8th Canadian Conference on Earthquake Engineering / 8ième Conférence canadienne sur le génie paraséismique
Vancouver - 1999

CALCULATED RESPONSE OF TILT-UP STRUCTURES

Fonseca, Fernando S.[1]; Hawkins, Neil M.[2]; and Wood, Sharon L.[3]

ABSTRACT

An analytical procedure for modeling the lateral load response of reinforced concrete tilt-up structures with plywood roof diaphragms was developed. The procedure was verified using the response of tilt-up specimens tested in the laboratory. Those specimens were representative of one-story warehouse tilt-up construction in the western U.S. The primary structural components of the specimens, such as concrete wall panels, plywood diaphragm, and roof framing system, along with important connections, were modelled explicitly. The analytical studies demonstrated that the lateral load response of reinforced concrete tilt-up structures with plywood roof diaphragms is controlled primarily by the behavior of the plywood roof diaphragm.

INTRODUCTION

Reinforced concrete tilt-up structures are economical to construct. Wall panels in tilt-up construction are cast horizontally on the floor slab, rather than in-place in vertical forms or in a prefabrication plant. Once set, these reinforced concrete wall panels are lifted (tilted) by a crane and set on prepared foundations to form the exterior walls. Next, the panels are connected to each other and to the foundation. Finally, the roof is attached to the panels. Because they are economical to construct, tilt-up structures comprise the primary structural system in approximately thirty percent of the existing low-rise industrial structures in the U. S. Tilt-up systems are susceptible to structural damage during earthquakes, and failure of the panel-to-roof or adjacent wall panel connections have often been cited as the cause of damage (Hamburger et al., 1988; Holmes and Somers, 1996; and Jennings, 1971).

The seismic performance of tilt-up construction is likely to improve if analytical models are developed to represent the lateral load response of the entire tilt-up system. The issues related to analytical modelling and structural details are intertwined because the system performance depends on the structural detail behavior. Many tilt-up buildings in use today rely on connections that are known to have performed poorly in recent earthquakes. An analytical model that can accurately predict the seismic response of tilt-up construction can also be used for evaluating the required strength and toughness of critical connections.

This paper briefly describes the analytical phase of a project to develop guidelines for evaluating seismic hazards in existing concrete tilt-up structures. A finite element model for calculating the lateral response of tilt-up structures is discussed. The results of analyses made with that model are compared with the measured response of one of two tilt-up specimens (specimen TU1) tested in the Newmark Civil Engineering laboratory. Detailed discussions of the experimental and analytical phases of the project are given in the reference by Fonseca (1997).

FINITE ELEMENT MODELS

Tilt-up structures have four major components: reinforced concrete wall panels, plywood roof diaphragm, foundation, and connections. For specimen TU1, the footings, the reinforced concrete wall panels, the plywood roof diaphragm, and the connections made between the wall panels and the roof diaphragm, between the wall panels and the footing, and between adjacent wall panels are shown schematically in Fig. 1.

Previous analytical studies of tilt-up systems have used simpler models (Adham, 1986 and Mehrain and Graf, 1990). Those models, however, necessitate a number of assumptions about the structural response of the system before beginning the analyses. In the study presented in this paper, the analytical model was built-up from individual structural elements so that the properties of each could be verified using available experimental data and the influence of the individual components on the system response evaluated. The models were designed to be sufficiently flexible to allow evaluation of the influence of different connection details on the seismic response. Accordingly, different types of finite elements were selected to model the structural elements of each major

[1] Assistant Professor, Brigham Young University, 368 Clyde Building, Provo, UT 84602

[2] Professor, University of Illinois, 205 North Mathews Avenue, Urbana, IL 61801

[3] Associate Professor, University of Texas, 10100 Burnet Road, PRC 177, Austin, TX 78758

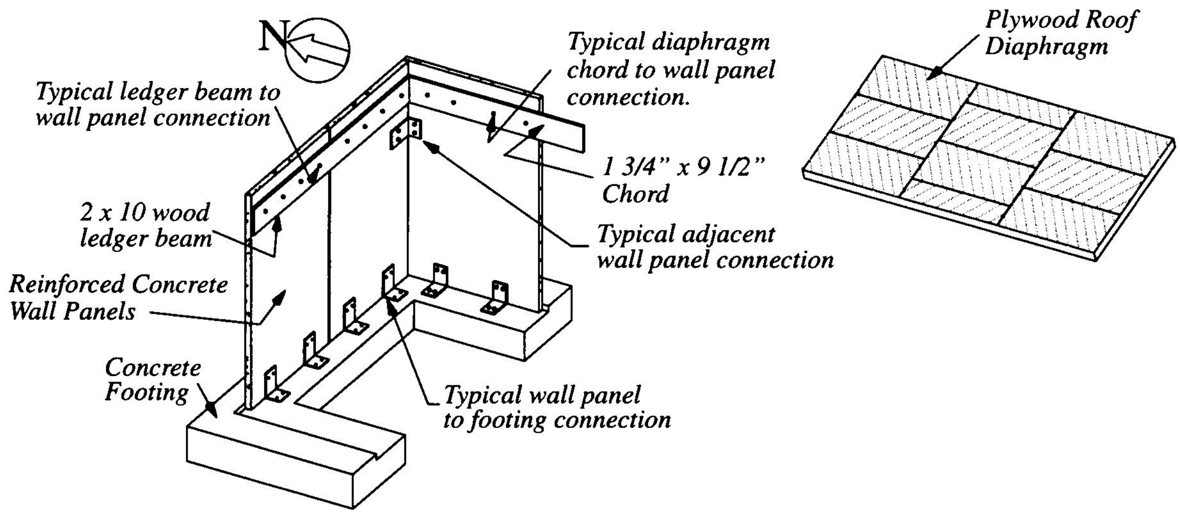

Fig. 1 Schematic View of Structural Components of Specimen TU1

component. The general-purpose finite element software ABAQUS (1992) was used for all analyses. The finite element representation of specimen TU1 is shown in Fig. 2.

Foundation and Boundary Condition along the Base of the Wall Panels

The foundation, as well as the connections between the wall panels and the foundation, were not intended to be representative of actual construction. The system was designed exclusively to provide a stable base for the wall panels of the test specimens. Therefore, the foundation and the connections to the wall panels were not explicitly modelled. Rather, an appropriate set of boundary conditions was established. The schematic representation of the boundary conditions used in the model along the base of the wall panels is shown in Fig. 2.

The boundary conditions were chosen based on test observations and measured responses. The in-plane measured deformations of the transverse wall panels were caused by several factors, including in-plane rocking, rigid body translation, and flexural and shear deformations. However, the transverse wall panels behaved essentially as rigid bodies and flexural; and shear distortions of the wall panels were very small and could be neglected. The in-plane and out-of-plane measured responses of the longitudinal wall panels indicated that these panels initially behaved as rigid bodies and simply rotated in the east-west direction about their bases. The bases of the transverse and longitudinal wall panels were allowed to rotate out-of-plane, but were restrained against translation in the vertical and horizontal directions. The arrows shown at the base of the wall panel in Fig. 2 indicate the directions in which the panels were restrained.

Reinforced Concrete Wall Panels

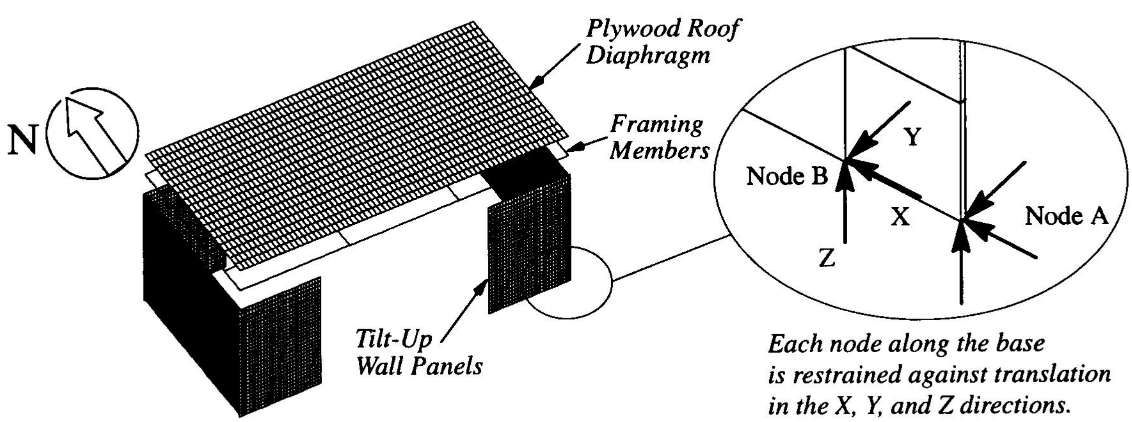

Fig. 2 F.E. Model of Specimen TU1 and the Boundary Conditions for the Base of the Wall Panels

A four-node shell element with reduced integration was used to represent the reinforced concrete wall panels (*S4R5* element of ABAQUS). In a tilt-up structure, the thickness of the wall panels is small compared to the length and height of the panel. Thus, transverse shear deformation is negligible and the wall panel may be modelled as a thin shell. The *S4R5* element is suitable for such applications. The element has only five degrees of freedom per node—three displacements and two rotations. The degree of freedom associated with transverse shear deformation is ignored. The deformations of the element are consistent with Kirchhoff plate theory.

The wall panels were modelled using linear methods. Measured flexural and shear distortions of the transverse wall panels were small compared to the overall displacements caused by the in-plane rocking of the panels. Thus, rocking was the only behavior considered necessary to model. Significant nonlinear behavior was observed for the longitudinal wall panels when they cracked diagonally at a load corresponding to approximately 85 percent of the capacity of the specimen. However, because of the difficulties involved in attempting to automatically include the effects of that cracking in any overall finite element model of a complete tilt-up structure, the longitudinal wall panels were also considered to remain elastic during the entire loading sequence. The possibilities of the concrete to crack in tension and its nonlinear response in compression were ignored.

Plywood Diaphragm

The roof diaphragm of the test specimen had two main components: plywood panels and framing members. The constant plane stress element *CPS8R* was selected to represent the plywood panel. This is a isoparametric, reduced-integration, eight-node quadrilateral element with a quadratic displacement field. The elements were assumed to be composed of isotropic material with linear stress-strain relationships.

Element *B21*, a two-node beam element that allows both bending and elongation, was selected to represent all framing members. The element uses a linear interpolation scheme and Timoshenko beam theory. The elements were also assumed to be isotropic with linear stress-strain relationships.

Connections

Plywood-to-Framing. Nails are the most widely used fastener in plywood diaphragms. In addition to connecting the plywood panels to the framing members, nails dissipate the energy accumulated during lateral loading. The non-linear behavior of plywood diaphragms is attributed exclusively to the non-linear behavior of the nails. This implies that the non-linear behavior of the framing members and the plywood panels have little effect on the overall response of the diaphragm. Also, it implies that the material representing the framing members and the plywood panels can be modelled using linear elastic properties without compromising accuracy of the results.

A new element was implemented in ABAQUS to represent the nails. The element, first used in a similar application by Easley *et al.* (1982), has two nodes, and each node has two translational in-plane degrees of freedom provided by a pair of springs extending parallel and perpendicular to the axes of the framing members. The spring pair represent the lateral stiffness of a nail, which is completely independent of direction. As the spring pair deforms, it has a component of the displacement in the two translational in-plane degrees of freedom. Those two components are combined to produce the total displacement which corresponds to the slip of the nail. The stiffness, however, is not combined. Using the calculated nail slip, the corresponding lateral stiffness is computed from non-linear nail force-slip curves. The load-slip curve used in the model was determined from a series of coupon tests (Dugan, 1995).

Framing-to-Framing. Within the roof framing system, sub-purlins were connected to purlins and purlins were connected to perimeter beams. Metal hangers, connected with nails, were used for these connections. The actual number of nails used in the connection depends on the size of the hanger. The nails connecting the metal hangers to the framing members dissipated energy in a manner similar to that of the nails connecting the plywood to the framing members. The connection between adjacent framing members could therefore be modelled by using the nail element. Because slip curves for hangers were not available, the connections were modelled as pinned connections with two coincident nodes using the multi-point constraint available in ABAQUS. The constraints were set so that only shear and axial forces were transferred between the two nodes. The rotations of the two nodes remained independent of each other and moments could not be transferred between the two nodes.

Bearing between Adjacent Plywood Panel. As the diaphragm was loaded, the initial gaps between adjacent plywood panels closed and the panels began to bear against each other. The element *INTER3* was selected to represent the contact between adjacent plywood panels. The element allowed the gaps between the plywood panels to increase without an increase in stiffness. If the gaps closed (*i.e.*, if one plywood panel touches another), there was an increase in stiffness. There was no separation between the surfaces once contact condition had been established, except in the case of unloading.

Roof Diaphragm-to-Wall Panel. The roof diaphragm was connected to the inside face of the concrete wall panels as shown in Fig. 1. Both ledger beams and diaphragm chords were connected to wall panels with four bolts per panel. An ABAQUS standard element, SPRING, was selected to represent the bolts.

Wall Panel-to-Wall Panel. There was no direct connection between the transverse wall panel. However, there was an indirect connection in the form of the continuity of the ledger beam across panels. In addition, a steel angle connected the transverse and longitudinal wall panels immediately below the roof elevation. It was assumed that the longitudinal and transverse wall panels had the same nominal displacements at the location of this connection. The assumption of equal displacements was accomplished in the finite element model by using the multi-point constraint option. The constraints were set so that the translational degrees of freedom of the transverse wall panel were linked to the translational degrees of freedom of the longitudinal wall panel at the node corresponding to the location of the connection. The corresponding rotational degrees of freedom of those nodes remained unconnected so that moments could not be transferred between longitudinal and transverse wall panels.

CALCULATED RESPONSE

Comparisons between measured and calculated results reported herein are between calculated and measured displacements, with the effects of rocking motions and slip at the base of the transverse wall panels removed (termed "modified measured response".) This procedure was necessary due to the boundary conditions, used in the finite element model, which did not allow for those behaviors. Further, the comparison between the calculated and modified measured responses is made using backbone curves.

In-Plane Displacement of Diaphragm

The number and location of nails connecting the plywood panels to the framing members influenced the response of the diaphragm. In addition to the base model, three models, J, K, and I were used to calculate the in-plane response of the diaphragm. In model J, the nails connecting the east side of the northwest and southwest plywood panels and the nails connecting the west side of the northeast and southeast plywood panels were eliminated. Model J was consistent with the nail survey completed approximately at 10 kips. In addition to the nails removed for model J, the nails connecting the south sides of the northwest and northeast plywood panels and the nails connecting the north sides of the southwest and southeast plywood panels were eliminated in Model K which was consistent with the nail survey completed after testing. In model I, the post-yield stiffness for the nail load-slip curve was taken as zero but no nails were removed.

The midspan modified measured response and the calculated response for these models are shown in Fig. 3. The measured response starts to become softer than the calculated response for the base model and model I at about a ram force of 4.2 kips. At a ram load of 10.0 kips, the computed displacements for the base model are approximately two-thirds of the measured displacements. By contrast, the computed displacements for model J at 10.0 kips are only about ten percent less than the modified measured displacements. Also, for models J and

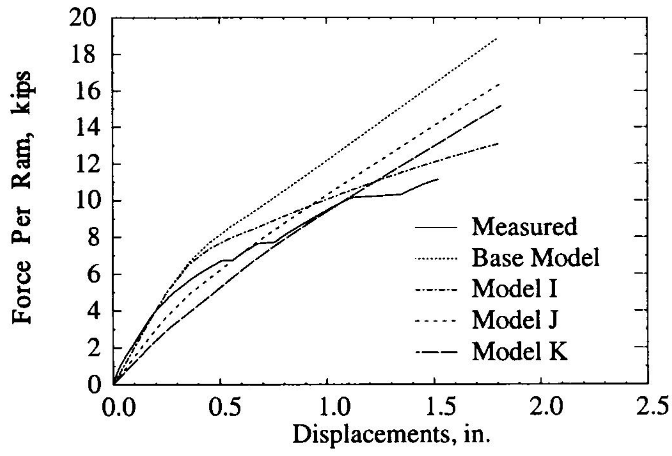

Fig. 3 Calculated Transverse Displacement at Midspan of the Diaphragm TU1

K, the rates of increase in displacements with increasing load are close to those measured at the load in which nail surveys were completed. Clearly, the number and position of the nails significantly influenced the results from the finite element model. For model I, the calculated displacements are approximately 80 and 95 percent of the measured displacements at loads of 6.0 kips and 10.0 kips, respectively. Furthermore, even after the cracking of the longitudinal wall panels at 10.0 kips, the calculated midspan displacement for model I is only 17 percent less than the modified measured displacements.

These results suggest that if the number of nails effective at a given load is known then the displacement at that load can be computed reasonably well using the analytical model described here—provided that longitudinal wall panels are uncracked. Unfortunately, that information would not be known unless a complex, step-by-step analysis is made in which nails are successively eliminated as their calculated slips reaches a previously defined limiting value. For design, such a complex analysis may not be necessary and displacements can be represented reasonably well using model I. On the average, the midspan displacements response calculated from model I is only 15 percent less than the modified measured displacements.

Displacements of the Longitudinal Wall Panel

The calculated response for the base model, model I and the modified measured out-of-plane displacements at mid-height and at the top of the south end of the northwest longitudinal wall panel are shown in Fig. 4. At low levels of applied loads, the modified measured response is slightly stiffer than the calculated responses. At higher values of applied loads, the difference between the calculated responses and modified measured response appears to be dependent of the location along the height of the wall panel. The instrument at mid-height was located just below the crack that developed in the longitudinal wall panel at about 10.0 kips, thus the displacements at that elevation decreased suddenly after formation of the crack. However, the instrument that was located at the upper corner of the longitudinal wall panel experienced a sudden increase in displacement at the time the crack developed.

The difference between the measured and calculated responses is not significant until cracking occurred at about 10.0 kips. The difference at loads less than 10.0 kips may be due to many reasons, including the increasing nonlinear behavior of the wall panels with increasing applied loads and numerical errors introduced during the calculation of the rocking component of the displacement. The large differences between the responses at 10.0 kips and above is due to the sudden increase and decrease in measured displacements after cracking of the wall panel, which was not modelled.

Another possible explanation for the difference between the measured and calculated responses may be the representation, in the finite element model, of the boundary conditions for the wall panel. In the analytical model the bases of the wall panels were not restrained against rotation. This may not accurately represent the actual boundary condition for the wall panels of the test specimens because some restraint against rotation was provided by the wall panel-to-footing connections and the notch in the concrete footing.

Because the correlation between the modified measured response and calculated responses is reasonably good, up to the cracking load, it is reasonable to conclude that if the model can correctly compute the load for cracking,

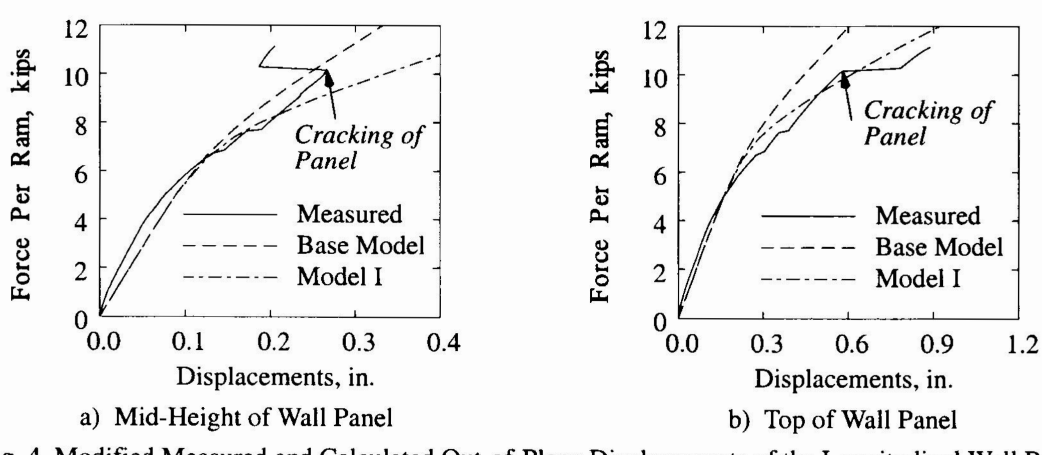

Fig. 4 Modified Measured and Calculated Out-of-Plane Displacements of the Longitudinal Wall Panel

then a simple elastic model, similar to the one presented here, can be used for the longitudinal wall panels in any model of the structure as a whole, for loads smaller than the cracking load.

CONCLUSIONS

The model developed was capable of representing well the overall measured response of the test specimen up to approximately forty percent of the maximum measured load. For higher loads, the measured response was increasingly softer, with an increase in applied loads, than the calculated response. Further modifications to that base model were necessary for continual representation of the measured response. Those modifications included recognition of when the nails became ineffective with increasing loads, and when cracking occurred in the longitudinal wall panels.

The comparisons between the calculated and measured responses indicated that the response of the specimen was controlled primarily by the response of the plywood diaphragm, which in turn was controlled directly by the inelastic behavior of the nails connecting the plywood panels to the framing members, and by the number of effective nails at a given applied load.

The ability of the finite element model to represent the measure response also seemed to be compromised by deviations between the load-slip curves for the nails, as determined from the measurements made in the tests and the curves obtained from coupon test results. There seemed to be three principal causes: the jamming of the plywood panels against the wall panels, out-of-plane deformations of the diaphragm caused by loading eccentricities and upward bowing of the plywood panels. Each of these effects, which were magnified with increasing load, enlarged the gap between the plywood panels and the framing members. This resulted in a lower load capacity for a given slip than that recorded in the coupon tests.

All these effects appear to be smeared when the post-yield stiffness of the nail load-slip curve was altered. Therefore, model I can be used to represent, reasonably well, the response of reinforced concrete tilt-up structures with plywood diaphragms.

REFERENCES

ABAQUS User's Manual. Version 5.2 (1992). Hibbitt, Karlsson & Sorenson, Inc., Pawtuckt, RI.

Adham, S.A. "Seismic Design Guidelines for Tilt–Up–Wall Buildings Based on Experimental and Analytical Models." *Proceedings.* Third U.S. National Conference on Earthquake Engineering. Earthquake Engineering Research Institute. 3 (1986): 1767–1777.

Dugan, K; Hawkins N.M.; and Wood. S.L. "Nail Slip Curves." Unpublished Report Urbana, IL, 1995.

Easley, J.T.; Foomani, M.; and Dodds, R.H. "Formulas for Wood Shear Walls." *Journal of the Structural Division*, ASCE, 108 (November 1982): 2460–2478.

Fonseca, Fernando S. "Cyclic Loading Response of Reinforced Concrete Tilt-Up Structures with Plywood Diaphragms." *Ph.D. Dissertation*, University of Illinois at Urbana-Champaign. Urbana, IL. May 1997.

Hamburger, R.O., D.L. McCormick, and S. Hom (1988). "The Whittier Narrows Earthquake of October 1, 1987 – Performance of Tilt–Up Buildings." *Earthquake Spectra.* Earthquake Engineering Research Institute. Vol. 4, No. 2, pp. 219–254.

Holmes, William T. and Peter Somers, technical editors (1996). "Northridge Earthquake of January 17, 1994: Reconnaissance Report, Vol. 2." *Earthquake Spectra,* Earthquake Engineering Research Institute, Supplement C to Volume 11.

Jennings, Paul C., editor (1971). "Engineering Features of the San Fernando Earthquake of February 9, 1971." California Institute of Technology. Division of Earthquake Engineering and Applied Science. *Earthquake Engineering Research Institute 71–02.* Pasadena, CA., pp. 230–258 and 297–298.

Mehrain, M. and Graf, W.P. "Dynamic Analysis of Tilt–Up Buildings." *Proceedings.* Fourth U.S. National Conference on Earthquake Engineering, Earthquake Engineering Research Institute. 2 (1990): 299–307.