STRUCTURAL ANALYSIS PROGRAM USING DIRECT STIFFNESS METHOD FOR THE DESIGN OF REINFORCED CONCRETE COLUMN

AHMAD FAZA AZMI

Engineering Faculty, Bhayangkara University Of Surabaya

Jalan Ahmad Yani 114, Surabaya 60231, Indonesia

E-mail: Faza@ubhara.ac.id

ABSTRACT

Most of the analyzes and calculations in civil engineering have been using computer. But many commercial structural analysis programs are relatively expensive. Hence one of the alternative way is using a self developed structural analysis program. The computer program is constructed using Visual Basic 6 programming language. Using direct stiffness method and theory of ACI 318-02, a numerical procedure along with a computer program was developed for the structural analysis and design of reinforced concrete member. From the direct stiffness method,in terms of structural analysis, the output that obtained from the program are internal element forces and support reactions node displacements. From ACI 318-02 codes, in terms of reinforced concrete member analysis, the program generate the amount of longitudinal and lateral reinforcement of concrete member based on the output in the previous structure analysis program. The program also reveals the biaxial bending-axial interaction diagrams in each direction of rectangular and circular column. The interaction diagram of the columns is a visual representation to determine the maximum axial load and moment exceeded the capacity of the column.

Keywords: *Structural, Analysis, Reinforced, Concrete, Column*

1. INTRODUCTION

Structural analysis and design is known to human beings since early civilizations. Structural analysis and design of a reinforced concrete column with biaxial bending and axial force is complicated without the assistance of a computer program. It takes a long time to calculate manually.

In fact, many commercial structural analysis programs are relatively expensive. Hence one of the alternative way is using a self developed structural analysis program. The developed structural analysis program is expected to be able to analyze the structure to produce internal element forces and support reactions node displacement. The internal forces is used to generate the amount of longitudinal and lateral reinforcement of reinforced concrete column. The computer program is constructed using Visual Basic programming language.

To analyze the structure so as to produce internal element forces, a direct stiffness method which is part of the finite element method is applied. There are several reasons that the direct stiffness method is used. The first is that the approach can be thorough and applicable to all types of structures. The second reason is that this approach is an efficient tool in describing the various steps in the analysis so that these steps can be easily programmed on the computer. The use of a matrix in this method is necessary to obtain internal element forces on computer computations, since a large number of numbers can be manipulated simply and efficiently [3].

The internal element forces that has been produced in the calculation of the direct stiffness method is then used as input data to calculate the number of required reinforcement of reinforced concrete column based on the calculation of reinforced concrete reinforcement in accordance with applicable regulations (ACI 318).

2. DIRECT STIFFNESS METHOD

Direct stiffness method is part of the finite element method. The direct stiffness method is particularly suited for computer-automated analysis of complex structure. The stiffness matrix relates forces acting at the nodes to displacements of the nodes.

The exposition is done by following the direct stiffness matrix steps applied to a simple plane structure. The method has two major stages (breakdown and assembly).

2.1 Degree of Freedom

Each type of element has a particular number and type of degrees of freedom. Each element is analyzed separately to generate member stiffness equations. The displacement and forces are related through the element stiffness matrix which depends on the configuration and properties of the element.

A truss element is able to proceed forces in tension or compression only. It implies each node has two degrees of freedom (DOF), vertical and horizontal displacement, in two dimensions. The resulting equation carries a four by four stiffness matrix.

$$\begin{bmatrix} f_{x1} \\ f_{y1} \\ f_{x2} \\ f_{y2} \end{bmatrix} = \begin{bmatrix} k_{11} & k_{12} & k_{13} & k_{14} \\ k_{21} & k_{22} & k_{23} & k_{24} \\ k_{31} & k_{32} & k_{33} & k_{34} \\ k_{41} & k_{42} & k_{43} & k_{44} \end{bmatrix} \begin{bmatrix} u_{x1} \\ u_{y1} \\ u_{x2} \\ u_{y2} \end{bmatrix} \tag{1}$$

A frame element can restrain bending moments in addition to compression and tension. It implies each node has three degrees of freedom, vertical displacement, horizontal displacement, and in plane rotation. The resulting equation carries a six by six stiffness matrix.

$$\begin{bmatrix} f_{x1} \\ f_{y1} \\ m_{z1} \\ f_{x2} \\ f_{y2} \\ m_{x2} \end{bmatrix} = \begin{bmatrix} k_{11} & k_{12} & k_{13} & k_{14} & k_{15} & k_{16} \\ k_{21} & k_{22} & k_{23} & k_{24} & k_{25} & k_{26} \\ k_{31} & k_{32} & k_{33} & k_{34} & k_{35} & k_{36} \\ k_{41} & k_{42} & k_{43} & k_{44} & k_{45} & k_{46} \\ k_{51} & k_{52} & k_{53} & k_{54} & k_{55} & k_{56} \\ k_{61} & k_{62} & k_{63} & k_{64} & k_{65} & k_{66} \end{bmatrix} = \begin{bmatrix} u_{x1} \\ u_{y1} \\ \theta_{z1} \\ u_{x2} \\ u_{y2} \\ \theta_{z2} \end{bmatrix} \tag{2}$$

2.2 Stiffness Matrix

In the matrix stiffness method, it is necessary to construct a stiffness matrix for each element [k] before obtaining a global stiffness matrix [K]. Several factor that affect the value of stiffness coefficient in 3D space frames are axial deformation, torsional deformation, flexural deformation, and shear deformation.

$$\begin{bmatrix} f_{x1} \\ f_{y1} \\ f_{z1} \\ m_{x1} \\ m_{y1} \\ m_{z1} \\ f_{x2} \\ f_{y2} \\ f_{z2} \\ m_{x2} \\ m_{y2} \\ m_{z2} \end{bmatrix} = \begin{bmatrix} k_a & 0 & 0 & 0 & 0 & 0 & -k_a & 0 & 0 & 0 & 0 & 0 \\ 0 & k_{z1} & 0 & 0 & 0 & k_{z2} & 0 & -k_{z1} & 0 & 0 & 0 & k_{z2} \\ 0 & 0 & k_{y1} & 0 & -k_{y2} & 0 & 0 & 0 & -k_{y1} & 0 & -k_{y2} & 0 \\ 0 & 0 & 0 & k_t & 0 & 0 & 0 & 0 & 0 & -k_t & 0 & 0 \\ 0 & 0 & -k_{y2} & 0 & k_{y3} & 0 & 0 & 0 & k_{y2} & 0 & k_{y4} & 0 \\ 0 & k_{z2} & 0 & 0 & 0 & k_{z3} & 0 & -k_{z2} & 0 & 0 & 0 & k_{z4} \\ -k_a & 0 & 0 & 0 & 0 & 0 & k_a & 0 & 0 & 0 & 0 & 0 \\ 0 & -k_{z1} & 0 & 0 & 0 & -k_{z2} & 0 & k_{z1} & 0 & 0 & 0 & -k_{z2} \\ 0 & 0 & -k_{y1} & 0 & k_{y2} & 0 & 0 & 0 & k_{y1} & 0 & k_{y2} & 0 \\ 0 & 0 & 0 & -k_t & 0 & 0 & 0 & 0 & 0 & k_t & 0 & 0 \\ 0 & 0 & -k_{y2} & 0 & k_{y4} & 0 & 0 & 0 & k_{y2} & 0 & k_{y3} & 0 \\ 0 & k_{z2} & 0 & 0 & 0 & k_{z4} & 0 & -k_{z2} & 0 & 0 & 0 & k_{z3} \end{bmatrix} = \begin{bmatrix} u_{x1} \\ u_{y1} \\ u_{z1} \\ \theta_{x1} \\ \theta_{y1} \\ \theta_{z1} \\ u_{x2} \\ u_{y2} \\ u_{z2} \\ \theta_{x2} \\ \theta_{y2} \\ \theta_{z2} \end{bmatrix}$$

Where :

$$\begin{gathered} k_a = \frac{EA}{L};\ k_{z1} = \frac{12EI_z}{L^3};\ k_{z3} = \frac{4EI_z}{L};\ k_{y1} = \frac{12EI_y}{L^3};\ k_{y3} = \frac{4EI_y}{L} \\ k_t = \frac{GJ}{L};\ k_{z2} = \frac{6EI_x}{L^2};\ k_{z4} = \frac{2EI_z}{L};\ k_{y2} = \frac{6EI_y}{L^2};\ k_{y4} = \frac{2EI_y}{L} \end{gathered} \tag{3}$$

2.3 Transformation Matrix

When [k] is the local stiffness matrix of the element and [R] is the transformation matrix of the element, then the global stiffness matrix [K] is expressed as follows :

$$[K]=[R]^T[k][R] \tag{4}$$

$$[R]=\left[\begin{array}{c|c|c|c} \lambda_{3x3} & & & \\ \hline & \lambda_{3x3} & & \\ \hline & & \lambda_{3x3} & \\ \hline & & & \lambda_{3x3} \end{array}\right] \tag{5}$$

$$[\lambda]=\begin{bmatrix} l_{xX} & l_{xY} & l_{xZ} \\ l_{yX} & l_{yY} & l_{yZ} \\ l_{zX} & l_{zY} & l_{zZ} \end{bmatrix} \tag{6}$$

$$X,rotation,3D=\begin{bmatrix} 1 & 0 & 0 & 0 \\ 0 & \cos\theta & -\sin\theta & 0 \\ 0 & \sin\theta & \cos\theta & 0 \\ 0 & 0 & 0 & 1 \end{bmatrix};Y,rotation,3D=\begin{bmatrix} \cos\theta & 0 & \sin\theta & 0 \\ 0 & 1 & 0 & 0 \\ -\sin\theta & 0 & \cos\theta & 0 \\ 0 & 0 & 0 & 1 \end{bmatrix}$$
$$Z,rotation,3D=\begin{bmatrix} \cos\theta & -\sin\theta & 0 & 0 \\ \sin\theta & \cos\theta & 0 & 0 \\ 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & 1 \end{bmatrix} \tag{7}$$

2.4 Force and Load

For nodal force in global axis direction, the force generates the load matrix directly to the corresponding displacement position occurs (Eqs.8). For the uniform load, it is necessary to find the equivalent load value for each nodal (Eqs.9). Furthermore equilibrium equation structure is written in Eqs.10 and Eqs.11.

$$[f]=\left[f_x f_y f_z m_x m_y m_z\right]^T \tag{8}$$

$$[p_s]=\left[f_{xeq} f_{yeq} f_{zeq} m_{xeq} m_{yeq} m_{zeq}\right]^T \tag{9}$$

$$\{U\}=[K]^{-1}\{Ps\} \tag{10}$$

$$\{f\}=\{f_0\}+[k]\{U\} \tag{11}$$

3. REINFORCED CONCRETE COLUMN

The method used in analyzing biaxial bending columns is by the exact analytical method. Biaxial bending columns are columns at the corners of the building or may also occur due to load imbalance. The strain distribution is considered linear. The following is a calculation phase to generate P-M interaction diagrams of the column:

1. Maximum axial load

$$P_0 = 0.85 f_c' A_g - A_{st} + f_y A_{st}$$
$$Pn = \phi P_0 \tag{12}$$

2. Determine the balance point

$$\varepsilon_s = \frac{C_b}{0.003} = \frac{d}{0.003 + f_y / E_s} \tag{13}$$

3. Determine the strain of the steel

$$\varepsilon'_{sn} = \left[\frac{C_b - d'}{C_b}\right]\varepsilon_{cu}$$
$$\varepsilon_{sn} = \left[\frac{d' - C_b}{C_b}\right]\varepsilon_{cu} \quad (14)$$

4. Determine the stress in the steel

$$f_{sn} = E_s \varepsilon_{sn} \quad (15)$$

5. Compute the forces and in the column

$$C_c = 0.85 f_c' b \beta_1 c$$
$$C_{sn} = A_{sn}(f_{sn} - 0.85 f_c')$$
$$T_s = A_s f_y \quad (16)$$
$$P_n = C_c + C_{sn} + T_s$$
$$M_n = C_c\left(\frac{h}{2} - \frac{a}{2}\right) + C_{sn}\left(\frac{h}{2} - d'\right) + T\left(d - \frac{h}{2}\right)$$

6. Calculate the other points by determining the other value of c
Each different c values, gives a different axial and moment values, as shown in figure 1.

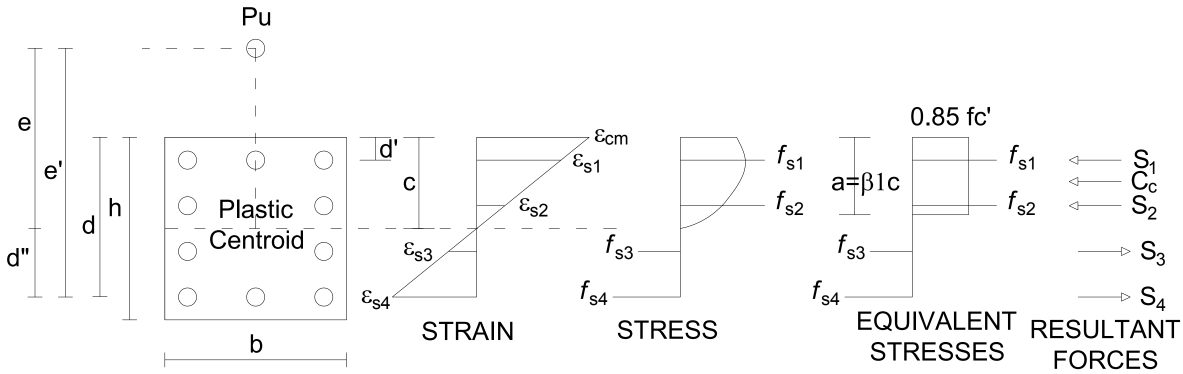

Figure 1. Eccentrically Loaded Column Section at the Ultimate Load (Park and Paulay, 1974)

4. RESULT AND COMPARISON

To examine the results and accuracy of the structural analysis program in performing the calculation processs, it is necessary to compare the program output to other program such as SAP2000 v.14.

4.1 Structural Analysis

The case example is a simple portal with two fixed restraints. The load assigned is a uniform load on the beam as shown in Figure 2. The concrete material assigned as follows :

Modulus of elasticity, E = 2.531×10^9 kg/m^2
Shear modulus, G = 1054583333 kg/m^2
Compressive strength f_c' = 27.8888062786627 MPa
Poisson's ratio = 0.2
β_1 = 0.85
Beam section = 0.4 m x 0.4 m
Beam length = 5 m
Column section = 0.5 m x 0.5 m
Column length = 5 m
Uniform Load on beam = 1000 kg/m

 ISSN : 2528 - 0260

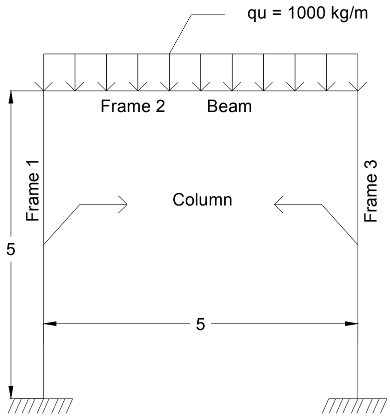

Figure 2 Simple Portal with Two Fixed Restraints and Uniform Load on Beam

Figure 3. Element forces of frame 1,2 and 3

Table 1. Comparison of Program and SAP2000 v.14

Structural Analysis			
Frame 1	**Element forces**	**Program**	**SAP2000 v.14**
	f_{x1} (kg)	2500	2500
	f_{y1} (kg)	-511.68	-511.69
	m_{z1} (kgm)	-837.19	-837.19
	f_{x2} (kg)	-2500	-2500
	f_{y2} (kg)	511.68	511.69
	m_{z2} (kgm)	-1721.23	-1721.23
Frame 2	**Element forces**	**Program**	**SAP2000 v.14**
	f_{x1} (kg)	511.68	511.69
	f_{y1} (kg)	2500	2500
	m_{z1} (kgm)	1721.23	1721.23
	f_{x2} (kg)	-511.68	-511.69
	f_{y2} (kg)	2500	2500
	m_{z2} (kgm)	-1721.233	-1721.23
Frame 3	**Element forces**	**Program**	**SAP2000 v.14**
	f_{x1} (kg)	2500	2500
	f_{y1} (kg)	511.68	511.69
	m_{z1} (kgm)	1721.23	1721.23
	f_{x2} (kg)	-2500	-2500
	f_{y2} (kg)	-511.68	-511.69
	m_{z2} (kgm)	837.19	837.19

4.2 Reinforced Concrete Column Analysis

The structural analysis output (Figure 3 and Table 1) is used to check the column reinforcement. The result of this reinforced concrete column analysis is a column interaction diagram (P-M). The data used in the column for comparison with PCAColumn is shown in Figure 4. And the output of reinforced concrete column analysis is revealed in Figure 5, furthermore the comparison of reinforced concrete column interaction diagrams between the program and PCACol is presented in Figure 6 and Table 2.

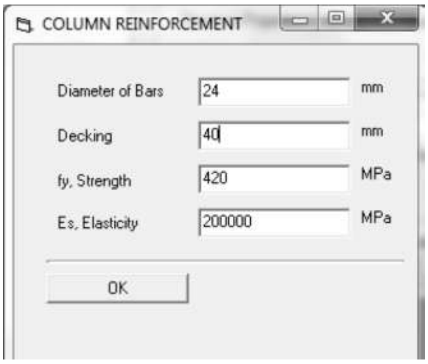

Figure 4 Column Reinforcement Properties

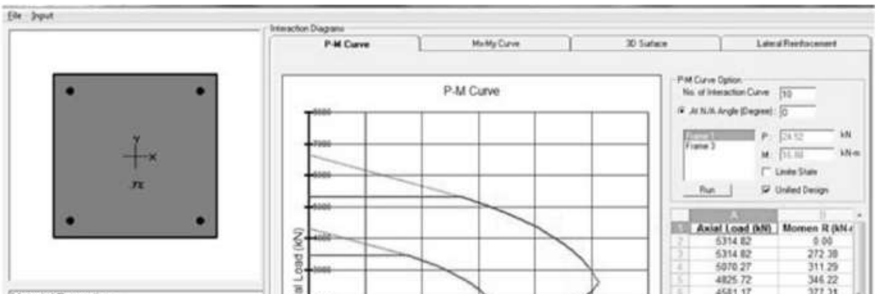

Figure 5 Output of Reinforced Concrete Column Analysis

 ISSN : 2528 - 0260

Table 2. Comparison of Reinforced Concrete Column Interaction Diagrams

Program		PCACol		Difference	
P (kN)	M (kN m)	P (kN)	M (kN m)	P (kN)	M (kN m)
3454.64	0	3454.60	0	0.04	0
3454.64	177.05	3454.60	177.00	0.04	0.05
3163.21	221.44	3163.00	221.00	0.21	0.44
2871.79	257.39	2872.00	258.00	0.21	0.61
2580.36	285.56	2580.00	285.00	0.36	0.56
2288.94	306.80	2289.00	307.00	0.06	0.20
1997.51	322.18	1998.00	322.00	0.49	0.18
1706.09	333.06	1706.00	338.00	0.09	4.94
1612.20	363.10	1612.00	363.00	0.20	0.10
1517.39	396.04	1517.00	397.00	0.39	0.96
1181.14	361.76	1181.00	361.00	0.14	0.76
787.42	299.79	787.00	300.00	0.42	0.21
393.71	227.54	394.00	228.00	0.29	0.46
0	148.90	0	149.00	0	0.10
-62.18	136.21	-62.00	137.00	0.18	0.79
-124.37	123.50	-124.00	124.00	0.37	0.50
-186.55	110.77	-187.00	110.00	0.45	0.77
-248.73	98.04	-249.00	99.00	0.27	0.96
-310.91	85.31	-311.00	86.00	0.09	0.69
-373.10	72.58	-373.00	73.00	0.10	0.42
-435.28	59.28	-435.00	60.00	0.28	0.72
-497.46	45.01	-497.00	45.00	0.46	0.01
-559.65	30.37	-560.00	30.00	0.35	0.37
-621.83	15.36	-622.00	16.00	0.17	0.64
-684.01	0	-684.00	0	0.01	0

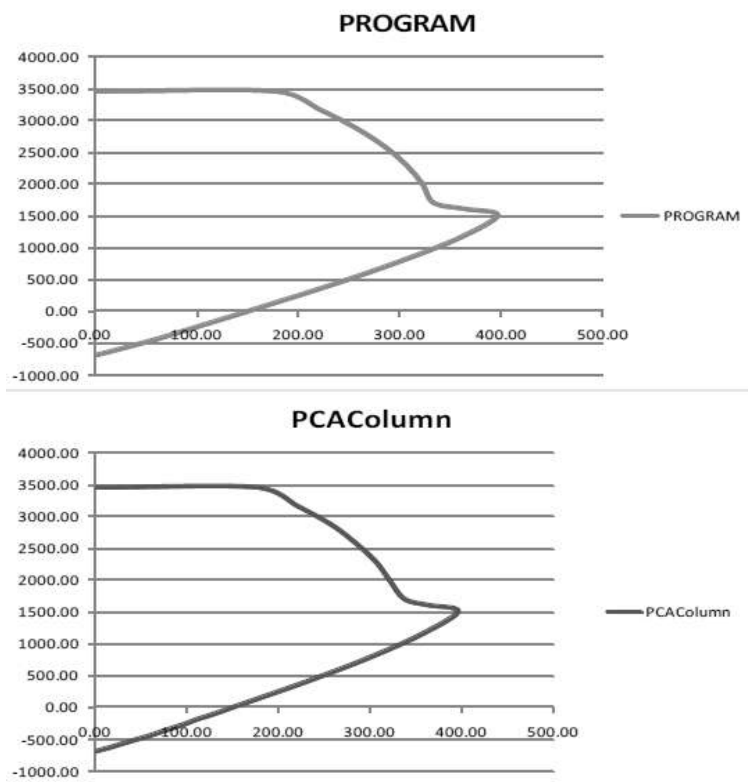

Figure 6. Comparison of Reinforced Concrete Column Interaction Diagrams

5. CONCLUSION

After comparing the results of the calculation of the program and SAP2000 and PCACol in some cases, it can be concluded as follows:

1. Structural analysis program using direct stiffness method can be used to obtain displacement, joint reaction and element forces of the plane frame structure
2. Structural analysis program using direct stiffness method has same output (Displacement, joint reaction, and element forces) as SAP2000 v.14
3. Structural analysis program using direct stiffness method can be performed to obtain the required longitudinal reinforcement on the column element in a complex structure.
4. Structural analysis program using direct stiffness method has same output (Axial and Moment) as PCACol

REFERENCES

[1] ACI, (2011). *Building Code Requirements for Structural Concrete and Commentary*, ACI 318-11, ACI Committee 318, American Concrete Institute, Farmington Hills, Michigan.

[2] Cook, R.D.; Malkus, D.S.; Plesha, M.E.; and Witt, R.J., (2001). *Concepts and Applications of Finite Element Analysis*. 4th edition, John Wiley & Sons, Inc ., New York.

[3] Gere, J.M., and Weaver, W. (1966). *Matrix Algebra for Engineers*, Van Nostrand, Princeton, NJ

[4] Gavin, H., (2011). *Matrix Structural Analysis*, Department of Civil and Environmental Engineering, Duke University, Durham.

[5] Logan, D., (2007). *A First Course in the Finite Element Method*, 4th edition, Thomson, USA.

[6] Nawy, E. G., (2003). *Reinforced Concrete – A Fundamental Approach*, 5th edition, Prentice Hall, Upper Saddle River, NJ.

[7] Park. R; Paulay. T., (1975). *Reinforced Concrete Structures*, John Wiley and Sons, New York, USA.

[8] Reddy.J.N., (2005). *Solutions Manual For An Introduction to the Finite Element Method*, 3rd edition. McGraw-Hill, New York.