International Journal of Research In Vocational Studies (IJRVOCAS)
Vol. 2 No. 4 (2023): IJRVOCAS – Special Issues – INCOSTIG – PP. 105~115
Print ISSN 2777-0168| Online ISSN 2777-0141| DOI prefix: 10.53893
https://journal.gpp.or.id/index.php/ijrvocas/index

Structural Analysis of Parking Barrier Made of Polymeric Resin Using Ansys Software

Zoanri Purba[1], Maraghi Muttaqin[2], Bustami Syam[*,2], Ade Irwan[2], Fakhrur Rozy[2]
[1] Undergraduate Student, Department of Mechanical Engineering, University of Sumatera Utara (USU), Indonesia
[2] Impact and Fracture Research Center, Dept. of Mechanical Engineering, Faculty of Engineering, University of Sumatera Utara (USU), Indonesia

Keywords:

Barrier gate
Software ansys
Fibreglass
Polymeric foam

ABSTRACT

A barrier gate is a stopping tool that has a security function in a place such as an office or agency entrance. This study aims to examine the response of the beam structure of the polymer composite material due to the support test. The parking portal material in this research is BQTN 157 EX resin, MEKPO catalyst, fiberglass. There are two parking portal compositions with impact test heights of 2 m and 4 m which will be simulated with ansys software. Overall, from two types of compositions, namely polymeric reinforced fiberglass composition A and composition B by looking at the results of the equivalent stress distribution (von-mises), it is concluded that polymeric reinforced fiberglass composition A has a higher maximum stress than polymeric reinforced fiberglass composition. reinforced with fiberglass composition B. From the comparison between simulation and experiment, it is known that the composition of rod A has a different damage position between simulation and experiment. While the composition of rod B in the simulation of the same damage position occurs, namely in the crossbar area connected to the transmission shaft and the experimental also has a damage position in the crossbar area connected to the transmission shaft.

Corresponding Author:

Bustami Syam,
Department of Mechanical Engineering,
Faculty of Engineering,
Almamater Road, Padang Bulan, Medan, North Sumatera, Indonesia.
Email: bustami@usu.ac.id

1. INTRODUCTION

Barrier gate is a stop device that has a security function in a place such as an office or agency entrance [1]-[2]. With so many types of bars, this research focuses on the study of alternative materials for the manufacture of bars using BQTN 157 EX fiberglass resin with MEKPO catalyst [3]-[5]. The material used, chopped strand mat (CSM) glass fiber was chopped and varied with 2 (two) different amounts of composition [6]-[10]. Analysis of the force applied to the crossbar (portal) of polymer material using BTQN 157 EX fiberglass resin with MEKPO catalyst in the form of an impact test in experimental testing to determine the position of damage that occurs in the crossbar and compare it with the position of the damage in the simulation results of ANSYS [11]-[13]. Before running the simulation, the material characteristics data will be used to be entered into the simulation analysis. The data was obtained from testing using a static tensile test. After getting the simulation results then compare the maximum equivalent stress value that can be accepted by the polymer composite portal bars of composition A and composition B.

The research objectives in this paper are:

1. To get the maximum stress distribution on the composite beam using Ansys simulation
2. Comparing the equivalent stress on the composite crossbar model of the two compositions using ansys simulation.
3. Comparing the position of the damage that occurs in the simulation of ansys with the position of the damage that occurs in the experimental test of free-fall impact.

Composite material is a combination of two or more materials that have several properties that are not possible for each of its components. Unsaturated polyester resin is a condensate polymer formed based on the reaction between polyol which is an organic combination with multiple alcohols or hydroxy functional groups and polycarboxylic containing composite double bonds at room temperature and atmospheric pressure conditions. The type of catalyst used is methyl ethyl ketone peroxide (MEKPO). Fiberglass is the same generic name as carbon fiber or steel. A variety of different chemical compositions. Usually, fiberglass has a general composition of 50%-60% SiO_2 and other alloys such as Al, Ca, Mg, Na, and others. Fiberglass usually has a low level of stiffness (about 70GPa) but has good dimensional stability. Fiberglass also has properties that are resistant to heat and cold as well as corrosion resistance [14]-[15].

1. E-Glass Fiber-glass: this glass fiber is the most popular glass fiber with the cheapest price. The letter "E" denotes the meaning of "electricity" which means glass fiber E-glass is a very good insulator. The composition of E-Glass is around 52-56% SiO_2, 12-16% Al_2O_3, 16-25% CaO, and 813% B_2O.
2. S-Glass: this fiber has stronger strength than E-Glass fiber. The letter “S” denotes strength. High-strength glass is commonly known as S-type glass in the United States, R glass in Europe, and T glass in Japan. S-Glass is commonly used in military and aerospace applications. S-Glass consists of Silica (SiO_2), Magnesia (MgO), and Alumina (Al_2O_3).
3. C-Glass: This glass fiber is a corrosion-resistant glass fiber and a chemical-resistant glass fiber. To protect against air erosion, a waterproof coating such as a silane compound is coated on the fibers during manufacture. The addition of resin during the composite forming process provides additional protection. These fibers are commonly used in the manufacture of storage tanks, pipes, and other chemical-resistant equipment.

For the modulus of elasticity, the density of fiberglass can be seen in table 1.

Tabel 1 Characteristic of *fibreglass* [16]

Material	Modulus Elastisitas (GPa)	Density (gr/cm^3)
Fibreglass	7.25	2.58
Carbon Fibres (Standard Modulus)	230	1.8
Carbon Fibres (Intermediate Modulus)	285	1.8
Carbon Fibres (High Modulus)	400	1.8
Epoxy Resin	2.4	1.14

2. RESEARCH METHOD

In this study, the composite beam model is made of rectangular hollow beams. The portal bars are generally made of iron with a circle model and aluminum with a hexagonal model, but for this research, we designed to manufacture portal bars funderstandsfiber-reinforced polymeric resin. For two-dimensional sketches and dimensions of composite bars with Solidworks, see Figure 1.

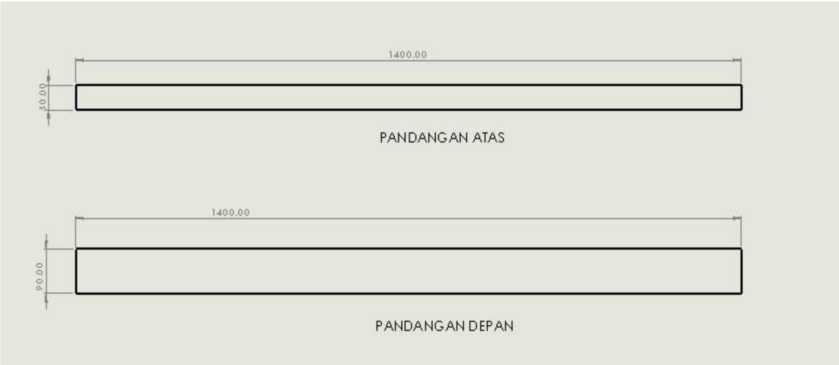


Fig 1. Composite Bar

The material developed in this research is fiberglass reinforced polymeric. The composition of the forming material (resin, fiber and catalyst), has been determined from previous research, namely with polymeric material of composition A and polymeric composition B. The composition of the composite material is shown in the material data used for engineering data in ansys as shown in table 2 [17].

Tabel 2. Tensile test result material data

Material	Density (kg/m^3)	Modulus Young (MPa)	Poisson’s Ratio
Composition A	0.126	274.686	0.040
Composition B	0.125	305.737	0.269

In this research, a new material is used, namely polymeric with fiberglass that has not been registered on the ansys workbench so we need to enter the material data into the engineering data. The steps for entering material data into engineering data are as follows:

1. Select the engineering data menu, then the "outline of schematic A2 Engineering Data" will appear.
2. Then click on the menu that say s "click here to add new material" and enter the name of the new material data that we want to simulate. Here the author enters the name of the new material “Polymeric reinforced fiberglass”.
3. Then on the toolbox menu select physical properties → density. Then enter the density value in the value column and specify the units used in the units column.
4. Then in the toolbox menu select linear elastic → isotropic elasticity. In the derive from line in the value column select "Young's Modulus and Poisson's Ratio". Then enter the value of Young's modulus in the value column and specify the units used in the units column. Next, enter the Poisson's ratio value in the value column.
5. Repeat these steps to enter other material data.

In step number three, the density value for polymeric material reinforced with fiberglass composition A in the tensile test is 0.126 kg/m3 and for polymeric reinforced fiberglass composition B is 0.125 kg/m3. Then the value of Young's modulus is taken from Table 3.4 tensile test for polymeric reinforced fiberglass composition A of 274,686 MPa and for polymeric reinforced fiberglass composition B of 305.737 MPa. Then the value of Poisson's ratio is taken from Table 3.4 for polymeric reinforced fiberglass composition A of 0.040 and for polymeric reinforced fiberglass composition B of 0.269. After all the steps are done, the engineering data filling process is complete. Then click return to project, it will return to the project schematic menu and a checklist will appear on the engineering data. The steps for the process of entering material data into engineering data can be seen in Figure 3.6. The picture shows a menu that appears at each step and is equipped with a sequence that has been arranged according to the steps above.

After completing the engineering data step, the next step is to do geometry modeling. The composite bar model that we have created in the solidworks software, we import into ansys. This step is done by right clicking on "geometry" then selecting "import geometry" then "browse" as seen in figure 2.

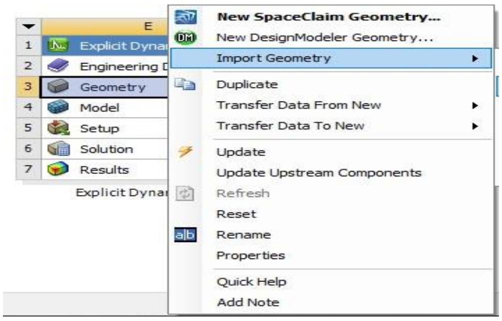
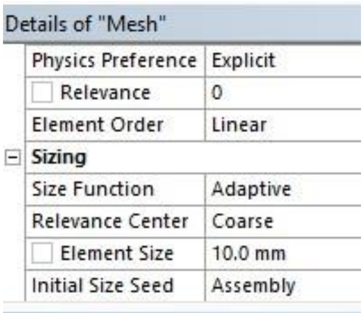


Figure 2. *Geometry*

Mesh is the division of objects into smaller parts. The smaller the meshing that is made, the more accurate the calculation results will be but it requires greater computing power as well. In this study the order mesh element used is linear and the mesh size used is 10 mm which can be seen in Figure 3. The meshing results can be seen in Figure 3.



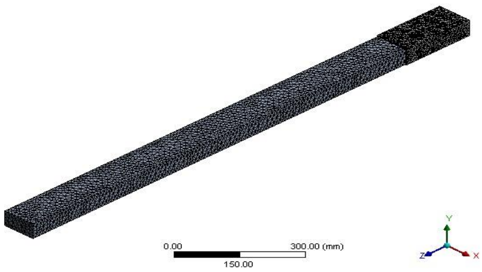


Figure 3. *Detail of meshing* Figure 3. *mesh*

To determine the analysis settings, in the analysis settings menu, enter the end time data, click on the analysis settings, the "details of analysis settings" menu will appear in the step controls → end time, which can be seen in Figure 4 below.

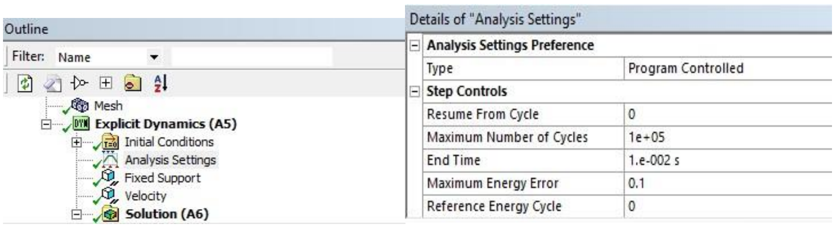


Figure 4. *Analysis settings*

Fix supports are fixed supports and are also used as boundary conditions in finite element analysis. In this study, the fix support is located on the transmission shaft as a barrier or as an area that is in physical contact with the bar. The fix support area can be seen in Figure 5.

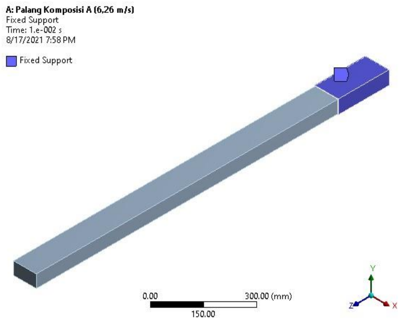


Figure 5. *fix support*

Determine the velocity to be simulated with the impact test by going to the project → explicit dynamics → insert → velocity, then the “details of velocity” menu will appear, in the scope → Geometry select the body that will provide the impact test load on the bars then definition → define by select components it will appear in the lower column of the coordinate system, enter the velocity value according to the direction of the Y-axis of the bar, as in Figure 6 and Figure 7.

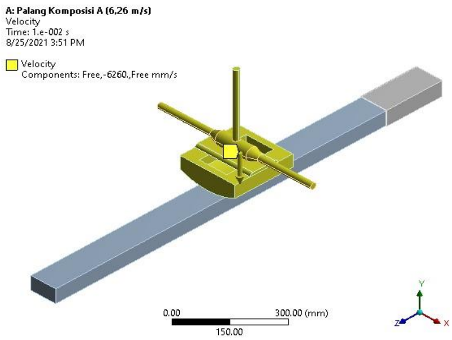


Figure 6. Velocity

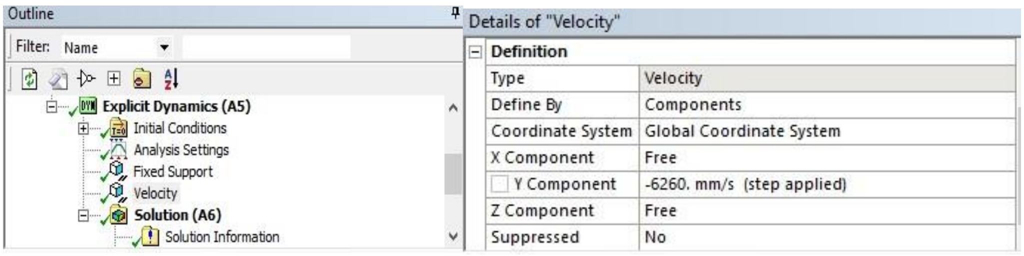


Figure 7. *Detail of velocity*

Free fall motion is the falling motion of an object that has no initial velocity (v0 = 0). As long as the object is falling, the air resistance is negligible, so its acceleration is constant. In addition, the time it takes an object to fall does not depend on its mass, but on its height. The acceleration experienced by any free-falling object is always the same, which is the same as the acceleration due to gravity (a = g) (g = 9.8 m/s2 and is often rounded to 10 m/s2). The following is the formula for determining the velocity of a free-falling object.

$$v = \sqrt{2.g.h} \quad (1)$$

Solution is a numerical analysis process carried out by ansys to obtain the desired parameters. In this study, the parameters to be obtained are total deformation, equivalent stress, x, y, and z axis stress due to stresses that occur on the bars with fiberglass composite material. The steps taken to get the results from the solution are right-clicking on the solution section and then selecting insert → stress → Equivalent von-Mises as shown in Figure 8.

https://doi.org/10.53893/ijrvocas.v2i4.175

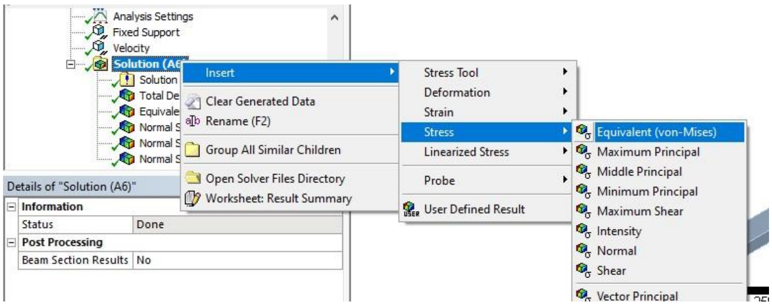
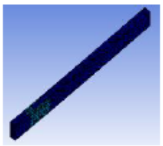


Figure 8. *Solution*

3. RESULTS AND ANALYSIS

The simulation results of fiberglass reinforced polymeric material bars with the impact test of composition A 2 m and B 2 m can be seen in table 3 and for composition A 4 m and B 4 m can be seen in table 4.

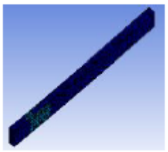
Table 3. Bar composition A and B height impact test 2 m

Pembebanan	Material	Deformation Total [mm]	Equivalent Stress [MPa]	Normal Stress X [MPa]	Normal Stress Y [MPa]	Normal Stress Z [MPa]
 *Uji impak ketinggian 2 m*	Polymeric diperkuat *fibreglass* komposisi A	80,835	2,221	0,332	0,174	2,031
	Polymeric diperkuat *fibreglass* komposisi B	69,739	1,818	0,284	0,122	1,722

In table 3 it can be concluded that the smallest normal stress on the X axis occurs in polymeric reinforced fiberglass composition B and the largest is experienced by polymeric reinforced fiberglass composition A. For the smallest normal stress on the Y axis occurs in polymeric reinforced fiberglass composition B and the largest is experienced by polymeric Reinforced fiberglass of composition A. For normal stress on the Z axis the smallest occurred in polymeric reinforced fiberglass of composition B and the largest was experienced by polymeric reinforced fiberglass of composition A.

By comparing the value of the equivalent von Mises stress of the two types of materials, it is found that the equivalent stress of composition A material is greater than that of composition B. So, the material is better to produce polymeric reinforced fiberglass of composition A. Also seen from the total deformation value of the two materials, it can be concluded that the value of the total deformation of the fiberglass reinforced polymeric material of composition B is smaller than that of composition A.

Table 4. Composite bar A and B impact test height 4m

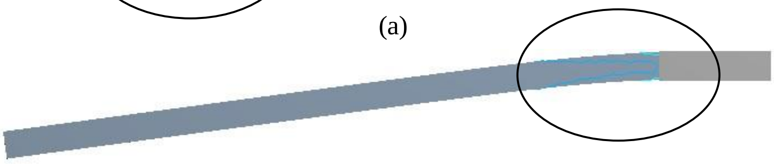
Pembebanan	Material	Deformation Total [mm]	Equivalent Stress	Normal Stress X	Normal Stress Y	Normal Stress Z
 *Uji impak ketinggian 4 m*	Polymeric diperkuat *fibreglass* komposisi A	145,03	3,50	0,524	0,272	3,171
	Polymeric diperkuat *fibreglass* komposisi B	101,85	3,106	0,535	0,247	2,966

In table 4 it can be concluded that the smallest normal stress on the X axis occurs in polymeric reinforced fiberglass composition A and the largest is experienced by polymeric reinforced fiberglass composition B. For the smallest normal stress on the Y axis occurs in polymeric reinforced fiberglass composition B and the largest is experienced by polymeric Reinforced fiberglass of composition A. For normal stress on the Z axis the smallest occurred in polymeric reinforced fiberglass of composition B and the largest was experienced by polymeric reinforced fiberglass of composition A.

By comparing the value of the equivalent von Mises stress of the two types of materials, it is found that the equivalent stress of composition A material is greater than that of composition B. So the material is better to produce polymeric reinforced fiberglass of composition A. Also seen from the total deformation value of the two materials, it can be concluded that the value of The total deformation of the fiberglass reinforced polymeric material of composition B is smaller than that of composition A.

In this study, a comparison of the results obtained in the impact simulation with the experimental free fall impact was carried out in order to determine the difference between the simulation damage position results and the experimental damage position results so that the results obtained were more accurate. The position of the simulated damage to the composite beam of composition A can be seen in Figure 9. It can be seen in the circled section of the beam and it is concluded that the position of the damage will occur in the area of the beam connected to the transmission shaft.

(a)

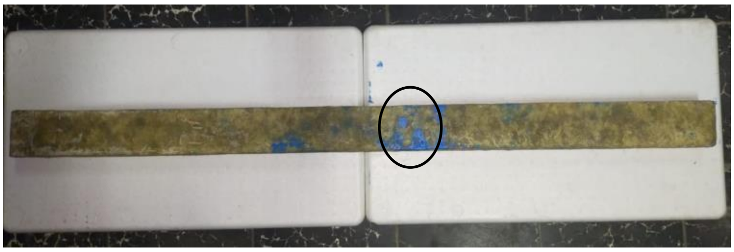


(b)

Figure 9. Simulation result of composition bar A (a) Left side (b) Right side (c) Top side (d) Bottom side

The position of the damage from the experimental test results on the composite beam of composition A can be seen in Figure 11. Based on the position of the damage shown in Figure 10, it can be concluded that the results of the simulation and experimental tests have different damage positions. In the simulation results, the position of the damage will occur in the crossbar area which is connected to the transmission shaft, while the experimental test results have a position of damage that will occur on the upper side and the right side in the middle of the bar.

https://doi.org/10.53893/ijrvocas.v2i4.175

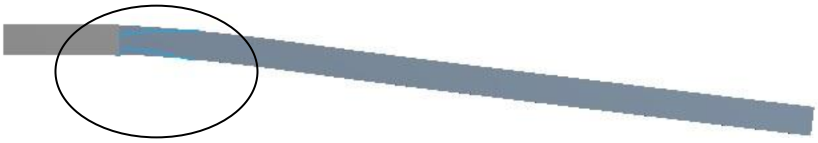


(a)

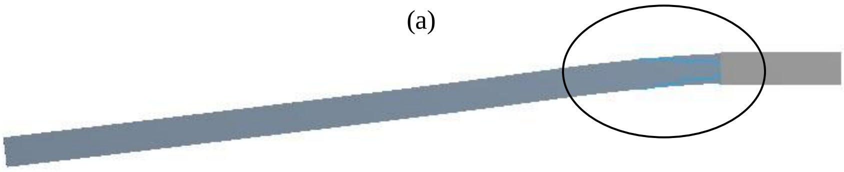
(b) (c)

Figure 10 Experimental test results of composition bars A (a) Condition of bars after testing (b) Top side (c) Right side

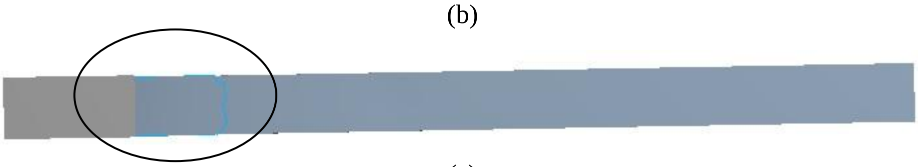
The position of the damage from the simulation results on the composite beam composition B can be seen in Figure 11.



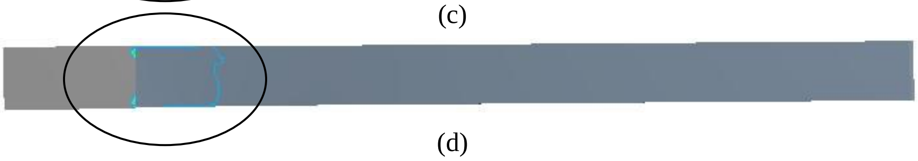
(a)



(b)



(c)

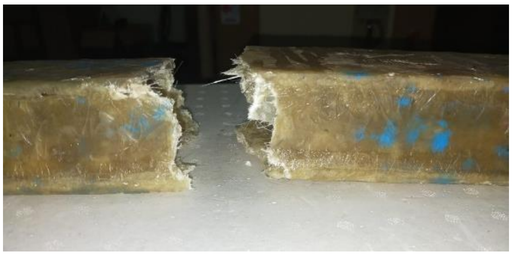


(d)

Figure 11. Simulation results of composition bars B (a) Left side (b) Right side (c) Top side (d) Bottom side

From Figure 11, it can be seen that the cross section is circled, and it can be concluded that the position of the damage will occur in the area of the beam connected to the transmission shaft. The position of the damage from the experimental test results on the composite beam composition B can be seen in Figure 12. From Figure 13 it can be seen, and the results show that the beam is in a position of damage that will occur in the crossbar area connected to the transmission shaft.

(a)



(b) (c)

(c) (d)

Figure 12 Experimental test results of bars composition B (a) Condition of bars after testing (b) Right side (c) Left side (d) Top side (e) Bottom side

From the results of the comparison between simulation and experiment, it is known that the composition A bar has a different damage position between the simulation and the experiment. While in the composition B bars in the simulation, the damage position occurs in the crossbar area connected to the transmission shaft and the experimental also has the same damage position, namely in the crossbar area connected to the transmission shaft.

4. CONCLUSION

After all the research was carried out and analyzed all the results obtained, the following conclusions were obtained:

1. From the composite bar design with a length of 1400 mm, a width of 50 mm, and a height of 90 mm, the total deformation, 1, x, y, z is obtained and after the impact loading is given as follows:

a. The total deformation that can be accepted by the bars with polymeric resin material reinforced with fiberglass of composition A impact test height of 2 m is 80,835 mm, σ_1 2,221 MPa, σ_x 0,332 MPa, σ_y 0,174 MPa, σ_z 2,031 MPa.

b. The total deformation that can be accepted by the bars with polymeric resin material reinforced with fiberglass of composition A impact test height of 4 m 145,03 mm, σ_1 3,50 MPa, σ_x 0,524 MPa, σ_y 0,272 MPa, σ_z 3,171 MPa.

https://doi.org/10.53893/ijrvocas.v2i4.175

c. The total deformation that can be accepted by the bars with polymeric resin material reinforced with fiberglass of composition B impact test height of 2 m 69,739 mm, σ_1 1,818 MPa, σ_x 0,284 MPa, σ_y 0,122 MPa, σ_z 1,722 MPa.

d. The total deformation that can be accepted by the bars with polymeric resin material reinforced with fiberglass of composition B impact test height of 4 m 101,85 mm, σ_1 3,106 MPa, σ_x 0,535 MPa, σ_y 0,247 MPa, σ_z 2,966 MPa.

2. Overall, from the two types of composition, namely polymeric reinforced fiberglass composition A and composition B by looking at the results of the equivalent stress distribution (von-mises), it was concluded that the polymeric reinforced fiberglass composition A had a higher maximum stress than the polymeric reinforced fiberglass composition B. for each impact test with a height of 2 m and 4 m.

3. From the comparison between simulation and experimental, it is known that the composition A bar has a different damage position between the simulation and the experimental. While in the composition B bars in the simulation the same damage position occurs, namely in the crossbar area connected to the transmission shaft and experimental also has a damage position in the crossbar area connected to the transmission shaft. Caused by several test parameters such as temperature, material defects, inhomogeneous density, calibration of test equipment and so on are factors that cause differences in simulation results using ansys software with experimental results of free fall impact tests.

ACKNOWLEDGEMENTS

The authors would like to thank the research facilities at the Impact Fracture Research Center (IFRC) mechanical engineering department.

REFERENCES

[1] Ahmad, Lutfi. 2007. "Perancangan Portal (Palang Pintu) Masuk Kampus Universitas Muhammadiyah Malang". [Skripsi] Malang: Universitas Muhammadiyah Malang.

[2] Puji, H. 2013. "Desain Barrier Gate Dan Sistem Kontrolnya Untuk Sistem Parkir Ramah Lingkungan Berbasis Programmable Logic Control (PLC) Dan Scada". [Skripsi] Semarang: Universitas Diponegoro.

[3] Hestiawan, H., & Jamasri, K. 2017. "Pengaruh Penambahan Katalis Terhadap Sifat Mekanis Resin Poliester Tak Jenuh". Teknosia, 3(1), 1-7.

[4] Rudydwi. 2010. "Mengetahui Sifat Mekanik Material Dengan Uji Tarik". https://rudydwi.wordpress.com/2010, diakses pada 25 April 2020 pukul 16.26.

[5] Kristianto, Laurensius, 2018. Pengaruh Persentase Serat Fiberglass Terhadap Kekuatan Tarik Komposit Matriks Polimer Polyester. [Skripsi], Yogyakarta: Universitas Sanata Dharma Yogyakarta.

[6] Kristianto, Laurensius, 2018. Pengaruh Persentase Serat Fiberglass Terhadap Kekuatan Tarik Komposit Matriks Polimer Polyester. [Skripsi], Yogyakarta: Universitas Sanata Dharma Yogyakarta.

[7] Wennie Fransisca, Ismet Danial Nasution. 2015. Pengaruh Penambahan Serat Kaca Dan Serat Poliester Terhadap Kekuatan Impak Bahan Basis Gigitiruan Resin Akrilik Polimerisasi Panas. Vol 2. Nomor 1.

[8] Moch. Razan Dwi, Djoko W. Karmiadji, Eko Prasetyo. 2020. Komposit Sandwich Hybrid Kombinasi Serat Tandan Kosong Kelapa Sawit Dan Serat Kaca Dengan Matrix Polyurethane, Vol.5.No.2.

[9] Zulhanif, Investigasi Sifat Mekanik Material Komposit Yang Terbuat Dari Pemanfaatan Limbah Batubara Dengan Matrik Resin Poliester Tak Jenuh. Jurnal Mechanical, 1(1), 2010,

[10] B. Syam, S Rizal. 2013. Pengembangan bahan komposit busa polimer (polymeric foam composite) diperkuat serat tandan kosong kelapa sawit (TKKS) untuk pembuatan berbagai produk industri transportasi dan olah raga. Final Report PENPRINAS MP3EI, DIKTI.

[11] D Hastrino, M Muttaqin, B Syam, dkk. 2018 Analisa struktur speed bump paduan bahan concrete foam dan polymeric foam dengan variasi rongga menggunakan sofware Ansys 14.5. Talenta Conference Series: Energy and Engineering (EE) 1 (1), 104-108.

[12] M Muttaqin. 2022 Static Simulation on Speed Bumps Made of Foam Concrete Foam with Durian Skin Fibers Using Ansys Software. Journal of Mechanical Engineering, Science, and Innovation 2 (1), 1-8.

[13] REK Siregar, B Syam, B Wirjosentono, M Muttaqin. 2018 Static simulation to horse shoes alternative materials based basic polymeric foam reinforced fiberglass with ANSYS software.

[14] Rios-Soberanis, C. R., R. H. Cruz-Estrada, J. Rodriguez-Laviada, dan E. Perez-Pacheco. 2012. Study of Mechanical Behavior of Textile Reiforced Composite Material. Engineering Journal DYNA: 115-123.

[15] Nuzuli dan Syam Bustami, 2018. Concrete Foam Composite Production and Testing, Universitas Sumatera pp. 1-6.

[16] S C, Liew. 2008. Pencirian Sifat Komposit Polimer Bergentian Semula Jadi Untuk Aplikasi Struktur, Master Thesis, Fakulti Kejuruteraan Awam Universiti Teknologi Malaysia.

[17] Liwandy, Sinaga. 2021. "Analisa Kekuatan Bending Dan Tarik Statik Komposit Berbahan Dasar Polymeric Resin Diperkuat Serat Kaca Sebagai Bahan Alternatif Pembuatan Palang Komposit". [Skripsi] Medan:

Universitas Sumatera Utara.

How to Cite

Purba, Z., Muttaqin, M., Syam, B., Irwan, A., & Rozy, F. (2023). Structural Analysis of Parking Barrier Made of Polymeric Resin Using Ansys Software. International Journal of Research in Vocational Studies (IJRVOCAS), 2(4), 105–115. https://doi.org/10.53893/ijrvocas.v2i4.175