Fatigue and Model Analysis of the CNC Cylindrical Grinder

Jui-Chang Lin [1], Cheng-Jen Lin [2,a]

[1]*Department of Mechanical Design Engineering, National Formosa University, Huwei,Taiwan, R.O.C*
[2]*Institute of Mechanical and Electro-Mechanical Engineering, National Formosa University, Huwei,Taiwan, R.O.C*

Abstract. The purpose of this study is to lower deviation of workpiece by meeting high stability and rigidity to prevent the resonance in producing procedure of the CNC universal cylindrical grinding machine. Using finite element analysis software ABAQUS in grinder machine tools for numerical simulation of several analyses for the following: structural rigidity analysis, optimized design, vibration frequency analysis and fatigue damage analysis. This work aims on state of the transmission of outer diameter spindle to proceed in stress and fatigue life analysis by FE-SAFE Subroutine. The max values of equivalent stress and average amount of displacement in structural rigidity analysis are 0.67(Mpa) and 0.92(μm). Optimization design effectively reducing extreme value of stress, the largest decline of about 5.43%. Modal analysis compared with the experimental, the average error percentage was less than 10% of parts. The whole structure error does not exceed 3%. The fatigue life of approximately 1,193,988 times, estimates into real life time can use more than sixty years, from the viewpoint of structural strength, spindle has a good high breaking strength is designed to be safe.

1 Introduction

In general, the influential factors of CNC machine accuracy are structural design, assembly errors and component machining problems. The structural vibration and deformation will affect CNC machine structure, the main reason is insufficient strength which is caused by improper reduction weight, some components of machine will shake during machine operation, in some situation the structural resonance may be happened, the machining quality must be influenced too. Therefore, experimental mode analysis and finite element analysis is necessary for structural design of CNC grinding machine. The improvement of the structural design errors, the avoidance of the vibration frequency and the structural rigid problems were an imperative task. FEM is one of the current analytic methods, which is obtained by numerical solution of engineering problems similar analysis.

FEM concept is to analyze the object which was meshed and cut into the appropriate size, number and shape, and then the basic theoretical principles (mainly elastic-plastic theory) combined with the material properties, the force conditions, and the boundary conditions, applying the limited cumulative iterations to solve the element and finally obtain the numerical solution. Finite element analysis software ABAQUS is widely used in automobiles, civil engineering, aerospace and mechanical industries. ABAQUS/CAE can analyze various mechanics problems from linear structure to non-linear material stress and strain including vibration and deformation. [1-4]

2 Theory

Mechanical systems are usually composed of several parts; the interaction is existed each other. A simpler physical model is represented in order to study the vibration characteristics of the actual mechanical system. Mechanics or mathematical theorems are applied to establish the physical model expression. A simple vibration system usually includes (1) spring or elastomer of stored potential energy; (2) the mass or inertia of stored kinetic energy; or (3) consumption of energy damping. When general structure of the vibration system can't be simplified to a point mass, the system will be treated as infinite degrees of freedom of the continuum. The FEM is used to solve equation of motion or equations of force for the complicated structure.

Vibration is the object which is treated as a reference point and moved gently back and forth with a small range of movement under the static equilibrium, so the vibration of the object is a regular cycle of movement. Four-bar linkage, slider-crank, swing and other periodic motion are not the so-called vibration. They only can be treated as dynamic system problems. In general, the vibration can be divided into two types: free vibration and forced vibration. Free vibration system is generated by the vibration exciting force in the initial conditions or after the abolition of the existing external vibration.

[a] Corresponding author: lcj@nfu.edu.tw

Forced vibration is caused by periodically or intermittently external forces. The two types of vibration can be a damping or undamped vibration, friction effect in the undamped vibration system is ignored; it can keep the vibration forever, but in reality the internal friction and external friction is exist, all objects are damping vibration.

Frequency is variable because of the structure shape, size, material properties and even the boundary conditions of the structure. Mode parameters is unique, constant, can be regarded as the characteristics of the system.

3 Static structural analysis for grinding machine

In this paper, FEM software was used to analyze the static structure of CNC cylindrical grinding ma-chine, including machine head, machine bed, and spindle. In the static analysis, the main structure analysis was made after the main structure assembly was finished, inspecting the stress changes and the deformation under the grinding load to find out the maximum strain of the mechanical structure, and enhance its structural rigidity by optimization.

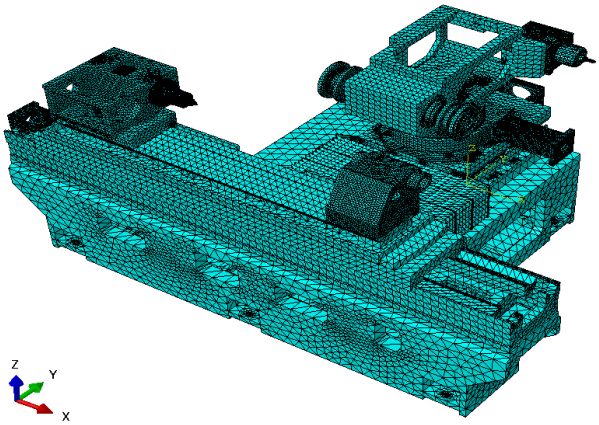

Figure 1. CNC grinding mechanical structure.

Table 1. The mechanical properties of machine material.

Material	Cast iron (FC300)
Density (ton/mm^3)	7.2×10^{-9}
Young's modulus (Gpa)	110
Poisson's ratio	0.28
yielding strength (MPa)	200

The Fig.1 showed the main structure of complex CNC cylindrical grinding machine including body, bed, head and other parts. The material of CNC cylindrical grinding machine is cast iron (FC300), the mechanical properties include density, Young's modulus, Poisson's ratio and yielding stress as listed in Table 1. The loading and support portion is added in the structure according to the actual situation and establishment of the basic conditions. Solving the results, the maximum displacement of the head of grinding machine is 0.016mm, and the maximum stress is 2.5MPa. However, the maximum stress at the grinding machine base is lower than the yielding strength of cast iron FC300 200MPa. The base structure of grinding machine does not produce permanent deformation owing to its grinding load, and reduce its precision and stability.

4 Grinding machine dynamical analysis and experiment

This study used finite element method to simulate the frequency response of complex CNC internal and external grinding machine, the frequency response was also actually measured to verify the accuracy. Therefore, the weakness of grinding machine structure will be known, in the working condition with spindle revolution, the resonance frequency should be avoided, and improved the machine structure.

4.1 Comparison between experiment and FEA

After modes experiment and modes analysis by FEA, the results showed that experiment and analysis was coincidence, the error was within 12 %, showed the comparison of modes analysis of whole machine in Fig.2. The rotational speed of complex CNC grinding machine is 30000 RPM, the corresponding frequency is 500 Hz. The figures of analysis model indicate that smaller parts have higher frequency, the first mode of the small most parts are larger than 500 Hz. It is evident that the most parts do not have vibration problems.

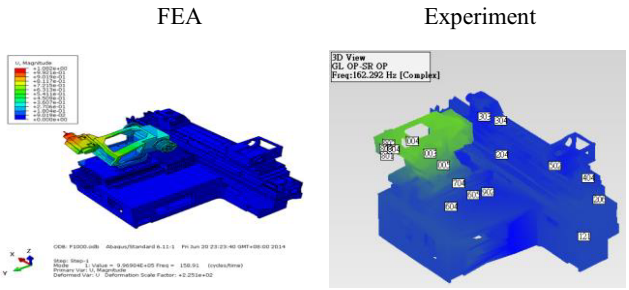

Figure 2. CNC grinding mechanical structure.

4.2 Fatigue estimate by finite element analysis

The finite element analysis (FEA) software ABAQUS was applied to analyze the fatigue of complex CNC grinding machine, including life of whole machine assembly structure. After modes of spindle analysis by FEA, the results showed in Fig.3. The fatigue life of approximately 1,193,988 times, estimates into real life can use more than sixty years, from the viewpoint of structural strength, spindle has a good high breaking strength is designed to be safe.

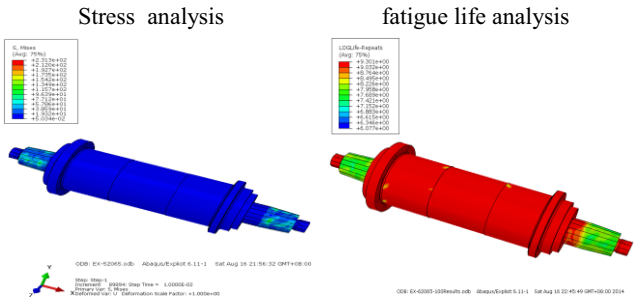

Figure 3. Fatigue analysis of spindle.

5 Topology optimization design

In the further research, topology optimization design may use to reduce the net weight of grinding machine in the case of without derogation strength. The process of optimization analysis need to make some restrictions in objective function, volume constraint and geometry, the structural symmetry factors also should be considered together. The element after topology optimization design should do the same structural strength analysis and compared with the original element. The results can be used to determine whether the topology optimization design to achieve the desired function.

This study makes first choice of outer wheel holder for topology optimization design. After the foregoing related settings of optimization analysis to reduce the weight of element, and execute related analysis of structural strength afterwards. The figure 4 showed the difference of stress and figure 5 shown the change of displacement. In brief the effect of topology optimization design, material was removed 15%, stress from 1.514(Map) down to 1.503(Mpa) and displacement from 2.00(μ m) increase to 2.033(μ m). These results are not too poor. But the outcome of the analysis of a single element can't determine the effect of optimization. The optimization design after the overall consideration of whole grinder structure, the final result is the basis of the feasibility of the topology optimization design.

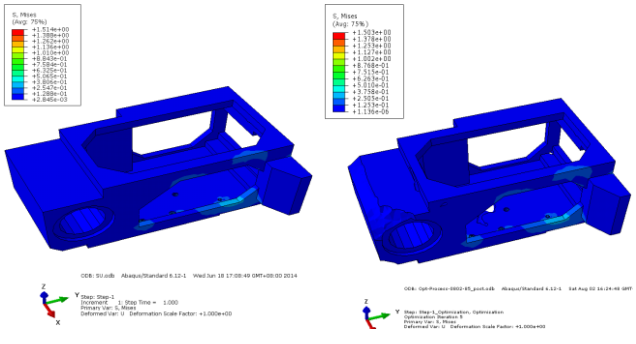

Figure 4. The comparison of stress of outer wheel holder before and after optimization design.

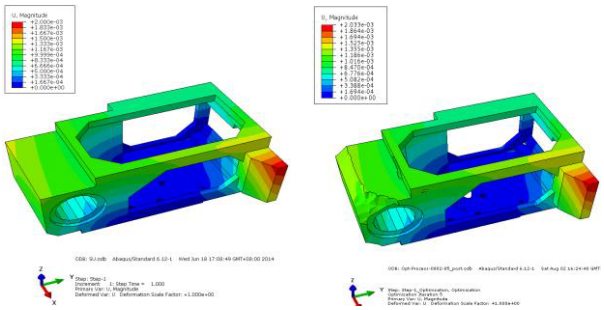

Figure 5. The comparison of displacement of outer wheel holder before and after optimization design.

6 Conclusion

By CAE software to improve the strength of the CNC machine structure, not only to reduce the cost and time of the CNC machine experimental, can be used as the reference of the development of new CNC grind machine structure design.

References

1. Ramezanali, Mahdavinejad, Finite element analysis of machine and workpiece instability in turning. International Journal of Machine Tools & Manufacture, **45**, 753-760 (2005)
2. J. A. Ewing, Humfrey, J. C., The Fracture of Metals under Repeated Alterations of Stress, Philosophical Transaction of the Royal Society, London, **A200**, 241-250 (1903)
3. J. C. Newman, Jr., A Finite Element Analysis of Crack Growth under Monotonic and Cyclic Loading, ASTM STP **637**, 56-80 (1977)
4. N. A. Fleck, Finite Element Analysis of Plasticity-Induced Crack Closure under Plane Strain Conditions, Engineering Fracture Mechanics, **25**, 441-449 (1986)