

Numerical Validation of the Stress-Strain Curve of JIS SM490 Steel in the Strain Hardening Range Using Finite Elements

Resky Ahmad ^{1✉}, Data Iranata ², Aniendhita Rizki Amalia ³
(1,2,3) Civil Engineering, Institut Teknologi Sepuluh Nopember, Surabaya, Indonesia

Abstract: SM490 steel is a structural steel that complies with JIS G3106 and is widely used in bridges and civil engineering structures due to its combination of strength, ductility, and good weldability. Before being applied to more complex analyses, the finite element model (FEM) for this material must be compared with experimental data to determine its reliability. This study developed an FEM model using Abaqus to simulate the true stress-true strain curve of SM490 steel under monotonic tensile loading and quantitatively evaluated the results against secondary experimental data. Validation was performed at nine points within the strain range of 2.151–7.450% using Mean Absolute Error (MAE), Root Mean Squared Error (RMSE), Mean Absolute Percentage Error (MAPE), Normalized RMSE (NRMSE), and the coefficient of determination (R^2). The evaluation results yielded an MAE of 12.96 MPa, an RMSE of 17.17 MPa, a MAPE of 3.06%, an NRMSE of 3.14%, and an R^2 of 0.910, indicating good agreement between the numerical and experimental results. The largest error of 8.66% occurred at a strain of 2.279%, specifically in the transition region from the yield plateau to strain hardening. Conversely, the best agreement, with a MAPE of 0.90%, was obtained in the strain range of 5.0–7.5%. These results indicate that the FEM model used adequately represents the strain-hardening behavior of SM490, although evaluations in the elastic and yield plateau regions still require numerical results with higher resolution.

Article history:

Received: 08 September 2026
Revised: 22 September 2026
Accepted: 26 September 2026
Published: 29 September 2026

Keyword:

abaqus, finite elements,
numerical validation, SM490,
stress-strain curve

This is an open-access article under the [CC-BY-SA License](https://creativecommons.org/licenses/by-sa/4.0/).



How to cite: Ahmad, R., Iranata, D., & Amalia, A. R. (2026). Numerical Validation of the Stress-Strain Curve of JIS SM490 Steel in the Strain Hardening Range Using Finite Elements. *RESWARA: Jurnal Riset Ilmu Teknik*, 4(4), 1334–1345. <https://doi.org/10.70716/reswara.v4i4.872>

INTRODUCTION

Structural steel is one of the primary materials used in the construction of bridges, buildings, and various engineering components because it offers high strength, good ductility, and is relatively easy to fabricate. One of the most widely used construction steels is SM490, a carbon-manganese steel that complies with the Japanese Industrial Standard (JIS G3106) for welded structures. This material is commonly chosen for bridge and civil engineering projects because it is relatively economical, easy to work with, and has good weldability (Fincato et al., 2023).

SM490 also demonstrates a relatively high level of ductility, with an elongation at fracture of more than 33%. This characteristic is particularly beneficial in seismic regions because it allows the steel to undergo substantial plastic deformation before failure, thereby improving structural safety under severe loading conditions (Fincato et al 2023). The combination of sufficient yield strength and strong deformation capacity makes SM490 well suited for structural members that are required to sustain significant loads while maintaining adequate ductile behavior during extreme events.

To make effective use of these mechanical properties in structural design, a reliable description of the full stress-strain response of SM490 is required, covering the material behavior from yielding, through strain hardening, to final fracture. This information is fundamental not only for conventional design calculations, but also for numerical modelling of SM490 structural members subjected to

complex or extreme loading conditions. Therefore, the accuracy of the stress-strain data becomes an important factor because it directly influences the reliability of both analytical assessments and numerical predictions.

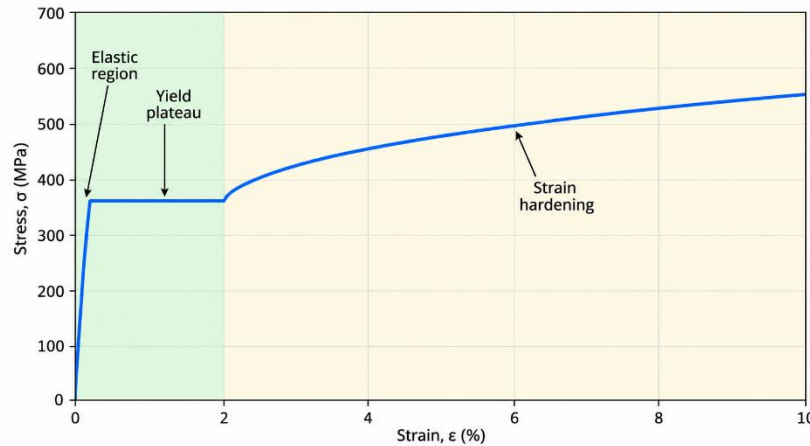


Figure 1. General schematic of the stress-strain curve for carbon steel, showing the elastic region, yield plateau, and strain hardening

The mechanical properties of materials, particularly the stress-strain relationship as shown in Figure 1, are fundamental to structural analysis and design. The availability of comprehensive and validated stress-strain data is also increasingly important in the development of data-driven engineering, as evidenced by the creation of large-scale databases compiled from scientific literature (Kumar et al., 2024). In analytical practice, the finite element method (FEM) is widely used to estimate the mechanical response of materials, thereby reducing the need for costly and time-consuming experimental testing (Schreiber et al., 2015). Abaqus is one of the most commonly used software packages for nonlinear simulation of metallic materials, including SM/SMA490 welded joints (Du et al., 2022; Xu et al., 2023a; Xu et al., 2023b), dual-phase steel roll-formed components (Song et al., 2024), and residual stress analysis resulting from heat treatment (Wang et al., 2024).

However, the accuracy of FEM simulations is greatly influenced by the precision of the material representation incorporated into the model. Several studies have shown that differences in the form of input data such as true stress-strain and engineering stress-strain as well as the methods used to approximate the plastic curve can lead to significant differences in the prediction of a component's plastic stiffness (Rahchamandi et al., 2026). Therefore, the results of numerical simulations must first be verified against experimental data before the model is used for parametric studies or structural analysis under more complex loading conditions (Sedmak et al., 2024; Milojević et al., 2025). The validation process is generally performed through a quantitative comparison between numerical and experimental results using statistical parameters, including MAE, RMSE, MAPE, NRMSE, and R^2 . Similar approaches have been used in the validation of deep learning-based stress-strain curves (Yang et al., 2025), data-driven models from indentation testing (Rom & Priel, 2025), as well as machine learning models to predict roll deviation in the 316L stainless steel rolling process (Lotfinia & Tayebi Arasteh, 2021; Rahchamandi et al., 2026).

Although previous studies have examined SM/SMA490-series steels using finite element methods, much of the available literature has concentrated on fatigue behavior and the performance

of welded joints under cyclic loading or crack-growth conditions (Du et al., 2022 ; Xu et al., 2023a; Xu et al., 2023b). In contrast, relatively limited attention has been given to direct, point-by-point validation of the monotonic true stress–true strain response in the strain-hardening region. This region is particularly important because it represents a substantial portion of the post-yield load-carrying capacity and energy dissipation of ductile steel. SM490 was selected for the present study because its combination of moderate yield strength and high ductility, with fracture elongation exceeding 33%, makes its strain-hardening response highly relevant to structural performance under seismic and other extreme loading conditions. However, this behavior has rarely been evaluated using explicit quantitative error measures in the existing literature. Accordingly, this study focuses on two main objectives: first, to assess the capability of an Abaqus finite element model to reproduce the true stress–true strain behavior of SM490 steel within the strain-hardening range, and second, to quantify the deviation between numerical and experimental results at each corresponding strain point rather than relying solely on a single overall error indicator.

The contribution of this study lies in two main aspects. First, the agreement between the numerical and experimental responses is evaluated quantitatively at each strain point within the strain-hardening region, rather than being assessed only through a visual comparison of the curve profiles. Second, the study examines the locations and possible causes of discrepancies between the Abaqus predictions and the experimental data, allowing the strain intervals with satisfactory agreement and those that still require further calibration to be identified more clearly. This approach provides a more detailed assessment than conventional FEM validation procedures that rely mainly on a single overall error value without showing how the error varies throughout the loading response.

RESEARCH METHOD

This study adopts a quantitative computational approach to assess the agreement between finite element simulation results and previously published experimental data. The reference dataset was obtained from the true stress–true strain response of SM490 steel presented in Fig. 7a of Fincato et al. (2023) in Archives of Civil and Mechanical Engineering. This dataset was chosen as the reference because it provides a well-documented monotonic tensile response for JIS-compliant SM490 steel, including both engineering and true stress–strain relationships, as well as the specimen geometry and testing conditions required to reproduce a comparable finite element model. Fincato et al. (2023) reported a 0.2% proof stress of 361 MPa, an ultimate tensile strength of 534 MPa, and an elongation at fracture of 33.2% from monotonic tensile tests conducted using a 40 mm gauge length and a loading speed of 2 mm/min.

For the present analysis, the true stress–true strain curve presented in Fig. 7a was digitized up to approximately 10% strain, where the corresponding true stress reached about 577 MPa. This value is higher than the reported nominal tensile strength because true stress accounts for the reduction in the instantaneous cross-sectional area during deformation, particularly at larger strains. Therefore, the digitized true stress value is physically consistent with the nominal tensile strength reported in the original study rather than being contradictory to it. The extracted data were subsequently used to define the tabulated hardening relationship adopted in the finite element model.

The validation dataset consisted of nine strain points generated directly from the discrete Abaqus output within the strain-hardening range of (2.151–7.450%). These points were therefore determined by the simulation output rather than selected manually. The lower limit of 2.151% represents the first recorded increment after the response moved beyond the yield plateau, while 7.450% corresponds to the final increment available before the numerical analysis ended. Since the strain values reported by Abaqus did not exactly match those in the experimental dataset, the experimental stress corresponding to each numerical strain value was estimated by linear interpolation between the two nearest experimental data points.

The plastic response of SM490 was implemented in Abaqus using a tabulated relationship between true yield stress and true plastic strain obtained from the experimental data of Fincato et al. (2023). The input curve started at 360.82 MPa at zero plastic strain and increased to 568.73 MPa at a plastic strain of 0.09, with an initial yield plateau extending to approximately 0.0155 plastic strain. This procedure follows common calibration approaches in which hardening parameters are derived directly from tensile stress–strain data (Kubaschinski et al., 2022; Świłło & Cacko., 2023).

It should be noted, however, that the same published dataset was used both to define the material input and to evaluate the numerical response. Accordingly, the comparison is intended mainly as a consistency check to verify that the finite element model, including the specimen geometry, mesh, boundary conditions, and solver configuration, can reproduce the prescribed monotonic tensile behavior. It should therefore not be interpreted as an independent prediction against previously unseen experimental data or as evidence of predictive accuracy outside the range covered by the digitized material input.

The SM490 tensile specimen was modeled in Abaqus/Explicit as a three-dimensional solid using eight-node linear brick elements with reduced integration (C3D8R), with the specimen dimensions reproduced from the reference study. The final mesh contained 3.016 elements and 3.710 nodes, with an approximate element length of 2.0 mm along the axial direction in the reduced gauge section. Material behavior was represented using an elastic-plastic constitutive model with a density of 7.85×10^{-9} tonne/mm³, a Young's modulus of 206 GPa, a Poisson's ratio of 0.30, and the previously defined tabulated hardening relationship. To account for material degradation beyond the calibrated hardening response, a ductile damage model was also included. The adopted parameters consisted of a fracture strain of 0.28, stress triaxiality of 0.333, strain rate of 8.33×10^{-4} s⁻¹, and a displacement-based damage evolution value of 0.165984 mm. Geometric nonlinearity was activated through (NLGEOM = YES), and the simulation was carried out using the Dynamic, Explicit procedure with a total analysis time of 1.0 s.

The tensile load was introduced through a kinematically coupled reference point located at the upper end of the specimen. An axial displacement of $U_2 = 10$ mm was imposed using a linear amplitude, while the other degrees of freedom at the reference point were restrained. At the opposite end, the lower gripping region spanning ($y = -50$ to -35 mm) was fully fixed using an ENCASTRE boundary condition, corresponding to $U_1 = U_2 = U_3 = 0$ for the solid nodes within this region. This boundary-condition arrangement eliminated rigid-body motion and reproduced a displacement-controlled uniaxial tensile test. To reduce computational cost while maintaining quasi-static loading behavior, semi-automatic whole-model mass scaling was employed with a target stable time increment of 1.0×10^{-4} s. Linear and quadratic bulk-viscosity coefficients of 0.06 and 1.2, respectively, were also

used. The main modeling parameters and analysis settings are summarized in Table 1 to facilitate reproducibility of the numerical procedure.

Table 1. Finite element model settings used in the Abaqus simulation

Item	Model setting
Element formulation	C3D8R, 8-node linear brick, reduced integration
Mesh	3.016 elements; 3.710 nodes; axial size ≈ 2.0 mm in reduced section
Elastic properties	$E = 206$ GPa; $\nu = 0.30$; $\rho = 7.85 \times 10^{-9}$ tonne/mm ³
Plasticity	True yield stress–true plastic strain table: 360.82 MPa at $\epsilon_p = 0$ to 568.73 MPa at $\epsilon_p = 0.09$; yield plateau to $\epsilon_p \approx 0.0155$
Ductile damage	$\epsilon_f = 0.28$; $\eta = 0.333$; strain rate = 8.33×10^{-4} s ⁻¹ ; displacement-based damage evolution = 0.165984 mm
Solver	Dynamic, Explicit; NLGEOM = YES; total step time = 1.0 s
Loading	Displacement control via kinematically coupled reference point; $U_2 = 10$ mm; linear amplitude
Boundary condition	Lower gripping region ($y = -50$ to -35 mm): ENCASTRE ($U_1 = U_2 = U_3 = 0$); loaded end: remaining DOFs (other than U_2) constrained at the reference point
Mass scaling / damping	Whole-model fixed mass scaling, target $\Delta t = 1.0 \times 10^{-4}$ s; bulk viscosity = 0.06 (linear) and 1.2 (quadratic)

A formal mesh-convergence analysis was not conducted in this study, as the numerical model was evaluated using only one mesh configuration. An axial element size of approximately 2.0 mm was adopted in the reduced gauge section and maintained throughout the analysis. As a result, the influence of mesh refinement on the predicted stress response could not be quantified, and the reported results should be regarded as those obtained from the selected mesh rather than from a verified mesh-independent solution. Within the validation range of 2.151–7.450% strain, the model produced a MAPE of 3.06%, an RMSE of 17.17 MPa, an NRMSE of 3.14%, and R2 of 0.910 when compared with the experimental stress–strain response, as discussed in the Results and Discussion section. Although these values indicate a reasonable agreement in the global response, they are not sufficient to establish mesh independence. A systematic convergence study using at least three progressively refined mesh configurations, such as coarse, medium, and fine meshes, is therefore required in future work. Such an assessment should compare key response parameters, including peak stress and the strain-hardening behavior, before the model can be regarded as fully mesh independent.

To ensure that the experimental and simulation results can be compared on an equal basis, the strain values from Abaqus which were originally in decimal form were converted to percentages. The strain points in the experimental data do not always coincide exactly with the numerical output points. Therefore, the experimental stress values corresponding to the Abaqus strain data were determined using linear interpolation. Next, the degree of agreement between the two datasets was calculated using the following five evaluation metrics (Xing et al., 2024).

$$Error_i (\%) = |\sigma_{Abaqus,i} - \sigma_{Exp,i}| / \sigma_{Exp,i} \times 100\% \quad (1)$$

$$MAE = (1/N) \sum |\sigma_{Abaqus,i} - \sigma_{Exp,i}| \quad (2)$$

$$RMSE = \sqrt{[(1/N) \sum (\sigma_{Abaqus,i} - \sigma_{Exp,i})^2]} \quad (3)$$

$$NRMSE = RMSE / \sigma_{Exp,max} \times 100\% \quad (4)$$

MAPE is used to indicate the average relative error at each data point, while R^2 describes the degree of agreement in the stress variation patterns between experimental and simulation results. Meanwhile, NRMSE is obtained by normalizing the RMSE value against the maximum experimental stress in the compared range. The simultaneous use of several metrics provides a more comprehensive evaluation and is consistent with the stress-strain model validation approach in previous studies (Yang et al., 2025; Rom & Priel, 2025).

RESULTS AND DISCUSSION

Figure 2 shows the mesh configuration of the SM490 specimen at the initial increment before loading is applied. At this stage, the von Mises stress is still zero in all elements. The mesh density was increased in the gauge length region so that the development of stress and plastic strain during the drawing process could be better recorded.

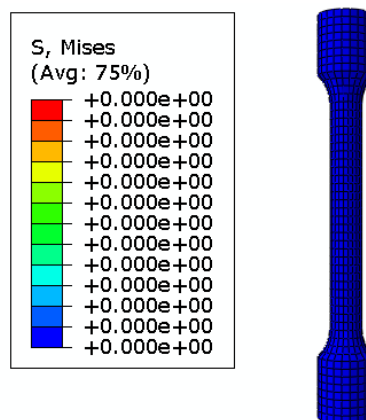


Figure 2. Finite element model of the SM490 specimen at the initial condition (increment 0) before loading is applied

As the tensile displacement was gradually increased, the von Mises stress distribution evolved in accordance with uniaxial loading conditions. The highest stresses occur primarily in the gauge length region. At the advanced loading stage shown in Figure 3, the maximum stress reaches 568.7 MPa. Meanwhile, Figure 4 shows stress concentration around the fillet radius with a maximum value of 481.5 MPa. This pattern is consistent with the general characteristics of stress distribution in dogbone-shaped tensile specimens.

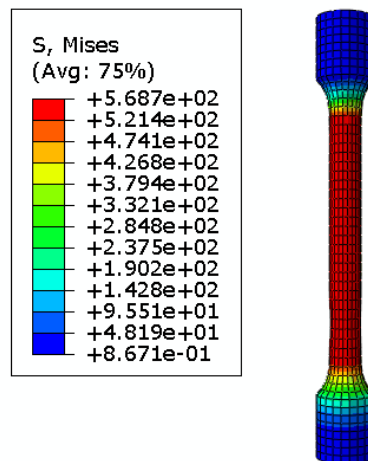


Figure 3. Von Mises stress contour on the SM490 specimen during the advanced loading stage ($\sigma_{\max} \approx 568.7$ MPa)

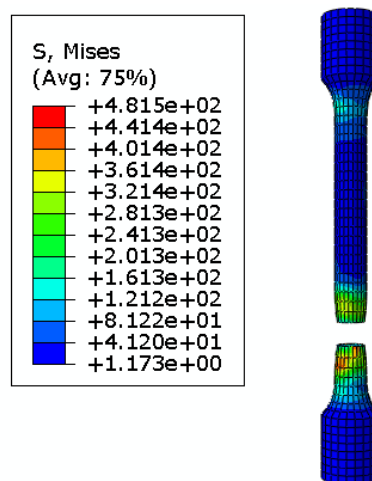


Figure 4. Von Mises stress contour showing stress concentration at the specimen's fillet radius ($\sigma_{\max} \approx 481.5$ MPa)

Table 2 shows a comparison at each point between the experimental stress obtained through linear interpolation and the stress from the Abaqus simulation at the same strain value.

Table 2. Comparison of experimental stress and Abaqus FEM results at the same strain

No.	Strain (%)	Experimental Stress (MPa)	Abaqus Stress (MPa)	Difference (MPa)	Error (%)
1	2.151	389.74	374.23	-15.51	3.98
2	2.279	396.78	362.44	-34.34	8.66
3	2.495	408.26	379.94	-28.32	6.94
4	3.264	441.14	425.84	-15.29	3.47
5	5.050	497.13	508.81	+11.68	2.35
6	5.271	503.18	504.15	+0.97	0.19

7	5.648	513.51	514.37	+0.86	0.17
8	6.544	529.38	536.79	+7.41	1.40
9	7.450	546.19	548.44	+2.25	0.41

The shapes of the two curves and the relative errors at each strain point are presented in Figure 5 and Figure 6.

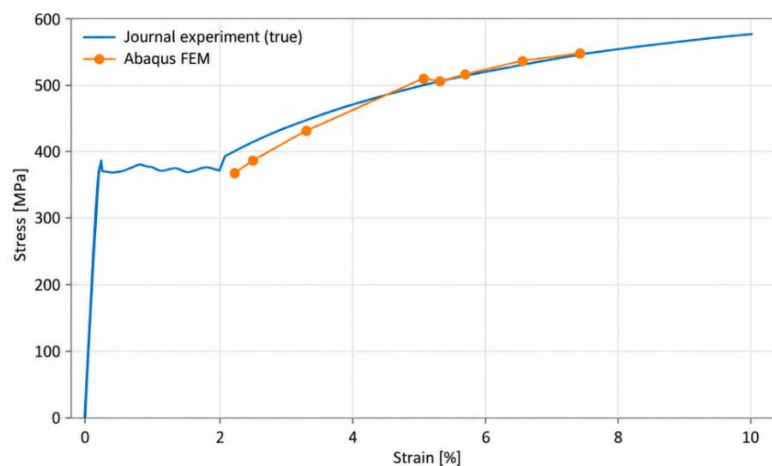


Figure 5. Comparison of the experimental true stress-true strain curve for SM490 and the numerical results from Abaqus FEM

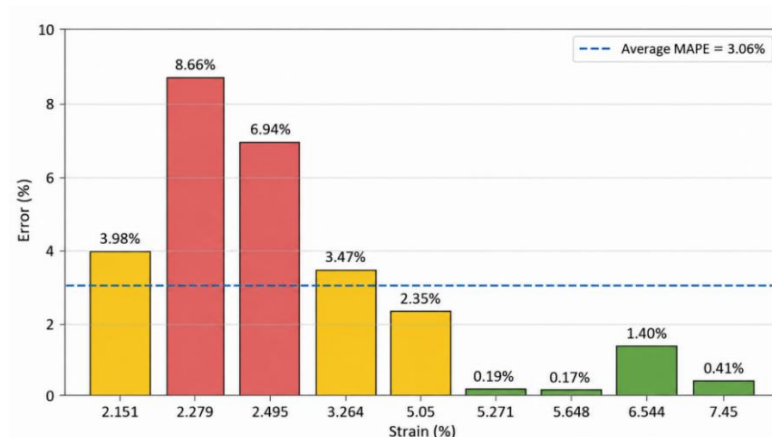


Figure 6. Graph of the relative error (%) of the Abaqus FEM results compared to the experimental data at each strain point

A summary of the statistical values used to evaluate the validation results is presented in Table 3.

Table 3. Summary of statistical data for the validation of the FEM model against experimental data

Parameter	Value
MAE	12.96 MPa
RMSE	17.17 MPa
MAPE	3.06%
NRMSE	3.14%
R^2	0.910

In general, the Abaqus FEM results curve was able to track the increase in stress in the plastic strain hardening region. The MAPE value of 3.06% and the NRMSE value of 3.14% indicate that the average deviation between the numerical and experimental results is relatively small. On the other hand, the R^2 value of 0.910 shows that the stress variation pattern from the simulation results agrees well with the experimental data. This level of agreement is consistent with a number of FEM validation studies on other steel grades, such as 22MnB5 steel (Eom et al., 2024) and QP 5Mn steel (Kassaye et al., 2025), which also demonstrate a high degree of agreement between numerical and experimental results in the plastic region.

In this study, the classification of the FEM results as showing “good agreement” with the experimental data was based on a validation criterion defined specifically for the present analysis rather than on a universally accepted standard. A MAPE below 5% together with an R^2 value above 0.90 was considered to indicate good agreement between the numerical and experimental stress-strain responses, whereas a MAPE greater than 10% was taken to suggest that further model refinement would be necessary. These limits were selected to provide a relatively strict basis for evaluating the overall curve while still allowing for localized deviations in regions where the response changes rapidly, particularly around the transition from yielding to strain hardening. They should therefore be viewed as study-specific criteria, since different limits may be more appropriate for other materials, loading conditions, or validation objectives.

Using these criteria, the present model satisfies the defined good-agreement range, with an overall MAPE of 3.06% and an R^2 of 0.910. The RMSE of 17.17 MPa and NRMSE of 3.14% provide additional information on the magnitude of the deviations and are more sensitive to larger local errors than the MAE. The fact that the RMSE is higher than the MAE of 12.96 MPa indicates that a limited number of points contribute disproportionately to the total error, particularly within the yield-to-hardening transition region. For this reason, the validation results are best interpreted by considering all five statistical measures together rather than relying on a single indicator.

Based on Figure 6, the highest discrepancy occurs at a strain of 2.279%, where the error reaches 8.66%. This strain level is located near the transition from the yield plateau to the strain-hardening region. A similar sensitivity has been reported by Rahchamandi et al. (2026), who showed that the way the plastic response is represented at the onset of yielding can substantially influence the accuracy of finite element predictions. Several factors may explain this localized deviation. First, the transition from the yield plateau to the strain-hardening region corresponds to the part of the true stress-true strain curve where the slope changes most rapidly. Since Abaqus interpolates tabulated plasticity data linearly between consecutive input points, predictions in this high-curvature region are more sensitive

to the spacing of the digitized data. As a result, stress values located between relatively distant input points may be locally underestimated or overestimated, whereas the smoother portion of the hardening branch is generally represented more accurately.

Second, the experimental response may contain small local fluctuations immediately after the yield plateau. Such behavior can occur in mild and structural steels during discontinuous yielding, including phenomena associated with Lüders-band propagation or serrated plastic flow. These local variations are difficult to reproduce using a smooth, monotonically increasing tabulated hardening curve. In addition, minor differences in gauge geometry between the experimental specimen and the numerical model may shift the strain level at which the deformation response begins to change, producing a local mismatch within this narrow transition range.

The agreement becomes considerably better at higher strains. Within the 5.0–7.5% strain interval, the calculated MAPE decreases to only 0.90%. This improvement is likely related to the smoother shape of the hardening curve in this region and the more adequate representation provided by the available digitized material data. The results therefore suggest that the adopted hardening parameters capture the SM490 response reasonably well at moderate plastic strains, while the yield-to-hardening transition remains the main source of residual error. Moreover, the stress contours presented in Figures 3 and 4 remain consistent with the expected uniaxial tensile response and show no apparent signs of numerical instability. This observation suggests that the remaining discrepancy is more closely related to the representation of the material input curve than to severe mesh distortion or solver convergence problems.

These results underscore the importance of evaluating models using quantitative statistical measures, as applied in various validation studies of stress-strain curves based on machine learning and data-driven approaches (Yang et al., 2025; Rom & Priel, 2025; Lotfinia & Tayebi Arasteh, 2021), as well as in microstructure-level calibration of constitutive models against tensile curves (Sim et al., 2025). However, the Abaqus output available in this study does not yet have a sufficient number of points in the elastic region and yield plateau, particularly at strains around 0–2%. Therefore, the results obtained are not yet sufficient to assess the elastic modulus or the shape of the yield plateau in detail. Validation covering the entire curve requires a history output with a higher frame density in that range, as well as an extension of the numerical output to approach the maximum experimental strain of approximately 10%.

CONCLUSION

Within the strain-hardening range evaluated in this study, from 2.151% to 7.450% strain, the Abaqus model reproduced the true stress–true strain response of SM490 steel with a satisfactory level of agreement. This is reflected by an MAE of 12.96 MPa, an RMSE of 17.17 MPa, a MAPE of 3.06%, NRMSE of 3.14%, and R^2 value of 0.910. These results fall within the agreement criteria discussed previously, namely a MAPE below 5% and an R^2 greater than 0.90. The largest deviation, 8.66%, occurred at a strain of 2.279%, which corresponds to the transition from the yield plateau to the strain-hardening region. This discrepancy is most likely associated with the relatively sparse digitized input data in this part of the curve, where the stress–strain response changes rapidly, rather than with numerical instability in the finite element model. In contrast, the best agreement was observed between 5.0% and 7.5% strain, where the MAPE decreased to 0.90%.

These findings should, however, be interpreted only within the strain range considered in the present analysis. The elastic region and yield plateau below approximately 2% strain were not assessed because the available numerical output in this range was too limited for a detailed comparison. Likewise, the model has not been validated beyond 7.450% strain or under cyclic, dynamic, or multiaxial loading conditions. In addition, since the hardening curve used as the Abaqus material input was derived from the same experimental dataset used for comparison, the present results mainly demonstrate the consistency and correct implementation of the numerical model rather than its independent predictive capability against unseen data. Further work should therefore focus on increasing the output resolution in the elastic and yield regions, extending the analysis to the full experimental strain range of approximately 10%, and validating the model against an independent experimental dataset. The development of a corresponding cyclic-loading model would also be valuable for assessing the applicability of the SM490 finite element model to a broader range of structural engineering problems.

REFERENCES

- Du, P., Liu, H., & Xu, X. (2022). Cyclic performance of structural steels after exposure to various heating-cooling treatments. *Metals*, 12(7), 1146. <https://doi.org/10.3390/met12071146>
- Eom, J. H., Jin, C. K., Ahn, D. Y., Babu, J. S. S., Jang, J. Y., & Lee, M. S. (2024). Comparison of fe simulation and experiment on tensile test of twb-hpf 22mnb5 steel. *Metals*, 14(10), 1176. <https://doi.org/10.3390/met14101176>
- Fincato, R., Yonezawa, T., & Tsutsumi, S. (2023). Numerical modeling of cyclic softening/hardening behavior of carbon steels from low- to high-cycle fatigue regime. *Archives of Civil and Mechanical Engineering*, 23, 164. <https://doi.org/10.1007/s43452-023-00698-4>
- Kassaye, F. T., Kori, T. H., & Kozłowska, A. (2025). Experimental Investigation and FEM Simulation of the Tensile Behavior of Hot-Rolled Quenching and Partitioning 5Mn Steel. *Materials*, 18(4), 868. <https://doi.org/10.3390/ma18040868>
- Kubaschinski, P., Gottwalt, A., Tetzlaff, U., Altenbach, H., & Waltz, M. (2022). Calibration of a combined isotropic-kinematic hardening material model for the simulation of thin electrical steel sheets subjected to cyclic loading. *Materialwissenschaft und Werkstofftechnik*, 53(4), 422–439. <https://doi.org/10.1002/mawe.202100341>
- Kumar, P., Kabra, S., & Cole, J. M. (2024). A database of stress-strain properties auto-generated from the scientific literature using ChemDataExtractor. *Scientific Data*, 11, 1275. <https://doi.org/10.1038/s41597-024-03979-6>
- Lotfinia, M., & Tayebi Arasteh, S. (2021). Machine learning-based generalized model for finite element analysis of roll deflection during the austenitic stainless steel 316L strip rolling. *arXiv*. <https://arxiv.org/abs/2102.02470>
- Milojević, I., Tarić, M., Klimenta, D., Grujić, B., Andriukaitis, D., Jovanović, S., & Čolović, M. (2025). Finite element analysis-based assessment of damage parameters for ultra-low-cycle fatigue behavior of structural steels. *Symmetry*, 17(4), 615. <https://doi.org/10.3390/sym17040615>
- Rahchamandi, I., Tavassoli, M. R., & Esmaeilabadi, R. (2026). Multi-Linear and Bi-Linear Stress – Strain Approximations for Finite Element Modelling of Extended End-Plate Moment Connections. *Buildings*, 16(1), 236. <https://doi.org/10.3390/buildings16010236>
- Rom, N., & Priel, E. (2025). A data-driven methodology for obtaining the stress-strain curves of metallic materials using discrete indentation tests. *Metals*, 15(1), 40. <https://doi.org/10.3390/met15010040>
- Schreiber, J. M., Smid, I., Eden, T. J., & Jann, D. (2015). Prediction of fragmentation and experimentally

- inaccessible material properties of steel using finite element analysis. *Finite Elements in Analysis and Design*, 104, 72–79. <https://doi.org/10.1016/j.finel.2015.06.001>
- Sedmak, A., Grbović, A., Gubeljak, N., Sedmak, S., & Budimir, N. (2024). Numerical simulation of fatigue crack growth and fracture in welded joints using XFEM—A review of case studies. *Materials*, 17(22), 5531. <https://doi.org/10.3390/ma17225531>
- Sim, G.-J., You, J., Choi, S., Kim, Y., Lee, C. A., Kim, H., Noh, D., & Lee, M.-G. (2025). A GND-based back stress model for reverse loading in metal sheets with consideration of GNB. *arXiv*. <https://arxiv.org/abs/2509.20996>
- Song, J., Lan, J., Zhu, L., Jiang, Z., Zhang, Z., Han, J., & Ma, C. (2024). Finite element simulation and microstructural analysis of roll forming for DP590 high-strength dual-phase steel wheel rims. *Materials*, 17(15), 3795. <https://doi.org/10.3390/ma17153795>
- Świłto, S., & Cacko, R. (2023). A new approach for evaluation true stress-strain curve from tensile specimens for DC04 steel with vision measurement in the post-necking phases. *Materials*, 16(2), 558. <https://doi.org/10.3390/ma16020558>
- Wang, A., Ling, C., Zhao, X., Wang, H., Wang, T., Tao, G., Fu, Y., & Cheng, T. (2024). Research on the analysis of residual stress in heat treatment of bellows using ABAQUS. *Materials*, 17(13), 3263. <https://doi.org/10.3390/ma17133263>
- Xing, M., Liu, Y., Lu, J., Huang, X., Wang, X., Xue, J., Zhang, F., & Du, F. (2024). Fracture criterion calibration and finite element simulation in SUS304 stainless steel. *Materials*, 17(23), 5711. <https://doi.org/10.3390/ma17235711>
- Xu, X., Xie, L., Zhou, S., An, J., Huang, Y., & Li, D. (2023). Fatigue crack propagation behavior and life prediction of welded joints of SMA490BW steel for bogies. *Processes*, 11(7), 1984. <https://doi.org/10.3390/pr11071984>
- Xu, X., Xie, L., Zhou, S., An, J., Huang, Y., Liu, Y., & Jin, L. (2023). Effect of welding defects on fatigue properties of SWA490BW steel cruciform welded joints. *Materials*, 16(13), 4751. <https://doi.org/10.3390/ma16134751>
- Yang, Z., Yang, R., Han, W., & Liu, Q. (2025). Enhancing stress-strain predictions with Seq2Seq and cross-attention based on small punch test. *arXiv*. <https://arxiv.org/abs/2506.17680>