DOI: 10.2478/aucts-2023-0004

ACTA UNIVERSITATIS CIBINIENSIS – TECHNICAL SERIES

Vol. 75 2023

CALIBRATING DC01 MATERIAL PROPERTIES FOR FINITE ELEMENT ANALYSIS WITH ABAQUS AND ISIGHT

POPP Ilie-Octavian
Faculty of Engineering, Department of Industrial Machines and Equipment, "Lucian Blaga" University, Sibiu, Romania, ilie.popp@ulbsibiu.ro

POPP Mihai-Octavian
Faculty of Engineering, Department of Industrial Machines and Equipment, "Lucian Blaga" University, Sibiu, Romania, mihai.popp@ulbsibiu.ro

OLEKSIK Mihaela
Faculty of Engineering, Department of Industrial Engineering and Management, "Lucian Blaga" University, Sibiu, Romania, mihaela.oleksik@ulbsibiu.ro

RUSU Gabriela-Petruța
Faculty of Engineering, Department of Industrial Machines and Equipment, "Lucian Blaga" University, Sibiu, Romania, gabriela.rusu@ulbsibiu.ro

CHICEA Anca Lucia
Faculty of Engineering, Department of Industrial Engineering and Management, "Lucian Blaga" University, Sibiu, Romania, anca.chicea@ulbsibiu.ro

Abstract: *This article explores the process of calibrating the properties of DC01 materials for finite element analysis using ABAQUS and Isight. Precise calibration of yield strength and tensile strength is crucial for accurately modeling material behavior under various loading conditions. The article details the sample preparation process, deformation measurements, and tensile tests to obtain experimental data. These data are then used to calibrate the DC01 material model in ABAQUS with the assistance of Isight, ensuring that the simulations accurately reflect real-world behavior. The results indicate the success of the calibration, facilitating the design and testing of products in a predictive virtual environment. Precise calibration is essential for predicting material behavior in real-world conditions and minimizing the risk of implementation failures.*

Key words: Finite Element Analysis, Calibration, Material Behavior, Model Adjustment.

1. Introduction

This article is dedicated to exploring the calibration procedure of DC01 material properties in the context of finite element analysis, with a particular emphasis on using ABAQUS software and the optimization tool Isight. Precise calibration of mechanical properties such as yield strength and tensile strength is vital for accurately modeling the material's behavior under different loading conditions [1]. This involves not only a deep understanding of the fundamental properties of the material but also the ability to adjust these properties within a virtual model to reflect the real material behavior. DC01, a high-quality cold-rolled steel, is widely used in engineering due to its remarkable mechanical properties, including excellent tensile strength and superior malleability [2]. This material plays an essential role in structural components, automotive bodies, and appliances, thanks to its wear resistance and adaptability in various manufacturing processes. Precise calibration of these properties is crucial in product design and testing, enabling realistic simulations in virtual environments.

Finite Element Analysis (FEA) is an essential computational method in engineering, providing the ability to simulate and analyze material behavior under various loading conditions [3, 4]. ABAQUS stands

out for its capability to handle complex simulation scenarios, including nonlinear modeling and dynamic analysis, essential for studying materials like DC01. Efficient use of ABAQUS in modeling DC01 critically depends on the accuracy with which material properties are represented in the software, requiring careful and rigorous calibration. Isight is an essential tool for optimizing and automating simulation processes, facilitating the calibration of FEA models. Through Isight, model parameters can be automatically adjusted to align with experimental data, ensuring that simulations in ABAQUS faithfully reflect the real behavior of DC01 material. Using Isight in combination with ABAQUS enhances the efficiency of simulations under various loading scenarios, contributing to the accuracy and reliability of designs [5].

This introduction aims to establish a detailed context for the calibration process, highlighting its complexity and importance within finite element analysis. We intend to provide a comprehensive perspective on how calibration influences material modeling and structural analysis, emphasizing the role of state-of-the-art technologies in optimizing this process. Continuing this introduction, we will discuss the stress-strain curve, a central element in understanding the mechanical behavior of DC01. The stress-strain curve illustrates how the material behaves under applied loads, providing vital information for calibration. It begins with an elastic region, followed by a yield point, a plastic zone, and ends at the fracture point. A precise understanding and modeling of this curve in Isight ensure that finite element simulations accurately reflect the material's behavior under different loading conditions.

The curve starts with an elastic region, where deformations are reversible, and the material returns to its original shape after loading [6]. This is followed by a yield point, where the material begins to deform plastically and permanently. The plastic zone of the curve continues up to the point of maximum strength, known as ultimate tensile strength. Finally, the curve ends at the fracture point, where the material fails under load. This curve is crucial for calibration in ABAQUS, as it allows engineers to adjust the material model to closely match real behavior. A precise understanding and modeling of this curve in Isight ensure that finite element simulations faithfully reflect the material's behavior under different loading conditions [7, 8].

2. METHODOLOGY

To calibrate the properties of DC01 material in ABAQUS, we began by determining the mechanical characteristics of the material through a series of tensile tests. These tests are essential to obtain real data that will be subsequently used in the calibration process in ABAQUS, using Isight.

A total of 6 specimens of DC01 steel were prepared for testing. The specimens were cut at 0° angles with respect to the rolling direction, following the SR EN ISO 6892-1:2020 standard. They had standardized dimensions: a total length of 160 mm, a gauge length of 75 mm, a width of 12.5 mm, and thicknesses of 0.8 mm. Cutting was performed using water jet technology from Famatech company, a method that ensures high precision and avoids structural transformations often associated with other methods, such as laser cutting.

Before testing, the specimens were prepared for strain measurement using the ARAMIS optical system. This process included degreasing the surfaces with technical alcohol, followed by applying a matte white RAL 9010 paint layer. After the paint dried, a network of black dots was applied by spraying with matte black RAL 9005 spray paint. This preparation allowed the ARAMIS system to accurately measure the strains in the specimens during testing. The specimens were then mounted on the Instron 5587 tensile testing machine, equipped with force and displacement transducers. The testing procedure was carried out using the Bluehill 2 software package, involving testing the specimens until they ruptured. These tests allowed obtaining conventional characteristic curves for each material and cutting angle, essential for further calibration.

The data obtained from the tensile tests were used to calibrate the DC01 material model in ABAQUS. This process requires the experimental data to be introduced into Isight, where material model parameters were adjusted to closely match real behavior. This iterative process ensured that simulations conducted in ABAQUS accurately reflected the mechanical properties of DC01, including the stress-strain curve and behavior under different loading conditions.

3. EXPERIMENTAL RESULTS

For DC01 steel, with a specimen thickness of 0.8 mm and cut at 0° relative to the rolling direction, the following experimental results were obtained:

1. Young's Modulus, representing the material's stiffness, varied between 78422.99 MPa and 84422.74 MPa, with an average of 81854.092 MPa. This is an indicator of material elasticity and is crucial for modeling the elastic behavior in ABAQUS.

2. The material's yield strength, the point at which it begins to deform plastically, was recorded between 152.26 MPa and 176.21 MPa, with an average of 168.498 MPa. This represents the values at which the material model in ABAQUS will start to exhibit plastic behavior.

3. The ultimate tensile strength, or the maximum point on the stress-strain curve, had values between 264.97 MPa and 273.23 MPa, with an average of 266.239 MPa. This information is crucial for defining the breaking point in the material model in ABAQUS.

4. The strain hardening coefficient, describing the material's strengthening as plastic deformation increases, showed relatively stable values, with an average of 0.252. This coefficient helps in modeling post-yield behavior in simulations.

5. The strength coefficient, which is another way of expressing tensile strength, showed an average of approximately 493.424 MPa. This can be used to validate or calibrate the material's tensile model in ABAQUS.

6. The elongation at break, measuring the material's ductility, was recorded with values between 0.316 mm/mm and 0.338 mm/mm, with an average of 0.326 mm/mm. This feature is vital for evaluating the material's ability to deform before rupture.

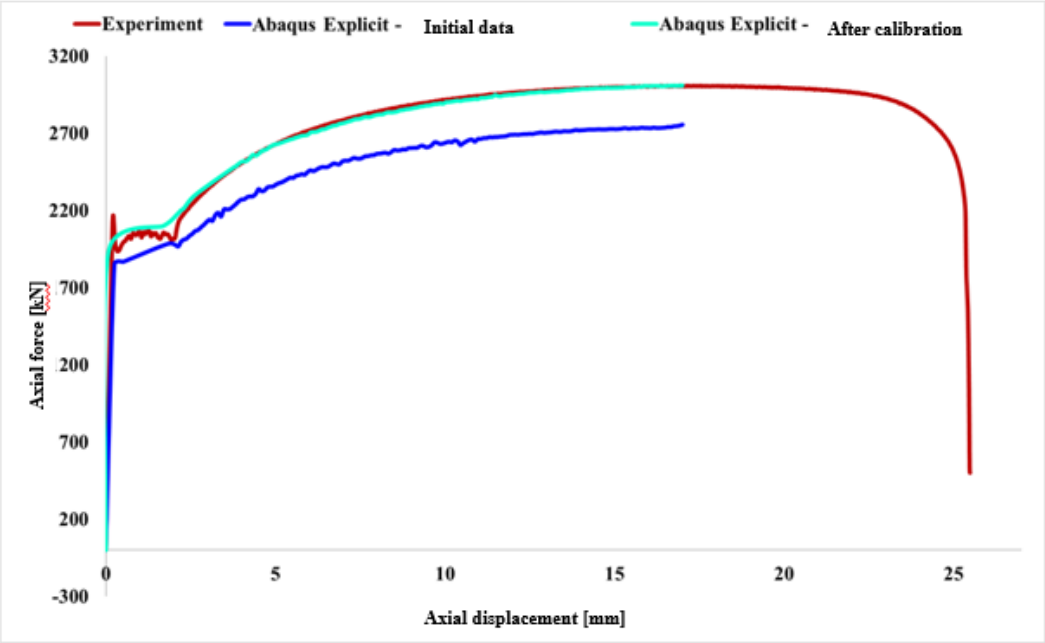
7. The observed standard deviations and coefficients of variation indicate the consistency of results between specimens, and the provided p-values suggest that variations in data are not statistically significant, indicating good data repeatability and reliability.

Specimen Thickness [mm]	Cutting Direction	Test Number	Young's Modulus [MPa]	Yield Strength [MPa]	Ultimate Tensile Strength [MPa]	Strain Hardening Coefficient	Strength Coefficient [MPa]	Elongation at Break [mm/mm]
0.8	0°	1.	78422.99	176.21	270.75	0.247	497.95	0.317
		2.	84422.74	170.36	268.57	0.251	496.54	0.338
		3.	83455.18	152.26	257.15	0.264	486.18	0.331
		4.	83123.20	167.40	264.97	0.252	491.96	0.316
		5.	80347.35	175.88	273.23	0.245	500.75	0.318
		6.	81353.10	168.88	262.76	0.251	487.17	0.335
		Average	81854.092	168.498	266.239	0.252	493.424	0.326
		Median	82238.150	169.618	266.771	0.251	494.249	0.325
		Standard deviation	2239.006	8.747	5.848	0.006	5.960	0.010
		Coefficient of variation	2.735	5.191	2.197	2.554	1.208	3.051
		p-value	0.716	0.131	0.892	0.106	0.538	0.156

These experimental data will be used to calibrate the DC01 material model in ABAQUS. By adjusting material parameters in Isight to match these average values and distributions, we can ensure that the model will accurately reflect the observed material behavior under load. The calibration process will include optimizing the material model to match the experimentally obtained stress-strain curve, ensuring that finite element simulations accurately reproduce DC01 behavior in real structural and mechanical applications.

4. CALIBRATION OF DC01 STEEL IN ABAQUS

Calibrating the DC01 material model in ABAQUS was accomplished by comparing experimental results with those obtained from ABAQUS Explicit simulations. The graph presented illustrates the material's response under applied load, representing the stress-strain curve obtained from physical tensile tests (red line) and the corresponding curve generated by the ABAQUS simulation (blue line).



Comparing the initial slopes of both curves, it is evident that Young's Modulus (stiffness) of the ABAQUS model is calibrated closely to the experimental values. This ensures that the initial elastic phase of the material's behavior is accurately represented in simulations. The yield point, where the material begins to deform plastically, seems to be well-adjusted in the ABAQUS model. This is crucial to simulate the moment when the material deviates from pure elastic behavior. The strain hardening zone, or material strengthening with increasing plastic deformation, appears to be represented with some variation between experimental data and the ABAQUS model. Fine-tuning the strain hardening coefficient in the model may be necessary to improve the accuracy of representing the strain hardening phase. The point of ultimate strength and elongation at break captured in the ABAQUS simulations align well with experimental results, with minor improvements possible for even more accurate representation.

The final calibration of the material in ABAQUS requires an iterative process of adjusting material parameters using Isight to closely match experimental data. The process would include adjustments to Young's Modulus, yield strength, strain hardening coefficient, and other relevant parameters until the simulation closely matches experimental results.

Precise calibration ensures that subsequent simulations conducted in ABAQUS will accurately reflect the material's behavior under load with a high level of accuracy, facilitating predictive design and structural analysis in a virtual environment. This is essential for anticipating material behavior in real operating conditions, enabling engineers to make informed decisions in the design phase and avoid unexpected failures in application.

5. CONCLUSIONS AND DISCUSSIONS

In this article, we have investigated the detailed process of calibrating DC01 material properties using finite element analysis in ABAQUS, coupled with the optimization capabilities provided by Isight. The aim was to ensure precise modeling of material behavior under different loading conditions, with a special focus on critical mechanical properties: yield strength and tensile strength. Through rigorous tensile tests, we obtained a set of experimental data that defined the stress-strain curve for DC01 steel, highlighting the elastic and plastic regions, the yield point, and the fracture point. This information was essential for adjusting the material model in ABAQUS to closely match observed behavior.

Within the methodology, we detailed the processes by which specimens were prepared and tested, and described the iterative calibration process in ABAQUS using Isight. This involved fine-tuning material parameters such as Young's Modulus and yield strength to align simulations with the material behavior recorded in physical tests. Young's Modulus of the ABAQUS model was calibrated to closely match the experimental values, providing a solid foundation for simulating the initial elastic phase of material behavior. The yield strength and strain hardening zone were adjusted with reasonable accuracy, though further optimizations may benefit to more accurately reflect material behavior in ABAQUS. The point of ultimate strength and elongation at break captured in ABAQUS simulations closely align with experimental results, with minor adjustments possible for even more accurate representation.

Calibrating DC01 material in ABAQUS, facilitated by the use of Isight, significantly enhances the efficiency and accuracy of simulations under various loading scenarios. This calibration process is not just a theoretical exercise but an essential step in product design and testing in a predictive virtual environment. The results indicate notable success in anticipating material behavior in real conditions, allowing engineers to make informed decisions and minimize the risk of failures in implementation.

In conclusion, the work presented here underscores the importance of a meticulous approach in calibrating material properties in finite element analysis and demonstrates the effectiveness of integrating ABAQUS and Isight as a powerful tandem for advanced structural and mechanical engineering.

References

1. Kurkin, O., Januška, M., Product Life Cycle in Digital factory. Knowledge management and innovation: a business competitive edge perspective, *International Business Information Management Association*, pp. 1881 – 1886 (2010)
2. Barlat, Frédéric., Constitutive Modeling for Sheet Metal Forming (2005)
3. Zhou, Zhi-Dong., Zhao, She-Xu., Kuang, Zhen-Bang., An integral elasto-plastic constitutive theory, *International Journal of Plasticity*, 19, pp. 1377 – 1400 (2003)
4. Liu, I. – S., Constitutive theories: basic principles, *Continuum Mechanics, Encyclopedia of Life Support Systems, The UNESCO Publications*, 1, pp. 198 (2011)
5. Banabic, Dorel., *Sheet Metal Forming Processes*, Springer-Verlag, Berlin (2010)
6. Butuc, M. C., Teodosiu, C., Barlat, F., Gracio, J. J., Analysis of sheet metal formability through isotropic and kinematic hardening models, *Journal of Mechanics - A/Solids*, 30, pp. 532 – 548 (2011)
7. Ahmadi, S., Eivani, A.R., Akbarzadeh, A., An experimental and theoretical study on the prediction of forming limit diagrams using new BBC yield criteria and M–K analysis, *Computational Materials Science*, 44 (2009)
8. Petrașcu, O.-L., Manole, R., Pascu, A. -M., The behavior of composite materials based on polyurethan resin subjected to uniaxial tensile test, *Materials Today: Proceedings*, 62 (2022)