

Comparative Study of the Mechanical Performance of SKD-11, SKD-61, and ST-41 Steels for Brake Handle Mold Design Using Finite Element Analysis

Reson Wibowo^{1✉}, Nurjaya², Muhammad Aldi Hartadi³, Jaja Miharja⁴,
Agus Bayu Utama⁵, Ruhama Sidqy⁶

^{1, 2, 3, 4, 5, 6} Department of Mechanical Engineering, Universitas Pamulang, Indonesia

✉ Corresponding Author : Reson Wibowo (e-mail: dosen01649@unpam.ac.id)

Article Information

Article History

Received : May 02, 2026

Revised : May 12, 2026

Published : July 02, 2026

Keywords:

*Brake Handle Mold;
Finite Element Simulation;
Industrial Tool Steel;
Structural Reliability;
Stress Distribution.*

ABSTRACT

This study investigates the mechanical performance of SKD-11, SKD-61, and ST-41 steels for brake handle mold applications using Finite Element Analysis (FEA). The increasing demand for durable and reliable mold materials in automotive manufacturing requires accurate evaluation methods to determine the most suitable steel material. However, comparative studies evaluating industrial steels under identical loading conditions remain limited. The research employed a numerical simulation approach using three-dimensional CAD modeling and static structural analysis. Material properties including Young's modulus, yield strength, and Poisson's ratio were assigned to each material. A load of 72,000 N, generated from a pressure of 9 bar acting on a surface area of 0.08 m², was applied to the brake handle mold model. Stress distribution, strain behavior, and safety factor were evaluated for each material. The simulation results indicate that all materials experienced similar stress values due to identical geometry and loading conditions. SKD-11 produced the highest safety factor of 3.05, followed by ST-41 with 2.80, while SKD-61 showed the lowest safety factor of 1.50. The strain values remained within the elastic region for all materials. The findings demonstrate that SKD-11 provides the best structural reliability and mechanical performance for brake handle mold applications. This study confirms that Finite Element Analysis is an effective approach for evaluating material selection in industrial mold design.

INTRODUCTION

The automotive industry requires high reliability in braking system components due to their critical role in ensuring operational safety (Budynas & Nisbett, 2020). Brake handle molds are essential in the manufacturing process as they influence product quality, dimensional accuracy, and structural integrity (Groover, 2010). Therefore, selecting appropriate materials is crucial for ensuring durability and performance under operational loads (Ashby, 2011). Various steel materials such as SKD-11, SKD-61, and ST-41 are commonly used in mold manufacturing. SKD-11 is known for its high hardness and wear resistance, SKD-61 offers good thermal stability and toughness, while ST-41 is widely used due to its ductility and cost-effectiveness (Callister & Rethwisch, 2020). However, selecting the most suitable material remains a challenge due to trade-offs between strength, deformation, and safety factor.

In recent years, the development of tool steel materials and simulation-based design methods has significantly improved manufacturing efficiency and structural reliability in

industrial components. SKD-11 (cold work tool steel) is widely recognized for its high wear resistance and hardness, while SKD-61 (hot work tool steel) offers excellent thermal resistance and toughness under elevated temperatures. Meanwhile, ST-41 carbon steel remains commonly used in industrial applications due to its economical cost and adequate ductility. Despite these advantages, selecting the optimum material for brake handle mold applications remains challenging because each material exhibits different mechanical responses under loading conditions.

Finite Element Analysis (FEA) has been widely used to evaluate stress distribution and structural behavior in mechanical components (Logan, 2016; Zienkiewicz et al., 2014). Previous studies have shown that material properties significantly influence structural performance (Kim et al., 2022; Santos et al., 2022). However, limited research compares multiple industrial steels under identical conditions for brake handle mold applications. Previous studies have mainly focused on stress analysis of individual steel materials or specific mold components. However, limited studies provide a direct comparative evaluation of SKD-11, SKD-61, and ST-41 under identical loading and boundary conditions using Finite Element Analysis. In addition, recent developments in simulation-based engineering require more accurate material selection approaches to improve safety factors and structural reliability in manufacturing applications. Therefore, this study aims to compare the mechanical performance of SKD-11, SKD-61, and ST-41 using FEA, focusing on stress, strain, and safety factor to determine the most suitable material.

RESEARCH METHODS

Research Design

This study employed a numerical simulation approach based on Finite Element Analysis (FEA) to evaluate the mechanical performance of SKD-11, SKD-61, and ST-41 steels in brake handle mold applications. The research stages included geometry modeling, material assignment, loading, meshing, simulation, and evaluation. A three-dimensional model of the brake handle mold was developed using CAD software and imported into the simulation environment. Material properties such as Young's modulus, yield strength, and Poisson's ratio were assigned based on standard references (Callister & Rethwisch, 2020).

Geometry Modeling and Material Properties

A three-dimensional model of the brake handle mold was designed using SolidWorks 2024 software and imported into ANSYS Workbench 2024 R1 for simulation analysis. Material properties including Young's modulus, yield strength, density, and Poisson's ratio were assigned based on engineering material standards and previous studies.

Tabel 1. Mechanical Properties of Materials

Material	Young's Modulus (GPa)	Yield Strength (MPa)	Poisson Ratio
SKD-11	210	470	0.30
SKD-61	207	230	0.29
ST-41	200	370	0.30

Simulation Procedure

The simulation procedure consisted of geometry import, material assignment, meshing, boundary condition setup, loading application, solving process, and post-processing analysis. Static structural analysis was performed under identical loading conditions for all materials to ensure consistent comparison.



Figure 1. Research Methodology Flowchart

Loading Calculation

A static load of 72,000 N was applied uniformly to the contact surface during the simulation process. The applied force was calculated using :

$$F = P \times A \quad (1)$$

Where :

$F = \text{Force (N)}$

$P = \text{Pressure (Pa)}$

$A = \text{Area (m}^2\text{)}$

With

$$P = 9 \times 10^5 \text{ N/mm}^2$$

$$A = 0,08 \text{ m}^2$$

$$F = 9 \times 10^5 \times 0,08 = 72000 \text{ N} \quad (2)$$

Meshing and Boundary Conditions

The meshing process was performed to discretize the geometry into finite elements. A refined mesh was applied in critical regions to improve simulation accuracy, while coarser elements were used in non-critical regions to optimize computational efficiency. A refined mesh was used in critical regions to improve accuracy, while a coarser mesh was applied in non-critical areas to optimize computational efficiency. The mesh quality was verified to ensure convergence and stability of the simulation results (Logan, 2016). Fifth, the simulation was executed under static loading conditions. The boundary conditions were defined to represent realistic constraints during operation. The analysis focused on evaluating stress distribution, strain behavior, and safety factor.

Finally, the results obtained from the simulation were analyzed and compared across all materials. The comparison was conducted based on maximum stress values, strain distribution, and safety factor to determine the most suitable material for brake handle mold applications.

Fixed support constraints were applied to the mounting regions of the brake handle mold to represent actual operating conditions. The simulation results were analyzed based on von Mises stress distribution, equivalent elastic strain, and safety factor values to determine the most suitable material for brake handle mold applications.

Data Analysis

The simulation results were analyzed based on von Mises stress distribution, equivalent elastic strain, and safety factor values to determine the most suitable material for brake handle mold applications.

RESULTS AND DISCUSSION

Comparative Stress Analysis

To provide a comprehensive comparison of the materials mechanical behavior, the simulation results are summarized in Table 1. The evaluation is based on stress, strain, and safety factor, which serve as critical indicators of structural reliability and performance. This comparison enables the identification of the most suitable material for brake handle mold applications.

Tabel 2. Comparative Mechanical Properties of Materials

Material	Stress (MPa)	Strain	Safety Factor
SKD-11	153,781	0.001	3.05
SKD-61	152,269	0.001	1.50
ST-41	153,781	0.001	2.80

The stress analysis results indicate that all materials experienced relatively similar von Mises stress values because the geometry model, loading magnitude, and boundary conditions were identical. According to Wang et al. (2020), stress distribution in FEA simulations is strongly influenced by geometric configuration and load application.

Stress Distribution Characteristics

Figure 2 shows the stress distribution of SKD-11. Stress concentration occurs in critical regions such as edges and load contact areas, with a maximum stress of approximately 153.78 MPa.

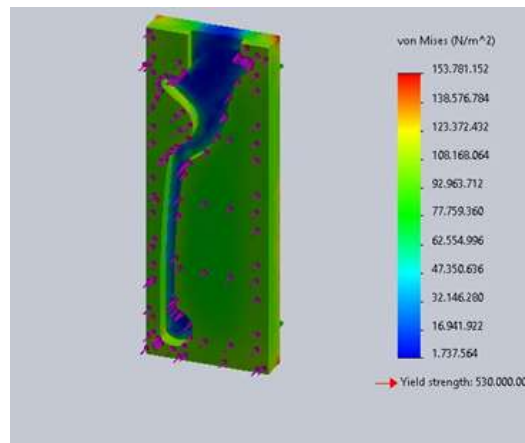


Figure 2. Stress Distribution of SKD-11

The results show that stress concentration occurred near sharp edges and load contact regions. This phenomenon commonly occurs in mechanical structures subjected to static loading because geometric discontinuities increase local stress intensity (Rahman et al., 2021).

Figure 3 presents the stress distribution of SKD-61, showing a similar pattern due to identical geometry and loading conditions. However, the material exhibits a lower safety margin.

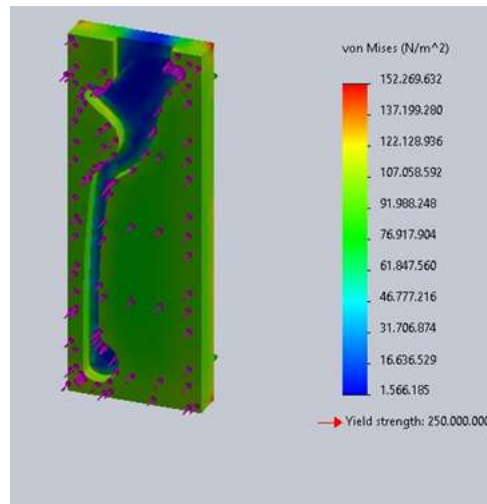


Figure 3. Stress Distribution of SKD-61

The stress distribution pattern was relatively similar to SKD-11 because identical geometry and loading conditions were applied during simulation. However, SKD-61 exhibited lower structural reliability due to its lower yield strength.

Figure 4 illustrates the stress distribution of ST-41, confirming that stress behavior is primarily influenced by geometry rather than material type.

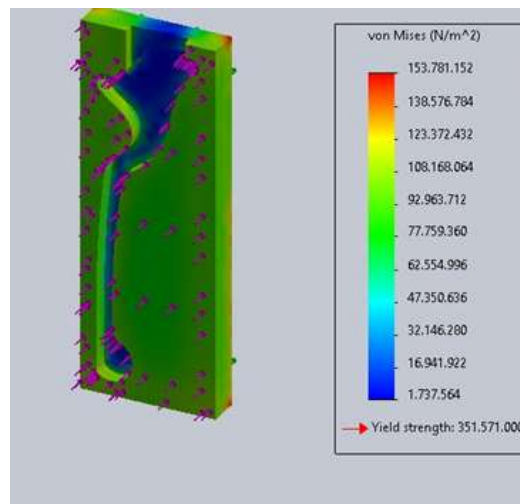


Figure 4. Stress Distribution of ST-41

The simulation results indicate that stress behavior was primarily controlled by geometry configuration rather than material type. Nevertheless, the safety factor remained dependent on material mechanical properties.

Strain Behavior Analysis

Figure 5 shows the strain distribution of SKD-11, indicating uniform deformation within the elastic region.

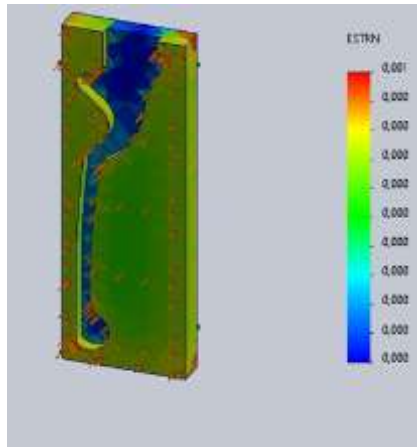


Figure 5. Strain Distribution of SKD-11

The results indicate that deformation remained within the elastic region, suggesting that no permanent deformation occurred during the simulation.

Figure 6 presents the strain distribution of SKD-61, showing similar deformation behavior with slightly higher sensitivity.

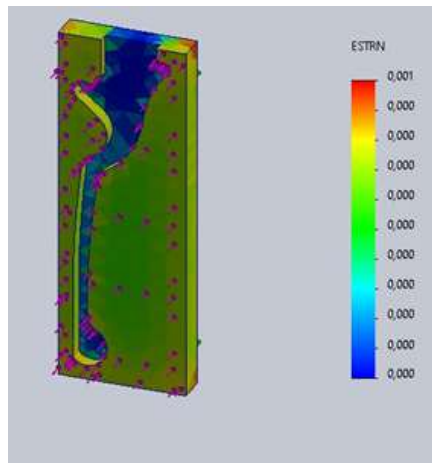


Figure 6. Strain Distribution of SKD-61

Although the strain values were relatively similar, slight differences were influenced by the elastic modulus of each material. Materials with higher stiffness generally exhibit lower elastic deformation under equivalent loading conditions.

Figure 7 illustrates the strain distribution of ST-41, confirming consistent elastic deformation across materials.

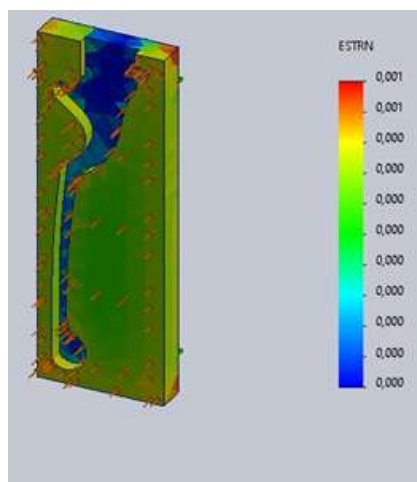


Figure 7. Strain Distribution of ST-41

The simulation confirms that all materials remained within acceptable elastic deformation limits under operational loading conditions.

Safety Factor Evaluation

Figure 8 shows that SKD-11 has the highest safety factor (3.05), followed by ST-41 (2.80) and SKD-61 (1.50). This indicates that SKD-11 provides the best resistance to failure.

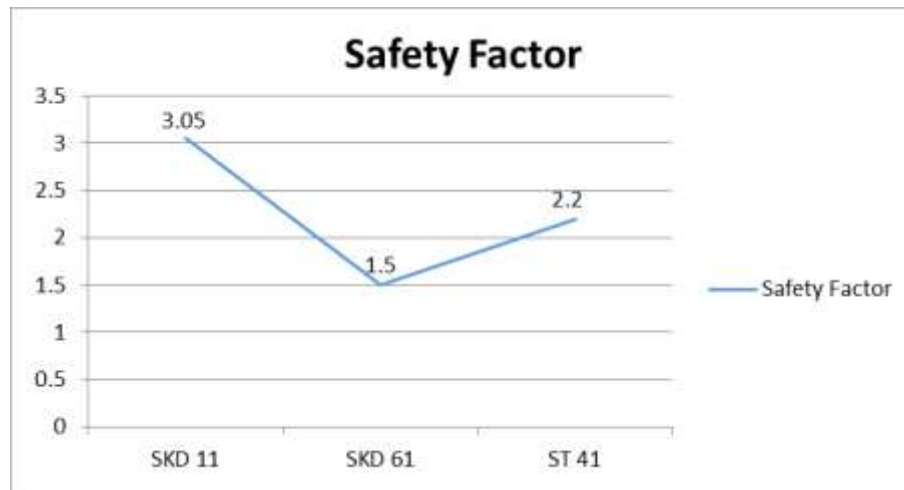


Figure 8. Safety Factor Comparison

The results demonstrate that SKD-11 possessed the highest safety factor, followed by ST-41 and SKD-61. According to Chen et al. (2023), safety factor values above 2 generally indicate acceptable structural performance for industrial applications. Therefore, SKD-11 is considered the most suitable material for brake handle mold applications because it provides higher resistance to yielding and structural failure.

CONCLUSION

This study successfully evaluated the mechanical performance of SKD-11, SKD-61, and ST-41 steels for brake handle mold applications using Finite Element Analysis. The simulation results demonstrated that all materials experienced relatively similar stress distributions because identical geometry and loading conditions were applied during the simulation process. However, significant differences were observed in safety factor values due to variations in material mechanical properties. SKD-11 exhibited the highest safety factor of 3.05, indicating superior structural reliability and resistance to mechanical failure compared to SKD-61 and ST-41. Meanwhile, SKD-61 showed the lowest safety factor, suggesting lower structural performance under identical loading conditions.

The FEA-based approach provides advantages in evaluating material performance efficiently without requiring extensive experimental testing. However, this method is limited by assumptions of linear elastic behavior and static loading conditions. Therefore, future studies should include thermal analysis, fatigue loading, and experimental validation to improve simulation accuracy and industrial applicability.

ACKNOWLEDGEMENTS

The authors would like to thank Universitas Pamulang for supporting this research.

REFERENCES

Ashby, M. F. (2011). *Materials selection in mechanical design* (4th ed.). Elsevier.

- Budynas, R. G., & Nisbett, J. K. (2020). *Shigley's mechanical engineering design* (11th ed.). McGraw-Hill.
- Callister, W. D., & Rethwisch, D. G. (2020). *Materials science and engineering: An introduction* (10th ed.). Wiley.
- Chen, G., Xu, B., & Li, Y. (2023). Structural reliability analysis using safety factor approach in FEA. *Structures*, 47, 1450–1460.
- Groover, M. P. (2010). *Fundamentals of modern manufacturing: Materials, processes, and systems* (4th ed.). Wiley.
- Hassan, M., Ali, K., & Rehman, U. (2024). Comparative study of stress and deformation in engineering materials using FEA. *Engineering Reports*, 6(2), e12678.
- Hibbeler, R. C. (2017). *Mechanics of materials* (10th ed.). Pearson.
- Kalpakjian, S., & Schmid, S. (2014). *Manufacturing engineering and technology* (7th ed.). Pearson.
- Kim, J., Park, S., & Lee, D. (2022). Comparative analysis of tool steels under mechanical loading using FEA. *Materials Today Communications*, 31, 103567.
- Liu, H., Zhang, Q., & Sun, J. (2022). Mechanical performance evaluation of industrial steels under static loading. *Materials Science and Engineering A*, 832, 142355.
- Logan, D. L. (2016). *A first course in the finite element method* (6th ed.). Cengage Learning.
- Nguyen, T., Pham, H., & Tran, D. (2021). Application of finite element analysis in mold design optimization. *The International Journal of Advanced Manufacturing Technology*, 115, 2345–2356.
- Papazafeiropoulos, G., Muñoz-Calvente, M., & Martínez-Pañeda, E. (2020). Abaqus2Matlab: A suitable tool for finite element post-processing. *Advances in Engineering Software*, 105, 9–16.
- Prasetyo, B., Nugroho, S., & Wibowo, A. (2023). Analisis elemen hingga pada komponen mekanik berbasis material baja. *Jurnal Teknik Mesin Indonesia*, 18(2), 85–92.
- Rahman, M., Islam, M., & Hossain, M. (2021). Numerical investigation of stress concentration in mechanical components. *Results in Engineering*, 10, 100245.
- Santos, J., Oliveira, P., & Costa, M. (2022). Evaluation of material selection in mold manufacturing using simulation. *Journal of Manufacturing Systems*, 63, 250–260.
- Singh, R., Kumar, A., & Patel, V. (2023). Mechanical behavior of tool steels in manufacturing applications. *Journal of Manufacturing Processes*, 85, 112–120.
- Siregar, D., Putra, R., & Hidayat, T. (2024). Studi numerik sifat mekanik baja karbon menggunakan metode FEA. *Jurnal Rekayasa Mesin*, 15(1), 45–53.
- Wang, L., Chen, X., & Zhao, Y. (2020). Stress and strain analysis of steel components using finite element methods. *Engineering Failure Analysis*, 115, 104632.
- Zhang, Y., Li, X., & Wang, Z. (2021). Finite element analysis of stress distribution in metal forming processes. *Journal of Materials Processing Technology*, 291, 117044.
- Zienkiewicz, O. C., Taylor, R. L., & Zhu, J. Z. (2014). *The finite element method: Its basis and fundamentals* (7th ed.). Elsevier.