Stress and strain analysis from dynamic loads of mechanical hand using finite element method

Iskandar Hasanuddin¹, Husaini¹, M. Syahril Anwar¹, Sandy Yudha B.Z.² and Hasan Akhyar³

¹Department of Mechanical and Industrial Engineering, Universitas Syiah Kuala Jl. Tgk. Syech Abdurrauf No. 7 Darussalam – Banda Aceh 23111, Indonesia
²Department of Mechatronics Engineering, Politeknik Aceh Jl. Politeknik Aceh, Pango Raya, Banda Aceh 23116, Indonesia
³Department of Mechanical Engineering, Universitas Gajah Mada Jl. Grafika No. 2 Sleman - Yogyakarta 55284, Indonesia

E-mail: iskandarhasanuddin@unsyiah.ac.id

Abstract. This research discusses the distribution of stress and strain due to the dynamic loads of mechanical hand. The stress and strain that occur on mechanical hand are the main concern for comparing the value of finite element analysis (FEA) and calculating for its material properties. The stress and strain analysis is done with a loading condition. The given loading condition is dynamic. The loading input condition in the simulation of using hydraulic hand dynamometer is from the grip strength measurement of ten samples. The form of the given loading to the mechanical hand is the increment value with a maximum of 708 N/m² within 1 minute. The amount of maximum stress (von Mises) simulation is 1.731 x 10⁵ Pa, and the amount of maximum strain is 7.441 x 10⁻⁷. The amount of maximum reaction force is 5.864 x 10⁻² N, while the amount of maximum displacement that occurs on the distal part is 1.223 x 10⁻² m. Based on the analysis, the maximum stress and strain were found both to occur at the extension part. The result of this study has shown that the stress and strain still occur far below from the yield strength and the shear strength from the material AISI 1010. It can be concluded that the mechanical hand is durable for the given loading and can hold an object with a minimum diameter of 45 mm.

1. Introduction
The hand is a multifunctional part of the body. It can do multiple things at the same time. Clenching the fingers can form a grip. Humans have hands which consist of four fingers and a thumb each. The main functions of the hand are to hold, to grab, and to grasp. In certain conditions, the hand cannot touch the target object directly, for instance when holding a beaker glass containing certain chemicals, lest the chemicals splash on the hand. In that case, a device is required to hold or handle the chemicals.

Nowadays, the development of such device has been greatly studied. Ganipineni Sai Krishna designed and analyzed the mechanical hand for industrial application for grasping oddly shaped objects. The result is that the mechanical hand can grasp the objects well [1]. Ivan Virgala analyzed mechanical hand modeling, but his model is only able to form a grip [2]. Sandy Yudha Barry Zaqy designed a mechanical hand and analyzed the strength in a static start-up loading condition with a load of 1000 N,
and the result is that the load applied to the finger does not exceed the strength of the elastic material [3]. However, there has not been any analysis regarding dynamic-implicit loading condition, and the load is given merely an assumption.

Due to the problem above, this research continues the dynamic-implicit loading condition using finite element analysis. The purpose of this research is to analyze the stress and strain forces caused by the dynamic load using finite element analysis.

2. Literature Review

2.1 Mechanical Hand
The mechanical hand is modeled using the software Autodesk Inventor Professional 2015. The movements of the mechanical hand mimic the movements of the human hand [3]. The mechanical hand has five fingers; the index finger, the middle finger, the ring finger, the little finger, and the thumb.

Stress is the quantity describing the ratio of the pull force that is applied to the cross-section area of an object. The stress that occurs affects the mechanical properties of the material of the object. Stress occurs due to pressure, pull [4]. Stress is defined as:

$$\sigma = \frac{F}{A}$$ (1)

where F is the force or the pull load applied, which is measured in the unit Newton (N), and A is the cross-section area before the load is applied (m²). Stress is measured in megapascal MPa (SI) (where 1 MPa = 10⁶ N/m²).

2.2 Strain Analysis
The strain is defined as the ratio of the added length of an object to the original length. The word strain relates to the relative change in the dimension or shape of the object where a pressure is applied, defined as:

$$\varepsilon = \frac{l - l_0}{l_0} = \frac{\Delta l}{l_0}$$ (2)

Where: $\varepsilon$ = strain, $\Delta$ = length of object (m), $l_0$ = original length (m) dan $\Delta l$ = length difference (m) [4].

2.3 Finite Element Analysis
FEA is a numerical study that can be used to solve problems like stress analysis of structure and deformation [5], because exact calculation cannot be applied in complicated cases. Using this method, static and dynamic analysis can also be done. The simulation of finite element analysis is done using the software ABAQUS™.

3. Research Method

3.1 Research Location
This research is done in the Computer Aided Mechanical Laboratory and the Work System Design and Ergonomics Laboratory, Faculty of Engineering, Syiah Kuala University and collaborated with Mechanical Engineering Department, Gajah Mada University.

3.2 Research Equipments
To find out the load that will be applied, a measurement is made on the strength of the grip using the device Hydraulic Hand Dynamometer, because the data is needed for the application to FEA.
3.3 Testing Material
The material used is AISI 1010, with the mechanical properties as shown in Table 1.

| Property                | Value  |
|-------------------------|--------|
| Modulus Young (E)       | 210 Gpa|
| Poisson's ratio (ν)     | 0.30   |
| Density (ρ)             | 8,030 Kg/m³|
| Yield Strength (σ_y)    | 305 MPa|
| Tensile Strength (σ_R)  | 365 MPa|
| Shear Strength (τ)      | 260 MPa|

3.4 Testing Procedure
In this research, the components in the mechanical hand that will be analyzed with the finite element analysis are the connecting pins in the mechanical hand, which are assumed to be the joints of the human fingers, as shown in Figure 1.

![Figure 1. Index finger component to be analyzed using stress and strain analysis [3]](image)

The load given is based on the increment value to time. The minimum increment value is 20 to 708 N/m² in 1 minute. The area observed can be seen in Figure 2. The position of the loading can be seen in Figure 3.

![Figure 2. Observed pins](image)
The boundary condition for the dynamic-implicit loading position is chosen at the base. The type chosen is ENCASTRE (U1 =U2 =U3 =UR1 =UR2 =UR3 =0).

![Figure 3. Loading position and boundary condition](image)

The meshing step is done separately or make dependent, with the seed part at 0.004, and the mesh type is a hexahedron. The number of nodes is found to be 10826 and the number of elements 5752.

4. Result and Discussion

4.1 Simulation Result

Below is a depiction of the simulation result of the finger of the mechanical hand. Figure 4 is the result of stress (von Mises), with the maximum value at node 358 in the extension part. A contour of red coloring is seen with the value of $1,731 \times 10^5$ Pa.

![Figure 4. Stress contour occurring at the extension part](image)

Meanwhile for the maximum displacement at node 35 is $1,223 \times 10^{-1}$ m as shown in figure 5, with the brightest red contour on the distal part.
Figure 5. Displacement contour occurring at mechanical hand finger

Figure 6 below shows the contour of the maximum reaction force at node 329 at the base area of the mechanical hand, seen with red contour at 5.864 x 10^{-2} N.

Figure 6. Contour of reaction force at mechanical hand finger

Figure 7 shows the contour of the maximum strain at node 135 at the extension part at 7.441 x 10^{-7}.

Figure 7. Contour of strain at mechanical hand finger

Considering all the parts, maximum stress (von Mises) occurs on the extension part at 1.731 x 10^5 Pa,
and maximum strain occurs at the extension part at $7.441 \times 10^{-7}$. Considering pin 1 through 7, maximum stress (von Mises) occurs to pin 5, which is the connecting part between the proximal and the distal part. Maximum strain occurs on pin 2, which is the connecting part of the extension part and the pull arm. Maximum stress displacement occurs on pin 1, which is the connecting part of the extension part and the base.

The yield strength of the material AISI 1010 is 305 MPa, maximum stress (von Mises) occurs on the mechanical hand finger is $1.731 \times 10^5$ Pa, maximum stress displacement is 260 MPa, stress displacement that occurs on the mechanical hand finger is $2.021 \times 10^4$ Pa. This means that the design of the mechanical hand is safe for the given load.

5. Conclusion

Based on the stress and strain analysis on the mechanical hand finger, with the dynamic load using the finite element analysis, it can be concluded that:

- Maximum stress value (von Mises) is $1.731 \times 10^5$ Pa, which occurs on the extension part.
- Maximum strain value is $7.441 \times 10^{-7}$, which occurs on the extension part.
- The yield strength of the material AISI 1010 is 305 MPa, with a stress displacement of 260 MPa and maximum stress displacement of $2.021 \times 10^4$ Pa. Therefore, it can be concluded that the mechanical hand is safe for the given load.
- The corner of the finger can move ±45 mm from the distal part of the base of the hand.

References

[1] Ganipinenei S K, and Venkata R N 2014 Grasp and manipulation of five-fingered hand robot in unstructured environments *International Journal of Engineering Research and Applications* 497-102

[2] Ivan V and Michal K 2013 Kinematic analysis of humanoid robot hand *American Journal of Mechanical Engineering* 1 443-446

[3] Sandy Y B Z 2016 Perancangan Pembuatan dan Analisa Tegangan pada Prototipe Tangan Mekanik dengan Pendekatan Kinematik Model (Tesis Teknik Mesin, Universitas Syiah Kuala, Darussalam Banda Aceh) (in Indonesian)

[4] M Souisa 2011 Analisa Modulus Elastisitas dan Angka Poisson Ratio Bahan Dengan Uji Tarik Jurnal Ilmu Matematika dan Terapan 51-6 (in Indonesian)

[5] Rao S S 2005 *The Finite Element Method in Engineering, 4th edition* (Elsevier)

[6] Ridha M, Husaini and Husni 2013 *Material Teknik* (Syiah Kuala University Press.)

[7] http://www.azom.com/article.aspx?Article1. (Accessed 25 February 2017)

[8] https://icme.hpc.msstate.edumediawiki/index.php/1010_Steel. (Accessed 05 February 2017)