Jurnal Teknik Industri, Vol. XX, No. Y, December 20XX
ISSN 1411-2485 print / ISSN 2087-7439 online DOI: 10.9744/jti.XX.Y..-..

Analysis of Component Strength in Shock Absorber Test Machines Using the FEA Method

Agustinus Valentino Sonny Nugraha[1], Dorothy Puspitasari Santoso[2], Adhi Setya Hutama[3*], Bondan Wiratmoko[4], Abram Pangeling[5], Th. Adi Nugroho[6]
[1-5)] Program Studi Perancangan Manufaktur, Politeknik ATMI Surakarta
Jl. Mojo No. 1, Karangasem, Laweyan, Kota Surakarta, Jawa Tengah, 57145, Indonesia
Email: setya.hutama@atmi.ac.id*, agustinus.20215001@student.atmi.ac.id, dorothy.20215012@student.atmi.ac.id, bondan.wiratmoko@atmi.ac.id, abram.pangeling@atmi.ac.id, adi.nugroho@atmi.ac.id
*Corresponding author

Abstract: Shock absorber is one of the components in motor vehicles that functions to dampen shocks or vibrations while driving on the road. To ensure its quality, testing is essential, and the testing machine itself must be reliable. One critical component of the testing machine is pusher plate. This study evaluates pusher plate component by analyzing stress, displacement, and the factor of safety (FoS), also comparing two material variations S45C and SS400 and three loading variations of 9.000 N, 10.000 N, and 11.000 N. The analysis method used is Finite Element Analysis (FEA), simulated using CAE software. The results show that pusher plate design using S45C material offers better performance. The maximum Von Mises stress ranges from 382.086 MPa to 470.515 MPa under loads of 9.000 N to 11.000 N, which is still below the yield strength of S45C material at 530 MPa. The maximum displacement is recorded between 0.697 mm and 0.854 mm, while the minimum factor of safety (FoS) ranges from 1.126 to 1.387. Based on these parameters, it can be concluded that pusher plate design is safe and feasible for use. The findings of this study are valuable as a reference for automotive companies or shock absorber manufacturers.

Keywords: Computer Aided Engineering, Finite Element Analysis, Press Pull Machine, Shock Absorber, Stress, Displacement, Safety Factor, SolidWorks, Shock Absorber

Introduction

Shock absorbers is one of the components in motor vehicles that serve to dampen shocks or vibrations when driving on uneven or potholed roads [1]. The primary function of a shock absorber is to keep the vehicle's wheels in contact with the road surface and ensure that the vehicle remains stable and comfortable to drive [2]. A shock absorber consists of a tube filled with fluid, equipped with springs and a piston that can absorb and dampen vibrations and shocks occurring in the vehicle [3]. Shock absorbers work by dampening vibrations caused by the up-and-down movement of the wheels through the movement of the piston inside the fluid-filled cylinder, thereby converting kinetic energy into frictional energy and keeping the vehicle stable [4]. Because shock absorbers play an important role in a vehicle's suspension system, accurate testing methods are required to ensure their quality and performance before use. Testing of shock absorbers is used to evaluate their ability to absorb shocks and maintain vehicle stability. Shock absorber testing must consider various parameters, such as damping force, fluid viscosity characteristics, and dynamic response to load changes [5]. To meet these requirements, a specialized testing machine is needed that can simulate the loads and real-world conditions experienced by shock absorbers when the vehicle is in operation. This machine is the press-pull machine (Figure 1).

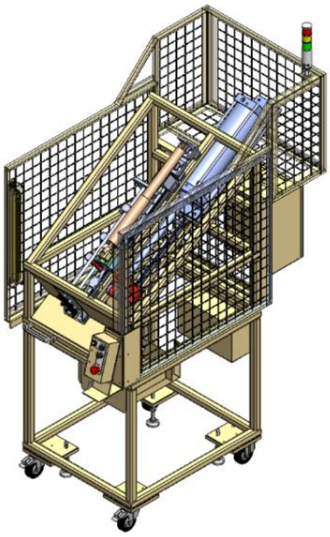


Figure 1 Press Pull Machine

In this study, the research object used for the study was the components of a pull press machine. The design of the pull press machine can be seen in (Figure 2). Through this research object, it is hoped that it can provide analysis results related to design strength.

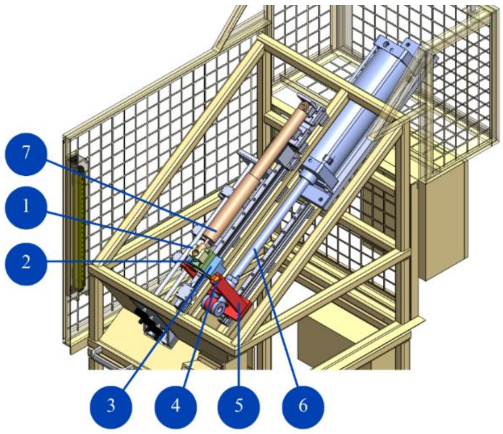


Figure 2 Analyzed Components of the Pull Press Machine

Table 1 Analyzed Components of the Pull Press Machine

No	Components
1.	Stopper
2.	Plate House
3.	Pusher Plate
4.	Hook
5.	Pusher Plate Hydraulic
6.	Hydraulic
7.	Shock Absorber

It is also known that there is one important component that receives special attention in the design of a pull press machine, namely pusher plate. Pusher plate in a pull press machine system functions to transmit pressure evenly to the surface of the shock absorber (Figure 3). This component acts as an intermediary between the hydraulic system and the workpiece, so that the pressure applied can be distributed symmetrically and stably without damaging the structure of the shock absorber. The primary function of the pusher plate is to maintain even force distribution and keep the shock absorber in the correct position and orientation throughout the testing process. Therefore, the pusher plate significantly influences the final production results and testing quality, particularly in mechanical strength testing. If the pusher plate is damaged, force distribution becomes uneven, potentially leading to test failure, shock absorber damage, and disruptions in the production process.

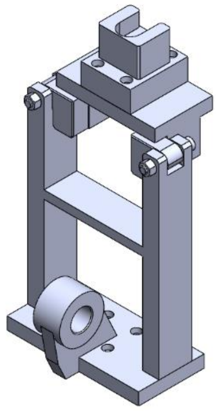


Figure 3 Pusher Plate Design

Based on this, a study was conducted to analyze the strength of the press pull machine with a focus on pusher plate, in order to determine its mechanical strength characteristics. This analysis was performed using the Finite Element Analysis (FEA) method, simulated through the Computer Aided Engineering (CAE) software SolidWorks. The analysis focused on stress, displacement, and safety factor parameters, with three load variations: 9.000 N, 10.000 N, and 11.000 N. Additionally, two material variations were used: SS400 and S45C. The objective of this research is to determine the success rate of the press pull design in terms of stress, displacement, and safety factor on the pusher plate component through simulation methods. The impact of this research is expected to be useful for automotive companies or shock absorber manufacturers to increase productivity and produce higher quality products.

Methods

Pusher Plate Testing

The tests conducted on pusher plate are tests to determine the characteristics of pusher plate part. Several tests that can be performed to determine these characteristics are:

- Stress
  Von Mises Stress testing in CAE is an FEA-based analysis method used to evaluate the strength and resistance of materials to mechanical loads. Von Mises Stress is used as a material failure criterion, especially for ductile materials such as steel and aluminum, which undergo plastic deformation before breaking [6]. Ductile materials will undergo yielding when the distortion energy per unit volume reaches or exceeds the distortion strain energy per unit volume for yield when the material is subjected to simple tensile or compressive stress [7].
- Displacement
  Displacement is one of the results of finite element analysis (FEA) that describes the shift (displacement) of points on an object due to external loading (force, torque, pressure, or other loads) [8].
- Safety Factor
  Safety of Factor (FoS) is the ratio between the material strength limit and the maximum stress caused by the working load. It indicates how safe a component or structure is against material failure [9]. In analysis using SolidWorks Simulation, the FoS value is typically calculated based on the comparison between yield strength and Von Mises Stress.

Finite Element Analysis (FEA)

- Definition
  Finite Element Analysis (FEA) is a numerical method used to solve various problems in engineering and mathematical physics [11]. In the process, the finite element method converts a physical system into a model consisting of small interconnected elements. Some of the fields that can be analyzed using this method include structural analysis, heat transfer, fluid flow, mass transfer, and electromagnetic waves.

In general, the main steps in applying the finite element method include:
1. Preprocessing: dividing the domain into finite elements.
2. Element formulation: determining the calculations for the elements.
3. Assembly: determining the calculations for the entire system from the calculations of each element.
4. Completing the calculations.
5. Postprocessing: interpreting the results.

Finite element method (FEM) is a very powerful tool that can be used to analyze various problems, both in structural and non-structural fields. In the structural context, FEM is often applied in the following aspects:
1. Stress Analysis
2. Failure and Buckling Analysis
3. Vibration Analysis
4. Impact Analysis

- CAE (Computer Aided Engineering)

  Computer Aided Engineering (CAE) is the use of computer software to assist in the engineering analysis of a product or system [12]. CAE covers various aspects of simulation, validation, and design optimization. The following is a flowchart of the CAE analysis process described in Figure 4.

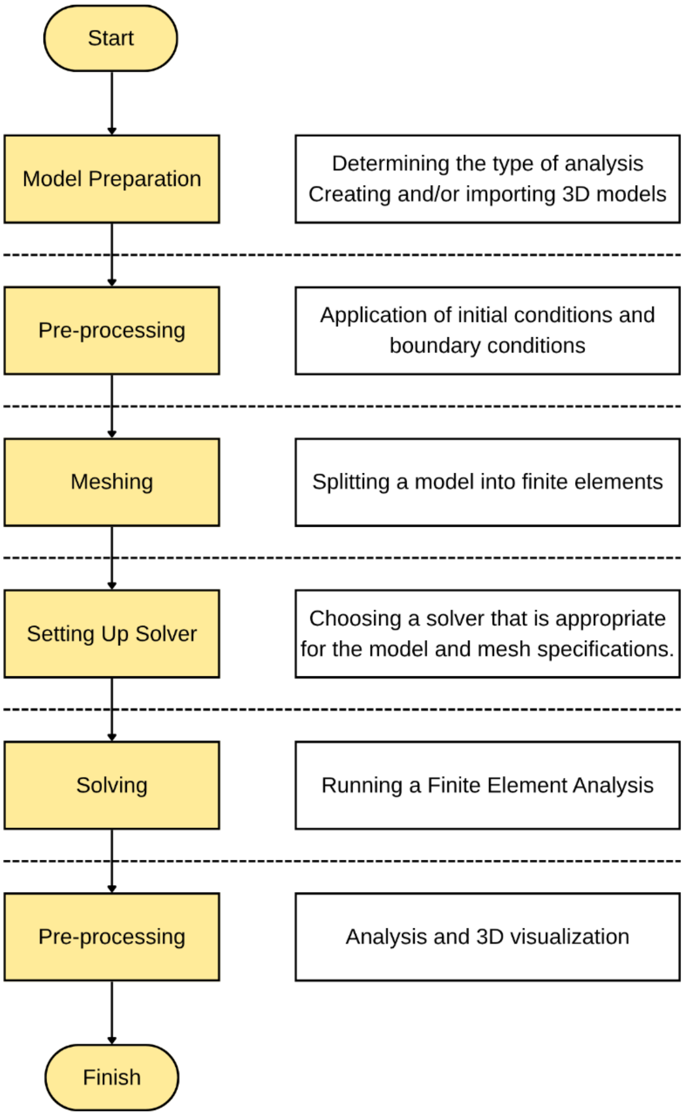


Figure 4 CAE Analysis Process Flowchart

Material

The purpose of analyzing pusher plate on a pull press machine using FEA method is to obtain a pull press machine that is safe during testing. A safe pull press machine is one that complies with the specifications regarding the materials used in its components. The material specifications used in the CAE simulation are presented in table 2 as follows:

Table 2 Material specifications

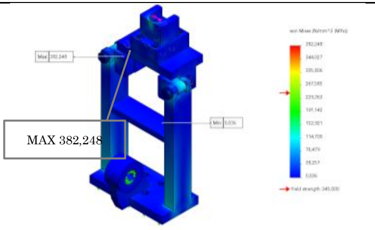
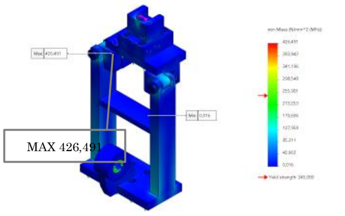
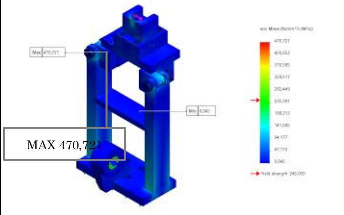
Material Properties	SS400	S45C
Young Modulus	210.000 Mpa	205.000 Mpa
Shear Modulus	77.000 Mpa	80.000 Mpa
Poison's Ratio	0,3	0,29
Density	7,85E-0.6 kg/*mm*3	7.850 kg/mm3
Yield Stress	245 Mpa	530 Mpa
Ultimate Stress	400 Mpa	565 Mpa

Results and Discussions

Result

1. SS400
   1.1. Stress

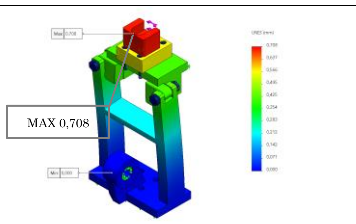
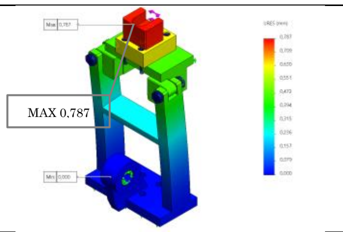
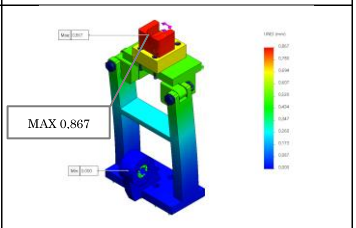
Table 3 Results of Von Mises Stress Analysis of SS400 Material

Load	9.000 N	10.000 N	11.000 N
Figure			

Based on table 3, the results of the Von Mises stress analysis with loads of 9.000 N, 10.000 N, and 11.000 N show that the maximum stress values are 382,248 MPa, 426,491 MPa, and 470,721 MPa, respectively. All stress values are still above the material's yield strength of 245 MPa, so it can be concluded that the component is unable to withstand the applied load due to plastic deformation under all three load conditions.

1.2. Displacement

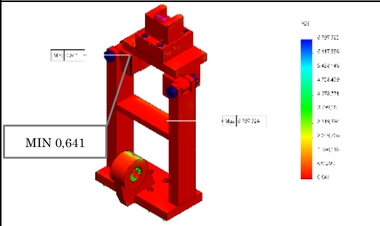
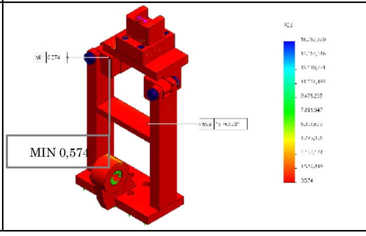
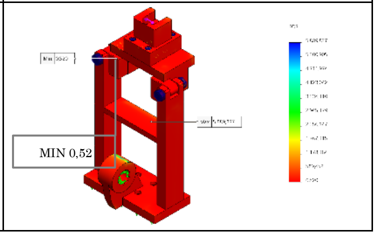
Table 4 Results of SS400 Material Displacement Analysis

Load	9.000 N	10.000 N	11.000 N
Figure			

Based on table 4, the displacement analysis results show that the greater the load applied, the greater the maximum displacement of the component. At a load of 9.000 N, the maximum displacement was 0,708 mm; at a load of 10.000 N, the maximum displacement was 0,787 mm; and at a load of 11.000 N, the maximum displacement was 0,867 mm.

1.3. Safety Factor

Table 5 Results of SS400 Material Safety Factor Analysis

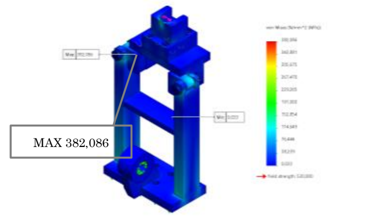
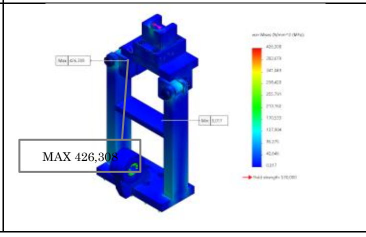
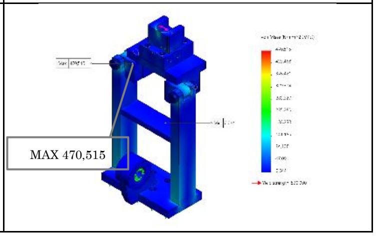
Load	9.000 N	10.000 N	11.000 N
Figure			

Based on table 5, the simulation shows that the minimum FoS value for all load variations is below the safety limit of 1. At a load of 9.000 N, the minimum FoS is 0,641, then decreases to 0,574 at 10.000 N, and 0,520 at 11.000 N. This indicates that the design is unsafe, so design improvements are necessary.

2. S45C

2.1. Stress

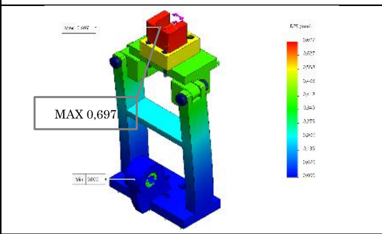
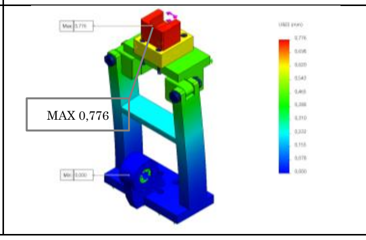
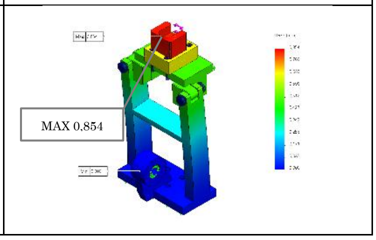
Table 6 Results of Von Mises Stress Analysis of S45C Material

Load	9.000 N	10.000 N	11.000 N
Figure			

Based on table 6, the results of the Von Mises Stress analysis with loads of 9.000 N, 10.000 N, and 11.000 N show that the maximum stress values are 382,086 MPa, 426,308 MPa, and 470,515 MPa, respectively. All stress values remain below the material's yield strength of 530 MPa, so it can be concluded that the component is capable of withstanding the loads without experiencing plastic deformation under all three loading conditions.

2.2. Displacement

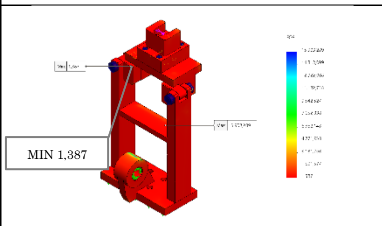
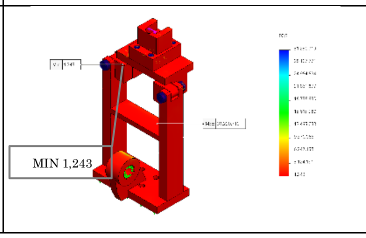
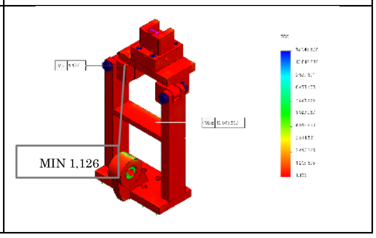
Table 7 Results of S45C Material Displacement Analysis

Load	9.000 N	10.000 N	11.000 N
Figure			

Based on table 7, the displacement analysis results show that the greater the load applied, the greater the maximum displacement of the component. At a load of 9.000 N, the maximum displacement was 0,697 mm; at a load of 10.000 N, the maximum displacement was 0,776 mm; and at a load of 11.000 N, the maximum displacement was 0,854 mm.

2.3. Safety Factor

Table 8 Results of S45C Material Safety Factor Analysis

Load	9.000 N	10.000 N	11.000 N
Figure			

Based on table 8 of the simulation results, the minimum FoS value at a load of 9.000 N is 1,387, then decreases to 1,243 at 10.000 N, and 1,126 at 11.000 N. This indicates that the design is safe because the resulting FoS value is greater than 1.

Conclusion

Based on the analysis results, it can be concluded that the pusher plate design with S45C material provides better results. The maximum Von Mises stress values obtained were 382,086 MPa, 426,308 MPa, and 470,515 MPa at loads of 9.000 N, 10.000 N, and 11.000 N, respectively. All stress values are still below the yield strength of the S45C material, which is 530 MPa. The maximum displacement values obtained are 0,697 mm at a load of 9.000 N, 0,776 mm at a load of 10.000 N, and 0,854 mm at a load of 11.000 N. The minimum safety factor (FoS) obtained is 1,387 at a load of 9.000 N, decreasing to 1,243 at 10.000 N, and 1,126 at 11.000 N. These results indicate that the pusher plate design is deemed safe, as all FoS values obtained remain above the minimum threshold, which is greater than 1.

Acknowledgment

The author would like to express sincere gratitude and hopes that this research will be beneficial for companies in the automotive industry and those involved in the manufacturing of shock absorbers, as well as for future studies related to testing equipment and the application of FEA–CAE methods.

References

[1] “Artikel AstraOtoshop,” Mengenal Komponen Shockbreaker | AstraOtoshop.com. Accessed: Jul. 31, 2025. [Online]. Available: https://astraotoshop.com/article/mengenal-komponen-shockbreaker

[2] “Artikel Suzuki Indonnesia,” Apa Itu Shock Absorber? Kenali Fungsinya! Accessed: Jul. 31, 2025. [Online]. Available: https://www.suzuki.co.id/tips-trik/apa-itu-shock-absorber-kenali-fungsinya

[3] “Artikel Hyundai,” Mengenal Fungsi Shockbreaker, Gejala Shockbreaker Rusak dan Perbaikan di Bengkel Shockbreaker Mobil. Accessed: Mar. 03, 2025. [Online]. Available: https://www.hyundai.com/id/en/hyundai-story/articles/mengenal-fungsi-shockbreaker,-gejala-shockbreaker-rusak-dan-perbaikan-di-bengkel-shockbreaker-mobil-0000000366

[4] “Artikel AstraOtoshop,” Kenali Fungsi Shock Absorber Mobil dan Motor | AstraOtoshop.com. Accessed: Apr. 14, 2025. [Online]. Available: https://astraotoshop.com/article/kenali-fungsi-shock-absorber-mobil-dan-motor

[5] M. R. A. Putra and H. L. Gutur, “Pemodelan dan Analisis Pengaruh Penggunaan Adaptive Shock Absorber Menggunakan Variabel Orifice Terhadap Karakteristik Gaya Redam dan Respon Dinamis Kendaraan,” JURNAL TEKNIK ITS, vol. Vol. 6, 2015.

[6] A. C. Ugural, Advanced strength and applied elasticity, 4th ed. Upper Saddle River, N.J: Prentice Hall PTR, 2003.

[7] R. G. Budynas, K. J. Nisbett, J. K. Nisbett, and J. E. Shigley, Shigley’s mechanical engineering design, 10. ed. in SI units. in Mcgraw-Hill series in mechanical engineering. New York, NY: McGraw-Hill Education, 2015.

[8] D. J. Dawe, Matrix and Finite Element Displacement Analysis of Structures. in Oxford Engineering Science. Birmingham: Clarendon Press, 1984.

[9] R. V. Petrova, Introduction to Static Analysis Using SolidWorks Simulation, 0 ed. CRC Press, 2014. doi: 10.1201/b17425.

[10] A. Diinil Mustaqiem and Nurato, “ANALISIS PERBANDINGAN FAKTOR KEAMANAN RANGKA SCOOTER MENGGUNAKAN PERANGKAT LUNAK SOLIDWORK 2015,” jtm, vol. 9, no. 3, p. 164, Nov. 2020, doi: 10.22441/jtm.v9i3.9567.

[11] D. L. Logan, A First Course in the Finite Element Method. CENGAGE Learning, 2015.

[12] K.-H. Chang, e-Design: computer-aided engineering design. Amsterdam ; New York: Academic Press, 2015.

[13] A. Saxena and B. Sahay, Eds., Computer Aided Engineering Design. in SpringerLink Bücher. Dordrecht: Springer Netherlands, 2005. doi: 10.1007/1-4020-3871-2.