

Aleksander ŚLADKOWSKI*, **Tadeusz OPASIAK**
Silesian University of Technology, Faculty of Transport
Kraśińskiego St. 8, 40-019 Katowice, Poland
*Corresponding author. E-mail: aleksander.sladkowski@polsl.pl

FEM MODEL OF FLEXIBLE COUPLING ASR TYPE

Summary. Main load in operation of flexible coupling in a drive system is concentrated on elastomer part. The paper presents modeling of contact operation of the hub claws and elastomer teeth in a single insert flexible coupling ASR type.

MODEL MES SPRZĘGŁA PODATNEGO TYPU ASR

Streszczenie. Na elemencie elastomerowym sprzęgła podatnego kumulują się główne obciążenia w czasie pracy sprzęgła w układzie napędowym. W artykule przedstawiono sposób modelowania współpracy kontaktowej łącznika podatnego z występami zębatymi w sprzęgle jednowkładkowym podatnym typu ASR.

1. INTRODUCTION

The flexible coupling which was modeled is built from two parts (passive and active) made from steel and a third elastomer element (with connector function), which is between the passive and active parts (Fig.1).

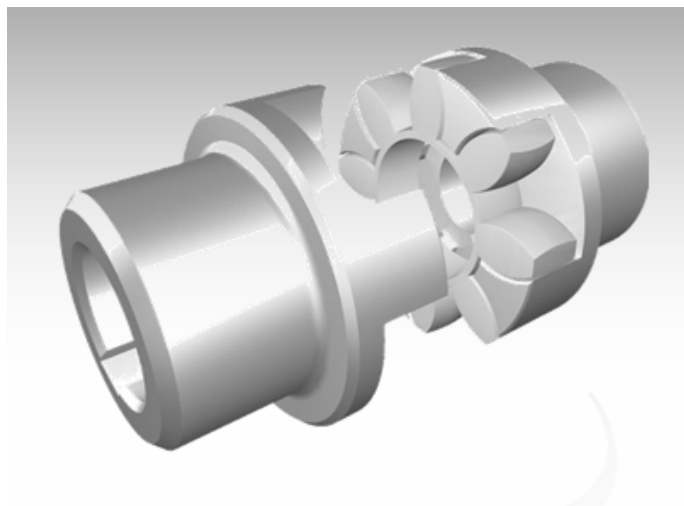


Fig. 1. Single insert flexible coupling ASR type
Rys. 1. Jednowkładkowe sprzęgło podatne typu ASR

An exploitation durability of a coupling of this kind depends on an elastomer insert between passive and active part of a hub. The elastomer insert is a basic element in this kind of coupling, which undergoes damage in a short time [1]. After pass permissible a torque the insert becomes crushed. Load size knowledge of surface pressures indispensable will be useful for qualification of the load size of a maximum torque does not accelerate destruction of the insert. High torque deforms the insert co-operating with steel element and become a durable deforms of the insert which initials the start of the crumpling process. Knowledge of load size of the stress which is transferred after the insert is indispensable.

Knowledge of the transferring loading permits to apply the right elastomer which borders impossible stress to fast destruction near the date of load level.

The kind of insert of the ASR coupling type works under press, so it is necessary to concentrate the model on the contact work of the insert from elastomer to steel performance of the hub claws in the coupling.

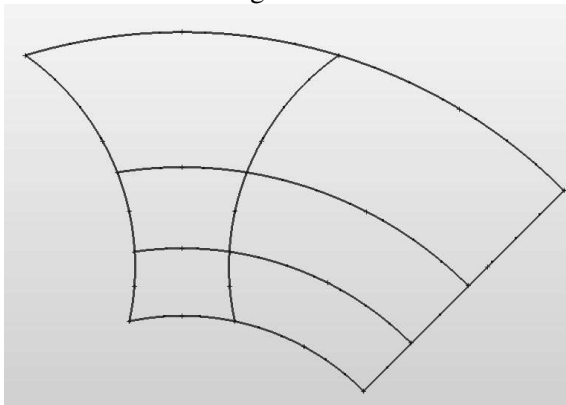
The best for testing the size of stress which transfers insert in this kind of coupling will be building a FEM model of this coupling in paying special attention to the contact connection between co-operation of the insert of flexible coupling with the hub claw.

The model of coupling was analyzed in MSC.MARC programmes which are in the packet program that uses the Finite Element Method calculation. The results received in MSC.MARC programme, from FEM analysis give the answer to a question which load sizes of stress step out on surface of elastomer of the insert in the ASR coupling would give the load size of a torque. For creating the 3D discrete model FEMAP programme was used. MSC.MARC software possesses the ability of converting from software computational models FEMAP and different such as an example of HYPERMESH. There are programs which make it possible to execute the compiled discrete model in our case of the model ASR coupling type. Program bases on the NASTRAN, but it is a much more modern application on Windows and very simple and friendly for a user.

2. GEOMETRICAL MODEL OF MAIN PARTS OF THE COUPLING

Modeling the couplings begun from creating a 2D model of the repeated hubs (Fig. 2). Figure 2a presents 6 basic zones which built ASR coupling of the FEM model creating of two modules type. Figure 2b presents 3 basic zones modeling the insert of the coupling.

a) Basic zones necessary for the coupling's hubs modeling



b) Basic zones necessary for the coupling's insert modeling

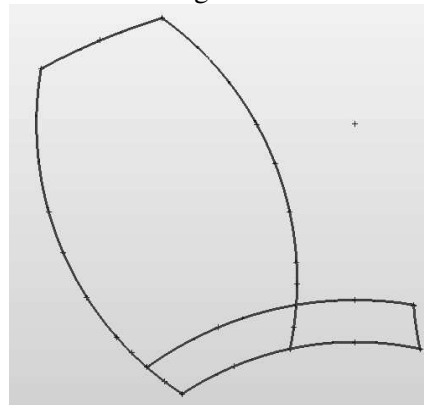


Fig. 2. Geometrical modelling of flexible part of the coupling
Rys. 2. Geometryczne modelowanie części sprzęgła

The geometrical FEM model starts from the basis of model of the smallest element repeated around one part, and the part was copied suitably and extruded. The ASR coupling type is the rotary block built from four symmetrical parts repeated around the hub claws. Such a model makes beginning of works possible by creating repeating elements (Fig. 1a,b), and then copying it suitably. Such approach shortens the number of operation. In time of modeling individual frequent parts of coupling following principles were applied. First, 2D shape was created and repeated around the middle of the part, next these elements were extruded. In this way we got a repeat of 3D shape which was copied around common rotary axis.

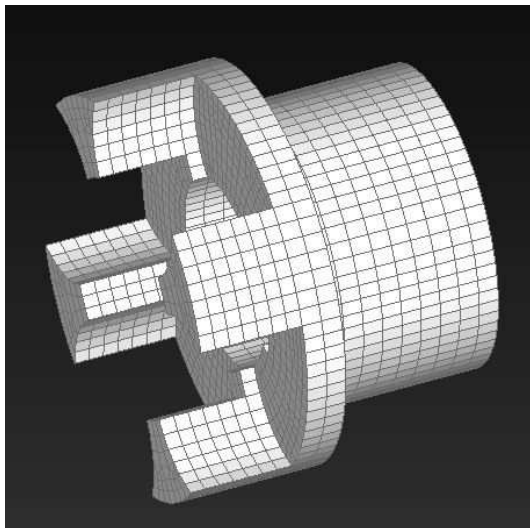
It put on the suitable mesh of FEM. Each part was made separately and it was added in a logical way. Such an approach to the problem made possible to suitably reduce the number of operation to the 3D model creation. For meshing, semi-automatic function was used. Such approach makes the adjustment in place of mutual contact the mesh two co-operating elements possible.

The most difficult element in modeling was elastomer of the insert which consisted from considerably larger operation graphic .It was indispensable and special attention was placed on compiled shapes, and made sure in simplifying in the time of building of the coupling model. Different kind on modules like curves and a chamfer of the coupling, these elements that bring nothing to final calculations of FEM are being simplified. The test was concentrated on exact imitation of the shape of the insert elastomer because it plays a high rule in time of exploitation. 3D models of the ASR type coupling were executed on basis of the original executive document, from Management of the FENA C.O. enterprise, situated in Katowice.

3. PROBLEM OF GENERATING THE FEM MESH OF FLEXIBLE COUPLING

Modeled elements in a dependent discrete model of coupling (Fig. 3a, b) have to be the final solution of problem covered with mesh in a satisfactory possible shortest time.

a) hub model



b) insert model

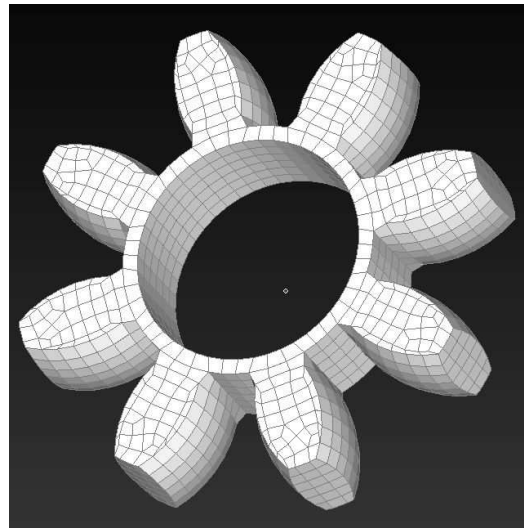


Fig. 3. Individual parts of flexible coupling FEM models
Rys. 3. Modele MES piasty i łącznika sprzęgła

Essential influence of exact calculations was the way of choosing the co-operating elements in modeling. The standard generator of meshes does not warn non agreement of mesh of one element co-operating from second element which causes a big mistake of calculations [2, 3].

Selection of semi-automatic generation of meshes FEM method [4, 5] allows avoidance of these problems (Fig. 4).

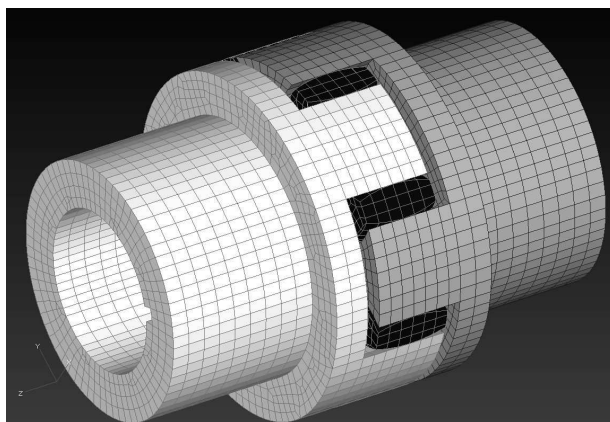


Fig. 4. Composition of modelled parts in the MSC.NASTRAN for Windows

Rys. 4. Zestawienie zamodelowanych części w programie MSC.NASTRAN for Windows

Regarding putting the automatic mesh co-operating with two elements that do not agree so putting the mesh of elements had to be given up, work was stopped with this method. The divergence of two co-operating elements and the divergence of meshes cause incorrect final results. Therefore, for receiving correct results in the coupling model putting on mesh was executed from omission to the automatic generator into semiautomatic meshes. The largest mistakes of delimitation of contact stresses are connected from irregularity of meshes of FEM and the incompatibility of contact a node of co-operating bodies. The decision about resignation in result of conducted analysis was undertaken from automatic meshes generating and the semi-automatic way mesh operation realization.

The conditions of generating on working surfaces of performances of the rectangular mesh of hub claw near this should be safe.

The contact activity of the node near their co-operation should work together. Fig. 3 presents the meshes of 4 considered elements and the coupling tested where the conditions of regularity in meshes and node are obligatory. It caused the largest difficulty in putting on this kind of mesh in adjustment between the insert and performance of half part of the coupling.

Difficulty depended on that the co-operating elastomers, the insert's surface rounded in relation to co-operating from cylinder surface. It required long fitting time. It was dictated that the co-operating surfaces did not fit to their profile.

Such approach to the question lengthened the time of modeling component elements in coupling but obtained results about larger credibility was the result. It was possible to say that they depend to a very large degree to the correctness of construction of FEM from the approach of the constructor to matter contact modeling two co-operating elements.

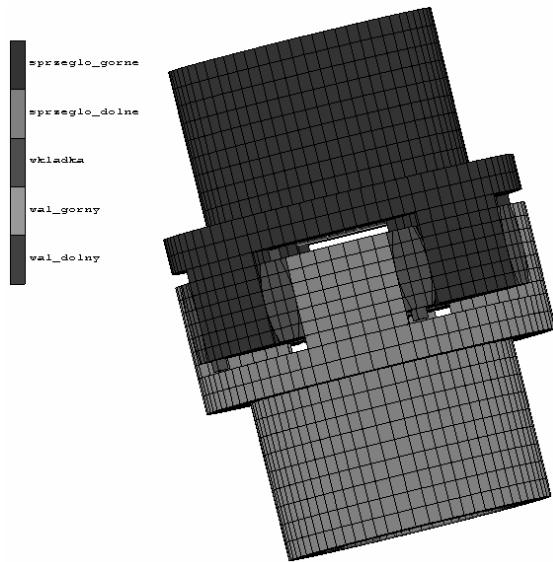
Regarding this, hub claws were made from S 275 and the insert modules from polyurethane and following sizes were accepted characterizing material model:

- for steel:
 - the coefficient of elasticity Younga $2,1 \times 10^5$ MPa,
 - the Poisson ratio 0,3,
- for polyurethane about hardness 95 SH°A
 - the coefficient of elasticity 26 MPa
 - the Poisson ratio 0,49.

The MES model created (Fig. 4) MSC was imported to MARC programme, where borderage conditions were given for considered problem (Fig. 5). In time of work on the flexible coupling the elastic element was only pressed and that investigation was accepted. Elastic insertion is only a deformed body. Only the hub claws of the flexible coupling was modeled as ideal stiff bodies. Hub claws on a stiff body are located on two sides of the elastomeric insert.

For distinctions one part was named bottom and the second upper part (Fig. 5). The bottom body is motionless, and the upper body has for all elements the possibility to turn around on common axis. Mutual contact among upper body and bottom body were holding for mediation of the elastomer insert, stiff body contacts from deformation body.

a) The deformed body



b) The rigid body

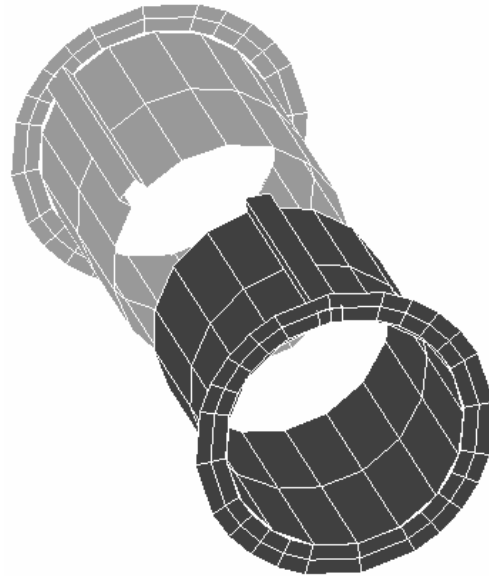


Fig. 5. Task of contact bodies in the MSC.MARC programme
Rys. 5. Zadanie ciał kontaktowych w programie MSC. MARC

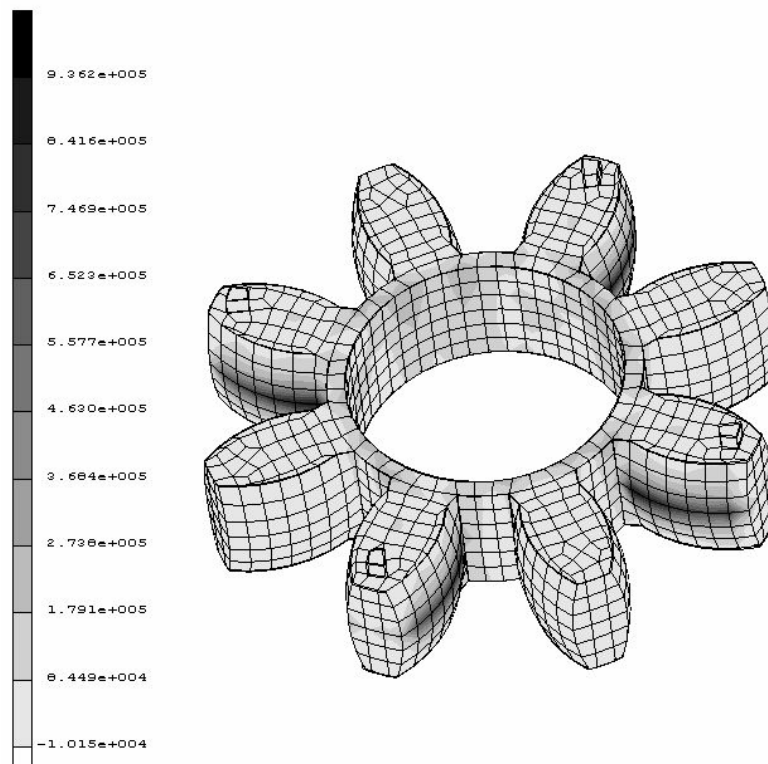


Fig. 6. Stress in the insert according to Huber-Mises-Hencky theory
Rys. 6. Naprężenia we wkładce wg teorii Hubera-Misesa-Hencky'ego

The schedule of stress on working surface elastomer the insert on figure 6 was reduced according to theory Huber-Mises-Hencky theory was introduced. The analysis of contact stress permits to draw out the following conclusions. The investigation of contact stress was not possible in case of automatic generator of mesh use. Such approach generates distorted results caused by passing the node put on mesh. The schedule of contact stress is situated along the central surface of elastomer insert. The insert has 8 shoulders, in time the work transferred by 4 shoulders. The maximum stress is part on the bottom central part of the insert. During the work of the cylinder part of performance the hub claws press on the rounded surface of insert. The central part was a subject of continuous stress.

4. CONCLUSIONS

Modeling of this kind of coupling should begin from modeling the smallest repeating part. Generating automatic mesh shortens the cycle of modeling costs but the results may be adequate to the problem of co-operation contact. Maximum contact stresses was situated in the central part of the insert on 4 shoulders. Arrangement stress can happen by curve of the surface co-operating between the hub claws and half part of the ASR type flexible coupling.

Literature

1. Markusik S., Opasiak T., Gruszka P.: *Napędy maszyn transportu ciągłego ze sprzęgłem podatnym*. XVI Konferencja Szkoły Jesiennej „Podstawowe problemy transportu przenośnikowego”, Transport Przemysłowy, nr 3/2006.
2. Sładkowski A., Kuminek T.: *Influence of the FE Discretization on Accuracy of Calculation of Contact Stress in a System Wheel – Rail*. Proceedings 3rd Scientific Conference of Jan Perner Transport Faculty “New Trends in Transport and Communications”, University of Pardubice, Pardubice 2003, s. 13 - 18.
3. Сладковский А. Контактное взаимодействие колес и рельсов. – Луганск: Русь, 2005, с. 189.
4. Sładkowski A., Gubaczewa L., Basow G. *Modelowanie ciernego tłumika drgań*. VI Konferencja Naukowa "Telematyka i Bezpieczeństwo Transportu", Katowice 2006, s.173 - 186.
5. Katalog firmy Fena. Polska 2007.

Received 2.10.2007; accepted in revised form 23.11.2007