fib Congress 2026
Lisbon - Portugal

High-fidelity finite element modeling for seismic assessment of safety-critical structures in nuclear power plants

Jan Cervenka[1*], David Kurmann [2], Marios Lacerda [2], Mohamad Ahmad[1], Gabriela Hulkova [1]

1. Cervenka Consulting s.r.o., Prague, Czech Republic

2. Axpo Power AG, Kernenergie, Baden, Switzerland

[*]Corresponding author email: jan.cervenka@cervenka.cz

Abstract

The paper presents a high-fidelity nonlinear finite element modeling approach for the seismic evaluation of safety-critical reinforced concrete structures using the ATENA software. The methodology combines advanced constitutive models for concrete and reinforcement, detailed representation of key structural components, dynamic time-history analysis, and consistent treatment of damping and boundary conditions.

The modeling uncertainty validated against the CAMUS III shake-table benchmark, demonstrating its ability to reproduce global response, damage evolution, and localized failure mechanisms under seismic loading. It is subsequently applied to the seismic assessment of selected reinforced concrete buildings forming part of a nuclear power plant's critical infrastructure. Soil–structure interaction is considered using a simplified substructure approach based on equivalent foundation springs.

The results highlight the importance of nonlinear material behavior and modeling assumptions in seismic response prediction and demonstrate the potential of high-fidelity finite element analysis to support more reliable safety evaluations of existing nuclear structures.

Keywords: Nonlinear seismic analysis, dynamic analysis, finite element modeling, nuclear power plant safety, benchmark, soil-structure interaction (SSI)

1. Introduction

Seismic safety assessment of nuclear power plant structures remains a key engineering challenge, particularly for reinforced concrete buildings designed and constructed before the introduction of modern seismic design standards. Many existing nuclear facilities were developed using simplified linear analysis methods and conservative design assumptions, which do not explicitly capture nonlinear material behavior, damage evolution, or complex interaction effects under strong ground motion.

Since the 1990s, significant efforts have been undertaken within the nuclear engineering community to improve the realism of seismic analyses through advanced numerical modeling. Early applications of nonlinear finite element methods to reinforced concrete nuclear structures were carried out within international benchmark programs coordinated by organizations such as the OECD/NEA and CEA, focusing on the ability of numerical models to reproduce experimentally observed seismic response and damage mechanisms (Sigmund et al. 2001). More recent benchmark initiatives additionally addressed nonlinear dynamic analyses and soil–structure interaction effects (Richard et al., 2016). These studies demonstrated that high-fidelity models are capable of capturing key response features, including cracking, reinforcement yielding, stiffness degradation, and localized failure.

Experience from these programs has shown that the seismic behavior of reinforced concrete nuclear structures cannot be adequately represented using linear or equivalent-static approaches. Although simplified nonlinear procedures, such as pushover analysis, are now included in major design codes, their applicability to safety-critical nuclear structures remains limited, particularly when dynamic effects, three-dimensional response, and soil–structure interaction play a significant role.

Advances in finite element technology, constitutive modeling of concrete and reinforcement, and computational efficiency have enabled the practical application of nonlinear dynamic analyses to large-scale nuclear structures (Lee and Fenves, 1998; Proske et al., 2013; Stäuble Akcay et al., 2019). At the same time, various studies have highlighted the importance of soil–structure interaction modeling, including nonlinear effects, for reliable seismic response prediction of nuclear facilities (Wolf, 1985; Stäuble Akcay et al., 2019).

This paper presents a nonlinear finite element modeling framework for the seismic assessment of reinforced concrete nuclear structures using the ATENA software. The approach is validated against the CAMUS III benchmark and applied to selected nuclear power plant buildings, with consideration of soil–structure interaction.

2. Nonlinear Modeling

This section presents the high-fidelity nonlinear modeling approach adopted for the seismic assessment of nuclear power plant structures. The modeling procedure is implemented using the finite element software ATENA and aims to capture material nonlinearity, damage evolution, soil–structure interaction, and the influence of key modeling uncertainties.

The nonlinear finite element model (Argyris 1954, Zienkiewicz and Cheung Y.K. 1967) is developed as a three-dimensional representation of a complex reinforced concrete structure intended for seismic response analysis. The nonlinear finite element technique involves approximations of a theoretical continuum by discrete numerical representations, as illustrated in Figure 1.

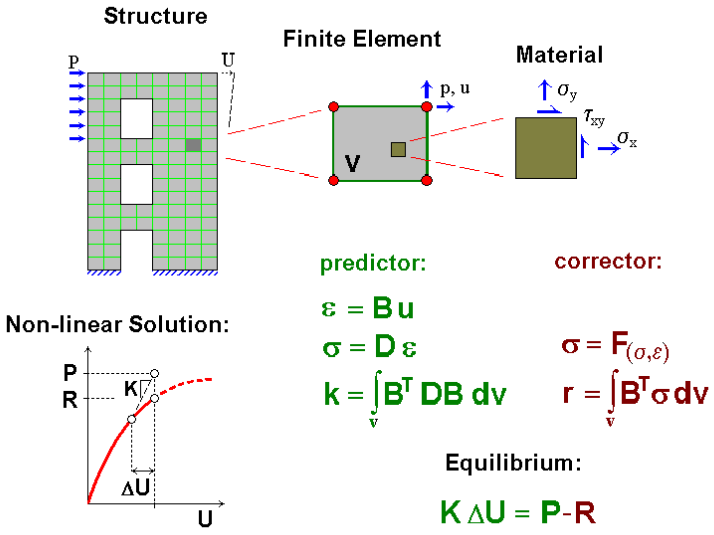


Figure 1. Key steps in nonlinear FE solution process.

For efficient modeling of reinforced concrete structures, the model needs to take into account key elements of reinforced concrete such as: nonlinear constitutive model, nonlinear interfaces or contacts between different materials, and reinforcement (see Figure 2). The mixed finite element types are recommended for modeling of complex reinforced concrete buildings, that combine solid and shell elements according to the structural function of individual building components.

Columns can be effectively represented using linear isoparametric solid elements, which provide an adequate description of stress redistribution and local nonlinear effects.

Structural walls and slabs are suitably modelled using quadratic three-dimensional shell elements specifically developed for reinforced concrete applications, as described by Jendele and Červenka (2014) and Červenka et al. (2025). These elements are defined by a three-dimensional geometry with twelve nodes and translational degrees of freedom only, which facilitates their seamless integration with standard solid elements within a single numerical model. Internally, the shell elements are formulated as layered shell elements incorporating both displacement and rotational degrees of freedom (Jendele and Červenka 2014, Figure 2). This formulation enables the definition of internal reinforcement layers and significantly simplifies the modeling of reinforcement in slabs and walls, while maintaining a realistic representation of stiffness, load-carrying capacity, and damage development.

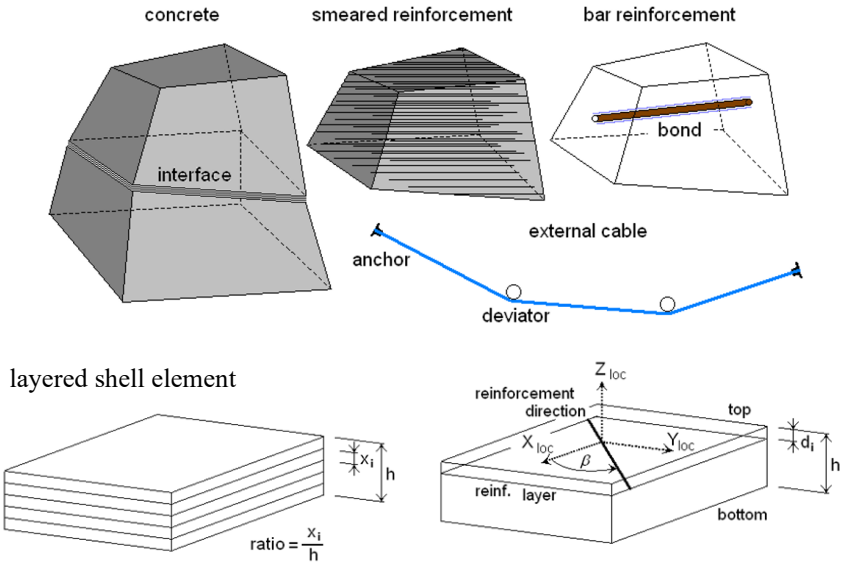


Figure 2. Key elements for high-fidelity large-scale models of reinforced concrete structures.

The reinforcement modeling strategy should be based on the identification of critical structural regions, where a detailed discrete reinforcement model is recommended. In these regions, reinforcement bars and stirrups are explicitly modelled as embedded truss elements within three-dimensional solid finite elements, enabling a realistic simulation of reinforcement yielding and interaction with the surrounding concrete. In the remaining regions of the structure, reinforcement can be modelled using a smeared reinforcement approach within a composite reinforced concrete material model, or through internal reinforcement layers in shell elements, as described in the ATENA documentation. This approach provides a balance between accuracy and computational efficiency.

Figure 4 and Figure 5 show an example of a reinforced concrete building taken from the application case in Section 4. The most critical beam and column elements are modelled with brick elements with discrete reinforcement option (Figure 4). In the case of reinforced concrete walls and slabs (Figure 5), it is more efficient to model the reinforcement using either the smeared option (see Figure 2) or internal reinforcement layers within shell elements.

In practical applications, the interaction between the structure and the underlying soil can be represented using a simplified approach, in which the foundation is modelled by equivalent elastic springs. This Winkler-type representation enables soil stiffness to be introduced through discrete springs defined at the foundation level. This approach is commonly used in nonlinear and dynamic analyses of reinforced concrete structures.

The adopted modeling strategy represents a compromise between numerical accuracy and computational efficiency, enabling the analysis of complex structural systems under nonlinear seismic loading. It

provides a reliable approach for investigating the influence of material nonlinearity, reinforcement detailing, and structural layout on the seismic response of reinforced concrete buildings.

The nonlinear material model for concrete in ATENA combines two constitutive models for compressive and tensile behavior, Červenka and Papanikolaou (2008). The Menetrey and Willam (1995) model is employed to describe concrete plasticity under multiaxial compression (Figure 3-a) with nonlinear hardening and softening behavior (Figure 3–b, c).

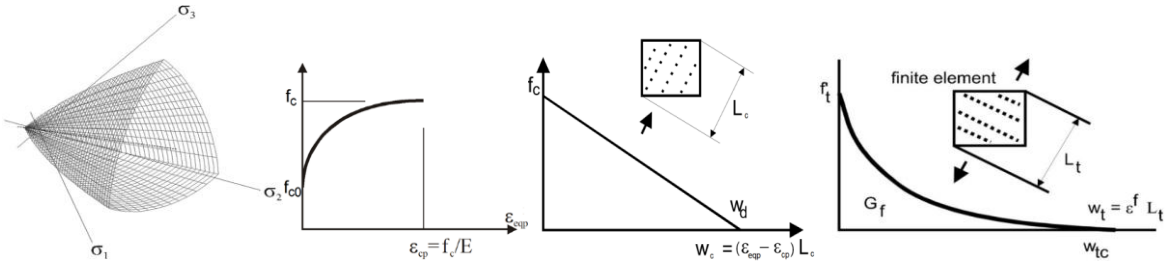


Figure 3. Three-parameter Menetrey and Willam (1995) concrete failure criterion in principal stress frame (a), hardening/softening law for plasticity model of concrete under compression (b, c) and tensile softening (d).

For tensile cracking, the Rankine criterion with exponential softening, as proposed by Hordijk (1991), is utilized (Figure 3d), where w_t represents the crack width. The crack band model of Bažant and Oh (1983) is applied to link crack opening displacement to fracturing strains. A similar approach is adopted for compression following Červenka et al. (2014). The crack band L_t and crush band sizes L_c are adjusted based on the crack orientation method Červenka et al. (2018). This paper also addresses the issue of large finite elements when the crack spacing limiter should be applied on L_t for tensile cracking and crush band limiter on L_c.

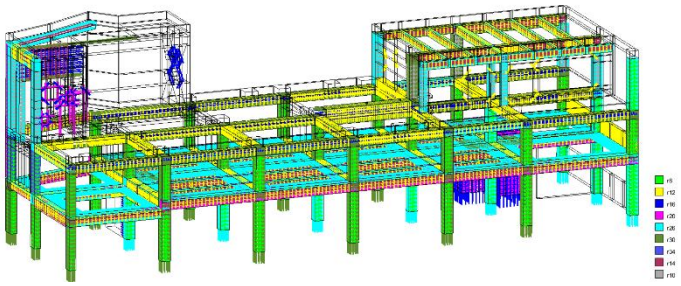


Figure 4. Discrete reinforcement model for the reinforced concrete building example

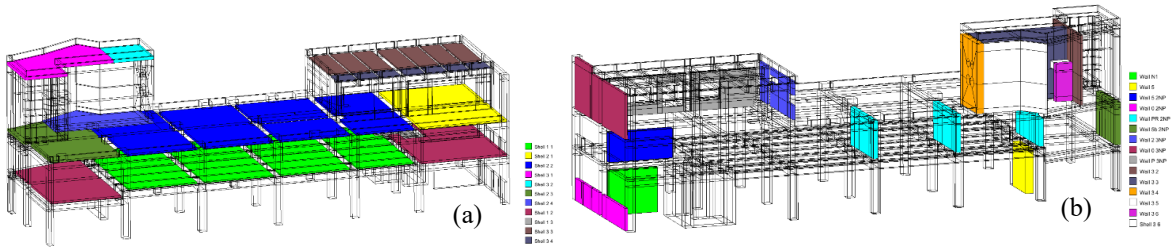


Figure 5. Structural elements (building 1UN-D) with smeared reinforcement model inside shell elements (a) and solid elements (b)

3. Model Validation and Benchmark Studies

Benchmark tests provide an effective means for validating modeling approaches applied to complex reinforced concrete structures, as the comparison of numerical results with experimental data allows the accuracy of the model and the associated model uncertainties to be assessed. This section briefly introduces the CAMUS III experiment, which was used to verify the selected modeling approaches

implemented in the ATENA software, and to compare the numerical predictions with available experimental measurements.

The CAMUS experimental program (*Conception et Analyse des Murs sous Séisme*) comprises a series of shake-table tests conducted at the Azalée facility of CEA-Saclay on scaled reinforced concrete wall structures subjected to in-plane seismic loading. Within this program, the CAMUS III (Figure 6) specimen was designed in accordance with early Eurocode 8 principles and has been widely adopted as an international benchmark for the validation of nonlinear numerical models with the focus on the ability to reproduce both global and local seismic response (Sigmund, Guljaš and Matoševič 2001).

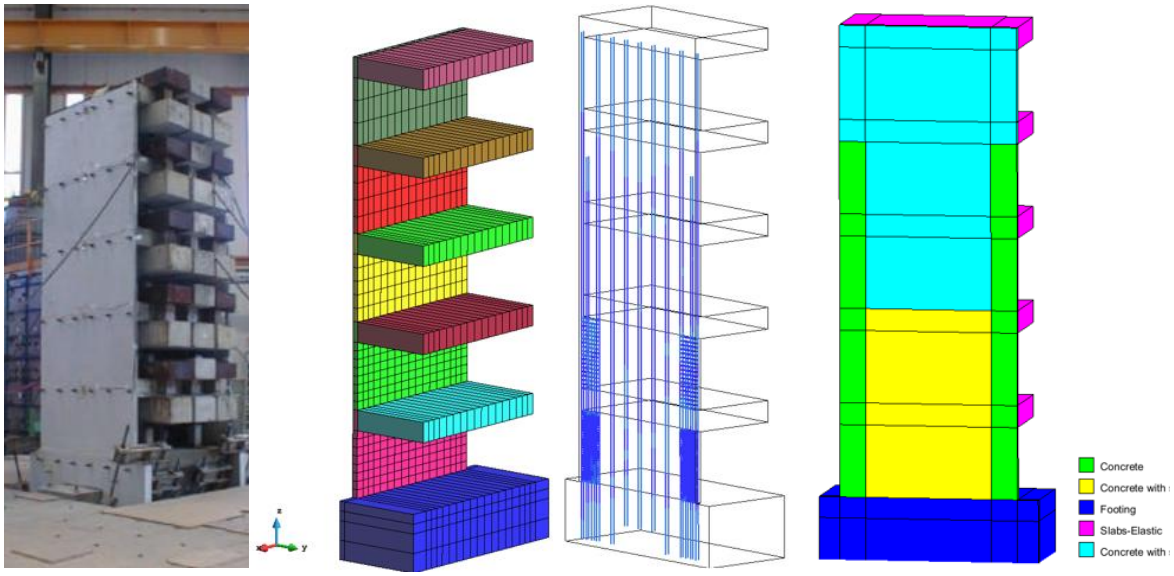


Figure 6. Experiment setup and finite element model for CAMUS III benchmark.

A nonlinear finite element model of the CAMUS III specimen was created in ATENA (Cervenka et al. 2025) to serve as a validation example for dynamic analysis. Owing to the structural symmetry, only half of the scaled five-story building was modelled. The numerical model focused primarily on the reinforced concrete wall, while the slabs were included mainly to represent the structural masses.

Reinforcement was modelled using a combined approach as recommended in Section 2. Horizontal reinforcement in the wall was represented by the smeared reinforcement model, whereas the critical vertical reinforcement bars were modelled discretely to account for their exact position and structural significance (Figure 7). Dynamic time-history analysis was performed, and the numerical response was evaluated in terms of floor displacements and damage development. The simulations reproduced crack opening and closing as well as localized concrete crushing, showing good agreement with the experimentally observed damage patterns (Figure 7).

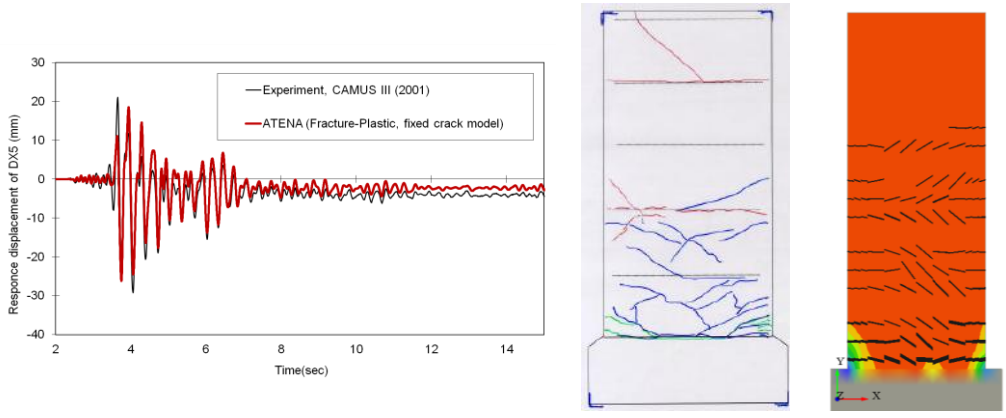


Figure 7. CAMUS-III response comparison

4. Case Study: Application to a Nuclear Power Plant Structures

The case study includes four reinforced concrete buildings forming part of a nuclear power plant complex and belonging to its critical infrastructure. The buildings are located in a seismic area and were constructed around the 1960s, reflecting the design practices of that period. All structures are 3-to-4-story reinforced concrete frame buildings. Due to their age, structural system, and seismic exposure, they represent suitable candidates for seismic assessment using nonlinear dynamic analysis.

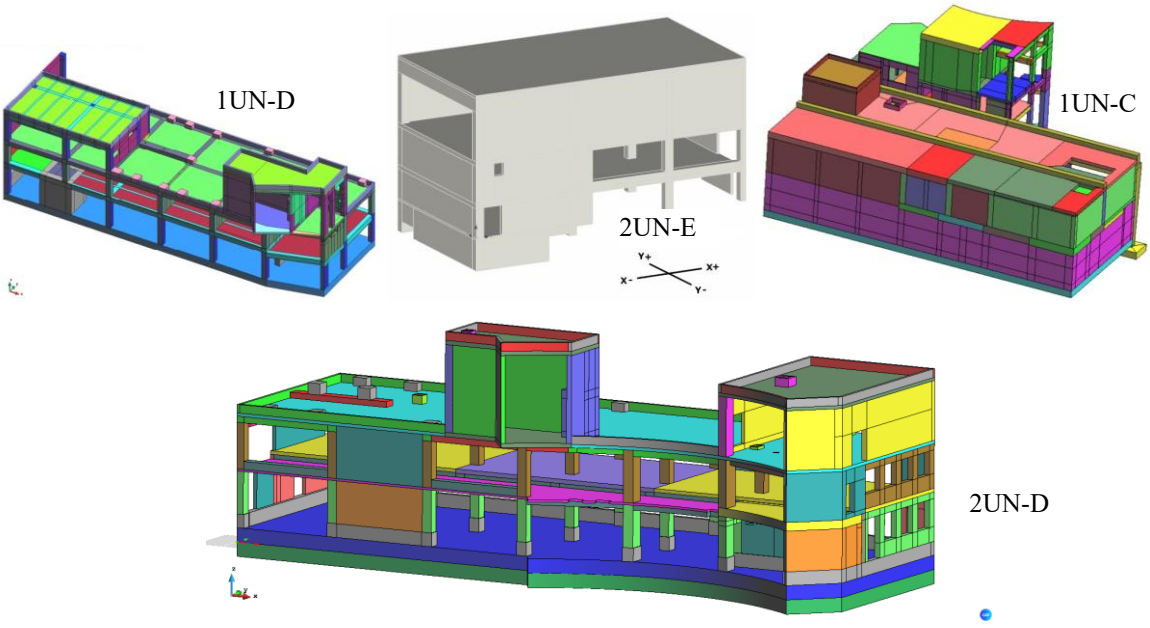


Figure 8. Numerical models of investigated nuclear structures.

Figure 8 shows the numerical models of the four analyzed buildings. The modeling strategy follows the principles described in Section 2 and has been adapted to the geometry and available documentation of the assessed structures. Columns are modelled using three-dimensional solid finite elements, while walls and slabs are represented by three-dimensional shell elements.

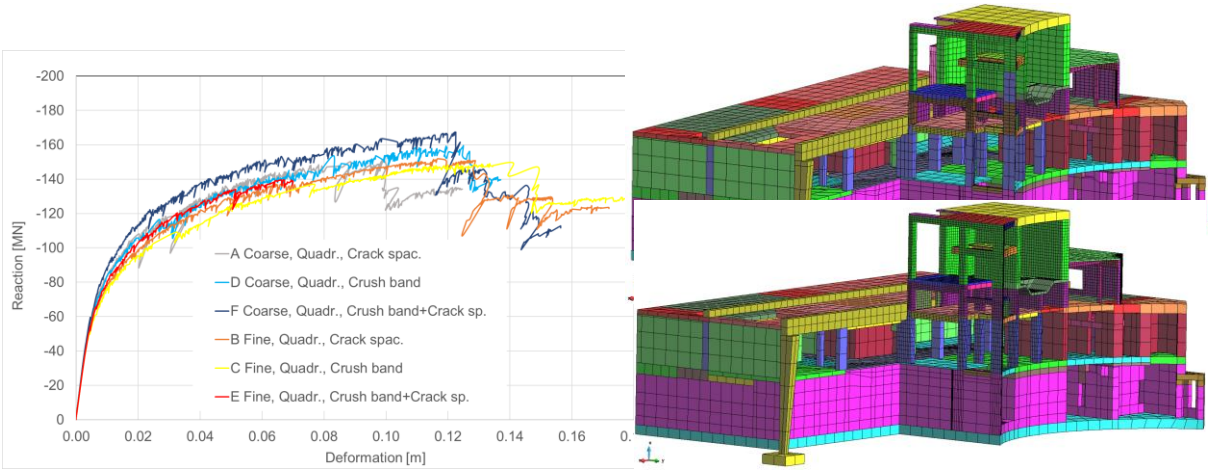


Figure 9. Sensitivity studies using pushover analyses for building 1UN-C, coarse (top), fine (bott.) mesh.

Critical reinforcement in columns and beams is modelled explicitly using discrete reinforcement bars represented by embedded truss elements, as illustrated in Figure 4, which shows the detailed reinforcement layout in one of the analyzed buildings. Reinforcement in slabs and walls is modelled using a smeared reinforcement approach, as shown in Figure 5, highlighting the regions where reinforcement is included through composite reinforced concrete material models.

The soil foundation interaction was simulated using the idealized spring approach as discussed in Section 2. The spring stiffnesses in normal directions, bending as well as torsion were calibrated based on the global linear elastic model in software SASSI (ACS-SASSI 2024).

Model uncertainty was addressed by sensitivity studies of critical model parameters. This is demonstrated for the building 1UN-C in Figure 9, which shows the influence of mesh size (0.9 m and 0.5 m) in the most critical part of the structure, and the effect of crack spacing and minimal crush band on the structural response as introduced in Section 2. The sensitivity study shows that the structural response is not strongly sensitive to the mesh size, but the effect of crack spacing can be quite significant, and increases the stiffness of the structure. It was decided to assume more conservative approach and not to use the crack spacing together with coarser meshes for faster analysis time.

Due to space constraints in this paper, only the results for the dynamic analysis of the building 2UN-D are reported here. The finite element mesh consists of quadratic brick elements as shown in Figure 10 for the building 2UN-D.

The damping of the structures was introduced using Rayleigh damping. The Rayleigh coefficients were calculated based on a specified damping ratio and the eigenfrequencies of the buildings, providing a consistent representation of energy dissipation in the dynamic analysis based on the linear analysis by SASSI software.

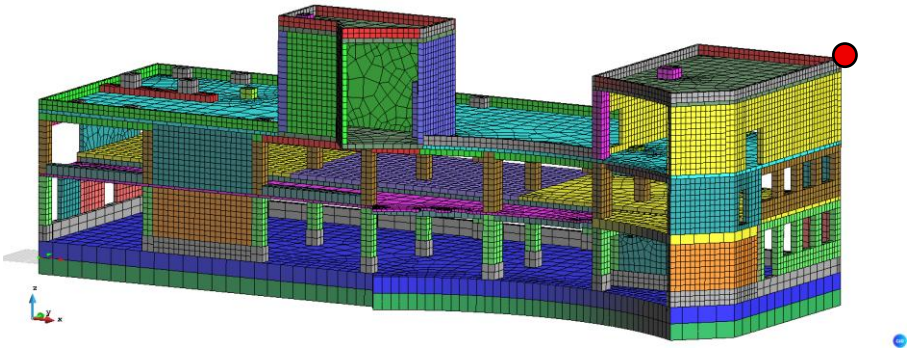


Figure 10. The finite element model for seismic analysis, building 2UN-D showing the point where displacements are reported in Figure 13.

Seismic loading was defined by sets of three-component acceleration time histories generated to match the target response spectrum shown in Figure 11 (Stäuble Akcay et al., 2019). Five different realizations (Seed 07, 08, 10, 13 and Seed 28) were specified in the seismic input definition. In the present study, the nonlinear dynamic analysis is presented for Seed 28, which represents one realization of the prescribed seismic action. Figure 12 illustrates the applied seismic input in terms of acceleration time histories for the horizontal and vertical components. The records show the temporal evolution of ground acceleration and serve as the input excitation for the dynamic response analysis presented in the subsequent parts of the paper.

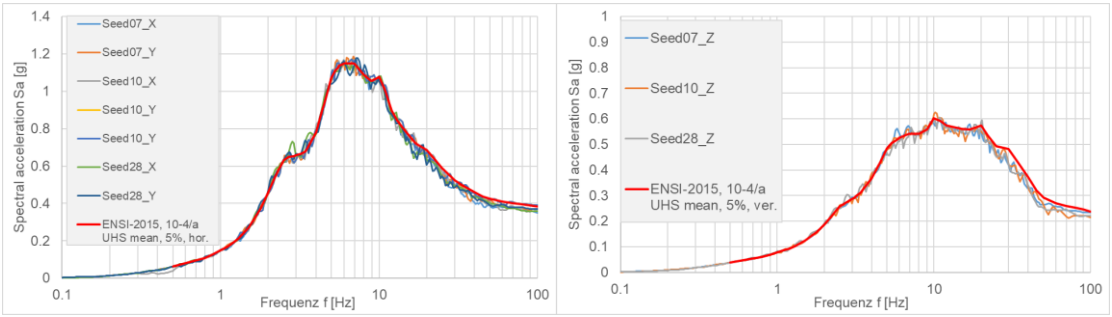


Figure 11: Response spectrum used for the generation of artificial accelerograms

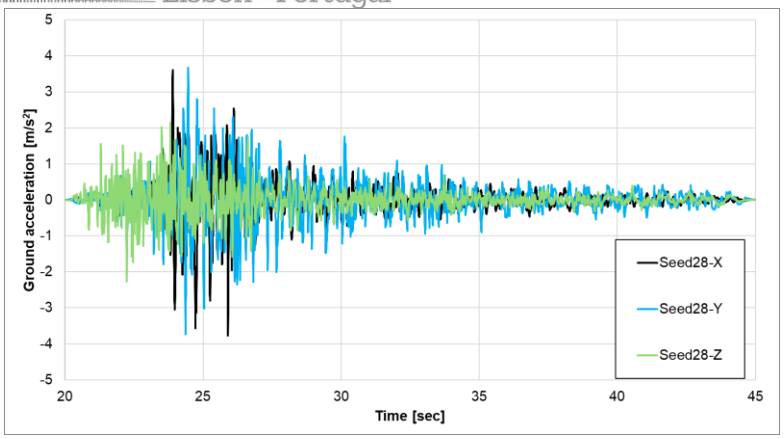


Figure 12. Artificial accelerogram of the seismic input (seed 28) in the two horizontal directions and the vertical direction used for the presented nonlinear dynamic analysis

The dynamic nonlinear analysis was performed by software ATENA (Cervenka et al. 2025) using an implicit approach. The whole analysis for the building 2UN-D for Seed 28 consisted of 5000 time steps and required approximately 30 hours on a workstation running Windows 11, equipped with 13th generation Intel Core i9 processor and 64 GB RAM . The selected results are shown in the subsequent figures. Figure 13 shows the evolution of displacements at the monitoring point located at the top of the third floor as shown in Figure 10 during the most critical first 8 seconds of the dynamic analysis. Figure 14 shows the strains in the reinforcement and concrete, which are the most important quantities that are being checked by the owner, and must satisfy the required limits. Figure 15 shows two selected plots of damage at the time with the highest observed top floor displacements. Figure 15a shows the deformed shape with crack pattern and calculated crack widths at 26s, and Figure 15b shows the location, where concrete yielding is observed. This corresponds to the state when the material reaches its nonlinear response in compression as shown in Figure 3b. During the dynamic analysis of the building the material in compression did not reach the compressive strength or the post-peak softening as shown in Figure 3c. This indicates that the structure remains within acceptable safety margins for the considered seismic input

The models were prepared and validated by the authors and will be used subsequently by the power plant owner to perform large number of analyses to determine the fragility curves of these safety critical buildings in the nuclear power plant.

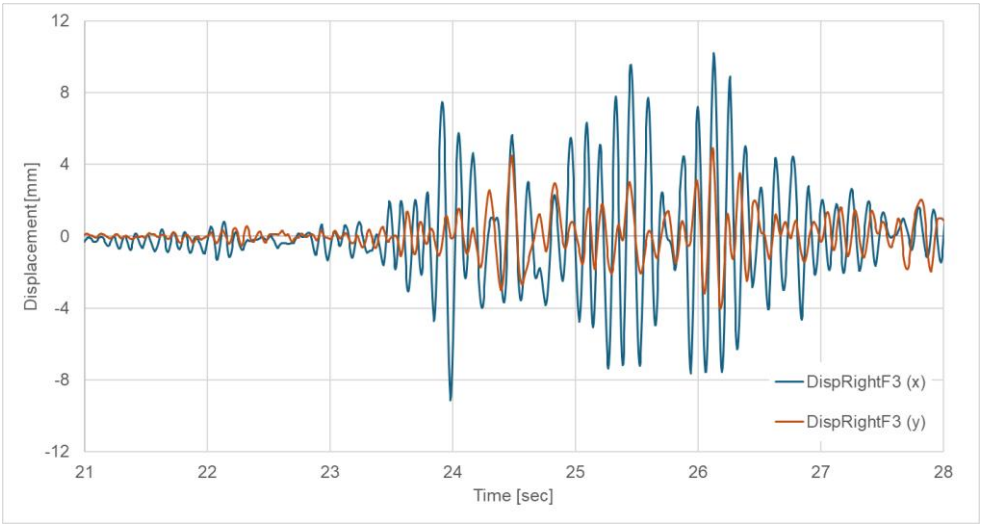


Figure 13: Displacement evolution in the building 2UN-D dynamic analysis

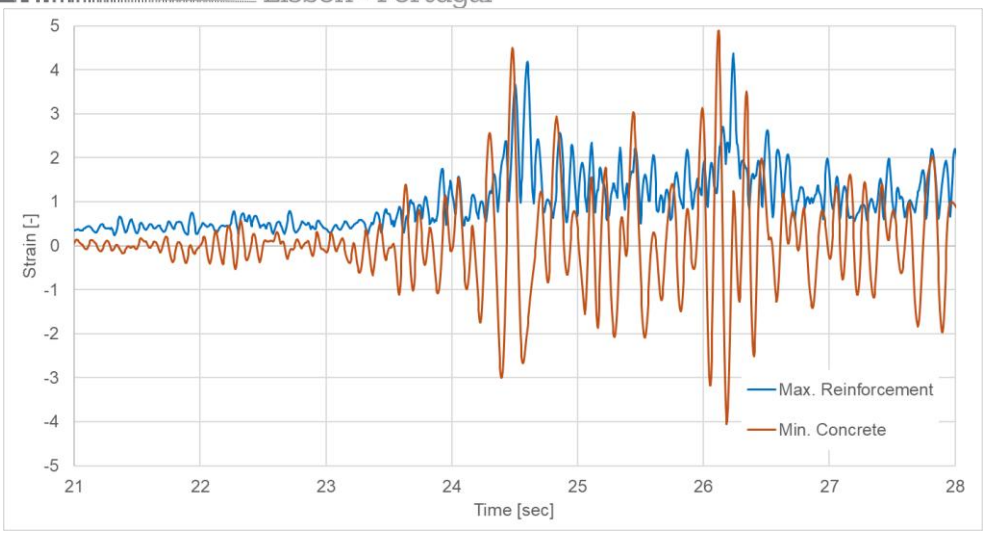


Figure 14: Evolution of concrete and reinforcement strains in the building 2UN-D dynamic analysis.

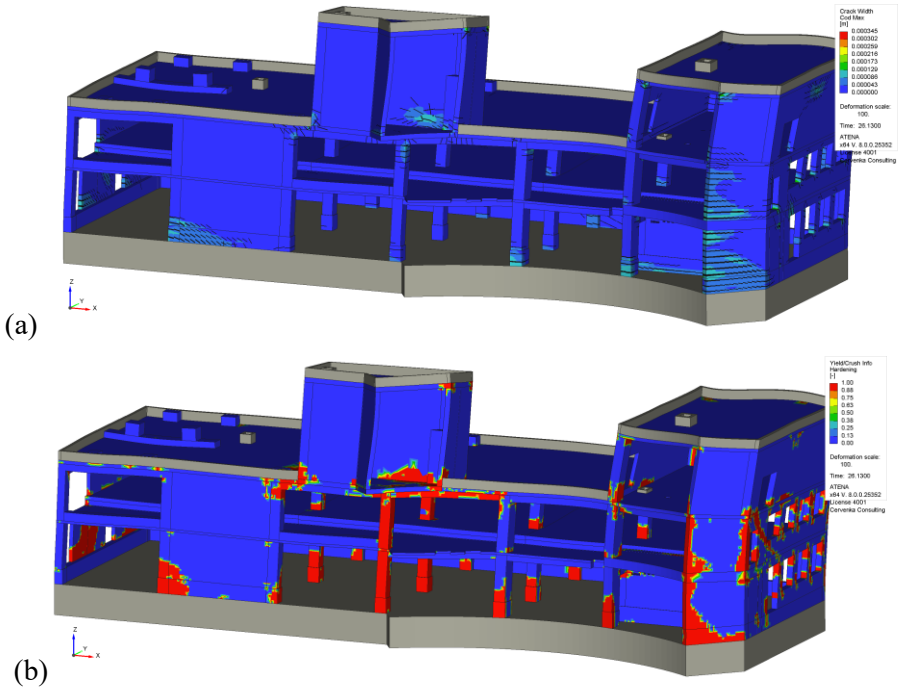


Figure 15: Crack width and pattern (left) and concrete yielding (right) at time 26.13s, building 2UN-D.

5. Conclusions

This paper presented a high-fidelity nonlinear finite element modeling framework for the seismic assessment of safety-critical reinforced concrete structures in nuclear power plants. The approach combines advanced constitutive models for concrete and reinforcement, mixed solid–shell finite element discretisation, nonlinear dynamic time-history analysis, and a consistent treatment of damping and soil–structure interaction.

The modeling strategy and uncertainties were first validated against the CAMUS III shake-table benchmark. The numerical simulations showed good agreement with experimental observations in terms of global dynamic response, crack development, and localization of damage, demonstrating the capability of the adopted approach to realistically capture key nonlinear seismic response mechanisms.

The validated methodology was subsequently applied to the seismic assessment of selected reinforced concrete buildings forming part of a nuclear power plant's critical infrastructure. The results highlighted the importance of directly accounting for material nonlinearity, reinforcement detailing, and modeling assumptions when evaluating seismic response and damage development. For the analyzed seismic input, the structures remained within acceptable response limits, with no indication of concrete crushing or of loss of load-carrying capacity.

The presented results confirm that high-fidelity nonlinear finite element analysis represents a robust and practical tool for the seismic safety reassessment of existing nuclear structures. When supported by benchmark validation and a balanced modeling strategy, such analyses can provide valuable insight into structural behavior and safety margins, and can serve as a reliable basis for subsequent fragility assessment and risk-informed decision-making.

6. Acknowledgements

The dynamic high-fidelity modeling approaches were developed during the project TQ05000005, STF4SW – "Sustainable Act for Construction Market: A new perspective on passive seismic protection" co-funded by the Technology agency of the Czech Republic within the SIGMA: M-ERA.NET program.

7. References

ACS-SASSI (2024), Soil Structure Interaction Software, www.acssassi.com

Argyris JH. (1954), Energy Theorems and Structural Analysis. Aircraft Engineering and Aerospace Technology. Emerald, 1954.

Červenka J. and Papanikolaou V. (2008). "Three Dimensional Combined Fracture-Plastic Material Model for Concrete," *Int Journal of Plasticity*. 24:2192-220.

Červenka, V., Červenka, J. and Jendele, L. (2025). "ATENA Program Documentation, Part 1: Theory," Cervenka Consulting s.r.o. www.cervenka.cz.

Červenka, J.; Červenka, V.; Laserna, S. On crack band model in finite element analysis of concrete fracture in engineering practice. Eng. Fract. Mech. 2018, 197, 27–47. https://doi.org/10.1016/j.engfracmech.2018.04.010.

Jendele, L., Červenka, J., Curvilinear Non-linear Three-Dimensional Isoparametric Layered Shell Elements, Paper 212, Civil-Comp Press, 2014, Proceedings of the Twelfth International Conference on Computational Structures Technology, B.H.V. Topping and P. Iványi, (Editors), Civil-Comp Press, Stirlingshire, Scotland,

Lee, J., Fenves, G.L. (1998). Plastic-damage model for cyclic loading of concrete structures. Journal of Engineering Mechanics, ASCE.

Menetrey, P.; Willam, K.J. (1995). "Triaxial failure criterion for concrete and its generalization," ACI Struct. J., 92 311–318. https://doi.org/10.14359/1132.

Proske, D., Kurmann, D., Cervenka, J. (2013), Seismische Tragfähigkeit eines Stahlbetongebaudes, Beton- und Stahlbetonbau, 108, Heft 8, DOI: 10.1002/best.201200080

Sigmund, V., Guljaš, I., & Matošević, Đ. (2001). "CAMUS 3" International Benchmark : After test comparison of numerical and experimental results.

Richard, B, Martinelli, P., Voldoire. F, Chaudat, T, Abouri, S, Bonfils, N, (2016), SMART 2008: Overview, synthesis and lessons learned from the International Benchmark, Engineering Structures, Volume 106, 2016, Pages 166-178, ISSN 0141-0296, https://doi.org/10.1016/j.engstruct.2015.10.029.

Stäuble Akcay, S., Nykyforchyn, A., Klügel, J.-U., Khalili Tehrani, P., Kosbab, B., & Talebinejad, I. (2019). Determining seismic safety margins by nonlinear soil-structure analysis. International Journal of Safety and Security Engineering, Vol. 9, No. 3, pp. 225–236.

Wolf, J.P. (1985). Dynamic Soil–Structure Interaction. Prentice Hall.

Zienkiewicz O.C., Cheung Y.K. 1967. The Finite Element Method in Structural and Continuum Mechanics. McGraw-Hill, London, 1967. p 274.