Vietnam Journal of Mechanics, NCST of Vietnam Vol. 21, 1999, No 2 (116 – 128)

CALCULATION OF THE GOFFERED MULTILAYERED COMPOSITE CYLINDRICAL SHELLS BY USING THE FINITE ELEMENT METHOD

PHAM THI TOAN
University of Communications and Transport

SUMMARY. In the present paper, the goffered multilayered composite cylindrical shells is directly calculated by finite element method. Numerical results on displacements, internal forces and moments are obtained for various kinds of external loads and different boundary conditions.

1. Introduction

The classical methods for analyzing shell structure yield governing differential equations whose complexity depends greatly on the shell geometry.

Analytical solutions of these equations are available only for shells with simple geometric forms and for restricted boundary conditions.

The finite element method have been suggested for more general shell shapes.

In [3], by using the effective modulus theory and Seydel's assumptions, Dao Huy Bich et al. replaced a goffered shell by a equivalent shallow shell and the equilibrium equations for the shallow shell were derived. In the present paper, the goffered multilayered composite cylindrical shells is directly calculated by the using finite element method.

2. Model and fundamental equations of the finite element method

2.1. Finite element mesh

Let R, L and h respectively are the radius, length and thickness of the goffered multilayered composite cylindrical shells.

The homogennization method for studying composite materials had been introduced in [1]. By this method the elastic layered-composite material reduces to the set of anisotropic elastic material.

The finite element method implies an idealization of the shell surface as an assemblage of discrete structural elements. In this study, the shell with goffered cylindrical shapes is represented by rectangular flat elements.

Consider a typical rectangular flat element subjected simultaneously to in-plane and bending actions

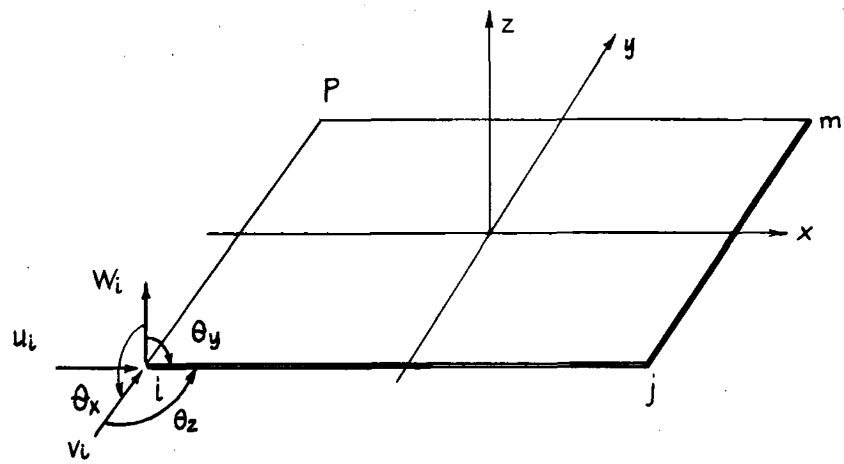


Fig. 1

2.2. The stiffness matrix for in-plane stress

The state of strain is uniquely described in terms of the u and v displacement of each typical node. The minimization of the total energy potential let to the stiffness matrices described there and gives

$$\{F^p\} = [k^p]\{\delta^p\}$$

where $\{\delta^p\}$ is the vector of nodal displacements

$$\{\delta^p\} = \{u_i\ v_i\ u_j\ v_j\ u_m\ v_m\ u_p\ v_p\}^T$$

$\{F^p\}$ is the vector of nodal forces

$$\{F^p\} = \left\{N_{xi}\ N_{yi}\ N_{xj}\ N_{yj}\ N_{xm}\ N_{ym}\ N_{xp}\ N_{yp}\right\}^T$$

The stiffness matrix of the rectangular elements in plane stress is defined from relationship as

$$[k^p] = \iint [B]^T [D] [B]\, hdxdy$$

$[B]$ is the strain displacement matrix in-plane stress and it is defined in chapter 4, [2].

Elasticity matrix [D] for orthotropic materials is given by [2].

2.3. The stiffness matrix for bending state

Similarly, when bending was considered, the state of strain was given uniquely by the nodal displacement θ_x and θ_y. This resulted in stiffness matrices of the type

$$\{F^b\} = [k^b]\ \{\delta^b\}$$

with

$$\{\delta^b\} = \left\{w_i\ \theta_{xi}\ \theta_{yi}\ w_j\ \theta_{xj}\ \theta_{yj}\ w_m\ \theta_{xm}\ \theta_{ym}\ w_p\ \theta_{xp}\ \theta_{yp}\right\}^T$$

$$[F^b] = \left\{N_{zi}\ M_{xi}\ M_{yi}\ N_{zj}\ M_{xj}\ M_{yj}\ N_{zm}\ M_{xm}\ M_{ym}\ N_{zp}\ M_{xp}\ M_{yp}\right\}^T$$

In which

$$[k^b] = \iint [B^*]^T[D^*][B^*]h\,dx\,dy$$

$[B^*]$ is the strain displacement matrix in bending of plate and it is defined in chapter 10, [2]; $[D^*]$ is the elasticity matrix for orthotropic material and it is defined in chapter 10, [2]

2.4. The stifness matrix for the shell element

The stiffness matrix for the shell element may therefore be obtained by superimposing of the two matrices: The stiffness matrix for the plane stress finite element and the stiffness matrix for the bending plate. The shell element has 6 degrees of freedom at each node

$$\{\delta_i\} = \left\{u_i\ v_i\ w_i\ \theta_{xi}\ \theta_{yi}\ \theta_{zi}\right\}^T$$

and the appropriate "forces" as

$$\{F_i\} = \left\{N_{xi}\ N_{yi}\ N_{zi}\ M_{xi}\ M_{yi}\ M_{zi}\right\}^T$$

The relationship between the corresponding nodal displacements is as follows

$$\begin{Bmatrix} \{F_i\} \\ \{F_j\} \\ \{F_m\} \\ \{F_p\} \end{Bmatrix} = \begin{Bmatrix} [k_{ii}] & [k_{ij}] & [k_{im}] & [k_{ip}] \\ [k_{ji}] & [k_{jj}] & [k_{jm}] & [k_{jp}] \\ [k_{mi}] & [k_{mj}] & [k_{mm}] & [k_{mp}] \\ [k_{pi}] & [k_{pj}] & [k_{pm}] & [k_{pp}] \end{Bmatrix} \begin{Bmatrix} \{\delta_i\} \\ \{\delta_j\} \\ \{\delta_m\} \\ \{\delta_p\} \end{Bmatrix}$$

or

$$\{F\}^e = [k]_{24\times 24}\{\delta\}^e.$$

In which, each stiffness matrix $[k_{rs}]$ is now made up from the following submatrices

$$[k_{rs}] = \begin{bmatrix} \begin{matrix} [k^p_{rs}] \end{matrix} & & 0 & 0 & 0 & 0 \\ & & 0 & 0 & 0 & 0 \\ 0 & 0 & & & & 0 \\ 0 & 0 & & [k^b_{rs}] & & 0 \\ 0 & 0 & & & & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix}$$

2.5. Transformation to global co-ordinates

The stiffness matrix derived in the previous section used a system of local co-ordinates. Transformation of co-ordinates to a common global system (which now will be denote by $x'y'z'$, and the local system by xyz) will be necessary to assemble the elements and to write the appropriate equilibrium equations. The forces and displacements of a node transform from the global to the local system by a matrix L giving

$$\{\delta_i\} = [L]\{\delta_i'\}, \quad \{F_i\} = [L]\{F_i'\}$$

in which

$$[L] = \begin{bmatrix} \lambda & 0 \\ 0 & \lambda \end{bmatrix}$$

with $[\lambda]$ being a three by three matrix of direction cosines of angles formed between the two sets of axes. The stiffness matrix of an element in the global co-ordinates becomes

$$[k'] = [T]^T[k][T]$$

2.6. Structural stiffness matrix

We can write equilibrium equations for structure

$$[K]\{\delta\} = \{R\} \qquad (2.1)$$

where $[K]$ is structural stiffness matrix, and it is assembled from the stiffness matrices element by the standard way, $\{\delta\}$ is the nodal displacements of the structure, and $\{R\}$ is the consistent load vector of the structure. The nodal displacements are obtained by solving the equation (2.1). The resulting displacements calculated are referred to the global system.

The nodal displacements in the local system $\{\delta\}^e$ are then computed by

$$\{\delta\}^e = [T]\{\delta'\}$$

These displacements can be decomposed into in-plane displacement $\{\delta^p\}$ and bending displacements $\{\delta^b\}$. The element stresses can be obtained as follows

$$\begin{aligned}\{\sigma^p\} &= [D][B]\{\delta^p\} = [S^p]\{\delta^p\}\\ \{\sigma^b\} &= [D^*][B^*]\{\delta^b\} = [S^b]\{\delta^b\}.\end{aligned}$$

The stress at arbitrary point in the element is a combination of the above in plane stresses and bending stresses. The strain at arbitrary point in the element is defined as follows

$$\begin{aligned}\{\varepsilon^p\} &= [B]\{\delta^p\}\\ \{\varepsilon^b\} &= [B^*]\{\delta^b\}.\end{aligned}$$

3. A procedure for solving the problem

FEA, consultancy department is a UK leader in the application of finite element technology. The LUSAS of FEA was recognized in the course of different application led to the development of more sophisticated general systems.

The goffered cylindrical shells is calculated by using LUSAS program.

At first, we must established the function coordinates refer to the co-ordinates of all the nodal points. The shell is devided into the subdivisions following

Subdivision 1

$$x_1 = 0$$

$$\begin{aligned}y_{1i} &= R\cos\frac{(i-1)\pi}{n}\\ z_{1i} &= R - R\sin\frac{(i-1)\pi}{n} \qquad (i = 1, 2, \dots, n+1)\end{aligned}$$

n is element numbers are devided refer to the directional cross vault

Subdivision 2

$$\begin{aligned}H_1 &= \frac{4H^2 + \ell_1^2 - 4\ell_1 a}{8H} = H\sin\left(\frac{2\pi a}{\ell_1}\right)\\ x_2 &= a\\ y_{2i} &= (H_1 + R)\cos\frac{(i-1)\pi}{n}\\ z_{2i} &= R - (R + H_1)\sin\frac{(i-1)\pi}{n}\end{aligned}$$

Subdivision 3

$$x_3 = \frac{\ell_1}{a}$$

$$y_{3i} = (H + R)\cos\frac{(i-1)\pi}{n}$$
$$z_{3i} = R - (R + H)\sin\frac{(i-1)\pi}{n}$$

Subdivision 4

$$x_4 = \ell_1 - a$$

$$y_{4i} = y_{2i}; \quad z_{4i} = z_{2i}$$

Subdivision 5

$$x_5 = \ell_1$$

$$y_{5i} = y_{1i}; \quad z_{5i} = z_{1i}$$

Subdivision 6

$$x_6 = \ell_1 + b$$

$$H_2 = \frac{4H^2 + \ell_2^2 - 4\ell_2 b}{8H} = H\sin\left(\frac{2\pi b}{\ell_2}\right)$$
$$y_{6i} = (R - H_2)\cos\frac{(i-1)\pi}{n}$$
$$z_{6i} = R - (R - H_2)\sin\frac{(i-1)\pi}{n}$$

Subdivision 7

$$x_7 = \ell_1 + \frac{\ell_2}{2}$$
$$y_{7i} = y_{3i}; \quad z_{7i} = z_{3i}$$

Subdivision 8

$$x_8 = \ell_1 + \ell_2 - b$$

$$y_{8i} = y_{6i}; \quad z_{8i} = z_{6i}$$

Subdivision 9

$$x_9 = \ell_1 + \ell_2$$

$$y_{9i} = y_{1i}; \quad z_{9i} = z_{1i}.$$

The second step is the determination of material properties. By using the homogenization method, the multilayered composite material reduces to the orthotropic material.

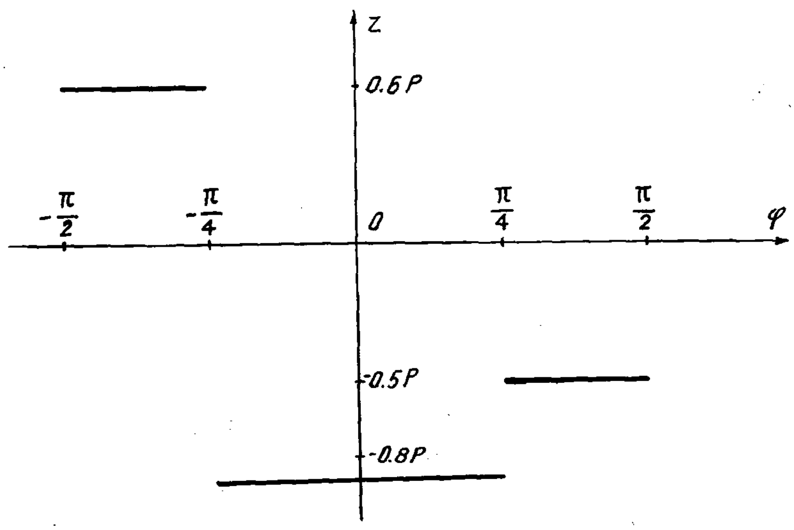
The third step is the insertion of prescribed loading conditions

1) Uniform distributed load p from outside

2) Uniform distributed load p from inside

3) Wind load

Diagram of wind load is under the form



The final step is the insertion of prescribed boundary conditions

The boundary lines along generating line are strictly clamped.

The remains are free or simply supported.

4. Numerical results

1) **Data input :** $n = 20$

$R = 6190\,\text{mm}$ $\quad$ $H = 188\,\text{mm}$

$\ell_1 = 370\,\text{mm}$ $\quad$ $\ell_2 = 190\,\text{mm}$

$L = 17000\,\text{mm}$ $\quad$ $h = 5\,\text{mm}$

$E_1 = E_2 = 25000\ \text{N/mm}^2,\ \gamma = 0.15,\ G = 3000\ \text{N/mm}^2,\ p = 0.00147\ \text{N/mm}^2$

2) Static analysis

Numerical results on displacements, internal forces and moments are obtained for various kinds of external loads and different boundary conditions. We represented some typical cases

Case 1 - Uniform distributed load p from outside

- The boundary lines along generating line are strictly clamped and the remains are free. Fig. 2 shows the deflection on middle line of the cross-section of the shell.

The internal forces indicated in Fig. 3, Fig. 4 and Fig. 5.

The bending moments indicated in Fig. 6, Fig. 7 and Fig. 8

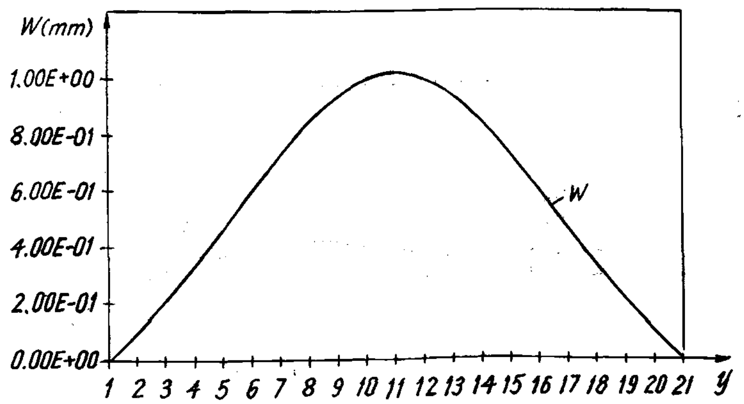


Fig. 2

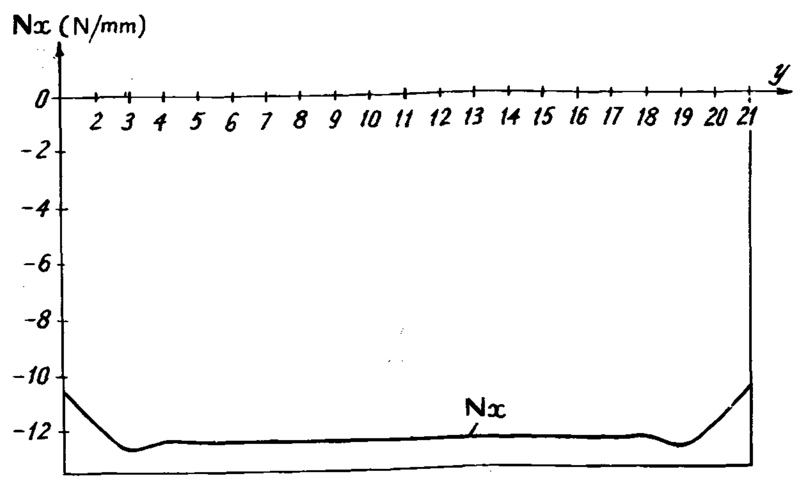


Fig. 3

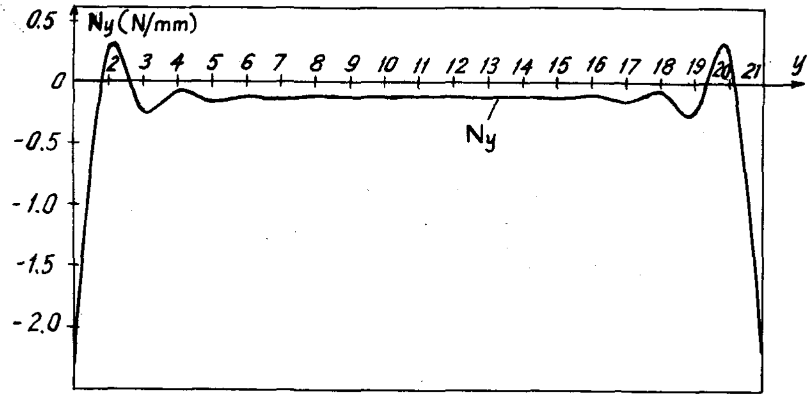


Fig. 4

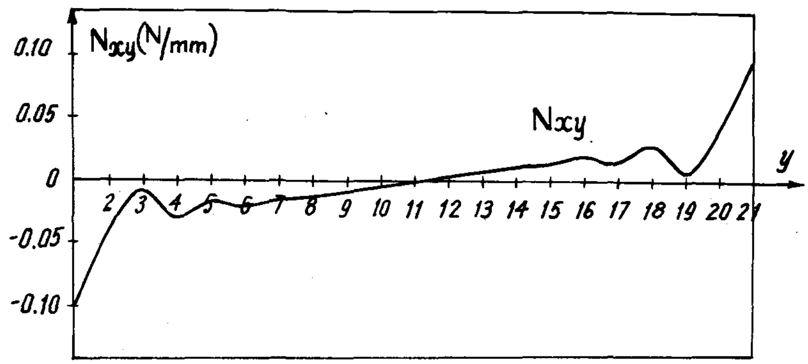


Fig. 5

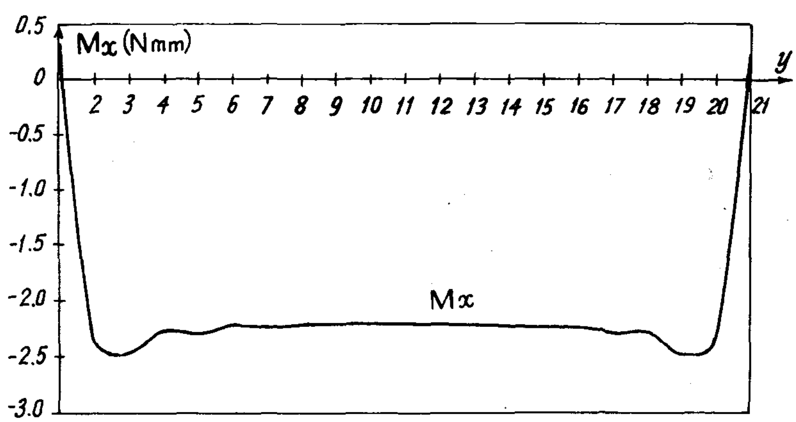


Fig. 6

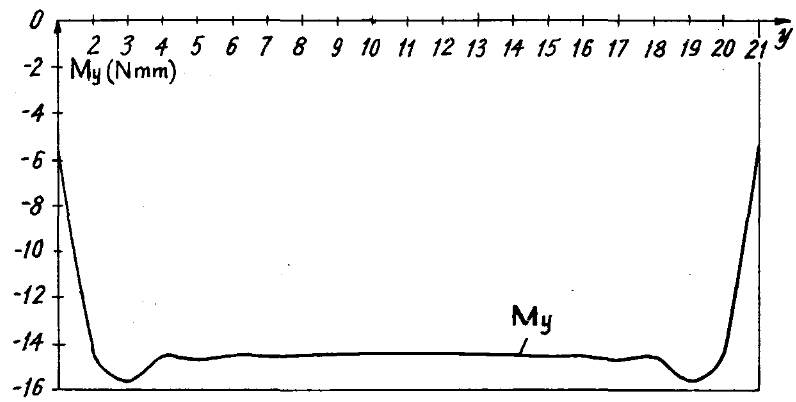


Fig. 7

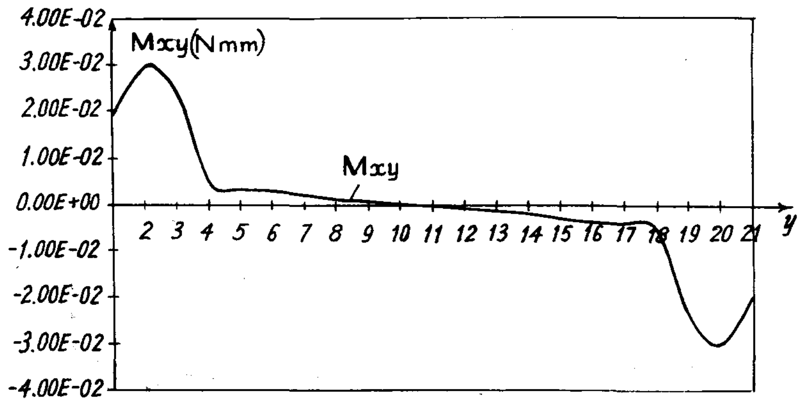


Fig. 8

Case 2. - Wind load

- The boundary lines along generating line are strictly clamped and the remains are. free Fig. 9 shows the deflection on middle line of the cross-section of the shell.

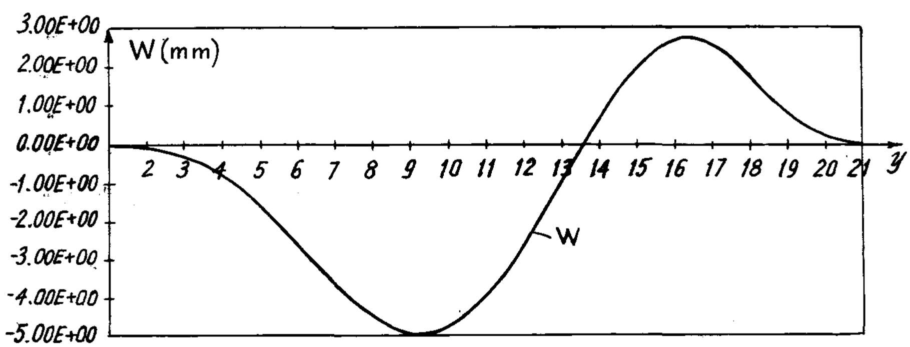


Fig. 9

The internal forces indicated in Fig. 9, Fig. 10, Fig. 11 and Fig. 12
The bending moments indicated in Fig. 13, Fig. 14 and Fig. 15

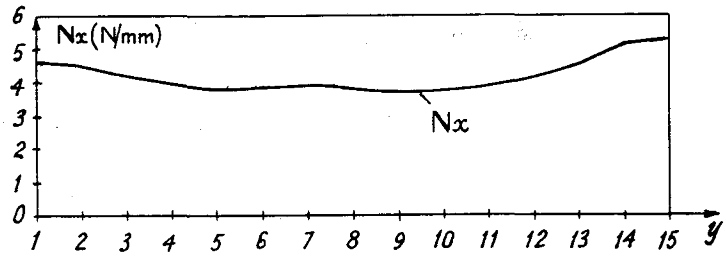


Fig. 10

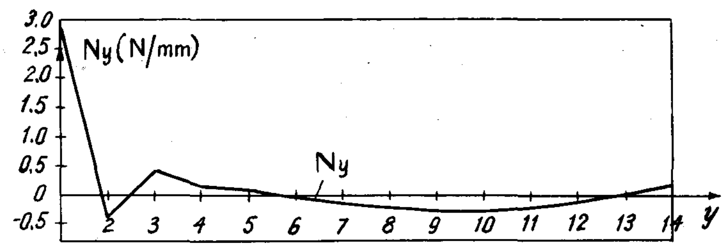


Fig. 11

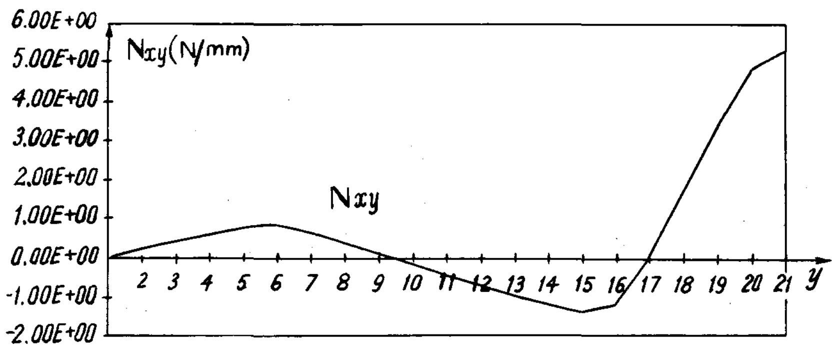


Fig. 12

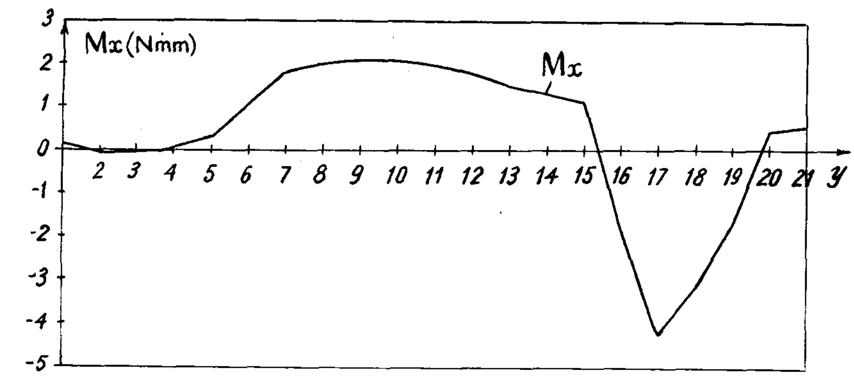


Fig. 13

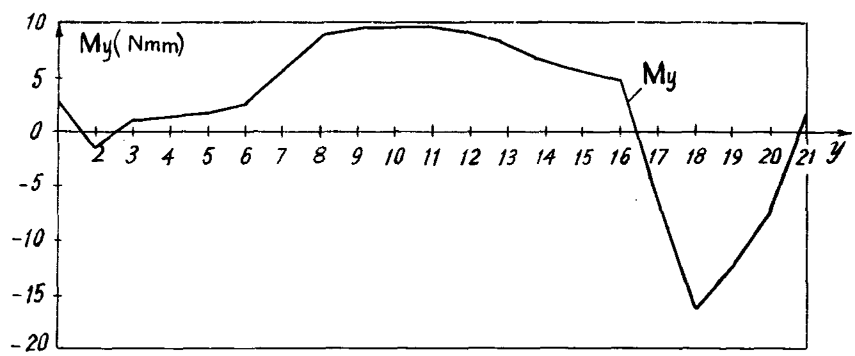


Fig. 14

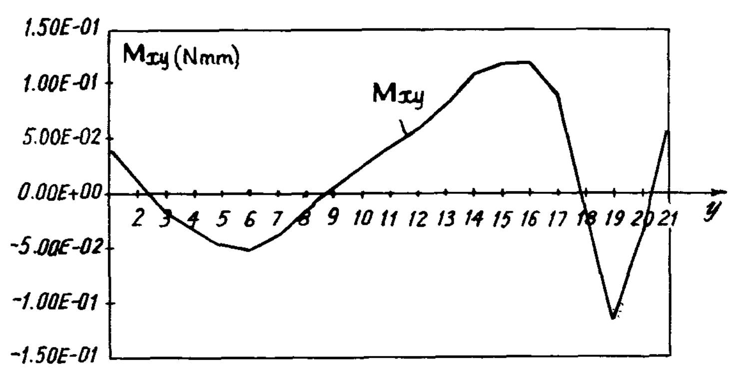


Fig. 15

Comment

The finite element method implies an idealization of the shell as an assemblage of discrete structural elements. Calculate results are found the results of argument. It is seem that the boundary conditions, loads and shape are symmetric then the deflection and internal forces are symmetric.

Conversely, it is not the case for the wind load (asymmetric).

5. Conclusions

In this paper, by using the finite element method, the goffred multilayered composite cylindrical shells is directly calculated without assumptions. The author would like to thank Prof. D. Sc. Dao Huy Bich for helping her in completing this work. This paper is completed with financial support from the National Basis Research Programe in Natural Sciences.

REFERENCES

1. Pobedrja B. E. Mechanics of composite materials, MGU, 1984.
2. Zienkiewicz O. C. Finite element method in engineering science, Mc Graw-Hill PCL, 1971.
3. Dao Huy Bich et al. Calculation of the goffered multilayered composite cylindrical shells. Journal of Mechanics, No 3, 1997.
4. Dao Huy Bich. Theory of Elasticity, Publishing House of Vocational University and Education. Hanoi 1979.
5. Timoshenko S. Woinowsky - Krieger S. Theory of plates and shells. Mc Graw-Hill Book Company, NY 1959.

Received March 25, 1999

TÍNH TOÁN VỎ COMPOSIT LƯỢN SÓNG NHIỀU LỚP
BẰNG PHƯƠNG PHÁP PHẦN TỬ HỮU HẠN

Trong bài báo này, vỏ trụ composite nhiều lớp lượn sóng được tính toán trực tiếp bằng phương pháp phần tử hữu hạn. Các kết quả số là chuyển vị, nội lực và mô men đạt được cho các loại khác nhau của tải trọng ngoài và các điều kiện biên khác nhau.