PERIODICA POLYTECHNICA SER. MECH. ENG. VOL. 48, NO. 2, PP. 133–143 (2004)

FE MODELLING OF THE MOUNTING PROCESS OF METAL SCREW AND COMPOSITE PART

Csilla ERDŐS–SÉLLEY

Insitute of Machine Design
Budapest University of Technology and Economics
H–1111 Budapest Műegyetem rkp. 3
Tel: (36 1) 463 1292
e-mail: selley@eik.bme.hu

Received: April 4, 2003

Abstract

The thread cutting process of threaded joint between metal screw and short glass fibre reinforced polymer part (PAGF30) was modelled by the Finite Element Method in order to establish the right joint geometry. An axial symmetrical model was developed for modelling the mounting process. The model takes into consideration the non-linear constitutive law of polymer, its large deformations and in-plane friction. In the contact process the metal screw was assumed to be rigid.

As a result of simulation, the thread profile in the nut and surface pressures over the contact area were specified, which can serve as a basis for taking thermal load and tangential friction into consideration. In addition, the pre-tightening process and limit force of damage were determined. The resulting thread profile and limit force are in good correlation with the experimental data. The deviation can be explained with the material softening due to the higher temperature.

The thermal process due to the frictional heat generation was separately analyzed. It was established that high temperature arises only in a small environment of the contact zone in the polymer part. The highest temperature was obtained on the second thread.

Keywords: FEM, thread cutting, composite.

1. Introduction

1.1. General Aspects

Joints formed by the connection of thread-cutting metal screw and polymer nut have been used in various fields of industry for decades (e.g. in cars, household machinery, hand machine tools, toys). The main reasons for application of this joint are economical mass production and rapid mounting. However, durability and load bearing capacity of this joint are affected by various factors, therefore designing it is a complex task. At present, engineers designing such joints may use recommendations based on long years of experimental work [1], supplemented by simple analytical considerations [2].

The creeping process was examined in more detail by Tome by experiments [5]. Relaxation of the thread cutting joint was represented by SOÓS et al. with finite element [4]. The effect of dynamic loading on the damage process was examined by DRATSCHMIDT [3] with hysteresis measuring method. On the basis of his observations by dynamical load the loadability of the metal screw is determinant. To the author's knowledge, there is no other simulation in this field to confirm the experimental results.

Two main requirements have to be met when designing the joint: the pre-tightening force must remain always above a safety limit and the joint must not be damaged either during the mounting or under operating loads. The pre-tightening force due to the stress relaxation can sink even by 80%, so the start value of the pre-tightening have to be chosen as high as possible. Accordingly, the limit force provides a basis for designing and it can be obtained by modelling the mounting process.

Characteristic stress types of the polymer part are pressure under the head, shear stress on the outer wrapper of the thread, contact pressure on the thread, radial and tangential stress according to the displaced volume and torsion.

The four types of failure are tearing up the bolt along the outer wrapper from the polymer tube, breaking the head of the bolt, the axial crack of the tube, and the rip of the tube in its cross section (*Fig. 1*).

For being repairable, the joint should be designed in such a way that the first art of damage is the standard damage which is caused also by the axial force.

Rising the mounting speed, in order to improve the productivity, the heat generation will also grow up on the contact surface, which „helps" to form the thread in the polymer. However, above a limit the pre-tightening force can hardly adjust, so it is recommended that the mounting speed should not exceed 500 1/min.

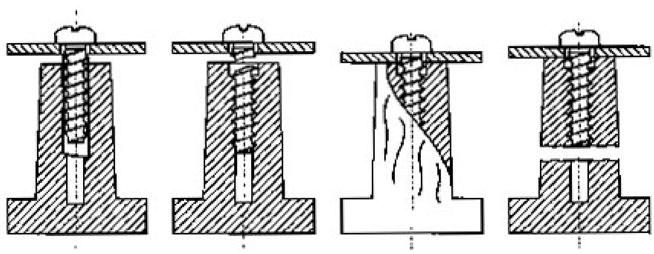

Fig. 1. Failure types of the joint

The aim of this study is to create an appropriate model to simulate the mounting process by the Finite Element Method. The analysis, by presenting the stresses and temperature distribution in a small environment of the thread, can give a more comprehensive explanation about the behaviour of the joint in the mounting process.

The limit force is a basic characteristic value of the joint, so comparing the calculated and measured values we can check the property of the applied numerical model. Having a right numerical model for the limit force, geometrical parameters assuring the desired strength of the joint can be given. In the analysis a short glass fibre reinforced Polyamide was examined (BASF Ultramid® B3EG3, PA6-GF30) which is frequently used e.g. in the car industry. For simulation the mechanical problem the MSC. Marc and for solving the thermal problem the COSMOS/M finite element program systems were used.

1.2. Permanent (Enduring) Processes

After the mounting process the polymer starts to creep due to the visco-elastic behaviour of the polymer activated by the pre-tightening force.

TOME [5] tested by experiment way the effect of different parameter of the residual pre-tightening force due to the relaxation process. He established that the increasing humidity and temperature decrease the residual pre-tightening force, while with increasing the glass fibre content smaller relaxation was observed. Geometrical parameters and the magnitude of the pre-tightening force (between 1 kN and 4kN) were found not to affect the process characteristics significantly. Enlarging the frontal area and choosing the right way for load transfer (on the frontal face instead of the foot of the tube) also influence favourably the remaining pre-tightening force.

Summarizing the observations above, it can be stated that geometrical parameters have less impact on the relaxation process but they influence mainly the limit force which is determined by the effects taking place in the course of the mounting and by the environmental circumstances, and so revealing the mounting contributes to the right design of this joint.

2. Describing the FE Model

2.1. The Mounting and Pre-tightening Processes

In the course of the mounting, the polymer nut enters into connection with the metal bolt and due to the mounting moment it deforms partly visco-elastically and partly plastically. In the course of this process heat is generated originating from the plastic strain dissipation and from the friction in the contact surface. A transient heat conduction process evolves implying an inhomogeneous temperature distribution and so the material property of the composite: in the vicinity of the contact surface the limit stress (yield stress) falls significantly.

At the same time, the strains in the metal bolt are much smaller and the material properties remain constant in the evolved temperature range. The increasing penetration implies small elastic or locally plastic strain of the bolt.

Dominant load case of bolt is probably the torsion due to the contact pressure and great friction. As torsion rises with the growth on the contact area, it can lead to damage of the metal bolt.

The following simplifications can be made:

- The bolt is considered as a rigid body due to the big difference in Young modulus of the connecting materials.

- At the stage of the static analysis, time dependency of the thread cutting is not fully regarded. Temporal behaviour of the thermal process is more interesting in the polymer nut. A two dimensional axisymmetrical model gives a properly detailed image about the load distribution and the developing thread profile, contact normal force, radial stresses; these properties define the axial limit force. On the other hand this model does not include directly the tangential friction (normal to the modelling plane). Limiting the thread length, damage of bolt can be avoided due to high frictional moment.
- In the course of the thread cutting process for proper modelling of the continuously increasing contact surface, a very fine finite element mesh should be generated in all steps.
- To model the thread cutting process, a rigid profile symbolizing the bolt indents into the polymer tube in 50 steps from the borehole diameter d (see in *Fig. 1*) to the nominal diameter of the bolt. Pre-tightening is considered with movement of this profile in axial direction.
- By mechanical modelling at this stage the thermal effects are neglected, the material law for tension is considered at 23°C in conditioned state [7]. Thermal effects have been analyzed separately due to the heat partition generated by the friction. Material softening effects due to the developed temperature distribution should have been considered later solving the coupled thermo-mechanical problem.
- Assuming a careful manufacturing process, the reinforced polymer can be considered as homogeneous, isotropic material. Practically only on the top of the tube occurs any fibre orientation but it exerts only local impact so it could be neglected. A finite element model including fibre orientation would be very time consuming because it must be also considered in the course of remeshing.
- For the material considered there is no information about the effect of the compressibility on the mechanical properties, which plays probably not a negligible role at the given load case. Measurements are required to explore this feature.
- Dynamic simulation is out of interest at the first stage of the modelling because in this case the bolt also has to be treated as deformable.

2.2. Thermal Process

In the course of the thread cutting process, on the one hand, a portion of the strain energy, and on the other hand, the friction will be converted to heat, which implies an inhomogeneous temperature field due to a transient heat conduction process. Owing to the temperature change, material properties of the polymer also vary greatly and so exert impact on the deformation process. The coupled thermo-mechanical problem can be solved in a later simulation. Modelling the temporal behaviour correctly involves a three dimensional model that implies enormous computation effort. The procession in the time can be approximated with a moving heat source from thread to thread. Considering the effects in the polymer part, this assumption is reasonable. The heat source represents the frictional heat; dissipation term (which is not supposed to be big) can be taken into account in a coupled problem. A sensitive and open proposition is how to assume the frictional relation; wanting detailed information, friction coefficient was assumed $\mu = 0.3$.

3. Results

3.1. Problem Definition

The examined geometry can be seen in *Fig.* 2 ($L = 11$ mm, $d = 4.5$ mm), the finite element mesh and boundary conditions in *Fig.* 3, the applied material law in *Fig.* 4. The connected element is also assumed to be rigid.

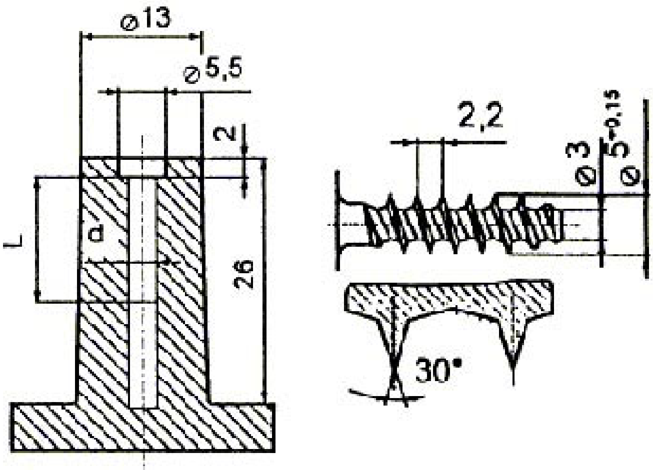

Fig. 2. Geometry of the examined polymer tube and the metal bolt

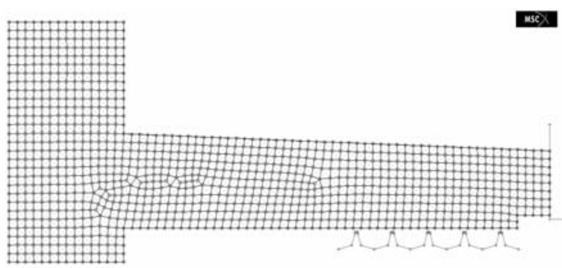

Fig. 3. Finite element model of bolt and nut

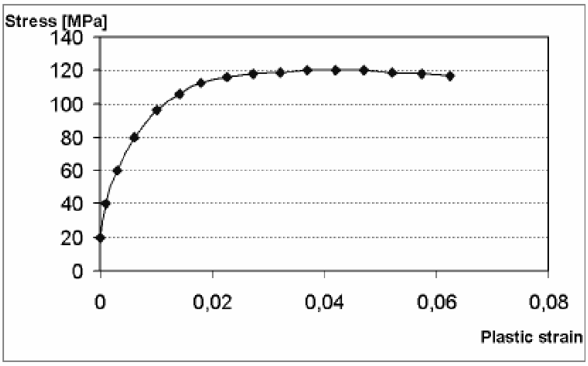

Fig. 4. Material law of PA6 GF30 after [7] at 20 °C

In thread cutting, the rigid profile symbolizing the metal bolt moves into the polymer tube in 50 uniform steps, while a small amount of the compressed volume goes to radial direction and the larger part comes out along the thread.

In the course of the simulation, special care should be taken about mesh distortion due to large deformation in order not to lead to braking the solving of the contact problem, and enough element should be placed on the increasing contact area for appropriate approximation of the contact pressure. Problem is solved with smaller linear triangular and quadrilateral elements as in the initial model (0.125 mm) and in each step a remeshing was applied. The remeshing process took about the half of the total solution time.

3.2. The Thread Cutting Phase

The deformed shape owing to the thread cutting process and the Cauchy equivalent stresses can be seen in *Fig. 5.a* and the strains in *Fig. 5.b*. The contact pressure distribution can be observed in more detail in earlier paper [6].

3.3. Pre-tightening Process

During the pre-tightening phase, the metal bolt is considered as it were moving out in axial direction from the tube. The pre-tightening force can be calculated from the reaction force on the connected element.

By simulation of the pre-tightening phase it can be observed how the initially symmetrical stress distribution becomes asymmetric as a result of the applied constantly rising pre-tightening force. In the 7. step of pre-tightening, one side of the metal bolt thread rises from the nut as a result of 6200 N axial force (*Fig.6*).

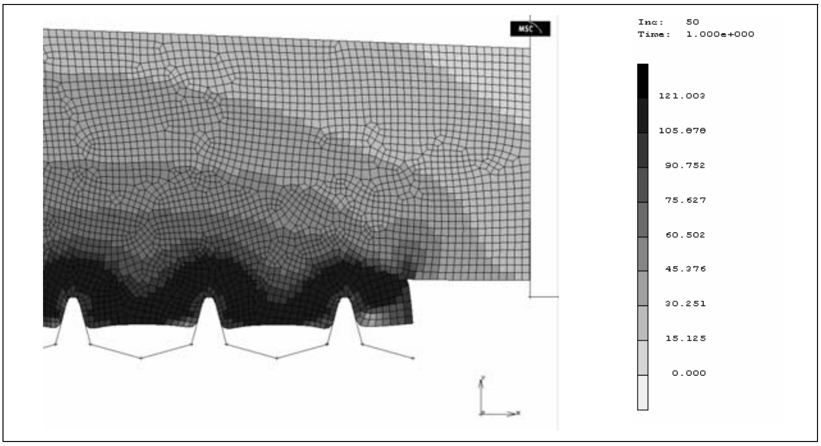

Fig. 5.a. Stress distribution after the thread-forming process

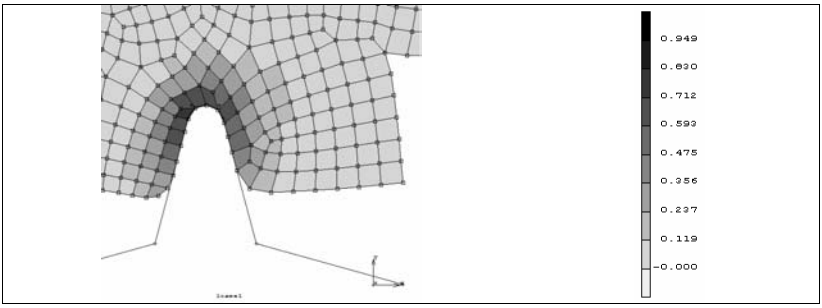

Fig. 5.b. Plastic strains after the thread-forming process

Following the change of the axial force, the limit force for damage can be given (*Fig. 7*); reaching it the bolt will scratch out of the nut in a continuously accelerating way. The axial force in this case is 7640 N, which is 25% higher than the measured value. The main reason for the difference is neglect of the thermal effect. The separately established thermal analysis in the next section shows a significant temperature increase in the highly loaded places, which implies material softening (*Fig. 8*).

3.4. Thermal Analysis

Axial moving of the heat source is substituted with separate heat sources on each thread, coming to act one after the other. After the delay needed to complete the previous threads, they follow the growth of the contact surface quadratically and become constant on the steady contact zone as it can be seen in *Fig.7*. This model

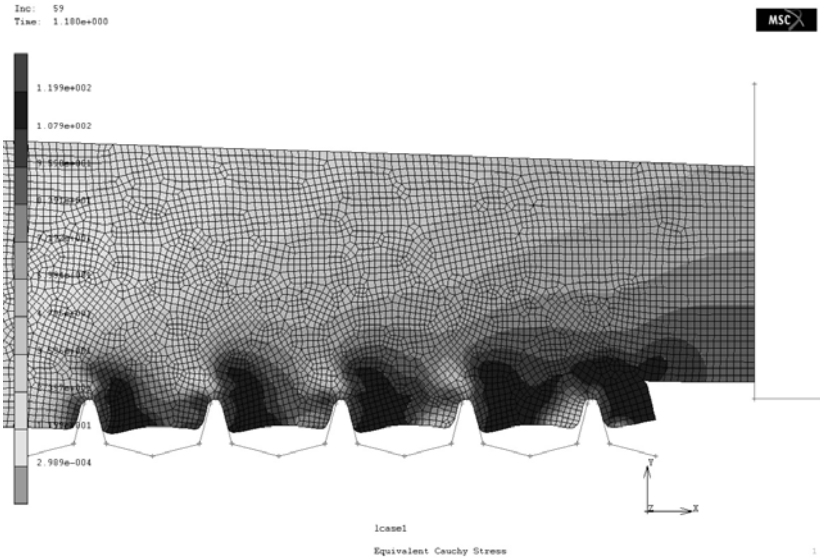

Fig. 6. Cauchy stresses at the moment of thread separation

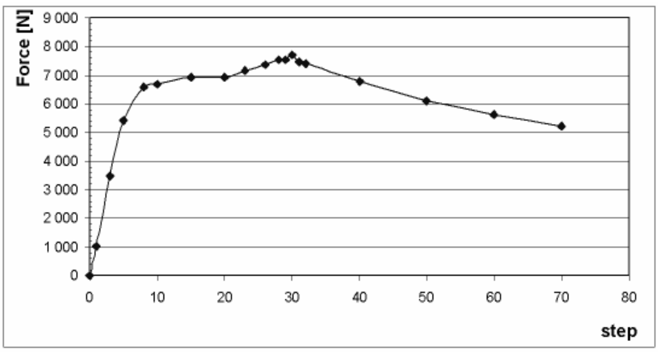

Fig. 7. Axial force in the course of the pre-tightening process

obviously does not follow correctly the heating process of the metal bolt, and its effect to the polymer – initial temperature of bolt on thread 5 is much higher than the original environmental temperature.

The frictional heat flux can be calculated by

$$q = \mu p v,$$

where μ denotes the friction coefficient, p is the contact normal pressure and v is the circumferential velocity (calculated at $n = 300$ 1/min. with average value acting on the middle diameter of the contact area). The temperature dependency

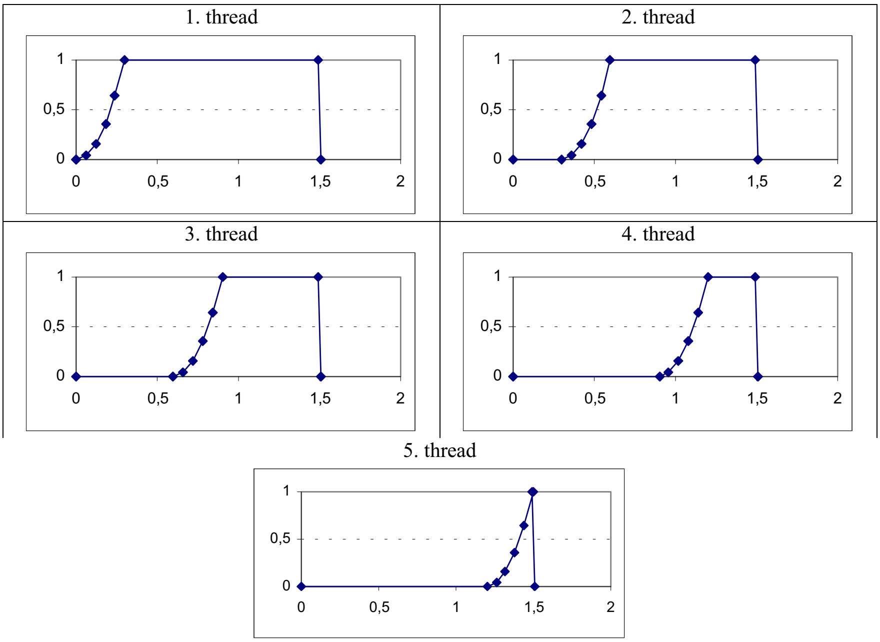

Fig. 8. Time curve for the heat source

for μ, without any direct information about it, was not considered; as the mounting moment shows a piecewise linear character, it can be assumed to be constant; its average value was supposed to be 0.3. By solving the coupled problem in a further examination, the frictional effects have to be explored by measuring methods. In the case of a separate problem there is no result for the contact normal force, so it can be approximated with the analytical formula given in [2]:

$$p = \frac{2}{\sqrt{3}}\sigma_F\,(1+\alpha)\,,$$

where $\alpha = 5.63^\circ$ for the given thread profile, σ_F is the yield stress.

Material softening depending on the temperature can be considered with the yield stress-temperature diagram (*Fig. 9*) that is completed according to the data of BASF [7].

The end state of the transient heat conduction can be followed in *Fig.10*. It can be observed that very high temperature (much above the glass transition temperature) arises in the very closed environment of the contact zone in the polymer part. The highest temperature develops on the second thread according to the results of this model. Explanation for it can be that although the heat generation is the highest on the first thread, but the cooling process can take place mostly at this place, which can equal or exceed the aforementioned effect.

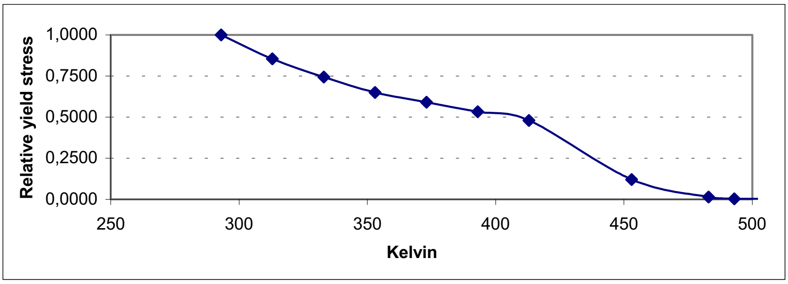

Fig. 9. Temperature dependency of yield stress

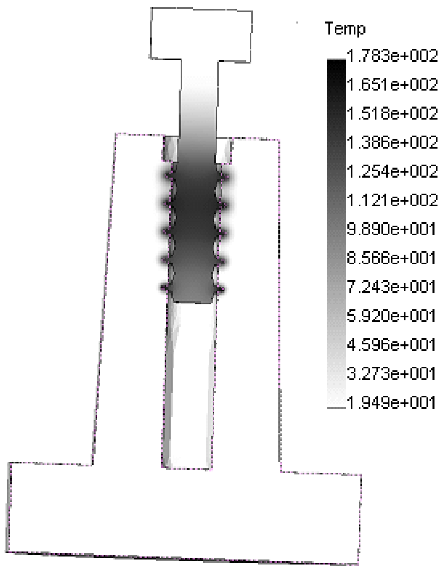

Fig. 10. Temperature distribution owing to the mounting at 1.5 s

After 1.5 s the contact surface is steady and there is no heat generation, however, the heat conduction process continues. Following the process further shows that in a few seconds a great amount of heat leaves, so the contact surface temperature reduces to about 40 °C.

4. Conclusion

The assumptions that had been made for separation of the processes taking place in the course of the mounting of a metal bolt in the glass fibre reinforced polymer nut prove true because

- the simulated deformed shape owing both to the thread cutting and to the pre-tightening follows the experimental observation with an acceptable deviation,
- by modelling the damage process, the limit force has a good correlation to the experimental results.

The model can be applied for reconciling the geometrical parameters because it reveals the weak points that are not to be determined with experiments and so it provides conducement by the design of the joint. The contact normal pressure is determined so knowing the frictional effect normal to the simulation plane, the mounting and limit moment can be calculated and the strength of the joint can be decided accordingly.

The thermal analysis shows a qualitative image about the real state in the polymer part, which confirms the results of the static analysis. This kind of modelling is proved suitable for the application in a coupled thermo-mechanical analysis in the future.

Acknowledgements

Research was sponsored by the National Fund for Scientific Research (OTKA – F 025546). The authors are grateful for assistance by Dr. Károly Váradi and István Elinger to clarify theoretical issues. Special thanks to Drechsen Kft. for their assistance in using the software and to Prof. Imre Bojtár to make MSC. Marc Mentat finite element system available for us.

References

[1] EHRENSTEIN, G. W. – ONASCH, J., Fügen von Kunststofftteilen mit gewindeformenden Metallschrauben, *Verbindungstechnik*, **9** (1981), 13. Jahrgang.
[2] EHRENSTEIN, G. W. – ONASCH, J., Berechnungsmöglichkeiten für das Verschrauben von Teilen aus Kunststoffen mit gewindeformenden Metallschrauben, *Kunststoffe* **72**, 1982.
[3] DRATSCHMIDT, F., Zur Verbindung von glasfaserverstärktem Polyamid, *Technisch-wissentschaftlicher Bericht*, Erlangen, 1999.
[4] SOÓS, E. – RENZ, R., Experimental and Numerical Investigations of a Thread-Cutting Screw Joint, *3rd Conference on Mechanical Engineering, Budapest University of Technology and Economics*, May 30–31. 2002, pp. 674–678.
[5] TOME, A. – EHRENSTEIN, G. W., Direktverschraubungen unter Temperaturbelastung, *Kunststoffe*, **90** Jahrgang, 2000.
[6] ERDŐS–SÉLLEY, CS., Numerical Simulation of the Thread Self-Tapping by Threaded Joint between Metal Bolt and Polymer Nut, *3rd Conference on Mechanical Engineering, Budapest University of Technology and Economics*, May 30–31. 2002, pp. 515–519.
[7] CAMPUS BASF database 4.52, 2001.