PROCEEDINGS OF THE YEREVAN STATE UNIVERSITY

Physical and Mathematical Sciences 2010, № 3, p. 44–50

Mechanics

COMPARISON OF DIFFERENT PLANE MODELS IN FINITE ELEMENT SOFTWARE IN STRUCTURAL MECHANICS

B. YAZDIZADEH*

Chair of Mechanics, YSU

In solution of plane problems of mechanics there are several elements used in finite element software. In ANSYS – one of the best finite element software – there are about six type of elements. We consider different Plane Models in simple bending problem and compare the various results to distinguish the type of elements, which are suitable for solving the problem under consideration. Comparing results with analytical solutions shows that the Plane 42 and 82 models are the most suitable ones among the others. Also results in Plane 42 with usual mesh is closer to the same problem solution with fine mesh than Plane 82 in the same case.

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

Introduction. The finite element method is a numerical method that can be used for the accurate solution of complex engineering problems. The method was first developed in 1956 for the analysis of aircraft structural problems. Thereafter, within a decade, the potentialities of the method for the solution of different types of applied science and engineering problems were recognized [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].

Nowadays, we live a curious situation. On one hand, most structural engineers and FE codes for computational solid mechanics are decanted. On the other, the observed mesh-size and mesh-bias dependence exhibited by these models make the academic world very suspicious about this format. Hence, a lot of effort has been spent in the last 30 years to investigate and remedy the observed drawbacks of this approach [3].

In some complicated problems, such as cracks and contacts with no effective analytic solution, numerical analysis is strongly recommended. Finite element

* E-mail: behyazd@gmail.com

analysis is one of the usual ways to solve this kind of problems. In ANSYS software, there are some elements to solve plane problems, so it is necessary to choose the right type of elements type complex problems to obtain correct results.

Theoretical Aspects. We assume simple plane bending as illustrated in Fig. 1.

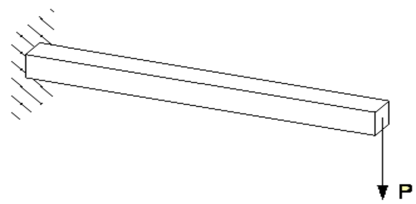

Fig. 1.

The displacement formula is [4]:

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

where E is the module of elasticity; I is the moment of inertia; y is the vertical displacement of the point in x position.

As we know for the beam of Fig. 1, solution of above differential equation with the following boundary conditions $\left.\frac{\partial y}{\partial x}\right|_{x=L} = 0$, $y|_{x=L} = 0$ is

$$y = \frac{p}{6EI}\left(2L^3 - 3L^2 x + x^3\right).$$

The maximum displacement (at $x = 0$) is $\frac{PL^3}{3EI}$.

The normal and shear stresses for the beam in Fig. 1 are obtained from relation (2) and (4) respectively [5]. 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 moment of area from y to natural axis; V is the shear force acting on section; t is the beam thickness; σ is the normal stress; τ is the shear stress; y is the distance of a such point, where the shear stress must be calculated; c is the maximum distance from beam surface to natural axis and A is the area of cross section of the beam. For more information see [6]. 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 bending frequency. Relation (5) is obtained from Rayleigh approximation method and the error of this approximated solution is less than 0.5% [7]:

$$f = \frac{1}{2\pi} k \sqrt{EIg / \omega L^4}\ , \tag{5}$$

where f is the first mode lateral frequency; ω is the weight per unit length; g is the volume coefficient that is chosen here to be equal to 1 and k is the constant that is equal to 3.53.

We proceed by solving the problem by different plane elements, and then comparing the results with the analytical solution.

Choosing plane element instead of various type of ANSYS element is discussed in [8]. The elements, that were used, are listed and explained below.

Different Plane Elements in ANSYS Software [9].

1. Plane 42 is used for two-dimension (2-D) modeling of solid structures as illustrated in Fig. 2. 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: translations in the nodal x and y directions. The element has plasticity, creep, swelling, stress stiffening, large deflection and large strain capabilities.

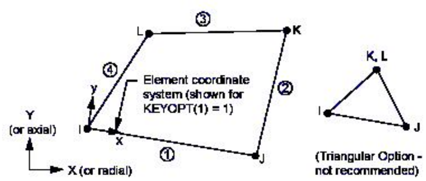

Fig. 2.

2. Plane 82 is a higher order version of the 2-D, four-node element (Plane 42) as illustrated in Fig. 3. It provides more accurate results for mixed (quadrilateral-triangular) automatic meshes and can be used for irregular shapes without much loss of accuracy. The 8-node elements have compatible displacement shapes and are well suited to model curved boundaries.

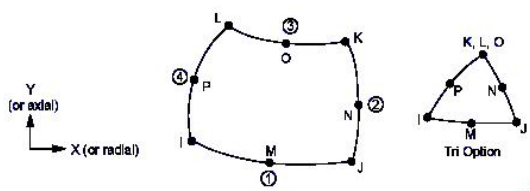

Fig. 3.

3. Plane 182 is used for 2-D modeling of solid structures like in Fig. 2. The element can be used as either a plane element (plane stress, plane strain or generalized plane strain) or an axisymmetric element. It is defined by four nodes, having two degrees of freedom at each node: translations in the nodal x and y directions. The element has plasticity, hyperelasticity, stress stiffening, 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.

4. Plane 183 is a higher order 2-D, 8-node or 6-node element like in Fig. 3. Plane183 has quadratic displacement behavior and is well suited to modeling irregular meshes.

This element is defined by 8 nodes or 6-nodes, having two degrees of freedom at each node: translations in the nodal x and y directions. The element may be used as a plane element (plane stress, plane strain and generalized plane strain) or as an axisymmetric element. This element has plasticity, hyperelasticity, creep, stress stiffening, 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. Initial stress import is supported.

5. Plane 25 is used for 2-D modeling of axisymmetric structures with non-axisymmetric loading as illustrated in Fig. 4. Examples of such loading are bending, shear or torsion. The element is defined by four nodes, having three degrees of freedom per node: translations in the nodal x, y and z direction. For unrotated nodal coordinates, these directions correspond to the radial, axial and tangential directions respectively.

The element is a generalization of the axisymmetric version of Plane 42 the 2-D structural solid element, where the loading need not be axisymmetric.

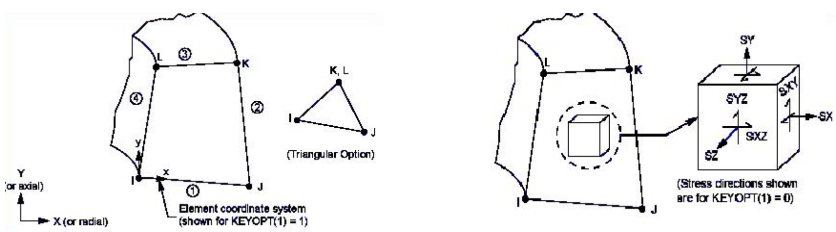

Fig. 4.

6. Plane 83 is used for 2-D modeling of axisymmetric structures with non-axisymmetric loading as illustrated in Fig. 5. Examples of such loading are bending, shear or torsion. The element has three degrees of freedom per node: translations in the nodal x, y and z direction. For unrotated nodal coordinates, these directions correspond to the radial, axial and tangential directions respectively.

This element is a higher order version of the 2-D, four-node element (Plane 25). It provides more accurate results for mixed (quadrilateral-triangular) automatic meshes and can tolerate irregular shapes without much loss of accuracy. The element is also a generalization of the axisymmetric version of Plane 82 the 2-D 8-node structural solid element, where the loading need not be axisymmetric.

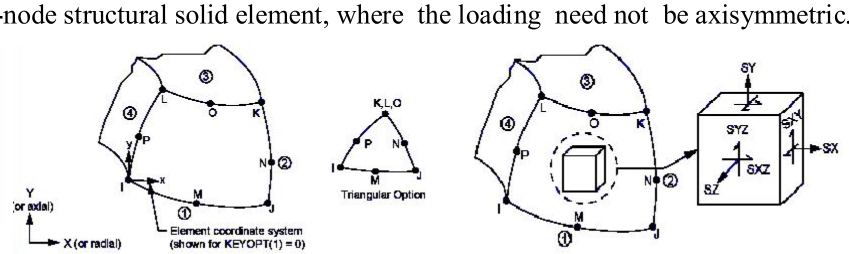

Fig. 5.

Solution of the Problems.

The following parameters are considered for the beam:

$$L = 2m,\ \ P = 50N,\ \ b = 0.01m,\ \ h = 0.2m,\ \ E = 2\times10^8\ N/m^2,\ \ \rho = 1700\,kg/m^3,$$

where h and b are the beam height and thickness respectively and ρ is the beam density.

Substituting this values in (1)–(4), after calculation we obtain the values of maximum displacement, normal and shear stresses and frequency: 0.2 *m*, 3×10^6 *N/m*^{2}, 7.5×10^4 *N/m*^{2} and 2.78148 s^{-1}, respectively.

Modeling in ANSYS software is implemented in two ways for each element:

1. Creation of the elements directly from nodes. In this way 3 samples (1 element, 4 elements and 10 elements) are used.

2. Creation of the region then meshing it. In this way 2 samples (usual and fine meshing) are used.

The analysis is done with three outputs: x and y displacement, normal and shear stresses and first mode frequency.

Results.

Table 1

The maximum displacement (in meters) and error results

№ plane	Displacement									
	1 element	err., %	4 elements	err., %	10 elements	err., %	usual elements	err., %	fine mesh elements	err., %
42	1.51E-01	24	1.98E-01	1	2.01E-01	0.5	2.01E-01	0.5	2.01E-01	0.5
82	1.51E-01	24	1.98E-01	1	2.01E-01	0.5	2.01E-01	0.5	2.01E-01	0.5
182	5.09E-03	97	5.75E-02	71	1.36E-01	32	1.97E-01	1.5	2.01E-01	0.5
183	5.09E-03	97	5.75E-02	71	1.36E-01	32	1.97E-01	1.5	2.01E-01	0.5
25	8.67E-05	100	1.38E-04	100	1.43E-04	100	1.46E-04	100	1.47E-04	100
83	1.65E-05	100	9.64E-05	100	1.09E-04	100	2.67E-04	100	1.47E-04	100

Table 2

The maximum normal stress and error results

№ plane	Normal stress									
	1 element	err., %	4 elements	err., %	10 elements	err., %	usual elements	err., %	fine mesh elements	err., %
42	1.50E+06	50	2.63E+06	12	2.85E+06	5	3.09E+06	3	3.05E+06	1.8
82	41667	98	8.24E+05	72	2.70E+06	10	3.33E+06	11	3.04E+06	1.7
182	41667	98	8.24E+05	72	2.11E+06	70	3.21E+06	107	3.97E+06	32
183	41667	98	8.24E+05	72	2.11E+06	70	6.12E+06	104	4.47E+06	49
25	1193.7	100	3390	100	4814.5	100	9827.9	100	31034	100
83	207.59	100	3326.8	100	5551.5	100	29352	100	60009	100

Table 3

The maximum shear stress and error results

№ plane	Shear stress									
	1 element	err., %	4 elements	err., %	10 elements	err., %	usual elements	err., %	fine mesh elements	err., %
42	50000	33	50000	33	50000	33	78254	4	84146	12
82	95833	27	52941	29	11111	85	70471	6	84425	13
182	95833	27	52941	29	11111	85	69484	7.3	70893	5.5
183	95833	27	52941	29	11111	85	87309	17	85604	14
25	0	100	75	100	185	99	1308	98	5166	93
83	197	100	146	100	435	99	2425	96	6930	90

Table 4

First mode frequency and error results

№ plane	Frequency									
	1 element	err., %	4 elements	err., %	10 elements	err., %	usual elements	err., %	fine mesh elements	err., %
42	2.71E+00	2.5	2.77E+00	0.4	2.75E+00	1.1	2.75E+00	1.1	2.75E+00	1.1
82	3.01E-05	8.2	5.12E+00	84	3.35E+00	20	2.75E+00	1.1	2.75E+00	1.1
182	1.48E+01	47	5.12E+00	84	3.35E+00	20	2.78E+00	0.05	2.75E+00	1.1
183	1.48E+01	47	5.12E+00	84	3.35E+00	20	2.75E+00	1.1	2.75E+00	1.1
25	3.64E-06	100	0.00E+00	100	0.00E+00	100	5.17E-06	100	0.00E+00	100
83	0.00E+00	100	0.00E+00	100	0.00E+00	100	6.29E-06	100	3.01E-05	100

Discussion.

The Table 1, representing the displacement, shows that Plane 25 and 83 are not appropriate for this purpose. Plane 182 and 183 are appropriate only in usual and fine mesh, and Plane 42 and 82 are appropriate for all conditions except in one element mesh. In usual and fine mesh element the Plane 42 and 82 errors are less than 1%.

The Table 2, representing the maximum normal stress, shows that Plane25 and 83 are not appropriate. Plane 182 and 183 only at fine mesh give some results that are close to analytical solution although the error is not acceptable. Plane 42 and 82 are appropriate for usual and fine element meshes. In fine mesh element the Plane 42 and 82 errors are less than 2%.

The Table 3, representing the maximum shear stress, shows that Plane 25 and 83 are not appropriate. Other Plane element (182, 183, 42 and 82) are appropriate only in usual and fine element meshes. There is about 5% error only in Plane 42 and 82 at usual mesh.

The Table 4, representing the first mode frequency, shows that Plane 25 and 83 are not appropriate. Plane 82, 182 and 183 are appropriate only in usual and fine meshes (error is less that 2%). Plane 42 is appropriate for all conditions.

Conclusion. The results show that the analysis of the simple beam bending with Plane 25 and 83 is not recommended in all cases. For displacement in usual and fine mesh, Plane 42, 82, 182 and 183 results are close to analytical solution, but in some problems fine mesh couldn't be applied and usual mesh is considered for use, so in this case Plane 42 and 82 are better to use. By the same reason as above Plane 42 and 82 are suggested for normal and shear stresses. Frequency results shows, that there is no difference between Plane 42, 82, 182 and 183, so each of them is suitable for use.

In some mixed problems or problems where all the data must be calculated in one procedure, Plane 42 or Plane 82 is recommended and better for use to solve plane problems.

Special thanks to my adviser and supervisor Prof. Karen Ghazarian and Dr. Haikaz Yeghiazaryan for their help.

Received 09.04.2010

REFERENCES

1. **Gupta K.K. and Meek J.L.** International Journal for Numerical Methods in Engineering, 1996, v. 39, p. 14.
2. **Singiresu S.R.** The Finite Element Method in Engineering. Fourth ed. Elsevier Science & Technology Books, 2004, 658 p.
3. **Cervera M.** Computer Methods in Applied Mechanics and Engineering, 2008, v. 197, p. 16.
4. **Popov E.P.** Engineering Mechanics of Solids. First ed. New Jersy: Prentice-Hall, 1990, 760 p.
5. **Beer F.P. and Johnson E.R.** Mechanics of Material. 2nd ed. NY: McGraw-Hill, 1925, 532 p.
6. **Timoshenko S.P. and Goodier J.N.** Theory of Elasticity. 3rd ed. NY: McGraw-Hill, 1970, 506 p.
7. **Timoshenko S.P.** Vibration Problems in Engineering. 2nd ed. NY: D. Van Nostrand Company, INC, 1937, 465 p.
8. **Yazdizadeh B. and Yeghiazaryan H.** Polytechnic University Journal (submited). Yer., 2010, p. 9.
9. **Workbench A.** ANSYS Workbench Help. NY, 2007.

Proc. of the Yerevan State Univ. Phys. and Mathem. Sci., 2010, № 3, p. 44–50.

Բ. Յազդիզադե

Կառուցվածքների մեխանիկայի վերջավոր տարրերի ծրագրում օգտագործվող տարբեր հարթ մոդելների համեմատությունը

Վերջավոր տարրերի ծրագրում մեխանիկայի հարթ խնդիրների լուծման ընթացքում կիրառվում են մի քանի տարբեր բազային տարրեր: ANSYS ծրագրում, որը վերջավոր տարրերի լավագույն ծրագրերից մեկն է, շուրջ վեց տեսակի տարր կա: Ծռման պարզ խնդրում մենք դիտարկում ենք տարբեր հարթ մոդելներ և ներկայացնում ստացված արդյունքների համեմատությունը՝ այն տարրերի տեսակը բացահայտելու նպատակով, որոնք կիրառելի են տվյալ խնդրի լուծման համար: Արդյունքների համեմատությունը ցույց է տալիս, որ 42 և 82 հարթ մոդելներն ավելի կիրառելի են, բացի այդ, սովորական բջիջներով 42-րդ հարթ մոդելի արդյունքներն ավելի մոտ են տվյալ խնդրին, քան մանր բջիջներով 82-րդ մոդելինը:

Б. Яздизаде.

Сравнение различных плоских моделей, используемых в программе конечных элементов механики конструкций

При решении плоских задач механики в программе конечных элементов используются несколько различных базовых элементов. В программе ANSYS, в одной из лучших программ конечных элементов, имеется около шести типов элементов. В простой задаче изгиба мы рассматриваем различные плоские модели и проводим сравнение полученных результатов с целью выявления типов элементов, которые применимы к решению данной задачи. Сравнение результатов показывает, что плоские модели 42 и 82 более приемлемы, а также что плоская модель 42 с обычной ячейкой дает лучшие результаты в данной задаче, чем плоская модель 82 с мелкой ячейкой.