ICENS 7TH INTERNATIONAL CONFERENCE ON ENGINEERING AND NATURAL SCIENCE
June 23-27 2021 (Hybrid Conference)

The Developed FEA-Based Program for Planar Static Analysis with a Special 12x12 Rectangular Element

Ragib Sabah[1], Namik Kemal Öztorun[2], Baris Sayin[3]

Abstract

There are many widely used structural programs to calculate static loads' effect on buildings, meanwhile, a set of calculations' assumptions are used as a basis of these programs. These assumptions may lead to unreal structural calculations and behaviors. One of the most important assumptions is the stiffness matrix of shear walls which is only composed of 8x8 elements ignoring the rotational stiffness values. In this study, a computational program based on the finite element method was developed using FORTRAN programming language to study the effect of static loads on planar structures. This program is called YAY2020. In the developed program, a special rectangular elements' stiffness matrix formulation with 12x12 will be used to consider rotational deformations of the share wall elements. The correctness of this formulation has been proven by comparing it with Timoshenko beam analytical analysis. The aforementioned formulation was not in the literature until recently introduced by Öztorun. All results obtained by YAY2020 will be compared with the commonly used SAP2000 software.

Keywords: FORTRAN, Finite element analysis (FEA), 12x12 Stiffness matrix, Planar static analysis, Structural analysis software, Timoshenko Cantilever Beam

1. INTRODUCTION

The calculation and analysis of exact loads affecting buildings is the most important part of the construction process because even if the structures are constructed correctly and there were structural analysis errors, results will be significantly reflected later causing economic and human losses. The scientific development witnessed in recent years accompanying the rapid development of programming and computer sciences have revealed a set of programs that have reduced the structural analysis calculation burdens and errors, but how are the assumptions used in these programs are correct and accurate. Therefore, the mechanism of each program must be understood and monitored before getting started using it, and always seek to develop these programs to access accurate analysis results.

Öztorun [1, 2] presented a formulation of the rectangular stress element. This formulation has three degrees of freedom at each rectangular stress element's nodes. This feature distinguished Öztorun formulation from the previous stress element formula that has only two degrees of freedom in each element node. With this formulation, 12×12 rectangular stress element stiffness matrices can be calculated. Wilson [3-6] wrote the first automated finite element structural analysis computer programs known as CAL family of programs. These programs were greatly successful and gave accurate results. In 1975 under the Computers and

[1] *Corresponding author: Istanbul University-Cerrahpasa, Institute of Graduate Sciences, 34320, Avcılar/ Istanbul, Turkey* ragib.sabah@ogr.iuc.edu.tr

[2] *Istanbul University-Cerrahpasa, Department of Civil Engineering 34320, Istanbul, Turkey,* kemal@iuc.edu.tr

[3] *Istanbul University-Cerrahpasa, Department of Civil Engineering 34320, Istanbul, Turkey,* barsayin@iuc.edu.tr

Structures Inc. firm's platform, he played a role in the development of structure programs such as SAP2000, SAFE and ETABS. In these programs, shear-wall stiffness matrices contain only 8x8 elements with no rotational stiffness values. Bathe [7, 8] was one of the pioneers in the field of finite element analysis and its applications. He worked with Wilson together to develop finite element analysis in structures. He wrote a book called "Finite Element Procedures" to explain the detailed procedure of finite element analysis. Turner [9] presented a set of arithmetic formulations for triangular and rectangular stress elements, but in these formulations, the rotational stiffness values were neglected. Griffiths [10] presented a stiffness matrix of the four-node quadrilateral element in the closed-form this quadrilateral element had just 8x8 stiffness matrix elements. Öztorun and others [11, 12] performed static and modal analyzes of a bridge using the GP-DYNA computer program developed by E. Öztorun in his doctoral thesis. As a result of the analyses, the first findings revealed that the bridge piers were strained under the effects of horizontal earthquakes, that piers and horizontal elements could be strained under vertical effects at frequencies close to the frequencies that cause the horizontal oscillations of the bridge piers; stated that this situation may be a collapse reason of a system.

In this study, a program based on the finite element method will be coded by the FORTRAN compiler [13, 14] to find the response of known geometry planar structures under static loads. The program will be called YAY2020. In principle, YAY2020 program calculates the system stiffness matrix by calculating and superposing the stiffness matrices of each system's element separately. The distinctive feature of this program, its uses of a special rectangular element formulation that add the perpendicular rotational stiffness values in the calculation of shear-wall elements' stiffness matrices with real and full-size 12×12 stiffness matrix elements. This aforementioned formulation was developed by Öztorun [1, 2]; the accuracy of this formulation had been proved by comparing it with the analytical solution presented by Timoshenko and Goodier [15]. The results of the YAY2020 program will be compared to commonly use finite element structural programs like SAP2000. In light of these features, this study aims to contribute to the literature by providing a program having more realistic structure analysis calculation methods than current programs.

2. MATERIALS AND METHODS

In this section, methods and formulations used in YAY2020 program will be explained.

2.1. Frame Elements Stiffness Matrix

Figure 1 shows a planar frame element with two nodes (i and j nodes) and three degrees of freedom in each node's ending. The axis across through i and j nodes is called the local x-axis and the perpendicular to the local x-axis is called the local y-axis, and these two axes formed the local coordinate system of the element. Frame element includes the properties of both beam element and truss element. Therefore, the stiffness matrix of the frame element can be written as Eq.1. [3, 8]

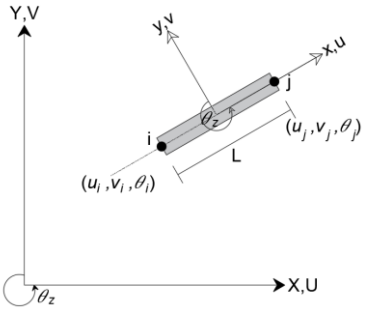

Figure 1. Planar frame element

$$K_{e.l.}^{\text{frame}} = \begin{bmatrix} AE/L & 0 & 0 & -AE/L & 0 & 0 \\ & 12EI/L^3 & 6EI/L^2 & 0 & -12EI/L^3 & 6EI/L^2 \\ & & 4EI/L & 0 & -6EI/L^2 & 2EI/L \\ & & & AE/L & 0 & 0 \\ & simetrik & & & 12EI/L^3 & -6EI/L^2 \\ & & & & & 4EI/L \end{bmatrix} \quad (1)$$

The stiffness matrix written in the Eq.1 is in the local coordinates of the element. For structural analysis, a planar transformation matrix should be used to transfer from the local coordinate system of the element to the global coordinate system of the structure. The planar transformation matrix is given in Eq.2. [3, 8]

$$R = \begin{bmatrix} Cos(\theta) & Sin(\theta) & 0 & 0 & 0 & 0 \\ -Sin(\theta) & Cos(\theta) & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & Cos(\theta) & Sin(\theta) & 0 \\ 0 & 0 & 0 & -Sin(\theta) & Cos(\theta) & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix} \quad (2)$$

Here;

$$Cos(\theta) = \frac{X_j - X_i}{L} \quad (3)$$

$$Sin(\theta) = \frac{Y_j - Y_i}{L} \quad (4)$$

L: Element length
The frame element global stiffness matrix can be obtained by multiplying the transpose of the transformation matrix by the element local stiffness matrix and then multiplied by the transformation matrix itself again, as written in Eq.5.

$$K_{e.g.}^{frame} = [R^T].\left[K_{e.l.}^{frame}\right].[R] \quad (5)$$

Here,
$K_{e.g.}^{\text{frame}}$: Frame element global stiffness matrix.
R^T: Transpose of the transformation matrix.
$K_{e.l.}^{\text{frame}}$: Frame element local stiffness matrix.

2.2. Shear Wall Elements Stiffness Matrix

The rectangular element is used to analyse shear walls in YAY2020. Shear walls count as one rectangular element on each floor of the building. To analyse rectangular elements, degrees of freedom must be determined for each node of the rectangular. The plane stress element which is shown in Figure 2 has three degrees of freedom at each node (1 rotation and 2 displacements). This element doesn't exist in the literature even it's presented by Öztorun [1]. Meanwhile, the previous rectangular stress element used in structural programs had just two degrees of freedom in each node (2 displacements) with no rotation. The plane stress element stiffness matrix formulation is given in Table 1.

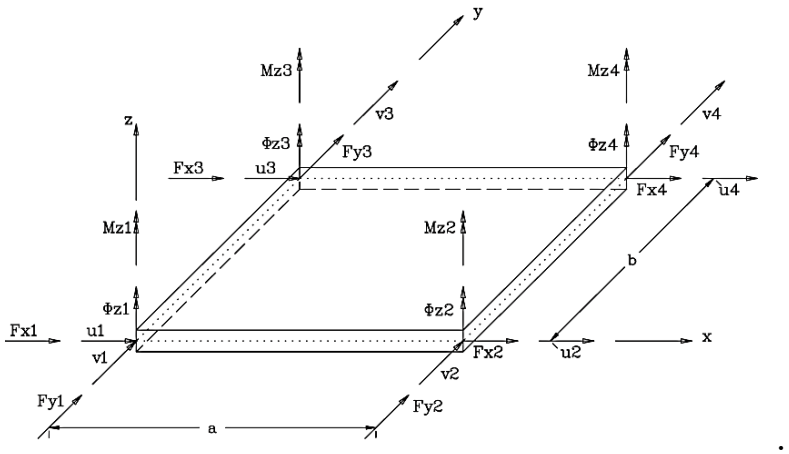

Figure 2. Generalized forces and displacements for finite plate elements for plane stress problems [1]

Table 1. Stiffness matrix of plane stress element [1]

Δ^T	Δ_{X1}	Δ_{Y1}	θ_{Z1}	Δ_{X2}	Δ_{Y2}	θ_{Z2}	Δ_{X3}	Δ_{Y3}	θ_{Z3}	Δ_{X4}	Δ_{Y4}	θ_{Z4}
I/J	1	2	3	4	5	6	7	8	9	10	11	12
1	$S_{1,1}$	$S_{1,2}$	$S_{1,3}$	$S_{1,4}$	$S_{1,5}$	$S_{1,6}$	$S_{1,7}$	$-S_{1,5}$	$S_{1,9}$	$S_{1,10}$	$-S_{1,2}$	$S_{1,12}$
2		$S_{2,2}$	$S_{2,3}$	$-S_{1,5}$	$S_{2,5}$	$S_{2,6}$	$S_{1,5}$	$S_{2,8}$	$S_{2,9}$	$-S_{1,2}$	$S_{2,11}$	$S_{2,12}$
3			$S_{3,3}$	$S_{1,6}$	$-S_{2,6}$	$S_{3,6}$	$-S_{1,9}$	$S_{2,9}$	$S_{3,9}$	$-S_{1,12}$	$-S_{2,12}$	$S_{3,12}$
4				$S_{1,1}$	$-S_{1,2}$	$S_{1,3}$	$S_{1,10}$	$S_{1,2}$	$S_{1,12}$	$S_{1,7}$	$S_{1,5}$	$S_{1,9}$
5					$S_{2,2}$	$-S_{2,3}$	$S_{1,2}$	$S_{2,11}$	$-S_{2,12}$	$-S_{1,5}$	$S_{2,8}$	$-S_{2,9}$
6						$S_{3,3}$	$-S_{1,12}$	$S_{2,12}$	$S_{3,12}$	$-S_{1,9}$	$-S_{2,9}$	$S_{3,9}$
7							$S_{1,1}$	$-S_{1,2}$	$-S_{1,3}$	$S_{1,4}$	$-S_{1,5}$	$-S_{1,6}$
8								$S_{2,2}$	$S_{2,3}$	$S_{1,5}$	$S_{2,5}$	$S_{2,6}$
9									$S_{3,3}$	$-S_{1,6}$	$-S_{2,6}$	$S_{3,6}$
10		SYMMETRIC								$S_{1,1}$	$S_{1,2}$	$-S_{1,3}$
11											$S_{2,2}$	$-S_{2,3}$
12												$S_{3,3}$

Parameters of the plane stress element stiffness matrix are given as follows: [1]

$$D_S = \frac{E_S . t_S}{(1-\upsilon_S{}^2)}$$

$$\beta = \frac{b}{a}$$

$$S_{1,1} = D_S . \left[\frac{13.\beta}{35} + \frac{(1-\upsilon_S)}{5.\beta}\right]$$

$$S_{1,2} = D_S . \frac{(1+\upsilon_S)}{8}$$

$$S_{1,3} = -D_S . a . \left[\frac{11.\beta^2}{210} + \frac{(9-19.\upsilon_S)}{240}\right]$$

$$S_{1,4} = D_S . \left[-\frac{13.\beta}{35} + \frac{(1-\upsilon_S)}{10.\beta}\right]$$

$$S_{1,5} = D_S . \frac{(3.\upsilon_S - 1)}{8}$$

$$S_{1,6} = D_S . a . \left[\frac{11.\beta^2}{210} + \frac{(3-13.\upsilon_S)}{240}\right]$$

$$S_{1,7} = D_S . \left[\frac{9.\beta}{70} - \frac{(1-\upsilon_S)}{5.\beta}\right]$$

$$S_{1,9} = D_S . a . \left[\frac{13.\beta^2}{420} - \frac{(9+\upsilon_S)}{240}\right]$$

$$S_{1,10} = -D_S . \left[\frac{9.\beta}{70} + \frac{(1-\upsilon_S)}{10.\beta}\right]$$

$$S_{1,12} = D_S . a . \left[-\frac{13.\beta^2}{420} + \frac{(3+7.\upsilon_S)}{240}\right]$$

$$S_{2,2} = D_S . \left[\frac{13}{35.\beta} + \frac{\beta.(1-\upsilon_S)}{5}\right]$$

$$S_{2,3} = D_S . b . \left[\frac{11}{210.\beta^2} + \frac{(9-19.\upsilon_S)}{240}\right]$$

$$S_{2,5} = D_S . \left[\frac{9}{70.\beta} - \frac{\beta.(1-\upsilon_S)}{5}\right]$$

$$S_{2,6} = D_S . b . \left[-\frac{13}{420.\beta^2} + \frac{(9+\upsilon_S)}{240}\right]$$

$$S_{2,8} = D_S . \left[-\frac{13}{35.\beta} + \frac{\beta.(1-\upsilon_S)}{10}\right]$$

$$S_{2,9} = -D_S . b . \left[\frac{11}{210.\beta^2} + \frac{(3-13.\upsilon_S)}{240}\right]$$

$$S_{2,11} = -D_S . \left[\frac{9}{70.\beta} + \frac{\beta.(1-\upsilon_S)}{10}\right]$$

$$S_{2,12} = D_S . b . \left[\frac{13}{420.\beta^2} - \frac{(3+7.\upsilon_S)}{240}\right]$$

$$S_{3,3} = D_S . a . b . \left[\frac{1}{105} . \left(\beta^2 + \frac{1}{\beta^2}\right) + \frac{(27-37.\upsilon_S)}{720}\right]$$

$$S_{3,6} = D_S . a . b . \left[-\frac{1}{420} . \left(4.\beta^2 + \frac{3}{\beta^2}\right) + \frac{(9+\upsilon_S)}{720}\right]$$

$$S_{3,9} = D_S . a . b . \left[-\frac{1}{420} . \left(3.\beta^2 + \frac{4}{\beta^2}\right) + \frac{(9+\upsilon_S)}{720}\right]$$

$$S_{3,12} = D_S . a . b . \left[\frac{3}{420} . \left(\beta^2 + \frac{1}{\beta^2}\right) - \frac{(9+\upsilon_S)}{720}\right]$$

2.3. Calculation of Joints Displacements, Support Reactions, and End Forces of Frame Elements

Static analysis performed by YAY2020 follows the regular finite element solution method YAY2020 software superposing element load vector to create a system's general load vector. Combining the loads affect the degrees of freedom of each element, the local load vector of the elements is created. Using Hooke's law which is given in Eq.6, unknown system joints displacements, support reactions, and end forces of frame elements can be calculated.

$$F_{(n*1)}^{system} = K_{(n*n)}^{system} * U_{(n*1)}^{system} \tag{6}$$

Here, n is the size of the system stiffness matrix and simultaneously the system degrees of freedom.

3. RESULTS AND DISCUSSION

In this section, two numerical examples of YAY2020 program will be given and compared with the SAP2000 program.

3.1. Timoshenko Cantilever Beam

Timoshenko cantilever beam as seen in Figure 3 shows a plate represented by a coarse mesh of 12 rectangular elements with a total vertical concentrated load of 1000N applied to the free edge nodes of the plate. The calculated values by YAY2020 program for u and v displacements in the x and y directions, respectively, are shown in Figure 3 adjacent to each node. In addition, the values shown in parentheses are analytical solutions for u and v displacements obtained by Timoshenko and Goodier from equations based on the classical theory of Elasticity. The plate element modulus of elasticity $E = 30 \times 10^{6} N/cm^2$, Poisson ratio $\upsilon = 1/3$ and plate thickness $t = 0.1\ cm$. Figure 3 indicates that YAY2020 gives almost the same results as Timoshenko's analytical solution and the error ratio is on the order of 1%. Table 2 shows the results getting with YAY2020 program is closer to Timoshenko's analytical solutions than Sap2000 with the same number of mashing elements. In the SAP2000, using more meshing element displacements results is getting closer to YAY2020 and therefore getting closer to Timoshenko analytical solution. Some results in YAY2020 have a different signal from those of SAP2000 due to the sign notation used in the programs.

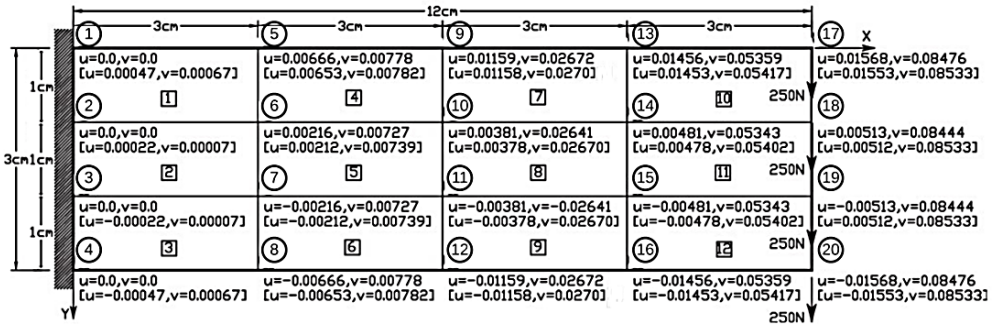

Figure 3. Comparison of displacements (cm) calculated by YAY2020 and elasticity theory [shown in parentheses][1]

Table 2. Joints displacements of Timoshenko beam calculated by YAY2020 and Sap2000

SAP2000 - Joint displacements				**YAY2020 - Joint displacements**			
Joint No	**U1 cm**	**U3 cm**	**R2 Radius**	**Joint No**	**U1 cm**	**U3 cm**	**R2 Radius**
1	0	0	0	1	0	0	0
2	0	0	0	2	0	0	0
3	0	0	0	3	0	0	0
4	0	0	0	4	0	0	0
5	8.01E-03	6.77E-03	4.87E-03	5	-7.78E-03	-6.66E-03	4.44E-03
6	7.42E-03	2.14E-03	4.55E-03	6	-7.27E-03	-2.16E-03	4.67E-03
7	7.42E-03	-2.14E-03	4.55E-03	7	-7.27E-03	2.16E-03	4.67E-03
8	8.01E-03	-6.77E-03	4.87E-03	8	-7.78E-03	6.66E-03	4.44E-03
9	2.72E-02	1.18E-02	7.96E-03	9	-2.67E-02	-1.16E-02	7.87E-03
10	2.70E-02	3.91E-03	8.01E-03	10	-2.64E-02	-3.82E-03	7.89E-03
11	2.70E-02	-3.91E-03	8.01E-03	11	-2.64E-02	3.82E-03	7.89E-03
12	2.72E-02	-1.18E-02	7.96E-03	12	-2.67E-02	1.16E-02	7.87E-03
13	5.46E-02	1.48E-02	1.02E-02	13	-5.36E-02	-1.46E-02	9.88E-03
14	5.45E-02	4.85E-03	9.91E-03	14	-5.34E-02	-4.81E-03	9.85E-03
15	5.45E-02	-4.85E-03	9.91E-03	15	-5.34E-02	4.81E-03	9.85E-03
16	5.46E-02	-1.48E-02	1.02E-02	16	-5.36E-02	1.46E-02	9.88E-03
17	8.61E-02	1.58E-02	1.07E-02	17	-8.48E-02	-1.57E-02	1.10E-02
18	8.60E-02	5.23E-03	1.07E-02	18	-8.44E-02	-5.13E-03	1.04E-02
19	8.60E-02	-5.23E-03	1.07E-02	19	-8.44E-02	5.13E-03	1.04E-02
20	8.61E-02	-1.58E-02	1.07E-02	20	-8.48E-02	1.57E-02	1.10E-02

3.2. Shear Wall-Frame Structure

To compare the shear wall-frame structure of the YAY2020 program with Sap2000, a 3-storey 4-span system is taken as shown in Figure 4. All Storey heights are $h = 350\ cm$ and span distances $L = 300\ cm$. Shear walls are located between the 3rd and 4th axes in the x-direction. All Frame elements are square in shape and have the same material properties. Frame elements cross-sectional area $A = 2500\ cm^2$, Modulus of elasticity $E = 13025000\ N/cm^2$, a moment of inertia $I = 520833.33\ cm^4$ and mass per unit volume $\rho = 0.00025\ kg/cm^3$. Shear wall elements have the same modulus of elasticity and mass per unit volume as

frame elements. Shear wall elements thickness $t = 50\ cm$ and Poisson ratio $\upsilon = 0.2$. $1000\ N$ single loads applied on the top-left and top-right points of the structure.

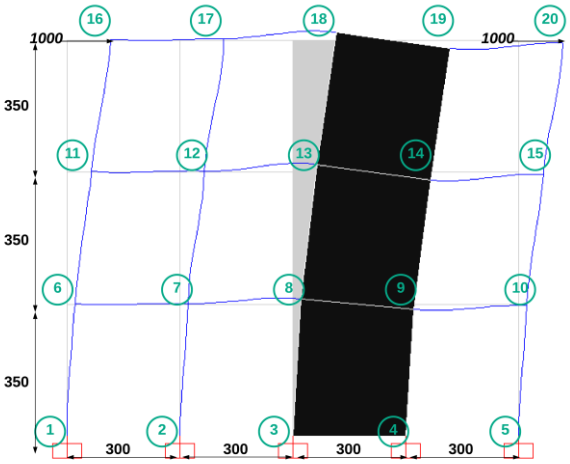

Figure 4. Shear Wall-Frame Structure

System joints deformations values obtained with YAY2020 compared with SAP2000 are given in Table 3. Some results in YAY2020 have a different signal from the SAP2000 results due to the sign notation used in the programs. System support reactions are given in Table 4. Selected five columns' and beams' end forces calculated by YAY2020 and SAP2000 are given in Tables 5 and 6.

Table 3. Joints displacements of Shear Wall-Frame Structure by YAY2020 and Sap2000

SAP2000 - Joint displacements				YAY2020 - Joint displacements			
Joint	**U1**	**U3**	**R2**	**Joint**	**U1**	**U3**	**R2**
No	**cm**	**cm**	**Radius**	**No**	**cm**	**cm**	**Radius**
1	0	0	0	1	0	0	0
2	0	0	0	2	0	0	0
3	0	0	0	3	0	0	0
4	0	0	0	4	0	0	0
5	0	0	0	5	0	0	0
6	1.96E-04	1.50E-05	6.40E-07	6	1.88E-04	1.48E-05	-6.15E-07
7	1.95E-04	1.90E-05	2.09E-07	7	1.87E-04	1.93E-05	-1.82E-07
8	1.91E-04	1.39E-04	9.28E-07	8	1.84E-04	1.33E-04	-9.35E-07
9	1.91E-04	-1.38E-04	9.31E-07	9	1.83E-04	-1.33E-04	-9.35E-07
10	1.93E-04	-3.40E-05	3.77E-07	10	1.85E-04	-3.53E-05	-3.46E-07
11	6.26E-04	2.50E-05	1.03E-06	11	6.00E-04	2.57E-05	-9.83E-07
12	6.24E-04	3.30E-05	3.97E-07	12	5.98E-04	3.35E-05	-3.61E-07
13	6.20E-04	2.06E-04	1.37E-06	13	5.94E-04	1.96E-04	-1.33E-06
14	6.20E-04	-2.05E-04	1.36E-06	14	5.94E-04	-1.95E-04	-1.33E-06
15	6.22E-04	-6.00E-05	7.34E-07	15	5.96E-04	-6.12E-05	-6.95E-07
16	1.20E-03	2.90E-05	8.82E-07	16	1.14E-03	2.97E-05	-8.44E-07
17	1.16E-03	4.00E-05	-6.83E-08	17	1.11E-03	4.09E-05	8.85E-08
18	1.14E-03	2.20E-04	1.41E-06	18	1.09E-03	2.07E-04	-1.34E-06
19	1.14E-03	-2.19E-04	1.43E-06	19	1.09E-03	-2.08E-04	-1.37E-06
20	1.17E-03	-7.10E-05	1.67E-07	20	1.12E-03	-7.29E-05	-1.41E-07

Table 4. Support reactions of Shear Wall-Frame Structure by YAY2020 and Sap2000

SAP2000 - Support reactions				YAY2020 - Support reactions			
Joint	**F1**	**F2**	**M2**	**Joint**	**F1**	**F2**	**M2**
No	**cm**	**cm**	**N-cm**	**No**	**N**	**N**	**N-cm**
1	-36.69	-331.28	-9446.43	1	-37.3408	-337.14	9445.47
2	-69.26	-425.07	-13110.66	2	-72.0459	-437.871	13470
3	-921.3	-3867.53	-59314.1	3	-915.959	-4627.6	-65504.4
4	-917.18	3842.16	-58652.07	4	-916.836	4600.8	-64137.4
5	-55.58	781.72	-11509.42	5	-57.8184	801.811	11753.8

Table 5. Selected five columns' end forces calculated of Shear Wall-Frame Structure by YAY2020 and Sap2000

Column	YAY2020 - Column end forces						SAP2000 - Column end forces					
	i end			j end			i end			j end		
	P	V2	M3	P	V2	M3	P	V2	M3	P	V2	M3
No	N	N	N-cm	N	N	N-cm	N	N	N-cm	N	N	N-cm
1	-337.14	37.34	9445.47	337.14	-37.34	3623.81	331.28	36.69	9446.43	331.28	36.69	-3393.52
3	801.81	57.82	11753.80	-801.81	-57.82	8482.65	-781.72	55.58	11509.42	-781.72	55.58	-7942.00
5	-321.95	146.50	26485.00	321.95	-146.50	24790.60	316.64	141.61	25669.83	316.64	141.61	-23893.73
7	-91.67	104.23	17582.50	91.67	-104.23	18899.30	90.64	103.66	17438.36	90.64	103.66	-18844.01
9	266.45	174.46	27904.40	-266.45	-174.46	33155.50	-263.86	171.06	27251.77	-263.86	171.06	-32619.40

Table 6. Selected five beams' end forces calculated of Shear Wall-Frame Structure by YAY2020 and Sap2000

Beam	YAY2020 - Beam end forces						SAP2000 - Beam end forces					
	i end			j end			i end			j end		
	P	V2	M3	P	V2	M3	P	V2	M3	P	V2	M3
No	N	N	N-cm	N	N	N-cm	N	N	N-cm	N	N	N-cm
1	23.91	-91.30	-16084.80	-23.91	91.30	-11304.10	-23.71	89.60	15815.36	-23.71	89.60	-11064.86
3	-48.39	-213.19	-35232.20	48.39	213.19	-28723.60	47.19	204.53	33741.36	47.19	204.53	-27618.59
5	111.91	-306.86	-40670.80	-111.91	306.86	-51387.00	-112.63	298.83	39456.15	-112.63	298.83	-50193.37
7	895.77	-91.67	-18899.30	-895.77	91.67	-8602.10	-896.34	90.64	18844.01	-896.34	90.64	-8346.82
9	-825.54	-266.45	-46779.80	825.54	266.45	-33155.50	828.94	263.86	46537.54	828.94	263.86	-32619.40

4. CONCLUSIONS

Frame elements give roughly the same results in both Sap2000 and YAY2020 because of formulas used in these programs are close to each other.

In rectangular elements (which are used in shear-wall elements), in Sap2000 if only the plane stress element is used, the rotation values will be neglected, resulting in unreal element behavior. In the rectangular element formulation which used in YAY2020 program, the lateral and vertical displacements in addition to the rotation of element joints can be calculated. This formulation is closer to the real element behavior.

To obtain the real behavior of the rectangular element in Sap2000, it is necessary to use a large number of meshing shell elements instead of using a plane stress element. However, this takes a long time to analyze the system and increases the computational burden.

YAY2020 rectangular elements analysis results are closer to Timoshenko's analytical solution than those obtained with Sap2000.

In the calculation of support reactions and end forces of elements, both YAY2020 and SAP2000 give close results. However, some results show different signs because of the sign notation acceptance of the used formulation.

The results of joints deformations in YAY2020 program are lower than those of SAP2000 because the stiffness matrix in YAY2020 program is bigger than the other used in SAP2000.

ACKNOWLEDGMENTS

This paper is prepared using the data of the master thesis titled “Behavior of High-Rise Planar Shear-Wall-Frame Structures under Dynamic Loads” by Ragib Sabah. The thesis is supervised by Prof. Namik Kemal Öztorun. The other author, Baris Sayin, put a lot of effort into organizing data, conceptualizing, methodology, writing-reviewing, and editing.

REFERENCES

[1]. Öztorun N.K., (2006) “*A rectangular finite element formulation”,* Finite Elem. Anal.Des. 42 (12), 1031–1052.

[2]. Öztorun N.K., (2007) *Reply to “discussion on: ‘Rectangular finite element formulation”*, Finite Elements in Analysis and Design 43 (2007) 733 – 736.

[3]. Ed. Wilson, (2002) *“Three-Dimensional Static and Dynamic Analysis of Structures”*, Computers and Structures Inc. University of California, Berkeley, USA.

[4]. Ed. Wilson, (1991) *“Computer Assisted Learning of Static and Dynamic Analysis of Structural Systems user’s manual”*, University of California, Berkeley, USA.

[5]. Ed. Wilson, E. Ray, W. Clough, (1952) *“Dynamic response by step-by-step matrix analysis”*, Computers in Civil Engineering Symposium.

[6]. Computers and Structures, Inc. (2016*) "SAP2000 CSI Analysis Reference Manual, Structural Analysis Program"*, California, Berkeley, USA.
[7]. Bathe K.J. Wilson, E. L. (1976) *"Numerical Methods in Finite Element Analysis"*, Prentice-Hall, New Jersey, 528s.
[8]. Bathe K.J., (2014*) "Finite Element Procedures Second edition"*, Prentice-Hall, Pearson Education, Inc. 978-0-9790049-5-7.
[9]. M. Turner, R. Clough, H. Martin, L. Topp, (1954) *"Stiffness and Deflection Analysis of Complex Structures"*, New York, January 25-29.
[10]. D. V. Griffiths, (1994) *"Stiffness matrix of the four-node quadrilateral element in the closed-form"*, Colorado School of Mines, Golden, Colorado 80401, USA.
[11]. E., Öztorun, 2020, *"Köprülerin Deprem Etkileri Altındaki Davranışları"*, Thesis (PhD), Istanbul University.
[12]. E., Öztorun, C., Alhan, S., Erdem ve N.K., Öztorun, 2017, *"Köprülerin Deprem Etkileri Altındaki Davranışları"*, 20. Ulusal Mekanik Kongresi, 05-09 Eylül. 2017, Uludağ Üniversitesi, Bursa, 938-947.
[13]. Adams J.C., Brainerd W.S., Martin J.T., Smith B.T., Wagener J.L. (1992) *" Fortran 90 Handbook"*. Library of Congress Catalog Card Number 91-77211.
[14]. William H. Press, Saul A. Teukolsky, William T. Vetterling, Brain P.Flannery, (1996) *"Fortran 90 Numerical Recipes"*. V.2 Library of Congress catalog card number 96-5567.
[15]. Timoshenko S., Goodier J.N., (1951) *"Theory of Elasticity"*, McGraw-Hill, New York.

BIOGRAPHY

Ragıb Sabah was born in 1993. He is also known as Ragheb Alsabbagh. In 2014 Mr. Sabah started his undergraduate degree in the Civil Engineering Department at Pamukkale University in Turkey and graduated in 2018 with high honor degree. In 2021, Mr. Sabah completed his master's degree at Istanbul University-Cerrahpasa. He is interested in computational sciences, applicable technology, and management areas.