58th ICMD 2017
6 - 8 September 2017, Prague, Czech Republic

THE CONCTACT ANALYSIS OF DESIGNING THE CAM MECHANISM BY USING ANSYS

Ha NGUYEN VAN[1], L. ŠEVČIK[2]

[1,2]Department of Design of Machine Element and Mechanism, Faculty of Engineering, Czech Technical University of Liberec, Studentska 1402/2, Liberec I, 461 17, Czech republic

Abstract
A cam mechanism is used to transmit rotary motion between two concentric shafts which was built by CATIA V5-R16 software. In this model, the cam mechanism is composed by rotary input cam shaft, a rotary output cam shaft, a middle part and two balls and a frame to mount all the parts. Due to the contact between the balls with their grooves on each part of the cam mechanism is very important, it could directly affect the lifespan and working ability of the structure. This paper mainly focuses on the determination of contact stress between the balls and straight grooves on output cam shaft, middle part as well as circular groove on the input camshaft by using fine element analysis. Through contact analysis, the changes could be shown in stress, strain, penetration, friction stress among the groove of the input, output, middle part and balls. Furthermore, the simulation results revealed that the computational values were consistent with theoretical values. It would provide a scientific basis for optimum design of the cam mechanism.

Key words: *cam mechanism, ball transmission, contact analysis, ANSYS.*

INTRODUCTION

Cam mechanisms are widely used in many types of modern machines because of their excellent property for operation speed, motion accuracy, structural rigidity and low production cost. Generally, cam mechanism used steel balls to transmit rotary motion between two concentric shafts, the main function of the balls is the transmission of forces and motion from the input cam shaft into the output cam shaft. So, when cam works each ball can be easily to move up and down with respect to their straight groove on the output cam shaft and middle part correspondingly. Moreover, these balls must move along with the circular groove on the input cam shaft. Therefore, the surface contact between the balls with their grooves is always changing when cam working. This contact phenomenon could be caused to make the balls can damage easily and result is failure of these balls [1, 5]. Therefore, in this work to study the stiffness, contact stress and deformation of the balls to optimize the cam mechanism.

Contact finite element analysis can show the information of the cam mechanisms under contact stress, strain, penetration and so on, which play a significant role in the optimum design of cam mechanism. Therefore, analysis of contact problem is a major concern in many engineering applications such as ball bearing, gears and pressure vessel attachment. The numerical modeling of practical contact problems requires special attention because the actual contact area between the contacting bodies is usually not known in advance. With the change of load, material, boundary condition or the other factors, touch or separation will take place between the surfaces. That is hard to predict, even is a abrupt change. Most frictional effects on the contact problems are needed to be considered. It may be disordered as well as nonlinear. So, ANSYS gives a good blue print for contact analysis which can take friction heat and electrical contact into account. It also has special contact guide which is conveniently for creating contact pairs. The internal expert system of contact analysis does not require any settings of related contact parameter in a general contact analysis. So, it can easily establish the contact analysis. This work is to study the contact between the balls and the grooves in the cam mechanism. Therefore, the mechanism has designed as an example and it discussed about the contact and built its finite element 3-D parameterized model by using Finite Element Analysis software ANSYS Workbench 18 and CATIA V5-R16. Based on these results, the nonlinear contact state was researched and analyzed.

MATERIAL AND METHODS

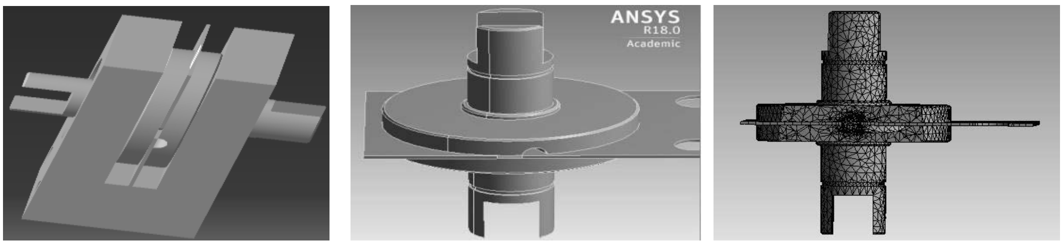
Assume that contact between the ball and circular groove is like contact of a sphere on a sphere and contact between the ball and straight groove is like contact of a sphere on a plat-plate. Hertz elastic contact theory has solved the calculation problems of contact stress and deformation of the balls and their grooves of the cam mechanism designed successfully. It used to make following assumptions to solve the contact shape and dimension and surface pressure distribution of elastic solids. The first, the objects contact with each other which only produce elastic deformation obey Hook's law. The second, the contact surface is smooth, which only have the effect of the normal force. The third, contact size is much smaller than the size of curvature radius of the contact bodies' surface. Contact problem of ball and groove basically corresponds with the Hertz assumption [2].

The basic steps in the cam mechanism's contact analysis are performed. In contact problem involved two boundaries, it is natural that take on boundary as target surface and take the other on as contact surface. Surface-surface contact is very suitable for those problems just as: interference fitting installation, or embedded contact, forging and deep-drawing. Surface-surface contact's analysis has many steps which include:

1) Build 3D geometry model and mesh;
2) Identify contact pairs;
3) Name target surface and contact surface;
4) Define target surface;
5) Define contact surface;
6) Set up element key options and real constants;
7) Define and control rigid goal's movement(only applicable in rigid-flexible contact);
8) Apply the necessary boundary condition;
9) Define solution options and load steps;
10) Solve contact problems, and look over and analyze results.

ANSYS software supports the surface contact element of rigid-flexible or flexible-flexible. The elements form contact pairs by using target and contact surface. For the rigid-flexible contact, it can be chosen as contact surface such as convex surface, dense meshing or little size surface, otherwise chosen as target surface. Under the condition that the model should be simplified as far as possible, so that the computational time could be reduced, simply and accurately reflects the mechanical property of the solid model. By use of CATIA the model is built (see Fig 1 a) [3, 4] and imported into ANSYS. In theory, it is feasible that model can be changed, such as material property, retrained displacement and applied load, and so on. Therefore, in this case the cam mechanism model which is created and imported into ANSYS without frame and bearings part. So, now the cam mechanism in ANSYS workbench is composed of the input cam shaft, output cam shaft, middle part and balls (see Fig 1b). The material of all parts made by steel, the elasticity model is 2 E5 MPa, the Poisson's ratio is 0.3. The meshing type is free, element size is set 0.5 mm for each contact pairs are shown in Fig 1 c.

In this work, the contact pairs are frictional between ball, groove and so on. Taking separately the groove surface of the input cam shaft, output cam shaft and middle part as target surface and taking correspondingly sphere surface of the balls as contact surface, two contact pairs can be built. It is necessary that to make sure the contact is rigid-flexible contact between balls and groove of each part, to set 0.15 as frictional factor value. Next step to set boundary condition and apply loads: Boundary condition restrained the all degrees of freedom(DOF) of the middle part (fix support), added frictionless support constraint to the input and output cam shaft, and applied moment load to the input cam shaft is given 1000 N.mm.



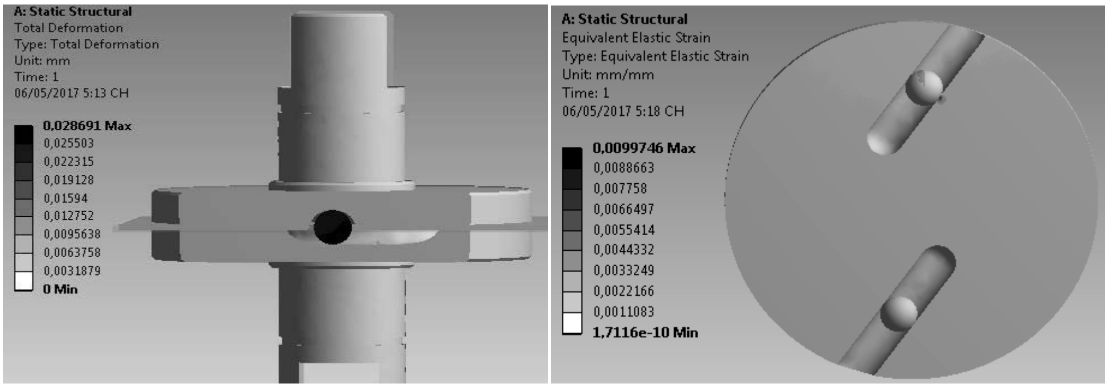
a b c

Fig. 1 (a) Cam mechanism assembly model, (b) the imported model in Ansys, c) meshed model

RESULTS AND DISCUSSION

According to the result was made by ANSYS, the phenomenon would be found that, the contact change status can be given, such as contact stress, strain, penetration, contact pressure among the input, output cam shaft and the balls, middle part.

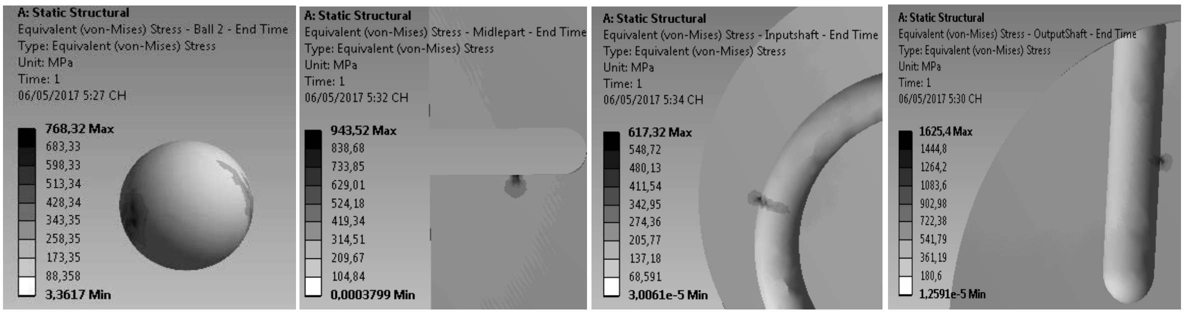
Fig. 2 a, b shows von-Mises total strain. The biggest total displacement and strain of cam mechanism respectively is 0.03 mm, 0.01 mm. The bigger contact displacement mainly concentrated on the balls, and the straight groove of the output cam shaft.



a b

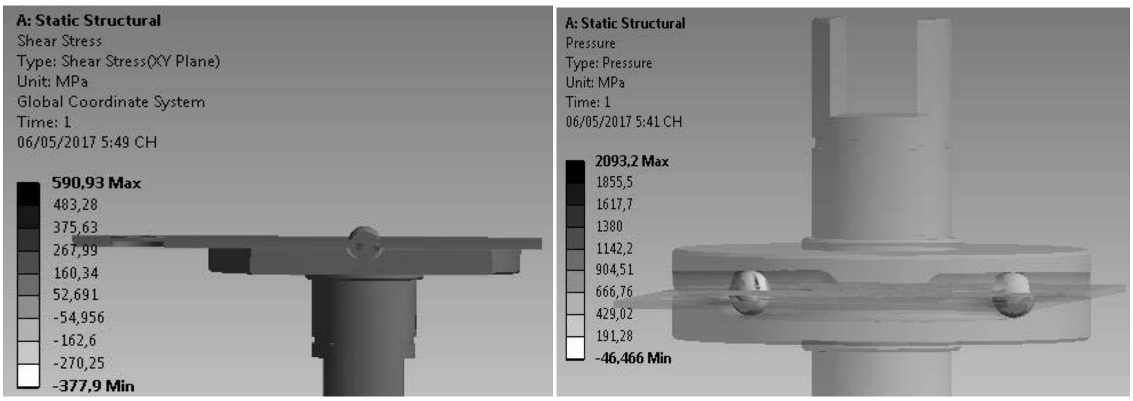
Fig. (2a) Deformation cam mechanism, (2b) equivalent strain cam mechanism

From Fig 3, it is victimized that the contact area had an approximate ellipse shape in contact area of the balls and the grooves, which was consistent with the Hertzian contact theory.



a b c d

Fig. 3 Equivalent stress of cam mechanism, (a) view on ball, (b) view on the groove of the middle part, (c) view on the circular groove of the input shaft, (d) view on the straight groove of the output shaft.



a b

Fig.4(a) Shear stress cam mechanism, (b) pressure stress cam mechanism.

Fig 4 shows the simulation of the calculation results of the maximal contact shear stress and contact pressure was 590 MPa, 2093 MPa respectively, while the Hertzian theory value was 651 MPa, 2100

MPa [3, 4]. The comparison revealed that there was good consistency between the Hertzian theory solution and finite element solution.

CONCLUSIONS

By using ANSYS 18 software to numerically simulate and analyze the stress and strain during cam mechanism contacts, the fine element solutions were shown the results, which had good consistency with the Hertzian theory. The contact analysis of finite element method can easily and intuitively get the stress and strain value as well as their cloud imagery, which can efficiently understand the parts running information, such as contact penetration, contact stress also. The results could provide a few of original ideas and would have a significant assistance for the life-design and structural optimum about cam mechanism.

ACKNOWLEDGMENT

In this occassion we would like to thank, Technical University of Liberec for providing the fund for this work under the project "Innovation of products and equipment in engineering practice" with the support of the specific University Research Grant, as provided by the Ministry of Education, Youth and Sports of the Czech Republic for the year of 2017.

REFERENCES

1. Gopalakrishnan, T., Ruban, M. (2015). Contact analysis of roller bearing using finite element method. Vels Journal of Mechnical Engineering,Vol-2, 30-33.
2. Norton, R. L.(2006). Machine Design an integrated approach, third edition. Pearson. ISBN 013-202012-2.
3. Nguyen Van, H., Sevcik, L. (2016). Designed calculation of the cam mechanism for transmission between two concentric shafts. In ICMD2017 57th International conference of Machine Design Departments (pp.137-140). ISBN 978-80-261-0609-8.
4. Nguyen Van, H., Sevcik, L. (2017). Apply steel balls for designing the Cam mechanism. Proceeding of 22 nd International Conference Mechanika (pp. 118-121). ISSN 1822-2951.
5. Tang Zhaoping, Sun Jianping. (2011). The contact analysis for deep groove ball bearing based on anasys. International Conference on Power Electronics and Engineering Application (pp. 423-428).

Corresponding author:
Ing. Ha Nguyen Van, Department of Design of Machine Elements and Mechanism, Faculty of Engineering, Czech Technical University of Liberec, Studentska 1402/2, Liberec I, 461 17, Czech republic, phone:+420485353318, e-mail: nguyenha.hut@gmail.com
Ing. Ladislav Sevcik, Prof. Department of Design of Machine Elements and Mechanism, Faculty of Engineering, Czech Technical University of Liberec, Studentska 1402/2, Liberec I, 461 17, Czech republic, phone:+420485353316, e-mail: ladislav.sevcik@tul.cz