IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)
e-ISSN: 2278-1684,p-ISSN: 2320-334X, Volume 15, Issue 1 Ver. II (Jan. - Feb. 2018), PP 01-11
www.iosrjournals.org

Analysis Of Gradually Built Structural System

Binsar H. Hariandja[1]
[1](*Professor of Civil Engineering, Bandung Institute of Technology, Bandung, Indonesia)*
Corresponding Auther: Binsar H. Hariandja

Abstract:*The paper deals withthe analysis of structural system that built gradually such that the structural stiffness varies with time of construction. The analysis of such structural system is carried out by applying finite element method. First, all of the finite elements are included in the computation of number of degrees of freedom. Then, the analysis is initiated by considering all of elements and the numbering of degrees of freedom are carried out. Assembled elements are denoted by assigning the value of modulus elasticity, and absent elements are denoted by assigning zero modulus of elasticity. Nodal degrees of freedom that are located in absent elements are suppressed from the global equilibrium equation. Hence, the analysis is performed with changing global structural stiffness. A computer package program in FORTRAN is written for the analysis. The new package program is applied in the analysis of several study cases.*
Keywords:*finite element method, gradually built structural system, suppressing of inactive degrees of freedom.*

Date of Submission: 03-01-2018 Date of acceptance: 18-01-2018

I. Introduction

A structural system ussually is built by assembling its components gradually. Hence, the structural total number of degrees of freedom and the structural stiffness vary according to the assembling process. For example, in case of multi-storey building, the system is built gradually by positioning columns of first storey and then beams of the next floor above. The same process is carried out for storeys. This case is particularly true for the case of structural system made of precast components.

Before the structural system is completed to its final configuration, the temporary system has to be capable of sustaining external forces. In this case, the total number and the numbering of degrees of freedom and hence the structural stiffness vary with time. So, the total number and numbering of degrees of freedom have to be carried out for each construction step.

The paper proposes a method in which the numbering of degrees of freedom is carried out for completed form of the structure. It means that the total number and numbering of structural degrees of freedom are kept constant during all construction steps. In a particular construction step, assembled components are indentified and given real values of modulus elasticity, and absent components are indentified and given zero values of modulus elasticity. By doing so, total number and numbering of degrees of freedom are kept constant for all construction steps. After stiffness and load assemblage, row and column members of stiffness matrix pertaining to the degrees of freedom belong to absent components would be all zero and this will make the stiffness matrix ill condition. To remedy this problem, the degrees of freedom may be suppressed from the equilibrium equation. If the suppressed degree of freedom is inside of the displacement vector, the suppression of the degree of freedom will decrease the total number of degrees of freedom and the new numbering of degrees of freedom has to be carried out. This kind of remedial step may take long execution time.

Another way to handle inactive degree of freedom is to retain the degree of freedom in the matrix form of equilibrium equation. All elements of row and column of stiffness matrix pertaining to the degree of freedom are set to zero, and the diagonal element of the stiffness matrix pertaining to the degree of freedom is set to some value. The element of load vector pertaining to the degree of freedom is set to zero. Therefore, the total number and the numbering of degrees of freedom are kept constant and the solution of global equilibrium equation results in zero displacements of inactive degrees of freedom.

II. Finite Element Method

To begin with, in this chapter, finite element method is resumed by formulating stiffness matrices and load vectors of some types of elements; i.e., grid element and eight load isoparametric plane element [3,4]. The development of the two elements are described in the following.

2.1 Grid Element

Grid element model is depicted in Fig. 1. The element consisting of two nodes, each node has one torsional rotation, and one bending rotation. That three displacements correspond to shear, torsion and bending moments. Hence, the element has six degrees of freedomand the matrix reads

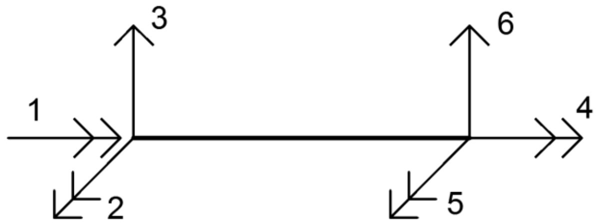


Figure1: Grid Element

$$[k] = \begin{bmatrix} \frac{EI_{\rho\rho}}{2(1+v)L} & 0 & 0 & -\frac{EI_{\rho\rho}}{2(1+v)L} & 0 & 0 \\ & \frac{4EI_{zz}}{L} & \frac{6EI_{zz}}{L^2} & 0 & \frac{2EI_{zz}}{L} & -\frac{6EI_{zz}}{L^2} \\ & & \frac{12EI_{zz}}{L^3} & 0 & \frac{6EI_{zz}}{L^2} & -\frac{12EI_{zz}}{L^3} \\ & & & -\frac{EI_{\rho\rho}}{2(1+v)L} & 0 & 0 \\ & symm. & & & \frac{4EI_{zz}}{L} & -\frac{6EI_{zz}}{L^2} \\ & & & & & \frac{12EI_{zz}}{L^3} \end{bmatrix} \quad (1)$$

For uniform element load q, the element load vector reads

$$\{p\} = \{p_1 \quad p_2 \quad p_3 \quad p_4 \quad p_5 \quad p_6\} \quad (2)$$

in which

$$p_1 = 0; \; p_2 = +{}^{qL^2}/_{12}; \; p_3 = -{}^{qL}/_{2}; \; p_4 = 0; \; p_5 = -{}^{qL^2}/_{12}; \; p_6 = -{}^{qL}/_{2} \quad (3)$$

1.1. Rectangular Bending Element

Rectangular bending element shown in Fig. 2 possessess for corner nodes, each node has three degrees of freedom; i.e., one vertical displacement and two bending rotations. Hence, there are twelve degrees of freedom, arranged in vector form

$$\{u\} = \{w_1 \quad \theta_{x1} \quad \theta_{y1} \quad \cdots \quad \cdots \quad \cdots \quad \cdots \quad \cdots \quad \cdots \quad w_4 \quad \theta_{x4} \quad \theta_{y4}\} \quad (4)$$

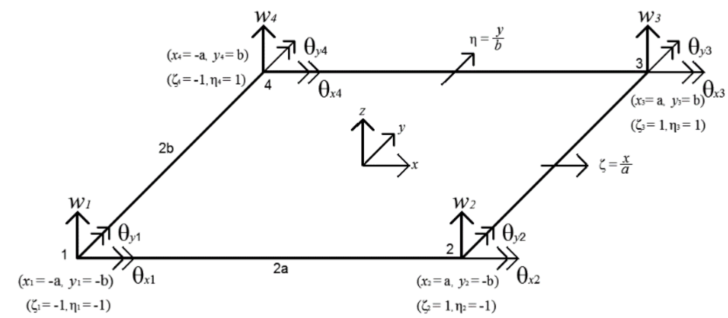


Figure 2: Rectangular Kirchhoff Bending Element

The stiffness matrix of rectangular bending element is derived by means of Kirchhoff bending rule. The form of the stiffness reads

$$[k] = D_b([k_{b1}] \quad [k_{b2}] \quad [k_{b3}] \quad [k_{b4}]) \quad (5)$$

in which

$$D_b = \frac{Eh^3}{12\,(1-v^2)} \quad (6)$$

where υ is the Poisson’s ratio, *E*elasticity modulus and *h*plate thickness. The matrices in Eqn. 3 read

$$[k_{b1}] = \frac{b}{6a^3}\begin{bmatrix} 6 &&&&&&&&&&& \\ 0 & 0 &&&&&&&&&& \\ -6a & 0 & 8a^2 &&&&&&&&& \\ -6 & 0 & 6a & 6 &&&&&&&& \\ 0 & 0 & 0 & 0 & 0 &&&&&&& \\ -6a & 0 & 4a^2 & 6a & 0 & 8a^2 &&&&&& \\ -3 & 0 & 3a & 3 & 0 & 3a & 6 &&&&& \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 &&&& \\ -3a & 0 & 2a^2 & 3a & 0 & 4a^2 & 6a & 0 & 8a^2 &&& \\ 3 & 0 & -3a & -3 & 0 & -3a & -6 & 0 & -6a & 6 && \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & \\ -3a & 0 & 4a^2 & 3a & 0 & 2a^2 & 6a & 0 & 4a^2 & -6a & 0 & 8a^2 \end{bmatrix} \tag{7}$$

$$[k_{b2}] = \frac{a}{6b^3}\begin{bmatrix} 6 &&&&&&&&&&& \\ 6b & 8b^2 &&&&&&&&&& \\ 0 & 0 & 0 &&&&&&&&& \\ 3 & 3b & 0 & 6 &&&&&&&& \\ 3b & 4b^2 & 0 & 6b & 8b^2 &&&&&&& \\ 0 & 0 & 0 & 0 & 0 & 0 &&&&&& \\ -3 & -3b & 0 & -6 & -6b & 0 & 6 &&&&& \\ 3b & 2b^2 & 0 & 6b & 4b^2 & 0 & -6b & 8b^2 &&&& \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 &&& \\ -6 & -6b & 0 & -3 & -3b & 0 & 3 & -3b & 0 & 6 && \\ 6b & 4b^2 & 0 & 3b & 2b^2 & 0 & -3b & 4b^2 & 0 & -6b & 8b^2 & \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix} \tag{8}$$

$$[k_{b3}] = \frac{\upsilon}{2ab}\begin{bmatrix} 1 &&&&&&&&&&& \\ b & 0 &&&&&&&&&& \\ -a & -2ab & 0 &&&&&&&&& \\ -1 & -b & 0 & 1 &&&&&&&& \\ -b & 0 & 0 & b & 0 &&&&&&& \\ 0 & 0 & 0 & a & 2ab & 0 &&&&&& \\ 1 & 0 & 0 & -1 & 0 & -a & 1 &&&&& \\ 0 & 0 & 0 & 0 & 0 & 0 & -b & 0 &&&& \\ 0 & 0 & 0 & -a & 0 & 0 & a & -2ab & 0 &&& \\ -1 & 0 & a & 1 & 0 & 0 & -1 & b & 0 & 1 && \\ 0 & 0 & 0 & 0 & 0 & 0 & b & 0 & 0 & -b & 0 & \\ a & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & -a & 2ab & 0 \end{bmatrix} \tag{9}$$

and

$$[k_{b4}]=\frac{\lambda}{15ab}\begin{bmatrix} 21 & & & & & & & & & & & \\ 3b & 8b^2 & & & & & & & & & & \\ -3a & 0 & 8a^2 & & & & & & & & & \\ -21 & -3b & 3a & 21 & & & & & & & & \\ -3b & -8b^2 & 0 & 3b & 8b^2 & & & & & & & \\ -3a & 0 & -2a^2 & 3a & 0 & 8a^2 & & & & & & \\ 21 & 3b & -3a & -21 & -3b & -3a & 21 & & & & & \\ -3b & 2b^2 & 0 & 3b & -2b^2 & 0 & -3b & 8b^2 & & & & \\ 3a & 0 & 2a^2 & -3a & 0 & -8a^2 & 3a & 0 & 8a^2 & & & \\ -21 & -3b & 3a & 21 & 3b & 3a & -21 & 3b & -3a & 21 & & \\ 3b & -2b^2 & 0 & -3b & 2b^2 & 0 & 3b & -8b^2 & 0 & -3b & 8b^2 & \\ 3a & 0 & -8a^2 & -3a & 0 & 2a^2 & 3a & 0 & -2a^2 & -3a & 0 & 8a^2 \end{bmatrix} \tag{10}$$

in which a is plate length in x direction and b is plate width in y direction. For uniform element load q, the element load vector reads

$$\{p\} = \{p_1 \quad p_2 \quad p_3 \quad \cdots \quad \cdots \quad \cdots \quad \cdots \quad \cdots \quad \cdots \quad p_{10} \quad p_{11} \quad p_{12}\} \tag{11}$$

in which

$$\begin{aligned} &p_1 = +qab^2/24;\ p_2 = -qa^2b/24;\ p_3 = +qab/4; \\ &p_4 = +qab^2/24;\ p_5 = +qa^2b/24;\ p_6 = +qab/4; \\ &p_7 = -qab^2/24;\ p_8 = +qa^2b/24;\ p_9 = +qab/4; \\ &p_{10} = -qab^2/24;\ p_{11} = -qa^2b/24;\ p_{12} = +qab/4 \end{aligned} \tag{12}$$

2.2 Plane Eight Node Isoparametric Element

The formulation of grid and rectangular plane bending elements are described in previous sections. In this section the formulation of plane eight node isoparametric element will be discussed. Isoparametric formulation enables the use of distorted shape of element.Element model shown in Fig. 3 possesses eight nodes, four corner nodes and four mid-side nodes. Hence, the element has eight nodes, each node has displacements in *x* and *y* direction and the element has sixteen degrees of freedom.

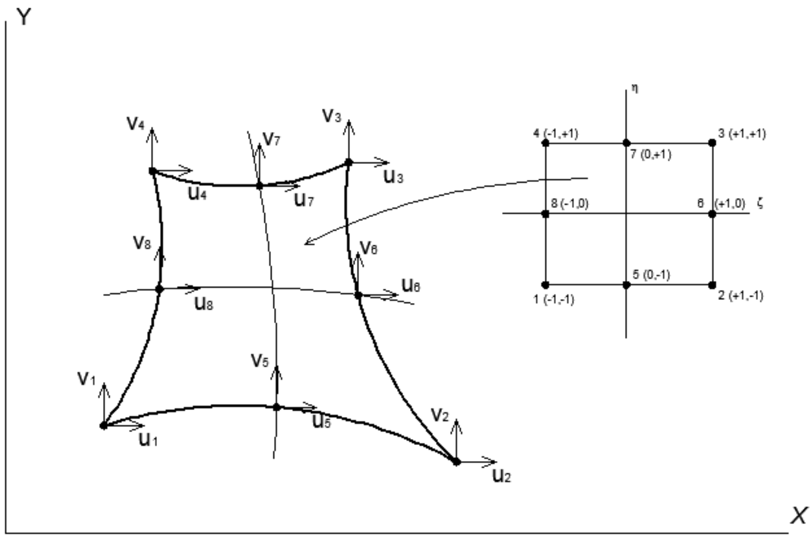


Figure 3: Plane Eight Node Isoparametric Element

The displacement vector is arranged in the form

$$\{\hat{u}\} = \{u_1 \quad v_1 \quad u_2 \quad v_2 \quad \ldots \quad \ldots \quad u_8 \quad v_8\}_{16x1} \tag{13}$$

and nodal coordinates in the form

$$\{\hat{x}\} = \{x_1 \quad y_1 \quad x_2 \quad y_2 \quad \ldots \quad \ldots \quad x_8 \quad y_8\}_{16x1} \tag{14}$$

In isoparametric formulation, element displacement and coordinates are interpolated from nodal displacement and coordinates with the same shape functions

$$\begin{Bmatrix} u \\ v \end{Bmatrix} = \begin{bmatrix} N_1 & 0 & N_2 & 0 & \dots & \dots & N_8 & 0 \\ 0 & N_1 & 0 & N_2 & \dots & \dots & 0 & N_8 \end{bmatrix}_{2x16} \{\hat{u}\} \tag{15}$$

and

$$\begin{Bmatrix} x \\ y \end{Bmatrix} = \begin{bmatrix} N_1 & 0 & N_2 & 0 & \dots & \dots & N_8 & 0 \\ 0 & N_1 & 0 & N_2 & \dots & \dots & 0 & N_8 \end{bmatrix}_{2x16} \{\hat{x}\} \tag{16}$$

in which

$$\begin{aligned} &N_i(\xi,\eta) = \frac{1}{4}\xi\eta(1-\xi)(1-\eta); \quad N_2(\xi,\eta) = \frac{1}{4}\xi\eta(1+\xi)(1-\eta) \\ &N_3(\xi,\eta) = \frac{1}{4}\xi\eta(1+\xi)(1+\eta); \quad N_4(\xi,\eta) = \frac{1}{4}\xi\eta(1-\xi)(1+\eta) \\ &N_5(\xi,\eta) = -\frac{1}{2}\eta(1+\xi)(1-\xi)(1-\eta); \quad N_6(\xi,\eta) = +\frac{1}{2}\xi(1+\xi)(1+\eta)(1-\eta) \\ &N_7(\xi,\eta) = +\frac{1}{2}\eta(1+\xi)(1-\xi)(1+\eta); \quad N_8(\xi,\eta) = -\frac{1}{2}\xi(1-\xi)(1+\eta)(1-\eta) \end{aligned} \tag{17}$$

In the formulation of element stiffness, some partial diferentiations are needed; i.e.,

$$\frac{\partial x}{\partial \xi} = \sum N_{i,\xi} x_i; \quad \frac{\partial x}{\partial \eta} = \sum N_{i,\xi} x_i; \frac{\partial y}{\partial \xi} = \sum N_{i,\eta} y_i; \quad \frac{\partial y}{\partial \eta} = \sum N_{i,\eta} y \tag{18}$$

with the summation are carried out for 1 until 8. The Jacobian matrix then reads

$$[J] = \begin{bmatrix} \sum_{i=1}^{8} N_{i,\xi} x_i & \sum_{i=1}^{8} N_{i,\xi} y_i \\ \sum_{i=1}^{8} N_{i,\eta} x_i & \sum_{i=1}^{8} N_{i,\eta} y_i \end{bmatrix} \tag{19}$$

and the determinant is

$$|\mathrm{J}| = \sum N_{i,\xi} x_i \sum N_{j,,\eta} y_j - \sum N_{i,\xi} y_i \sum N_{j,,\eta} x_j \tag{20}$$

and

$$\bar{J}_{11} = -\sum N_{i,\eta} y_i; \quad \bar{J}_{12} = -\sum N_{i,\xi} y_i; \bar{J}_{21} = -\sum N_{i,\eta} x_i; \quad \bar{J}_{22} = \sum N_{i,\xi} x_i \tag{21}$$

The $[B]$ matrix relating displacement vector to straint vector is

$$[\mathrm{B}] = \frac{1}{|J|} \begin{bmatrix} \tilde{B}_1 & \tilde{B}_2 & \dots & \tilde{B}_8 \end{bmatrix}_{(3x16)} \tag{22}$$

in which

$$[\tilde{\mathrm{B}}_{\mathrm{j}}] = \begin{bmatrix} a_j & 0 \\ 0 & b_j \\ b_j & a_j \end{bmatrix} \tag{23}$$

The elements in Eqn. 23 are

$$a_j = \bar{J}_{11} N_{j,\xi} + \bar{J}_{12} N_{j,\eta}; \quad b_j = \bar{J}_{21} N_{j,\xi} + \bar{J}_{22} N_{j,\eta} \tag{24}$$

in which

$$a_j = \sum_{i=1}^{8} N_{i,\eta} y_i N_{j,\xi} - \sum_{i=1}^{8} N_{i.\xi} y_i N_{j,\eta}; \; b_j = -\sum_{i=1}^{8} N_{i,\eta} x_i N_{j,\xi} + \sum_{i=1}^{8} N_{i.\xi} x_i N_{j,\eta} \tag{25}$$

Stiffness matrix of element may be formed by using formulation

$$[k_i] = [\iiint_V [B_i]^T [E][B_i] dV] \tag{26}$$

In the process, elements a_j and b_j in matrix $[B]$ are in quadratic forms in ξ and η such the integrand in qubical forms in respective ξ and η. Therefore, the form in Eqn. 26 has to be integrated by means of Gauss integration with the number of integration points $m = (3+1)/2 = 2$ in respective *x* and *y* direction. The location of integration points is shown in $\xi_i = \eta_i = \pm 0.577350269$ ... . Then the stiffness matrix becomes

$$[k]_{8x8} = \sum_{i=1}^{4} w_i x \frac{\left[\tilde{B}(\xi_i,\eta_i)\right]^T [E]\left[\tilde{B}(\xi_i,\eta_i)\right]}{\left|J(\xi_i,\eta_i)\right|} \tag{27}$$

For the vertical load$\bar{t}$, the equivalent element load takes the form

$$\{p_e\} = \int_{-1}^{+1}\int_{-1}^{+1} [N]^T t(\xi,\eta)|J| d\xi d\eta \tag{28}$$

Considering the order of $\bar{t}(\xi,\eta)$ in ξ and η, while $N_i(\xi,\eta)$ are in quadratic form in ξ and η, then the number of integration Gauss points m_ξ and m_η in ξ and η directions may be computed, and element equivalent load vector becomes

$$\{p_e\} = \sum_{j=1}^{m_\eta}\sum_{i=1}^{m\xi} w_i w_j \left[N(\xi_i,\eta_j)\right]^T t(\xi_i,\eta_j)\left|J(\xi_i,\eta_j)\right| \tag{29}$$

III Computer Programming

Several computer package programs are constructed based on the finite element method described in the previous chapter. One computer package program is constructed for grid structural system, and another computer program is for plane half medium system. The computer packages follow standard formation of FORTRAN computer program. The new feature that incorporated in both programs are the suppression of inactive degrees of freedom. If displacement component U_i is suppressed, then global equilibrium equation is modified by zeroing all row and column elements $K_{ij} = K_{ji} = 0$ and setting $K_{ii} = 1.0$ and $P_i = 0$. For example, if $U_3$is suppressed, then

$$[K]\{U\} = \begin{bmatrix} & & 0 & & \\ & & 0 & & \\ 0 & 0 & 1 & 0 & 0 \\ & & 0 & & \\ & & 0 & & \end{bmatrix}\begin{Bmatrix} \\ \\ U_3 \\ \\ \\ \end{Bmatrix} = \begin{Bmatrix} \\ \\ 0 \\ \\ \\ \end{Bmatrix} \tag{30}$$

The solution of Eqn. 30 will result in zero for U_3.

IV Case Study

In the study, two cases are performed; i.e., a bridge structure as a grid system and a half plane space with a tunnel. The first case is a bridge consisting of several girders positioned initially, then some diapraghms are installed. The second case consists of half plane medium with own weight of the medium, and a houle is digged beneath the soil surface.

4.1. Bridge Structure with Inserted Diapraghm

As a first case, a bridge structure shown in Fig. with span 40.0 meters and width 18 meters is considered [2]. The bridge consists of girders made of post-tensioned concrete system, floor slab, and cross beams at supports. Bridge cross section, depicted in Fig 5, consists of five girders. The girder is made of I beam with the height 2.0 meters, flange width 2.0 meters, flange thickness 0.3 meter, and thickness of web 0.3 meter. The height of diapraghm is 0.6 meter and the width is 0.3 meter. The thickness of floor slab is 0.3 meter. The concrete is made of material with characteristic compression strength f_c' = 40 MPa and elastic modulus E = 20,000 MPa. The loading considered consists of structural own weight q_c= 24.0 kN/m^3, uniform live load 7.83 kN/m^2 and line live load 49.0 kN/m.

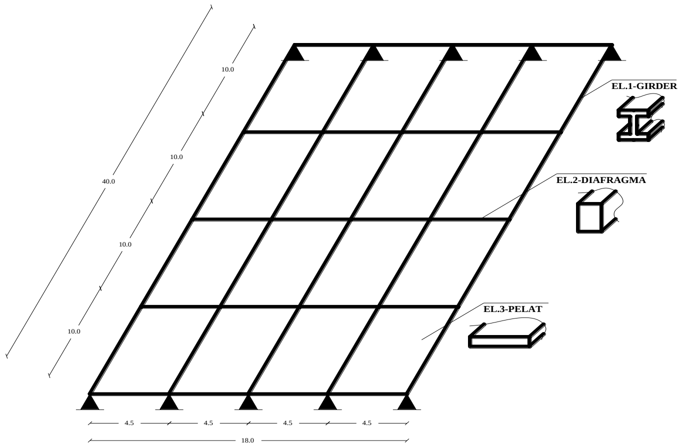


Figure 4: Bridge Grid System

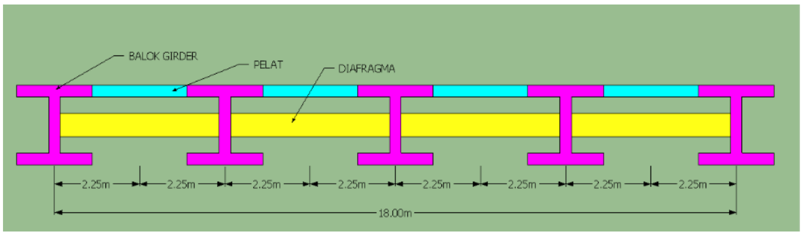


Figure 5: Bridge Cross Section

Finite element meshing of the bridge is shown in Fig. 6. The discrete model consists of 28 grid element representing girders, 16 rectangular bending plate element representing floor slab and 12 beam element representing diapraghms. Two cases of loading is considered; i.e., (1) dead load only, and (2) dead load with uniform live load at plate elements 42, 43, 46 and 47, and line live load at diapraghm 22, 23, 24, 27 and 28. Beside, three structural sub cases are considered, i.e., (1) the bridge without diapraghm in all loading cases , denoted by T-T, (2) the bridge without diapraghm in load case 1 and with diapraghm in load case 2, denoted by T-D, and (3) the bridge with diapraghm in all load cases, denoted by D-D.

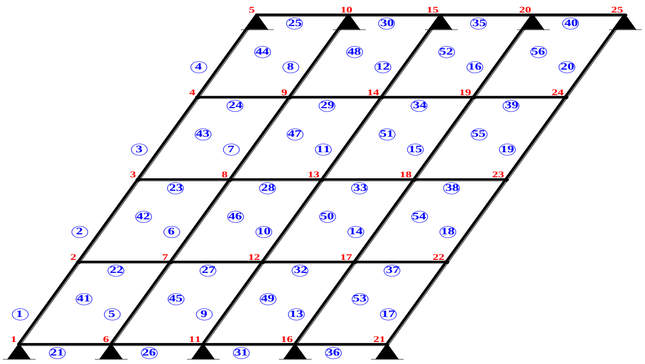


Figure 6: FiniteElement Meshing of Brigde

The analysis was carried out with a package program specially written for the algorithm. Displacements and stresses within the structure were all computed; but due to limitation of space, only displacements are disscused for load case 2 and for structural case T-D. The displacements at mid span diapraghm is shown in Fig. 7. The maximum differential vertical displacement between two adjacent girders in case T-T is 0.0039 meter. In this sub-case, the differential vertical displacement could not withstood by floor slab and wide cracks developed at floor slab.

The floor slab cracking was then remedied by insterting diapraghms in the existing adjacent girders. The diapraghms were made of precast concrete. The displacement for structural case T-D was recorded at mid span diapraghm. The maximum differential vertical displacement between two adjacent girders in case T-D is 0.0016 meter. The cracks in floor slab were then filled by epoxy. The insertion of the diapraghms overcame the crack problem of floor slab.

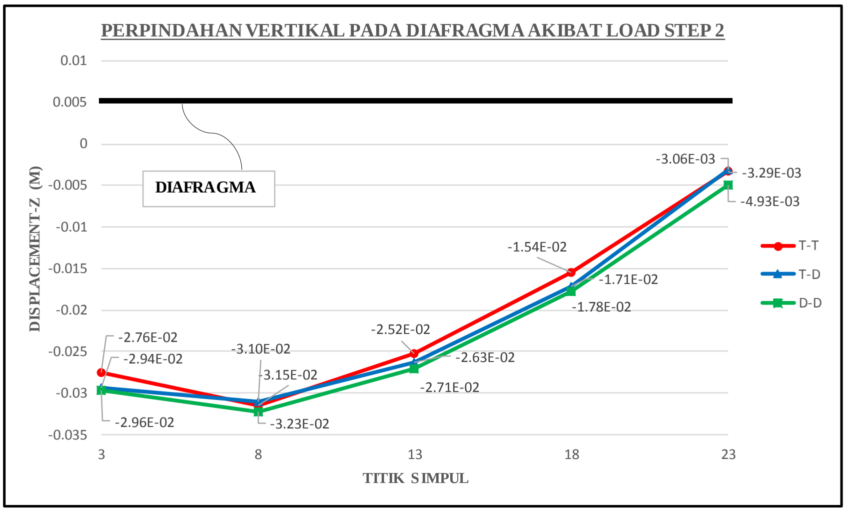


Figure 7: Plot of Displacement at Bridge Mid Span Cross Section

4.2. Plane Medium with Tunnel

A soft medium resting on hard layer is considered in this second case [1]. The region considered is the medium portion with width 20.0 meters and thickness 20.0 meters. The region is represented by a discrete model with 100 eight node isoparametric elements shown in Fig. 8. Two loading cases are considered; i.e., (1) own wieght of medium only, and (2) own weight of medium plus uniformly distributed load at medium surface. Four structural cases are considered; i.e., (1) Medium without tunnel at all loading cases, denoted by T-T, (2) medium with opening in load case 2, denoted by T-B, (3) medium with tunnel in load case 1 and then closed in load case 2, denoted by B-T, and (4) medium with tunnel in all load cases, denoted by B-B. Uniform distributed load is performed in three load steps, i.e., 0.0, 1,000.0 and 2,000.0 kN/m.

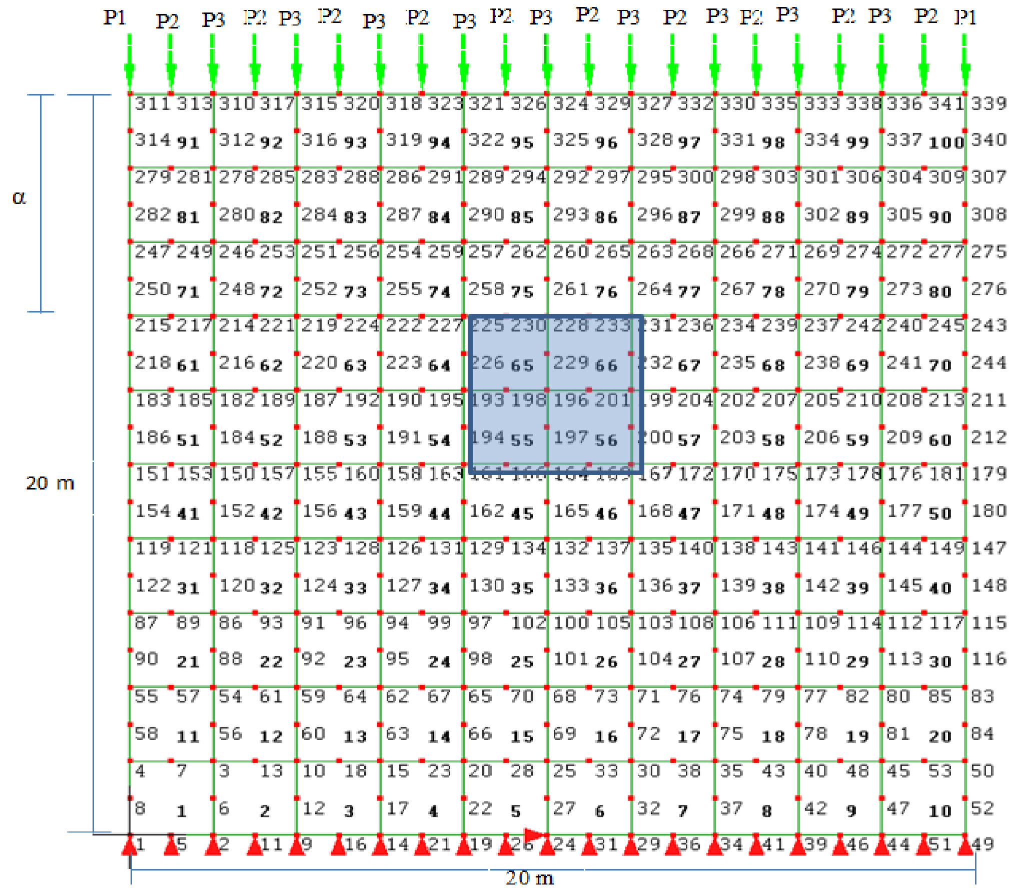


Figure 8: Finite Element Meshing of Medium

The tunnel dimension is 4.0 meters x 4.0 meters represented by 2 x 2 = 4 elements. The tunneling depth is denoted by α. Displacements, strains and stresses are computed for all cases, but due to limitation of space, only displacements are considered. Three observations are made; i.e., (1) the vertical displacment at node 228 for structural cases T-T, T-B, B-T and B-B; (2) the vertical displacment at node 228 in structural case T-B due to sequence of opening of the four elements and (3) the vertical displacement at nodes 100, 196 and 292 according to tunneling depth.

Table 1: Displacements at Node 228 for the Four Structural Case

Step Pembebanan(kN)	Displacement(m)			
	TT	BB	TB	BT
0	0	0	0	0
1000	-0.2114	-0.3222	-0.2114	-0.3222
2000	-0.4228	-0.6444	-0.5336	-0.5336

For observation 1, displacements at node 228 are given in Table 1 and drawn in Fig. 9, The smallest displacement occurs in structural case T-T, the largest displacement occurs in case B-B, while the displacements in case T-B and B-T are the same.

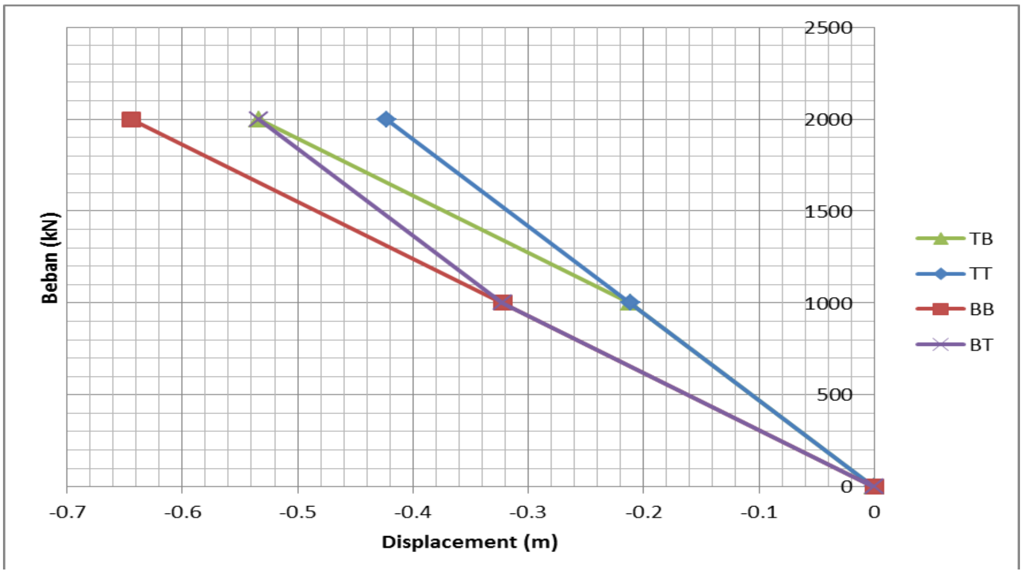
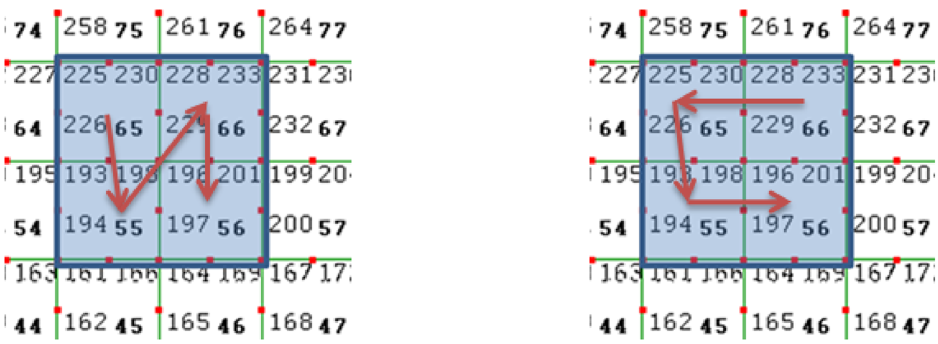


Figure 9: Displacement Curves at Node 228

For observasion 2, two cases of digging sequence are considered; i.e., (1) sequence 65, 55, 66 and 56; and (2) sequence 66, 65, 55 and 56, shown in Fig. 10.



(a) sequence 1

(b) sequence 2

Figure 10: Sequence of Tunneling

The displacements at node 228 are tabulated in Table 2 and drawn in Fig. 11 for loading levels. The displacement in sequence 2 is slightly larger than that of sequence 1 since sequence 2 causes larger decrease of structural stiffness.

Table 2: Displacement at Node 228 According to Tunneling Sequence

Load Step (kN)	Displacement (m)	
	Sequence 1 (65-55-66-56)	Sequence 2 (66-65-55-56)
0.0	0.0000	0.0000
1000.0	-0.2114	-0.2114
2000.0	-0.4459	-0.4462
3000.0	-0.6873	-0.7544
4000.0	-1.0040	-1.0720
5000.0	-1.3270	-1.3940

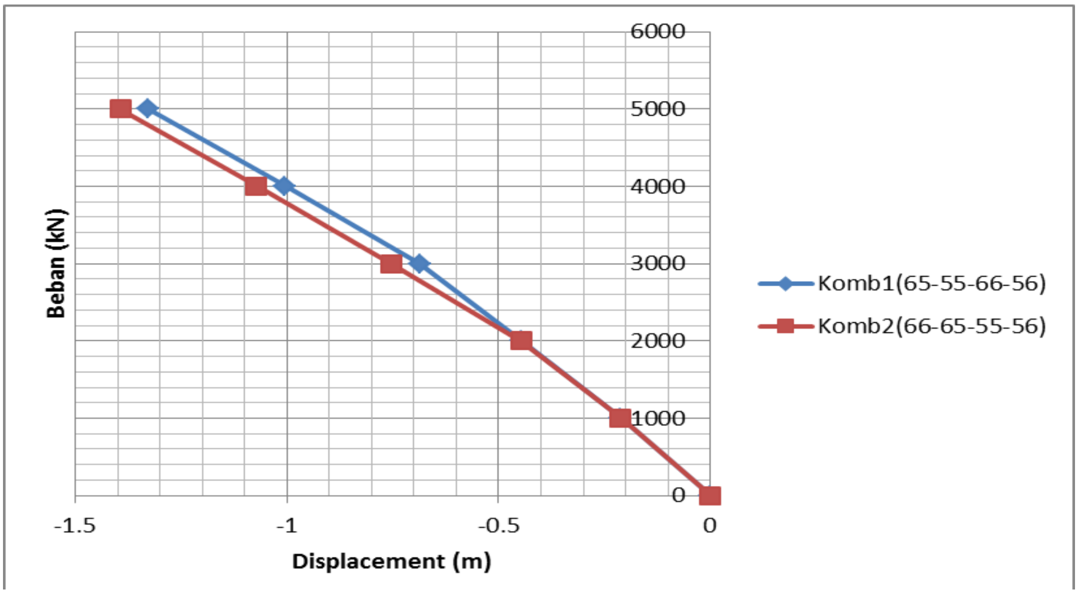


Figure 11: Displacement Curves at Node 228 According to Tunneling Sequence

The third observation deals with the influence of the tunneling depth with respect to displacement of node 100, 196 and 292. These nodes are the nodes at mid side of upper side of tunnel hole. The displacements at these nodes are tabulated in Table 3. Since node 292 is closest to medium surface, then the largest displacement occurs here. Since node 100 is farest to medium surface, then the smallest displacement occurs there.

Table 3: Displacements at Node 100, 196 and 292 According to Tunneling Depth

Load Step (kN)	Displacement (m)		
	α = 1 m (node 292)	α = 4 m (node 196)	α = 7 m (node 100)
0.0	0.0000	0.0000	0.0000
1000.0	-0.4691	-0.2835	-0.1816
2000.0	-0.9382	-0.5669	-0.3632

V Conclusions

A computer package program was prepared for the analysis of sructural systems with changing geometry. Several cases are capable of analyzed; i.e., structural system with constant elements, structural system with reducing components, structural system with adding elements. In all cases, all elements and hence all nodes are included in computation of total number and numbering of structural degrees of freedom. For the case of reducing components, inactive components are not taken into account by setting zero value for modulus of elasticity, and degrees of freedom of nodes pertaining to inactive elements are suppressed.

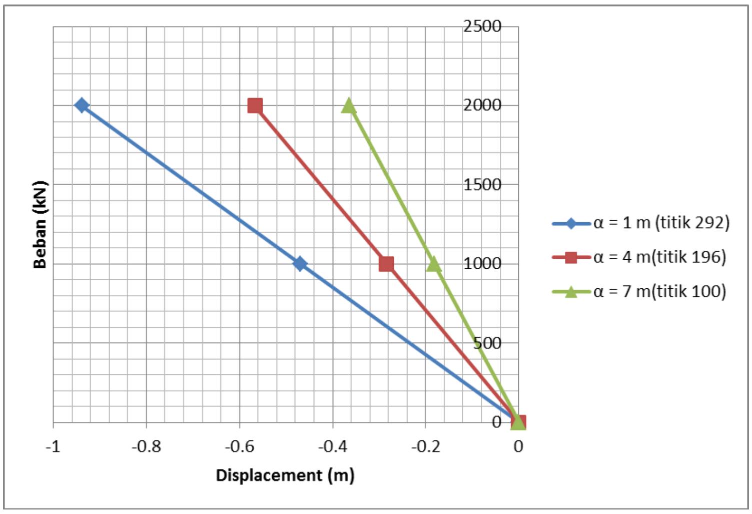


Figure 12: Displacement Curves at Node 100, 196 and 292 According to Tunneling Depth

Two cases are considered; i.e., bridge structure with and without diapraghms and plane medium with tunnel. In the first case, the decrease or increase of elements results in the decreace or increase of structural stiffness which in turn affects the displacements, strains and stresses. In the second case, the computer results may demonstrate the influence of the size and the depth oftunneling.

Acknowledgements

The paper is written partly based on the thesis of Mr. Jeply Murdiaman Guci and the thesis of Mr. Muhammad Azhar Dwitama, carried out at Civil Engineering Department, Bandung Institute of Technology, Bandung, Indonesia, in which the author was the academical advisor. Typing and drawing of the paper were carried out by Mr. Donni Canra, to which the author extend his cordial and sincere appreciation.

References

[1]. Dwitama, M.A., Analisis Sistem Struktur Bawah Tanah Dengan Metoda Elemen Hingga, thesis, Civil Engineering Department, Bandung Institute of Technology (2017)
[2]. Guci, J.M., Pengaruh Diafragma Terhadap Perilaku Sistem Struktur Jembatan, thesis, Civil Engineering Department, Bandung Institute of Technology (2016)
[3]. Hariandja, B., Analisis Struktur Berbentuk Rangka Dalam Formulasi Matriks, Penerbit Aksara Hutasada, Bandung (1997).
[4]. Hariandja, B., Metoda Elemen Hingga, Penerbit Teknik Sipil, Universitas Pancasila, Jakarta (2015).

Binsar H. Hariandja "Analysis Of Gradually Built Structural System" IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) , vol. 15, no. 1, 2018, pp. 01-11