MODELING STRATEGIES FOR THREE DIMENSIONAL ANALYSIS OF PRECAST PANEL BUILDINGS UNDER SEISMIC LOAD

Serdar ASTARLIOGLU[1], Ali M MEMARI2[2] And Andrew SCANLON[3]

SUMMARY

This paper describes a computer program (SNAPS-3D) being developed to perform nonlinear dynamic analysis of three-dimensional precast concrete building structures. The finite element library consists of three-dimensional brick elements to model wall panels and horizontal diaphragm components, beam elements with plastic hinges to model coupling beams, and spring elements to model behavior of joints between panels and diaphragms. The spring elements are designed to model potential crushing, gap opening, and sliding. Of particular interest is the performance of the joints at the interface between diaphragm and vertical wall panel elements. Truss elements are used to model vertical post-tensioning. The paper includes an assessment of the applicability of the modeling techniques used.

INTRODUCTION

Construction of precast concrete wall panel buildings in seismic zones has been limited due to uncertainties concerning the performance of these types of structures during seismic events. Precast concrete large panel construction consists of vertical and horizontal panels connected to each other by different kinds of joints to form a box type structure. The usual dry connections (using bolting, welding, etc.) form planes of weakness due to low strength and stiffness of the connection in the structure and during a severe earthquake, highly localized inelastic deformations can be introduced due to sliding and rocking at joint locations.

Analytical work by Schricker and Powell [1980], Kianoush and Scanlon [1988], Pekau and Hum [1991] and Kianoush *et al*. [1996] have been limited to analysis of simple walls or coupled walls with various horizontal and vertical connection models. Such attempts at modeling response of precast building structures have generally used two-dimensional analysis of wall systems isolated from the remainder of the structure and with the assumption of rigid diaphragms and concentration of inelastic action only at connection regions. However, the behavior of the structure may be strongly influenced by interaction between precast diaphragms with flexible connections and lateral load resisting systems that may include combinations of simple wall and coupled wall arrangements, particularly if walls in both directions or unsymmetrical wall arrangements are present. In such cases and in configurations with core walls or flexible floor diaphragms, two-dimensional analysis may not provide sufficient accuracy. It is to be noted that in short buildings with reinforced concrete shear walls, the stiffness of walls are usually much larger than horizontal elements. According to Unemori *et al*. [1980], in such cases, the distribution of seismic forces based on the usual assumption of rigid diaphragms will not be accurate.

Modeling wall panels using linearly elastic finite elements may not give accurate results. In such cases, because of joint opening due to rocking mechanism, panel corners may experience high compressive

stresses during uplift of the opposite side, as reported by Oliva *et al*. [1988] and Caccese and Harris [1987]. The objectives of the study described in this paper are to develop an analytical model for inelastic analysis of three-dimensional large panel structures and determine the effects of diaphragm flexibility, rocking and sliding mechanisms on overall response of the building and its stability.

[1] *Department of Civil and Environmental Engineering, The Pennsylvania State University.*
[2] *Department of Architectural Engineering, The Pennsylvania State University.*
[3] *Department of Civil and Environmental Engineering, The Pennsylvania State University.*

OVERALL DESCRIPTION OF SNAPS-3D COMPUTER PROGRAM

This section describes the overall modeling aspects of SNAPS-3D finite element analysis program being developed to perform nonlinear dynamic analysis of three-dimensional precast concrete building structures. The finite elements consist of three-dimensional brick elements to model wall panels and horizontal diaphragm components, beam elements with end plastic hinges to model regular beams or coupling beams, and spring elements to model the behavior of joints between panels and diaphragms. The spring elements are designed to model potential crushing of concrete, sliding of panels at horizontal and vertical joints, and joint openings. Of particular interest is the performance of the joints at the interface between diaphragm and vertical wall panel elements. Truss elements are used to model vertical post-tensioning.

The program is developed in modular form, where different tasks are performed by separate subroutines. The program consists of the following modules: input, loading, incremental loading, stiffness matrix assembly, solution of equations, residual force calculations, convergence check, and output results. A schematic flow chart for the program is shown in Figure 1. The organization of the program follows the layout suggested by Owen and Hinton [1980]. The module for input data consists of information on geometry, boundary conditions, and material properties of the finite element model. The loading module takes into account the loading contribution of each element and currently considers static loads that are applied to the structure at the nodes. When completed, it will be possible to apply ground accelerations in three orthogonal directions as well (two horizontal such as N-S component and E-W component, and vertical acceleration.) The module for incremental loading controls the loading increments. The load incrementing module when fully implemented will calculate the load vector at each time step, so that equilibrium equations can be evaluated at those time points. In static nonlinear analysis time step has no meaning, however, incrementing the load, instead of placing the full amount, results in efficiency in the iterations that follow. If the structure stays linear elastic during a load increment, equilibrium is satisfied automatically. However, if nonlinear behavior is observed, then iterations must be carried out until the residual forces are eliminated. Once the equilibrium for that load increment is satisfied (i.e., the residual forces are zero,) the next load increment is applied.

For each load increment, the following parameters will be used: maximum number of iterations, output frequency, the value of load increment, and the limit of convergence tolerance. The module for stiffness matrix assembles the structure stiffness matrix using the element stiffness matrices for solution based on displacement method of finite element analysis. The stiffness matrix is assembled starting with the first element. Depending on the type of the element, the element stiffness matrix is allocated (e.g. 24*24 for brick, 6*6 for truss element) and appropriate element stiffness module is called to calculate the components of the stiffness matrix. Following this step, the element stiffness matrix is added to the global stiffness matrix. The solution of simultaneous equations basically follows the Gauss Elimination technique, using the frontal solution method [Irons, 1970]. The essence of the frontal method is that before the complete global stiffness matrix is formed, certain degrees of freedom are eliminated from finite elements as they are being assembled. In fact, only those stiffness matrices that are required for static condensation of a specific degree of freedom are assembled first, and after the elimination of the degree of freedom, another element stiffness matrix is added for the elimination of another degree of freedom. This way, of course, a reduced structure stiffness matrix is assembled.

The nonlinear dynamic analysis of structural systems is accomplished using an implicit time integration scheme. Similar to the case of linear analysis, the equilibrium of the system at $t+\Delta t$ is considered. However, in nonlinear analysis, this consideration requires an iteration to be performed at each time increment [Bathe, 1996].

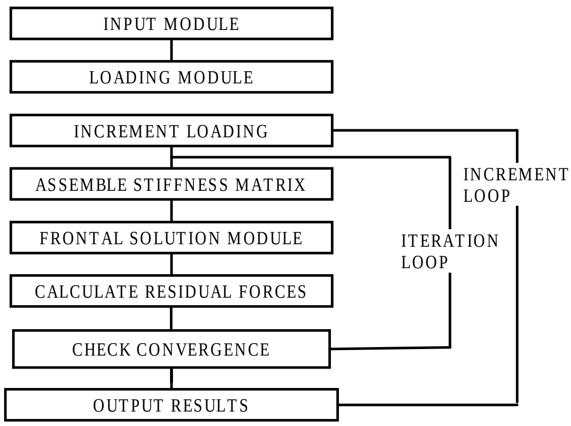

Figure 1: Schematic flow chart for SNAPS-3D

DESCRIPTION OF THE FINITE ELEMENTS USED

In this section, the modeling of finite elements used in the program SNAPS-3D is discussed. Some of the modeling assumptions are as follows: 1) the behavior of wall panels is assumed to be linear elastic; 2) the behavior of beams is assumed to be elastic, except at the ends of coupling beams, where concentrated inelasticity is assumed; 3) the floor slab is assumed to remain linearly elastic but is not infinitely rigid; and 4) the shear and compressive/tensile behavior across horizontal and vertical panel connections are assumed to be independent.

The wall panels and the floor slab panels are modeled using eight node (hexahedral) brick elements that have three translation degrees of freedom (dof) per node. This solid element, shown in Figure 2, has a total of 24 dof and is assumed to remain elastic during static and dynamic loading. When these elements are connected to beams, any nonlinearity that may occur is assumed to be limited to the beam ends. A wall panel can be modeled using several brick elements.

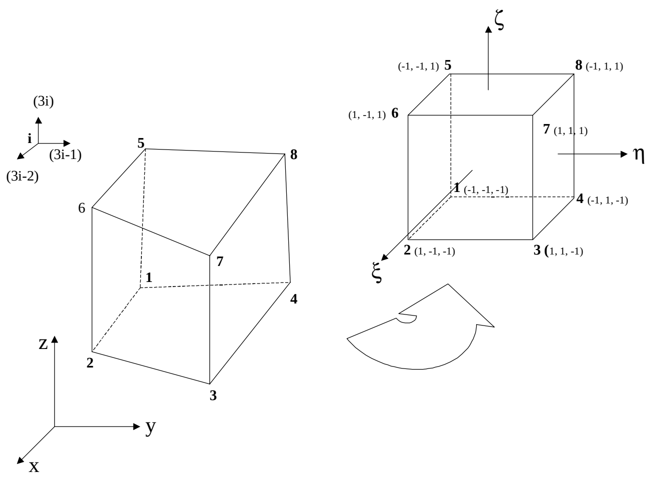

Figure 2: 8-Node brick element

Beams are idealized as beam elements with six dof at each end (three translations and three rotations.) In this program, beams are assumed to be connected to walls or slabs. Since at each end the beam is assumed to have a common node with the adjacent brick element, incompatibility arises because of the rotation dof on beam ends. To resolve the incompatibility, the rotation dof at beam ends are expressed in terms of translation dof of some of the adjacent brick nodes. Figure 3 shows the brick compatible beam element between nodes 1 and 5, where the rotation dof at nodes 1 and 5 are expressed, respectively, as translations at node sets 2, 3, 4, and 6, 7, 8. The rotations at node 1 are calculated by dividing the relative translations between node 1 and nodes 2, 3 and 4 by respective distances shown in Figure 3. Similarly, the rotations at node 5 are calculated by dividing the relative translations between node 5 and nodes 6, 7, and 8 by respective distances. The only restriction is that the additional nodes must be located on the beam's local coordinate system (e.g., line 12 should be on x' axis and line 13 should be on y' axis.) However, the rest of the nodes on the adjacent brick elements do not have to be in cubic (all 90° angles) arrangement and can be isoparametric, as shown in Figure 4. An appropriate transformation matrix is used in the element formulation to accommodate the needed compatibility.

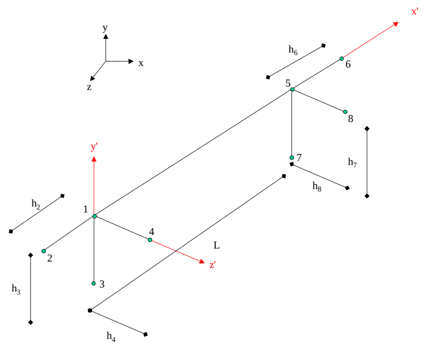

Figure 3: Brick compatible beam element

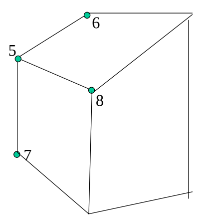

Figure 4: Adjacent brick element

The post-tensioning bars, assumed to be ungrouted, can be modeled using the 3-D truss element that has three translation dof at each node. This element, shown in Figure 5, is also assumed to have a linear elastic behavior. In later stages of development, this element will have elastoplastic behavior options as well.

The horizontal joints between two wall panels or between a wall panel and a slab or foundation and the vertical joints between wall panels can be idealized by using the gap shear friction element, shown in Figure 6. The element has two nodes, each with three translation dof, of which two perpendicular dof are in a plane parallel to the joint and one is normal to the joint. Compressive, tensile, and shear stiffness values can be specified independently.

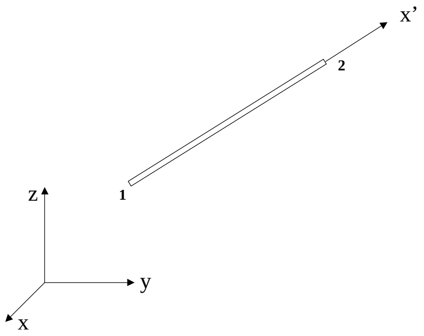

Figure 5: Truss element

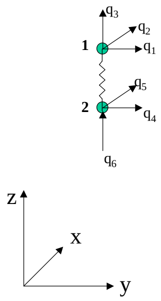

Figure 6: Gap-shear friction element

ELASTIC ANALYSIS OF AN EXAMPLE STRUCTURE

To illustrate the ability of the computer program to model response of three dimensional panel type structures, a static linear elastic analysis of the simple two-story structure shown in Fig. 7 has been performed. The structure, consisting of wall panels on three sides and coupling beams on the fourth side, is symmetrical with respect to the x-axis and unsymmetrical with respect to the y-axis. This configuration was elected to assess the ability of the model to demonstrate torsional effects on the structural response. Walls were connected to each other using vertical connections, and slabs were connected to the walls using horizontal connections. The plan dimensions are 360 cm x 360 cm, and the floor-to-floor height of each story is 250 cm. The walls are 20 cm thick, the floor and roof slabs are each 25 cm thick, and the coupling beams are 20 cm x 75 cm (including the slab thickness.) In addition to self-weight of members, a total lateral load of 4400 kN is applied in the y-direction, with 2200 kN at each floor level, half of that load at x = 0 cm and the other half at x = 360 cm. In this example, the floor planks have been assumed to be monolithic, however by using gap elements with appropriate stiffness values it will be possible to model individual floor planks and any potential relative movement.

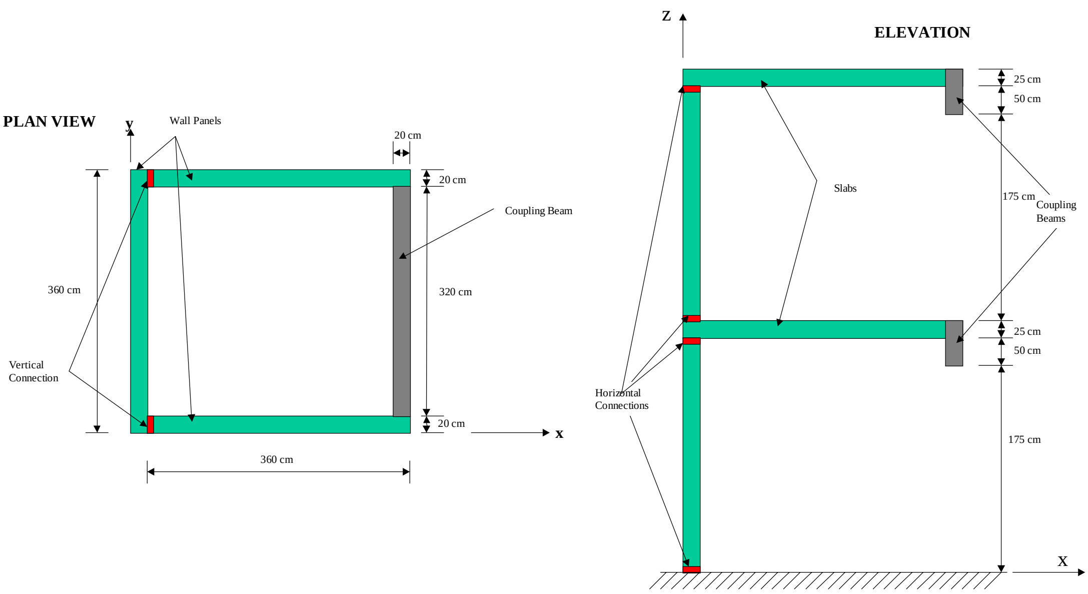

Figure 7: The plan and elevation section of the structure

The finite element model of the structure is shown in Figures 8.a and 8.b, where Figure 8.a shows the elements on the X-Z plane at the exterior (Y=0) side of the building and Figure 8.b shows the elements on the X-Z plane at Y=120 cm. In this representation, panels and slabs are modeled using brick elements, beams shown in two-dimensional cross-section are actually (line) beam elements connecting two nodes, and the panel-to-panel and panel-to-slab connections are modeled with gap friction elements. The finite element model has a total of 368 nodes, 106 brick elements, 2 beam elements, and 144 gap friction elements.

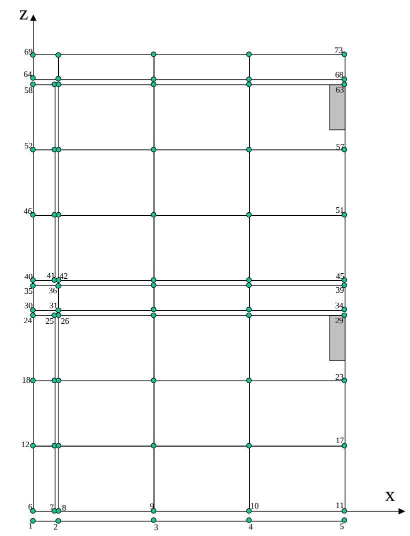

Figure 8.a: The Finite Element Model at Y=0 on the X-Z plane

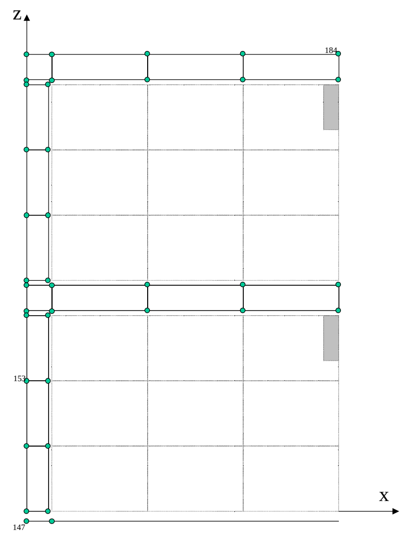

Figure 8.b: The Finite Element Model at Y=120 cm on the X-Z plane

In the finite element modeling, zero length gap elements are inserted at the base of the structure, above and below the first floor slabs and immediately below the roof slab. Since a linear elastic analysis is being performed, the axial stiffness of the gap element has been assumed to be equal in tension and compression. For panels, the modulus of elasticity and poisson's ratio are assumed as $30x10^3$ MPa and 0.2, respectively. At horizontal connections, 28 gap elements are used with the assumption of 17.5 kN/mm axial stiffness and 175 kN/mm shear stiffness per element. To restrain the movement in the vertical gap elements, very high shear and axial stiffness values are assumed (these values will be modified when the nonlinear program is developed.) Strictly speaking, because zero length elements have been used, the response of the gap elements should be rigid in compression. In an iterative non-linear analysis, tensile and compressive stiffnesses would be specified independently to model the effects of gap opening and potential crushing in the compressive zone as discussed in the next section. Also, the gap elements can be assigned a non-zero length.

With reference to Figure 8.a, the lateral loading is applied as concentrated nodal loads with 1100 kN in the y-direction at each of the following nodes: 35,39, 69, and 73. Results of the linearly elastic static analysis are presented in Figures 9.a, 9.b, and 10 in the form of deformed configurations in elevation and plan views. Figure 9.a shows deformations in the wall along the y-axis demonstrating the tendency for gap opening and slip to occur at the horizontal joints. The numerical results show that the total gap opening is 36 mm, 69% of which occurs at the foundation level horizontal connection and the rest at the first floor level, with no gap opening at the roof level. Figure 9.b shows the deformation pattern at a section located at x = 56 cm, again indicating gap opening and slip. Twisting of the unsymmetrical structure under symmetrical loading is clearly indicated by the deformed configuration of the roof in plan view as indicated in Fig. 10. The resulting nodal displacements show that the roof slab experiences a counterclockwise rotation of 0.0021 radians. These preliminary results indicate the potential for the program to model three-dimensional response of precast buildings under lateral load. Many of the non-linear modeling techniques developed previously for two-dimensional analysis [Kianoush and Scanlon, 1986] can now be incorporated to provide a better understanding of overall response of precast structures.

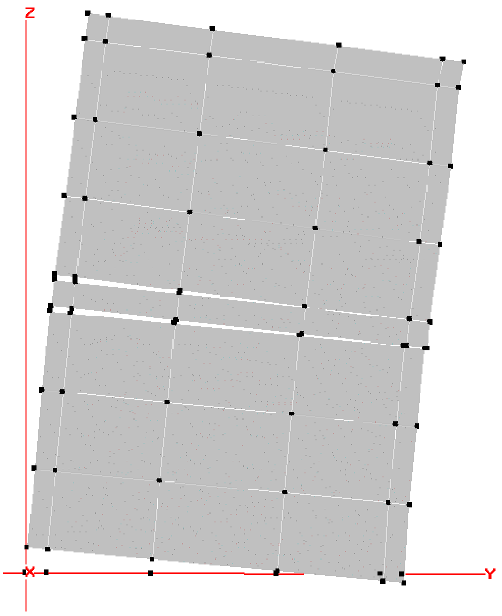

Figure 9.a: Deflected shape (X=0)

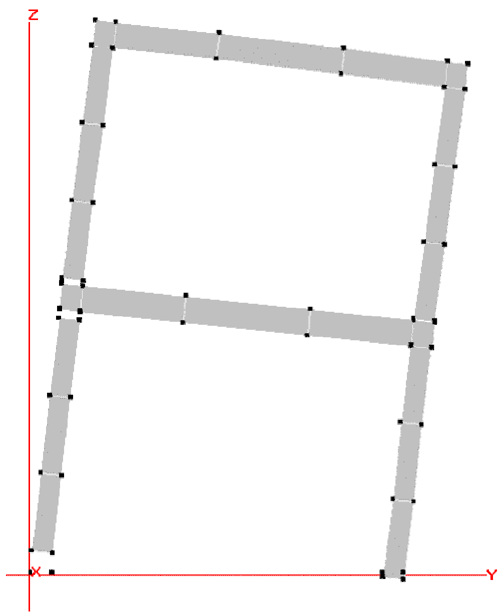

Figure 9.b: Deflected shape (X=140 cm)

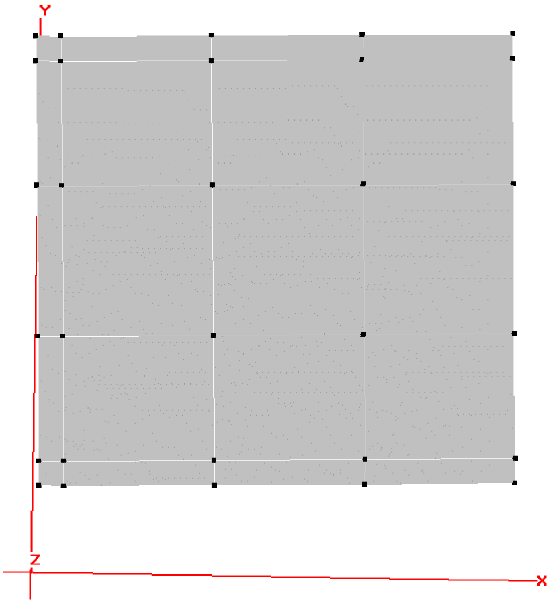

Figure 10: Deflected shape (Z=500 cm)

MODELING OF NONLINEAR RESPONSE

Techniques for non-linear modeling including plastic hinges at the ends of coupling beams, gap opening, slip, and crushing at horizontal and vertical joints, and yielding of reinforcement at horizontal joints have been developed in previous studies of two-dimensional structures as mentioned above. Several of these models are currently being incorporated in SNAPS-3D. For example, non-linear response of the gap shear-friction element can be specified as shown in Fig. 11 so that slip occurs at a specified normal stress. Nonlinear response in compression can be modeled by a multi-linear stress-strain diagram, and tension across the gap can be modeled by zero or low tension stiffness. Reinforcement across the gap can be modeled either by a separate gap element or in a combined compressive/tensile element. In addition to modeling potential slip in the longitudinal wall direction, three-dimensional modeling will provide information on potential slip in the direction transverse to the wall.

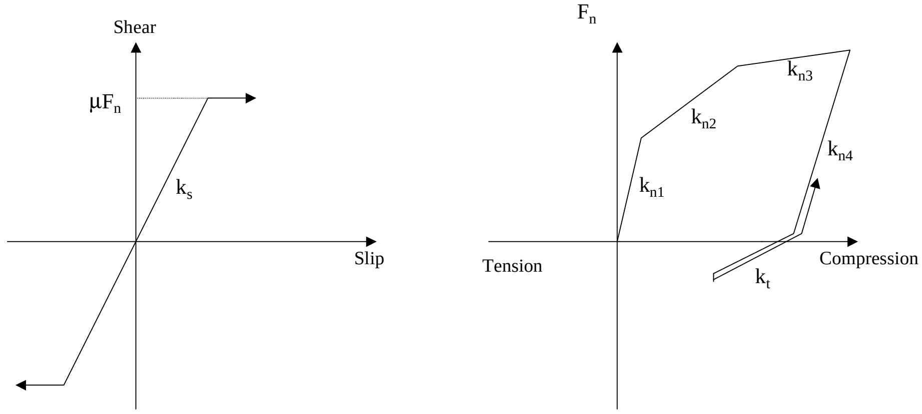

Figure 11: Force-displacement relationships for the gap-shear friction element

CONCLUDING REMARKS

A complete description of seismic response of precast concrete structures requires an understanding of three-dimensional structural behavior. The computer program being developed in this study, as an outgrowth of previous studies on two-dimensional modeling, promises to provide useful insights into the overall seismic performance of precast concrete structures.

ACKNOWLEDGEMENT

Financial support provided by (Canakkale Onsekiz Mart University) in the form of scholarship to the first author is gratefully acknowledged.

REFERENCES

Bathe, K.-J. (1996), *Finite Element Procedures*, Prentice Hall, Englewood Cliffs, NJ.

Caccese, V., and Harris, H. G. (1987), "Seismic Resistance of Precast Concrete Shear Walls," *Earthquake Engineering and Structural Dynamics*, Vol. 15, pp. 661-677.

Irons, B. M. (1970), "A Frontal Solution Program for Finite Element Analysis," *International Journal for Numerical Methods in Engineering*, Vol. 2, pp. 5-32.

Kianoush, M. R., and Scanlon, A. (1986), *Inelastic Seismic Response of Precast Concrete Large Panel Coupled Shear Wall System*, Structural Engineering Report No. 134, University of Alberta.

Kianoush, M. R., Elmorsi, M., and Scanlon, A. (1996), "Response of Large Panel Precast Wall Systems: Analysis and Design," *PCI Journal*, Vol. 41, No. 6, pp. 90-108.

Oliva, M. G., Clough, R. W., Velkov, M., and Gavrilovic, P. (1988), *Correlation of Analytical and Experimental Responses of Large Panel Precast Building Systems*, Report No. UCB/EERC-83/20, University of California, Berkeley.

Owen, D. R. J., and Hinton, E. (1980), *Finite Elements in Plasticity: Theory and Practice*, Pineridge Press Limited, Swansea, U.K.

Pekau, O. A., and Hum, D. (1991), "Seismic Response of Friction Jointed Precast Panel Shear Walls," *PCI Journal*, Vol. 36, No. 2, pp. 56-71.

Schricker, V., and Powell, G. H. (1980), *Inelastic Seismic Analysis of Large Panel Buildings*, Report No. UCB/EERC-80/38, University of California, Berkeley.

Unemori, A. L., Roesset, J. M., and Becker, J. M. (1980), "Effect of Inplane Floor Slab Flexibility on the Response of Cross-Wall Building Systems," *Reinforced Concrete Structures Subjected to Wind and Earthquake Forces*, ACI Publication SP-63, Detroit, MI, pp. 113-134.