

# An Interactive Program for the Analyses of Membrane Structure

Wei Xi<sup>1</sup>  
Shigeo Goto<sup>2</sup>  
Hiroyuki Obiya<sup>1</sup>  
Lei Liu<sup>3</sup>  
Katsushi Ijima<sup>4</sup>

## ABSTRACT

This paper describes the development and application of a Windows program for the form finding and load analysis of membrane structures using the finite-element technique. The large displacement effect is formulated based on the Tangent Stiffness method with the separation of element stiffness and geometric stiffness. Compared with the traditional DOS version program, the Windows version has stronger data processing capability and more friendly user interface. Moreover, since it runs under Microsoft Windows, this new program can conduct multiple instances simultaneously and share the other Windows resources.

## 1. INTRODUCTION

The design of membrane structures combines the architecture and engineering from the first beginning. The principle reason for this is that tensile membrane or cable network shows the shear-free pre-stressed mechanisms, in which form defines the path to resist the loads. It is very interesting and useful to know the interrelationships between structural performances and geometrical configurations. Also, it is important to find an equilibrium form which can satisfy not only the aesthetic and functional requirement, but also the fabric stress and boundary load criteria. In this paper, we introduce a Windows program for the form-finding and load analysis of pre-stressed membrane structures with considering the

geometric non-linearity. This Windows program particularly concentrates on the visualization of analysis and the flexibility of shape adjustment. Its features can be summarized as follows.

- Considering the large data size in Finite Element Method, this program employs the "paging" technique to allocate global memory so as to extern the basic data segment from the original 64Kb to a size limited only by the Windows systems and the hardware configurations. This technique considerably expands the data processing capability.

- In membrane structure design, graphic display is essential for visual interpretation of both form and analytic results, while numerical techniques are essential to the problem solving. The current Microsoft Windows application is just pointing to

1 Research Associate, Dept. of Civil Eng., Faculty of Science and Eng., Saga Univ.

2 Professor, Dept. of Civil Eng., Saga Univ.

3 Ph.D. Student, Dept. of Civil Eng., Saga Univ.

4 Associate Professor, Dept. of Civil Eng., Saga Univ.

offer great computational power with closely coupled graphics functions. Especially, the tedious manual manipulation of large input data file is replaced with on-screen interactive model generation and modification.

• Furthermore, this membrane structure analysis program for Windows can employ the other Windows resources as a part of itself. Other Windows software can be launched to serve this program freely during the analysis. Therefore, this program can realize a comparatively complete function with no more work.

## 2. NUMERICAL METHOD

The mechanical flexibility of membrane structure results in large deformations with the structure shape accommodating to loads. Thus, the form-finding and load analysis of membrane structure must be considered as a geometrically non-linear problem. In this program, we adopt a so-called Tangent Stiffness Method<sup>[1][2]</sup> to analyze the large-displacement effect while meshing the membrane surface with triangular finite elements.

According to the equilibrium equation of each joint node,

$$\mathbf{X} = \boldsymbol{\alpha} \mathbf{S} \quad (1)$$

its complete differential expression is written as,

$$\delta \mathbf{X} = \boldsymbol{\alpha} \delta \mathbf{S} + \delta \boldsymbol{\alpha} \mathbf{S} \quad (2)$$

Where  $\mathbf{X}$  is the joint force vector,  $\mathbf{S}$  is the element force vector and  $\boldsymbol{\alpha}$  is the coordinate transformation from local element coordinates to the global structural coordinate. The first term in the right hand of Eq.2 represents the effect of the change of element forces, whereas the second term depends only on the structural geometric

deformation and states of stress which are independent from material characteristics. From this point of view, the tangent stiffness is formulated with two parts named as element and geometric stiffness, respectively.

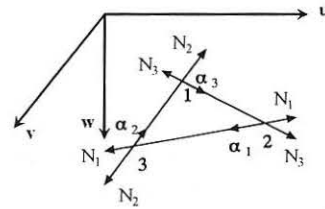


Fig. 1

To generate the tangent stiffness matrix of the structure in the updated Lagrangian approach, we select the three sides of the triangular element as the local coordinate system which is shown in Fig. 1. The element force vector  $\mathbf{S}$  is designated as the vector  $\mathbf{N}$  in this local coordinate system and vector  $\boldsymbol{\alpha}_1$ ,  $\boldsymbol{\alpha}_2$  and  $\boldsymbol{\alpha}_3$  are the direction cosines of the three sides of the element in the global Cartesian coordinate system  $\mathbf{uvw}$ .

The membrane facet defined in this local coordinate system has comparably small deformation. So, it is reasonable and convenient to be assumed as the constant-strain triangle. We develop element stiffness in incremental form as following,

$$\Delta \mathbf{N} = \mathbf{k} \Delta \mathbf{L} \quad (3)$$

where  $\mathbf{k}$  is the conventional linear element stiffness matrix with respect to the length increments of the triangular sides  $\Delta \mathbf{L}$ . Transforming Eq.3 to global coordinate system, leads to Eq.4 which is expressed by the element stiffness matrix  $[\mathbf{K}_0]^e$  and the vector of nodal displacement increments  $\Delta \mathbf{u}^e$ .

$$\boldsymbol{\alpha} \Delta \mathbf{N} = \boldsymbol{\alpha} \mathbf{k} \boldsymbol{\alpha}^T \Delta \mathbf{u} = [\mathbf{K}_0]^e \Delta \mathbf{u}^e \quad (4)$$

Geometric stiffness is derived from the second right term of Eq.2. It occurs when the element force changes its direction with the structural deformation. First, we consider the element force  $N_i$  in side  $i$  direction.

$$\begin{aligned} \Delta \alpha_i \cdot N_i &= N_i \cdot \Delta \begin{Bmatrix} \alpha \\ \beta \\ \gamma \end{Bmatrix}_i = N_i \cdot \Delta \begin{Bmatrix} (u_k - u_i) / L_i \\ (v_k - v_j) / L_i \\ (w_k - w_j) / L_i \end{Bmatrix} \\ &= \frac{N_i}{L_i} \begin{bmatrix} 1 - \alpha^2 & -\alpha\beta & -\alpha\gamma \\ -\alpha\beta & 1 - \beta^2 & -\beta\gamma \\ -\alpha\gamma & -\beta\gamma & 1 - \gamma^2 \end{bmatrix}_i \cdot \Delta \begin{Bmatrix} u_k - u_i \\ v_k - v_j \\ w_k - w_j \end{Bmatrix} \\ &= \mathbf{k}_{Gi} \Delta \mathbf{u}_{kj} \end{aligned} \quad (5)$$

where  $\Delta \mathbf{u}_{kj}$  is the difference between coordinate increments of two end joints of side  $i$ . And subscript  $i, j$  and  $k$  run in the order of 1, 2 and 3. Then, according to Fig.1, the joint force vector of triangle element can be written as

$$\Delta \alpha \mathbf{N} = \begin{bmatrix} N_2 \Delta \alpha_2 - N_3 \Delta \alpha_3 \\ N_3 \Delta \alpha_3 - N_1 \Delta \alpha_1 \\ N_1 \Delta \alpha_1 - N_2 \Delta \alpha_2 \end{bmatrix} \quad (6)$$

Substituting Eq.5 into Eq.6, we obtain the geometric stiffness matrix  $[\mathbf{K}_G]^c$ .

$$\begin{aligned} \Delta \alpha \mathbf{N} &= \begin{bmatrix} \mathbf{k}_{G2} + \mathbf{k}_{G3} & -\mathbf{k}_{G3} & -\mathbf{k}_{G2} \\ -\mathbf{k}_{G3} & \mathbf{k}_{G3} + \mathbf{k}_{G1} & -\mathbf{k}_{G1} \\ -\mathbf{k}_{G2} & -\mathbf{k}_{G1} & \mathbf{k}_{G1} + \mathbf{k}_{G2} \end{bmatrix} \cdot \begin{Bmatrix} \Delta u_1 \\ \Delta u_2 \\ \Delta u_3 \end{Bmatrix} \\ &= [\mathbf{K}_G]^c \Delta \mathbf{u}^e \end{aligned} \quad (7)$$

Geometric stiffness is important for the forming of membrane structure, because it accounts for the change in potential energy associated with the rotation of volume element under load. The membrane force acts tangent to the mid-surface of

the structure, so that in the initial flat configuration no stiffness in the normal direction resists the lateral loads if the geometric stiffness is neglected. Therefore, geometric stiffness is supplied not only for realizing precise analysis, but also for avoiding the ill-condition of the tangent stiffness matrix.

Assembling the element stiffness and geometric stiffness of each triangle into the structure matrix, we get the tangent stiffness equation of the structure in its current configuration,

$$\Delta \mathbf{U} = ([\mathbf{K}_0] + [\mathbf{K}_G]) \Delta \mathbf{u} \quad (8)$$

According to the Newton-Raphson method, the current deformation state is used as the reference state prior to the next incremental step of the solution. The unbalance force  $\Delta \mathbf{U}$  is the sum of the loads externally applied and the forces applied by element in each joint. In the equilibrium configuration, the unbalance force  $\Delta \mathbf{U}$  is zero. Instead, displacement increment  $\Delta \mathbf{u}$  will be calculated by solving Eq.8. Adding  $\Delta \mathbf{u}$  to the displacement vector  $\mathbf{u}$  accumulated in the preceding iteration, the local coordinates are updated to produce a new reference state. Then the next iteration is in order. Repeating this process, equilibrium shape is attained with the unbalance forces approaching to zero.

### 3. THE USER INTERFACE

This Windows program accepts the standard MDI ( Multiple Document Interface) technique to create a clear and efficient interface to ease the practical use. It allows the user to alternate the working window freely, so that multiple functions can be performed simultaneously.

In this program, totally, seven window classes are registered. From base to top, they are the Frame

window, the MDI Client window and four MDI Child windows named as Model, Form, Membrane and View. This four MDI Child windows are respectively created for 2-D original model buildup, 3-D form and real membrane graph visualization and old results review. All 3-D graphics can be rotated in the three axial directions and their effect of expression be changed by using the Function Buttons in the MDI Child windows. Additionally, a Speed Control Bar is set up to execute the major functions conveniently.

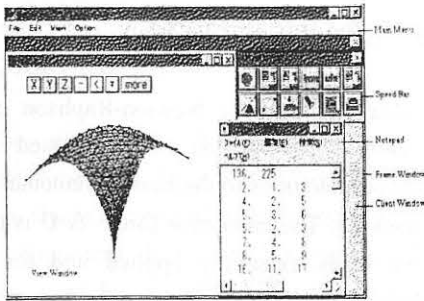


Fig.2 Main Menu

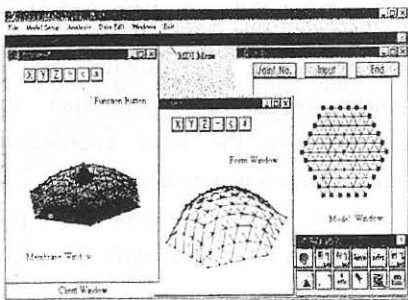


Fig.3 MDI Menu

As the program starts, Frame window, Client window and a Speed Control Bar show up simultaneously with a main command menu on the top of the frame. In the main menu [Fig.2], 'File' is the gate to enter the analysis regime, whereas 'View' and 'Edit' are used for reviewing the old results in both graph and text. In addition, an 'Option' item is designed here to facilitate user to assign directories for source data, final results and

middle temporary data.

While the sub-item, 'New' or 'Open', of the 'File' in the main menu, is selected to create a new or open an old 2-D original model, the main menu is replaced by a MDI menu. In the MDI menu [Fig.3], the 'Model Setup' can access to three menu items, 'Boundary', 'Stiffener' and 'Load', which are used to define the boundary conditions, the stiffener positions and the load vectors, respectively. The 'Analysis' function includes the form-finding and the real membrane calculation which are symbolized by menu items 'Form' and 'Membrane'. The 'Data Edit' runs the Notepad program to view and modify data files. Moreover, user can rearrange or close all MDI windows on the screen with 'Windows' and 'Exit' items.

#### 4. ANALYSIS PROCEDURE AND EXAMPLE

The analysis procedure consists of Original Model Setup, Form-finding and real Membrane analysis which are all visualized in the corresponding windows. Dialogue and message boxes are the interactive tools through which calculation required data are inputted and corrected as well as instant results and error information are offered. In order to get a reasonable design, the following three steps are performed repeatedly while the initial model, inner pressure and support condition are modified according to the analysis results of stress distribution and joint coordinates.

##### 4.1 Original Model Setup

Model setup is to generate a 2-D figure which includes the finite element mesh and conditions of boundary, stiffener and load. Instead of the tedious input data file, the original model is created with an on-screen work in this program. To begin a model setup, the vertex coordinates of a polygon and

some other parameters are keyed into the dialog box of new model. A finite element graph is then shown on the Original Model window with the mesh generated automatically [Fig.4].

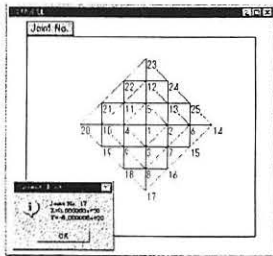


Fig.4 Initial mesh

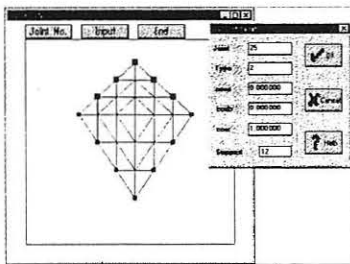


Fig.5 Adjusted mesh with boundary

User can adjust the mesh freely by pushing the left mouse button down on the selected node, dragging this node to any desired position with reference to the displayed nodal coordinate at the button of the window and releasing the button to fix the node in new position. The new nodal position can also be eliminated with the Undo Function of the right mouse button. If the changed joint is just a vertex of the polygon, mesh will be reformed in accordance with the new polygon style by clicking the left mouse button again. Fig.5 shows the mesh adjusted from Fig.4.

An other eminent feature of this program is the easy boundary definition. Under the boundary input status, boundary node is simply chosen by clicking mouse button on the corresponding position. Then a dialog box of boundary condition [Fig.5] prompts showing the nodal number chosen and other default parameters. While this support input is completed

and 'OK' button is selected, a label indicating the support type is immediately presented on the 2-D model, such as the square and circle in Fig.5. Similarly, joint loads and stiffeners are set on the membrane surface which are labeled with capital letter 'F' and red line, respectively.

## 4.2 Form Finding

The menu item 'Form' in the 'Analysis' function conducts the calculation under the assumption of no rigidity element. This assumption is quite different from actual structures, but the soap film shape produced this way is proved reasonable to be the initial form of the membrane structure<sup>[3]</sup>. From a 2-D original model with the given membrane stress and the boundary condition, the tangent stiffness matrix is generated only by geometric stiffness under the constant pre-stress condition. In this method, material characteristics are not necessary to be considered, so that the calculation is greatly simplified. Since the isotonic condition and force equilibrium condition can not be satisfied simultaneously, we control the configuration by iteration times instead of the nodal unbalance force in form-finding period.

Different isotonic surfaces can be obtained through adjusting the inner pressure and the constant element stress imposed on the soap film. Boundary condition and stiffener position are also variable for form-finding. Once more, dialog box undertakes the communication between user and computer. When 'Form' is selected, the dialog box in Fig.6 is displayed with default values. Users can modify these parameters according to requirements of calculation. Fig.6 shows a soap film form generated from the 2-D original model of Fig.5 under the inner pressure of  $0.01 \text{ tf/m}^2$  with element stress of  $0.04 \text{ tf/m}$  after 2 times iteration. Fig.7.2 shows a modified model with reference to the

initial one in Fig.5 by adding some stiffeners in the thick line positions. With the increased inner

pressure of  $0.012\text{tf/m}^2$  and the checked cable-box, form-finding gives the shape shown in Fig.7.1.

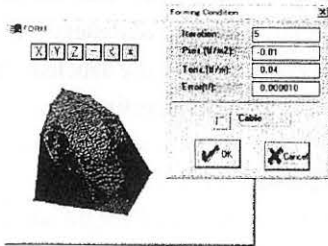


Fig.6 Form-finding

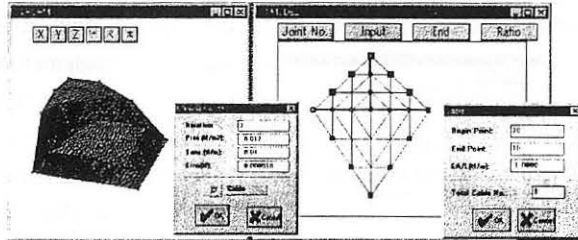


Fig.7.1 Form with stiffener

Fig.7.2 Original model

### 4.3 Membrane Analyses

Menu 'Membrane' in 'Analysis' function is arranged for real membrane analyses with the tangent stiffness matrix considering both geometric and element stiffness. In this process, the effects of material characteristics and load conditions are calculated for the decision of equilibrium shapes of the membrane structure. Also, the fabric stress are studied to evaluate the reasonableness of the formed shape.

We perform the membrane analysis with the initial shape in Fig.6 under the same boundary condition and inner pressure. The Et (Elastic modulus times material thickness) and Poisson ratio of the membrane fabric are  $0.12\text{tf/m}$  and  $0.4$ , respectively. For the soap film shape shown in Fig.6, its maximum nodal force imbalance is  $0.04\text{tf}$ . After 12 times of iteration in the real membrane solving, an equilibrium shape with unbalance force less than  $10^{-4}\text{tf}$  is obtained. However, the relative height difference between these two figures is less than 3%. Thus, the isotonic theory used in this program is practical for the form-finding of membrane structures. Fig.8 gives another example for the geometric comparison of the isotonic soap film and equilibrium membrane shape.

The membrane calculation can be started from either spatial soap film shape or flat pre-stressed

original model. In the first way, users can change the boundary and load condition or stiffener position of the 3-D configuration before performing real membrane calculation. For example, the equilibrium membrane shapes in Fig.9.2 and Fig.9.3 are all calculated from the initial form of Fig.9.1 but with different boundary conditions and calculation parameters. In the second way, equilibrium membrane shapes can be directly generated from flat model without the isotonic assumption. Fig.10.1 is an example created from the flat model in Fig.10.2 whose boundaries are all free except four fixed vertex points. No soap film shape can be generated from this kind of flat model, but it has solution in the real membrane calculation.

Membrane structures have large deformation under the loads. Load condition affects the membrane shape directly. In this program, user can easily investigate the shape changing of one structure under various loads. Fig.11 gives an example of different configurations generated from the initial shape in Fig.11.1 by changing its inner pressure and joint load condition.

Although this program focus on the structural graphing, data results are never neglected. Joint coordinates and element stresses are all saved in data files automatically. They are accessible at anytime of calculation. It is helpful for user to recognize the structure thoroughly.

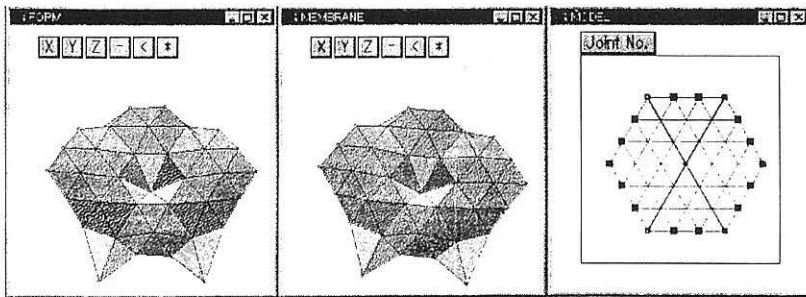


Fig.8 the comparison of isotonic shape and equilibrium shape

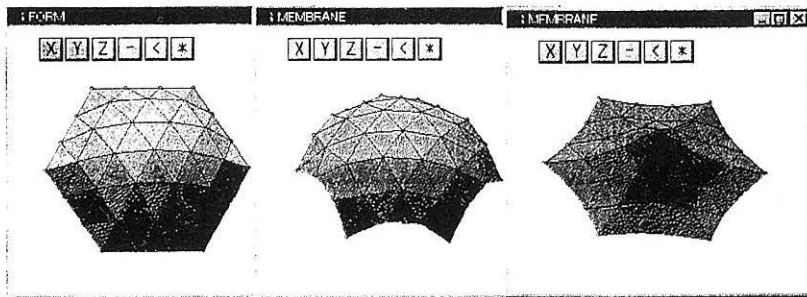


Fig.9.1 Soap film shape

Fig.9.2 membrane shape

Fig.9.3 membrane shape

(boundary condition changed) (inner pressure=0 & central point fixed)

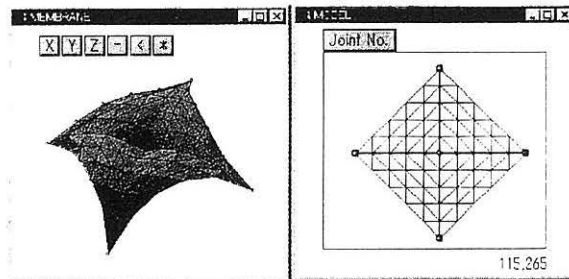


Fig.10.1

Fig.10.2

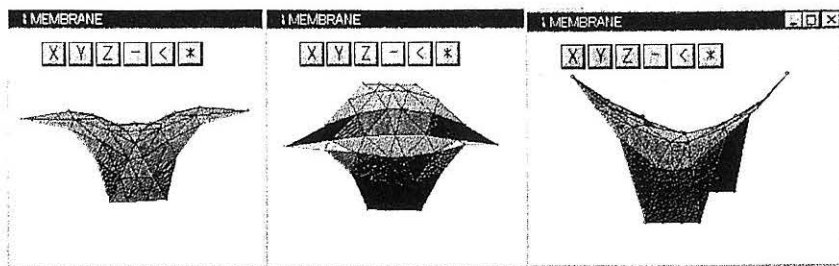


Fig.11.1

Fig.11.2

Fig.11.3

## 5. SUMMARY

This program employs the Tangent Stiffness method in the problem solving. It benefits from the separation of Geometric and Element stiffness. The initial shapes can be generated by merely considering the Geometric stiffness which has no relation to real material characteristics. Therefore, this method is also useful for form-finding of other large deformation structures, such as shell and cable structure. And this program is potential to be expanded to all kinds of large space structural analyses only if the element stiffness matrix is amended to represent their practical material property.

On the other hand, the present Windows program is designed from an engineer's standpoint. It is easy to learn and convenient to use. Windows

program method has shown significant flexibility and efficiency in the structural analysis field.

## References

1. Shigeo Goto, A study on tangent geometric stiffness without including inherent stiffness of element itself, Proceedings of Japanese society of Civil Engineering, 1991.
2. Shigeo Goto, Analyses of isotonic curved surface and membrane structure by the method with the separation of the element stiffness, Proceedings of Japanese society of Civil Engineering, 1991.
3. Shigeo Goto, A study on problems of determining curved surface forms using geometrical stiffness, Computational Mechanics, ISBN 9054103337

### 膜構造解析・形状解析のための対話型汎用プログラム

シー ウエイ\*1  
後 藤 茂 男\*2  
帯 屋 洋 之\*1  
劉 磊\*3  
井 嶋 克 志\*4

### 梗 概

要素剛性分離の手法による接線剛性法を用いた、明解な統一理論による膜構造汎用解析法の一層の実用化のため、その形状決定問題から、インフレ、デフレ時の解析、外力作用時の変形・応力解析を包括する汎用プログラムを開発した。本プログラムは、ウインドウズの機能を十分に活用する対話型処理形式となっており、従前のDOSバージョンに較べて、ユーザーの多様化する入出力条件のニーズに対して、より柔軟な対応が可能となっている。また、接線剛性法の理論的特性により、ユーザーにとっては、ブラックボックスの使用を強制されずに、ユーザー自身の力学的感覚を解析結果に反映できる使用環境を実現した。

\*1 佐賀大学理工学部建設工学科助手 \*2 同教授 \*3 同大学院博士後期過程 \*4 同助教授