

ANALYSIS AND MODELING OF WEIGHBRIDGE CONSTRUCTION REINFORCEMENT AT PT BUKIT ASAM

Praja Dilla Atos*

Department of Mechanical Engineering, Faculty of Engineering, Universitas Sriwijaya, South Sumatera, Indonesia, Indonesia

ABSTRACT

Coal mining is a process that consists of several stages, and each stage must be passed as an inseparable part of the other stages. One of the important stages in the coal mining process is the weighing process used to measure the achievement of coal production and as a transactional tool. The weighing of coal is done using a bridging scale. The importance of the role in mining productivity must have strived for no failure in its main components, one of which is the weighbridge construction, where several occurrences of deformation occur in the main beam construction. Repair and strengthening efforts are carried out to overcome the deformation problems based on analysis and modeling using the software-based Finite Element Method. The material used is modeled using the Finite Element Method with loading $FA = 41,618.18 \text{ N}$, $FB = 51,087.56 \text{ N}$, and $FC = 26,203.35 \text{ N}$ towards the Z axis ST52-3 with Density $7,850 \text{ g/cm}^3$, Yield Strength 345 Mpa , Tensile Strength 560 Mpa . The meshing of 946,278 elements and 1,728,745 nodes. Construction analysis using finite element-based software Inventor 2019. The simulation results can be concluded that the greatest von mises stress occurs in the direction of the Z axis with a value of 184 MPa , and the largest displacement occurs in the direction of the Z vector with a value of 7.98 mm .

Keywords: Weighbridge, FEM, Autodesk Inventor, von mises stress, displacement.

1 INTRODUCTION

PT Bukit Asam always strives to increase coal production every year. One of the efforts made is to increase the carrying capacity of dump trucks from 20 tons to 30 tons. Mining contractors in each mining area carry out the mining process itself. In the case of contractor payments, it is calculated from the weight of the coal transported by dump truck from the front area to the stockpile area. The weighing of coal haulage is carried out using a weighbridge.

Meanwhile, mining operations that are required to continue producing and will use weighbridges as a measurement tool for production results and payment of contractor tariffs need to be maintained for reliability, and repairs must be carried out in a fairly short time without disrupting production.

2 LITERATURE REVIEW

2.1 Weighbridge

A weighbridge is a tool consisting of several components arranged so that it can be used to measure the mass of a light vehicle unit to a heavy vehicle both in unloaded and unloaded conditions. According to J.L. Cannon and L.G. Kraige in their book "Engineering Mechanics Statics Vol. I 15th Edition", that mass is a measure of the inertia of a body, which is its resistance to a change of velocity. Mass can also be considered a body's quantity of matter. The mass of a body affects the gravitational attraction force between it and other bodies. This force appears in many applications in statics. To carry out its function, a set of weighbridges generally consists of several main components, including a foundation, platform, and electronic instruments such as load cells, weighing indicators, and junction boxes.

In general, the working principle of the weighbridge is that when the vehicle enters and is

*Corresponding author's email: atosprajadilla@gmail.com

above the platform, the load cell will directly detect the mass or compressive force resulting from the reaction on the platform pedestal and then convert it into electric current or voltage. The electric current or voltage generated is not always constant. And tends to fluctuate following changes in the condition of the vehicle being weighed. Furthermore, the electric current or voltage will be sent to the weighing indicator to be processed on digital analog to become a scale value or digital number from the scales.

2.2 Software of Autodesk Inventor

Autodesk Inventor is a software product developed by a US-based company. Autodesk Inventor is a CAD (Computer Aided Design) program with solid three-dimensional modeling capabilities for making 3D prototype objects visually, simulating, drafting, and documenting the data [1]. In Autodesk Inventor, a designer can create a 2D sketch of a product and model it into 3D to continue with the visual prototyping or even more complex ones [2], Namely, the Autodesk Inventor simulation has several advantages that can facilitate the design and a more attractive and real appearance because of the material facilities provided.

2.3 Structural Analysis on Autodesk Inventor

Software technology in engineering is growing rapidly in helping the process of completing the analysis, specifically in analyzing a structure. Conducting structural analysis using Autodesk Inventor can produce a structural calculation including stress or stress that occurs in a component, deflection, and motion simulation that makes it easier for machinists to describe or realize a digital prototype form by providing simulation parameters for real approach conditions before making the shape [3].

One of the important points in conducting structural analysis using Autodesk Inventor is loading and selecting the right and appropriate material because this greatly affects the analysis results. So to get accurate results, it must be ensured that the properties of the material we use represent the material that will be used. Likewise, restraints and loads, these two things must be able to describe the actual conditions that a structure will accept.

Stress Analysis on Autodesk Inventor uses linear static analysis based on the Finite Element Method (FEM) to calculate stress [4]. FEM is a

reliable analytical method for engineering design. This method replaces complex problems with several simple problems. This method divides the model into smaller parts with simple shapes called elements. Each element is further divided into points called nodes. The analysis method using FEM is called Finite Element Analysis (FEA).

2.4 Material Force Theory (Strength of Material)

The procedure for selecting a material according to the application conditions is determined in designing a structure. The strength of the material is not the only criterion that must be considered in the design of the structure [5]. The stiffness of a material is equally important to a lesser degree; properties such as hardness, and toughness are determinants of material selection. An experimental tensile test on the specimen from the stress due to the tensile force it is subjected to.

Some of the material properties that are considered in the selection of material:

- a. Tenacity is the property of a material that allows it to absorb energy at high stresses without breaking, which is usually above the elastic limit.
- b. Elasticity is the property of a material's ability to return to its original size and shape after an external force is removed. This property is important in all structures subjected to variable loads.
- c. Stiffness is a property based on the extent to which a material can resist deformation. A measure of the stiffness of a material is its modulus of elasticity, which is obtained by dividing the unit stress by the unit deformation caused by the stress.
- d. Foreseeability is the property of a material whose shape can be changed by applying compressive stresses without damage.
- e. Strength is the ability of a material to withstand stress without breaking. Some materials, such as structural steel, wrought iron, aluminum, and copper, have nearly the same tensile and compressive strengths. In contrast, their shear strength is approximately two-thirds of their tensile strength.

2.5 Structural Failure Theory

In designing a machine or steel construction, what must be considered is knowing how the material

is at work. Certain characteristics or properties of the material to be applied must be known in advance. Usually, the material's characteristics can be known by performing a tensile test (Tensile Test).

In this tensile test, the load is continuously added to a material to be studied. It is known how much load and elongation occurs in the material until the material breaks. The stress that occurs is calculated by dividing the load by the cross-sectional area (cross-sectional area) of the material to be tested. The amount of elongation or strain can be determined by dividing the change in length that occurs due to the load's addition by the material's initial length. The tensile test value obtained can be one of the parameters to determine the failure of a structure [6].

Overall types of failure in the material can be formed, such as fatigue, wear (wear), corrosion, fracture, impact, and others. Failure can occur due to several factors, one of which is caused by static loads and mechanical loads, so stresses often arise due to loads exceeding yield strength.

2.6 Maximum Shear Stress Theory

Maximum Shear Stress Theory is often used in ductile materials [7-8]. The value of the maximum shear stress at yielding (yield) is $\tau_{max} = S_y/2$. Under general stress conditions, the three principal stresses can be determined and arranged in such a way that $\sigma_1 \geq \sigma_2 \geq \sigma_3$. Then the maximum shear stress is $\tau_{max} = (\sigma_1 - \sigma_3)/2$ as illustrated in Figure 1.

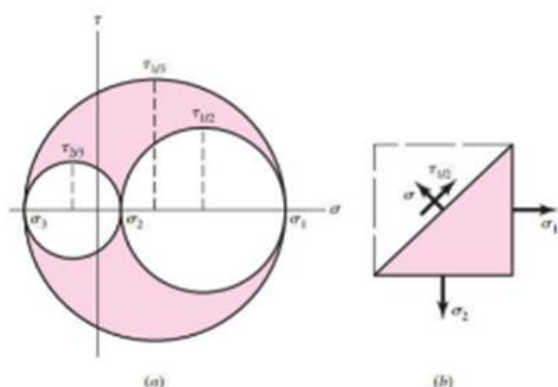


Figure 1 Mohr's circle for three-dimensional stress

2.7 Design Factors

The general approach compares the allowable load with the load that occurs due to loss of function in obtaining the design factor [2]. This factor is then referred to as the safety factor (n).

Safety factors have the same definition as design factors but are generally different numerically.

So to provide a safety limit and protect against failure caused by the unpredictable, the allowable stress must be less than the stress that results in failure. Working stress is the stress value for a material used to determine safe stress [9].

General recommendations for safety factor values are given in Table 1 [7-10]:

Table 1 General recommendations for safety factors

Safety Factor	Use
1,3 – 1,5	For use on highly reliable materials—design structures that accept static loads with a high confidence level for all design data.
1,5 – 2,0	Design of machine elements that accept dynamic loading with an average confidence level for all design data
2,0 – 2,5	For use with ordinary materials where loading and environmental conditions are not severe—the average confidence level for all design data.
2,5 – 3,0	For materials of unknown type and brittle materials where loading and environmental conditions are not severe. Designs for static structures or on-machine elements subjected to dynamic loading with uncertainties regarding loads, material properties, stress, or environmental analysis
3,0 – 4,0	For applications where material properties are unreliable and where loading and environmental conditions are not severe, or where reliable materials will be used in difficult loading and environmental conditions

3 INTRODUCTION

3.1 General Research Process

In general, this research process can be seen in Figure 2.

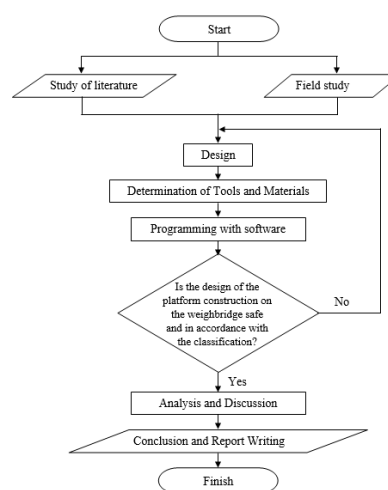


Figure 2 General research process flowchart

3.2 Finite Element Solving Procedure

This procedure describes the steps in simulating to get the value of the structural strength of the platform construction on the portable truck scale, as shown in Figure 3. The completion steps are as described below:

Pre-Processing, in this step, the procedure for modeling the geometry of the platform construction on a portable truck scale is explained, determining the type of material to be used, the constraints, the procedure for doing meshing, and giving the forces or loading points. Further explanation as explained below:

- a. Platform Construction Geometry Modeling,
- b. Entering Material Types,
- c. The process of determining area constraints,
- d. Determining Loads,
- e. Meshing Process.

Simulation Running process, running finite element-based software stress analysis function with static analysis method.

Post Processing the final process in performing analysis procedures using finite element software. The simulation results were obtained from von misses stress, displacement, and safety factor in this process.

- a. Von Misses Stress
- b. Displacement
- c. Safety Factor

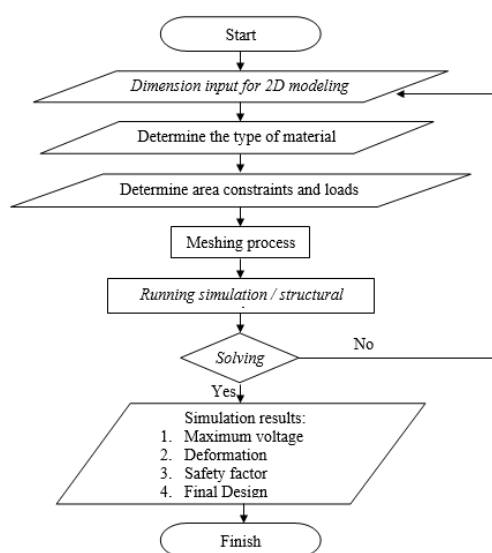


Figure 3 Flowchart of the solving process with finite elements

4 RESULT AND DISCUSSION

4.1 Construction Simulation

To determine the size of the construction that will be made in making this 2D sketch, it refers to the dimensions of the dump truck that will be used in this case, namely the Kamaz 6520 (6x4) Dump Truck. After completing the 2D sketch, 3D modeling is carried out using the Design feature in Autodesk Inventor.

4.2 Assumption of Steel Profile used

In making a 3D geometry model, as shown in Figure 4, the author assumes to use a steel profile, as shown in Table 2.

Table 2 List of steel profiles used

No Item	Steel Profile Dimension	Standard	Material Type
1	I-600x200x11	JIS G 3192 I-Shape	ST52-3
2	I-300x150x10	JIS G 3192 I-Shape	ST52-3
3	C-200x75x8,5	DIN 1026-1	ST52-3
4	C-200x75x8,5	DIN 1026-1	ST52-3

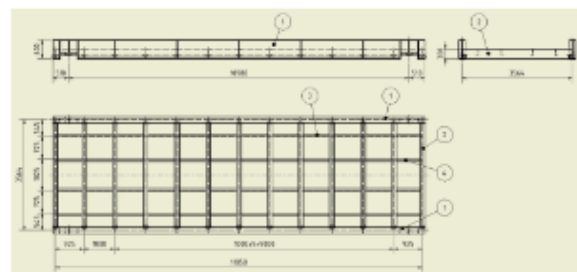


Figure 4 3D drawing of the platform construction

4.3 The Process of Determining Area Constraints

The platform construction will rest on a load cell, where the surface contact between the load cell and the area on the platform construction will become area constraints, as shown in Figure 5 and Figure 6. Because the assembly system between the construction and the pedestal uses the bolting method, then fixed constraints are used.



Figure 5 Side view of the constraint area



Figure 6 Front view of the constraint area

4.4 Determination of Loads

The platform construction load on this portable truck scale is a dump truck unit with a maximum gross weight of 40,000 kg, as for the load distributed to each dump truck wheel, totaling 10 Ea, as shown in Figure 8. If it is known that the Center of Gravity of the dump truck is located at a distance of 5,429 mm from the front of the truck, the distribution of the load is determined as described in Figure 7.

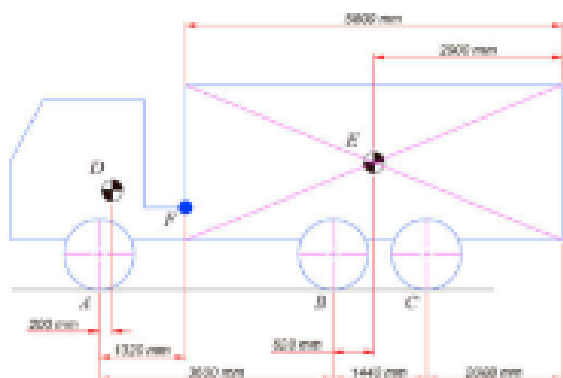


Figure 7 General dimensions of dump truck

Information:

- A = Front wheels totaling 2 (1 each left and right)
- B = Center wheel totaling 4 (2 each left and right)
- C = 4 rear wheels (each left and right)
- D = Center of mass/gravity of dump truck cab
- E = center of mass/gravity of dump truck body
- F = Assuming for connection between the cab and tailgate located at point F (which is equivalent to an elastic but smooth pin)

The static load on the series of structures for platform construction is 40 tons distributed by each wheel with a force of $F_A = 41,618.18 \text{ N}$, $F_B = 51,087.56 \text{ N}$, and $F_C = 26,203.35 \text{ N}$.

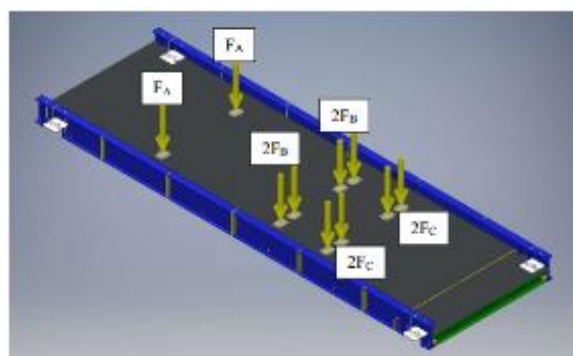


Figure 8 Distribution of FA, FB, and FC. Loads

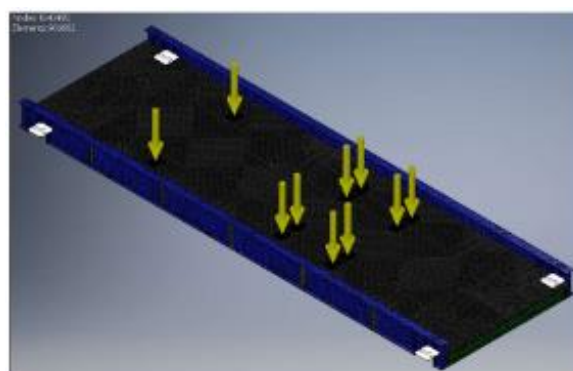


Figure 9 Meshing results

4.5 Meshing Process

The construction will be made into 946,278 elements and 1,728,745 nodes, as shown in Figure 9.

4.6 Results of Von Mises stress

The maximum equivalent stress value occurs in one of the middle sections of the construction of 184 MPa, as can be seen in Figure 11. In comparison, the minimum equivalent stress occurs at 0 MPa, which occurs in one of the middle sections of the construction, as shown in Figure 12.

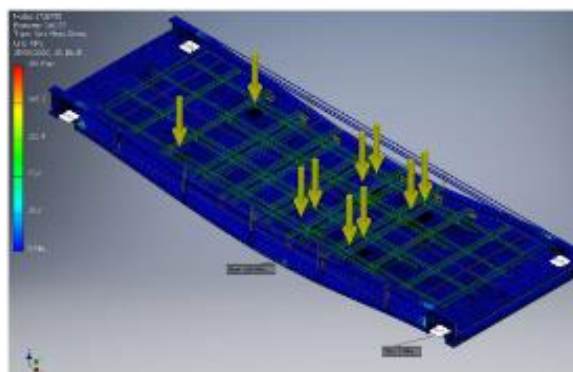


Figure 10 Display of the results of Von Mises stress

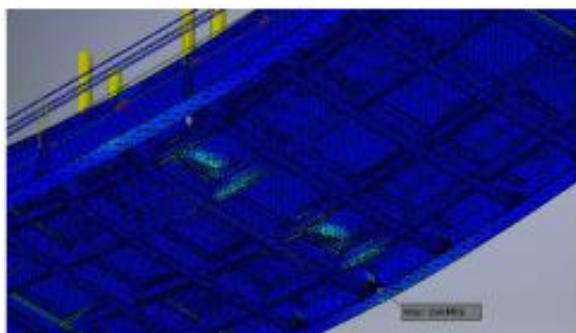


Figure 11 Display of maximum equivalent voltage

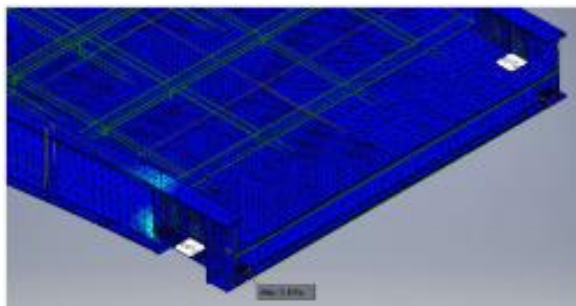


Figure 12 Display of the minimum equivalent voltage

4.7 Displacement Results

The simulation results show that the largest total deformation is in the center of the platform construction at 7.98 mm. The smallest total deformation is in one part close to the fixed constraints area of 0 mm, as shown in Figure 13. The deformation values can be broken down towards the X, Y, and Z axes. Then this displacement component can be called a directional deformation. The largest deformation or displacement value only occurs on the Z axis or the axis in the direction of the gravitational force. From the simulation results, the stress analysis shows that the largest and smallest deformation on the X-axis only occurs at the top of the I-Beam steel profile, which is close to the position of the constraints with each value of 0.572 mm and 0.4643 mm, as shown in Figure 14. The largest and smallest deformation in the Y-axis direction occurs in the I-Beam profile construction, where the FB and FC loads are placed with each deformation value of 0.8818 mm and 0.8783 mm, as shown in Figure 15. In comparison, the displacement is in the Z-axis or the same direction. With the largest gravitational force occurring at the bottom frame or where the dump truck wheels are at rest, with a deformation value of 7.987 mm, the smallest deformation value occurs at each end of the construction with a

deformation value of 0.439 mm, as shown in Figure 16.

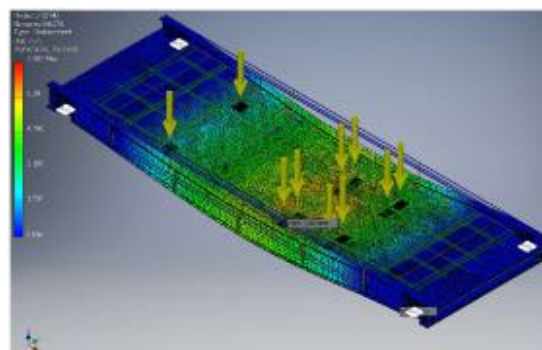


Figure 13 Display of deformation results

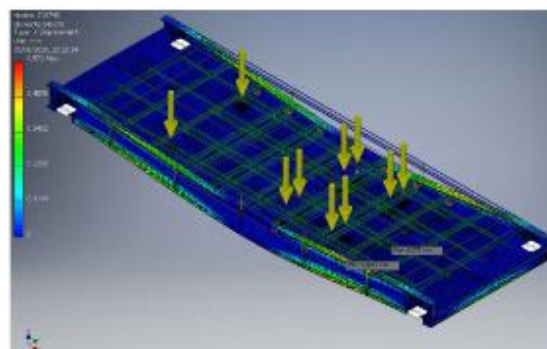


Figure 14 Display of deformation in the X direction

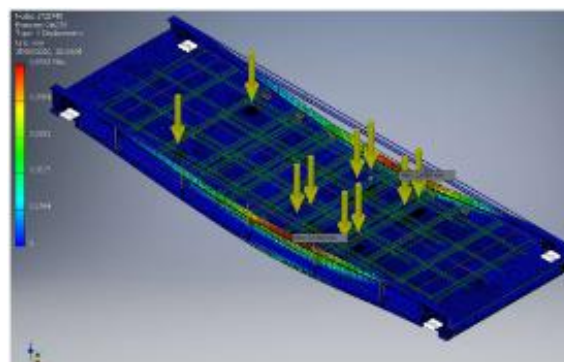


Figure 15 Display of deformation in the Y direction

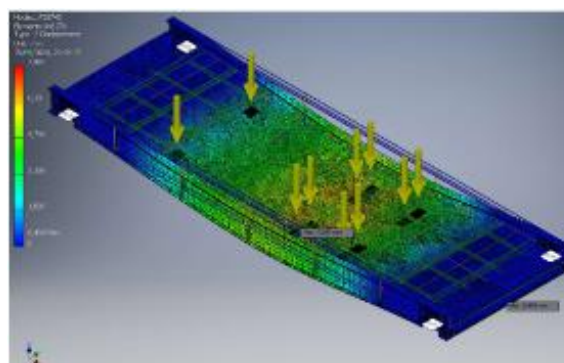


Figure 16 Z direction deformation display

4.8 Safety Factor

The simulation results show that the maximum safety factor value is 15, and the minimum safety factor value is 1.49, as shown in Figure 17.

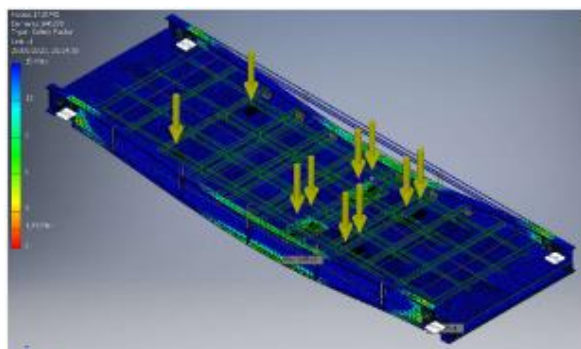


Figure 17 Display of safety factor results

Thus, the analysis and simulation results can be recapitulated, as shown in Table 3.

Table 3. Recapitulation of simulation results

Simulation Results		Gross Weight 40.000 kg
Von Mises	Maximum	184 MPa
	Minimum	0 MPa
Displacement	Maximum	7,98 mm
	Minimum	0 mm
Safety Factor	Maximum	15
	Minimum	1,49

5 CONCLUSION

As with the purpose of this research, it can be concluded that based on the geometry of the weighbridge construction modeled using Autodesk Inventor software, the construction can accommodate changes in the hauling distance of dump trucks from the front area to the weighbridge with quite flexible capabilities. In addition, based on the simulation results of the construction analysis that has been carried out using the Autodesk Inventor 2019 software, it can be concluded that the construction is safe and feasible to be loaded with dump trucks with a gross weight of 40 tons.

In this study, the authors suggest several things, namely efforts to find out how much the lifetime of the weighbridge construction is; it is necessary to carry out a simulation method of advanced construction analysis using dynamic analysis by considering external factors. Then it is necessary to conduct further trials on whether the weighbridge construction can use other materials or cross-sectional steel profiles to choose a more economical alternative construction. In implementing advanced simulations such as

dynamic analysis, it is better to calculate the effect of vibration on the feasibility of the construction.

REFERENCES

- [1] Ambrosius, L. (2008). AutoCAD 2009 & AutoCAD LT 2009 All in One Desk Reference for Dummies. Indianapolis: Wiley Publishing, Inc.
- [2] Budynas, R. G., & Nisbett, J. K. (2011). Shigley's Mechanical Engineering Design (9th ed.). McGraw-Hill Companies, Inc.
- [3] Asroni. (2015). Stress Analysis pada Stand Shock Absorbers Sepeda Motor dengan Menggunakan Software Inventor 2015. TURBO ISSN 2301-6663 Vol.4 No.1.
- [4] Waguespack, C. (2014). Mastering Autodesk Inventor 2015 and Autodesk Inventor LT 2015. Indianapolis: John Wiley & Sons, Inc.
- [5] Meriam, J. L., & Kraige, L. G. (2002). Engineering Mechanics Static (fifth ed., Vol. 1). United States of America: John Wiley & Sons, Inc.
- [6] Schijve, J. (2001). Fatigue of Structures and Materials. Dordrecht: Kluwer Academic Publishers.
- [7] Oberg, E., Jones, F. D., Horton, H. L., & Ryffel, H. H. (2000). Machinery's Handbook (Vol. 26th Edition). New York: Industrial Press Inc.
- [8] Timoshenko, S. (1948). Stength of Materials (II ed.). California: Lancaster Press, Inc.
- [9] George E. Dieter, J. (1961). Metallurgy and Metallurgy Engineering Series. USA: McGraw-Hill Book Company, Inc.
- [10] Setyono, B., Mrihrenaningtyas, & Hamid, A. (2016). Perancangan dan Analisis Kekuatan Frame Sepeda Hibrid "Trisona" Menggunakan Software Autodesk Inventor. Jurnal IPTEK Vol. 20 No. 2 ISSN:1411-7010.