PROCEEDINGS OF THE YEREVAN STATE UNIVERSITY

Physical and Mathematical Sciences 2013, № 1, p. 38–43

Mechanics

SOME FEM MODELS FOR BENDING AND VIBRATION PROBLEMS OF BEAM

BEHROOZ YAZDIZADEH[*1], H. H. YEGHIAZARIAN[2]

[1] *Chair of Mechanic YSU, Armenia*
[2] *Institute of Mechanics NAS RA*

The problem of beam bending and proper frequency under load has been considered. The displacement, stresses and the frequency of the beam have been calculated using finite elements such as the *Beam*, *Shell, Plane* and *Solid.* The calculations have been carried out by applying the ANSYS software package and comparisons show that the results of modeling using the *Plane* elements prove more accurate.

Keywords**:** beam, bending, frequency, finite element.

Introduction. The finite element analysis is a method of numerical modeling employed for accurate solution of complex engineering problems. First developed for analyses of problems of aircraft structures in 1956, it gained acceptance during the following decade due to its potentiality in solution of different types of applied science and engineering problems [1]. Over the years, the finite element technique has been so well established that today it is considered one of the best methods for solving a wide variety of practical problems efficiently. In fact, the method has become one of the active research areas for applied mathematicians. One of the main reasons for the popularity of the method in different fields of engineering is that once a general computer program is written, it can be used for the solution of any problem simply by changing the input data [2].

Theoretical Aspects. In Fig. 1 is shown the displacement formula for a beam

$$EI\partial^4 y/\partial x^4 = 0, \tag{1}$$

where E is the elasticity modulus; I is the moment of inertia; y is vertical displacement of a point in x position [3]. The solution of above differential equation for a beam in Fig. 1 under the boundary conditions

$$dy/dx|_{x=L} = 0,\ \ y|_{x=L} = 0$$

is $y = P(2L^3 - 3L^2x + x^3)/6EI$. The maximum displacement (at $x = 0$) is $PL^3/3EI$.

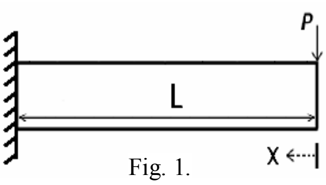


Fig. 1.

[*] E-mail: behyazd@gmail.com

The normal and shear stresses for beam (Fig. 1) are obtained from relation (2) and (4) respectively [4]. Relation (4) is obtained from relation (3) applying the beam conditions:

$$\sigma = \frac{My}{I}, \tag{2}$$

$$\tau_{xy} = \frac{VQ}{It}, \tag{3}$$

$$\tau_{xy} = \frac{3}{2}\cdot\frac{V}{A}\left(1-\frac{y^2}{c^2}\right), \tag{4}$$

where Q is the first or statical moment of area from y to natural axis; V is the shear force acting on section; t is the beam thickness; σ is normal stress, τ is the shear stress; y is the distance of the point, the shear stress at which ought to be calculated; c is the maximum distance from the beam surface to natural axis [5]. It is clear that maximum normal stress $\sigma_{\max} = \frac{Mc}{I}$, and maximum shear stress $\tau_{\max} = \frac{3}{2}\cdot\frac{V}{A}$.

There are several ways to calculate the beam frequency. Relation (5) was obtained from Rayleigh approximation method and the error of this approximate solution is less than 0.5% [6]:

$$f = \frac{1}{2\pi}\sqrt{\frac{90\cdot 9}{65}}\cdot\sqrt{\frac{EIg}{\omega L^4}}, \tag{5}$$

where f is the lateral frequency of the first mode; ω is weight per unit length; g is volume coefficient choosen here to be equal to 1.

Different Element Using in ANSYS Software.

1. *2D Elastic Beam* is a uniaxial element with tension, compression and bending capabilities. The element has three degrees of freedom at each node: translations in the nodal x and y directions and rotation about the nodal z-axis.

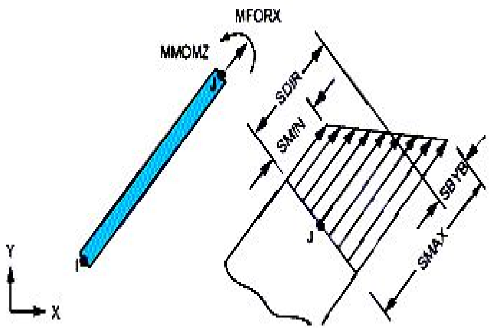


Fig. 2. Beam geometry.

Fig. 2 shows the geometry, node locations and the coordinate system for this element [7]. The element is defined by two nodes, the cross-sectional area, the inertia moment of area, the height and the material properties. The initial strain in the element (ISTRN) is given by Δ / L, where Δ is the difference between the element length L (as defined by the I and J node locations) and the zero strain length. The initial strain is also used in calculating the stress stiffness matrix, if any, for the first cumulative iteration.

The element could be used in an axisymmetric analysis, if hoop effects are negligible such as for bolts, slotted cylinders etc. The area and moment of inertia should be introduced on a full 360^0 basis for an axisymmetric analysis. The shear deflection constant (SHEARZ) is optional. To neglect the shear deflection a zero value of SHEARZ is used. The shear modulus (GXY) is used only with the shear deflection. One can specify an added mass per unit length with a ADDMAS real constant.

2. *Elastic Shell* has both the bending and the membrane capabilities. Both in-plane and normal loads are permitted. The element has six degrees of freedom at each node: translations in the nodal x, y and z directions and rotations about the nodal x, y and z-axis. The stress stiffening and large deflection capabilities are included. A consistent tangent stiffness matrix option is available for use in large deflection (finite rotation) analyses.

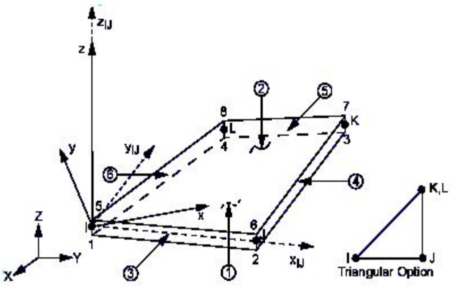


Fig. 3. Shell geometry:
x_{IJ} = element x-axis, if ESYS is not supplied;
x = element x-axis, if ESYS is supplied.

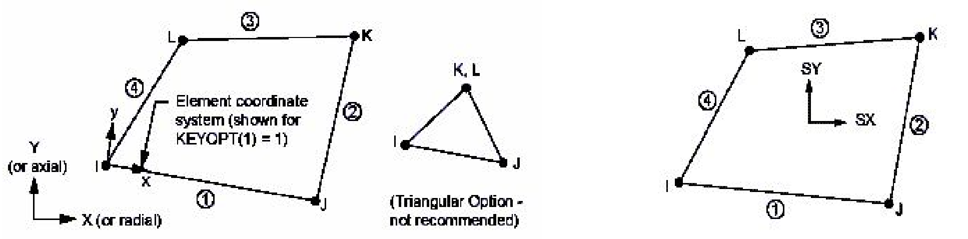
The geometry, node locations and the coordinate system for this element are shown in Fig. 3 [7]. The element is defined by four nodes, four thicknesses, elastic foundation stiffness and the orthotropic material properties. The directions of orthotropic material correspond to the directions of element coordinate. The element x-axis may be rotated by an angle θ (in degrees). The thickness is assumed to vary smoothly over the area of the element with the thickness input at the four nodes. If the element has a constant thickness, only *TK*(*I*) shall be entered. If the thickness is not constant, all four thicknesses must be entered. The stiffness of elastic foundation (EFS) is defined as the pressure required for production of a unit normal deflection of the foundation. The elastic foundation capability is bypassed, if EFS is less than or equal to zero.

If not entered the stresses are based on the input of thicknesses. ADMSUA is the added mass per unit area. Pressures may be input as surface loads on the element edges as shown by the circled numbers on Fig. 3. The positive pressures act on the element. The edge pressures are entered as the force per unit length. The equivalent element load produces more accurate stress results with flat elements representing a curved surface or elements supported on an elastic foundation since certain fictitious bending stresses are eliminated.

3. *Plane* (2D *Structural Solid*) is used for 2D-modeling of solid structures. The element can be used either as a plane element (plane stress or plane strain) or as an axisymmetric element. The element is defined by four nodes having two degrees of freedom at each node: the translations in the nodal x and y directions. The element has the plasticity, creep, swelling, stress stiffening, large deflection and large strain capabilities. An option is provided for suppression of extra displacement shapes.

The geometry, node locations and the coordinate system for this element are shown in Fig. 4, a [7]. The input data for this element includes four nodes, a thickness (for the plane stress option only) and the orthotropic material properties. The directions of orthotropic material correspond to the element coordinate directions.

The element loads are described in Node and Element Loads. The pressures may be entered as the surface loads on the element edges as shown by the circled numbers on Fig. 4, a. The positive pressures act on the element. The temperatures and fluences may be entered as the element body loads at the nodes. The node *I* temperature *T*(*I*) defaults to TUNIF. If all other temperatures are not specified, they default to *T*(*I*). For any other input pattern, the unspecified temperatures default to TUNIF. Similar defaults occurs for fluence except that zero is used instead of TUNIF.



a b

Fig. 4. Plane: a – geometry; b – stress output.

The nodal forces, if any, should be input per unit depth for the plane analysis.

The initial state conditions previously handled via the ISTRESS command will be discontinued for this element. The INISTATE command will provide increased functionality. To continue using Initial State conditions in the future versions of ANSYS, the switching to the appropriate Current Technology element is considered.

Several items are illustrated in Fig. 4, b. The element stress directions are parallel to the element coordinate system. Surface stresses are available on any edge. The surface stresses on edge *IJ*, for example, are defined parallel and perpendicular to the *IJ* line and along the *z*-axis for the plane analysis or in the hoop direction for the axisymmetric analysis.

4. *3D 8-Node Structural Solid or Layered Solid* is used for 3D-modeling of solid structures. It is defined by eight nodes having three degrees of freedom at each node translations in the nodal *x*, *y* and *z* directions. The element has plasticity, hyperelasticity, stress stiffening, creep, large deflection and large strain capabilities. It also has mixed formulation capability for simulating deformations of nearly incompressible elastoplastic materials, and fully incompressible hyperelastic materials.

Solid is available in two forms: 1 – Structural Solid; 2 – Layered Solid.

Structural Solid is suitable for modeling general 3D solid structures. It allows for prism and tetrahedral degenerations when used in irregular regions. Various element technologies such as B-bar, uniformly reduced integration and enhanced strains are supported.

The geometry and node locations for this element are shown in Fig. 5, a [7]. The element is defined by eight nodes and the orthotropic material properties. The coordinate system of default element is along the global directions. One may define an element coordinate system using ESYS, which forms the basis for orthotropic material directions.

The element loads are described in Node and Element Loads. The pressures may be entered as surface loads on the element edges as shown by the encircled numbers. Positive pressures act on the element. The temperatures may be input as element body loads at the nodes. The node *I* temperature *T*(*I*) defaults to TUNIF. If all other temperatures are not specified, they default to *T*(*I*). For any other input temperature pattern, the unspecified temperatures default to TUNIF. Similar defaults occurs for the fluence except that zero is used instead of TUNIF.

For the case of hyperelastic materials, the output of stress and strain is always with respect to the global Cartesian coordinate system rather, than following the material/element coordinate system. The effects of pressure load stiffness are automatically included for this element. Several items are illustrated in Fig. 5, b. Stress directions are shown for global directions.

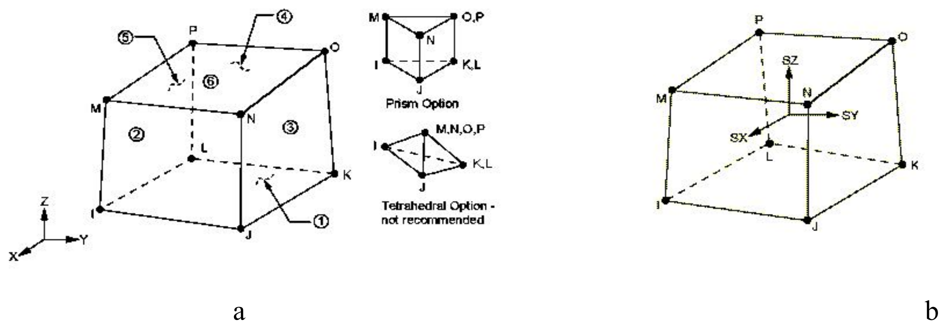


Fig. 5. Structural Solid: a – geometry; b – stress output.

Problem Solving. These parameters are considered for beam:

$$L = 1\ m,\ P = 10\ kN,\ b = 0.1\ m,\ h = 0.1\ m,\ E = 2{\cdot}10^{12}\ N/m^2,\ \rho = 7800\ kg/m^3,$$

where h is the beam height, b – thickness and ρ is the beam density.

After giving value to variables of relations (1)–(4) and calculation, the following values for maximum displacement, normal and shear stresses and frequency are obtained as $2{\cdot}10^{-4}$ *m*, $6{\cdot}10^7$ N/m^2, $1.5{\cdot}10^6$ N/m^2 and 259.707 s^{-1}, respectively.

The finite elements method can be easily implemented for beam elements since the matrices of stiffness and generalized geometrical stiffness of a noncracked beam are already commonly known [8].

Results and Discussion. The results of maximum displacement, maximum normal stresses, maximum shear stresses and frequency obtained analytically and numerically are shown in Table.

Problem results in different way

	Max. displacement, *m*	Max. normal stresses, N/m^2	Max. shear stresses, N/m^2	Frequency, ρ^{-1}
Analytical Solution	$0.2{\cdot}10^{-03}$	$6{\cdot}10^7$	$1.5{\cdot}10^6$	259.707
Beam	$0.2{\cdot}10^{-03}$	$4.4{\cdot}10^7$	$1{\cdot}10^6$	258.17
Plane	$0.20145{\cdot}10^{-03}$	$6.0113{\cdot}10^7$	$1.4208{\cdot}10^6$	257.04
Shell (membrane)	$0.20104{\cdot}10^{-03}$	$6.189{\cdot}10^7$	$2.9347{\cdot}10^6$	257.10
Shell (include bending)	$0.1998{\cdot}10^{-03}$	$6.747{\cdot}10^7$	$3.874{\cdot}10^6$	258.16
Solid (normal mesh)	$0.2001{\cdot}10^{-03}$	$7.0243{\cdot}10^7$	$3.047{\cdot}10^6$	257.87
Solid (fine mesh)	$0.1998{\cdot}10^{-03}$	$6.747{\cdot}10^7$	$3.874{\cdot}10^6$	258.16

Percentage of maximum displacement errors of each kind of element from analytical solution are as follows: *Beam* element – 0%; *Plane* element – 0.73%; *Shell membrane* element – 0.52%; *Shell bending* element – 1.11%; *Solid* element – 0.1%; *Solid (fine mesh)* element – 0.05%.

Percentage of maximum normal stress errors of each kind of element from analytical solution are as follows: *Beam* element – 26.67%; *Plane* element – 0.19%; *Shell membrane* element – 3.15%; *Shell bending* element – 19.48%; *Solid* element – 12.45%; *Solid (fine mesh)* element – 17.07 %.

Percentage of maximum shear stress errors of each kind of element from analytical solution are as follows: *Beam* element – 33.33%; *Plane* element – 5.27%; *Shell membrane* element – 95.67%; *Shell bending* element – 319.47%; *Solid* element – 158.27%; *Solid (fine mesh)* element – 103.13%.

Percentage of frequency errors of each kind of element from analytical solution are as follows: *Beam* element – 0.59%; *Plane* element – 1.03%; *Shell membrane* element – 1%; *Shell bending* element – 1%; *Solid* element – 0.60%; *Solid (fine mesh)* element – 0.71%.

Conclusion. Although for cases in question the *Beam* element provided higher accuracy, but it is better to use *Plane* element for the problem in this case, because it has acceptable accuracy for all results. In addition, such structural problems as fracture problems need *Plane* or *Solid* element for modeling the problem, because the crack cannot be inserted in the *Beam* element. If the problem is two dimensional the *Plane* element is considered. *Solid* element is used for unsymmetrical and complicated problems that cannot be modeled in the *Beam* or other element. Because the modeling in *Solid* element is not easy and *fine mesh,* it needs more time for calculation. So, the choice of any kind of element for solving the mechanic problems depends on what output results are requested and how much accuracy is desirable.

Special thanks to my supervisor Prof. Karen Ghazaryan for his special helps.

Received 02.12.2010

REFERENCES

1. **Gupta K.K., Meek J.L.** A Brief History of the Beginning of the Finite Element Method. // International Journal for Numerical Methods in Engineering, 1996, v. 39, p. 14.
2. **Singiresu S.R.** The Finite Element Method in Engineering(4th ed.). Elsevier Science & Technology Books, 2004, 658 p.
3. **Popov E.P.** Engineering Mechanics of Solids (1st ed.). New Jersy: Prentice-Hall, 1990, 760 p.
4. **Beer F.P., Johnson E.R.** Mechanics of Material (2nd ed.). NY: Mc Graw-Hill, 1925, 532 p.
5. **Timoshenko S.P., Goodier J.N.** Theory of Elasticity (3rd ed.). NY: Mac Graw-Hill, 1970, 506 p.
6. **Timoshenko S.P.** Vibration Problems in Engineering (2nd ed.). NY: D. Van Nostrand Company, INC, 1937, 465 p.
7. **Workbench A.** Ansys Workbench help. NY, 2007.
8. **Bhatt P.** Structures. Harlow, Essex, England : Addison Wesley Longman, 1999, 597 p.