



Research Article / Araştırma Makalesi

A NOVEL APPROACH FOR STATIC ANALYSIS OF PLANE FRAME SYSTEMS

Orkun YILMAZ*, Abdurrahman ŞAHİN

Yildiz Technical University, Department of Civil Engineering, Davutpasa-İSTANBUL

Received/Geliş: 29.09.2014 Accepted/Kabul: 21.12.2014

ABSTRACT

This paper presents a novel approach for analyzing plane frame systems by using Finite Element Method. The topology matrices are used to analyze the system. The releases may be easily assigned to elements and any boundary conditions may be applied to the system. In analysis process, all frame elements are assumed as having rigid connections, even though elements have releases or not. Additional freedoms are added to the nodes to consider releases. The coordinate system of nodes can be changed and supports can be assigned with any angle. Additional freedoms and the coordinate system rotations of nodes are taken into account by using topology matrices. An educational program was developed to analyze frame systems by using proposed approach. A numerical example is taken into account to explain the analysis process. The results of numerical example have compared with SAP2000 structural analysis program and suitable results are obtained.

Keywords: Finite element method, topology matrix, static analysis, frame systems.

DÜZLEM ÇERÇEVE SİSTEMLERİN STATİK ANALİZİ İÇİN YENİ BİR YAKLAŞIM

ÖZET

Bu makale, düzlem çerçeve sistemlerin sonlu elemanlar yöntemi ile analizi için yeni bir yaklaşım sunmaktadır. Sistemin analizi için topoloji matrisleri kullanılmaktadır. Mafsallar elemanlara kolayca atanabilir ve herhangi sınır koşulları kolayca sisteme uygulanabilir. Analiz sürecinde bütün çerçeve elemanlarının, mafsala sahip olsun ya da olmasın, rijit bağlantıya sahip olduğu varsayılır. Mafsalları hesaba katmak için düğüm noktalarına ilave serbestlikler eklenir. Düğüm noktalarının koordinat sistemleri değiştirilebilir ve mesnetler herhangi bir açıyla atanabilir. İlave serbestlikler ve düğüm noktalarının koordinat sistem dönme, topoloji matrisleri kullanılarak göz önüne alınır. Çerçeve sistemleri, tasarlanan yaklaşımı kullanarak analiz etmek için bir eğitim programı geliştirilmiştir. Analiz sürecini açıklamak için sayısal bir örnek dikkate alınmıştır. Sayısal örneğin sonuçları SAP2000 yapısal analiz programı ile karşılaştırılmış ve uygun sonuçlar elde edilmiştir.

Anahtar Sözcükler: Sonlu elemanlar yöntemi, topoloji matrisi, statik analiz, çerçeve sistemler.

1. INTRODUCTION

Finite element method (FEM) is a numerical technique which gives approximate solutions to boundary value problems for differential equations are found. Variational methods to minimize an error function and produce a stable solution are used [1-6]. FEM is used for a wide range of

* Corresponding Author/Sorumlu Yazar: e-mail/e-ileti: orkun.yilmaz@hotmail.com.tr, tel: (212) 383

engineering analysis such as progressive collapse analysis of buildings [7-9], simulating the blast effects on the structural elements [10-12], simulating the aircraft structures [13, 14], simulating aircraft impact on structures [15, 16] and human body simulation [17], etc.

In classical solution algorithm of frame systems with FEM, different element stiffness matrices are formed depending on the release conditions. Then, these matrices are combined to generate system stiffness matrix. In proposed approach, the stiffness matrix formed for rigid element is used for all elements which have different release conditions. After forming the stiffness matrices, new freedoms are added to the nodes which have releases. The added freedoms are given in Figure 1 for a plane frame element; a) for moment release, b) for shear release, and c) for axial release.

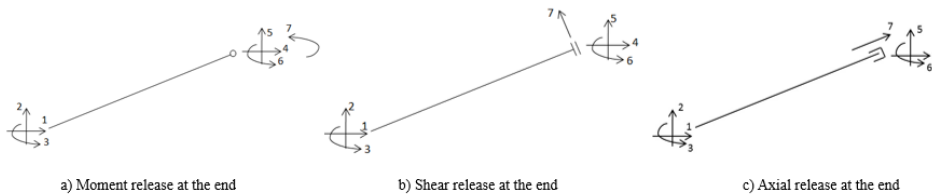


Figure 1. Degrees of freedom (DOF) and directions of a plane frame element which has releases at the end

2. PROPOSED APPROACH FOR SYSTEM ANALYSIS

2.1. Topology Matrix for Elements and Releases

Topology matrix is used to define element location in the system in terms of degrees of freedom (DOF). The row and column dimensions of the topology matrices are the DOF of element and system, respectively. If the freedom of an element coincides with the freedom of system, values are defined as “1” at that freedom number while the other values are assigned as zero. The detailed use of topology matrices may be found in [5].

In proposed algorithm, two topology matrices for an element must be obtained. These are topology matrix of element (τ_e) and topology matrix of releases for an element (τ_r). In this study, the column numbers of topology matrices are increased with the number of added releases. The proposed methodology may be explained on an example. A simple frame which has 4 nodes, 3 elements and 14 DOF is presented as given in Figure 2. DOF numbers from 1 to 12 are related to the joints and numbers 13, 14 are related to the releases.

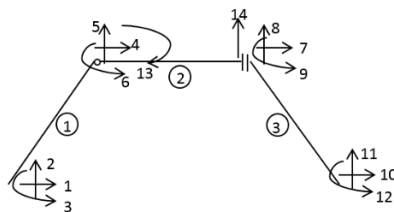


Figure 2. A simple frame system with moment and shear releases

The element 2 has two releases. There is moment release at initial node and shear release at end node. Other elements have not any releases. The topology matrix of element 2 and the topology matrix of releases for element 2 are obtained as follows:

$$\begin{matrix}
 & 1 & 2 & 3 & 4 & 5 & 6 & 7 & 8 & 9 & 10 & 11 & 12 & 13 & 14 \\
 \tau_e^2 = & \begin{bmatrix}
 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
 0 & 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
 0 & 0 & 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
 0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 \\
 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 & 0 \\
 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 \\
 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 \\
 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 & 0 & 0 \\
 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 & 0 \\
 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 \\
 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0
 \end{bmatrix} \\
 & 1 & 2 & 3 & 4 & 5 & 6 & 7 & 8 & 9 & 10 & 11 & 12 & 13 & 14 \\
 \tau_r^2 = & \begin{bmatrix}
 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 & 0 & 0 \\
 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 & 0 \\
 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0
 \end{bmatrix}
 \end{matrix}$$

The topology matrices of releases of other elements are zero matrices with same dimensions. The general illustration of topology matrix is given in Figure 3.

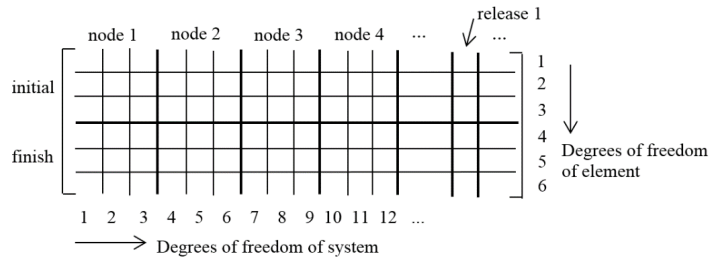


Figure 3. General illustration of proposed topology matrix

Total DOF of system is obtained as follows:

$$DOF = 3 \cdot n + r \tag{1}$$

Where:

n: number of nodes of system, r: number of releases of system

The dimensions of each topology matrices are equal to 6xDOF. After obtaining topology matrices of each element (τ_e), the topology matrices of releases (τ_r) located on the related element must be produced. The directions of added new freedoms depend on the releases. Topology matrices of releases must be multiplied by element transformation matrix to change the directions from local axes to global axes. After the transformation process, the directions of added new freedoms are obtained in global axis and the topology matrices of releases are obtained in global axis (τ_h).

The topology matrix of element and the topology matrix of releases located on element in global coordinate system must be combined. After that, topology matrix of all freedoms of element (τ_g) are obtained.

2.2. Node Rotations in the System

Coordinate systems of nodes can be changed to assign supports with any angle. The node transformation matrix of element is used to change coordinate systems of nodes and this matrix is as follows:

$$t_j^i = \begin{bmatrix} c_j & -s_j & 0 & 0 & 0 & 0 \\ s_j & c_j & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & c_j & -s_j & 0 \\ 0 & 0 & 0 & s_j & c_j & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix} \tag{2}$$

Where:

c_j : cosine of rotation of node, s_j : sine of rotation of node

After applying changes in coordinate systems of nodes, these changes need to be applied to the system. Topology matrices developed for each element need to be transformed into new DOF system as follows:

$$\tau_l^i = t_j^i \cdot \tau_g^i \tag{3}$$

2.3. The Stiffness Matrix of the System

Global stiffness matrices of each elements are extended into to system size by using topology matrices and obtained as follows:

$$K_{gg}^i = (\tau_l^i)^T \cdot K_g^i \cdot \tau_l^i \tag{4}$$

Where:

K_g^i : Global stiffness matrice of element and are obtained by local stiffness matrice and element transformation matrice [1-6].

For generating system stiffness matrix, all extended stiffness matrices are combined.

2.4. Load Vector Producing

External load vector is produced by using joint load vector (F_j) and element load vector ($\Sigma f_{gg}^{(i)}$). These vectors are combined and system load vector is obtained. External joint load vector is easily defined by adding the applied loads into the related DOF in the vector. Element load vectors are generated by using fictive point loads at the ends of the element (f_i). The distributed loads are transformed into these fictive point loads. The load vector must be defined in the global axis (f_g) and extended to system size (f_{gg}) by using topology matrix of related element. Fictive point loads due to distributed load as given in Figure 4 are obtained as given with Equation 5 [1-6].

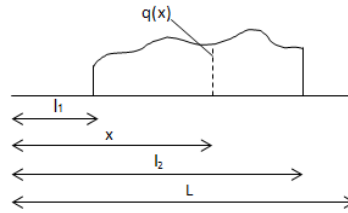


Figure 4. A distributed load example.

$$f_l^i = \int_{l_1}^{l_2} q(x) \cdot \begin{bmatrix} N_1 \\ N_2 \\ N_3 \\ N_4 \\ N_5 \\ N_6 \end{bmatrix} \cdot dx \tag{5}$$

Where:

l_1 : Starting point of distributed load, l_2 : End point of distributed load, $q(x)$: The value of distributed load to x , L : Length of element, $N_1=0$, $N_2=1-3x^2/L^2+2x^3/L^3$, $N_3=x-2x^2/L+x^3/L^2$, $N_4=0$, $N_5=3x^2/L^2-2x^3/L^3$, $N_6=x^3/L^2-x^2/L$

2.5. Obtaining Internal Forces

The boundary conditions must be taken into account before the analysis. The system stiffness matrix and external load vector must be reduced depending on the boundary conditions. Reduced displacement vector is obtained as follows:

$$U_r = (K_r)^{-1} \cdot F_r \tag{6}$$

The reduced displacement vector needs to be extended to system size (U). Restricted freedoms are considered as “0”. The size of system displacement vector is equal to total DOF of system. The additional displacements of releases given in Figure 5 are also added into the obtained load vector.

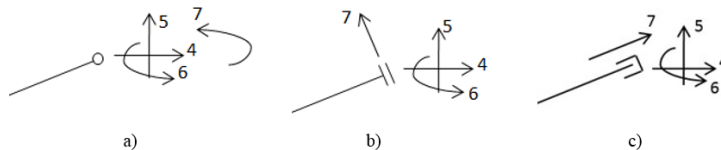


Figure 5. Three nodes which have different releases

In Fig. 5-a, a moment release is given. The rotation value of this node is obtained by combining displacements no. 6 and 7.

In Fig. 5-b, shear release has been assigned. The displacement 7 is transformed into global axis, firstly. The translation values at this node are calculated by combining the transformed values of displacement no.7 with displacements 4 and 5.

In Fig. 5-c, axial release has been assigned. The displacement 7 is transformed into global axis, firstly. In the same way, the translation values at this node are calculated by combining the transformed values of displacement no.7 with displacements 4 and 5. Finally, the displacement vectors of each elements are obtained as follows:

$$U_g^i = U_c^i + t^i \cdot U_h^i \tag{7}$$

Where:

U_g : The displacement vector of each element with all freedoms, U_c : The displacement vector of each element, t : Transformation matrix of element, U_h : The displacement vector of releases on each element.

The displacement vector of each element with all freedoms have been obtained for changed coordinate systems of nodes. The displacement vectors are transformed into the global axis (X, Y) by using equation 8.

$$U_{eg}^i = t_j^i \cdot U_g^i \tag{8}$$

Where:

U_{eg} : The displacement vector of each element with all freedoms in the global axis.

The global axis element displacement vectors need be transformed to local element displacement vectors (U_l).

The internal force vector of each element is obtained as follows:

$$S_l^i = K_l^i \cdot U_l^i - f_l^i \tag{9}$$

3. NUMERICAL APPLICATION

A numerical example is taken into account to explain the analysis process. Geometric shape and external loads are given in Figure 6. The element and node numbers, freedoms and their directions are also presented in Figure 7.

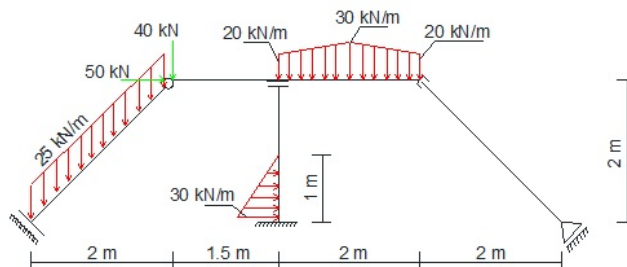


Figure 6. Geometric shape and external loads

The sample system has 6 nodes and 3 releases. There are 21 DOF in this example. Element 1 has a moment release at the end node, element 3 has a shear release at the end node, and element 5 has an axial release at the initial node.

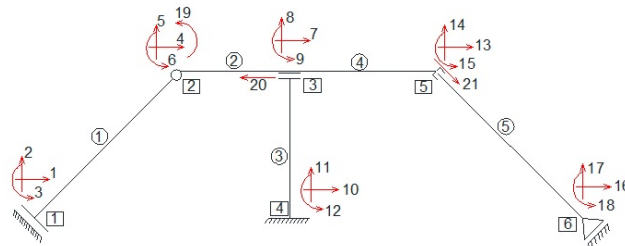


Figure 7. Element and node numbers, freedoms and their directions.

Table 1. Some information about elements

| Element | Section | Moment of inertia (cm ⁴) | Axial Area (cm ²) | cs | sn | Initial Node | End Node | l (m) |
|---------|---------|--------------------------------------|-------------------------------|-------|--------|--------------|----------|-------|
| 1 | IPE200 | 1940 | 28.5 | 0.707 | 0.707 | 1 | 2 | 2.83 |
| 2 | IPE300 | 8360 | 53.8 | 1 | 0 | 2 | 3 | 1.5 |
| 3 | HE200A | 3690 | 53.8 | 0 | 1 | 4 | 3 | 2 |
| 4 | IPE300 | 8360 | 53.8 | 1 | 0 | 3 | 5 | 2 |
| 5 | IPE200 | 1940 | 28.5 | 0.707 | -0.707 | 5 | 6 | 2.83 |

For all elements modulus of elasticity is assigned as 2.1×10^8 kN/m².

The local stiffness matrices, transformation matrices and global stiffness matrices for all elements are obtained. The dimensions of each topology matrices are equal to 6x21. The topology matrices of elements and the topology matrices of releases on elements are obtained. Element 2 and 4 have not any releases, so their topology matrices of releases on elements are zero matrices with the same dimensions. The directions of added new freedoms change depending on the releases. Therefore, the topology matrices of releases of element 3 and 5 must be multiplied by transformation matrices of elements. Both topology matrices are combined and the topology matrices of elements with all freedoms are obtained. As it can be seen in Figure 6, there are sloping supports in the system. The node transformation matrices of elements must be obtained to consider sloping supports. The topology matrices of elements with all freedoms are multiplied by the node transformation matrices of elements and the topology matrices for new coordinate systems are obtained. Coordinate systems of nodes for elements 2, 3 and 4 are not changed. Therefore, their node transformation matrices are identity matrices. After using node transformation matrices, new directions of freedoms are obtained as shown in Figure 8.

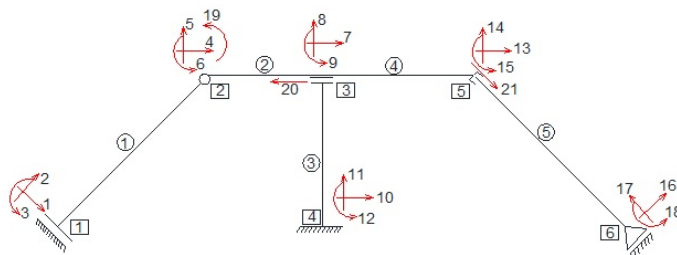


Figure 8. Freedoms and their new directions

$$\tau_l^s = \begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0.707 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 & -0.707 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0.707 & -0.707 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0.707 & 0.707 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 \end{bmatrix}$$

The extended stiffness matrices of elements are obtained by using equation 4. The extended stiffness matrices of elements are combined and the system stiffness matrix is obtained. Then, node loading vector and element loading vectors are combined and external load vector is obtained. The boundary conditions must be taken into account. For this numerical example, freedoms no. 2, 3, 10, 11, 12 and 17 are restricted. Row and column numbers coinciding with these numbers must be deleted at stiffness matrix of system. For external load vector, the row numbers coinciding with these numbers must also be deleted. Then, the reduced system stiffness matrix (K_r) and reduced external loading vector (F_r) are obtained. Displacements are obtained by using equation 6. The freedom numbers of displacements are shown in parenthesis.

$$U_r = [(1)17.77 \quad (4)8.5 \quad (5)-8.44 \quad (6)5.94 \quad (7)8.5 \quad (8)-0.037 \quad (9)4.92 \quad (13)8.5 \dots \\ (14)9.5 \quad (15)4.73 \quad (16)26.12 \quad (18)4.73 \quad (19)-2.67 \quad (20)13.41 \quad (21)0.71]^T \cdot 10^{-2}$$

Note that displacement 19, 20 and 21 belongs to releases. Restricted freedoms are considered as “0” and extended displacement vector (U) is obtained. The element no. 1 has a moment release at the end node. The rotation of node which have moment release is obtained by adding displacement no. 6 and 19. The element no. 3 has a shear release at the end node and element no. 5 has an axial release at the initial node. Translations of these nodes are obtained by combining transformed displacements of releases to the global axes with displacement of nodes. The displacement vectors of each element with all freedoms are obtained as follows:

$$U_g^1 = [17.77 \quad 0 \quad 0 \quad 8.5 \quad -8.44 \quad 3.27]^T \cdot 10^{-2}$$

$$U_g^2 = [8.5 \quad -8.44 \quad 5.94 \quad 8.5 \quad 0.037 \quad 4.92]^T \cdot 10^{-2}$$

$$U_g^3 = [0 \quad 0 \quad 0 \quad -4.91 \quad -0.037 \quad 4.92]^T \cdot 10^{-2}$$

$$U_g^4 = [8.5 \quad -0.037 \quad 4.92 \quad 8.5 \quad 9.5 \quad 4.73]^T \cdot 10^{-2}$$

$$U_g^5 = [9 \quad 9 \quad 4.73 \quad 26.12 \quad 0 \quad 4.73]^T \cdot 10^{-2}$$

The displacement values for elements 1 and 5 are obtained in changed node coordinate systems. They need be transformed into the global coordinate system by using equation 12.

$$U_{eg}^1 = [12.56 \quad -12.56 \quad 0 \quad 8.5 \quad -8.44 \quad 3.27]^T \cdot 10^{-2}$$

$$U_{eg}^5 = [9 \quad 9 \quad 4.73 \quad 18.47 \quad 18.47 \quad 4.73]^T \cdot 10^{-2}$$

Global displacement vectors are transformed to local axis.

$$U_l^1 = [0 \quad -17.77 \quad 0 \quad 0.045 \quad -11.98 \quad 3.27]^T \cdot 10^{-2}$$

$$U_i^3 = [0 \ 0 \ 0 \ -0.037 \ 4.9 \ 4.9]^T \cdot 10^{-2}$$

$$U_i^5 = [0 \ 12.74 \ 4.73 \ 0 \ 26.12 \ 4.73]^T \cdot 10^{-2}$$

Finally, internal loading vectors are obtained by using equation 9.

$$S_i^1 = [-70.71 \ 0 \ -70.71 \ 120.71 \ 50 \ 0]^T$$

$$S_i^2 = [0 \ -160.71 \ 0 \ 0 \ 160.71 \ -241.07]^T$$

$$S_i^3 = [210.71 \ 15 \ -186.07 \ -210.71 \ 0 \ 191.07]^T$$

$$S_i^4 = [0 \ 50 \ 50 \ 0 \ 0 \ 0]^T$$

$$S_i^5 = [0 \ 0 \ 0 \ 0 \ 0 \ 0]^T$$

4. OVERVIEW OF FRAMESOLVER EDUCATION PROGRAM

An educational program which named FrameSolver was developed by using proposed algorithm. FrameSolver is developed by utilizing MATLAB [18] which is a mathematical tool developed by The Mathworks. MATLAB is very easy and quick tool for developing engineering software. The numerical example was analyzed by FrameSolver. The frame system may be created with an interactive drawing screen and the external loadings, material properties, frame sections, node releases and boundary conditions may be assigned easily. The geometrical shape of the numerical example is drawn in FrameSolver as shown in Figure 9.

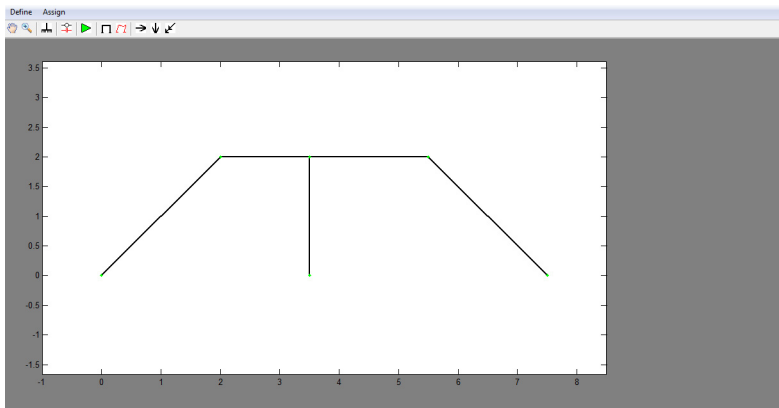


Figure 9. The geometrical shape of the numerical example

After the structural analysis is completed, deformations and internal force values are obtained and may be evaluated over the graphical screen. The deformed shape, axial force, shear force and moment diagrams may be shown in the diagram as shown in Figure 10 – 13.

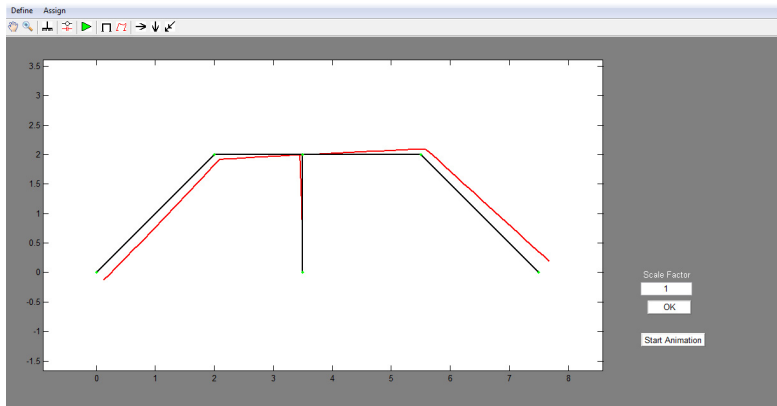


Figure 10. Deformed shape of frame obtained from FrameSolver

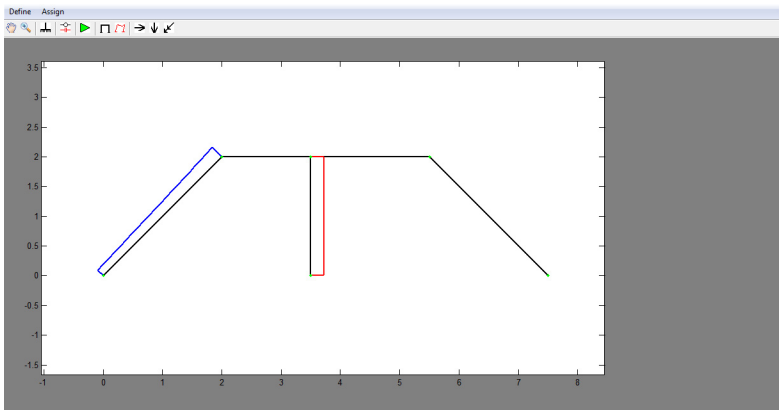


Figure 11. Axial force diagram of frame obtained from FrameSolver

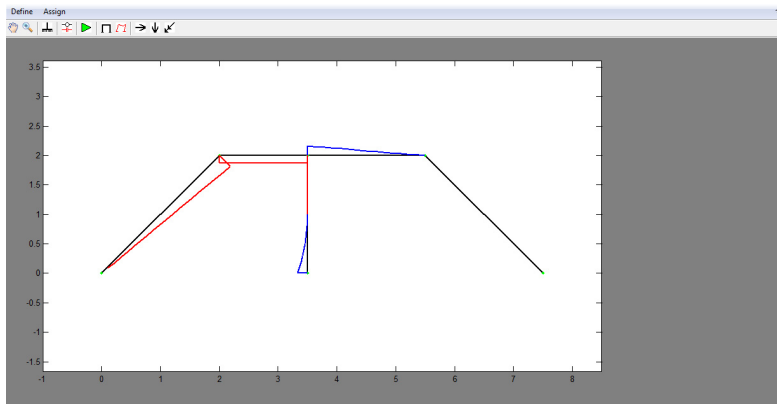


Figure 12. Shear force diagram of frame obtained from FrameSolver

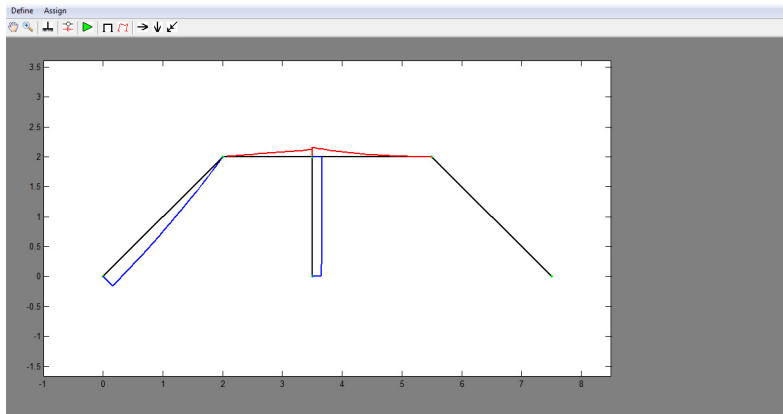


Figure 13. Moment diagram of frame obtained from FrameSolver

5. NUMERICAL VALIDATION OF RESULTS

The results of this numerical example have been compared with SAP2000 [19] structural analysis program and suitable results are obtained. The same geometry, material properties, external loadings, boundary conditions, frame sections and releases are used. The same displacements and internal force values are obtained as shown in Table 2 - 4. Deformed shape, axial force diagram, shear force diagram and moment diagram are as shown in Figure 14 – 17. Comparison of displacements are taken into account without displacement of releases as displacement of releases cannot be observed by SAP2000.

Table 2. Comparison of displacement values (for changed coordinate systems)

| Node No. | Displacement Values of numerical example (m, rad)*10 ⁻² | | | Displacement Values of SAP2000 (m, rad)*10 ⁻² | | | Percent Difference | | |
|----------|--|--------|------|--|--------|-------|--------------------|------|-----|
| | 1 | 2 | 3 | 1 | 2 | 3 | 1 | 2 | 3 |
| 1 | 17.77 | 0 | 0 | 18.03 | 0 | 0 | 1.45 | - | - |
| 2 | 8.5 | -8.44 | 5.94 | 8.64 | -8.58 | -5.94 | 1.63 | 1.65 | 0 |
| 3 | 8.5 | -0.037 | 4.92 | 8.64 | -0.037 | -4.91 | 1.63 | 0 | 0.2 |
| 4 | 0 | 0 | 0 | 0 | 0 | 0 | - | - | - |
| 5 | 8.5 | 9.5 | 4.73 | 8.64 | 9.48 | -4.73 | 1.63 | 0.21 | 0 |
| 6 | 26.12 | 0 | 4.73 | 26.18 | 0 | -4.73 | 0.23 | - | 0 |

Table 3. Internal Force Values of numerical example

| Element No. | Internal Force Values of numerical example (kN) | | | | | |
|-------------|---|---------|---------|---------|--------|---------|
| | N | | V | | M | |
| 1 | 70.71 | 120.71 | 0 | -50 | 70.71 | 0 |
| 2 | 0 | 0 | -160.71 | -160.71 | 0 | -241.07 |
| 3 | -210.71 | -210.71 | 15 | 0 | 186.07 | 191.07 |
| 4 | 0 | 0 | 50 | 0 | -50 | 0 |
| 5 | 0 | 0 | 0 | 0 | 0 | 0 |

Table 4. Internal Force Values of SAP2000

| Element No. | Internal Force Values of SAP2000(kN) | | | | | |
|-------------|--------------------------------------|---------|--------|--------|---------|---------|
| | N | | V | | M | |
| 1 | 70.71 | 120.71 | 0 | 50 | 71.71 | 0 |
| 2 | 0 | 0 | 160.71 | 160.71 | 0 | -241.07 |
| 3 | -210.71 | -210.71 | -15 | 0 | -186.08 | -191.07 |
| 4 | 0 | 0 | -50 | 0 | -50 | 0 |
| 5 | 0 | 0 | 0 | 0 | 0 | 0 |

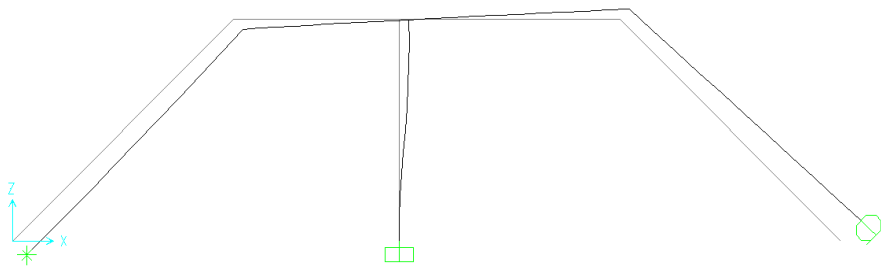


Figure 14. Deformed shape of frame obtained from SAP2000

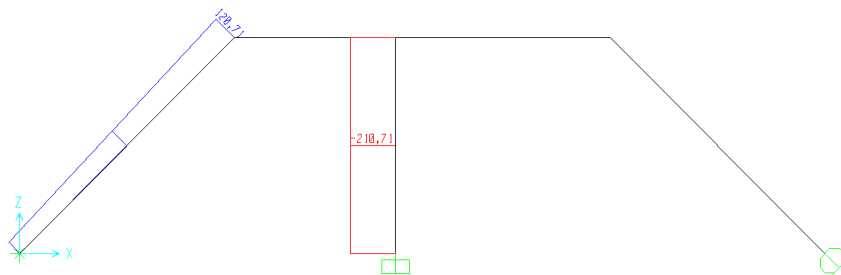


Figure 15. Axial force diagram of frame obtained from SAP2000

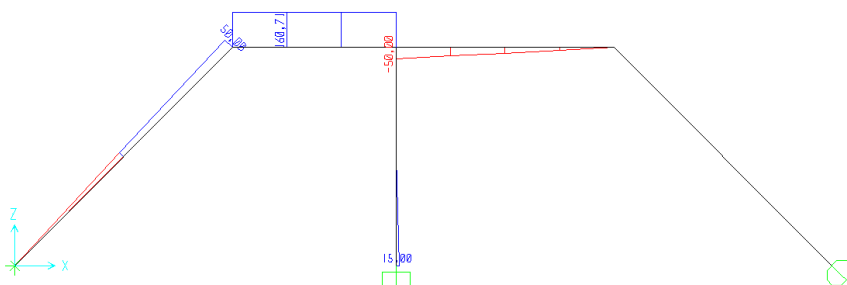


Figure 16. Shear force diagram of frame obtained from SAP2000

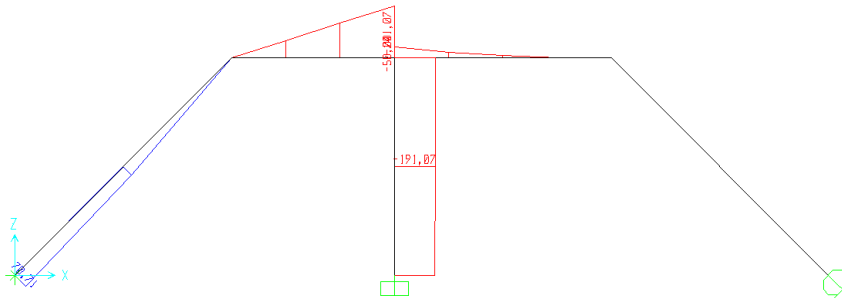


Figure 17. Moment diagram of frame obtained from SAP2000

6. CONCLUSIONS

This paper presents a new algorithm for analysis plane frame systems with Finite Element Method by using topology matrices. It is easy to assign releases to elements and change coordinate systems of nodes by using topology matrices. Additional freedoms are added to the nodes which have releases. An educational program called FrameSolver was developed by using proposed algorithm. The obtained results are compared with those of SAP2000. A perfect harmony is observed between FEM results. Finally, it should be emphasized that, in case of assigning releases to all elements of a node, the system becomes unstable as shown in Figure 18. In this example, releases are assigned towards the same node and the system becomes unstable. It is not possible to analyze the system. One release must be deleted in this node to analyze the system.

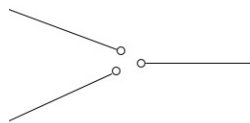


Figure 18. A simple node which all elements have releases

REFERENCES / KAYNAKLAR

- [1] O.C. Zienkiewicz, *The Finite Element Method in Engineering Science*, McGraw-Hill, London, 1977.
- [2] Zienkiewicz, O.C.; Taylor, R.L.; Zhu, J.Z. *The Finite Element Method: Its Basis and Fundamentals (Sixth ed.)*. Butterworth-Heinemann, 2005.
- [3] Bathe, K.J. *Finite Element Procedures*. Cambridge, MA: Klaus-Jürgen Bathe, 2006.
- [4] Cook, R.D.; Malkus D.S.; Plesha M.E.; Witt R.J, *Concepts and Applications of Finite Element Analysis (Fourth ed.)*. Wiley, 2001.
- [5] Kasımzade, A.A., *Finite Element Method: Fundamentals and Application on Structural Mechanics* (Language: Turkish), Birsen Yayınevi, 2004.
- [6] Şahin, A., “Advanced Computer Programming for Structural and Earthquake Engineering”, Class Notes, Yıldız Technical University – Graduate School of Natural and Applied Sciences, 2013.
- [7] Fu, F., Progressive collapse analysis of high-rise building with 3-D finite element modeling method, *Journal of Constructional Steel Research* 65, 1269–1278, 2009.

- [8] Lu X., Lu X., Guan H., Zhang W., Ye L., Earthquake-induced collapse simulation of a super-tall mega-braced frame-core tube building, *Journal of Constructional Steel Research* 82, 59–71, 2013.
- [9] Fu, F., 3-D nonlinear dynamic progressive collapse analysis of multi-storey steel composite frame buildings — Parametric study, *Engineering Structures* 32, 3974–3980, 2010.
- [10] Wu C., Sheikh H., A finite element modelling to investigate the mitigation of blast effects on reinforced concrete panel using foam cladding, *International Journal of Impact Engineering* 55, 24–33, 2013.
- [11] Sabuwala T., Linzell D., Krauthammer T., Finite element analysis of steel beam to column connections subjected to blast loads, *International Journal of Impact Engineering* 31, 861–876, 2005.
- [12] Lin X, Zhang Y.X. and Hazell P. J., Modelling the response of reinforced concrete panels under blast loading, *Materials and Design* 56, 620 – 628, 2014.
- [13] Jiapeng T., Ping X., Baoyuan Z. and Bifu H., A finite element parametric modeling technique of aircraft wing structures, *Chinese Journal of Aeronautics*, 26(5), 1202–1210, 2013.
- [14] Guo H., Bastien C., Blundell M., Wood G., Development of a detailed aircraft tyre finite element model for safety assessment, *Materials and Design* 53, 902–909, 2014.
- [15] Sadiq M., Yun Z.X., Rong P., Simulation analysis of impact tests of steel plate reinforced concrete and reinforced concrete slabs against aircraft impact and its validation with experimental results, *Nuclear Engineering and Design* 273, 653–667, 2014.
- [16] Lee K., Han S.E., Hong J.W., Analysis of impact of large commercial aircraft on a prestressed containment building, *Nuclear Engineering and Design* 265, 431–449, 2013.
- [17] Shabana A.A., Gantoi F. M., Brown M.A., Integration of finite element and multibody system algorithms for the analysis of human body motion, *Procedia IUTAM* 2, 233–240, 2011.
- [18] MATLAB The Language of Technical Computing (R2010a), The MathWorks, Inc.
- [19] SAP2000 Structural Analysis Program, Computers and Structures, Inc., 2009.