FLYWHEEL: JURNAL TEKNIK MESIN UNTIRTA

Home page journal: http://jurnal.untirta.ac.id/index.php/jwl

Stress Analysis of Varian of Plastic Material for Fence Ornament Products Using Finite Element Method

Hendra[1,2a*], Husain Haafizh[1], Shofiatul Ula[1], Reski Septiana[1], Harly Demustila[3], Rispandi[4], Hernadewita[5], Hermiyetti[6], Gusri Akhyar Ibrahim[7]

[1]*Department of Mechanical Engineering, Universitas Sultan Ageng Tirtayasa, Jl. Jendral Sudirman KM 3 Cilegon 42435, Indonesia*
[2]*Laboratory of Design, Mechanical Materials, and Manufacturing, Center of Excellence Building, Universitas Sultan Ageng Tirtayasa, Jl. Jendral Sudirman KM 3 Cilegon, 42435, Indonesia*
[3]*Department of Chemical Engineering, Universitas Sultan Ageng Tirtayasa, Jl. Jendral Sudirman KM 3 Cilegon 42435, Indonesia*
[4]*Department of Mechanical Engineering Universitas Andalas, Kampus Limau Manis Padang, Indonesia*
[5]*Master of Industrial Engineering, Universitas Mercubuana, Kampus Meruya, Jakarta, Indonesia*
[6]*Faculty of Economics, Universitas Bakrie, Jl. HR. Rasuna Said, Jakarta, Indonesia*
[7]*Department of Mechanical Engineering, Universitas Lampung, Jl. Prof. Dr. Sumantri Brojonegoro No. 1 Bandar Lampung 35145*

**hendra@untirta.ac.id*

ARTICLE INFO

Received: 21/01/2025
Revision: 24/03/2025
Accepted: 27/03/2025
Available online: 30/04/2025

ABSTRACT

Plastic is the most widely used material in Indonesia. The most commonly processed types of plastic are PET (Polyethylene terephthalate), HDPE (High-density polyethylene), and PP (Polypropylene). With advancements in materials technology, plastic is increasingly in demand as a material for fence decoration due to its advantages such as lightness, flexibility, weather resistance, and economy. Using composite materials makes it possible to optimize both strength, flexibility, and resistance to various environmental factors that occur, so as to produce more durable fence ornaments. This study aims to obtain the maximum stress and displacement values of fence ornament products using PP (Polypropylene), PET (Polyethylene Terephthalate), and HDPE (High Density Polyethylene) materials and different variations. And also compare the simulation results of the mechanical test of the fence ornament with the mechanical test conducted in the material laboratory. The finite element method is a numerical method for solving engineering problems of mathematics and physics. This method can be applied to problems with complex geometries, loads, and material properties, which cannot be solved analytically. Simulation results on fence ornament products with PP, PET, and HDPE plastic material composition with a thickness of 10 mm produced a stress value of 61.84 MPa, to the largest 70.26 MPa, and a displacement value of 0.038 mm, to 0.074 mm. In the experimental results obtained a stress value of 14.572 MPa, in the best simulation results obtained a stress value of 63.12 MPa, with a displacement value of 0.038 mm. This can occur due to factors that affect the results of simulation and testing.

Keywords: Maximum Stress, Deformation, PET, PE, PP

1. INTRODUCTION

Plastic is the most widely used material in Indonesia. One way to overcome this problem is by recycling. The most commonly processed types of plastic are PET (Polyethylene terephthalate), HDPE (High density polyethylene), and PP (Polypropylene). Fence ornaments are now an important part of modern building design. With advances in materials technology, plastic is increasingly in demand as a material for fence ornaments due to its advantages such as lightness, flexibility, weather resistance, and economy [1-2].

To be able to produce plastic fence ornament products in accordance with the author's wishes, more innovation is needed. Using this plastic material allows us to optimize both strength, flexibility, and resistance to various environmental factors that occur, so as to produce more durable fence ornaments.

By using simulation, engineers can analyse various aspects of the product, such as stress, strength, durability, and deformation [3-10]. Static test simulation allows designers to obtain mechanical properties in the form of maximum stress and deflection.

This research aims to simulate the static properties in the maximum stress and displacement on fence ornaments that have been designed using a combination of plastic materials namely PET (Polyethylene Terephthalate), HDPE (High Density Polyethylene), and PP (Polypropylene) with adjusted composition. The results of this study are expected to provide information about the mechanical properties of plastic composites in fence ornaments.

Stress is the reaction that arises throughout the specimen in response to a given load. If the small cross-sections are summed up to reach the cross-section of the specimen, then the sum of the forces per unit area that appear inside the material must be equal to the load outside [11].

The Finite Element Method (FEM), also known as Finite Element Analysis (FEA), is a numerical computational procedure that can be used to solve engineering problems such as structural stress analysis, heat transfer, electromagnetism, and fluid flow [3-10].

Deflection is the change in shape of a beam in the y-direction due to vertical loading applied to a material bar. Deformation in a beam can be described based on deflection according to the properties of the material, from its position before loading [11].

2. RESEARCH METHODOLOGY

In this study, simulations were carried out on fence ornament products with mixed materials in the form of PP, PET, HDPE plastics using the Solidwork application. The simulation carried out is a static simulation to find the maximum stress value and displacement that occurs. The research flow can be seen in Figure 1.

2.1 Experiment method

In this study using the experimental method, the experimental method carried out is to simulate fence ornament products with mixed materials in the form of 50% PP plastic, 30% PET, and 20% HDPE. The simulation carried out is a static simulation to find the value of the stress and displacement that occurs.

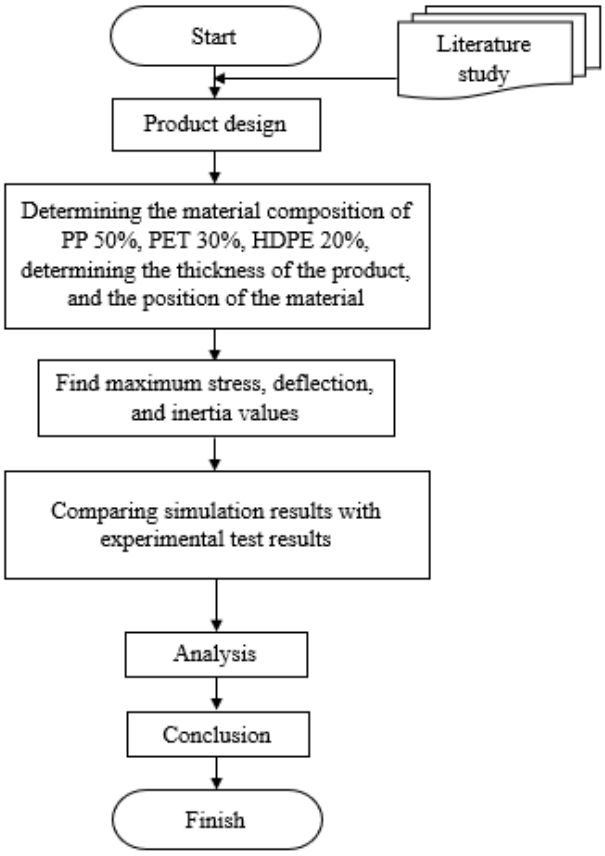


Figure 1. Research flow chart

2.2 Fence ornament design

A mold is a supporting tool that helps form the desired object. In this research there is a design of fence ornaments that will be made, the following design of fence ornaments will be simulated using finite element method (FEM).

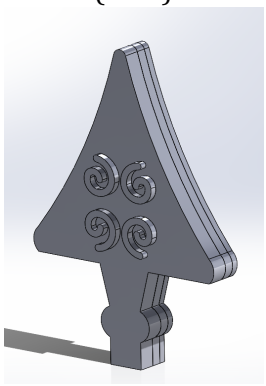


Figure 2. Fence ornament design

2.3 Simulation testing phases

To get the maximum stress and displacement results on the fence ornament design, there are stages that will be carried out:

1. Create a fence ornament design before conducting stress analysis on the fence ornament product. Figure 2. shows the fence ornament product used in this study.
2. Applying materials to the design of the fence ornament. After the model of the fence ornament product is completed, the next step is to select the appropriate material and apply it to the model. Figure 3. shows the selection and application of materials to the fence ornament products used in this research.

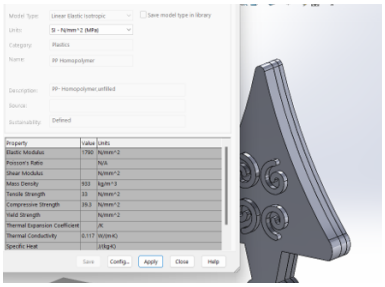


Figure 3. Application of materials on fence ornaments

3. Entering the force value for static simulation. After selecting and applying materials in the form of PP, PET, and HDPE to the fence ornament design, the next step is to enter the force value that will be used to perform a simulation of 160 N. In Figure 4. are the points that will be applied with the assumption that the fence ornament is nailed to the board.

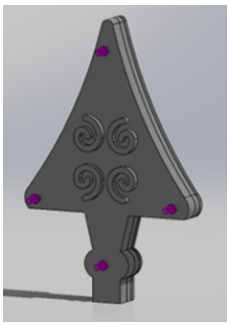


Figure 4. Application of style value to fence ornament design

4. Set up the mesh before the fence ornament design is simulated. Before performing static simulation to find the stress value on the fence ornament product, it is necessary to set up the mesh on the model. Figure 5. shows the mesh setup process used in this study.

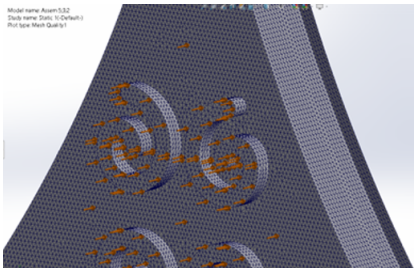


Figure 5. Set mesh on fence ornament design

5. Running a simulation of the fence ornament design. After all of the above stages are ready, the simulation on the fence ornament product model is then run to get the analysis results.

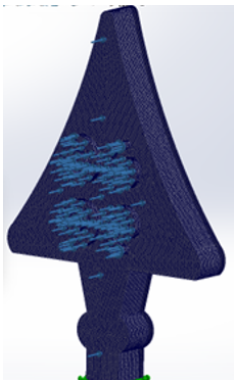


Figure 6. Running simulation on fence ornament design

6. After all is done, the results of the simulation on the product design of the fence ornament can be seen in Figure 7.

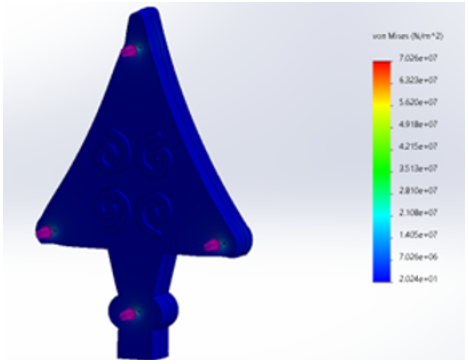


Figure 7. Simulation results on fence ornament design

3. RESULTS AND DISCUSSION

Simulations carried out on fence ornament products were carried out with variations in the composition of 50% PP, 30% PET, 20% HDPE, and thickness variations of 5 mm, 10 mm, 15 mm. The following are the results of the simulation that has been carried out.

3.1 Simulation results 5 mm thickness

In Figure 8. shows the simulation results with a material composition of HDPE 1 mm, PP 2.5 mm and PET 1.5 mm, with a thickness of 5 mm, the maximum stress generated is 62.15 MPa. Figure 9 shows the simulation results in the area of potential failure with a material composition of HDPE 1 mm, PP 2.5 mm and PET 1.5 mm, with a thickness of 5 mm. Figure 10 shows the displacement generated in the design of the fence ornament with the maximum value obtained of 0.072 mm.

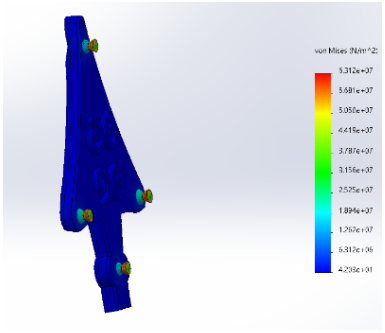
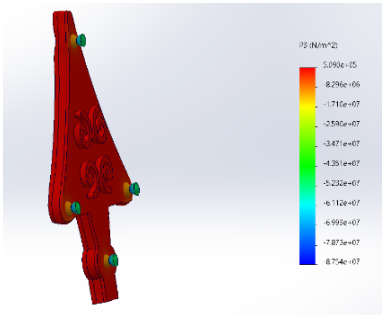
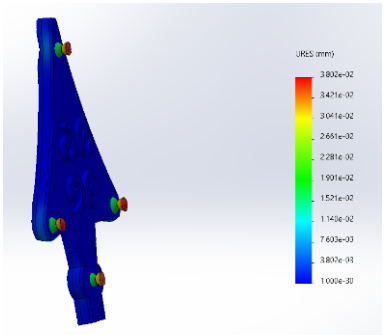


Figure 8. Maximum stress 5 mm



Gambar 9. 3rd Principal 5 mm



Gambar 10. Displacement 5mm

3.2 Simulation results 10 mm thickness

In Figure 11. shows the simulation results with a material composition of PET 3, PP 5 mm, and HDPE 2 mm, with a thickness of 10 mm, the maximum stress generated is 62.43 MPa. Figure 12 shows the simulation results on the potential failure area with a material composition of PET 3, PP 5 mm, and HDPE 2 mm, with a thickness of 10 mm. Figure 13. shows the displacement generated in the design of the fence ornament with the maximum value obtained of 0.038 mm.

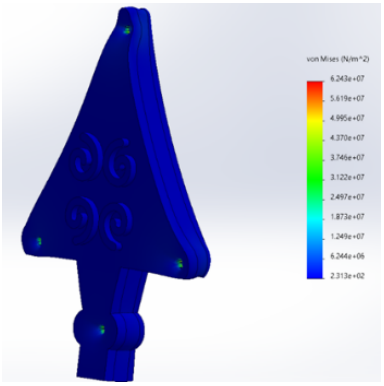


Figure 11. Maximum stress 10 mm

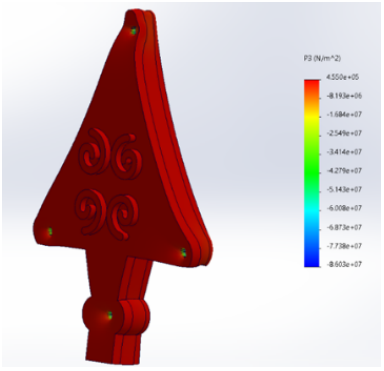


Figure 12. 3rd Principal 10 mm

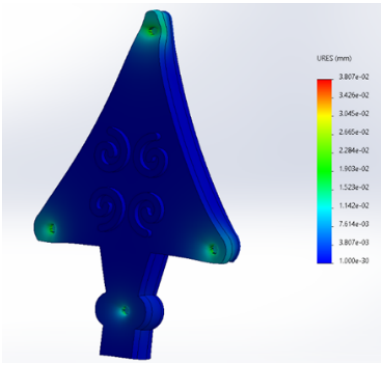


Figure 13. Displacement 10 mm

3.3 Simulation results 15 mm thickness

Figure 14 shows the simulation results with a material composition of 4.5 mm PET, 3 mm HDPE, and 7.5 mm PP, with a thickness of 15 mm, the maximum stress generated is 62.29 MPa. Figure 15 shows the simulation results on the potential failure area with a material composition of 4.5 mm PET, 3 mm HDPE, and 7.5 mm PP, with a thickness of 15

mm. Figure 16 shows the displacement generated in the design of the fence ornament with the maximum value obtained of 0.041 mm.

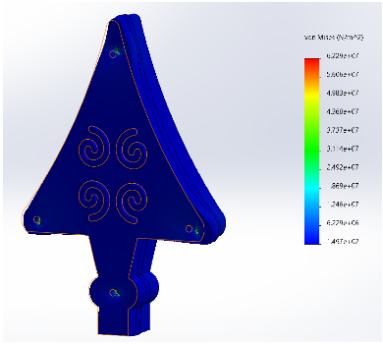
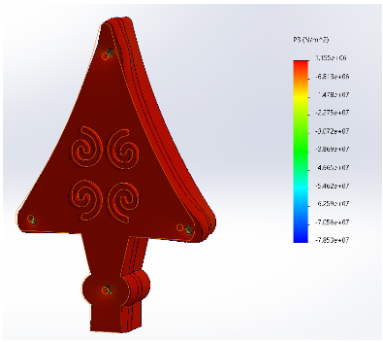
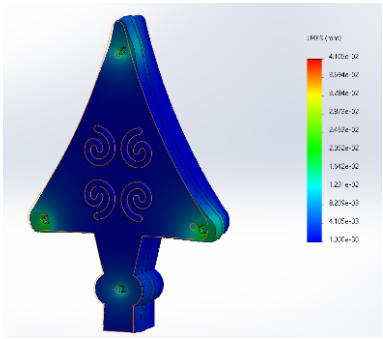


Figure 14. Maximum stress 15 mm



Gambar 15. 3rd Principal 15 mm



Gambar 16. *Displacement* 15 mm

3.4 Discussion of simulation

The best results at a thickness of 5 mm when considering both variable results, namely maximum stress and displacement with a material composition of PP in layer 2, PET in layer 1 and HDPE in layer 3, with a stress value of 63.12 MPa and a displacement value of 0.038 mm.

The best results at a thickness of 10 mm when considering both variable results, namely maximum stress and displacement with a material composition of PP in layer 2, PET in layer 1 and HDPE in layer 3, with a stress value of 62.43 MPa and a displacement value of 0.038 mm.

The best results at a thickness of 15 mm when considering both variable results, namely maximum stress and displacement, are in material composition number 16 with PP material composition in layer 3, PET in layer 1 and HDPE in layer 2, with a stress value of 62.29 MPa and a displacement value of 0.041 mm.

3.5 Comparison of simulation results with experimental results experiment

Based on the test results that have been carried out, the second test result is the best for comparison with the simulation results, the experimental results obtained a stress value of 14.572 MPa, in the best simulation results obtained a stress value of 63.12 MPa, with a displacement value of 0.038 mm. The comparison between the experimental results and the test results is clearly very different, this can occur due to several factors that cause the stress value to be far adrift. First, in the experimental sample, the raw materials used are mixed into 1 so that they are considered to be evenly mixed, while in the simulation, the fence ornament product is divided into three layers, which in each layer uses different raw materials. Second, when taking samples that can be influenced by various factors such as the quality of the materials used, environmental conditions when conducting tests. Third, the behavior of the material, if you look at the sample that will be carried out tensile testing, there are several cavities on the outside and inside, this will affect the test results.

4. CONCLUSION

After conducting research in the form of simulations and experiments on fence ornaments made from composites in the form of PP, PET, and HDPE, conclusions can be drawn, the following are the conclusions of the research results:

1. Simulation results on fence ornament products with PP, PET, and HDPE plastic material compositions with a thickness of 5 mm and different material compositions obtained maximum stress values between 60.53 MPa to 65.48 MPa and the resulting displacement value of 0.038 mm, to 0.072 mm, the results of this displacement indicate that the material remains stable with little deformation. In fence ornament

products with a thickness of 10 mm, the stress value is 61.84 MPa, up to the largest 70.26 MPa, and the displacement value is 0.038 mm, up to 0.074 mm. In fence ornament products with a thickness of 15 mm, the stress value is 61.76 MPa, up to 65.14 MPa, and the displacement value is 0.041 mm, up to 0.090 mm.

2. In the experimental results obtained a stress value of 14.572 MPa, in the best simulation results obtained a stress value of 63.12 MPa, with a displacement value of 0.038 mm. The comparison between the experimental results and the test results is clearly very different, this can occur due to several factors that cause the stress value in the fence ornament product to be different. First, due to the mixture of different raw materials in the experimental product, the three main ingredients are mixed into one, in the simulation the three materials are made into three layers with different materials, second, the quality factor of the materials used, third, the material behavior factor.

REFERENCE

1. Astuti AD, Wahyudi J, Ernawati J, Aini AQ. Kajian Pendiri Usaha Biji Plastik di Kabupaten Pati, Jawa Tengah. Jurnal Litbang: Media Informasi Penelitian, Pengembangan dan IPTEK, 2020; 16(2): 95-112.

2. Khairiyati L. Pengolahan Limbah Plastik Untuk Menjaga Kelestarian Lingkungan dan Meningkatkan Perekonomian, 1 ed., Nagari Koto Baru: INSAN CENDEKIA MANDIRI, 2021.

3. Indriani A, Hendra, Suhartini Y, Aswata, Rispandi, Hernadewita, Tanjung A. Stress Analysis on the Frame Holder of Generator Translation Motion on Horizontal Direction for Sea Wave Power Plant Using Finite Element Method (FEM). International Conference on Advanced Mechanical and Industrial Engineering, Cilegon, IOP Conf. Series: Materials Science and Engineering 909-012033. 2020. doi:10.1088/1757- 899X/909/1/012033.

4. Suhartini Y, Indriani A, Hendra, Rispandi, Hernadewita. Finite Element Method for Stress Analysis in the Frame Holder of Generator Translation and Rotation Motion on Vertical Direction Mechanism for Sea Wave Power Plant", ICOMERA, Malang, IOP Conf. Series: Materials Science and Engineering 1034-012008. 2020. doi:10.1088/1757-899X/1034/1/012008.

5. Hendra, Syukriah, Silalahi M, Hernadewita, Hermiyetti. Finite Element Method Analysis for Manufacturing Design Drum Dryer of Rotary Dryer Machine. ICIME, Medan, IOP Conference Series: Materials Science and Engineering 505 Issue: 1. 2019.

6. Hendra, Qiram SA, Susilo S, Nugraha K, Hernadewita, Hardian F. Analisa Tegangan Pada Desain Empat Mata Potong Untuk Mesin Pencacah Plastik Menggunakan Metode Elemen Hingga. Jurnal Teknik Mesin. 2023; 16(2): 118-126.

7. Hendra, Putra DM, Dwinanto, Hernadewita, Notonegoro HA, Analisa Tegangan Maksimum Pada Desain Struktur Mesin Pengering Bekatul dengan Tebal 0.7 mm dan Variasi Material Menggunakan Metode Elemen Hingga. Jurnal Teknik Mesin. 17(1): 14-19.

8. Maulana Ari, Yusup IM, Syafrizal F, Adha R, Satria D, Hendra. Thermal Stress Analysis for Barrel and Screw Component in The Plastic Waste Machine Using Finite Element Method. METAL: Jurnal Sistem Mekanik dan Termal. 2023; 7(2).

9. Hendra, Indriani A, Hernadewita, Mardian A, Kholik N, Rispandi. Analisis tegangan pada struktur mesin pencacah plastik menggunakanmetode elemen hingga (MEH) dan uji kerja mesin pencacah plastic. METAL: Jurnal Sistem Mekanik dan Termal: 2021; 5(1).

10. Kurniawan RNA, Romahadi D, Fitri M. Implementasi Metode ELemen Hingga Menggunakan Solidworks Untuk Mengoptimalkan Desain Pelek Depan Sepeda Motor Tipe Casting Wheel. Jurnal Teknik Mesin. 2023; 12(2).

11. Dearni R, Suryanita R, Ismediyanto I. Analisis Sifat Mekanik Bata Ringan Cellular Lightweight Concrete Menggunakan Program LUSAS V17. Jurnal Sainstek. 2019; 7(2): 73-79.