International Journal of Applied Engineering Research ISSN 0973-4562 Volume 12, Number 20 (2017) pp. 10233-10241


# The Optimum Design of Micro Gripper for Lifetime Improvement Based on Fatigue Analysis and Six Sigma Analysis

**Pisut Boonkaew[1] and Jatuporn Thongsri[2*]**

*Computer Simulation in Engineering Research Group, College of Advanced Manufacturing Innovation, King Mongkut's Institute of Technology Ladkrabang, Bangkok, 10520, Thailand.*

*[1] ORCID: 0000-0001-8318-5132, [2]ORCID: 0000-0002-0957-4639*
*[2]Scopus ID: 26531958500*

**Abstract**

This article reports a problem in hard disk drive manufacturing (HDD) process and a practical solution. A traditional micro gripper of stainless steel 304, practically used in an HDD factory, has a low life expectancy. Therefore, the fatigue analysis was employed to solve this problem. Firstly, 27 cases of possible designs of micro gripper were proposed according to the full factorial design of six sigma ($6\sigma$) analysis. Each case is different regarding the micro grippers shapes which are degree, radius, and length. Secondly, 3D models of the micro gripper for all cases were created and transferred to the finite element analysis to simulate the equivalent stress and the compressive force. After the simulation completed, we found that the optimum design of micro gripper has 457.9 MPa of the equivalent stress and 0.991 N of the compressive force, which was less than those of the traditional design. Finally, the optimum design was brought to analyze in more details by using the fatigue analysis. The comparing results of simulation showed that the maximum equivalent stress and the compressive force of the optimum design compared to the traditional design were reduced 14.5% and 19.4%, respectively. These make approximately extending 136.5% of the life cycle and increasing 17.5% of the safety factor. This improvement was expected that it could be able to extend the lifetime of the micro gripper and save up to 60 million dollars annually.

**Keywords:** Finite element analysis, ANSYS, Fatigue analysis, six sigma, Micro gripper

## INTRODUCTION

Nowadays, a hard disk drive (HDD) is an electronic storage equipment which is widely used worldwide. One of the important parts in the HDD is a slider. The slider is a component suspended in a head gimbal assembly (HGA) which works as a read/write function. When an HDD is operating, the slider flies above a rotating disc. Air flows below the slider creating a lift force that assists the slider's flying. Before assembling to the HDD, the slider will be brought to a flying test to check a quality of slider and flying stability in the testing process. For support the testing process, the micro gripper was designed in a U-shape and was fabricated as a fixture for holding the slider. The proper micro gripper will support the flying test with the good stability and reliable result of the slider's testing quality process. On the other hand, an improper micro gripper might cause the distorted result of the slider's quality, also might cause the damage on the slider. Previously, in order to find a suitable force action on the slider of the micro gripper in the testing process, a new design of micro gripper will be tested by a trial and error process. However, it consumed a long-time improvement and wasting high cost. The way to solve this problem is employing the computer simulation to find a suitable methodology to improve the design of micro gripper and to extend the lifetime.

The methodology of this research has benefited from literature search of many researches. For example, Chakherlou et al. [1] reported the use of finite element analysis (FEA) and fatigue analysis (FA) in the experimental and numerical investigated the effect of clamping force on the fatigue behavior of bolted plates. They found a scheme to improve a lifetime of bolt plates as expected. The work of Yang and Guang [2] studied a 3D model and finite element analysis which were applied to simulate and investigate the stress and deformation on a printer gripper. They also found a methodology to reduce both stress and deformation to extend the lifetime of the printer gripper. Bassoli [3] investigated the effect of electro discharge (ED) drilling holes on the fatigue life of Inconel 718. In his work, the rotating bending fatigue tested was compared with the traditional cutting tool in the same condition. They found that the ED drilling causes a decrease of the fatigue resistance. According to the superposition effect principle which quantified the fatigue limit variation reported in this work, ED drilling causes an increase of stress concentration factor around 25% compared to traditional drilling. Schreiber et al. [4] used FEA to simulate fragmentation, impact and explosion behaviors of high strain-rate properties of materials for commercial and military applications. Their results were applied to reduce the number of experiments and expense needed to characterize how a material performs at high-strain-rate. Ali and Haneef [5] employed the FEA and FA to investigate the fatigue life of a connecting rod of piston and found ways to extend the lifetime. Computer simulation with

ANSYS has been widely applied to solve many manufacturing problems such as water condensation problem in clean room [6], particulate contamination problem in automated assembly machines [7-8].

Because of the FEA, FA and computer simulation are powerful tools for improving the equipment and solving manufacturing problems as mentioned above. Thus, they were applied together with the six sigma analysis in this research to determine the optimum design of micro gripper which has a longer lifetime.

## THERORETICAL BACKGROUND

Finite element analysis (FEA) is a numerical method for solving problems of engineering and physics. This methodology is applied to analyze a material or object to find out the effects that might be occurred when any load was applied. The advantage of FEA is to determine the weak point in a design before fabricated. The analysis is done by creating the mesh in the object, which breaks down a physical structure into substructures called "element". Therefore, the system of the simultaneous equation was established by the combination of equations in each element [9]. The precision of result depends on the amount and size of elements and boundary condition setting. The setting of interpolation functions of each element which can be written in the system equation as:

$$[K]_{sys}\{u\}_{sys} = \{F\}_{sys}, \quad (1)$$

where $[K]_{\mathbf{sys}}$ is element stiffness matrix. $\{u\}_{\mathbf{sys}}$ is the vector of nodal displacements and $\{F\}_{sys}$ is a vector of a nodal force of the system.

In each element, every variable must be calculated in a global coordinate system. Between two nodes, there is the modulus of elasticity ($E$), cross-sectional area ($A$), length ($L$) and force ($F$). When all these variable combines under the equivalent, it causes vector of nodal displacements $\{u\}$ at each point. The system matrix can be shown as

$$\begin{bmatrix} \frac{AE}{L} & -\frac{AE}{L} \\ -\frac{AE}{L} & \frac{AE}{L} \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \end{Bmatrix}. \quad (2)$$

Stress-stain relationship discusses material relationship for linear materials. The stress is related to the strains by

$$\{\sigma\} = [D]\{\varepsilon^{el}\}, \quad (3)$$

where $\{\sigma\}$ is a stress vector which in the components $x$, $y$ and $z$ written by $\left[\sigma_x \sigma_y \sigma_z \sigma_{xy} \sigma_{yz} \sigma_{xz}\right]^T$. $[D]$ is an elasticity or elastic stiffness matrix. $\{\varepsilon^{el}\}$ is an elastic strain vector [10]. To calculate all unknown parameters in the equations (1) - (3), it is very difficult and takes a long time to solve all equations. So, ANSYS mechanical simulation software will be the best equipment in use for the FEA. Once we have corrected the right setting, the result will be accurate and reliable.

Fatigue analysis is the analytical solution about the weakening of a material caused by the repeatedly applied load. The nominal maximum stress that causes the damage might be less than yield stress limit. But once, the material is subjected to repeated load and unload, it will begin to form the stress concentrators and gain interfaces. Eventually, a crack will reach a critical size, and the structure will be fractured [11]. For low-cycle fatigue, strain life was brought to use. The relationship of strain to stress, Neuber's rule was used. The equation of Neuber's rule was shown in (4)

$$\varepsilon\sigma = K_t^2 eS, \quad (4)$$

where $\varepsilon$ is a local (total) strain. $\sigma$ is a local stress. $K_t$ is an elastic stress concentration factor. $e$ is a nominal elastic strain and $S$ is the nominal elastic stress.

For the full factorial, this analysis is one of six sigma ($6\sigma$) methodology. It is a methodology focusing on the improvement of strategy to minimize defects and variations in the product design, production and administrative process [12]. In this study, three levels of three factors of degree, radius, and length will be analyzed by

$$3^{k-p} = the\ number\ of\ runs, \quad (5)$$

which $k$ represents the number of factors and $p$ is the number of design generators [13].

## METHODOLOGY

This section describes the experiment, FEA and Linear regression equation which are our previous work reported in reference [14].

### *Experiment*

The micro gripper was fabricated by designing as a U-shape to hold the slider for supporting the flying testing. It was designed as shown in figure 1. In the experiment, 10 micro grippers with the traditional design were tested in a load cell device which is controlled by a high precision motor to determine a relationship between opening displacement and compressive force. Opening the micro gripper caused the displacement which was converted to force and recorded in the load cell device. To measure the compressive force, the micro gripper was stretched out from 0 to 0.1 mm, 0.01 at a time. Hence, the compressive force in every 0.01 mm in each distance would be provided.

### *Finite element analysis (FEA)*

To determine the compressive force, 3D models of 7 micro gripper designs including the traditional design were simulated by using the FEA using ANSYS simulation software. Figure 2 shows the boundary condition setting in ANSYS consisting of fixed support points (A and C) and contact point (B). The fixed

support and the displacement were applied on the micro gripper model in ANSYS to simulate and represent the micro grippers movement. At the contact point, the frictional function is 0.62 which is the friction between stainless steel and stainless steel [15]. In mesh model, there are 64,682 elements and 83,307 nodes of hexahedral mesh with the maximum skewness of 0.82. The hexahedral mesh provides the most accurate result [16] than other mesh types. Using the FEA, the simulated results of compressive force and equivalent stress for all designs were obtained. We found that the equivalent stress of the traditional micro gripper was 535.78 MPa, where the compressive force was 1.183 N at distance of 0.1 mm. The critical area was observed that appeared in the angle area which may be a cause of short lifetime. After analyzing all data, it was concluded that degree (*D*), radius (*R*) and Length (*L*) are not proper. Therefore, the traditional micro gripper needed to redesign. The traditional micro gripper is a design which we believe that it will give the best result and suitable for remodeling.

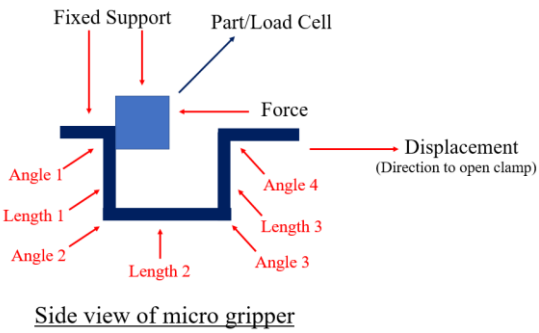


**Figure 1:** Side view of micro gripper and the free-body diagram.

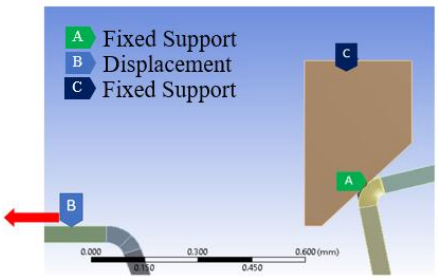


**Figure 2:** Boundary condition setting

### *Linear regression equation*

The linear regression equation was defined for purpose of predicting the compressive force which closed to the realistic. This methodology will provide a linear regression equation to present the relationship between the experiment and simulation by correlating the force together. Also, the tolerance between these two data can be emitted [17,18]. The linear regression equation expressed in (6),

$$Y = 1.4696X - 0.0564. \qquad (6)$$

where $Y$ is the predicted compressive force that might be provided in the realistic and $X$ is the compressive force from the simulation.

In this work, it was derived from the results of the experiment in previous section of experiment and FEA. This equation was employed for predicting the compressive force that might be represented as the actual compressive force by using the compressive force from the simulation as the variable. Comparing the predicting compressive force (*Y*) with the reference value of the factory, the error must not over ±30% which is acceptable following the factory's criterion. For example, at the displacement 0.1 mm, using equation (6), the optimum design gave 1.3998 N of the predicting compressive force which is in the factory' criterion; therefore, it was allowed to the fatigue analysis in the next section. Additionally, a brief workflow of our previous work and this work was shown in figure 3. This workflow shows our successful methodology to solve this problem.

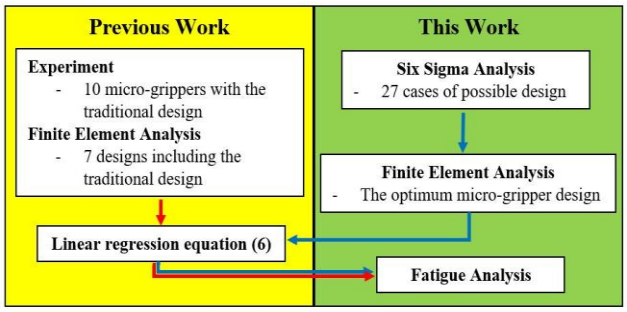


**Figure 3:** A methodology of workflow

### *Full factorial in six sigma Analysis*

As the conclusion mentioned in the section of FEA that the traditional design is not proper, the full factorial in six sigma analysis was applied to generate the possible cases regarding the shape of micro gripper such as degree (*D*), radius (*R*) and Length (*L*). A six sigma is a well-known methodology to improve the capability of the manufacturing. The best way of the improvement is about to increasing the performance and decreasing the process variation. The full factorial design is one of the equipment tools in six sigma which generally used with the two or more input values. This statistical tool will explore the output values of all possible combinations of input values. Once all possible cases were generated, then the 3D models following shape for each case were simulated by using the FEA to see the best design of reducing force and stress. In this research, we used $k = 3$, which are the number of factors related to the shape of micro gripper: increased 5% (denoted by +1), no change (denoted by 0) and decreased 5% (denoted by -1). The number of design generator was none ($p = 0$). The degree (*D*), radius (*R*) and length (*L*) constrained the shape of micro gripper were generated from the Key Process Input Variable (KPIV). Using equation (5) and information mentioned above, there were 27 cases to run the simulation, excluding the

repeated sequence. All boundary conditions were set with the same way as the traditional design setting. After the simulation finished, the simulated results of the equivalent stress of the micro gripper for all cases using ANSYS Mechanical software were recorded and reported as seen in Table 1.

The equivalent stress is a principal factor affected on the lifetime of the micro gripper; therefore, the design of micro gripper in the run number 21 (marked) is the optimum design since it has the minimum equivalent stress. This design is decreasing 5% of both degree and length (coded -1/0/-1) in which the maximum equivalent stress appeared in the critical area of angle as seen in figure 4. Using the shape of the optimum design, the 3D model was created, and then the FEA was applied to simulate the compressive force and the equivalent stress. The simulated result was compared with that of the traditional result. The comparison of equivalent stress was shown in figure 4. The maximum equivalent stresses of the traditional and optimum designs were 535.78 MPa and 457.90 MPa, respectively. The simulated results of the compressive forces for the traditional and optimum designs at displacement 0.1 mm, were 1.183 N and 0.991 N, respectively. The comparison between compressive forces of the traditional and the optimum designs at any distance was shown in Table 2. The optimum design gave the compressive force in the factory's criterion which was verified by equation (6) for all distances. Hence, the optimum designs will bring to the fatigue analysis to prove that this design is the best design of micro gripper.

**Table 1:** The equivalent stress for all possible cases

| Run Order. | Degree | Radius | Length | Stress(MPa) |
|---|---|---|---|---|
| 1 | 1 | 1 | -1 | 525.77 |
| 2 | -1 | 0 | 0 | 460.12 |
| 3 | -1 | -1 | 0 | 600.48 |
| 4 | 0 | 1 | -1 | 468.56 |
| 5 | 0 | -1 | 0 | 462.01 |
| 6 | 0 | 0 | -1 | 516.94 |
| 7 | 1 | -1 | -1 | 511.88 |
| 8 | 0 | 0 | 0 | 535.78 |
| 9 | -1 | 1 | -1 | 606.41 |
| 10 | 1 | -1 | 1 | 470.99 |
| 11 | 1 | 1 | 1 | 559.27 |
| 12 | -1 | 1 | 0 | 464.8 |
| 13 | 1 | -1 | 0 | 511.88 |
| 14 | 1 | 0 | 0 | 517.16 |
| 15 | 0 | -1 | 1 | 514.26 |
| 16 | 1 | 0 | -1 | 535.32 |
| 17 | 1 | 1 | 0 | 553.60 |
| 18 | -1 | 1 | 1 | 606.41 |
| 19 | 0 | -1 | -1 | 516.38 |
| 20 | -1 | -1 | -1 | 582.64 |
| 21 | -1 | 0 | -1 | 457.9 |
| 22 | 0 | 1 | 1 | 461.52 |
| 23 | 1 | 0 | 1 | 504.29 |
| 24 | 0 | 1 | 0 | 595.62 |
| 25 | -1 | -1 | 1 | 582.53 |
| 26 | 0 | 0 | 1 | 525.81 |
| 27 | -1 | 0 | 1 | 583.10 |

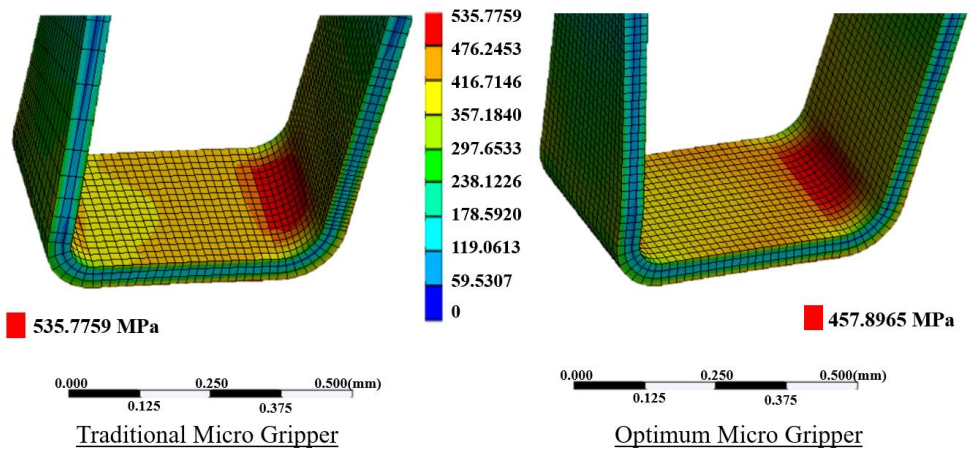


**Figure 4:** Equivalent stress of traditional micro gripper (left) and optimum micro gripper (right)

**Table 2:** Result of force

| Position (mm) | Compressive force (N) | | % Improvement | Results verified by equation (6) |
|---|---|---|---|---|
| | Traditional Micro gripper | Optimum Micro gripper | | |
| 0.0100 | 0.1121 | 0.1005 | -11.6211% | Pass |
| 0.0200 | 0.2289 | 0.1973 | -16.0119% | Pass |
| 0.0300 | 0.3450 | 0.2999 | -15.0474% | Pass |
| 0.0400 | 0.4611 | 0.3969 | -16.1641% | Pass |
| 0.0500 | 0.5793 | 0.4939 | -17.2790% | Pass |
| 0.0600 | 0.7058 | 0.5967 | -18.2805% | Pass |
| 0.0700 | 0.8234 | 0.6939 | -18.6625% | Pass |
| 0.0800 | 0.9443 | 0.7910 | -19.3848% | Pass |
| 0.0900 | 1.0632 | 0.8938 | -18.9551% | Pass |
| 0.1000 | 1.1835 | 0.9909 | -19.4337% | Pass |

## RESULTS AND DISCUSSION

In this research, the fatigue analysis was applied to investigate hysteresis curve, life circle, fatigue sensitivity and safety factor of the traditional and optimum design of micro gripper using the boundary condition for displacement at 0.1 mm. Figure 5 show a boundary condition setting in ANSYS mechanical software for this purpose.

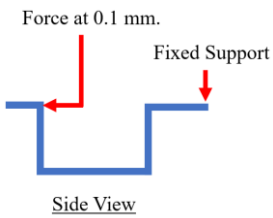


**Figure 5:** Boundary condition setting

In conducting numerical simulation, the bilinear kinematic hardening material model was used to simulate the hysteresis curve represented the stainless steel 304 stress-train behavior. In fact, there is no measurement that can make the tensile test which is an important data input. Therefore, the comparison of life time will be used as a reference between simulation and the manufacturing data, which the life time from simulation shown that it was nearby the actual one, approximately 16000 runs. The hysteresis curve will show the yield information on fatigue degree, as well as a stress-strain characteristic curve with the fact that time variation of load increase fatigue, decrease strength of mechanical and, implicitly, modification of hysteresis loop [19,20], as the figure 6. The life circle of the simulation was presented in fatigue 7. The minimum life circle is in the corner area of micro gripper. This also confirms the report from the factory that the critical area having the maximum equivalent stress is commonly observed in the corner. The minimum life circles of the traditional and optimum design are 16,796 and 39,729, respectively. This implies that the optimum design increases 136.5% of life circle.

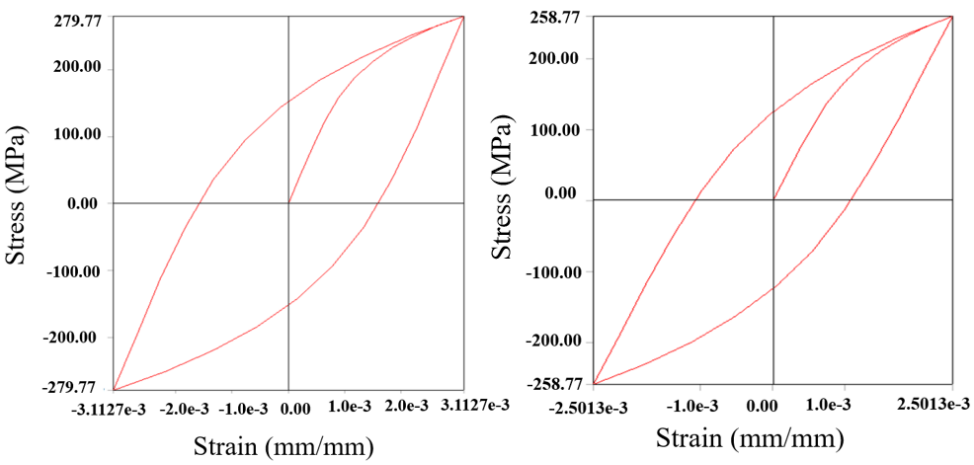


**Figure 6:** The hysteresis curve of the traditional micro gripper (left) and the optimum micro gripper (right)

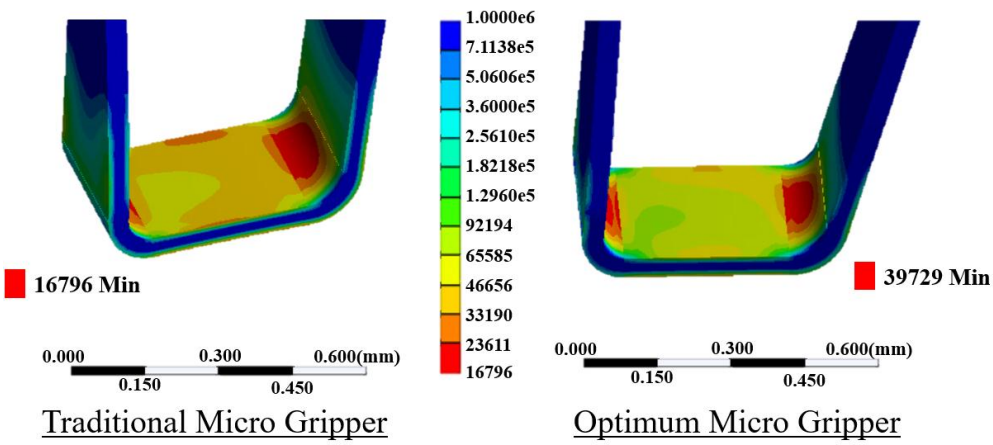


**Figure 7:** Life cycle of traditional (left) and optimum micro gripper (right)

A comparison of the fatigue sensitivity on both designs was reported as a graph in figure 8. The fatigue results changed as a function of the loading at the critical point on the model. The study of the lifetime sensitivity was varied from 50% to 150 % of the load. This graph can be seen that once the load was applied by 50% of the traditional load, the life cycle of both micro grippers can run up to $1.00 \times 10^6$ cycles. In the other hand, if the load was applied 150%, the life cycle of the traditional micro gripper will be 2376.6 cycles while the optimum micro gripper is 4857.4 cycles.

The design circle life condition was limited as $1.00 \times 10^6$ cycles in which the load was applied by the compressive force at a distance 0.1 mm. In figure 9, the results of the safety factor for both designs were shown. The minimum value also observed at the corner area of the micro gripper as expected since this area has the maximum equivalent stress. The minimum safety factors are 0.51885 and 0.61072 for the traditional and the optimum designs, respectively. This result showed that the safety factor was increased up to 17.7%. To converted life cycles to the amount of daily use, the optimum micro gripper can be utilized for 36 years, 3 months and 2 days, the data was calculated with the UPH (units per hour) recorded data from company owner in July 2017. According to the results of hysteresis curve, life circle, fatigue sensitivity, safety factor and the report from factory including the research methodology as described in this article, we are quite confident that the optimum design of micro gripper which is reduced 5% of both degree and length from the traditional design, gives the best performance for further development to practical use in real testing process of the HDD factory.

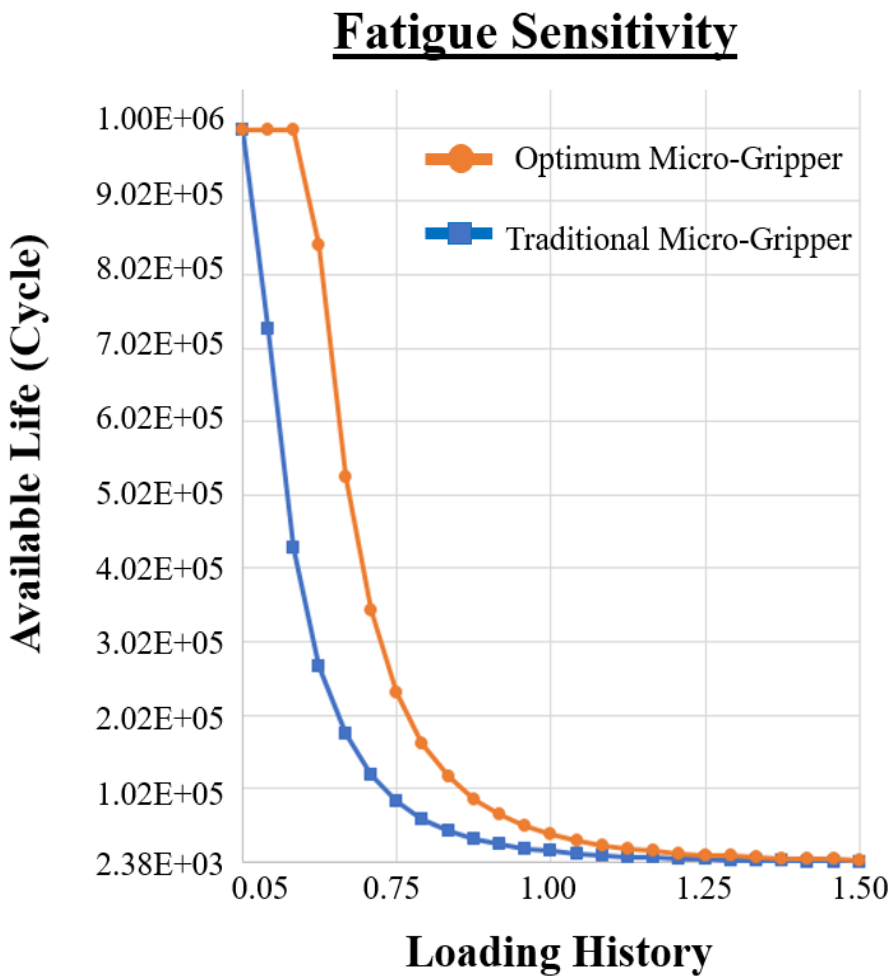


**Figure 8:** The comparison of fatigue sensitivity

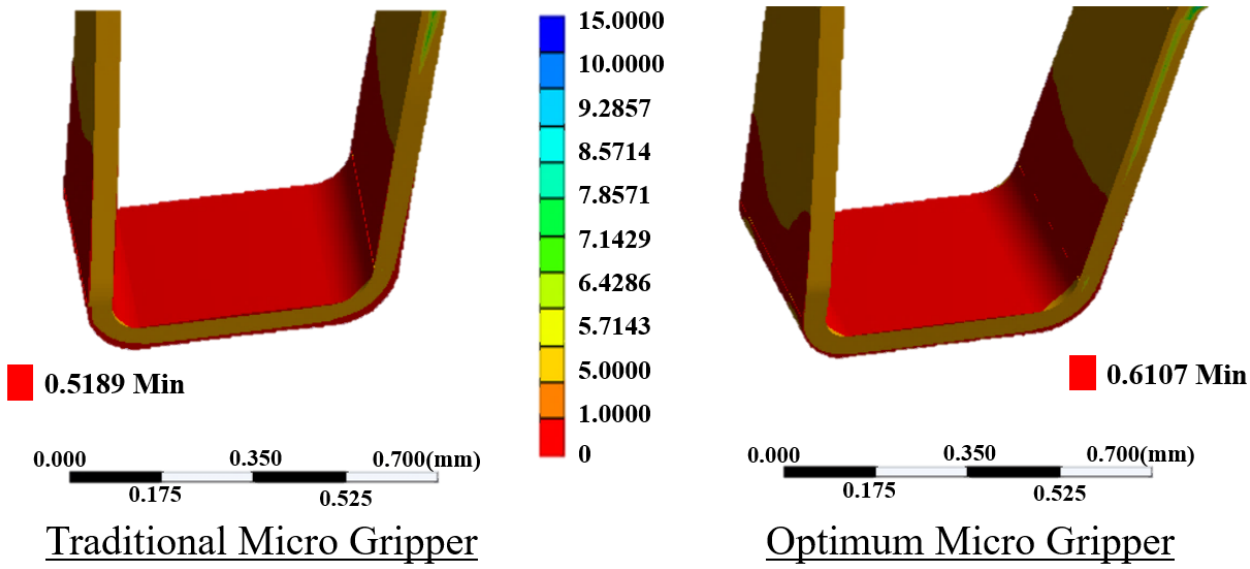


**Figure 9:** Safety factor of traditional micro gripper (left) and optimum micro gripper (right).

## SUMMARY

In a hard disk drive (HDD) manufacturing factory, a micro gripper was designed as a fixture tool for holding the slider in the testing process; however, it has a short lifetime usage. Previously, to research to find the optimum design of micro gripper with the acceptable compressive force and has longer lifetime, the micro gripper was tested by trial and error methodology. However, it consumes a long-time period and wasting high cost. Therefore, the fatigue analysis and six sigma analysis were employed in this research as important tools to improve the design. This article was intended as a practical solution to extend the lifetime of micro gripper. Firstly, a micro gripper was used to simulate the equivalent stress and the compressive force based on finite element analysis (FEA) by the ANSYS Mechanical simulation software. The FEA revealed that the maximum equivalent stress and the compressive force of traditional design were 535.8 MPa and 1.183 N, respectively, which may cause a short lifetime of the micro gripper. The critical area with the maximum equivalent stress is in the corner areas of micro gripper, which was confirmed by report from the factory. Secondly, to extend the lifetime, 27 cases of possible designs which excluding the repeated designs were proposed according to the full fractional rule of six sigma (6σ) analysis. Each case is different regarding the micro gripper shapes such as degree, radius, and length. The 3D model for all cases was created and transferred to the FEA to calculate the maximum equivalent stress and the compressive force. The simulation results for all cases were obtained. It was found that the best optimum design is the design that reduced 5% of both degree and length of micro gripper, which has the minimum equivalent stress and compressive force at 457.9 MPa and 0.991 N, respectively. Using the optimum design, the maximum stress and the compressive force were reduced 14.5% and 19.4%, respectively, compared with the traditional design. Finally, the optimum design was brought to the fatigue analysis. These make approximately extending 136.5% of life circle and increasing 17.5% of the safety factor. This finding was submitted to the HDD factory and was accepted for further improving the testing process. It was expected that would be able to extend the lifetime of the micro gripper and save up to 60 million dollars annually.

## ACKNOWLEDGMENTS

This research was financially supported by a contract no. MSD5810073 of the Research and Researcher for Industry (RRI) under the Thailand Research Fund (TRF). Facilities and miscellaneous materials were supported by College of Advanced Manufacturing Innovation (AMI) and King Mongkut's Institute of Technology Ladkrabang (KMITL).

## REFERENCES

[1] Chakherlou, T.N., Oskouei, R.H., and Vogwell, J., 2008, "Experimental and numerical investigation of the effect of clamping force on the fatigue behaviour of bolted plates," *Engineering Failure Analysis*, 15(5), pp. 563-574.

[2] Zhang, Y., and Li, G., 2009, "ANSYS-based finite-element analysis and mechanical analysis of printer gripper," *In proceeding of International Forum on Computer Science-Technology and Applications*, pp. 414-416.

[3] Bassoli, E., 2017, "Fatigue life of electrodischarge drilled Inconel 718," *International Journal of Applied Engineering Research*, 12(14), pp. 4067-4071.

[4] Schreiber, J.M., Smid, I., Eden, T.J., and Jann, D., 2015, "Prediction of fragmentation and experimentally inaccessible material properties of steel using finite element analysis," *Finite elements in analysis and design*, 104, pp.72-79.

[5] Ali, H.M.M., and Haneef, M., 2015, "Analysis of fatigue stresses on connecting rod subjected to concentrated loads at the big end," *Materials today: proceedings*, 2(4-5), pp.2094-2103.

[6] Thongsri, J., 2017, A successful CFD-based solution to a water condensation problem in a hard disk drive factory," *IEEE Access*, 5, pp. 10795-10804.

[7] Thongsri, J., 2017, A problem of particulate contamination in an automated assembly machine successfully solved by CFD and simple experiment," *Mathematical Problems in Engineering*, 2017, Article ID 6859852, 9 pages.

[8] Thongsri, J., and Pimsarn, M., 2015, "Optimum airflow to reduce particle contamination inside welding automation machine of hard disk drive production line," *International Journal of Precision Engineering and Manufacturing*, 16(3), pp. 509-515.

[9] Dechaumphai, P., "Finite element method in engineering", *Chulalongkorn University Press*, Edition 4th, 2007.

[10] Zienkiewicz, O.C., and Robert, L.T., "The finite element method", *McGraw-Hill*, Vol. 1, 2000.

[11] Kim, W.H., and Laird, C., 1978, "Crack nucleation and state I propagation in high strain fatigue-II mechanism", *Acta Metallurgica*, 26(5), pp. 789–799.

[12] Adan, V., Jaime, S., Salvador, N., and Berenice G.N., 2009, "Implementation of six sigma in a manufacturing process: A case study", *International journal of industrial engineering*, 16(3), pp. 171-181.

[13] Teruhiro, S., 1993, "Factorial factorial designs of two and three levels", *Discrete mathematics*, 116(1-3), pp. 99-135.

[14] Boonkaew, P., and Thongsri. J., 2017, "Effect of clamp structural on the compressive force behaviour based on finite element analysis", *in Proceeding of the 32nd International Technical Conference on Circuits/Systems, Computers and Communications (ITC-CSCC)*, pp. 361-364.

[15] Hsu, K. L., Ahn T. M., and Rigm D. A., "Friction, Wear, and Microstructure of Unfubricated Austenitic Stainless Steel", *ASME Wear of Material*, 1979.

[16] Benzley, S.E., Perry, E., Merkley, K., and Clark B., 1995, "A comparison of all hexagonal and all tetrahedral finite element meshes for elastic and elasto-plastic analysis", *In Proceeding of the 4th International meshing roundtable Sandia National Laboratories*, 17, pp.179-191.

[17] Miller, J., and Haden, P., "Statistical Analysis with the general linear model", *Creative Commons*, pp. 119-204, February 2013.

[18] Rencher, A.C., and Schaalje, G.B., "Linear models in statistics", *John Wiley & Sons*, Edition 2nd., pp. 127-227, 2008.

[19] Köksal, N.S., Kayapunar, A., and Çevik, M., 2013, "Fatigue analysis of a notched cantilever beam using ANSYS workbench", *In Proceeding of the 4th International Conference on Mathematical and Computational Applications*, pp. 111-118.

[20] FRUNZĂ G., and Diaconescu E.N., "Hysteresis and mechanical fatigue", *The annuals of university Y "DUNĂREA DE JOS" OF GALAŢI*, ISSN 1221-4590, 2006.