International Academic Institute for Science and Technology

International Academic Journal of Science and Engineering
Vol. 4, No. 2, 2017, pp. 226-241.
ISSN 2454-3896

International Academic Journal of Science and Engineering

www.iaiest.com

Numerical modelling of cross-section variations of metal structures

Masoud Mahmoudabadi[a], Seyed Mohammad Reza Hasani[b]

[a]*PhD., Faculty of Engineering, Department of Civil Engineering, University of Qom, Iran.*

[b] *Young Researchers and Elite Club, Qom Branch, Islamic Azad University, Qom, Iran.*

Abstract

In this paper, the modelling of variable cross-sections of the members used in metal structures has been done. For this purpose, a four nodes finite element is used. The finite element stiffness matrix for the case of changes during the cross-section is limited to linear gain. Nodal loads equivalent to the weight to the case that the cross-sectional area changes during the linear component is limited and The nodal loads equivalent centrifugal force to form the cross-section is a fixed limit, obtained. To evaluate the accuracy of model, two samples including a rotary bar and a rotary rod has been checked. Conclusions show that modelling with this method gives appropriate results.

Keywords: numerical modeling method, one dimensional four nodes finite element, rotary bar.

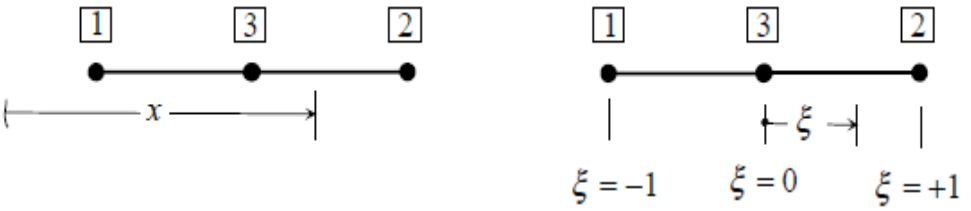
Introduction

Numerical methods are techniques by which mathematical problems are formulated so that they can be solved with arithmetic operations. Although there are many kinds of numerical methods,they have one common characteristics: they invariably involve large numbers of tedious arithmetic calculations. It is little wonder that with the development of fast, efficient digital computers, the role of numerical methods in engineering problem solving has increased dramatically in recent years [1, 2]. Finite element method is one of the most common numerical methods which has been used in a variety of research areas such as seismic design [3], structural stability analysis [4].

one dimensional three nodes finite element

The stiffness matrix of one dimensional three nodes finite element has been presented. This three nodes finite element is shown in figure1 [5].

Figure 1: one dimensional three nodes finite element in x coordinates and ξ dimensionless coordinates



Note that in local number system in reference [6] node 1, 2 and 3 represent the first ,last and middle nodes of this finite element respectively. The node 3 is for crossing the quadratic curve and is called internal node.

The symbol x_i is used to demonstrate the coordinates of any node,which i=1,2,3 is the number of node. In addition, the nodes displacement vector in local coordinates is $q = [q_1, q_2, q_3]^T$,which q_1, q_2 and q are the displacement of 1,2 and 3 nodes. Following equation represents a coordinate transformation from the x coordinates system to the dimensionless coordinates system ,ξ,as follows:

(1)

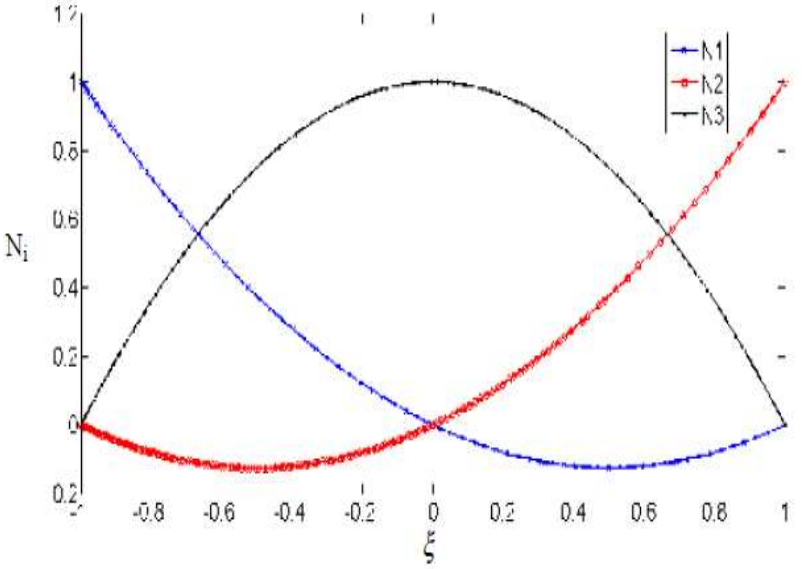
Where the l_e is the length of the finite element.From Eq. (1),it follows that ξ for the 1,2 and 3 nodes.in the dimensionless coordinates system, ξ , the quadratic shape functions, , and are defined with ξ,where:

(3)

[N]

The variations of quadratic shape functions with ξ is shown in figure 2.

Figure 2: shape functions



It follows from this figure that the value of in node 1 is 1 and in node 2 or 3 is 0.It is the same for the other shape functions, N

The displacement of internal plots of the finite element with the displacement of the nodes is:

Which $N = [N$] is a 1×3 shape function vector,and $q = [q_1, q_2, q_3]^T$ is a nodes displacement vector.in node ١ , and $N_2 = N_3 = 0$,so $u = q_1$. Similarly, in node 2 ,and in node 3, . Eq. (6) shows a quadratic interpolation function that crosses from nodal displacement

The strain of each finite element plot is equal to [7]:

— —— —— —[— — —].

It is follow from Eq. (2) to Eq. (4)that:

—[— ——— ——— ξ].

Where:

—[— ——— ——— ξ]

Using the Hooks low, it follows that:

Since are the quadratic shape functions, B in Eq. (9) is a linear function of ξ.It means that strain and stress in finite element vary linear.

Assume that we have u,ε and $\left(\frac{e}{}\right)d\xi$ and with constant ,using the Eq. (11) and Eq. (12) to obtain stiffness matrix and body force of the finite element .The process of calculations is given in reference [5].

$$\text{——}\int [\boldsymbol{B}\ \boldsymbol{B}]d\xi \quad \text{——}\begin{bmatrix} & & & 8 \end{bmatrix}$$

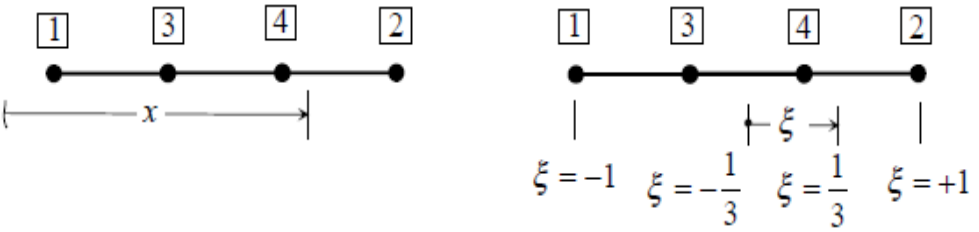
$$\text{——}\int \begin{matrix} {}^{1}/_{6} \\ {}^{1}/_{6} \\ \left({}^{2}/_{3}\right) \end{matrix}$$

Note that $\boldsymbol{E_e}, \boldsymbol{A_e}, \boldsymbol{l_e}$ and $\boldsymbol{\gamma_e}$ are the modulus of elasticity, cross-section, finite element length and volumetric weight, respectively.

one dimensional four nodes finite element with constant cross-section

in this section, one dimensional four nodes finite element stiffness matrix with constant cross-section, and equivalent nodal forces for body forces are calculated, and is plotted in figure 3. Note that in the local numbering system that used, node1 , 2 , 3 and 4 represent the first , last and middle nodes

Figure 3: one dimensional four nodes finite element in x coordinates and ξ dimensionless coordinates



of this finite element respectively. The nodes 3, 4 are for crossing the cubic curve and called internal nodes.

The symbol x_i is used to demonstrate the coordinates of any node,which i=1,2,3,4 is the number of node. In addition, the nodes displacement vector in local coordinates is $q = [q_1, q_2, q_3]^T$,which q_1, q_2 ,q_3 and q are the displacement of 1,2,3 and 4 nodes. Following equation represents a coordinate transformation from the x coordinates system to the dimensionless coordinates system ,ξ,as follows:

——— ———

Where the l_e is the length of the finite element. From Eq. (13),it follows that ξ – – for the 1,2 , 3 and 4 nodes.in the dimensionless coordinates system, ξ , the cubic shape functions, , ,N_3 and are defined with ξ,where [8]:

$$\text{—} \quad)\left(- \quad \xi\right)\left(-- \quad \xi\right)$$

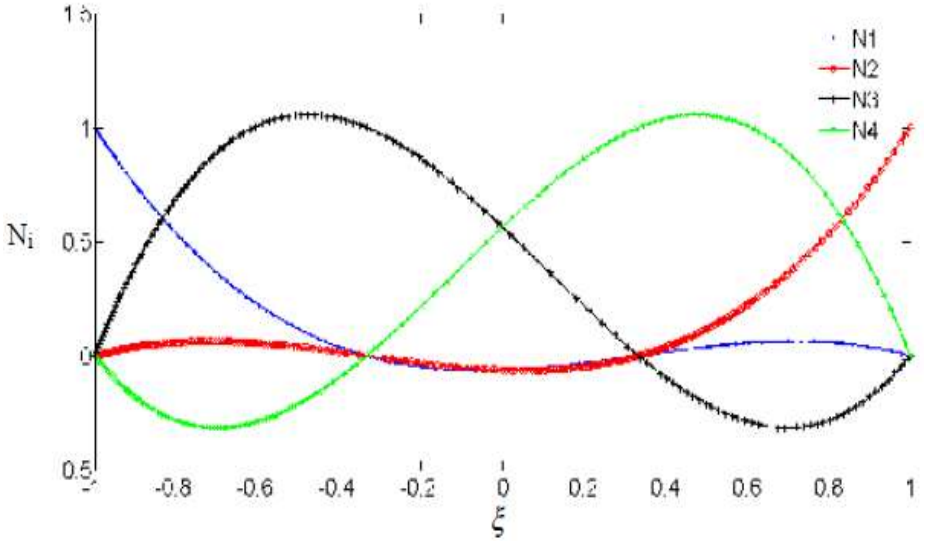
$$\text{—} \quad)\left(- \quad \xi\right)\left(-- \quad \xi\right)$$

$$\text{—} \quad)\left(- \quad \xi\right)$$

$$= [N_1, N_2, N_3, N_4]$$

The variations of cubic shape functions , N_1, N_2 ,N_3 and with ξ is shown in figure 4.

Figure 4: shape functions



It follows from this figure that the value of in node 1 is 1 and in node 2 ,3 and 4 is 0.It is the same for the other shape functions, N

The displacement of internal plots of the finite element with the displacement of the nodes is :

Which $N = [N$] is a shape function vector,and $[q$ $]^T$ is a nodes displacement vector.in node ١ , and ,so . Similarly, in node 2 , in node 3, and in node 4, . Eq. (19) shows a cubic interpolation function that crosses from nodal displacement q ,

The strain of each finite element plot is equal to:

$$— \quad —— \quad —— \quad -\left[—— \quad —— \quad —— \quad ——\right].$$

It is follow from Eq. (14) to Eq. (17) that:

$$-\left[—\left(- \quad -\right), —\left(3 \quad -\right), —\left(3 \quad - \quad 1\right), \quad —\left(3 \quad - \quad 1\right)\right].$$

Where:

$$-\left[-\left(-\qquad -\right),-\left(3\qquad -\right),-\left(3\quad -\quad 1\right),\ -\left(3\quad -\right.\right.$$
$$\left.\left.-1\right)\right]$$

Using the Hooks low, it follows that [8]:

Since $N_i's$ are the cubic shape functions, B in Eq. (22) is a quadratic function of ξ.It means that strain and stress in four nodes finite element vary quadratic.Assume that we have u,ε, σ and $dx = \left(\frac{e}{-}\right) d\xi$ and with constant A_e,using the reference [13] method to obtain stiffness matrix and body force of the finite element . The resultant equations are:

$$---\int\ [\boldsymbol{B}\ \ \boldsymbol{B}]d\xi\quad ---\left[\qquad\qquad\qquad\right]$$

$$---\int \begin{matrix} 1/8 \\ 1/8 \\ 3/8 \\ \left(3/8\right) \end{matrix}$$

Note that in Eq .(24) and Eq .(25) , $\boldsymbol{E_e}, \boldsymbol{A_e}, \boldsymbol{l_e}$ and $\boldsymbol{\gamma_e}$ are the modulus of elasticity, cross-section, finite element length and volumetric weight of four nodes finite element, respectively.

Nodal force equivalent with volumetric force due to bar rotation is calculated using Eq .(26).Note that when lumped mass m rotates about O with constant angular velocity and constant radius r , centrifugal force is necessary to keep this mass on gyrate path .so, volumetric force due to bar rotation for a rotary bar with constant cross-section, and constant density ,ρ,is equal to:

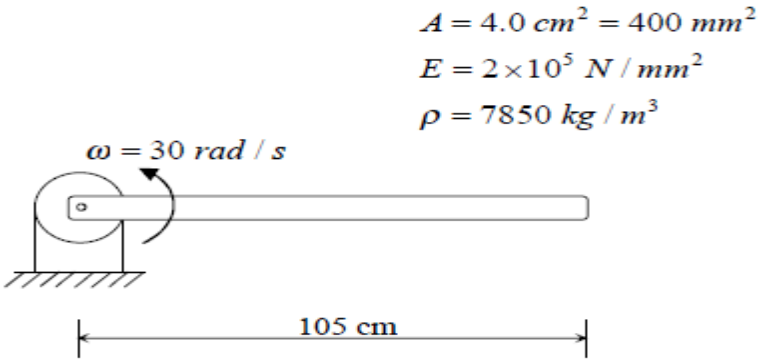
$$---\int \qquad --- \begin{matrix} 2/60 \\ 13/60 \\ 9/60 \\ \left(36/60\right) \end{matrix}$$

In Eq .(26) finite element mass is m and l_e is the length of four nodes finite element.Note that using Eq . (13) , $x = \frac{e}{-}(\xi + 1) + x_1$,and x because node 1 (first node) is assumed to be the center of rotation. So in Eq .(26) ,x , is replaced by $-$).

Rotary bar example

In this section, the example of rotary bar using the two, three nodes finite elements and one ,four nodes finite element is analysed . characteristics of this rotary bar are given in reference [5] but the units are transformed into SI system. Figure 5 shows the rotary bar with finite elements numbering and its nodes. Figure 5-1 shows the rotary bar schema and its characteristics. Figure 5-2 shows the corresponding finite element model composed of two ,three nodes finite elements. This model has five degrees of freedom. Using the Eq .(11) , stiffness matrix of finite element 1 and 2 are:

Figure 5-1:rotary bar with constant angular velocity



Figure(5-2):finite element model composed of two, three nodes finite elements

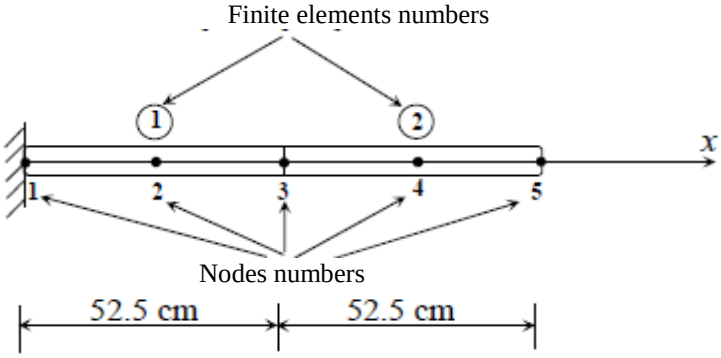
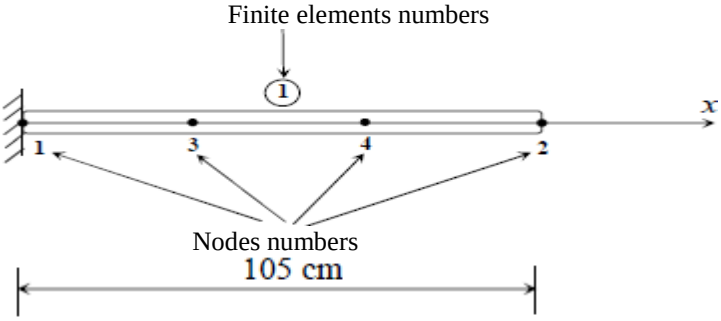


Figure 5-3: finite element model composed of one, four nodes finite element



$$\text{———————}\left[\qquad\qquad 8\right] \quad {}^{N}/_{mm}$$
$$\text{———————}\left[\qquad\qquad 8\right] \quad {}^{N}/_{mm}$$

Using above matrixes ,Stiffness matrix of total system is:

$$\text{———————} \qquad\qquad {}^{N}/_{mm}$$
$$\lfloor \qquad\qquad\qquad\qquad \rfloor$$

Using the equation [9] $f = mr\omega^2$ to calculate the volumetric force f due to centrifugal force .Observe that f is related to r. So it is necessary to calculate the centrifugal force f for each element in the center of that finite element averagely. Therefore, it follows that:

Also it follows from Eq .(12) that:

$$\begin{matrix} {}^{1}/_{6} \\ {}^{1}/_{6} \\ \left({}^{2}/_{3}\right) \end{matrix} \quad \left\{ \qquad \right\}$$

$$\begin{matrix} {}^{1}/_{6} \\ {}^{1}/_{6} \\ \left({}^{2}/_{3}\right) \end{matrix} \quad \left\{ 194\ 73 \right\}$$

Assembling the above equations give:

$$\left(194.73\right)$$

It follows from removing the first row and column in stiffness matrix by Eq .(29) and removing the first row of nodal force vector of total system (Eq .(34))that the equilibrium equation is:

$$\text{———————}\left[\qquad\qquad\qquad\qquad\right]\left\{\begin{matrix} \\ 3 \\ \end{matrix}\right\} = \left\{\qquad\right\}$$

Solving the above equation ,gives the displacement vector, that:

$$\left\{ \quad {}^{3} \right\} = \left\{ \qquad\qquad \right\}$$

It is observed from figure 5-2 that the 2 and 4 nodes are internal nodes for 1,2 finite element respectively. Using the Eq. (10) and nodal displacement of each three nodes finite element ,to calculate the stress of each node in finite element. Results of computation of the stresses in the first, middle and end nodes of 1,2 finite elements is given in Table 1.

Table 1: values of stresses in the first, middle and end nodes of three nodes finite element of rotary bar in Mpa unit

End node	Middle node	First node	Finite element number
2/291	3/408	3/895	1
0/0	1/460	2/921	2

In figure 5-3 the rotary bar is modeled with a four nodes finite element [10].This finite element has 4 degrees of freedom.Because only one ,four nodes finite element ,is used for modeling the rotary bar , the degrees of freedom of finite element is defined in local system.Using the Eq .(24) , the stiffness matrix of four nodes finite element is:

$$\text{———————}\left[\qquad\qquad\qquad\qquad 378 \right] N/mm$$

Using the Eq .(26) to calculated the volumetric force vector due to centrifugal force.so, the volumetric force vector due to centrifugal force for four nodes finite element is:

$$\text{———}\begin{matrix} {}^{2}/_{60} \\ {}^{13}/_{60} \\ {}^{9}/_{60} \\ \left({}^{36}/_{60}\right) \end{matrix} \qquad \begin{matrix} {}^{2}/_{60} \\ {}^{13}/_{60} \\ {}^{9}/_{60} \\ \left({}^{36}/_{60}\right) \end{matrix} \left\{ \qquad \right\}$$

It follows from removing the first row and column in stiffness matrix and removing the first row of nodal force vector, that:

$$\text{———————}\left[\qquad\qquad\qquad 594 \right]\left\{ q_3 \right\} = \left\{ 233 \quad \right\}$$

Solving the above equation ,gives the displacement vector, that:

$$\left\{ q_3 \right\} = \left\{ 6.563 \times 10^{-3} \right\}$$

One dimensional four nodes finite element with variant cross-section

In this part, One dimensional four nodes finite element with variant cross-section and corresponding nodal forces for volumetric forces are calculated [11]. cross-section variant along the finite element assumed to be linear.so if the four nodes finite element cross-section in the first nodes (in ξ

)assumed to be A_1,and in the end node (in assumed to be A_2,in arbitrary plot ξ assumed to be $\left(—\right)A$ $\left(—\right)A_2$.in this case,finite element stiffness matrix is equal to:

$$—\int [\boldsymbol{B}\ \boldsymbol{B}]A$$

Using the B matrix as defined in Eq .(22) and assume that $\left(—\right)A$ $\left(—\right)A$, four nodes finite element stiffness matrix is equal to:

$$—\left(\left[\quad\right]A\ \left[\quad\right]A_2\right)$$

Nodal forces vector due to weight force is equal to:

$$—\int \boldsymbol{N} \begin{matrix} {}^{13}/_{120} & {}^{1}/_{60} \\ {}^{1}/ & {}^{13}/_{120} \\ {}^{3}/_{10} & {}^{3}/_{40} \\ {}^{3}/_{40} & {}^{3}/_{10} \end{matrix}$$

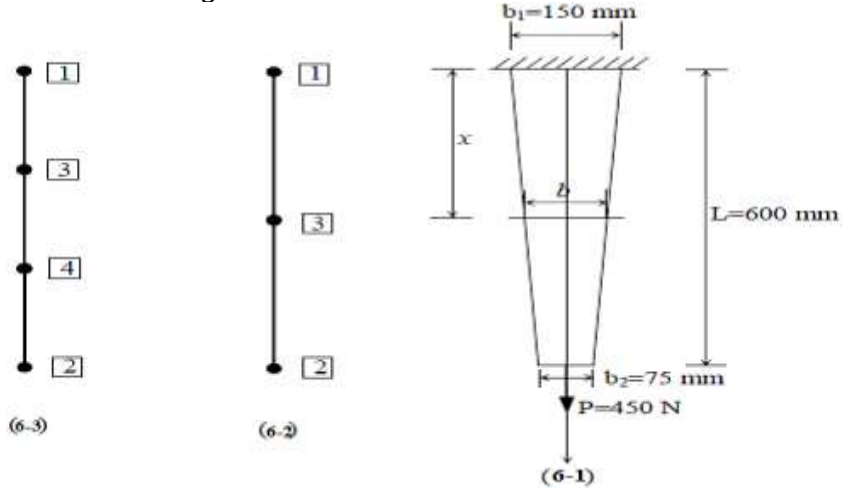
trapezium shape steel sheet example

In this part,use the example 3.3 of reference [5] to verify the formulation presented for ,four nodes finite element with variant cross-section.Note that units are transformed to SI system and concentrated load that was in center of sheet in reference [5], has transformed to the free end of the sheet.The shape of this steel sheet and support conditions and its loading has shown in figure (6).

Figure 6-1: trapezium shape steel sheet in vertical plate

Figure 6-2:three nodes finite element model

Figure 6-3: four nodes finite element model



In figure 6-1 the shape of this trapezium steel sheet and its dimensions is drawed. Sheet thickness , t=25 mm , modulus of elasticity,E=2 —— ,and density is —.Because the sheet is statically determinate, the axial average stress, in each cross-section of sheet(axial stress in x direction) can be obtained by division the sum of P force and weight of that part of the sheet that is in distance equal to x from fixed support ($W_1\ weight$) on area of cross-section:

That in Eq .(44) , b, is the width of the sheet that is in distance equal to x from fixed support, equal to:

And the force is equal to:

In this example in the first step, we use a three nodes finite element with constant cross-section for modeling this sheet. The numbers of the nodes of this three nodes finite element is shown in figure 6-2. This model totally has 3 degrees of freedom. Using the Eq .(11) ,stiffness matrix of this finite element is equal to:

$$\left[\qquad\qquad 8\right] N/mm$$

Nodal forces vector due to sheet weight force calculated using the Eq .(12) and is equal to:

$$\begin{matrix} 1/6 \\ 1/6 \\ (2/3) \end{matrix} \quad \left\{22\ 08\right\}$$

Note that in Eq .(47) and Eq .(48) instead of ,we replace the finite element average cross-section, that is:

It follows from removing the first row and column in stiffness matrix (Eq .(47)) and removing the first row of nodal forces vector (Eq .(48)) and adding the concentrated force, $P = 450\ N$,to number 2 degree of freedom, that the equilibrium equation is:

$$\left[\qquad\qquad\right]\left\{q^{2}\right\} = \left\{\qquad\qquad\right\}$$

Solving the above equations ,it follows that the displacement vector is:

$$\left\{q^{2}\right\} = \left\{5.507 \times 10^{-4}\right\}$$

Observe that free end displacement of trapezium shape steel sheet ,resulted from this three nodes model with constant cross-section, is $5.507 \times 10^{-4} mm$.

Using the energy method [3],and constant stress $\sigma(x)$ over trapezium cross-section assumption, free end displacement of trapezium shape steel is equal to:

$$\int \left(\right)\left(\right) dV$$

$$\ln\left(\frac{1}{}\right)$$

$$\text{—} \left[\text{–} \quad \text{———} \quad \text{————} \quad \text{—)} \right]$$

In Eq .(52) and Eq .(53) ,$\Delta_1$and Δ_2 are the displacement due to concentrated force P and displacement due to steel sheet weight force, respectively.
In this example in the second level,a four nodes finite element with constant cross-section was used for modeling this sheet.Number of the nodes of this four nodes finite element is shown in figure 6-3.This model has totally 4 degrees of freedom.Using the Eq .(24) stiffness matrix of this element is :

$$\text{————————} \left[\qquad\qquad\qquad 378 \right] N/mm$$

Nodal forces vector due to weight force of the sheet is calculated using the Eq .(25),that is :

$$\begin{matrix} 1/8 \\ 1/8 \\ 3/8 \\ \lfloor 3/8 \rfloor \end{matrix} \quad \left\{ \quad \right\}$$

Note that in Eq .(54) and Eq .(55) instead of ,we replace the finite element average cross-section,that is:

$$\text{————————}$$

It follows from removing the first row and column in stiffness matrix (Eq .(54)) and removing the first row of nodal forces vector (Eq .(55)) and adding the concentrated force, $P = 450\ N$,to number 2 degree of freedom,that the equilibrium equation is:

$$75 \left[\qquad\qquad\qquad 594 \right] \left\{ q_3 \right\} = \left\{ \qquad \right\}$$

Solving the above equation ,gives the displacement vector, that:

$$\left\{ q_3 \right\} = \left\{ 1.993 \times 10^{-4} \right\}$$

In this example in the third level,a four nodes finite element with variant cross-section was used for modeling this sheet.Number of the nodes of this four nodes finite element is shown in figure 6-3.This model has totally 4 degrees of freedom.Using the Eq .(42) stiffness matrix of this element is :

$$\left[\qquad\qquad\qquad\qquad\qquad\qquad\qquad\qquad\qquad\qquad \right] N/mm$$

Nodal forces vector due to weight force of the sheet is calculated using the Eq .(43),that is :

$$\begin{pmatrix} 13/120 & 1/60 \\ 1/60 & 13/120 \\ 3/10 & 3/40 \\ 3/40 & 3/10 \end{pmatrix} \begin{Bmatrix} \\ \\ \end{Bmatrix}$$

Note that in Eq .(58) and Eq .(59) instead of A_1 ,we replace the finite element cross-section in 1 node and instead of A_2 ,we replace the finite element cross-section in 2 node,3750 mm^2 and 1875 respectively.

It follows from removing the first row and column in stiffness matrix (Eq .(58)) and removing the first row of nodal forces vector (Eq .(59)) and adding the concentrated force, $P = 450\ N$,to number 2 degree of freedom,that the equilibrium equation is:

$$\begin{bmatrix} \end{bmatrix} \begin{Bmatrix} q_3 \end{Bmatrix} = \begin{Bmatrix} \end{Bmatrix}$$

Solving the above equation ,gives the displacement vector, that:

$$\begin{Bmatrix} q_3 \end{Bmatrix} = \begin{Bmatrix} 1 \\ \end{Bmatrix}$$

Result:

The objective of this paper is to model the cross-section variations of metal members of civil engineering using the four nodes finite elements. In the section 3.1, we analysed the example of rotary bar using the two,three nodes finite elements and one ,four nodes finite element.The displacement vectors of rotary bar using the two, three nodes finite elements are obtained in equation (36), and displacement vectors of rotary bar using the one ,four nodes finite element are obtained in equation (40). Observe that the free end displacement of rotary bar ,achieved from this model,($13.631\times 10^{-3} mm$) is 6.67 % greater than free end displacement of achieved from finite element model composed of 2 three nodes finite element (12.779× mm).

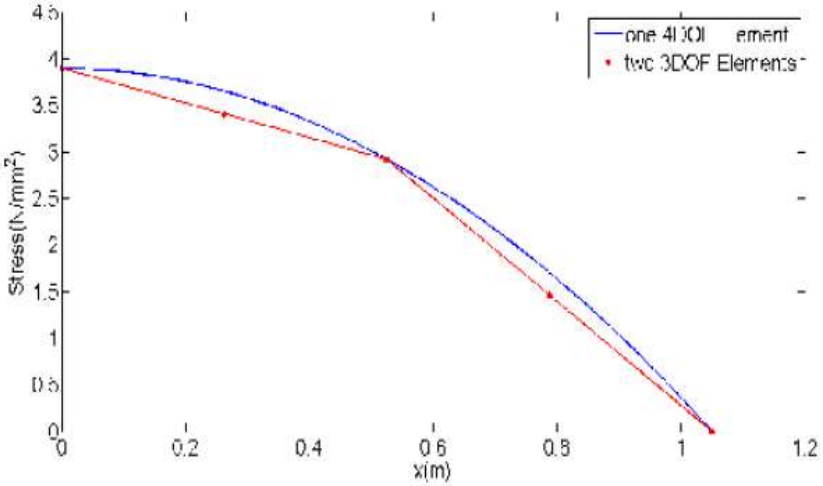
Using the Eq .(22) and Eq .(23) and nodal displacement of four nodes finite element to calculate the stress in any plot of this finite element.Therefor, according to figure (6-3),stresses in the first and last of the nodes of 1,2 , are 3.895 Mpa and 0 , respectively.

from the theory,stress in any point , x ,of a rotary bar with constant cross-section ,A, is obtained using the ——),note that, L is the length of the bar and is the angular velocity of rotary bar and,x, is the origin of coordinates. Observe that the stress variations,is a quadratic function of x , and stresses obtained from four nodes finite elements are correspond to the stresses obtaind from theory equation.Reason is that the stress variations in four nodes finite element,is a quadratic function of ξ,according of Eq .(22) and Eq .(23) ,and therefor ,modelling the rotary bar with only one,four node finite element leads in a exact solution.

Stresses [12] obtained from finite element model,composed of two,three nodes finite elements,only in the first and last nodes are corresponding to stresses obtained from theory relation.and in internal nodes,stresses obtained from finite element model are lesser than its exact and theory value .The reason is that the stress variations in three nodes finite element according to Eq .(9) and Eq .(10) , is a linear function of ξ , and so, modelling the rotary bar even with two ,three nodes finite elements, leads in an

approximate solution.Figure (7) shows the stress variations in a rotary bar using two,three nodes finite elements and one,four nodes finite element (exact solution) .

Figure 7: stress variations in rotary bar with modeling by a four nodes finite element and two,three nodes finite elements



In the section 4.1 we analysed the trapezium shape steel sheet example. In this example in the first step,we use a three nodes finite element with constant cross-section for modeling this sheet.The displacement vectors are obtained in equation (50).Next we used the energy method ,and constant stress x) over trapezium cross-section assumption, to obtain free end displacement of trapezium shape steel, that is in equations (51),(52) and (53). Observe that free end displacement of trapezium shape steel sheet,achieved from three nodes model with constant cross-section,is 1% lower than displacements achieved from Eq .(51) to Eq .(53) .Using the Eq .(9) and Eq .(10) and nodal displacement of three nodes finite element, to calculate the stresses of each plot of this finite element.Therefor,stresses in the first,middle and last nodes (1,3 and 2 nodes) of this finite element is 0.207,0.184 and 0.160 Mpa respectively.

The axial average stress achieved from Eq .(44) for trapezium sheet fixed end, middle of the length of the sheet and the free end is 0.155,0.180 and 0.240 Mpa respectively.if we compare this values with stresses obtained for three nodes finite element in the first,middle and last nodes the difference ratios are 33.5 ,2.2 and -33.3 %.

In this example in the second level,a four nodes finite element with constant cross-section was used for modeling this sheet.The displacement vectors are obtained in equation (57). Observe that free end displacement of trapezium shape steel sheet ,resulted from this four nodes model with constant cross-section, is 5.507×10^{-4} as before.

Using the Eq .(22) and Eq .(23) and nodal displacement of four nodes finite element to calculate the stress in any plot of this finite element.Therefor,stresses in the first node ,– node , – node and the last node(1,3,4,2 nodes) is 0.207,0.194,0.176 and 0.160 Mpa , respectively.

The axial average stress achieved from Eq .(44) for trapezium sheet fixed end, – of the length of the sheet from fixed side, , – of the length of the sheet from fixed end, and the free end is 0.155,0.169,0.194 and 0.240 Mpa respectively.if we compare this values with stresses obtained for four nodes finite element in the first node, – node, – node and last nodes the difference ratios are 33.5 ,14.8,-9.3 and -33.3 %. In this

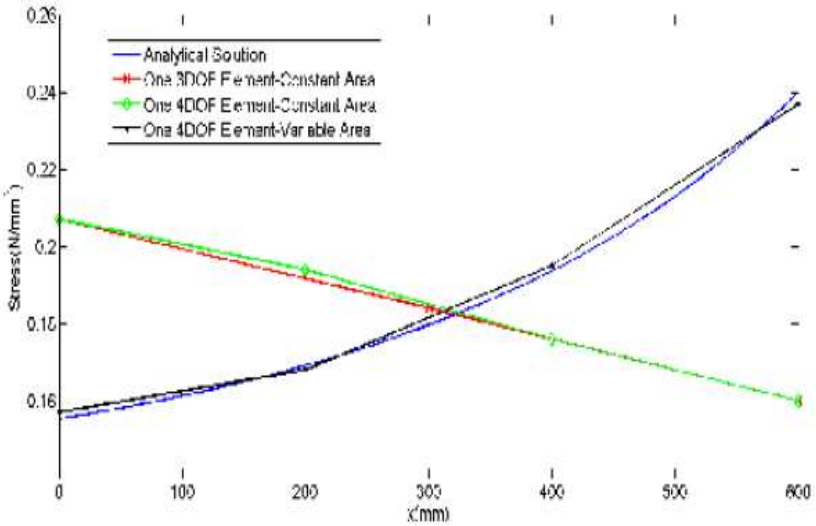
example in the third level,a four nodes finite element with variant cross-section was used for modeling this sheet. The displacement vectors are obtained in equation (61). Observe that free end displacement of trapezium shape steel sheet ,resulted from this four nodes model with variant cross-section, is 5. mm the same as values obtained in energy method(Eq .(51) to Eq .(53)).

Using the Eq .(22) and Eq .(23) and nodal displacement of four nodes finite element to calculate the stress in any plot of this finite element.Therefor,stresses in the first node ,– node , – node and the last node(1,3,4,2 nodes) is 0.157,0.168,0.195 and 0.237 Mpa , respectively.

The axial average stress achieved from Eq .(44) for trapezium sheet fixed end, – of the length of the sheet from fixed side, , – of the length of the sheet from fixed end, and the free end is 0.155,0.169,0.194 and 0.240 Mpa respectively.If we compare this values with stresses obtained for four nodes finite element with variant cross-section in the first node, – node, – node and last nodes the difference ratios are 1.3 ,-0.6,0.5 and -1.3 %.Observe that the differences are insignificant and could be the result of digit rounding.

In the figure (8), axial average stress achieved from Eq .(44) along the trapezium sheet has been drawn.Also,stresses achieved from three nodes finite element and four nodes finite element is given in this figure.

Figure 8: analytical stress variations in trapezium shape steel sheet ,also with modeling by a three nodes finite element and a four nodes finite element



As observed in calculations,stresses obtained from four nodes finite element with variant cross-section ,has conformity with axial stress in Eq .(44) .But, stresses obtained from four nodes finite element with constant cross-section and stresses obtained from three nodes finite element with constant cross-section ,have abundant differences with axial stresses obtained in Eq .(44) .Also,observe that with increase in x ,axial stress in Eq .(44) increases ,where stresses achieved from three nodes and four nodes finite element with constant cross-section have descend trend and with increase in x , they decrease.

Conclusion

According to discussed subjects in previous parts,following conclusions are imported:

1-as stress and strain variations in four nodes finite element is a quadratic function,so in rotary bars with constant cross-section,using the Eqs .(24) and .(26) for finite element stiffness matrix and finite element

nodal forces,will lead to the results that have conformity with analytical method.Whereas,if three nodes finite element formulation is used,as stress and strain variations in three nodes finite element is linear,stresses in the first and last node have conformity with analytical method,but the value in the middle node is lesser than analytical method.

2-in structural elements with linear cross-section variation,using the Eq .(42) and Eq .(43) for finite element stiffness matrix and four nodes finite element nodal forces vector due to weight force,will lead to the stresses that have conformity with analytical method.Whereas,if three or four nodes finite element formulation with constant cross-section is used ,stresses in the first and last nodes of finite element have abundant differences with stresses obtained from analytical method,but as expect,stress in finite element nodes ,have few differences with analytical method.

3-totally it is said that using the presented formulation in this article will lead to increase in calculation accuracy and decrease in costs.Because with variant cross-section,for increasing in calculations accuracy in the case of using the finite element formulation with constant cross-section ,the element should divided into many finite element .Whereas with presented formulation in this article it will be done only with one ,four nodes finite element with variant cross-section with high calculation accuracy.

References:

[1]. Chapra, Steven C (2005). Numerical Methods for Engineers, Fifth Edition.

[2]. Y.H. Zeinali, S.M. Jamali, and S. Musician, “General Form of the Stiffness Matrix of a Tapered Beam-column”, International Journal of Mining, Metallurgy & Mechanical Engineering (IJMMME), Vol. 1, Issue 2, pp. 149-153, 2013.

[3]. A. Mellati, M. Hadikhan Tehrani, Y. Zeinali, and F. Paytam, "Evaluation of Overstrength Factor of Steel Moment Resisting Frames", Proceeding of the International Conference on Advances in Structural, Civil and Environmental Engineering (SCEE 2013), pp. 100-103, 2013.

[4]. Y. Zeinali, M.M. Alinia, S. M. Hoseini, S.M. Mirfattah, "Geometrical Imperfections Sensitivity of a Four-node Membrane Element Using Perturbation Analysis Method”, International Journal of Advancements Civil Structural and Environmental Engineering, Vol. 1, Issue 1, pp. 71-74, 2013.

[5]. Tahouni, Shapor (1996). Finite Element Structural Analysis, 2nd ed. Elm Va Adab Publications.

[6]. Baker, A J., PEPPER, D. W., (1991). Finite Elements 1-2-3. New York: McGraw-Hill,

[7]. Beytschko, T., Moran, B., and Lju,.W. K. (2000). Nonlinear Finite Elements for Continua and Structures, New York: Wiley.

[8]. ZIENKIEWICZ, 0. C., and TAYLOR, R. L. (1991). The Finite Element Method, Vol. 2. New York: McGrawHill.

[9]. Lepi, S.M. (1998)., Practical Guide to Finite Elements: A Solid Mechanics Approach.

[10]. Marcel Dekker, Moaveni, S. (1998)., Finite Element Analysis: Theory and Applications with Ansys. Upper Saddle River: Prentice Hall.

[11]. Desai, C. S., and KUNDU, T. (2000)., Introductory Finite Element Method. CRC Press.

[12]. Cook, R. D. (1995), Finite Element Modeling for Stress Analysis. New York: Wiley.

[13]. Reddy. J. N., and Gartling, D. K.(2000). The Finite Element Method in Hear Transfer and Fluid Dynamics CRC Press.