Jurnal
Teknologi

Full Paper

FINGERTIP STRUCTURAL ANALYSIS–A SIMULATED DESIGN EVALUATION

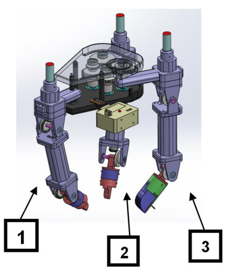
Jaafar, J.*, Shauri, R. L. A.

Faculty of Electrical Engineering, Universiti Teknologi MARA, Shah Alam, Selangor, Malaysia

Article history
Received
9 February 2015
Received in revised form
13 April 2015
Accepted
31 May 2015

*Corresponding author
jo.raiton@gmail.com

Graphical abstract



Abstract

This paper discusses the result on structural analysis of a recently designed fingertip for a three-fingered robot hand. This analysis is necessary to understand the structural changes occur under certain range of external force acting on the fingertip. The key parameters to be observed are deformation and equivalent (von-misses) stress. These simulated tests had given good results since there was no permanent deformation occurred and no yielding happened even in the area that had been considered critical. The fingertip will be produced as a prototype and equipped with the miniature load cells for force control of the three-fingered robot hand in the future.

Keywords: Finite element analysis, fingertip structure, deformation, von-misses, stress and strain

1.0 INTRODUCTION

Developing a robot hand to handle objects of various shapes and sizes is very challenging especially in meeting requirements such as ability to provide firm grasping, reduced size and weight to match human hand, housing sensory equipment and accessories and, to comply with tight budgeting. Hence, changes in the design are inevitable but every change needed to be analyzed for practicality. However, inefficient analysis technique used could resolve into error instead of better design, time consuming and could even waste valuable resources available. In order to avoid these inefficiencies, virtual prototyping [1][2] and structural analysis are required prior to hardware construction.

Finite Element Analysis (FEA) is one of the widely used methods for structural analysis to understand stress or strain experienced in a mechanical design due to applied force or pressure to the body. The FEA has been applied in various areas including validation for the structure of a tower crane [3], cantilever support of a foundation pit [4] and crank shaft of an engine [5]. This shows the reliability of FEA in validating mechanical design in terms of its integrity, safety and practicality before being actually manufactured and applied in the working environment. ANSYS Simulation Technology has also been used to analyze several mechanical designs of robots. The Carbon Fiber Reinforced Plastic (CRFP) body frame of Crabster200 seabed robot [6] has used ANSYS to analyze for its strength and durability to operate under water. Liaol [7] has analysed the natural frequency of the base of a welding robot to avoid resonance during operation. Rigidity and intensity of lower extremity exoskeleton has been analyzed by [8] in order to optimize its structure. Meanwhile, Kim [9] has made a comparison between stress result from a derived equation to calculate strain of a beam and the one provided by ANSYS. From above examples, it can be shown that ANSYS structural analysis software suite [10] has proved to be a trusted tool to analyze structural design using Finite Element Analysis method (FEA).

The design of a three-fingered robot hand using SolidWorks software has resulted to a working prototype in a previous work by [11]. However, the fingertip design requires few enhancements to improve the grasping capability and to accommodate sensors for force control. In this paper, an analysis of a new fingertip structural design through simulated test using CAE software by ANSYS, is presented. The FEA method is applied via ANSYS Simulation Technology to calculate the stress and strain characteristics and limitation of the fingertip. The structural analysis is required to avoid failure of the structure and possible overloading occurrence towards the miniature load cells when the fingertip is subjected to an external force of 15N maximum.

2.0 STRUCTURAL ANALYSIS

Structural analysis is required in this study since the fingertip will have a direct contact with an object during grasping and exerted with external forces during operation. The external force will become a load or stress felt by the structure of the fingertip. The alteration of shape and size of a structure can occur due to the load or stress. This is a condition known as deformation and in designing a structure; permanent or plastic deformation is the condition that needs to be avoided. Plastic deformation is a process in which enough stress is placed on metal or plastic to cause the object to change its size or shape in a way that it is not reversible. Deformation can occur if a structure is experiencing tension, compression or torsion depending on the magnitude and direction of force applied to the structure. Tension and compression is a condition when a structure is subjected to an external force, while the measurable parameter related to these two conditions are stress and strain.

In order to measure and understand the limit of a mechanical structure, important properties of the material which are the Young's Modulus and Tensile Yield Strength are taken as reference during analysis. Figure 1 shows the stress-strain curve of Aluminum Alloy and Mild Steel. As shown in Figure 1, the initial loading is indicated by the linear relationship between stress and strain. This linear relationship is similar between mild steel and aluminum alloy regardless of the magnitude of stress. Within this region, the deformation is completely recoverable. The linear relationship, where stress σ is directly proportional to strain ε is known as Hooke's Law; $\sigma = E\varepsilon$. The co-efficient E is the Young's Modulus or Modulus of Elasticity. Most mechanical structures are designed to function within their linear elastic region only. This is because after the stress reaches a critical value, the deformation becomes irrecoverable or irreversible. The corresponding stress is known as the yield stress or yield strength of the material beyond which the material is said to start yielding. Further application of stress beyond the yield stress will cause necking followed by breaking. During this stage, the material is permanently deformed.

In this paper, FEA tool in ANSYS can provide the value of deformation and the equivalent (Von-Misses) stress. Deformation which is measured as a displacement in meter (m) is not a usable parameter by itself in order to understand and analyze the behavior of a material under loading or stress. A quantity called strain defines the deformation in a better way. A tensile load applied to a material will cause elongation and compressive load could cause contraction. Both elongation and contraction can be represented as change in the original length denoted as δ. Strain can be simply defined as deformation per unit length and thus $\varepsilon = \delta / L$, where L is the length of material after elongation or contraction. Even though strain is a more meaningful quantity for analysis, ANSYS only provides information in terms of quantity of deformation. Fortunately, the deformation is converted into equivalent stress by ANSYS. This equivalent (Von Misses) stress is crucial for the analysis because this value can be compared to the Tensile Yield Strength. As long as the equivalent (Von Misses) stress is less than the value of Tensile Yield Strength, it can be concluded that no yielding occurs and the deformation is recoverable. This also denotes that the deformation is within the linear elastic region of the stress-strain curve as in Figure 1.

Von Misses criterion and Tresca criterion are two main structural failure criterion used in engineering design. Von Misses criterion or distortion energy theory suggests that yielding will occur when the maximum distortion or shear energy equals the maximum energy at yielding under uniaxial tension test. On the other hand, the Tresca yield criterion suggests that yielding will occur when the maximum shear stress under multiaxial loading reaches the value of the shear stress under uniaxial tension test. Since this structural analysis is performed on aluminum alloy which is a ductile material, yielding is the product of distortion rather than dilatational states in ductile materials. In addition to that, distortion can be associated with shear stress rather than with uniaxial tensile (or compressive) stress which is why von Misses stress analysis is more relevant for this study.

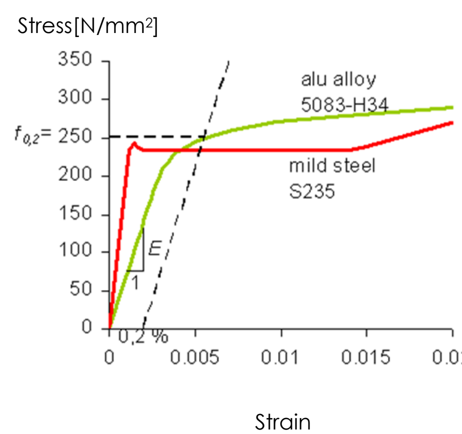


Figure 1 Stress-strain curve

3.0 METHODOLOGY

The design was developed using SolidWorks software and the analysis is carried out using Finite Element

Analysis (FEA) tool in ANSYS Simulation Technology, which provides visual and numerical information regarding stress-strain and deformation. The newly designed fingertip mainly focused on providing slots to install the load cells, improve grasping ability and to transmit the external forces acting on the fingertip to the load cell that enables the measurement of the external force. In relation to that, the purpose of this analysis and study is not to avoid deformation from developing within the structure but to ensure the deformation is within an allowable and safe limit. This is because the effect of deformation within the structure is the actual source of input to the load cell. The active surface on the fingertip is one part of the structure that is purposely designed to displace and press the load button on the load cell when the fingertip is subjected to external forces. The relative pressure from the external force will be sensed by the load cell. Figure 2 shows the active surfaces.

Any model produced in SolidWorks 2013 is recognized and is a supported format in ANSYS Simulation Technology. The compatibility between the two platforms has made it very easy to accomplish structural analysis. The analysis will be based on external force of 10N and 15N exerted to the fingertip at the direction of x-axis and z-axis. Measurement of deformation and von-misses stress provided by ANSYS Simulation Technology will be used to understand the relation and effect of stress and strain on the structure during loading.

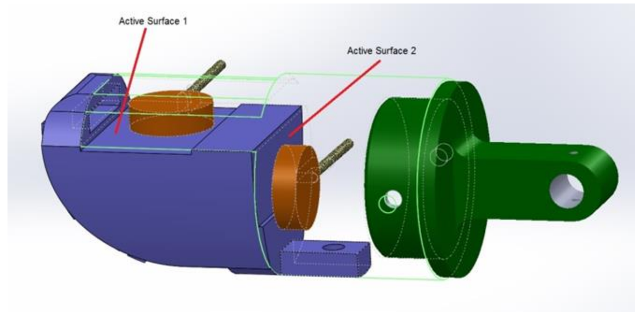


Figure 2 Active surfaces

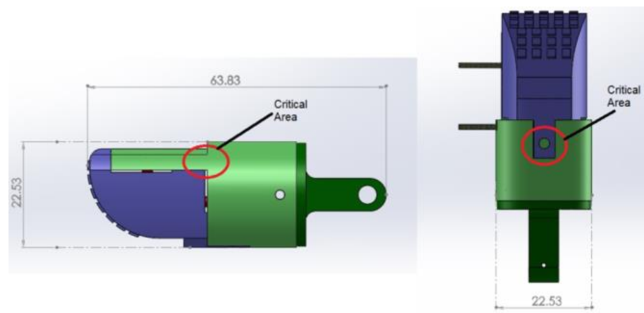


Figure 3 Dimension and critical area of fingertip

4.0 DIMENSION OF THE FINGERTIP

The dimension of the fingertip and the structure's critical area are shown in Figure 3. The area categorized as critical is due to its small dimension and the weakest region within the structure when subjected to external force. However, when the two different parts (green and purple color) are assembled to form the fingertip, these weak regions are enforced by complementing each other as seen in the figure. Figure 4 shows the application of the new fingertip design on the three-fingered robot hand. Referring to Figure 4, the fingers labeled as 1 and 2 are installed with the existing fingertip and the new fingertip is installed to finger 3.

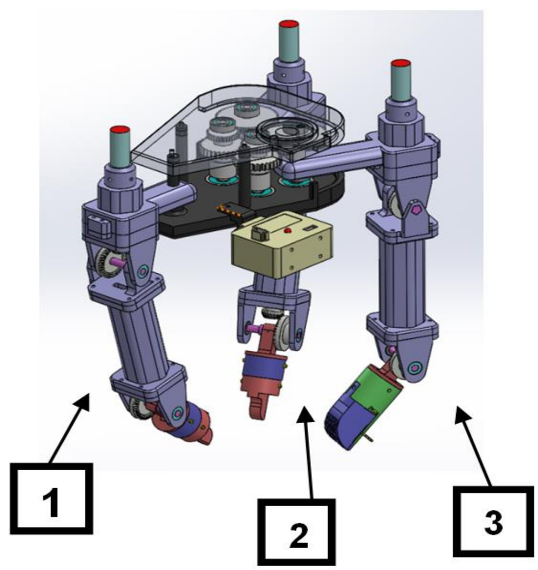


Figure 4 New and existing fingertip

5.0 EXPERIMENTAL RESULTS AND DISCUSSION

The concentration of the structural analysis will be on the effect of forces applied in the x-axis and z-axis of the fingertip. This directional orientation is based on the design made in SolidWorks and designation used for force control later in the project. However, ANSYS recognizes the x-axis as y-axis and remains the same for z-axis. This paper will retain the x-axis as y-axis for the sake of better understanding. The material chosen for the analysis is Aluminum Alloy, with Young's Modulus of 7.1E+10 Pa and Tensile Yield Strength of 2.8E+08 Pa.

The first test is to observe the structural deformation when a 10N force is applied in z-axis direction to the fingertip. As shown in Figure 5, only the tip-end experienced the highest deformation visualized by the red colored area and the deformation reduces towards the rear-end visualized by color changes from yellow, green to blue. Highest deformation captured is 2.2020E-8 meter maximum at the tip-end.

In Figure 6, the highest deformation is at 3.0304E-8 meter maximum when a 15N force is applied in z-axis direction to the fingertip. Other parts of the structure behave similarly as the first test but at higher magnitude of deformation. From both tests, it can be concluded that the structure can withstand the external force without failure or permanent deformation to the structure.

The equivalent (Von-Misses) stress when the fingertip was subjected to 10N force at z-axis is as per Figure 7. The maximum stress from the test is 4.1489E+05 Pa which is lower than the Tensile Yield Strength of the Aluminum Alloy. As discussed earlier, this shows that stress and strain effect on the fingertip under 10N of force is proportional to each other and the deformation is completely reversible.

Figure 8 shows the equivalent (Von-Misses) stress when the fingertip was subjected to 15N force at z-axis. The maximum stress from the test is 6.2233E+05 Pa which is still lower than the Tensile Yield Strength of the Aluminum Alloy of 2.8E+08 Pa. Again, the stress and strain effect on the fingertip under 15N of force is still proportional to each other and the deformation is completely reversible. As expected from the design, the surface of the fingertip that is in contact with an object or external force will experience the highest magnitude of stress. The stress is reducing within the structure which is further away from the source of stress. The transition of colors that denotes the severity of stress also shows that the deformation decreases as the position of structure is farther away from source of stress. The flow of deformation within the structure is very important to prove that the active surface is displaced to transmit the relative force for the load button on z-axis.

The subsequent test is carried out to see the effect of exerting the external force at the y-axis of the fingertip. As shown in Figure 9, a maximum of 1.7299E-7 meter deformation (red colored area) was felt by the fingertip after a 10N force applied at the y-axis. The equivalent (Von-Misses) stress was 7.4011E+5 Pa maximum but still lower than the Tensile Yield Strength, as shown in Figure 10. The deformation of the structure for this test is also within the elastic region of the stress-strain relation curve. The flow of deformation within the structure as analyzed from Figure 9 proved that the active surface is displaced to transmit the relative force for the load button on y-axis, as the proposed design requires.

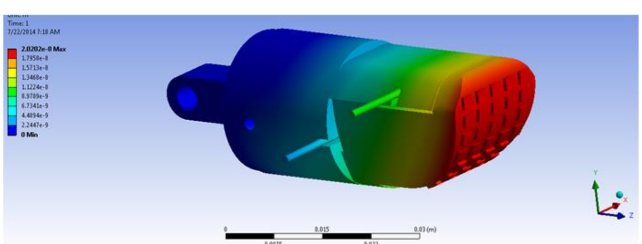


Figure 5 Deformation - 10N force at *z-axis*

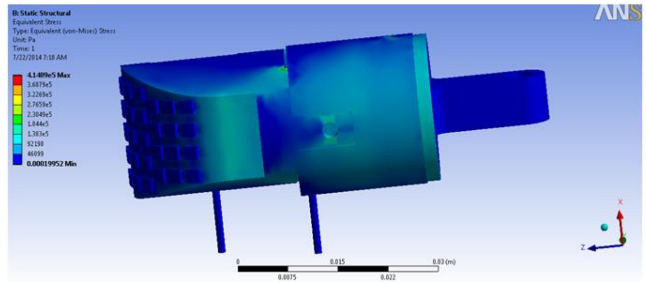


Figure 6 Deformation - 15N force at z-axis

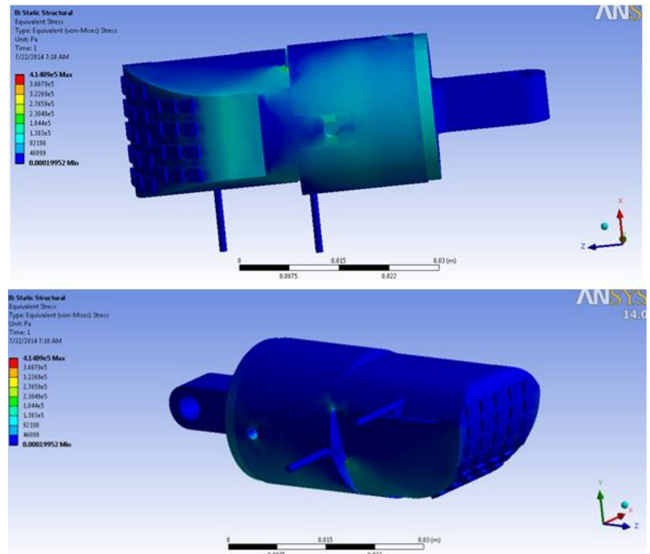


Figure 7 Von-Misses stress -10N force at z-axis

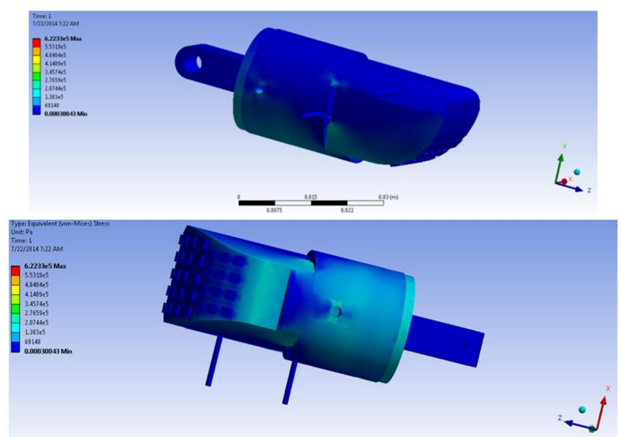


Figure 8 Von-Misses stress - 15N force at z-axis

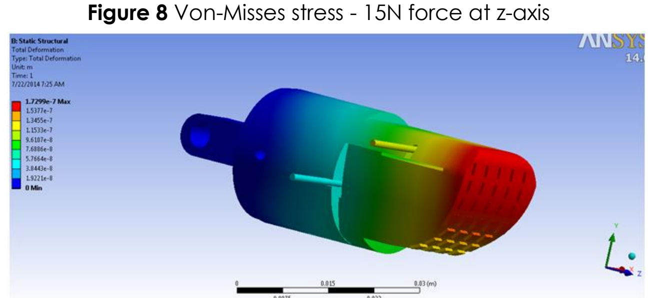


Figure 9 Deformation - 10N force at y-axis

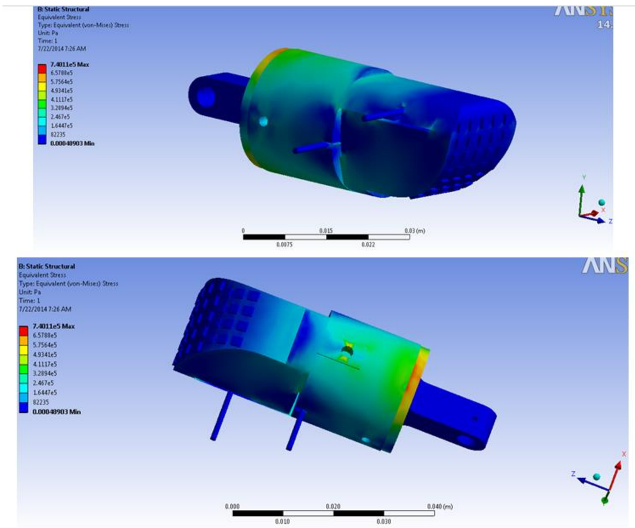


Figure 10 Von-Misses stress - 10N force at *y-axis*

6.0 CONCLUSION

The analysis has concluded that the fingertip design is a success in terms of the design and the reliability of its structure in handling designated external force without fail.

This analysis is significant not only to see the structural integrity of the fingertip but also to ensure that the stress delivers the desired deformation within the structure. The deformation will displace the active surface to transmit the relative force to the load button on the miniature load cell. On the contrary, severe deformation within the structure could overload the load cell. Therefore, it is very important to avoid this situation that could damage the sensor as it is the crucial part for the application of force control in the future works.

Acknowledgement

This research was supported by Ministry of Higher Education fund (600-RMI/FRGS 5/3 (86/2013)). The authors would also like to thank Research Management Institute (RMI) and Faculty of Electrical Engineering, UiTM for providing the financial supports and equipment to conduct this research.

References

[1] Chen, C. 2012. Virtual Model for a Multi-Finger Robot Hand Design. 2012 4th IEEE Int. Conf. Comput. Intell. Commun. Syst. Networks, 2012. *British Telecommunications Engineering*. 17: 291-295.

[2] Chen, C. 2012. Mechatronics Design of Multi-Finger Robot Hand. *2012 12th Int. Conf. Control. Autom. Syst*. Boston.

[3] Yang, Z. and Jianzhi, Z. 2011. Static Structural Finite-Element Analysis of Tower Crane Based on FEM. *IEEE Toronto International Conference*. 1-6.

[4] Xin, H. L., Ping, Q. and Bao, H. Z. 2008. The Finite Element Analysis of Cantilever Support in Foundation Pit. *2008 Int. Conf. Inf. Manag.* Berlin.

[5] Peng, L., Yingyun, H., and Guidong, K. 2003. Crankshaft Strength Calculation Based on Multi-body Systems Dynamic Analysis and Finite Element Analysis. *Int. Conf. Adv. Technol. Des. Manuf, 2003*. 1: 453-456.

[6] Yoo, S., Jun, B., Shim, H., Lee, P. and Kim, B. 2013. Finite Element Analysis of Carbon Fiber Reinforced Plastic Body Frame for Seabed Robot, Crabster200. *2013 IEEE*.

[7] Liaol, X., Gong, C., Lin, Y. and Wang, W. 2010. The Finite Element Modal Analysis of the Base of Welding Robot. *2010 3rd Int. Conf. Adv. Comput. Theory Eng.*

[8] Ding, B., Qian, J., Shen, L. and Zhang, Y. 2012. Finite Element Analysis and Optimized Design of Exoskeleton for Lower Extremity Rehabilitation Training. *Proceeding 2012 IEEE Int. Conf. Robot. Biometics*.

[9] Kim, G. 2001. Design of Two-axis Force Sensor for Robot's Finger. *ICASE Inst. Control. Autom. Syst. Eng. Korea*. 66-70.

[10] Structural Analysis Solutions-ANSYS. [Online]. From: http://www.ansys.com/Products/Simulation+Technology/Structural+Analysis.

[11] Jaafar, J. and Shauri, R. L. A. 2013. Three-fingered Robot Hand for Assembly Works. *2013 IEEE 3rd Int. Conf. Syst. Eng. Technol*. Aug 2013. 237-241.