DOI: 10.2507/36th.daaam.proceedings.xxx

ANALYSIS OF A TYPICAL 1960S LARGE PANEL BUILDING SEISMIC RESISTANCE

Romano Jevtić Rundek & Mario Uroš

This Publication has to be referred as: Katalinic, B[ranko]; Park, H[ong] S[eok] & Smith, M[ark] (2025). Title of Paper, Proceedings of the 36th DAAAM International Symposium, pp.xxxx-xxxx, B. Katalinic (Ed.), Published by DAAAM International, ISBN 978-3-902734-xx-x, ISSN 1726-9679, Vienna, Austria
DOI: 10.2507/36th.daaam.proceedings.xxx

Abstract

Construction of standardised precast large panel buildings intensified during the 1960s in the city of Zagreb. Many were built before the first significant seismic code was introduced (1964). Modern understanding shows that non typical and potentially dangerous failure modes are possible in these large panel buildings. In this article, some methods for analysis are presented, using software ETABS and Abaqus. High fidelity models of structural details are created in Abaqus and used to approximate their capacity curves. These curves are then used to calibrate a simpler lumped plasticity model in ETABS. The calibration process is validated by comparative analysis between wall assemblies in both software's to confirm appropriate interaction between elements of the lumped plasticity model. As a result, a simplified numerical model is provided capable of estimating complex failure mechanisms associated with large panel buildings to a reasonable degree.

Keywords: seismic analysis; Large panel buildings; Failure mechanism; Numerical model calibration

1. Introduction

The construction of standardized large-panel buildings intensified in the 1960s in Zagreb and across the Balkans. The first significant seismic code in the region was introduced in 1964, after many such buildings had already been constructed. Large-panel systems built in Zagreb before 1964 are characterized by a low amount of reinforcement, smooth reinforcement bars, a possible lack of longitudinal walls, and details prone to non-standard failure mechanisms. Resistance of a building portofolio can have a significant impact to seismic risk as shown in (1). Research into the seismic resistance of Large panel buildings gained momentum in the 1980s, notably with a report from the National Technical Information Service – NTIS (2), which consolidated experimental data, engineering recommendations, computational methods, and guidelines for numerical models. Among the research conducted in the Balkans, the master's thesis of Professor Zamolo (3) stands out. Large-panel buildings were predominantly constructed as semi-prefabricated reinforced concrete structures, composed of precast reinforced concrete panels. Typical 1960s large-panel buildings exhibit weak and brittle connections between elements, low reinforcement ratios, and the use of smooth rebar. Relevant failure mechanisms vary across individual buildings but generally include shear failure along horizontal or vertical panel joints, uplift of horizontal joints, and panel crushing at horizontal joints. When analysing such structures, it is necessary to account for the nonlinear behaviour of the connections to identify the governing failure mechanism of a large panel building. In the studies

Working Paper of 36th DAAAM Symposium

conducted at the time, due to limited computational capacity, analyses were restricted to smaller models and 2D simulations. Today, more advanced computational software such as Abaqus (4) and ETABS (5) is available. Simulations of kinematic and plastic behaviour of load-bearing elements and element interactions in assemblies can be seen in the literature, conducted using Abaqus (6)(7). Furthermore, examples of nonlinear building models applying concentrated plasticity can be found, which are used in modern quantification of seismic performance in the form of Probabilistic Seismic Hazard Analysis (PSHA) (8). From this, it can be concluded that a well-designed and calibrated model with concentrated plasticity would enable the application of PSHA to large-panel buildings. Additionally, it can be observed that there are significant variations in the shape and dimensions of large-panel elements, as well as in the overall geometry of the buildings themselves. Consequently, there are substantial differences in the resistance of details across different types of large-panel buildings. This paper describes the calibration procedure of a numerical model of one large panel building. Detailed local models are applied, along with a simplified global model calibrated against the detailed model. This approach allows sufficiently accurate modelling of the entire structure on a desktop computer, due to the significantly lower numerical complexity of the simplified model. This approach is likely to enable a more accurate PSHA analysis of buildings of this type, as well as global failure mechanism analysis and development of subsequent retrofit solutions.

2. Model definition

Two modelling approaches are employed: a detailed and computationally demanding 3D volumetric finite element model developed in Abaqus (4), and a simplified model based on concentrated plasticity elements developed in ETABS (5). The Abaqus model (Figure 1, left) is constructed using volumetric finite elements to simulate concrete and beam elements to simulate reinforcement, inserted as “embedded elements.” For the constitutive material model, the Concrete Damaged Plasticity (CDP) model (9) is selected. Its calibration, based on a chosen set of experiments, corresponds to a concrete with compressive strength of 20 MPa and is not described in this paper. The reinforcement is modelled using a bilinear constitutive law, equivalent to steel grade S235. The interaction between precast elements is defined either by friction alone or by friction and cohesion, depending on the model. The friction coefficient is taken as 0.6, with a sensitivity analysis performed.

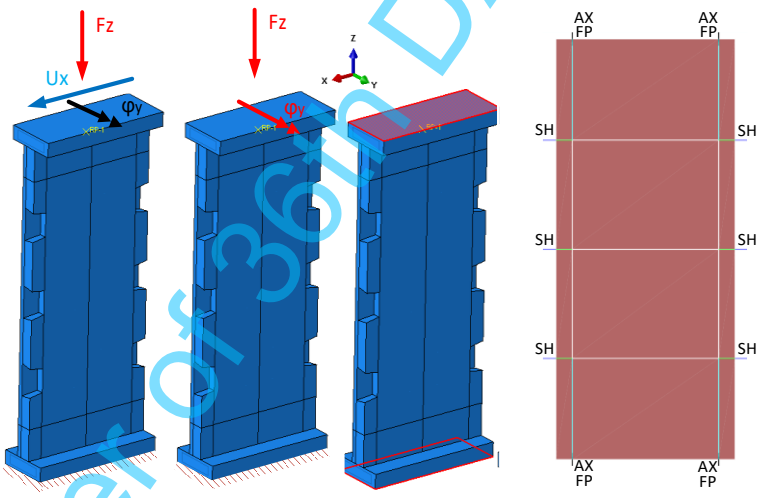

Fig. 1. One panel modelled in Abaqus (left) and ETABS (right)

The ETABS model (Figure 1, right) consists of linear shell and frame elements, and nonlinear link elements. Shell and frame elements form individual panels that behave elastically, while the links are arranged as shown in Figure 1 and simulate the behaviour of connections between adjacent elements. Links labelled SH simulate shear-keys, links labelled AX simulate panel uplift and crushing, and links labelled FP simulate panel sliding along horizontal joints. Links AX and SH are defined as bilinear (multilinear plastic) in the relevant degrees of freedom, while the FP link is defined as a connection with a friction coefficient that differs under tension and compression states (T/C friction pendulum).

3. Analysis definition

Two levels of analysis are performed. The first level involves multiple analyses of individual elements, and the second level involves multiple analyses of assembled walls. The first level is used for the initial calibration of nonlinear connection behaviour, while the second level is used to verify the interaction of calibrated connections within an assembled wall. Specifically, the first level includes pushover analyses of a single RC panel under different axial force levels, as well as analysis of a shear keys, allowing the simulation of all relevant failure mechanisms mostly independently. This level of analysis is conducted in both Abaqus and ETABS. Once the ETABS model is calibrated so that the behaviour of a single panel adequately matches the Abaqus results, the second level of analysis is performed to

Working Paper of 36th DAAAM Symposium

check the interaction of panels in an assembled wall. Pushover analyses are performed on several wall configurations with varying slenderness, thereby inducing different failure mechanisms (shear failure, flexural failure). The shape of pushover curves, failure mechanisms, and force distributions within elements is examined, assuming the more complex Abaqus model is accurate, while the simpler ETABS model is calibrated to achieve equivalent results.

4. Analysis results

Results are presented for a wall three panels wide and three panels high, as this configuration exhibits two failure mechanisms, enabling the demonstration of both shear and flexural failure. Figure 2 shows the Abaqus model (left) and ETABS model (right) after pushover analysis. Figure 3 presents the pushover curves of the entire walls in both models. Furthermore, Figure 4 shows the diagrams of horizontal forces in individual panels during pushover analysis for the Abaqus model, and Figure 5 for the ETABS model.

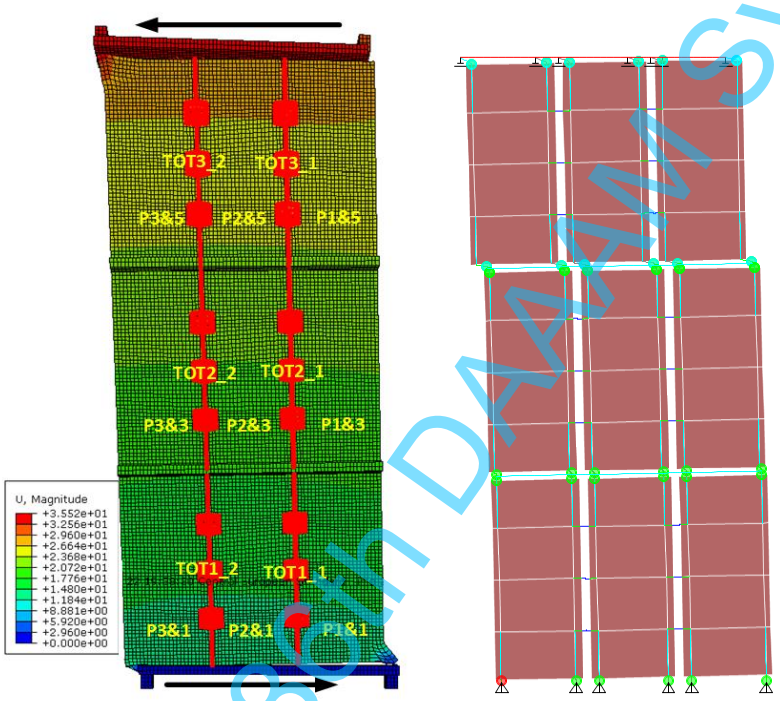

Fig. 2. Failure mechanism in Abaqus (left) and ETABS (right)

The pushover curve of the entire wall indicates good agreement in stiffness and strength of the assembled wall, as seen in Fig 3. Therefore, at minimum, good predictions of story drifts can be expected in the global model. The following discussion focuses on the distribution of forces and damage in individual elements of the assembled wall. In Figure 2, panel uplift and tie-beam sliding on the top can be observed in both models. Observed only in the ETABS model are concrete crushing (bottom left of panel P3&1) and sliding of the top panels, while in the Abaqus model sliding of the bottom of the wall is observed. While there is good overlap in the failure mechanisms, differences in the observed failure mechanism do exist and are attributed to the simplification of the ETABS model. There are two significant intrinsic limitation of the ETABS model as shown here. One is the inability to accurately simulate the location of the resultant axial force at the end of the panel, which will in reality vary depending on the axial loading. In the lumped model this is fixed. The second is the axial resistance of the axial link being one fixed value. Axial resistance of the link depends on the stress distribution, which depends on the axial force. This is modelled conservatively in the ETABS model. This approach is the reason for compressive axial failure in the ETABS model, while no failure occurred in the Abaqus model.

Working Paper of 36th DAAAM Symposium

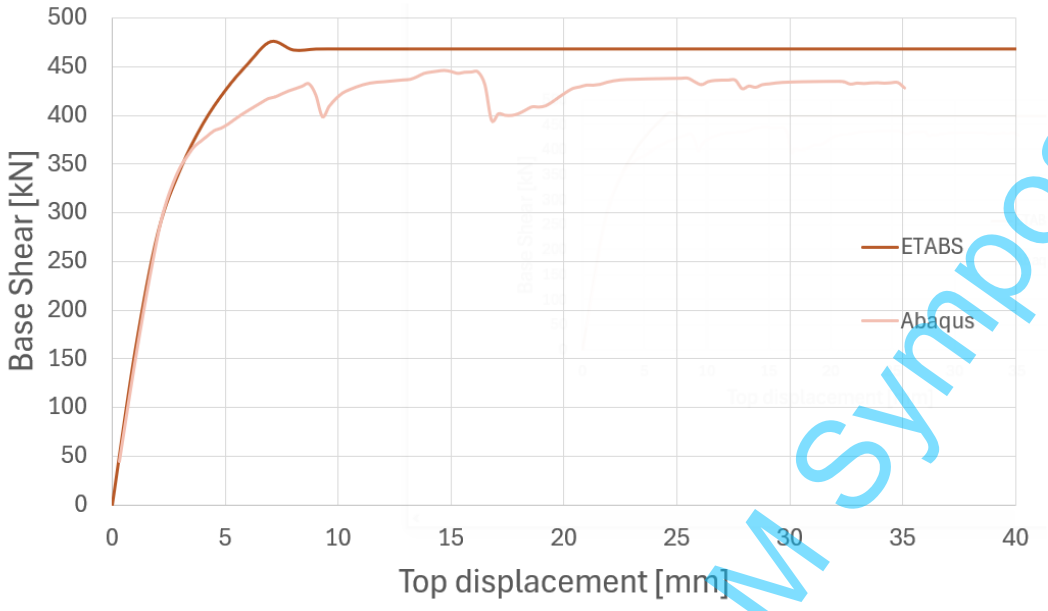

Fig. 3. Pushover curves of the assembled walls

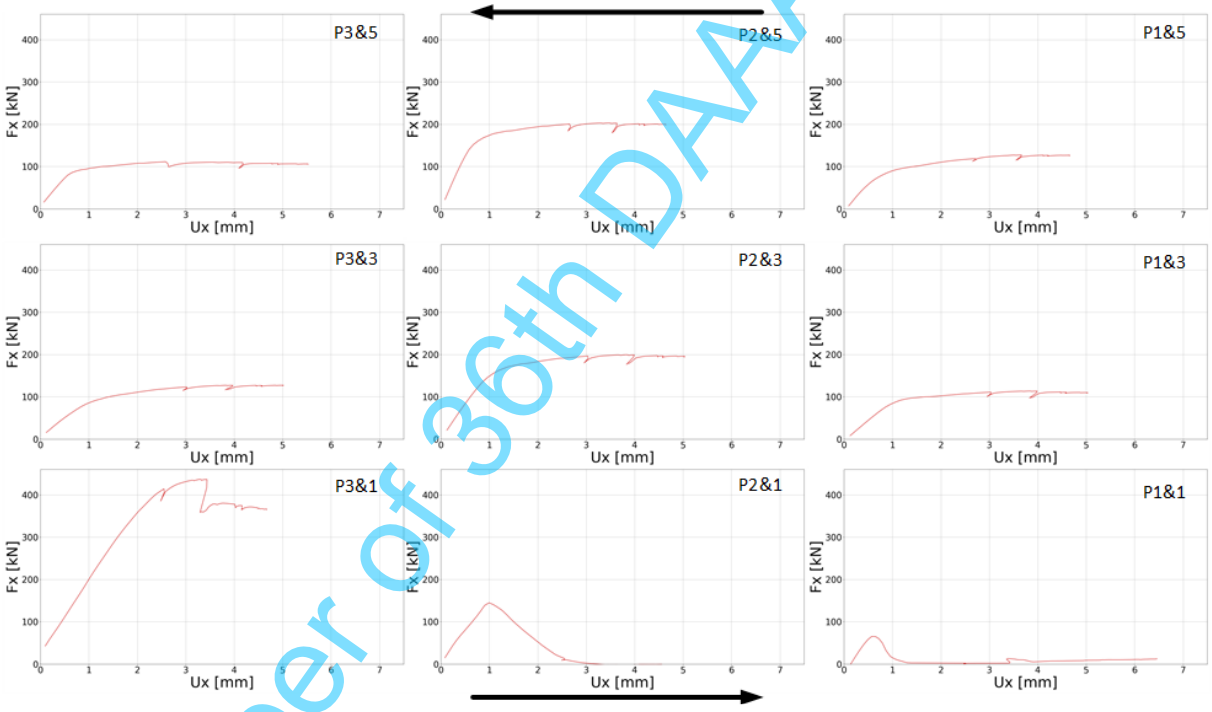

Fig. 4. Horizontal components of forces in individual panels (Abaqus)

Comparison of force distribution between Abaqus (Fig 4) and ETABS (Fig 5) shows almost identical distributions in the bottom row of panels, while differences appear in the upper layers of the wall. The Abaqus model shows the typical distribution of shear stress, with peak in the middle of the wall, indicating the wall is acting like a single deformable body, while the ETABS model shows a different distribution, more in line with shear transfer correlated to the axial loading of individual elements. The main cause of this discrepancy lies in the simulation of cohesion in the Abaqus model, which ETABS does not replicate, relying only on a friction coefficient. This should result with conservative results, and it seems that effect on failure mechanisms is small enough. In this example, differences with regards to internal forces are most pronounced in the top of the wall, with visible sliding of the top wall panel row in the ETABS model and no sliding in the ABAQUS model, which is also visible in the force distribution.

Working Paper of 36th DAAAM Symposium

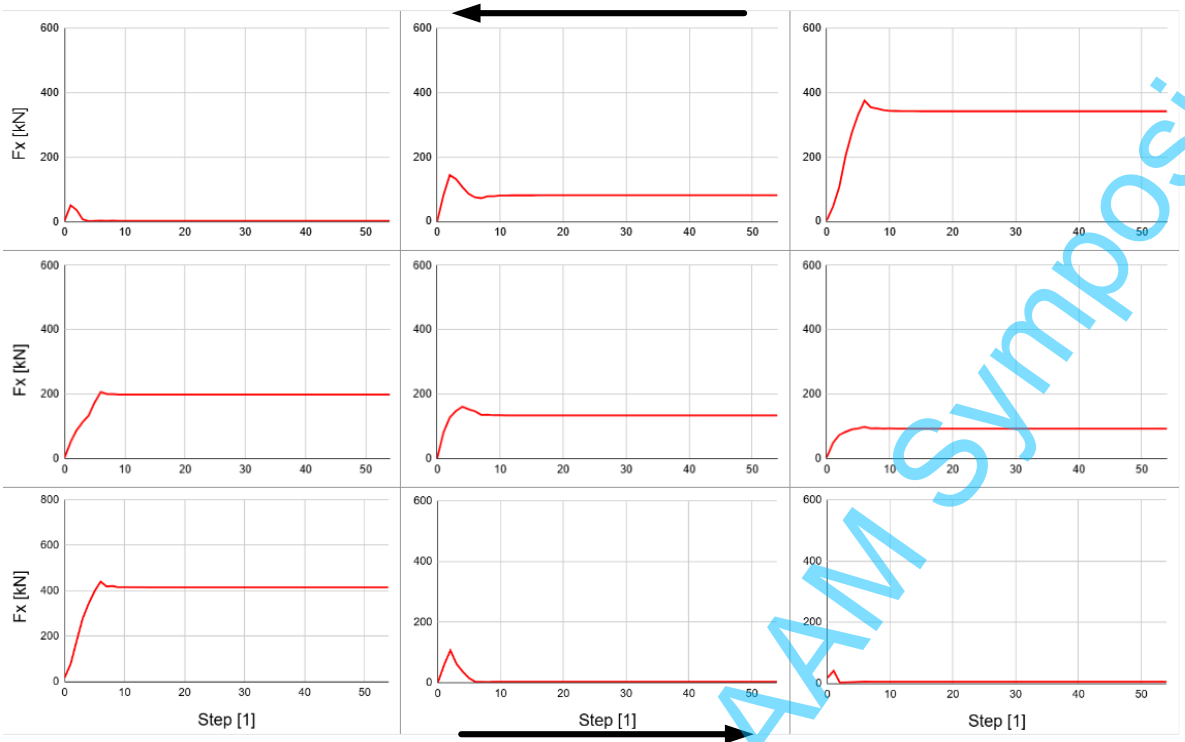

Fig. 5. Horizontal components of forces in individual panels (ETABS)

5. Conclusion

A set of pushover analyses has been conducted for various wall configurations, with results shown for a wall configuration which exhibits both relevant failure mechanisms and maximises differences between the models. Certain differences in force distribution and damage between the two models are observed, but these differences are acceptable given the simplified nature of the ETABS model and are generally conservative. Despite relying solely on a friction coefficient and bilinear behaviour, the ETABS model is able to reproduce the complex behaviour of large panel building walls reasonably well, enabling conducting a PSHA or similar analyses on a global 3D model of the entire Large Panel building. Limitations of this general approach are dependent on the choices in modelling used by the researcher and available data for calibration. Limitations of the model shown in this article mainly stem from assumptions made during simplification of the wall panel model, and its interaction of resistance and axial force as described in previous chapters. Furthermore, while the simplified model exists to make a global analysis of such buildings feasible, the computational load can still be very significant. This will be further developed, and a comprehensive PSHA analysis will be conducted in future work. Further research on simplification of complex numerical models is warranted.

6. References

[1] Zelinschi, G. A. (2011). The fiscal impact of the direct losses caused by a future earthquake to the public buildings in Romania. Annals of DAAAM & Proceedings, 1129-1131.

[2] Schricker, V.; Powell, GH. (1980). Inelastic Seismic Analysis of Large Panel Buildings.

[3] Mihaela, Zamolo. (1988). Ponašanje spojeva krupnopanelnih konstrukcija u seizmičkim područjima [Magistarski rad]. [Zagreb]: Građevinski fakultet sveučilišta u Zagrebu;

[4] Abaqus 2016 Documentation, link (http://130.149.89.49:2080/v2016/index.html), pristupljeno 2025 May 12

[5] Users Guide ETABS 2016, link (https://ottegroup.com/wp-content/uploads/2021/02/ETABS2016-Users-Guide.pdf), accessed 2025 May 12

[6] She, J.; Zou, Y.; Liu, Y.; Li, ZH.; Li, KW. (2014). Nonlinear Numerical Analysis of Precast Concrete Shear Wall, Applied Mechanics and Materials, 1192–6, 651–653, DOI: 10.4028/www.scientific.net/AMM.651-653.1192

[7] Henry, RS.; Sritharan, S.; Ingham, JM. (2016). Finite element analysis of the PreWEC self-centering concrete wall system, Eng Struct.; 115:28–41, https://doi.org/10.1016/j.engstruct.2016.02.029

[8] Paudel, S.; IIham, Maulana, TIiM.; Prayuda, H. (2024). Seismic Vulnerability Assessment of Regular and Vertically Irregular Residential Buildings in Nepal, Journal of the Civil Engineering Forum, 20;199–208, DOI: 10.22146/jcef.10316

[9] Michał, S.; Andrzej, W. (2015). Calibration of the CDP model parameters in Abaqus, Advances in Structural Engineering and Mechanics, Incheon, Korea, 25-29.8.

Working Paper of 36th DAAAM Symposium