ABSTRAK

Pertumbuhan transportasi di Indonesia sangatlah pesat, khususnya pada bidang transportasi darat. Hal ini yang mendasari team E-Niaga untuk membuat kendaraan roda tiga. *Chassis* merupakan komponen utama pada kendaraan yang didesain mempunyai kekakuan yang tinggi agar dapat menahan gaya aksial, gaya normal, dan momen.Untuk mengetahui kekuatan pada *Chassis* yang akan didesain, maka *Chassis* akan dilakukan simulasi dengan dua model pembebanan. Pembebanan yang akan dilakukan adalah pembabanan saat kendaraan berakselerasi, dan pembebanan saat kendaraan mengerem. Beban yang akan digunakan meliputi beban *Chassis, Cabinet, Driver, Traybody,* dan Payload dengan besaran sesuai dengan model *CAD* yang dibuat. Simulasi yang dilakukan menggunakan *Software Ansys Workbench 18.1* dan untuk pemodelan *Chasiss* menggunakan *Software Autodesk Inventor 2022.* Perhitungan yang dilakukan, kendaraan pada saat berakselerasi didapatkan gaya sebesar 1082,28 N dan saat mengerem didapatkan gaya sebesar 6900 N. Simulasi berfokus pada pembebanan akibat pengereman dikarenakan memiliki nilai gaya yang lebih besar dari pada pembebanan akselerasi. Setelah dilakukan simulasi didapatkan nilai tegangan *von mises* sebesar 75,5 MPa dengan batas tegangan ijin material ASTM A36 sebesar 250 MPa, tegangan *maximum shear* sebesar 28 MPa dengan batas tegangan geser maximum sebesar 58,3 MPa, dan nilai *Safety factor* diperoleh sebesar 3,3 dengan batas aman untuk kendaraan dengan beban dinamis 2,1 – 3. Dengan hasil simulasi yang dilakukan maka desain *chassis* dapat digunakan pada kendaraan E-Niaga dengan factor keamanan yang baik.

Kata Kunci: *Stress Analysis, Finit Element Method,* Analisa pembebanan Statis

ANALYSIS AND SIMULATION OF STATIC LOADING ON THE FRAME OF THE GENI BIRU THREE-WHEEL E-COMMERCIAL ELECTRIC VEHICLE USING THE FINITE ELEMENT METHOD

ABSTRACT

Plastic chairs are furniture that is often found in the general public because of its easy use to carry everywhere and also save storage because it can be stacked. In everyday use, plastic chairs are often damaged. Damage to the plastic chair is generally in the form of broken. Damage to plastic chairs is influenced by the load received on the plastic chair is greater than the load that can be held (yield strength) of the plastic chair. The purpose of this research is to analyze the loading of plastic. Based on data from the Centers for Disease Control and Prevention (2022), human body weight in the obesity category is in the range of 90 kg – 120 kg. From this data, the figure of 120 kg is taken as a reference in the analysis of the load on the plastic chair design. The analysis was carried out in a static loading structure at Solidworks, with the load being a force of 1200 Newtons (120 kg load) acting downwards (Y-axis) along the four legs of a fixed plastic chair. The analysis method uses von-mises stress, displacement, and safety factor simulations in Solidworks software. The analysis was carried out on three alternative designs, namely the design of the plastic chair without ribs, plastic chair with the ribs of the X model, and plastic chair with the ribs of the square model. Based on the analysis, the results of the von-mises stress value of the three alternative designs are 38.30 MPa; 30.81 MPa; and 16.65 MPa with the allowable stress limit by ABS material is 28.00 MPa. The safety factor values of the three alternative designs are 0.73; 0.91; and 1.68. The minimum safe limit for static loads is 1.25 according to Dobrovolsky's "machine element" book. Thus the alternative design that meets the requirements is a plastic chair with square ribs with a safety factor value of 1.68. As for the other two designs it is not safe. The unsafe location occurs at the corner of the base of the plastic chair legs.

Keywords**:** *Plastic Chair*, *Rib*, *Von Mises Stress, Displacement, Safety Factor.*