[1.]Ha NGUYEN VAN, [2.]Ladislav ŠEVČIK

BALL FOR CAM MECHANISM DESIGN

[1,2] Technical University of Liberec, Studentska 1402/2,461 17 Liberec 1, CZECH REPUBLIC

Abstract: This paper propose is to design a novel cam mechanism, which is used to transmit rotary motion between two concentric shafts. Unlike a conventional cam mechanism design. With this cam model design, is composed mainly of a rotary input camshaft, a rotary output camshaft, a middle part, and two balls and a frame to mount all these parts. The balls were used to intermediate between the input camshaft and output camshaft. The balls carry out pure rolling without sliding while rolling along a circular groove on the surface of the input camshaft, simultaneously each the ball must roll along a straight groove on the middle part and output camshaft corresponding. In this case, the contact between the balls and the grooves is the same as in all ball bearing in general while. Due to the contact between the balls with their grooves on each part of the cam mechanism is very important could directly affect the lifespan and be ability working on the structure. Sodetermining the contact stress between the balls and straight grooves on output camshaft, middle part as well as a circular groove on the input camshaft by using fine element analysis is a required task of the designer. Through contact analysis, the changes could be shown in stress, strain, penetration, friction stress on the cam mechanism. Furthermore, the simulation results revealed that the computational values were consistent with theoretical values. It would provide a scientific basis for the optimal design of the cam mechanism.
Keywords: Bearing for gearing, cam mechanism; a novel cam; contact analysis

1. 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.So they are usually applied in machine tools, automatic assembly lines, paper processing machines, packing machines and many automated manufacturing devices.The paper deals with a configuration cam mechanism with a rotary input camshaft, a rotary output camshaft, a middle part, two steel balls and the frame to mount all these parts. Geometric profiles of the cam mechanism are evident from Figure 1.

a)The input camshaft   b) the output camshaft   c) the balls were assembled

c) the middle part   d) the frame   e) cam assembled
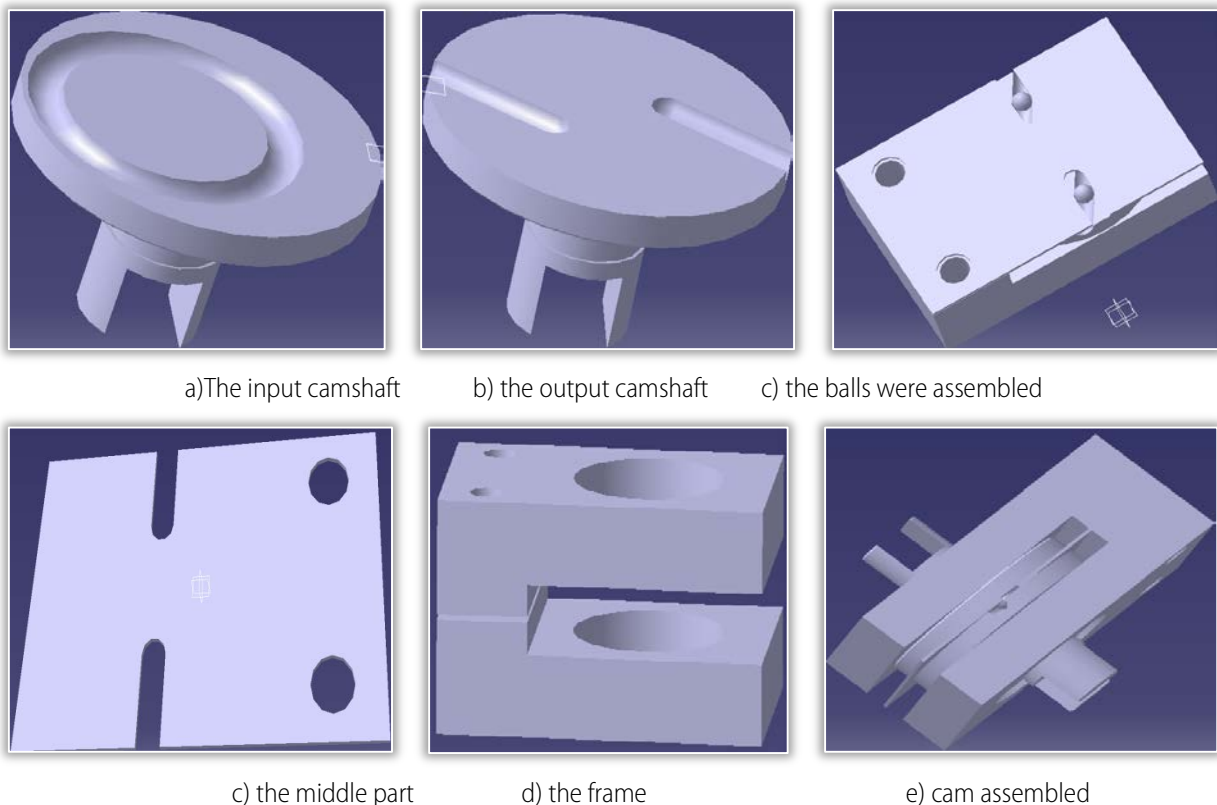
Figure 1: Cam mechanism geometry

The cam model uses two 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 camshaft into the output camshaft. So, when cam works each ball can be easy to move up and down with respect to their straight groove on the output camshaft 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 the result is a 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 were investigated in this article.

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 an 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 blueprint 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 was designed as an example and it discussed 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.

2. MATERIAL AND METHOD

Assume that contact between the ball and circular groove is like the contact of a sphere on a sphere and contact between the ball and straight groove is like the contact of a sphere on a flat-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 the 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 the boundary as target surface and take the other asa 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:

— Build 3D geometry model and mesh;
— Identify contact pairs;
— Name target surface and contact surface;
— Define target surface;
— Define contact surface;
— Set up element key options and real constants;
— Define and control rigid goal's movement(only applicable in rigid-flexible contact);
— Apply the necessary boundary condition;
— Define solution options and load steps;
— 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 the 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 Figure 2 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 camshaft, middle part, and balls (see Figure 2b). The material of all parts made of 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 Figure 2 c.

In this work, the contact pairs are frictional between the ball, groove and so on. Taking separately the groove surface of the input cam shaft, output camshaft,and middle part as target surface and taking correspondingly sphere surface of the

balls as a 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 camshaft, and applied moment load to the input cam shaft is given 1000 N.mm.
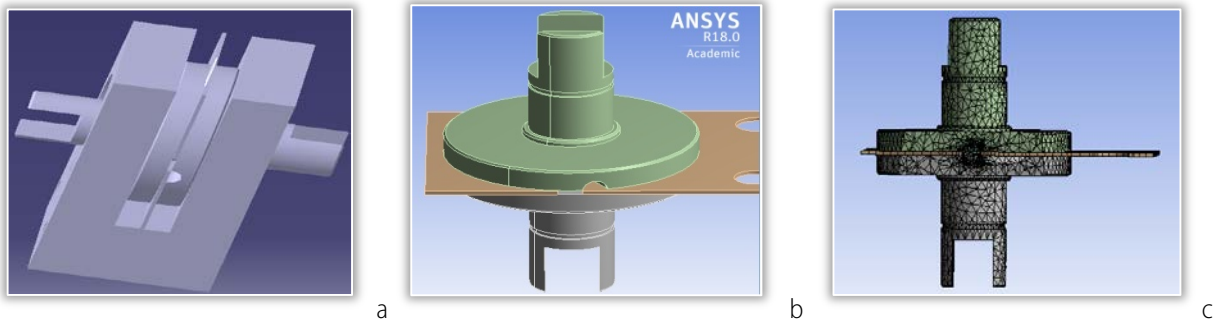

Figure 2. (a) Cam mechanism assembly model, (b) the imported modelin Ansys, c) meshed model

3. 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 camshaftand the balls, middle part. Figure 3 a, b shows von-Mises total strain. The biggest total displacement and strain of cam mechanism respectively is 0.03 mm, 0.01mm. The bigger contact displacement mainly concentrated on the balls and the straight groove of the output camshaft.
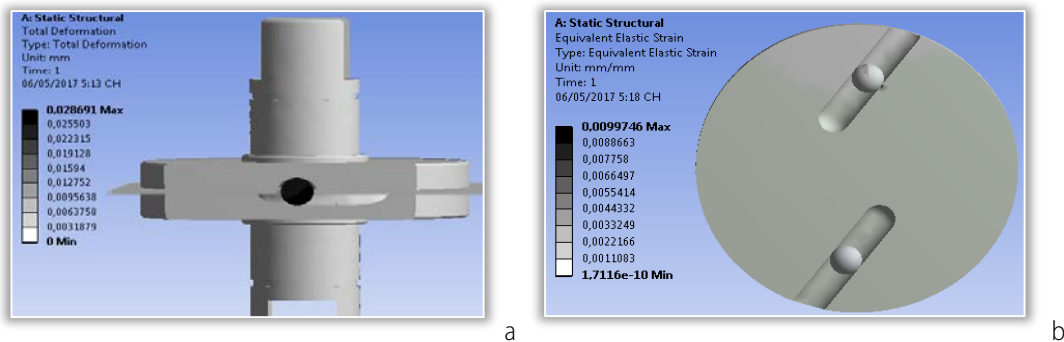

Figure (3a) Deformation cam mechanism, (3b) equivalent strain cam mechanism
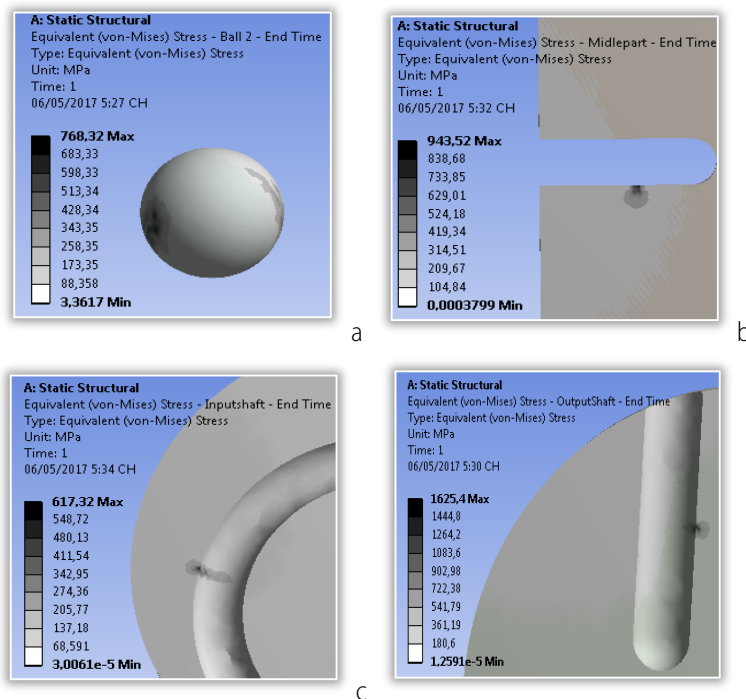

Figure 4. Equivalent stress of cam mechanism, (a) view on the ball, (b) view on the groove of the middle part, (c) view on the circular groove of the input shaft, (d) view on the straight groove of the output shaft

From Figure 4, it is victimized that the contact area had an approximate ellipse shape in the contact area of the balls and the grooves, which was consistent with the Hertzian contact theory.
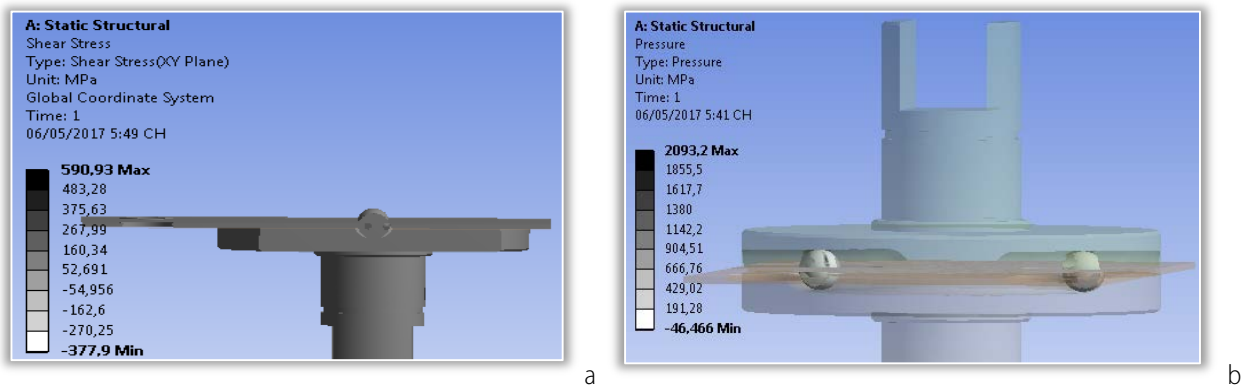

a   b

Figure 5. (a) Shear stress cam mechanism, (b) pressure stress cam mechanism

Figure 5 shows the simulationof 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.

4. CONCLUSIONS

By using ANSYS 18 software to numerically simulate and analyse the stress and strain during cammechanismcontacts, the fine element solutions were shown the results, which had a 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 of the cam mechanism.
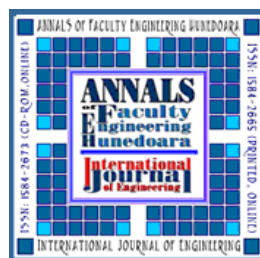
Acknowledgment

Note

This paper is based on the paper presented at **12th International Conference for Young Researchers and PhD Students – ERIN 2018**, organized by **Slovak University of Technology in Bratislava, Faculty of Mechanical Engineering** and **Slovak Association of Mechanical Engineers (SASI)**, in **Častá-Papiernička, SLOVAKIA**, 2 – 4 May 2018.

References

[1] Gopalakrishnan, T., Ruban, M. (2015). Contact analysis of roller bearing using finite element method. Vols Journal of Mechanical 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. Proceedings 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 the analyst. International Conference on Power Electronics and Engineering Application (pp. 423-428).