JTUS, Vol. 4, No. 3 March 2026
E-ISSN: 2984-7435, P-ISSN: 2984-7427
DOI: https://doi.org/

Journal Transnational Universal Studies

Numerical Validation of Floor Plates Due to Concentric Axial Loads Using the Steel Stress-Strain Model Approach in Abaqus Software

Septa Melani*, Rosidawani, K.M. Aminuddin, Aurora Lutffeoneta
Universitas Sriwijaya, Indonesia
Email: melanisepta@gmail.com*, rosidawani@ft.unsri.ac.id, kmaminuddin@ft.unsri.ac.id

Abstract
This study aims to validate the numerical model of plates without openings (S1) and plates with central openings (S2) due to concentric axial loads using the ABAQUS-based Element Up Method. The model is developed with a nonlinear approach, then calibrated and verified against the results of previous research. The results of the load deflection curve calibration showed a percentage difference of 9.1% for S1 and 3.1% for S2, which is still within the tolerance limit. Comparison of the deformation distribution shows a maximum value difference of 3.2% (S1) and 1.2% (S2). There are differences in the visualization of the contour deformation that occur due to the difference in the condition of the elements and the modeling approach. The damage pattern in the S2 model shows similarities to the experimental results carried out. Overall, the numerical model developed has a good level of accuracy and validity and is suitable for advanced analysis of open plates.

Keywords: Floor plate, ABAQUS, displacement, calibration, FEM.

INTRODUCTION

In modern construction practice, slabs often require a central opening to provide a path for mechanical and electrical installations, such as HVAC lines, water pipes, or power lines (W. Grondzik et al., 2026; W. T. Grondzik & Kwok, 2019; Kiessling et al., 2014). As a result, the bearing capacity of the plates decreases, cracking occurs faster, and the deformation of the plates can increase significantly if no additional treatment is provided.

Previous studies have shown that plates with openings without special treatment have a 30–50% decrease in shear and flexural capacity compared to whole plates (Hassan et al., 2020; Meghdadaian & Ghalehnovi, 2019; Robert et al., 2021; Sabouri-Ghomi & Mamazizi, 2015). For example, a study by (Santos et al., 2024) demonstrated through finite element modeling that maximum stress concentrations occur at the corners of the opening, which are the initial crack-prone points. Meanwhile, research (El-Shafiey et al., 2022) experimentally showed that the shape and position of the opening strongly influenced the fracturing and deformation behavior of the slab.

The decrease in shear and bending capacity of plates with openings, as well as stress concentrations at the opening corners, indicate the need for modeling capable of capturing the behavior of structures in detail. In this case, the Finite Element Method (FEM) is an effective tool for predicting the structural response to various loads before physical testing.

The finite element method discretizes a continuous problem into a finite number of degrees of freedom, thereby simplifying the solution process (Rahma Nindya Ayu Hapsari, Adhe Verdianto Hidayat, 2022). The Finite Element Method (FEM) enables more optimal design by predicting how a model responds to internal stresses, temperature changes, and other mechanical forces before real-world implementation (Imron, 2025). In addition, experimental methods remain a crucial approach in validating numerical simulation results.

Validity in quantitative research refers to the level of accuracy and precision of an instrument in measuring what it is intended to measure (Subhaktiyasa, 2024). A measurement result is considered reliable only when repeated measurements of the same group of subjects yield consistent results (Ramadhan, 2024). The relationship between analytical and numerical results strengthens the validity of the approach used, as well as demonstrating how theoretical methods remain relevant in supporting the computer-aided design process (Yopi Zekri Jenizah Putra, 2025).

Through the process of calibrating numerical models against experimental results that have been available in the literature, it is hoped that this model can accurately represent real conditions. Furthermore, an analysis was carried out on the deformation graph, deformation distribution, and crack pattern on the plate.

The research conducted (Agung Prakasa, 2023) showed that the comparative results had different deformation values due to the influence of the mesh element conditions on each approach and the plane strain conditions in the analysis. Furthermore, in other comparative studies, the results of analysis using various types of mesh, both low and high order, show different abilities in capturing the stress distribution in the structure, therefore, the selection of the type of mesh must be carried out based on the specific criteria of the case being analyzed (Febrianto et al., 2024).

Based on the above background, this study was conducted to validate the numerical model of the opening plate developed using a numerical approach by modeling the stress-strain relationship of steel in the ABAQUS software referring to the plate configuration from the study, previously both experimental and numerical (El-Mawsly et al., 2022)

This research is expected to be able to make a scientific and practical contribution to the development of an efficient, economical, and easy-to-implement opening-plate reinforcement method in modern structural engineering practices with the existence of a numerical model that has been validated with experimental methods that have been carried out by the research (El-Mawsly et al., 2022), this model can subsequently be used for further research on slabs with and without openings using different types of materials.

This study aims to analyze the behavior of reinforced concrete slabs without openings and slabs with central openings subjected to concentric axial loads using a numerical modeling approach with the ABAQUS software. In addition, this study seeks to validate the developed numerical model by comparing the simulation results with previous experimental and numerical research findings. The study also evaluates the load–deflection relationship, deformation distribution, and damage patterns occurring in reinforced concrete slabs based on Finite Element Method (FEM) analysis. Through this process, the research is expected to assess the accuracy of the numerical model employing the steel stress strain model approach (Park & Paulay) in representing the structural behavior of reinforced concrete slabs and to produce a validated numerical model that can be used for further analysis of slabs with and without openings.

METHODS

The ABAQUS program is one of the *finite element method* programs that are used to simulate the process of making a material component and to analyze the strength of the material, then conclusions can be drawn from the results of the simulation and analysis as a result of the approach of the actual production process. After modeling is carried out with reference to the form and material from the research (El-Mawsly et al., 2022) The results will then be validated to determine whether the results of the *numeric* research with the stress-strain model approach are the same as the results obtained with *numeric* and experimental studies.

Structural Modeling

Structural modeling in this study references experimental specimens from (El-Mawsly et al., 2022), consisting of flat slabs measuring 1100 × 1100 × 120 mm, including one reference slab without openings and one slab with a central opening of 300 × 300 mm.

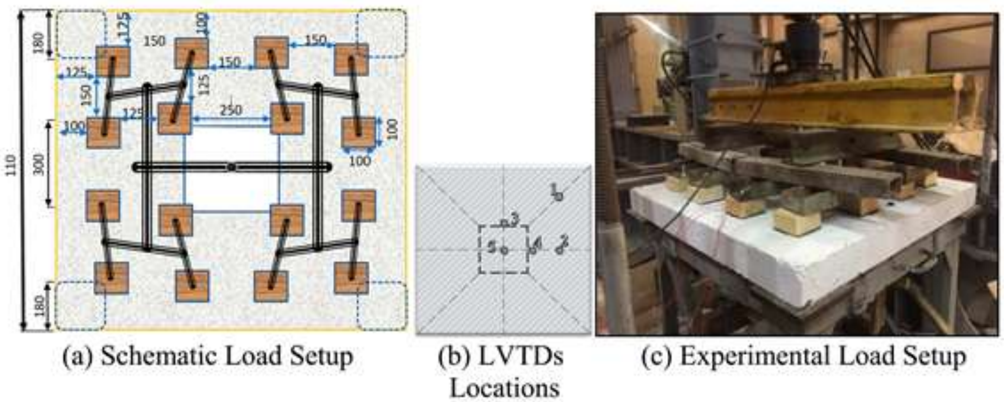


Figure - 1 Test set-up conducted (El-Mawsly et al., 2022)

The picture above is a *test set-up* of the load configuration applied to the slab, arranged to simulate uniformly distributed loading.

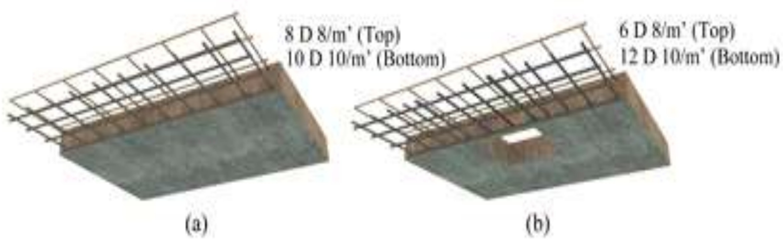


Figure - 2 Forms of 3-dimensional plate modelling performed (El-Mawsly et al., 2022)

The image above is the shape of the element that will be used at this stage. The stages begin with sketching the section cut and length of the element.

Tabel 1. Data Concrete Damage Plasticity (CDP)

Material	**C50**	**The parameters of CDP model**	
		Ψ	40^0
Concrete elasticity		ε	0.11
E (Mpa)	27,593.819	f_{b0}/f_{c0}	1.67
v	0.2	K	0.667
Concrete compression hardening		**Concrete compression damage**	
Stress [Mpa}	**Crushing strain [-]**	**Damage C [-]**	**Crushing strain [-]**
37.3514901	4.6382E-05	0	0.0007
44.4704111	0.000188392	0	0.008
51.2	0.000919183	0	0.000919
46.2148809	0.002199844	0.075702	0.0022
39.8990929	0.003228728	0.202018	0.003229
33.7709395	0.004250812	0.324581	0.004251
28.5019203	0.005241762	0.429962	0.005242
24.17198	0.006198679	0.51656	0.006199
20.6646998	0.007125782	0.586706	0.007126
17.8257574	0.008028666	0.643485	0.008029
15.5150767	0.008912405	0.689698	0.008912
13.9609147	0.009753243	0.720782	0.009753
12.4063207	0.010609582	0.751874	0.01061
11.1008798	0.011456891	0.777982	0.011457
Concrete tension hardening		**Concrete tension damage**	
Stress [Mpa]	**Cracking strain [-]**	**Damage T [-]**	**Cracking strain [-]**
7.8	0	0	0
8	0.00005	0	0.00005
7.6	0.000115968	0.05	0.000116
6.5	0.00092412	0.1875	0.000924
5.2	0.002333856	0.35	0.002334
4.7	0.004178472	0.4125	0.004178
3.5	0.01436916	0.5625	0.014369
2.5	0.0289014	0.6875	0.028901

Concrete Damage Plasticity (CDP) data refers to the stress and strain properties of concrete beyond its elastic limit (Lutffeoneta et al., 2025). This data is required for the estimation of concrete properties in the ABAQUS program. The following material data were obtained in experimental and numerical tests carried out (El-Mawsly et al., 2022).

Table 2. Rebar material data

Diameter (mm)	Elongation %	Yield Stress F_y (N/mm^2)	Ultimate Stress F_u (N/mm^2)
8	25	302	456
10	21	413	644

Plate reinforcement consists of a lower reinforcement with a diameter of 10 mm every 100 mm and an upper reinforcement with a diameter of 8 mm every 125 mm.

The equation approach of R. Park & T. Paulay (1975) is carried out by replacing the actual steel tension–strain curve with an ideal elastic–plastic model (bilinear) in order to simplify calculations in reinforced concrete analysis

a. Elastic Range

$f_s = E_s \varepsilon_s$

b. Yield strain

$$\varepsilon_y = \frac{f_y}{E_s}$$

c. *Strain Hardening*

$$f_s = \begin{cases} E_s \varepsilon_s \\ f_y + E_{sh}(\varepsilon_s - \varepsilon_y) \end{cases}$$

Table 3. Material data using the park and paulay method

Diameter 10		Diameter 8	
fy	413	*fy*	302
fu	644	*fu*	456
E (Mpa)	200000	*E (Mpa)*	200000
εy	0.002	*εy*	0.002
εsh	0.02	*εsh*	0.02
εsu	0.2	*εsu*	0.2
q	0.18	*q*	0.18
n	40.96	*n*	40.96
m	107.1396	*m*	102.977
Mass density	78.53982	*Mass density*	50.26548
Stress [Mpa]	**strain [-]**	**Stress [Mpa]**	**strain [-]**
413	0	302	0
413	0.002	302	0.002
413	0.003	302	0.003
413	0.004	302	0.004
413	0.005	302	0.005
413	0.006	302	0.006
413	0.007	302	0.007
413	0.008	302	0.008
413	0.009	302	0.009

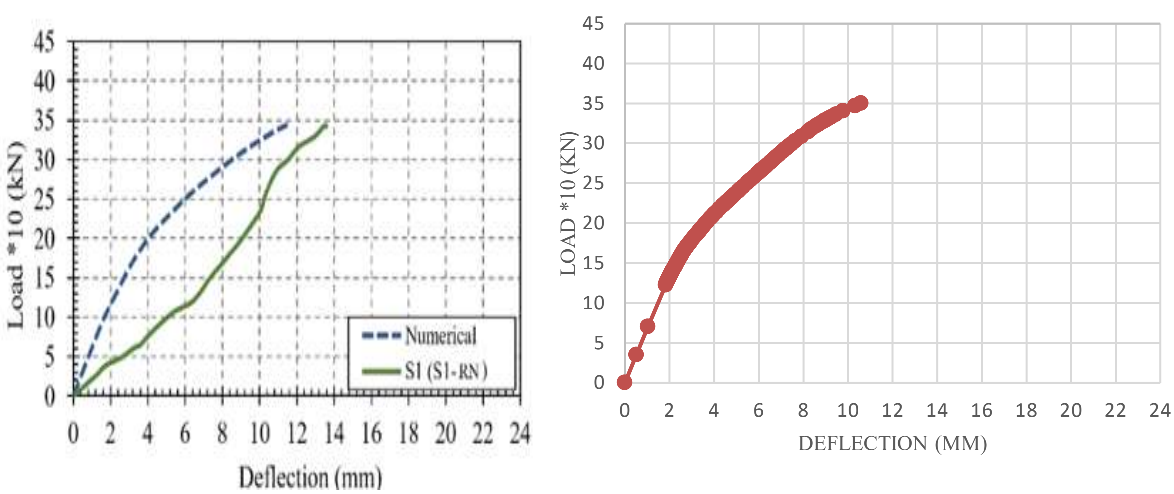
413	0.01	302	0.01
413	0.011	302	0.011
413	0.012	302	0.012
413	0.013	302	0.013
413	0.014	302	0.014
413	0.015	302	0.015
413	0.016	302	0.016
413	0.017	302	0.017
413	0.018	302	0.018
422.2131	0.019	308.1421	0.019
430.8913	0.02	313.9276	0.02

The mesh on the modeling of plate structures uses hexahedral elements. The approach on the model is given *a boundary condition* around the plate with *a spring / elastic foundation* which serves as a support condition for the slab.

RESULTS AND DISCUSSION

Load Curve Analysis – Deformation

The plate models used in this study are based on specimens from the study (El-Mawsly et al., 2022) The elements used are only elements S1 (Non opening) and S2 (Opening) by being modeled using *a finite element* auxiliary program. The results of modeling with *the finite element* assistance program (Abaqus 2020) are an approach. However, there are limitations that are used as a control that numerical results can be validated. So the results are not exactly the same as the results of the previous research. The following is presented a load-displacement graph as a verification with previous research.



Gambar - 3 Graphic load-deflection S1 Plate verification *(Non Bukaan)*

If you look at the graph in Figure - 3, it can be concluded that in general the two graphs have the same trend. From the graph, it can also be seen that the deflection value of the S1 modeling results (Non-opening) is smaller when compared to the modeling results (El-Mawsly et al., 2022) at the end of the deflection, however, the difference is not significant.

Table 4. Verification of Previous Research and S1 Modelling

No	Variation	*Load-Deflection*	
		Load (kN)	**Deflection (mm)**
1	Study (El-Mawsly et al., 2022)	35	11,75
2	S1 (*Non Bukaan*)	35	10.68
	Difference	0	1.07
	Percentage %	9.1%	

The table above shows the difference between the previous study and the S1 modeling results. There is a deflection difference of 1.07 mm, representing a percentage difference of 9.1%.

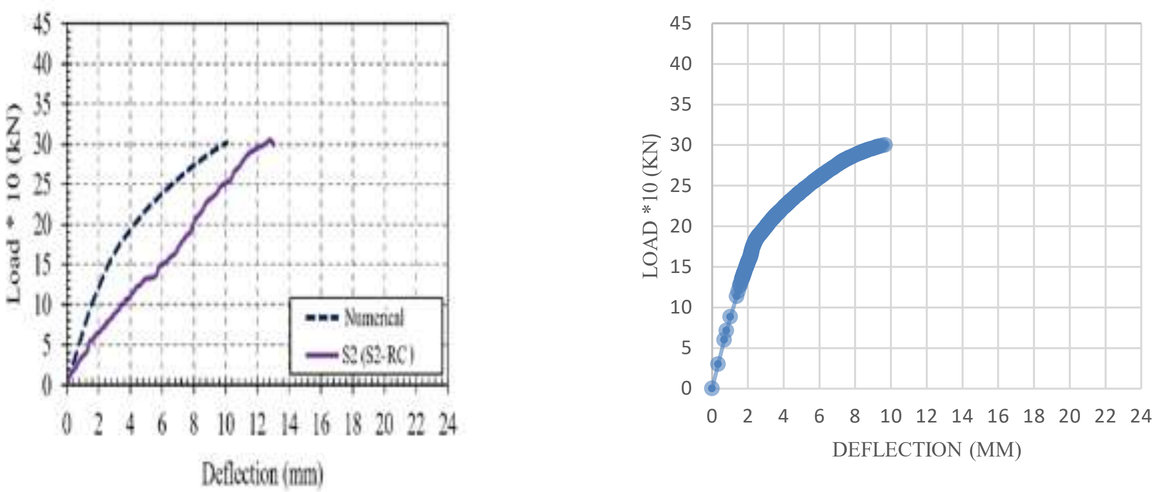


Figure - 4 Load-deflection verification S2 Plate (Opening)

As shown in Figure - 4, the two graphs follow the same general trend. From the graph, it can also be seen that the deflection value of the S2 modeling results (Opening) is slightly smaller when compared to the modeling results (El-Mawsly et al., 2022) at the end of the deflection, however, the difference is not significant.

Table 5. Verification of Previous Research and S2 Modelling

No	Variation	*Load-Deflection*	
		Load (kN)	**Deflection (mm)**
1	Study (El-Mawsly et al., 2022)	30	10
2	S2 *(Bukaan)*	30	9.69
	Difference	0	0.31
	Percentage %	3.1%	

The table above shows the difference between the previous study and the S2 modeling results. There is a deflection difference of 0.31 mm, representing a percentage difference of 3.1%.

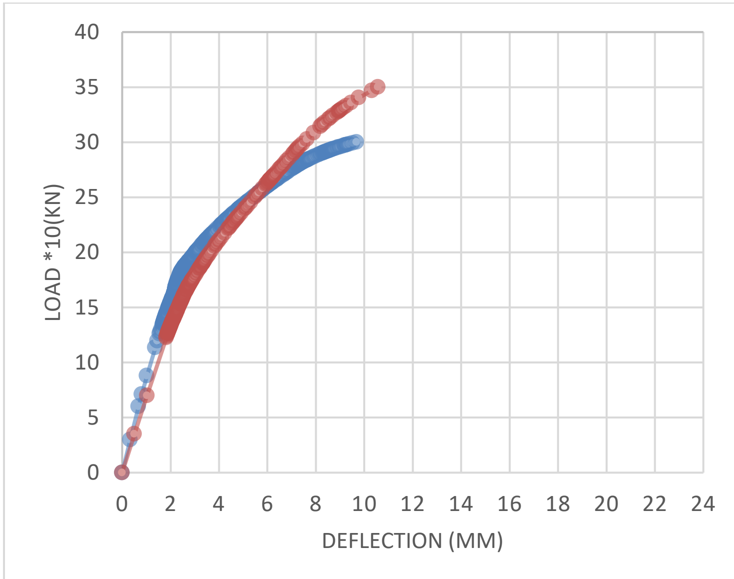
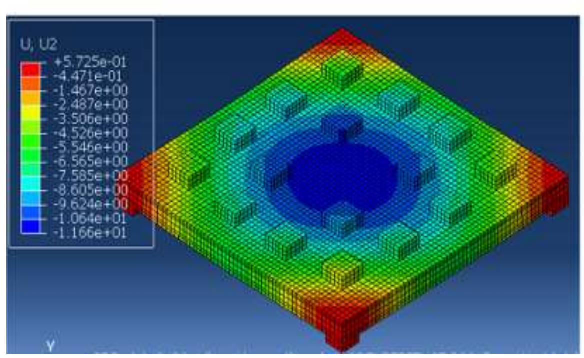


Figure - 5 Modelling verification recapitulation graph

The modeling graph has verified that the results obtained do not have significant differences in the results of the study (El-Mawsly et al., 2022) and the results of S1 (Non-opening) and S2 (Opening) modeling.

Comparison of Deflection Distribution on Plates

The results verified in the modeling are the similarity of the contour distribution and the value results in the table next to the modeling, The modeling results with *the finite element* auxiliary program (Abaqus 2020) is an approach. Thus, the results are not exactly the same as the results of the previous research.

a. Research (El-Mawsly et al., 2022)	

b. S1 *(Non Bukaan)*

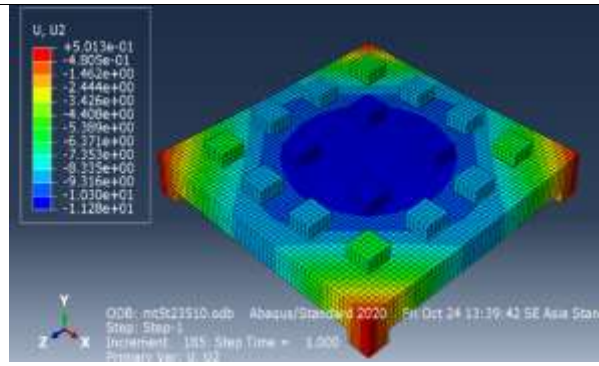


Figure - 6 S1 modelling deflection distribution

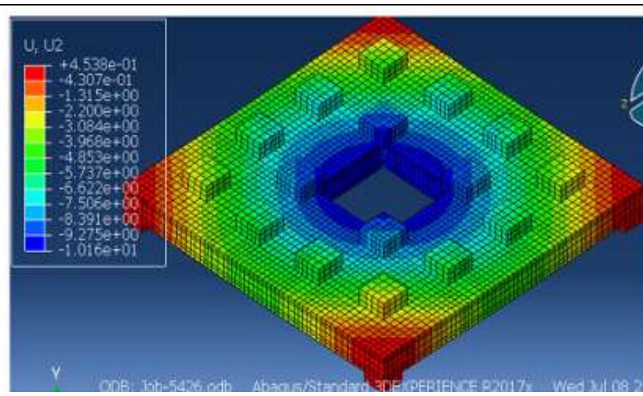
The contours of the structure that underwent deformation in S1 Modeling, show different visualizations; however, referring to the tabulated values alongside the model, the results can be summarized as follows:

Table 6. Comparison Table of S1 Modelling Distribution Results

No	Variation	***U – U2***	
		Min (mm)	**Maks (mm)**
1	Research (El-Mawsly et al., 2022)	0.56725	0.56725
2	S1 *(Non Bukaan)*	0.5013	0.5013
	Difference	0.05422	0.38
	Percentage %	9.5%	3.2%

Furthermore, verification was carried out on the S2 modeling (Opening), the following are the results obtained in the ABAQUS program.

a. Research (El-Mawsly et al., 2022)



b. S2 *(Bukaan)*

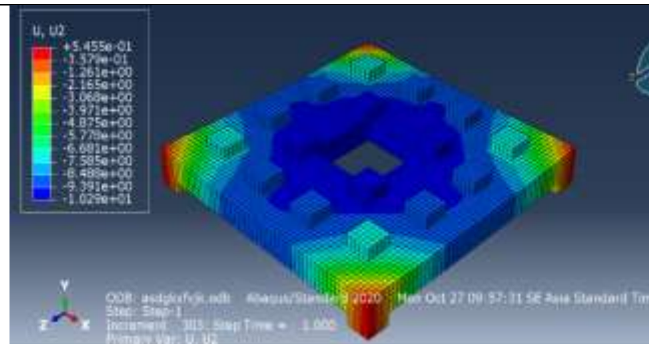


Figure - 7 S2 modelling deflection distribution.

The contours of the structure that underwent deformation in S2 Modeling, show different visualizations; however, referring to the tabulated values alongside the model, the results can be summarized as follows:

Table 7. Table of Results of Comparison of Modelling Distribution S2.

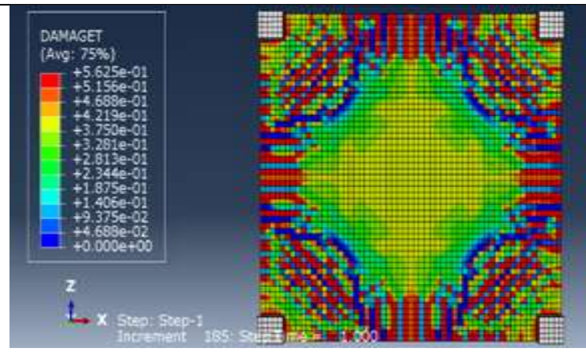
No	Variation	***U – U2***	
		Min (mm)	**Maks (mm)**
1	Research (El-Mawsly et al., 2022)	0.4538	10.16
2	S2 *(Bukaan)*	0.5455	10.29
	Difference	0.0925	0.13
	Percentage%	20.3%	1.2%

The verification results obtained can be used because there is no significant difference in values in the study (El-Mawsly et al. 2022) and S1 (Non-opening) and S2 (Opening) modeling.

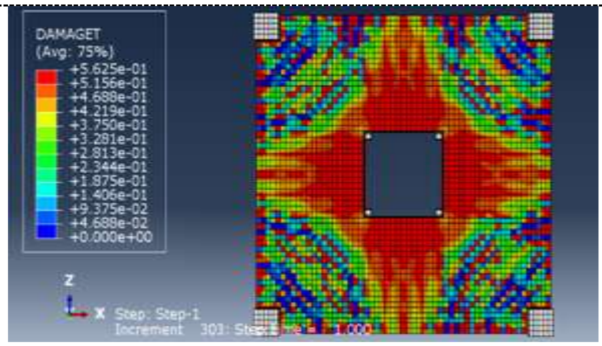
Pattern of Damage to Plates

The damage pattern in the Abaqus program is obtained in the *damageT* display resulting from numerical simulations using the Tensile Damage parameter, which closely resembles the experimental damage pattern. This suggests that the damage variable (damageT) can be used to visualize and predict damage patterns in concrete structures, particularly when the CDP model is employed. The material data and parameters used have been calibrated against relevant experimental data.

a. S1 (Non Bukaan)



b. S2 (Bukaan)



c. Research (El-Mawsly et al., 2022)

Figure - 8 Damage patterns in experimental research and plate modelling verification

Verification of the results of the damage pattern approach was carried out in S2 (Opening) modeling because it could be compared with the results of the research (El-Mawsly et al. 2022). The damage pattern obtained in S2 modeling is slightly wider than the results obtained in experimental research.

CONCLUSION

Based on the results of modeling validation, it can be concluded that the numerical model developed shows a good level of conformity with previous research. At the load-deflection calibration stage, S1 (non-opening) modeling had a percentage difference of 9.1%, while S2 (opening) modeling had a difference of 3.1%, both within acceptable tolerance limits. The difference in visualization of the contour deformation that occurred was influenced by the difference in the condition of the elements and the modeling approach, when compared to previous research. However, the comparative value of the deformation distribution showed a relatively small difference in the maximum value, which was 3.2% for S1 and 1.2% for S2, thus strengthening the validity of the model. In addition, the damage pattern in S2 modeling (opening) using the ABAQUS program shows similarities to the results of experimental research conducted by (El-Mawsly et al. 2022). Thus, numerical models using the steel stress-strain model approach (Park & Paulay, 1975) can accurately represent the structural behavior and are suitable for further analysis.

BIBLIOGRAPHY

Agung Prakasa, & Dwiyanto, M. (2023). *Studi komparasi analisis numerik 2D dan 3D pada kasus galian dalam di Jakarta Pusat dengan metode elemen hingga* (Vol. 3, No. 2).

El-Mawsly, E. H., El-Kashif, K. F. O., Shawky, A. A., & Abdalla, H. A. (2022). Experimental and numerical investigation on strengthening of RC flat slabs with central opening. *Case Studies in Construction Materials, 16*, e00974. https://doi.org/10.1016/j.cscm.2022.e00974

El-Shafiey, T. F., Atta, A. M., Hassan, A., & Elnasharty, M. (2022). Effect of opening shape, size and location on the punching shear behaviour of RC flat slabs. *Structures, 44*, 1138–1151. https://doi.org/10.1016/j.istruc.2022.08.076

Febrianto, P., Muzakki, M. G., & Sastranegara, A. (2024). Studi komparatif mesh hexahedral dan tetrahedral pada analisis statis crane hook dengan ANSYS. *22*(1), 63–76.

Grondzik, W., Overbey, D., & Kwok, A. (2026). *Mechanical and electrical equipment for buildings*. John Wiley & Sons.

Grondzik, W. T., & Kwok, A. G. (2019). *Mechanical and electrical equipment for buildings*. John Wiley & Sons.

Hassan, A., Atta, A. M., & El-Shafiey, T. F. (2020). Restoration of the shear capacity for RC beams with web openings using precast SHCC plates. *Structures, 25*, 603–612.

Hapsari, R. N. A., Hidayat, A. V., & E. S. (2022). Jurnal Teknik Sipil Unaya. *8*(1), 108–117.

Imron, J. (2025). *Analisis desain dan keandalan bejana bertekanan pada industri energi pendekatan numerik dan eksperimental*, 4(April).

Kiessling, F., Nefzger, P., Nolasco, J. F., & Kaintzyk, U. (2014). *Overhead power lines: Planning, design, construction*. Springer.

Lutffeoneta, A., & Aminuddin, K. M. (2025). Pengaruh variasi jarak tulangan spiral terhadap perilaku kolom akibat beban aksial konsentris menggunakan finite element analysis. *10*(6), 1–13.

Meghdadaian, M., & Ghalehnovi, M. (2019). Improving seismic performance of composite steel plate shear walls containing openings. *Journal of Building Engineering, 21*, 336–342.

Putra, Y. Z. J. (2025). Analisis komparatif metode Rayleigh dan Dunkerley terhadap hasil. *5*(2), 16327–16339.

Rahma Nindya Ayu Hapsari, A. V. H., & E. S. (2022). Jurnal Teknik Sipil Unaya. *8*(1), 108–117.

Ramadhan. (2024). Validitas dan reliabilitas. *06*(02), 10967–10975.

Robert, J. B., Prabhavathy, R. A., Joanna, P. S., Singh, S. C. E., Murugan, S., Rajkumar, S., & Sharma, S. (2021). Flexural behaviour of RC beams with a circular opening at the flexural zone and shear zone strengthened using steel plates. *Advances in Civil Engineering, 2021*.

Sabouri-Ghomi, S., & Mamazizi, S. (2015). Experimental investigation on stiffened steel plate shear walls with two rectangular openings. *Thin-Walled Structures, 86*, 56–66.

Santos, J. B. dos, Melo, G. de S., & Fernández Ruiz, M. (2024). Punching performance of flat slabs with openings accounting for the influence of moment transfer and shear reinforcement. *Engineering Structures, 303*, 117461. https://doi.org/10.1016/j.engstruct.2024.117461

Subhaktiyasa, P. G. (2024). Evaluasi validitas dan reliabilitas instrumen penelitian kuantitatif: Sebuah studi pustaka. *Journal of Education Research, 5*(4), 5599–5609.