한국복합신소재구조학회 논문집 제4권, 제1호, 2013년 03월
J. Korean Soc. Adv. Comp. Struc. Vol. 4, No. 1, pp. 1-8, March 2013
DOI http://dx.doi.org/10.11004/kosacs.2013.4.1.001
ISSN 2093-5145(Print)
ISSN 2288-0232(Online)

Bar와 Beam 구조물의 기본적인 유한요소 모델의 수치해석

류용희[1] · 주부석[1] · 정우영[2] · Suchart Limkatanyu[3]

노스캐롤라이나주립대학교 토목공학과 박사[1], 강릉원주대학교 토목공학과 교수[2]
[3]Associate Professor, Department of Civil Engineering, Faculty of Engineering, Prince of Songkla University, Thailand

Numerical Evaluation of Fundamental Finite Element Models in Bar and Beam Structures

Ryu, Yong-Hee[1] · Ju, Bu-Seog[1] · Jung, Woo-Young[2] · Limkatanyu, Suchart[3]

[1]Ph. D., Department of Civil Engineering, North Carolina State University, Raleigh, NC, USA
[2]Professor, Department of Civil Engineering, Gangneung-WonJu National University, GangNeung, South Korea
[3]Associate Professor, Department of Civil Engineering, Faculty of Engineering, Prince of Songkla University, Thailand

Abstract: The finite element analysis (FEA) is a numerical technique to find solutions of field problems. A field problem is approximated by differential equations or integral expressions. In a finite element, the field quantity is allowed to have a simple spatial variation in terms of linear or polynomial functions. This paper represents a review and an accuracy-study of the finite element method comparing the FEA results with the exact solution. The exact solutions were calculated by solid mechanics and FEA using matrix stiffness method. For this study, simple bar and cantilever models were considered to evaluate four types of basic elements - constant strain triangle (CST), linear strain triangle (LST), bi-linear-rectangle(Q4),and quadratic-rectangle(Q8). The bar model was subjected to uniaxial loading whereas in case of the cantilever model moment loading was used. In the uniaxial loading case, all basic element results of the displacement and stress in x-direction agreed well with the exact solutions. In the moment loading case, the displacement in y-direction using LST and Q8 elements were acceptable compared to the exact solution, but CST and Q4 elements had to be improved by the mesh refinement.

Key Words: finite element analysis; matrix stiffness method; accuracy-study; constant strain triangle; linear strain triangle; bilinear rectangle; quadratic rectangle

1. Introduction

The finite element analysis (FEA) has been widely used and has increased the interest in the accuracy-study of existing basic elements. This approach is originally related to the matrix analysis, and although the matrix analysis can be used for frame structures, FEA can be used to analyze surface structures. Mathematically, this method is to find a numerical solution of field problems using different types of elements for a given loading condition. Courant (1943) introduced the variational methods using triangle elements based on piece wise linear approximations. This was the first appearance of FE technique. Olson and Bearden (1979) studied about the convergence using the 18 degree of freedom flat triangular shell elements of constant strain triangle (CST) and linear strain triangle (LST). The following year, Robinson (1980) introduced a four-node quadrilateral membrane element (Q4) and an eight-node membrane element (Q8) adding a rotational degree of freedom at each node associated with a moment. Sze et al. (1992)

주요어: finite element analysis, matrix stiffness method

Corresponding author: Jung, Woo-Young
Department of Civil Engineering, Gangneung-Wonju National University, 7 Jukhun-gil, Gangneung, South Korea.
Tel: +82-033-640-2421, Fax: +82-033-646-1391 , E-mail:woojung@gwnu.ac.kr

투고일: 2013년 1월 10일 / 수정일: 2013년 2월 7일 / 게재확정일: 2013년 2월 28일

studied about the accuracy of quadrilateral elements such as Q4 and Q8 elements compared to the exact solution. In their studies, a field quantity is allowed to have only a simple spatial variation described by linear terms such as x or y as well as polynomial terms such as x^2,xy, and y^2. The actual variation in the region spanned by an element is certainly more complicated, so finite element method (FEM) provides an approximate solution, which means the exact stiffness in elements cannot be calculated. It shows a fundamental difference between solid mechanics' and FEA solutions. As a result, the evaluation of FEA using basic elements is necessary for the accurate analysis. This paper examines the accuracy of FEA using commonly used basic elements－CST, LST, Q4, and Q8 by comparing FEA results with an exact solution referred to solid mechanics. Practice models were subjected to uniaxial or moment loadings, and solid mechanics, matrix stiffness method, and FEM using basice lements were used to find the solutions for each case. There view of basic concepts and shape functions for each approach were conducted to develop stiffness matrix for the matrix stiffness method and FEM. The FEA performed in MATLAB, and the FEA results were compared with the exact solution. The structure model was improved by the mesh refinement if the analysis result was not acceptable compared to the exact solution.

2. Numerical Methods

2.1 Solution of Solid Mechanics

In order to calculate the exact solution for uniaxial loading case, we used a bar model which was subjected to uniformly distributed loads as shown in Fig. 1. Bernoulli Euler beam theory was used in the approach with the assumptions as: 1) the beam is initially straight, unstressed and symmetric, and long and slender, 2) plane sections remain plane and perpendicular to the neutral axis during bending, 3) Deflections are small, 3) the cross section needs to have a vertical axis of symmetry. 4) material is linearly elastic, isotropic, and homogeneous, 5) the beam has no twisting, 5) proportional limit is not exceeded. Having those assumptions, the solutions for the displacement and stress can be calculated by Equations (1) and (2):

$$\delta = \frac{wx}{E} \tag{1}$$

$$\sigma_x = w \tag{2}$$

Where, w is the applied stress; E is the Modulus of Elasticity. For bending moment loading case, a cantilever beam is subjected to the bending moment as shown in Fig. 2. The same assumptions were made as the uniaxial loading problem for this case. The exact solutions for the displacement and stress can be calculated by the solid mechanics' Eq. (3)(4):

$$\delta_y = \frac{Mx^2}{2EI} \tag{3}$$

$$\sigma_x = \frac{M}{I} y \tag{4}$$

Where, I is the moment of inertia calculated by ; t is thickness; h is height; y is the distance from the neutral axis of the cross section.

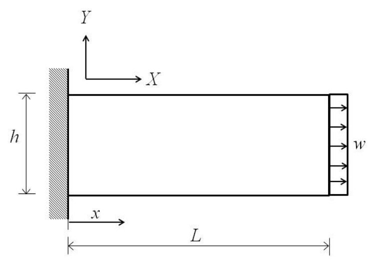

Fig. 1 Bar Model

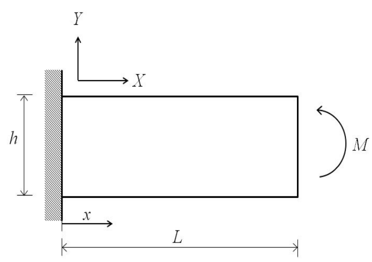

Fig. 2 Cantilever Model

2.2 Solution of Matrix Stiffness Method

The linear bar element is defined as straight prismatic members in one-dimensional elements for the uniaxial or lateral deformation as shown in Fig. 3. This element has a node at each end, and each node has one degree of freedom (DOF) for translations. In this problem, the assumptions were made as: 1) the material properties are not affected by applied forces or moment loading 2) geometric nonlinearity is not considered, 3) material remains linearly elastic, isotropic, and homogeneous. The shape functions N for the bar elements can be defined by Eq. (5):

$$N1 = \frac{x2 - x}{L}\ ,\ N2 = \frac{x - x1}{L} \tag{5}$$

The strain matrix (B) can be calculated by Eq. (6):

$$(B) = \frac{d}{dx}(N) \tag{6}$$

The stiffness matrix [K] can be calculated by Eq. (7):

$$[K]\int_{Element}(B)^T EA(B)dx \tag{7}$$

Displacements $\{u\}$ at each node are calculated by the inversed stiffness matrix times applied forces. The stress $\{\sigma\}$ can be obtained for each element from Eq.(8):

$$\sigma = E(B)u \tag{8}$$

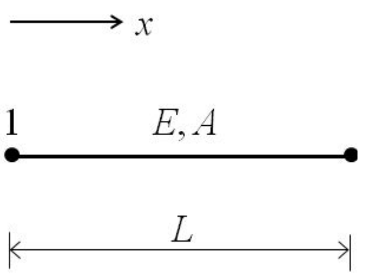

Fig. 3 Linear Bar Element

The linear beam element is characterized as straight prismatic members in two-dimensional elements for the lateral translation and rotation deformations as shown in Fig. 4. This element has a node at each end, and each node has two DOFs.

The same assumptions were made as the bar element problem for this case. The shape functions for the elements are given by Eq. (9):

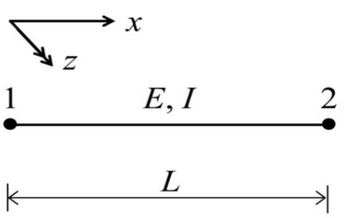

Fig. 4 Linear Beam Element

$$N1 = 1 - \frac{3x^2}{L^2} + \frac{2x^\delta}{L^\delta}\ ,\ N2 = 1 - \frac{2x^2}{L} + \frac{x^\delta}{L^\delta} \tag{9}$$

$$N3 = \frac{3x^2}{L^2} - \frac{2x^\delta}{L^\delta}\ ,\ N4 = -\frac{x^2}{L^2} + \frac{x^\delta}{L^\delta}$$

The strain (B) matrix can be calculated by Eq. (10):

$$(B) = \frac{d^2}{dx^2}(N) \tag{10}$$

The stiffness matrix [K] can be calculated by Eq.(11):

$$[K]\int_{Element}(B)^T EI(B)dx \tag{11}$$

Displacements $\{u\}$ at each node are calculated by the inversed stiffness matrix times applied forces. The stress $\{\sigma\}$ can be calculated by the given Eq. (8).

2.3 Solution of Finite Element Method using Basic Elements

Matrix stiffness method can only be used for framed structures. Finite element analysis using basic elements such as constant strain triangle (CST), linear strain triangle (LST), a four-node quadrilateral membrane element (Q4), and an eight-node membrane element (Q8) is originated as an extension of matrix analysis to surface structures. Finite element method (FEM) is derived by work-energy principles from assumed displacement or stress functions, and FEA result generally produces approximate results. Three conditions such as the compatibility, equilibrium, and boundary must be satisfied in this analysis. One of the basic models is the CST element which is a plane triangle element. In this model, a field quantity is allowed to have only a simple spatial variation described by linear terms such as x or y as shown in Fig. 5.

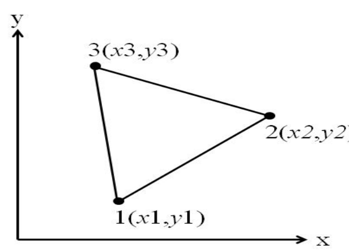

Fig. 5 Constant Strain Triangle(CST) Element

In this approximate solution, a linear displacement fields in stress analysis yields a constant strain field throughout the element's domain. The shape functions for this element are given as Eq. (12):

$$N1=\frac{(y3-y2)(x-x2)-(x3-x2)(y-y2)}{(y3-y2)(x1-x2)-(x3-x2)(y1-y2)}$$
$$N1=\frac{(y2-y1)(x-x1)-(x2-x1)(y-y1)}{(y2-y1)(x3-x1)-(x2-x1)(y3-y1)}$$
$$N1=\frac{(y1-y3)(x-x3)-(x1-x3)(y-y3)}{(y1-y3)(x2-x3)-(x1-x3)(y2-y3)} \quad (12)$$

The strain matrix $[\mathrm{B}]_{3\times 6}$ can be calculated by Eq. (13):

$$[B]=\begin{bmatrix} \frac{d}{dx}Ni & 0 \\ 0 & \frac{d}{dx}Ni \ ... \\ \frac{d}{dy}Ni & \frac{d}{dx}Ni \end{bmatrix} \quad (13)$$

Where, i is the number of the shape function. For the bar and cantilever models in two-dimensional problem, the plane stress condition was considered because the thickness of the model was assumed to be small compared to the other dimensions. In this case, the equation for plane stress can be given as Eq. (14).

$$[D]=\frac{\delta}{1-v^2}\begin{bmatrix} 1 & v & 0 \\ v & 1 & 0 \\ 0 & 0 & \frac{1-v}{2} \end{bmatrix} \quad (14)$$

Where, v is Poisson's ratio. The approximation of stiffness matrix [K] is given by Eq. (15):

$$[K]=t\iint_{Element}[B]^T[D][B]dxdy \quad (15)$$

Displacements $\{u\}$ at each node can be calculated by the inversed stiffness matrix times applied forces, and the stress vector $\{\sigma\}$ is obtained for each element by Eq. (16):

$$\sigma=[D][B]u \quad (16)$$

The second basic element is LST element which is a quadratic triangle element. This element has side nodes in addition to vertex nodes such that a field quantity is allowed to have only a spatial variation described by polynomial terms such as x2,xy, and y2 as shown in Fig. 6.

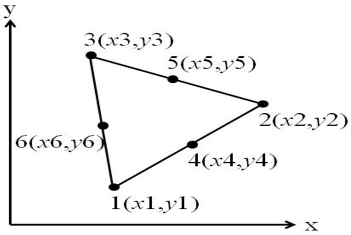

Fig. 6 Linear Strain Triangle(LST) Element

The shape functions for the elements are given by Eq. (17):

$$N1=\frac{(x23(y-y3)-y23(x-x3))(x46(y-y6)-y46(y-y6))}{(x23y13-y23x13)(x46y16-y46y16)}$$
$$N2=\frac{(x31(y-y1)-y31(x-x1))(x54(y-y4)-y54(y-y4))}{(x31y21-y31x21)(x54y24-y54y24)}$$
$$N3=\frac{(x21(y-y1)-y21(x-x1))(x56(y-y6)-y56(y-y6))}{(x21y31-y21x31)(x56y36-y56y36)} \quad (17)$$
$$N4=\frac{(x31(y-y1)-y31(x-x1))(x23(y-y3)-y23(y-y3))}{(x31y41-y31x41)(x56y36-y56y36)}$$
$$N5=\frac{(x31(y-y1)-y31(x-x11))(x21(y-y1)-y21(y-y1))}{(x31y51-y31x51)(x21y51-y21y51)}$$

Where, $x_{ij}=x_i-x_j$; $y_{ij}=y_i-y_j$. The strain matrix $[\mathrm{B}]_{3\times 12}$ can be calculated by Eq. (13), and the approximation of stiffness matrix [K] can be calculated by Eq. (15). The stress vector $\{\sigma\}$ can be obtained for each element using Eq. (14) and Eq. (16). The third element is the Q4 element which is a bilinear rectangle element. This element has a four node plane element with eight DOFs in which the field quantity is allowed to have only a simple spatial variation described by linear terms such as x or y as shown in Fig. 7.

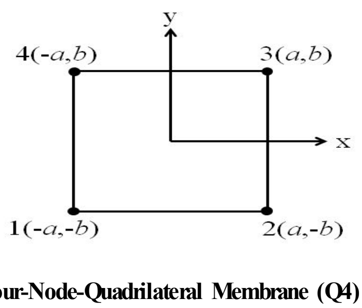

Fig. 7 Four-Node-Quadrilateral Membrane (Q4) Element

In this approximate solution, a linear displacement fields in stress analysis yields a constant strain field throughout the element's domain. The shape functions for the element are given as Eq. (18):

$$N1 = \frac{1}{4ab}(a-x)(b-y),\ N2 = \frac{1}{4ab}(a+x)(b-y)$$
$$N3 = \frac{1}{4ab}(a+x)(b+y),\ N4 = \frac{1}{4ab}(a-x)(b+y) \quad (18)$$

The strain matrix [B]3X8 can be calculated by Eq. (13), and the approximation of stiffness matrix [K] can be calculated by Eq. (15). The stress vector $\{\sigma\}$ can be obtained for each element using Eq. (14) and Eq. (16). The last element is Q8 element which is a quadratic rectangle element. This element is obtained by adding side nodes as LST element such that a field quantity is allowed to have only a spatial variation described by polynomial terms such as x^2, xy, and y^2 as shown in Fig. 8.

This element is allowed to have curved shapes, and the shape functions for the element are described by Eq. (19):

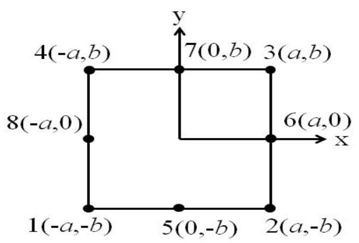

Fig. 8 Eight-Node Membrane (Q8) Element

$$N1 = \frac{1}{4a^2b^2}\{\ (b-y)bx^2 + abxy + (a-x)ay^2 - a^2b^2\ \}$$

$$N2 = \frac{1}{4a^2b^2}\{\ (b-y)bx^2 - abxy + (a+x)ay^2 - a^2b^2\ \}$$

$$N3 = \frac{1}{4a^2b^2}\{\ (b+y)bx^2 + abxy + (a+x)ay^2 - a^2b^2\ \}$$

$$N4 = \frac{1}{4a^2b^2}\{\ (b+y)bx^2 - abxy + (a-x)ay^2 - a^2b^2\ \} \quad (19)$$

$$N5 = \frac{1}{2a^2b^2}\{(y-b)bx^2 - a^2by + a^2b^2\}$$

$$N6 = \frac{1}{2a^2b^2}\{-(x+a)bx^2 - ab^2y + a^2b^2\}$$

$$N7 = \frac{1}{2a^2b^2}\{-(y+b)bx^2 + a^2by + a^2b^2\}$$

$$N8 = \frac{1}{2a^2b^2}\{(x-a)ax^2 - ab^2x + a^2b^2\}$$

The same process such as the Q4 element is taken also for Q8 elements to calculate displacement and stress for each node. The strain matrix $[B]_{3\times16}$ can be calculated by Eq. (13), and the approximation of stiffness matrix [K] can be calculated by Eq. (15). The stress vector $\{\sigma\}$ can be obtained for each element using Eq. (14) and Eq. (16).

3. Evaluation of Finite Eelment Analysis

3.1 An Uniaxial Loading Case

In order to evaluate FEA for the uniaxial loading case, the bar model was used as shown in Fig. 1. This model has dimensions as h = 0.25m, L = 0.5m, and t = 0.025m. The plane stress condition was considered because the thickness of the model is to be small compared to the other dimensions in two-dimensional analysis. For the material properties, the Modulus of Elasticity E was used as 150,000 Pa and the Poisson's ratio v as 0.3. The uniformly distributed uniaxial load w was applied as 3000 kN/m^2in X-direction as shown in Figure 1. From the Eq. (1), the displacement at middle and end locations were calculated as 0.005 m and 0.01 m, respectively. Fig. 9 shows the bar model for the matrix stiffness method. One DOF is used for the translation in X-direction. Fig. 10 shows the FE model using CST elements. Four plane triangle elements were used for this analysis. The field quantity is allowed to have linear displacements in X and Y directions. Next, two quadratic triangle elements were used for FEA using LST elements as shown in Fig. 11.

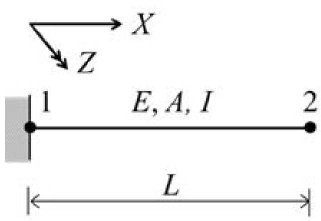

Fig. 9 Bar and Beam Model

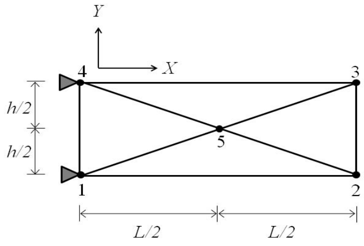

Fig. 10 CST Element Model

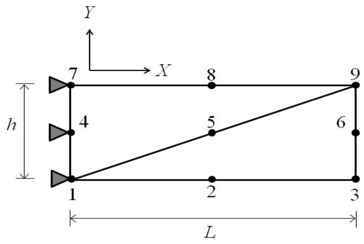

Fig. 11 LST Element Model

This element is allowed to be an arbitrary shaped quadrilateral. Fig. 12 shows the FE model using Q4 elements, and two plane bilinear rectangle elements were used for this analysis. Next, one quadratic rectangle element was used for FEA using Q8 elements to calculate displacements and stresses as shown Fig. 13.

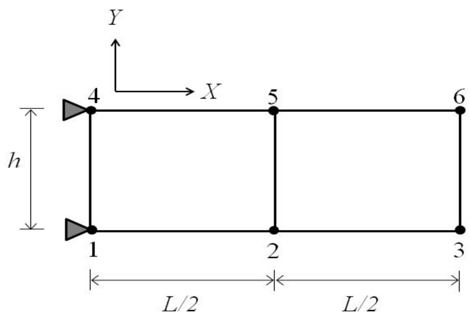

Fig. 12 Q4 Element Model

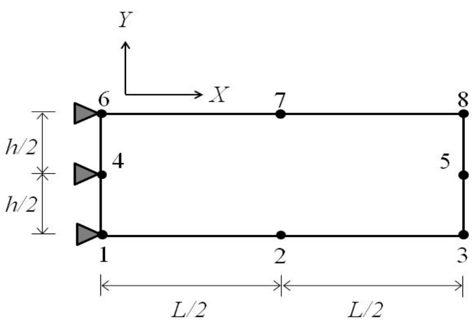

Fig. 13 The Q8 Element Model

Fig. 14 shows displacement results conducted by MATLAB using the solid mechanics' equation, matrix stiffness method, and FEM using basic elements - CST, LST, Q4, and Q8. There was no difference between matrix stiffness method and solid mechanics' results because matrix stiffness method is the solution to use the exact stiffness of the member. However, there were errors using FEA with basic elements. Fig. 15 shows percentage errors at middle and end locations from the analyses. Because FEM is derived by work-energy principles from assumed displacement or stress functions, there are about 8% error using CST element and less than 4% error using LST, Q4, and Q8 elements at middle location compared to the exact result. Furthermore, Fig. 16 and Fig. 17 show the results of stresses in x and y-directions. Based on the equilibrium condition, the stress at end location is the same at the applied stress 3000 kN/m^2.Fig. 16 shows the stress σ_x using FEA using basic elements agree well with the exact solution, but analysis results using CST, LST, Q4, and Q8 elements show about 6% of the stress σ_y compared to the stress σ_x even though stress in y-direction is not existed in Fig. 17.

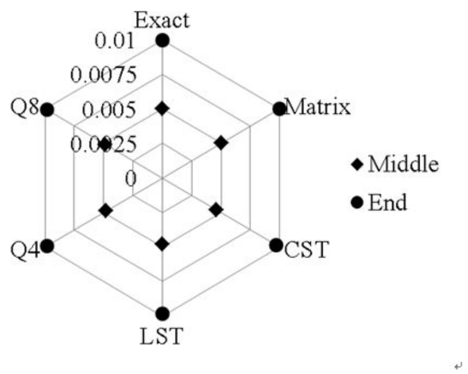

Fig. 14 Displacements (m) in X-Direction

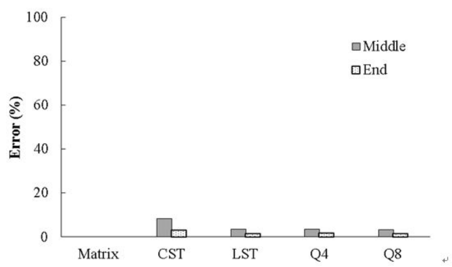

Fig. 15 Error in Percentage (%) of Displacements in X-Direction

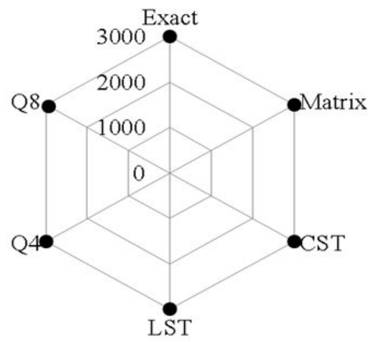

Fig. 16 Stress σ_x(kN/m^2)

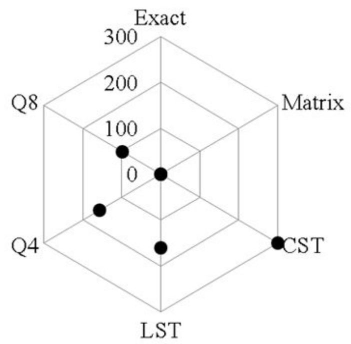

Fig. 17 Stress σ_y(kN/m^2)

3.2 A Bending Moment Loading Case

For the evaluation of FEA for bending moment loading case, the cantilever model was considered for analysis as shown in Fig. 2. This model is chosen to have dimensions as h = 0.25 m, L = 0.5 m, and t = 0.025 m. The plane stress condition was considered, and the material properties were used as E = 384,000 Pa and v = 0.3. The moment load was applied as M = 1 kN-m at the end of the beam as shown in Fig. 2. The same models were used for matrix analysis and FEA using basic elements. FEA results were compared to an exact solution referred to solid Mechanics for the uniaxial loading case. Fig. 18 shows displacement results conducted by MATLAB using the solid Mechanics' equation, matrix stiffness method and FEM using basic elements. From the Eq. (3), the displacement at middle and end locations were calculated as 0.0025 m and 0.01 m, respectively. Solid Mechanics' equation and the matrix analysis yield the same result, but there were significant errors using FEA with basic elements. Fig. 19 shows percentage errors of displacements in Y-direction at middle and end locations. There are about 70 % error using CST element and about 33% error using Q4 element at end location compared to the exact result. The displacements at the end using CST and Q4 elements were 0.003 m and 0.007 m, respectively. It should be noted that in FEA, the displacement or stress functions is assumed. As a result, good shape functions provide a better solution to anticipate structure behavior. For the beam bending case, the structure has the quadratic deformation instead of the linear deformation. Therefore, an engineer should choose LST and Q8 elements if the quadratic behavior is expected in a structure, and the linear triangle and bilinear rectangle elements must be improved by the mesh refinement if used.

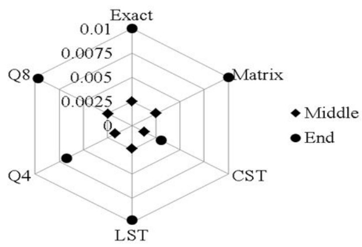

Fig. 18 Displacements (m) in Y-Direction

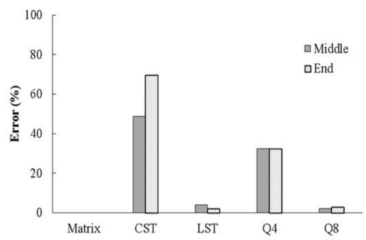

Fig. 19 Error in Percentage (%) of Displacements in Y-Direction

3.3 Consideration of An Improved Model

The mesh refinement can improve a structure model using CST and Q4 elements to have quadratic deformation in X and Y directions. Finer mesh was used to evaluate the improvement of analysis results. Having more degree of freedoms (DOFs) in the model, the percentage error reduced significantly as shown in Fig. 20. Therefore, to find the accurate solution using FEA, the engineer should check two conditions; 1) the shape function of the element has the suitable deformed shape to anticipate the structure behaviors 2) enough DOFs were used for analysis to consider appropriate deformations of the model.

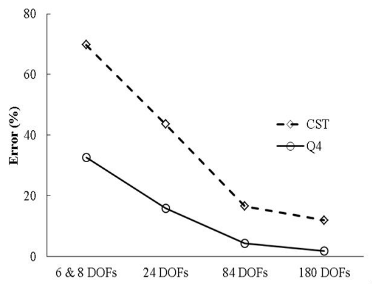

Fig. 20 Percentage Error fo CST and Q4 Elements

4. Conclusion

In this paper, the principle concept of finite element analysis (FEA) was reviewed, and FEA using basic elements such as constant strain triangle (CST), linear strain triangle (LST), a four-node quadrilateral membrane element (Q4), and an eight-node membrane element (Q8) were evaluated by comparing the FEA results with the exact answer. For the application, practice models were subjected to uniaxial or bending moment loadings, and then the solid mechanics, matrix stiffness method, and FEM using basic elements were used to find the solutions for each case. In the uniaxial loading case, all results of displacements and stresses in x-direction agreed well compared to the exact solution, but the stress in y-direction were found even though stress in y-direction was not existed. This error can cause a significant difference in analysis. In bending moment loading case, FEA using LST and Q8 elements provided the acceptable results of displacements in Y-direction, but the errors were significant when using CST and Q4 elements compared to the exact answer. The mesh refinement was conducted for those elements by increasing element numbers. The percentage error reduced significantly as finer mesh improved the model to have the quadratic behavior. In this paper, we examined the accuracy of FEA using commonly used basic elements - CST, LST, Q4, and Q8 by comparing FEA results with an exact solution referred to solid mechanics. FEM is a numerical technique to find approximate solutions. Therefore, an engineer must check the shape function of the element such that the element has the suitable deformed shape or the enough elements are considered to anticipate the structure behaviors.

Acknowledgements

This work was supported by the Natural Disaster Prevention Research Institute of Kangnung-Wonju National University, South Korea.

References

Cook, R. D. (2002). Concepts and applications of finite element analysis. 4th ed., Wiley, NewYork, USA.

Courant, R. L. (1943). "Vibrational Methods for the Solutionof Problems of Equilibrium and Vibration.", *Bulletin of the American Mathematical Society*, Vol. 49, pp. 1-23.

Iqbal, R. (2007). "Development of a finiteelement program incorporating advanced element types." *M.S. Thesis*, Osmania University Hyderabad, India

Kassimali, A. (2011). Matrix Analysis of Structures, Pacific Grove, Cengage Learning, CA, USA.

Kattan, P. I. (2007). MATLAB guide to finite elements: an interactive approach. 2nd ed. NewYork :Springer.

Melosh, R. J. (1985). "Self-qualification of finite element analysis results by polynomial extrapolation." *Finite Elements in Analysis and Design,* Vol. 1, pp. 49-60.

Olson, M. D. and Bearden, T. W. (1979). A simple flat triangular shell element revisited, *Int. J. Numer. Meth. Engng.,* Vol. 2, pp. 51-68.

Robinson, J. (1980). Four-node quadrilateral stress membrane element with rotational stiffness, *Int. J. Numer. Meth. Engng.*, Vol. 3, pp. 1567-1569.

Sze, K. Y., Wanji, C., and Cheung, Y. K. (1992). "An efficient quadrilateral plane element with drilling degrees of freedom using orthogonal stress modes." *Comp. Struct.,* Vol. 42, No. 5, pp. 695-705.

Timoshenko, S. P. (1953). History of Strength of Materials, McGraw-Hill Book Co. New York.