

Simulation of Material Compression Testing on Piston Heads and Molds Using Finite Element

Tessalonika Bori¹, Farrel Azhar Putra Adrian², Rafi Galih Prakoso², Zazkia Istiqomah Hayashi Fazrin Sujana¹, Stefanus Nathanael Mahesa¹, Dandun Mahesa Prabowoputra^{1*}

¹Program Studi Teknik Mesin, Fakultas Teknik, Universitas Jenderal Soedirman, Jl. Raya Mayjen Sungkono, Purbalingga, Jawa Tengah, Indonesia

²Program Studi Teknik Industri, Fakultas Teknik, Universitas Jenderal Soedirman, Jl. Raya Mayjen Sungkono, Purbalingga, Jawa Tengah, Indonesia,

*Corresponding author: dandun.mahesa@unsoed.ac.id

Abstract. This study aims to evaluate and compare the mechanical response of materials in piston components and their molds through compression test simulation based on the finite element method (Finite Element Analysis/FEA) using SolidWorks software. The research stage includes three-dimensional modeling of the piston and mold, mesh generation to ensure the stability of the simulation results, and compression testing with load variations of 5000 N, 10000 N, 15000 N, 20000 N, and 25000 N. The materials analyzed for the piston include Alloy Steel, Cast Alloy Steel, and A286 Iron-Based Superalloy, while the mold materials include Alloy Steel, AISI 4340, and AISI 4130. The primary parameter analyzed is stress distribution as an indicator of stress distribution and potential numerical errors. The simulation results show that Alloy Steel in the piston produces the lowest stress variation values compared to other materials, indicating better stress stability. Meanwhile, for the mold, AISI 4340 was selected as the best material because it has the smallest stress variation values, indicating a lower level of simulation error. Based on these results, Alloy Steel is recommended as the piston material, while AISI 4340 is considered the most optimal for the piston mold application.

Keywords: compression test, finite element analysis, material selection, piston, stress variation.

How to cite (in APA style):

Bori, T., Adrian, F.A.P., Prakoso, R.G., Istiqomah, Z., Sujana, H.F., Mahesa, S.N., & Prabowoputra, D.M. (2026). Simulation of Material Compression Testing on Piston Heads and Molds Using Finite Element. *Proceeding of IJC-MaSTEd*, 1(1), 305-310.

Copyright © 2026 Auhors.

DOI: <https://doi.org/10.31571/ijcmasted.v1i1.1031>

1. Introduction

Piston components are a critical part of internal combustion engines because they play a role in transferring force from gas expansion to the crankshaft; therefore, the stress distribution and mechanical response of the piston material must be thoroughly analyzed to ensure its strength and durability under actual operating loads. One approach that is often used to evaluate the mechanical performance of components is Finite Element Analysis (FEA), which enables numerical modeling of structural behavior under various loading conditions. The FEA method has been widely used in stress analysis of pistons to identifying regions with maximum stress concentrations and predicting the mechanical performance of these components (Sowmyashree & Aziz, 2018). Several studies on the application of FEA to pistons have been published in the last decade. Sowmyashree and Aziz applied FEA to map stress distributions on internal combustion engine pistons using ANSYS software, which identified critical regions of maximum stress on the top of the piston during operation (Sowmyashree & Aziz, 2018). Other research has also used FEA for structural analysis of pistons, including static and thermal analysis to understand stress and deformation behavior under combined load conditions (Patel & Chauhan, 2019). However, most previous studies have focused

on aluminum materials and specific piston geometries, while comparative research on high-strength structural metals under a wide range of compressive loads remains limited (Kuttin et al., 2024).

In addition to the analysis of the piston itself, the selection of mold materials is also critical for the manufacturing process of high-precision components. Mold materials such as AISI 4340 and AISI 4130 possess distinct mechanical properties, such as high tensile strength and resistance to deformation, which can influence the quality of component production. Studies on the use of advanced mold materials in FEA simulations of pistons are relatively scarce, so there remains a need for research that combines the evaluation of piston and mold materials simultaneously (Rahman et al, 2025).

Based on this literature review, there is a research gap in the literature regarding the integration of high-level comparative analysis of piston and mold materials under a broader range of compressive loads (5,000 N to 25,000 N). Furthermore, the scarcity of studies evaluating variations in stress modeling as indicators of stress stability and numerical errors also highlights the need for further research in this field. This is what distinguishes this study from previous ones as it focuses on a comparative analysis of three piston materials (Alloy Steel, Cast Alloy Steel, A286 Iron-Based Superalloy) and three mold materials (Alloy Steel, AISI 4340, AISI 4130) using FEA simulation (Kuttin et al., 2024).

The main issue addressed in this study is how the stress distribution characteristics (stress modeling) of various piston and mold materials behave when subjected to varying compressive loads, as well as which materials exhibit the most mechanically stable performance in the context of FEA simulations. To address these issues, this study employs a numerical approach through three-dimensional modeling, mesh refinement, and compression test simulations in SolidWorks software with specified variations in compressive loads. The key parameters analyzed are variations in stress modeling that indicate the stability of simulation results and potential modeling errors (Slamet, 2016).

The primary objective of this study is to provide recommendations on the most suitable materials for pistons and molds based on stress performance generated in the FEA simulation. The scientific contributions of this study include the provision of comparative data on the stress behavior of high-strength structural metals under a wide range of compressive loads and the identification of materials that exhibit the best stress stability for piston and mold applications (Kuttin et al., 2024).

2. Method

This study employs a numerical simulation method using SolidWorks Simulation software to analyze the mechanical response of the piston and mold. Prior to the main simulation, a mesh study was conducted to ensure that the results of the Finite Element Analysis (FEA) were stable, accurate, and not significantly influenced by mesh size. Next, a three-dimensional model of the piston and mold was created with geometries representing actual conditions. After the design was completed, the materials to be used were specified, namely Alloy Steel, Cast Alloy Steel, and A286 Iron-Based Superalloy for the piston, as well as Alloy Steel, AISI 4340, and AISI 4130 for the mold.

The next step is a simulation of static compression testing on the piston and mold using various compression forces. Before the testing begins, initial parameters are set, including geometric dimensions, material, and loading conditions on the piston head. The initial parameters used are a cylinder with a diameter of 80 mm, a length of 120 mm, and a cross-sectional area of 5024 mm². The material used is Alloy Steel with an axial load of 10,000 N. These parameters are used to calculate stress analytically as a basis for validating the simulation results. To obtain accurate results, a mesh

convergence study was conducted with element sizes ranging from 6 mm to 2 mm. The stress values from the simulation were compared with analytical stresses to determine the optimal mesh size.

Table 1. Mesh size testing

No	Size Mess (mm)	Number of Elements	Stress Level (Mpa)	Error
1	6	15953	2,013	1,12%
2	5	27089	1,989	0,07%
3	4	47580	1,989	0,07%
4	3	99592	1,989	0,07%
5	2	259431	1,989	0,07%

After determining the optimal mesh size, load variations of 5,000 N, 10,000 N, 15,000 N, 20,000 N, and 25,000 N were applied. Next, the piston head was modeled using SolidWorks software and analyzed with Finite Element Analysis (FEA) method. A mesh check was performed again under the initial loading conditions to ensure the consistency of the simulation results.

Table 2. Experiment on Style Variations and Error Value

No	Variations Styles	Value Stress Modeling	Stress Value Analytical	Error
1	5.000	0,995	0,99522293	0,02%
2	10.000	1,989	1,99044586	0,07%
3	15.000	2,984	2,98566879	0,06%
4	20.000	3,979	3,98089172	0,05%
5	25.000	5,019	4,97611465	0,85%

The simulation results, consisting of stress distributions and stress values, were analyzed for each variation in material and load. Based on these results, an evaluation was conducted to determine the piston and mold materials with the most stable mechanical performance and to establish safe compression force limits for use in the production process.

3. Results and Discussion

An analysis was conducted using Finite Element Analysis (FEA) to evaluate the stress behavior in the piston head under static loading with variations in force magnitude and material type. The simulation results show that the magnitude of the stress in the piston head increases as the applied force increases, while the stress distribution pattern remains relatively similar under all test conditions. The highest stress generally occurs in the upper area of the piston head, which serves as the support point and critical point due to high pressure forces.

The simulation results for a load of 5000 N are shown in Table 3. Under these conditions, the average stress values generated by the three materials show no significant differences, except for the A286 Iron-Based Superalloy, and all three remain well below the yield strength of their respective materials. This indicates that at low load levels, the piston head structure remains within the elastic range, and the effect of differences in material mechanical properties on structural safety is not yet significantly apparent.

Table 3. Piston Loading Simulation

No	Mesh	Style	Variations Materials	Value Stress Modeling	Yield Strenght
1	5	5.000	Alloy Steel	0,617	620.422
2	5	5.000	Cast Alloy Steel	0,645	241.275
3	5	5.000	A286 Iron Base Superalloy	1,145	275.000

The results of the Finite Element Analysis (FEA) show the von Mises stress distribution on the piston head due to static loading. The maximum stress recorded is 0.853 MPa, while the average stress is in the range of 0.617 MPa. The minimum stress value is 0.194 MPa, with a relatively uniform stress distribution across the piston head surface show the von Mises stress distribution on the piston head due to static loading. The maximum stress recorded is 0.853 MPa, while the average stress is in the range of 0.617 MPa. The minimum stress value is 0.194 MPa, with a relatively uniform stress distribution across the piston head surface.

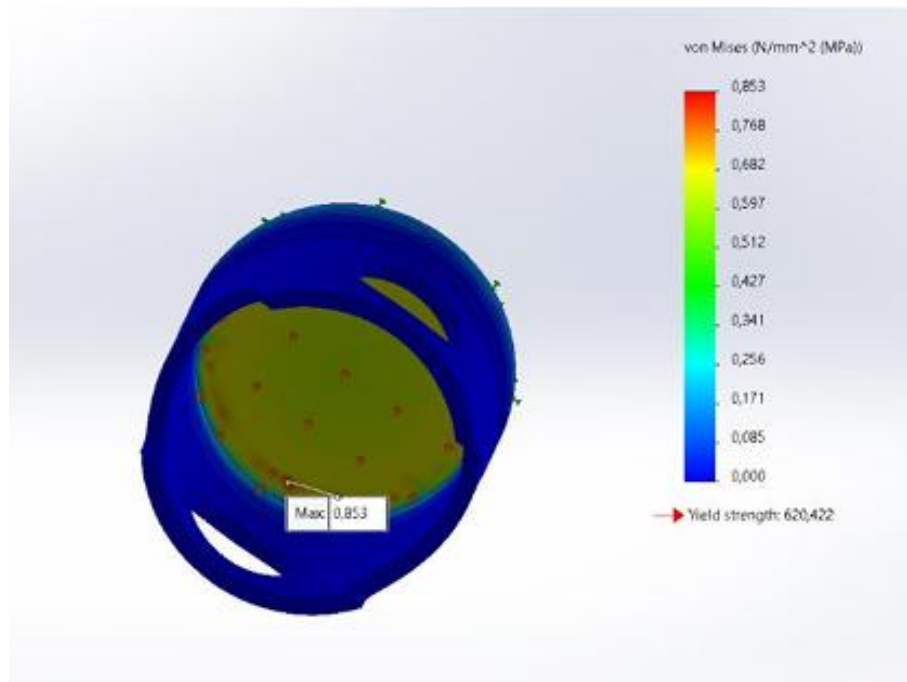


Figure.1. Von Mises stress distribution on the piston head for alloy steel material under a loading force of 5,000 N.

The highest stress concentrations are localized in the surface area of the piston head that bears the load directly, particularly around the center of the compression surface. Nevertheless, all the resulting stress values remain well below the yield strength of the Alloy Steel material at ± 620 MPa, so it can be concluded that the piston head is still operating under elastic conditions and is safe under the applied load. These results indicate that the piston head design has the ability to withstand loads effectively without the risk of plastic deformation, and the resulting stress distribution shows no signs of excessive stress concentration in critical structural areas.

In addition to testing the structural integrity of the piston head, an analysis was also conducted on the mold to evaluate the stress response caused by the pressure of the manufacturing process. High-Pressure Die Casting (HPDC) is a casting process in which molten metal is injected into the mold cavity under high pressure; during the intensification stage, the pressure applied to the molten metal is typically in the range of 70–150 MPa, depending on the process parameters and materials used, thereby ensuring proper filling of the mold cavity and reducing porosity in the final product (Shabadi et al., 2015).

The molding analysis was conducted by varying the mold material, namely Alloy Steel, AISI 4340 Steel, AISI 4130 Steel. Simulation results show that the von Mises stress distribution in the mold exhibits a relatively similar pattern across all materials, with maximum stress concentrations

occurring in the inner regions, edges, and cavity geometry of the mold. These areas are critical points due to the combination of high contact pressure and sharp geometric changes.

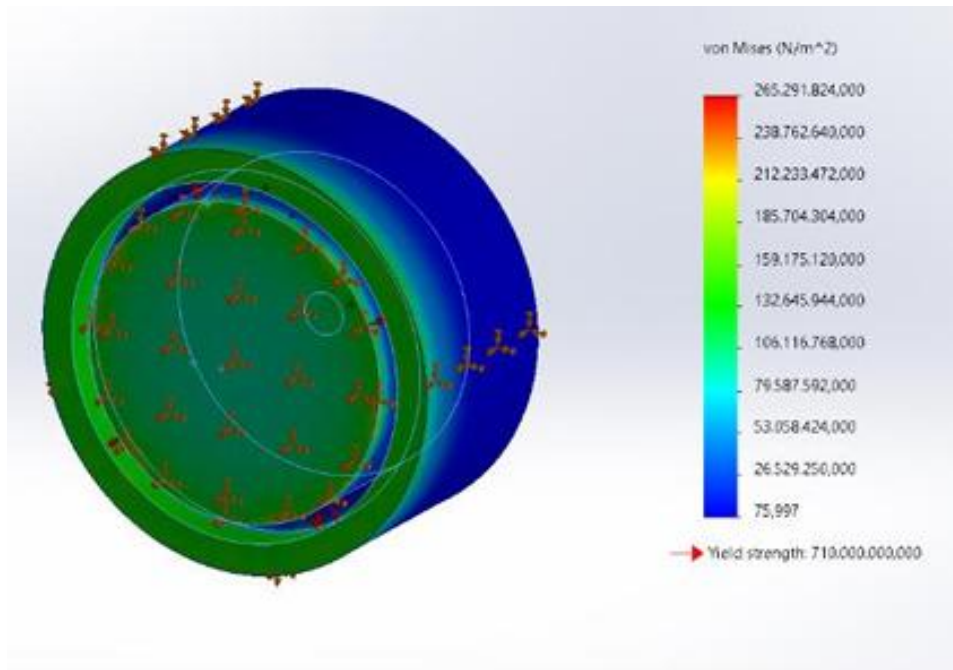


Figure.2. Von Mises stress distribution in piston castings made of AISI 4340 under a loading force of 150 N.

The results of the Finite Element Analysis (FEA) show the von Mises stress distribution in the piston mold component made of AISI 4340 under a molding pressure of 150 MPa. The maximum stress observed was ± 249.8 MPa, while the average stress was recorded at approximately 42.6 MPa. The minimum stress value was in the range of 3.04 MPa, indicating that most of the die area experienced relatively low stress.

The stress distribution shows that the highest stress concentrations are localized at the edges and in the areas where the mold geometry changes, particularly in regions in direct contact with the molding pressure. This phenomenon occurs due to changes in cross-section and the direction of force flow, which cause stress concentrations in those areas. Meanwhile, the outer wall of the mold exhibits lower and relatively uniform stress (Sivák et al., 2023).

When compared to the yield strength of the mold material, which is 710 MPa, all stress values obtained from the simulation remain well below that limit. This indicates that the piston mold structure is still operating under elastic conditions and is safe under the applied molding pressure, and is not at risk of plastic deformation during the molding process.

4. Conclusion

Based on the results of compression test simulations using Finite Element Analysis (FEA), it can be concluded that the stress response at the piston head and its mold increases linearly with increasing compressive load, but the stress distribution pattern remains relatively consistent across all variations in loading and material. The results of the analysis show that the Alloy Steel material in the piston produces the lowest stress variation values compared to Cast Alloy Steel and A286 Iron Base Superalloy, thus demonstrating greater stress stability and better mechanical performance under compressive loads up to 25,000 N. All stress values remain well below the material's yield strength,

indicating that the piston head operates under elastic conditions and is safe under the analyzed loading conditions.

In the analysis of the piston die, AISI 4340 material demonstrated the most optimal performance compared to Alloy Steel and AISI 4130, as evidenced by smaller stress variation values and a more stable stress distribution. The maximum stress concentration in the die is localized at the edge areas and the geometric transitions of the cavity due to the pressure of High Pressure Die Casting (HPDC). However, the maximum stress values obtained remain well below the yield strength of the die material. This indicates that the mold design and the material used are capable of withstanding molding pressures up to 150 MPa without the risk of plastic deformation.

Overall, the results of the study recommend Alloy Steel as the piston material and AISI 4340 as the piston die material based on stress stability and structural safety as determined by FEA simulations. For further research, it is recommended to conduct thermal and thermomechanical analyses, as well as experimental validation to obtain a better understanding of material behavior under conditions closer to actual operating conditions and to improve the reliability of the simulation results.

Acknowledgments

The author would like to express his deepest gratitude and highest appreciation to Mr. Dandun Mahesa Prabowoputra and Mr. Sugeng Waluyo, the instructors for the for their guidance, direction, and support provided throughout the conduct of this research. Their contributions and feedback played a crucial role in ensuring the smooth progress of the modeling, analysis, and preparation of this research, enabling it to be successfully completed.

Reference

- Kuttin, K. W., Yaw, A. O., Duodu, E. A., & Kuttin, A. W. (2024). Thermo-structural comparative analysis of internal combustion engine piston by finite element methods. *Journal of Engineering Research and Reports*, 26(11), 46–61. <https://doi.org/10.9734/jerr/2024/v26i111314>
- Patel, H., & Chauhan, V. (2019). Comparative study and static analysis of piston using finite element analysis. *International Journal for Research in Applied Science and Engineering Technology*, 7(6), 245–250.
- Rahman, H. A., Ghani, J. A., Rasani, M. R. M., Mahmood, W. M. F. W., Yaaqob, S., & Aziz, M. S. A. (2025). Application of finite element analysis and computational fluid dynamics in machining AISI 4340 steels. *Tribology International*, 207, 110616. <https://doi.org/10.1016/j.triboint.2025.110616>
- Slamet, S., & Hidayat, T. (2016). Perancangan Mesin High Pressure Die Casting/HPDC untuk memproduksi produk gamelan (Studi kasus: prototipe gamelan jenis saron). In *Seminar Nasional Teknologi dan Informatika 2016*. Muria Kudus University.
- Sowmyashree, P., & Aziz, A. (2024). FEM analysis and design optimization of the piston. *Journal of Scientific Research and Technology*, 6-18.
- Sivák, P., Delyová, I., & Bocko, J. (2023). Comparison of stress concentration factors obtained by different methods. *Applied Sciences*, 13(24), 13328. <https://doi.org/10.3390/app132413328>
- Shabadi, R., Avettand-Fènoël, M. N., Simar, A., Taillard, R., Jain, P. K., & Johnson, R. (2015). Thermal conductivity in yttria dispersed copper. *Materials & Design (1980-2015)*, 65, 869–877. <https://doi.org/10.1016/j.matdes.2014.09.083>