3D MODELING, FEM ANALYSIS AND DETAIL DESIGN FOR A TESTING DEVICE WITH SPHERICAL JOINT

Cornel Cătălin GAVRILĂ
Transilvania University of Brasov , cgavrila@unitbv.ro

***Abstract*—** 3D Modeling of mechanical structures in the aim of real-time simulation of the product dynamic behavior, using computer performing software, is a necessity. In the paper there are presented some aspects regarding the 3D modeling FEM analysis and also the detailed design of a testing device, to be used in tribological tests of small pieces. First there are presented some aspects of the device virtual modeling and drafting, using CATIA software. Then are defined the geometrical and kinematical restrictions between the parts. Finally, there are presented some aspects of the Finite Element Analysis and the results of this, using ANSYS software. In the final part of the paper, there are presented the conclusion.

***Keywords*—** 3D modeling, detail design, finite element analysis, simulation.

I. INTRODUCTION

THE paper presents some aspects regarding the virtual modeling, FEM analysis and detail design of a testing device, to be used to for tribological tests on small pieces, as plates [1], [2], [3].

Generally, the tribometers are used to study the friction between two surfaces which are in contact [4], [5]. The contact surfaces belong to real pieces made of different materials, which are adjusted to be a mechanical structure. Generally, the pushing force and movement can be set on tribometer. The working principle is presented in Fig. 1.

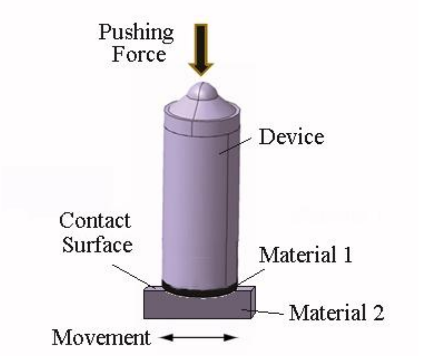


Fig. 1. The usual device, main parts and movements.

For testing different pieces, the tribometer has its own devices, to fit the studied parts together. But, when these parts have particular shapes, these cannot be fitted in the tribometer standard devices without modifying geometry (some cutting, trimming or polishing operations). Usually, for small pieces, as plates or chain links, this could lead to the part destruction. Also, there are situations when, because of surfaces misalignments, the existent devices cannot assure the surface contact between the studied parts. To avoid this, is necessary to design an adequate special device, to be used [1], [2].

The new device has, as requirements (Fig. 2):

1) *the device top head has to fit with the tribometer pushing system;*
2) *the main part, cylindrical has to guide in the tribometer device;*
3) *the bottom mobile part, which has to keep the material to be tested (material 1);*
4) *the spherical joint, between the main cylinder and bottom mobile part, which has to assure the surface contact between the tested parts;*
5) *the spherical joint has to stay permanent assembled (and also to compensate eventual surfaces misalignments), by some components placed inside.*

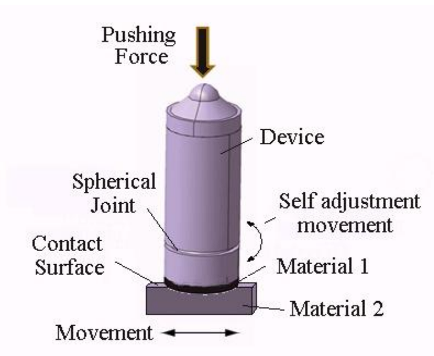
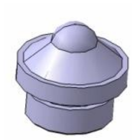


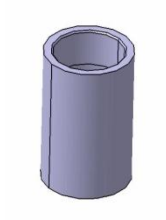
Fig. 2. The device with spherical joint, main parts and movements.

II. THE 3D MODELING

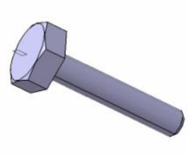
For virtual modeling, the CATIA software is used [1], [2], [3]. The first step is to sketch the testing device parts. To fit the device in the tribometer, the main dimension is corresponding to this. Then, using Part Design module, the each part's virtual model is completed. Inside the device, to assembly the spherical joint, were adopted some standard parts, as screw and washer, which can be chosen from the software virtual library. The device parts obtained are presented in Fig. 3, a - e.



a. Top head



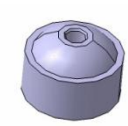
b. Main cylinder



c. Screw

d. Plate

e. Spring



f. Bottom part

g. Material 1 for tests

Fig. 3. Model parts

Then, in Assembly Design module, the device assembly is computed, using the adequate constraints.

The obtained assembled model for the device is presented in figure 4.

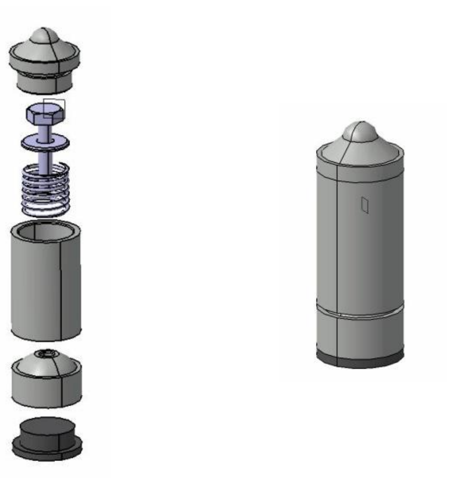


Fig. 4. The assembled model

III. THE FINITE ELEMENT ANALYSIS

For the finite elements method analysis, the ANSYS software is used. For analysis, in the model are not considered the inner components as a screw, plate and spring, which are not relevant here.

The device wears part is the spherical joint, which serves to compensate eventual misalignments. This spherical joint, between the main cylinder and the bottom part, is composed of the spherical surface on the bottom part and, also, by the conic surface inside the main cylinder (Fig. 5). So, the contact between these surfaces is a tangent circle. On this surfaces contact is necessary to be determined the contact pressure. Another contact surface where is necessary to study the contact pressure is the surface between the two materials (in Fig. 1 and 2 the material 1 and 2), which are subjected to the friction study on the tribometer.

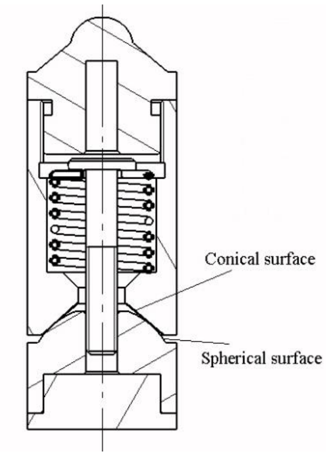
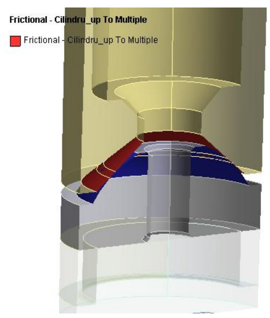


Fig. 5. Contact surfaces in spherical joint

The contact between surfaces is defined, as is shown in Fig. 6.



Definition	
Type	Frictional
Friction Coefficient	0.28
Scope Mode	Automatic
Behavior	Program Controlled
Trim Contact	Program Controlled
Trim Tolerance	0.19403 mm
Suppressed	No
Advanced	
Formulation	Program Controlled
Detection Method	Program Controlled
Penetration Tolerance	Program Controlled
Elastic Slip Tolerance	Program Controlled
Normal Stiffness	Program Controlled
Update Stiffness	Program Controlled
Stabilization Damping Factor	0.
Pinball Region	Program Controlled
Time Step Controls	None
Geometric Modification	
Interface Treatment	Adjust to Touch
Contact Geometry Correction	None

Fig. 6. Contact settings

Also, the material properties, as Young modulus, Poisson coefficient, Tensile Yield Strength, Ultimate Strength are defined as is shown in Fig. 7. For both parts of the device spherical joint, the material is steel. For the study case for contact between materials 1 and 2, the material 2 is also metallic, respectively material 1 is not metallic, for example polyamide [6], [7].

Property	Value	Unit
Density	7850	kg m^-3
⊟ Isotropic Secant Coefficient of Thermal Expansion		
Coefficient of Thermal Expansion	1.2E-05	C^-1
Reference Temperature	22	C
⊟ Isotropic Elasticity		
Derive from	Young's Modulus and Poisson'...	
Young's Modulus	2E+05	MPa
Poisson's Ratio	0.3	
Bulk Modulus	1.6667E+11	Pa
Shear Modulus	7.6923E+10	Pa
⊞ Alternating Stress Mean Stress	Tabular	
⊞ Strain-Life Parameters		
Tensile Yield Strength	250	MPa
Compressive Yield Strength	250	MPa
Tensile Ultimate Strength	460	MPa

Fig. 7. Defining material properties

For the analysis, the force is F=15 N, and the friction coefficient between the spherical joint surfaces is 0.3 [4]. The model with defined force is presented in Fig. 8.

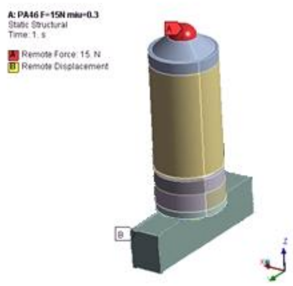


Fig. 8. The model with defined force

Also, the model needs to be meshed into finite elements, solids and deformable, type hexahedral and tetrahedral, with integration node [8].

Fig. 9. The finite element model

For rigid parts, can be used rigid elements, to simplify the model – since the soft is not calculating the strength and deformation for this [8]. In Fig. 9, the finite element model is presented.

IV. SIMULATION AND RESULTS

After the model simulation, the postprocessing results are the deformations and the contact pressure in spherical joint and,also, between material 1 and 2. In Fig. 10, is presented the model with the deformation values. It can be observed that, the maximum deformation value is 0.0016 mm.

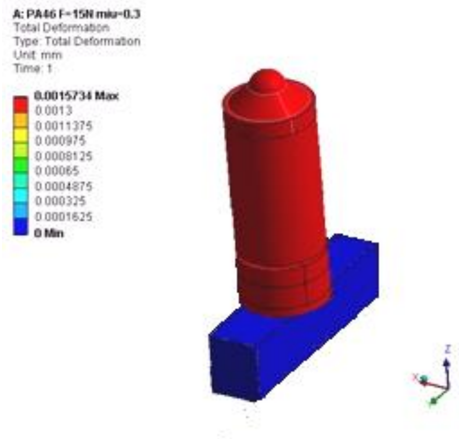


Fig. 10. Model with deformation values

The contact pressure in the spherical joint is presented in Fig. 11; the maximum value is 10,98 MPa. It can be observed that, the contact zone is circular. The discontinuities in the contact area resulted from meshing errors and an insufficient number of the nodes there.

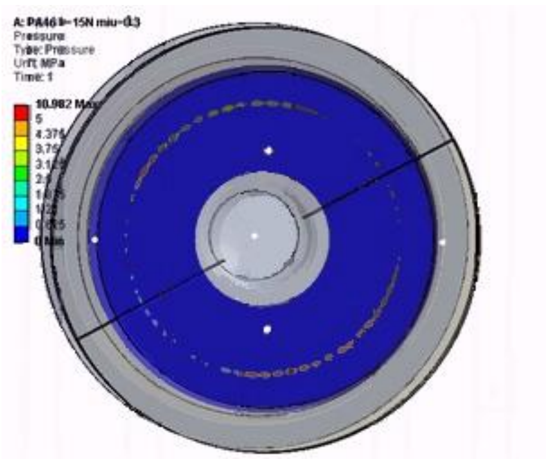


Fig. 11. Deformation values in spherical joint

The contact pressure between the material 1 and material 2 is presented in Fig. 12; here, the maximum value is 1,3 MPa. It can be observed that, the contact zone is uniform, because of the device spherical joint which allows adjusting the surface contact between the materials 1 and 2.

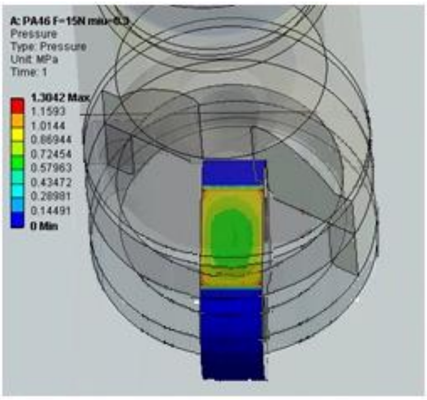


Fig. 12. Deformation values between materials 1 and 2

V. Detail Design and Manufacturing

For testing device manufacturing, the technical documentation is required [2]. The detail drawings and the ensemble drawing are computed in the CATIA Drafting module (Fig 13).

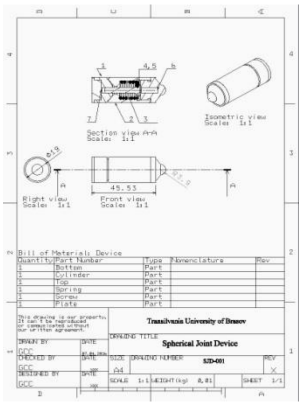


Fig. 13. Device ensemble drawing

After elaborating technical documentation, the device was manufactured (Fig. 14) and it is in use for tribological tests at Tribology Laboratory (DESMe research group), from ICDT Institute, Transilvania University of Braşov.

Fig. 14. Manufactured spherical joint device

VI. Conclusion

There are situations when, because of surfaces misalignments, the tribometer existent devices cannot assure the surface contact between the studied parts. To avoid this, an adequate device with the spherical joint was designed to be used.

Because of the reduced applied force and also because of the model rigidity, deformation values resulted is reduced.

On the studied 3D model, the maximal value of the contact pressure in the spherical joint is 10,98 MPa, which leads to the conclusion that the surfaces will resist to the stresses caused by loading force.

Since the discontinuities in the contact area resulted from meshing errors and/or an insufficient number of the nodes there, for accurate results, the mesh can be improved in studied area.

The contact pressure distribution between the material 1 and material 2 is uniform, so the spherical joint allowed to adjust the surface contact.

From 3D model, using the CATIA Drafting module, the technical documentation was elaborated - ensemble and detail drawing, for product manufacturing.

The FEM method is a rapid calculus method, which allows studying on different 3D models the stresses, contact pressures. The obtained values are close to the real situation, as the model is refined as well. But, the required hard resources can increase.

Similar, can be created and simulated a lot of 3D models for any needed device. In this way, for a company, can be reduced the research costs.

References

[1] C. Alexandru, *Mechanical Articulated Systems Simulation on Computer (Simularea pe calculator a sistemelor mecanice articulate)*, Ed. Lux Libris, Braşov, 2000 (in Romanian).

[2] C.C. Gavrilă, Velicu, R. *A Transversal Mobile Coupling Virtual Modelling*, In: Annals of the Oradea University, Fascicle of Management and Technological Engineering, Volume XIII, (XXIII) 2014, Oradea 2014, pp. 159 –162.

[3] I.G. Ghionea, *CATIA v5. Application in Mechanical Engineering (CATIA v5. Aplicaţii în ingineria mecanică)*, Editura Bren, Bucureşti, 2009 (in Romanian).

[4] M. T. Lateş, *Finite element modeling of the bush chains bolt-bush contacts.* In: Annals of the Oradea University, Fascicle of Management and Technological Engineering, vol. XI (XXI), Nr.1, Editura Universităţii din Oradea, 2012, pp.2.72-2.76.

[5] M. T. Lateş, *Stresses in skate/chain contacts.* In: Annals of the Oradea University, Fascicle of Management and Technological Engineering, vol. XII (XXII), ISSUE I, mai 2013, pp.163-166.

[6] M. T. Lateş, C. C. Gavrilă, *Friction phenomenon in polyamide – steel plate front face type contacts.* In: Annals of the Oradea University, Fascicle of Management and Technological Engineering, vol. XIII (XXIII), ISSUE 1, mai 2014, pp.75-78.

[7] M. T. Lateş, R. Velicu, R. Papuc, *Multiscale modeling of chain-guide contact by using tests and FEM.* 11th World Congress on Computational Mechanics WCCM XI, Barcelona, Spain, 20-25 July, 2014, pp.1062-1069.

[8] H.H., Lee, *Finite Element Simulations with ANSYS Workbench 14. Theory, Applications, Case Studies.* Schroff Development Corporation, Kansas, USA, 2012.