MECHANIKA
CZASOPISMO TECHNICZNE
TECHNICAL TRANSACTIONS
MECHANICS
WYDAWNICTWO POLITECHNIKI KRAKOWSKIEJ
2-M/2010
ZESZYT 8
ROK 107
ISSUE 8
YEAR 107

PAWEŁ KOT*

OPTIMIZATION OF SHAPE AND PARAMETERS OF A LATCH SUPPORTING THE BOTTOM AND THE COVER OF A FUSE BOX

OPTYMALIZACJA KSZTAŁTU I PARAMETRÓW ZATRZASKU MOCUJĄCEGO SPÓD I POKRYWĘ SKRZYNKI BEZPIECZNIKOWEJ

Abstract

In the design process, the topic of designing of a polymer latch mounting the bottom and the cover of a fuse box in a vehicle was considered. In this work, we presented the results of resistance calculations for the designed element, which were carried out in the Abaqus system. In this work, it is also presented a short characteristic of the Abaqus system, together with its advantages in terms of designing elements made from plastic materials.

Keywords: latch, strength analysis, plastic material, designing

Streszczenie

W toku projektowym podjęto temat konstrukcji polimerowego zatrzasku mocującego spód i pokrywę skrzynki bezpiecznikowej pojazdu. W referacie opisane zostały wyniki obliczeń wytrzymałościowych projektowanego elementu, które zostały wykonane w systemie Abaqus. W pracy przedstawiono także krótką charakterystykę systemu Abaqus, oraz przedstawiono jego zalety w aspekcie projektowania elementów wykonanych z tworzyw sztucznych.

Słowa kluczowe: zatrzask, Abaqus, analiza wytrzymałościowa, tworzywo sztuczne, optymalizacja

* MSc. Paweł Kot, Institute of Applied Informatics, Faculty of Mechanical Engineering, Cracow University of Technology.

1. Introduction

Advanced computer aided systems for engineering tasks CAD/CAM/CAE offer a wide array of possibilities in terms of geometric modelling. Individual commands in various programs differ with specific attributes, though in each and single one of them there are some generic methods of geometric modelling. [1, 2]

The same geometric model can be created in a digital form in many different ways. In case of a complex geometric model, the number of such ways is unlimited. In this unlimited number of hypothetical ways of creating geometric models, only a small share guarantees proper construction of the given geometric model. It is critical since contrary to graphic software, the main focus of which is to provide a good look of the designed model, a model in the advanced CAD/CAM/CAE is created in such a way so that it can play a number of various tasks and could be then applied to many further design processes, including among the others: development of technical documentation, kinematic analysis, analysis of element and component dynamics, verification of product functionality, engineering calculations etc. using the finite element methods, product visualization, storing additional information obtained in the design process or introduction of changes in the project. All such tasks must be carried out based on a set of previously prepared geometric models. That is why, applications and models for geometric modelling must be selected in a very careful way and their familiarity is necessary for their proper application. When selecting such methods, a designer must take into account further design process in such a way so that the computer model can be the basis for further product development, and don't bring new design problems. [1]

2. Goals

Here, we restrict the scope of the paper to strength analysis for a latch mounting the bottom and the cover of a fuse box.

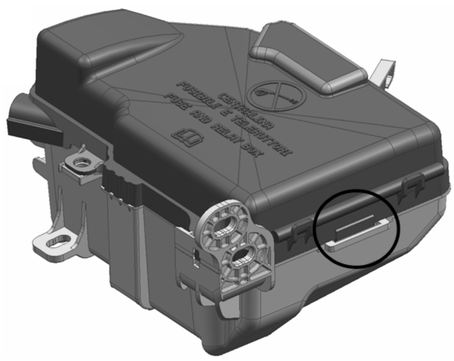

Fig. 1. A 3D model of a fuse box, together with indicated latch mounting the box cover

Rys. 1. Model 3D skrzynki bezpiecznikowej, z zaznaczonym zatrzaskiem pokrywy skrzynki

Taking into consideration the fact that the designed element is planned to be manufactured from polymer materials, the author decided to carry out the calculations using the Abaqus system, which is well prepared for solving problems of non-linear character. Fig. 1 presents the 3D model of a fuse box designed in the Unigraphics software. In this paper, it is also presented also stages of the optimization process for a latch mounting the box cover.

3. Application of engineering software for simulation purposes

3.1. General characteristics of the Abaqus software

ABAQUS system is a collection of software packages, designated for solving complicated problems and carrying out engineering simulations for various branches of the industry. One of the main branches of the industry where the Abaqus system is used is the motor vehicle, airplane and shipyard industry, along with machine, chemical and optical industries. Abaqus system comprises several modules: ABAQUS/Standard, ABAQUS/Explicit, ABAQUS/CAE, ABAQUS/Viever [9].

When running mechanical stress simulations, as presented in this article, ABAQUS/Standard module was used, equipped with the following functions:

- linear and non-linear static,
- linear analysis of dynamic load,
- thermal conductivity and analysis of thermal stress,
- analysis of thermal and electrical phenomena,
- analysis of flow of liquids through porous media coupled with generation of mechanical stress,
- analysis of mass diffusion processes,
- acoustic analysis,
- analysis of piezoelectric effect,
- analysis of shock waves in fluids,
- mechanics of fractures.

The ABAQUS/Standard module is based on the finite element method (FEM). It is also possible to utilize all commonly known types of finite elements in one, two or three dimensions. It is also possible to apply various types of special elements [8].

The superiority of the Abaqus system when compared with other similar systems is the fact that this particular system allows for modelling of various materials, including metals, rubber, plastics, soil, rocks or wood. The aforementioned materials might be isotropic or anisotropic, where the deformation may be slow or rapid, and the processes may be adiabatic and isothermal. Additionally, the ABAQUS/Standard module allows for modelling of various types of boundary conditions and loads, together with definition of contact surfaces [6].

4. Selection of the proper material and application of filling materials

Material used for manufacturing the designed latch should be primarily sufficiently rigid, thus it is necessary to guarantee proper mechanical durability. When designing the examined latch supporting the cover of a fuse box, PBT GF20 XF (Extreme Flow) material was selected. This particular material features enhanced flow properties, which is particularly useful when manufacturing tiny elements, guaranteeing perfect balance between rigidity and durability. It is also characterized with good mechanical properties at higher temperatures and exhibits very good electrical properties [10].

When selecting the optimum filling material, it is necessary to take into account the ratio between the required component rigidity and fibre prices. In the examine case, it was decided to use inorganic fibres, with the key role played by glass and boron fibres. Even though glass fibres have worse mechanic properties when compared with boron fibres, the price of this filling material is substantially lower. The difference in mechanical strength of the elements made from materials containing such filling materials limited application of glass materials in more critical structures, though such fibres are generally used for filling materials. When carrying out examination, it was decided to utilize glass fibres, thanks to which it was possible to achieve higher rigidity and proper resistance, with relatively lower cost and low proper weight. It is a crucial feature of composite materials, which are increasingly frequently used in various branches of industry, where high strength and low weight are critical.

5. Methodology of examinations

5.1. 3D model of designed latch

It is quite simpler to notice that the designed latch is symmetric, therefore to save time only ¼ of the element may be examined. This simplification should not impact negatively the obtained results [4, 5].

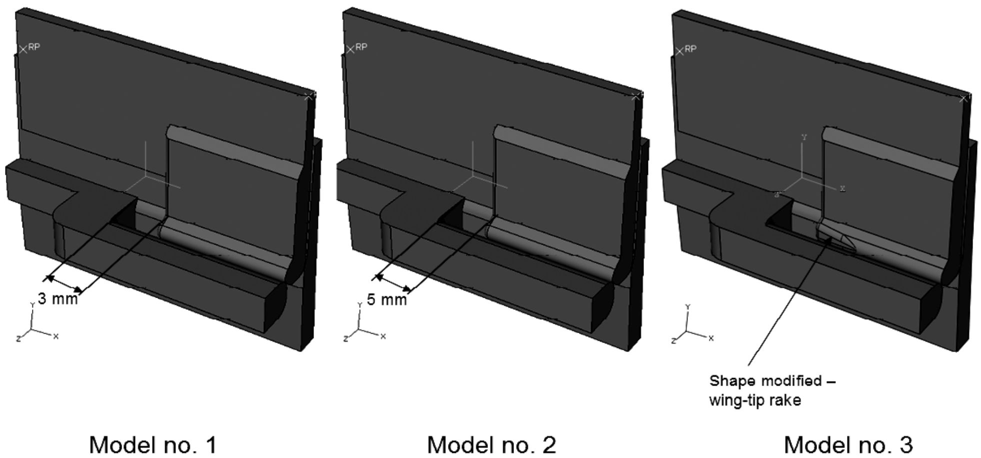

Fig. 2. Three versions of the designed mounting latch

Rys. 2. Trzy wersje projektowanego zatrzasku mocującego

When examining the designed element, it is necessary to perform discretization of the target model – this process is a critical stage for creation of a proper calculation model. The discretization method used for the given area defines the number of unknowns, size and shape of the elements and indirectly impacts the precision of the final problem solution. Figure 2 presents the three versions of the designed mounting latch, the surface of which was divided into sub-elements using Finite Elements Method (FEM).

5.2. Definition of boundary conditions in Abaqus system

Analyses for several versions of the latch were conducted in the Abaqus system. The main focus was paid to verification of deformations and mechanical stress in the designed component when closing and opening the fuse box cover. A phenomenon was simulated by using rigid components and also by defining appropriate boundary conditions. The simulation was carried out for the ambient temperature of 23°C and the humidity of 50%.

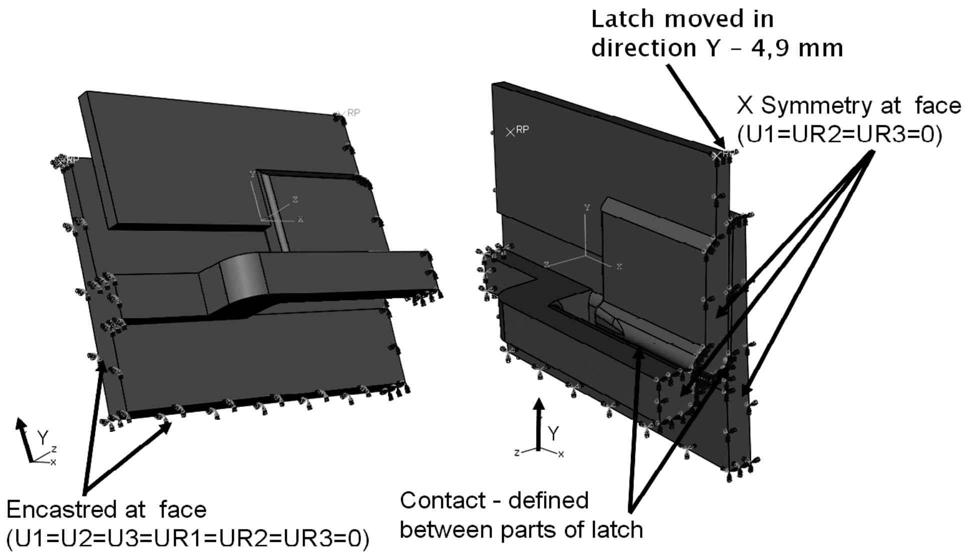

Fig. 3. Definition of contacts, boundary conditions as well as displacements occurring in the fixture

Rys. 3. Określenie kontaktów, warunków brzegowych, a także przesunięć występujących w uchwycie mocującym

6. Analysis of results

The Abaqus system was used to carry out three resistance analyses for the designed latch. Based on the information provided by LANXESS [3] manufacturing the selected material, it can be concluded that the said material can be deformed up to 2,6% with the maximum stress of 120 MPa. Assuming high safety coefficient, it can be concluded that the limiting value of the average strain for the designed component is equal to 2%. A safe limit of 75 MPa was assumed for the mechanical stress. Figures 4, 5 and 6 present distortion maps generated by the Abaqus system. Once the obtained results were examined in detail and cross-checked with the specification of the designed component, a geometry model number 2 was selected. The reaction strength obtained in model 3 was too low, while strain observed when opening the latch model 1 exceeded the value of 5%, which substantially exceeds the limit guaranteed by the material manufacturer.

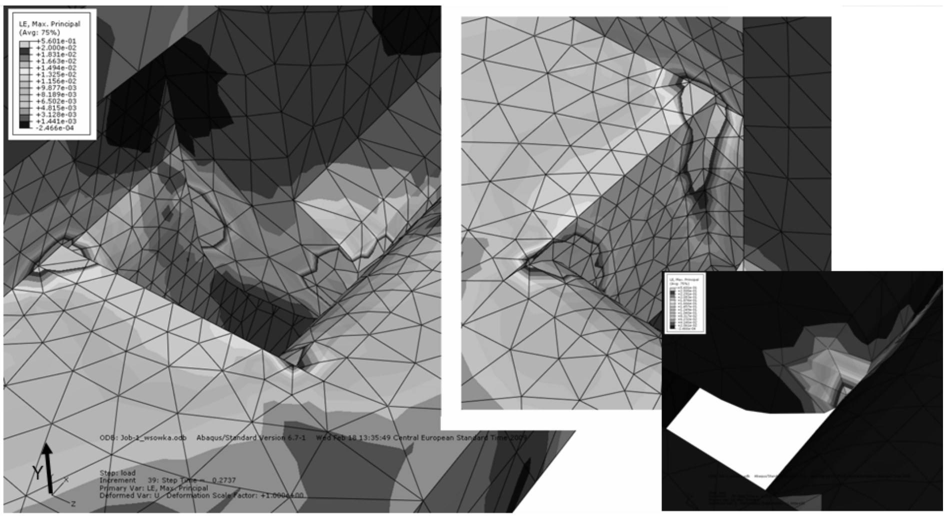

Fig. 4. Average strain for model 1; reaction strength in the direction Y is equal to 134N

Rys. 4. Odkształcenie średnie dla modelu 1; siła reakcji w kierunku Y wynosi 134 N

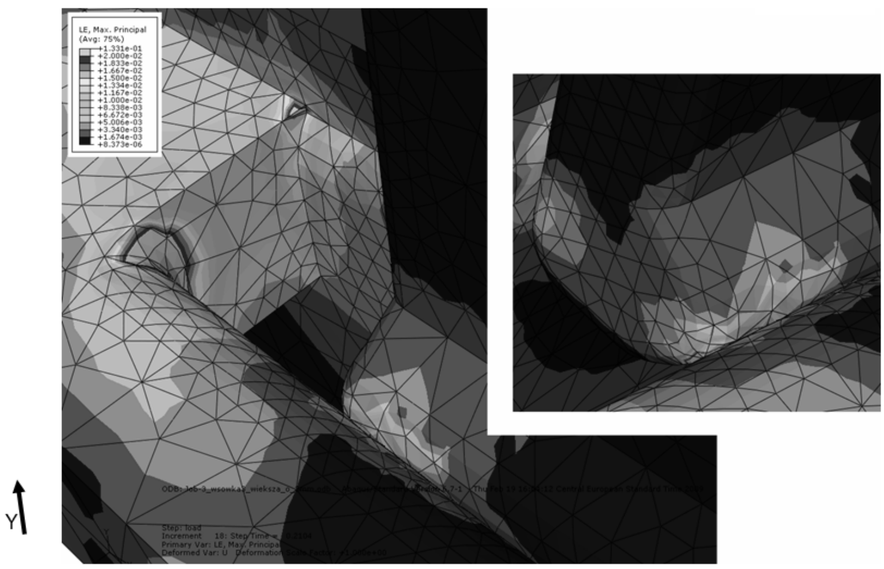

Fig. 5. Average strain for model 2; reaction strength in the direction Y is equal to 72N

Rys. 5. Odkształcenie średnie dla modelu 2; siła reakcji w kierunku Y wynosi 72 N

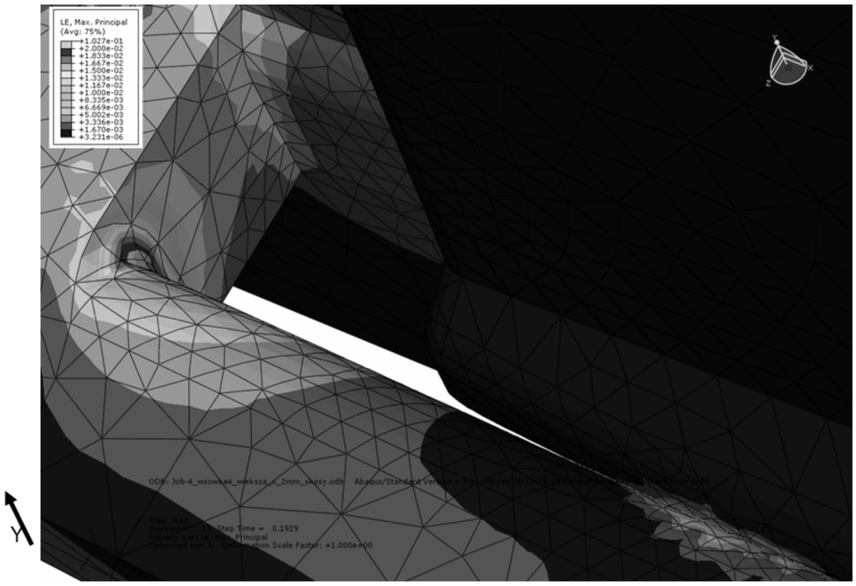

Fig. 6. Average strain for model 3; reaction strength in the direction Y is equal to 32N

Rys. 6. Odkształcenie średnie dla modelu 3; siła reakcji w kierunku Y wynosi 32 N

7. Conclusions

Meeting recommendations related with proper creation of computer geometric models for machine elements requires the designers to have large designing experience and familiarity with technological processes for the given type of elements, especially when elements are made from plastic materials. Moreover, it is also necessary to have experience in designing and modelling in computer CAD systems as well as to have skills required to link the knowledge in terms of CAX systems, basic machine design issues, design evolution as well as technological aspects. Therefore, the best solution during the process of designing machine parts of a specific type with a complex shape, is connected with elaboration of internal standards related with element structure, taking advantage of knowledge of experienced designers. Failure to meet such recommendations may cause problems in terms of modelling and extend the duration of this process, causing also loss of certain, partial model data. Even improper order of operation may increase the work-load related with geometric modelling.

Below, we present a few most often mentioned recommendations, which should be observed by designer when modelling [7].

- During the modelling process, it is necessary to strive to mimic the manufacturing technology during the design stage as closely as possible, also in terms of individual operations and their sequence.
- The modelling methods should follow the general concept, which is also the recommendation requiring a wide design experience.

– It is also necessary to separate technological operations in the created model as well as attempt to group model features bound together in terms of construction and technology, which in turns allows to order its structure, though sometimes it may cause excessive increase in the complexity of a structural tree of the model features.
– It is necessary to know the manufacturing and design basis in the elements, in the effect of which during the design process, a designer should define the construction and technological bases for the designed elements.
– It is suggested to attempt to maintain all the associations between the model and documentation.

References

[1] ABAQUS – Analysis user's Manual, 4'2009.
[2] Ćwikła K., *Projektowanie 2D i 3D wady i zalety*, CADCAM Forum, maj 2000.
[3] Dietrich M., *Podstawy konstrukcji maszyn*, WNT, Warszawa 1995.
[4] Frącz W., *Projektowanie i wytwarzanie elementów z tworzyw sztucznych*, Oficyna Wydawnicza Politechniki Rzeszowskiej, Rzeszów 2005.
[5] Lecyk P., *Co to jest CAD/CAM?* http://masters.ckp.pl/plecyk/, kwiecień 2009.
[6] Skarka W., Mazurek A., *CATIA. Podstawy modelowania i zapisu konstrukcji*, Wydawnictwo HELION, Warszawa 2005.
[7] Wełyczko A., *CATIA V5. Przykłady efektywnego zastosowania systemu w projektowaniu mechanicznym*, Wydawnictwo HELION, Warszawa 2005.
[8] www.CAMPUSplastics.com, CAMPUS 5.1, Baza danych materiałowych, marzec 2009.
[9] Samek A., *Projektowanie uchwytów obróbkowych,* Wydawnictwo Politechniki Krakowskiej, Kraków 1971.
[10] Wiśniewski P., *Efektywność zintegrowanych systemów CAD/CAM*, CADCAM Forum, październik 2007.