Article citation info:
Dębski H, Koszałka G, Ferdynus M. Application of fem in the analysis of the structure of a trailer supporting frame with variable operation parameters. Eksploatacja i Niezawodnosc – Maintenance and Reliability 2012; 14 (2): 107–114.

Hubert DĘBSKI
Grzegorz KOSZAŁKA
Mirosław FERDYNUS

APPLICATION OF FEM IN THE ANALYSIS OF THE STRUCTURE OF A TRAILER SUPPORTING FRAME WITH VARIABLE OPERATION PARAMETERS

WYKORZYSTANIE MES W ANALIZIE STRUKTURY NOŚNEJ RAMY NACZEPY O ZMIENNYCH PARAMETRACH EKSPLOATACYJNYCH*

This work presents a numerical analysis of the structure of a trailer frame of adjustable length and an increased load capacity designed for the transportation of oversize loads. The study was aimed at developing adequate numerical FEM models which would allow identification of the effort of the structure and the state of strain under operating loads. The analysis was carried out using ABAQUS/Standard, a numerical tool which enables calculations in the geometrically non-linear range with the use of the incremental–iterative Newton–Raphson method. As a result of the analysis, trouble areas in the frame were found in which dangerous stresses occurred. This enabled modification of the structure, leading to a reduction of the stresses to a safe level.

Keywords: *finite element method, supporting frame, strength calculations, trailer, oversized transport.*

W pracy zaprezentowano numeryczną analizę konstrukcji ramy naczepy o zmiennej długości i podwyższonej nośności przeznaczonej do transportu ładunków ponadnormatywnych. Prowadzone badania miały na celu opracowanie adekwatnych modeli numerycznych MES umożliwiających identyfikację wytężenia konstrukcji oraz stanu odkształcenia w warunkach obciążeń eksploatacyjnych. Zastosowanym do analizy narzędziem numerycznym był program Abaqus/Standard, umożliwiający prowadzenie obliczeń w zakresie geometrycznie nieliniowym z wykorzystaniem przyrostowo-iteracyjnej metody Newtona-Raphsona. W wyniku przeprowadzonych prac ustalono newralgiczne obszary ramy w których występowały niebezpieczne naprężenia. Umożliwiło to modyfikację konstrukcji pozwalającą na zmniejszenie naprężeń do bezpiecznego poziomu.

Słowa kluczowe: *metoda elementów skończonych, rama nośna, obliczenia wytrzymałościowe, naczepa, transport nienormatywny.*

1. Introduction

Design of modern structures is a complex task that requires taking account of numerous material and mechanical features when optimizing the geometry of the structure being designed. This concerns, in particular, critical structural elements, whose load capacity determines the strength of the entire supporting system. The process of designing such elements necessitates the use of modern tools which make it possible to search for the best design solutions [10–12]. A contemporary tool that provides a wide range of possibilities of analyzing the strength parameters of a structure being designed is CAE numerical software, which uses the finite element method [3–6, 8]. Such software currently finds broad application in many branches of industry, in particular the aerospace, aircraft and automotive industries.

A group of supporting structures that have to meet high strength and stiffness requirements are the frames used in the modern means of transport. Among design solutions that are subjected to the action of particularly high operating loads are frames of trailers for the transportation of oversize or overweight loads. Designing of this type of structures requires taking account of the various operating configurations of the trailer which make it possible, when necessary, to extend the length and width of the load deck [9–10]. A change of the configuration of the structure during its operation considerably affects the change in the character of its loading, which requires additional consideration in the process of designing and construction of the structure.

This study presents a conception of a central frame whose length can be changed over a wide range depending on the operating needs. A solution based on two mating thin-wall box beams was adopted in which frame length was adjusted by mutual sliding of the load-bearing members in and out of each other. In the calculations, structural details were taken into account, such as process holes, which could have a significant effect on the assessment of the stress-strain state of the structure of the frame. The calculations were done us-

---

(*) Tekst artykułu w polskiej wersji językowej dostępny w elektronicznym wydaniu kwartalnika na stronie ***www.ein.org.pl*** ing the Abaqus/Standard software, a modern numerical tool that uses the finite element method in the geometrically and physically non-linear range [1, 2, 6, 8].

2. Goal and scope of the study

The aim of the study was to assess the strength and stiffness of the design solution developed for a trailer support frame intended for the transportation of long loads of up to 45 tons. In the adopted concept of the frame, consisting of two independent parts – front and rear, length could be easily changed by sliding the front part of the frame in and out of its rear part. This enabled adjustment of the length of the frame to the transported load. The new structural solution required precise analysis of the state of strain and effort, which was conducted using the finite element method.

Prior to FEM calculations, the bending moments acting on the trailer frame were determined analytically for different combinations of its length and distribution of the transported load. The analysis covered only those cases which were deemed probable and permissible (e.g., loading of a maximally extended frame with a point force, of a value corresponding to the maximum permissible load weight, applied in the middle of its length was considered unlikely and was also listed in the trailer user's manual as impermissible). On the basis of the above considerations, cases for FEM analyses were chosen (Fig. 1).

Because the analysis of the structure was carried out in the range of operating loads, thus not allowing permanent deformations to develop in structure members, a decision was taken to adopt a linear elastic material model. Taking into consideration the contact interactions applied to the numerical model and the possibility of occurrence of large deformations, it was decided that the calculations should be carried out in the geometrically non-linear range using the incremental–iterative Newton–Raphson method [2, 6, 8].

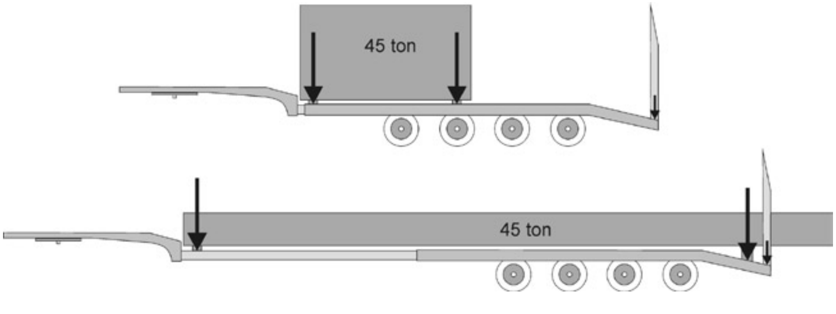

Fig. 1. Schematic diagrams of loading of a retracted (a) and an extended (b) frame with a load

3. Development of a numerical FEM model

Two configurations of frame operation were considered: a retracted frame configuration and a maximally extended frame configuration. In both variants, the same design and material parameters were used. The structure of the frame was designed so that its rear part, being the basic element of the structure for which the boundary conditions are determined by the vehicle suspension system, should simultaneously provide support for the front part, the movable element of the frame. The length of the frame could be changed by sliding the front part in and out of the rear part structure of the frame. A general view of the numerical model of the frame is shown in Fig. 2

The loading of the numerical model consisted of the structure's own weight as well as point mass loading deriving from the weight of the load of 45 tons, additionally taking into account 20% of dynamic surplus. A schematic diagram of the distribution of the external load in the form of point masses for the two frame configurations is shown in Fig. 2.

The structure of the spatial discrete model of the frame was based on shell-type elements having six degrees of freedom per element node [1]. In the joining area between the front and rear part of the frame, a solution was adopted which consisted in using flexible cushions which enlarged the mating area, and, by the same token, eliminated adverse local edge pressures in the zone of the joint. The mutual interactions between the front and the rear part of the frame were reproduced by defining contact interactions between the mating elements.

4. Boundary conditions of the discrete model

The principal task in developing a discrete model of the structure of the frame was to reproduce the operating conditions of the air suspension, which ensures identical reactions for each trailer wheel axle during operation of the vehicle. It was decided that the control arm would be reproduced by means of beam elements of specific stiffness for which different support conditions were defined in the front and rear node of the model of the control arm. The connection between the control arm and the frame in the front node ensured rigid transfer of load from the suspension onto the structure of the frame, allowing the arm only to rotate relative to the wheel axles. The rear node of the control arm was connected with the frame via an spring reproducing the air bag of the air suspension. The stiffness of the element modelling the air bag was selected so as to ensure similar reactions in the axles of all trailer wheels. The support of an individual wheel was realized by depriving the middle node of the control arm (in the connection node between beam elements) of the possibility of displacement in the direction of the X axis – Fig. 3a.

For the front part of the frame, support of the kingpin was defined by disabling displacement of kingpin nodes in the direction of the Y and Z axes – Fig. 3b. Moreover, in the direction of the X axis (vertical movements of the kingpin), support of nodes was introduced in the form of elastic response of an appropriately specified stiffness. Additionally, for the entire model, axial symmetry conditions were defined (along the axis of the frame – the Y axis) by depriving the nodes located on the model's symmetry axis of the possibil-

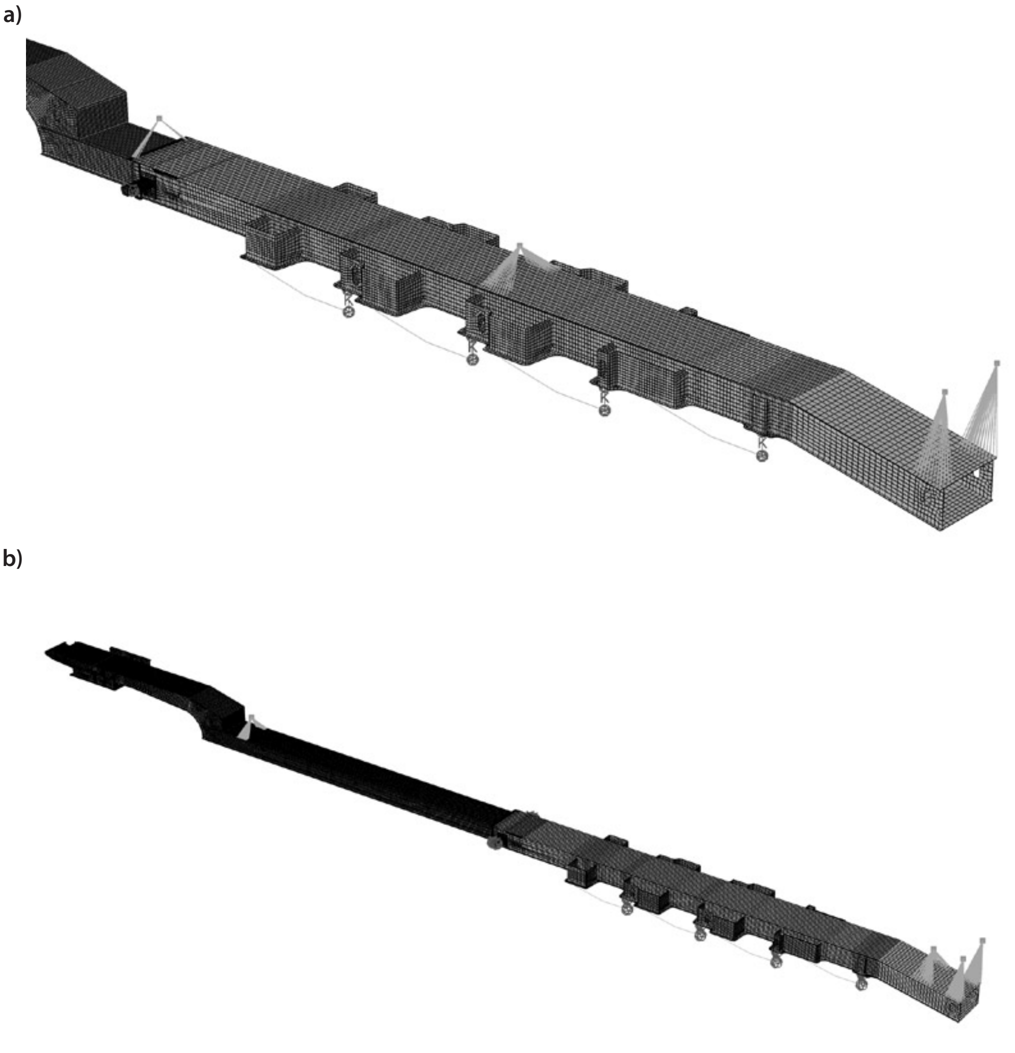

Fig. 2. Discrete models of the frame indicating the manner in which external loading was modelled: a) retracted configuration, (b) extended configuration

ity of displacement in the direction of the Z axis and rotation relative to the X and Y axes. Elimination of these degrees of freedom ensured stable work of the model without unwanted displacements of the structure in the direction crosswise to its axis (sideways) – Fig. 3b.

5. Results of numerical analysis

The numerical calculations conducted in this study made possible evaluation and comparison of the state of strain and effort of the structure of the frame for both of the considered operating variants – the retracted and maximally extended frame. The obtained maps of distribution of H-M-H reduced stresses in the elements of the structure pointed to the areas in which those stresses considerably exceeded the adopted yield point of the material Re = 360 MPa. The most adverse stress gradients occurred in the sliding part of the frame and in the joining zone between the two load-bearing members, both in the case of the retracted and the extended configuration of the frame – Fig. 4.

The very high levels of reduced stress occurring in some areas required elimination, which called for a better design solution. Therefore, modifications of structural details were introduced in the current solution. They primarily involved a change in the location of the process holes in the side walls of the frame, which were a source of dangerous stress gradients. The openings were removed and replaced with one hole in the lower beam wall. Moreover, flat bars for the reinforcement of the side walls of the frame were introduced in the risk zone and reinforcing ribs in the gooseneck were added – Fig. 5. As a result of such activities, successive variants of the structure were being developed, which were then subjected to FEM analysis. The newly introduced modifications made possible elimination of the trouble zones in the supporting system. Fig. 6. shows the results of FEM calculations for the final variant of the structure.

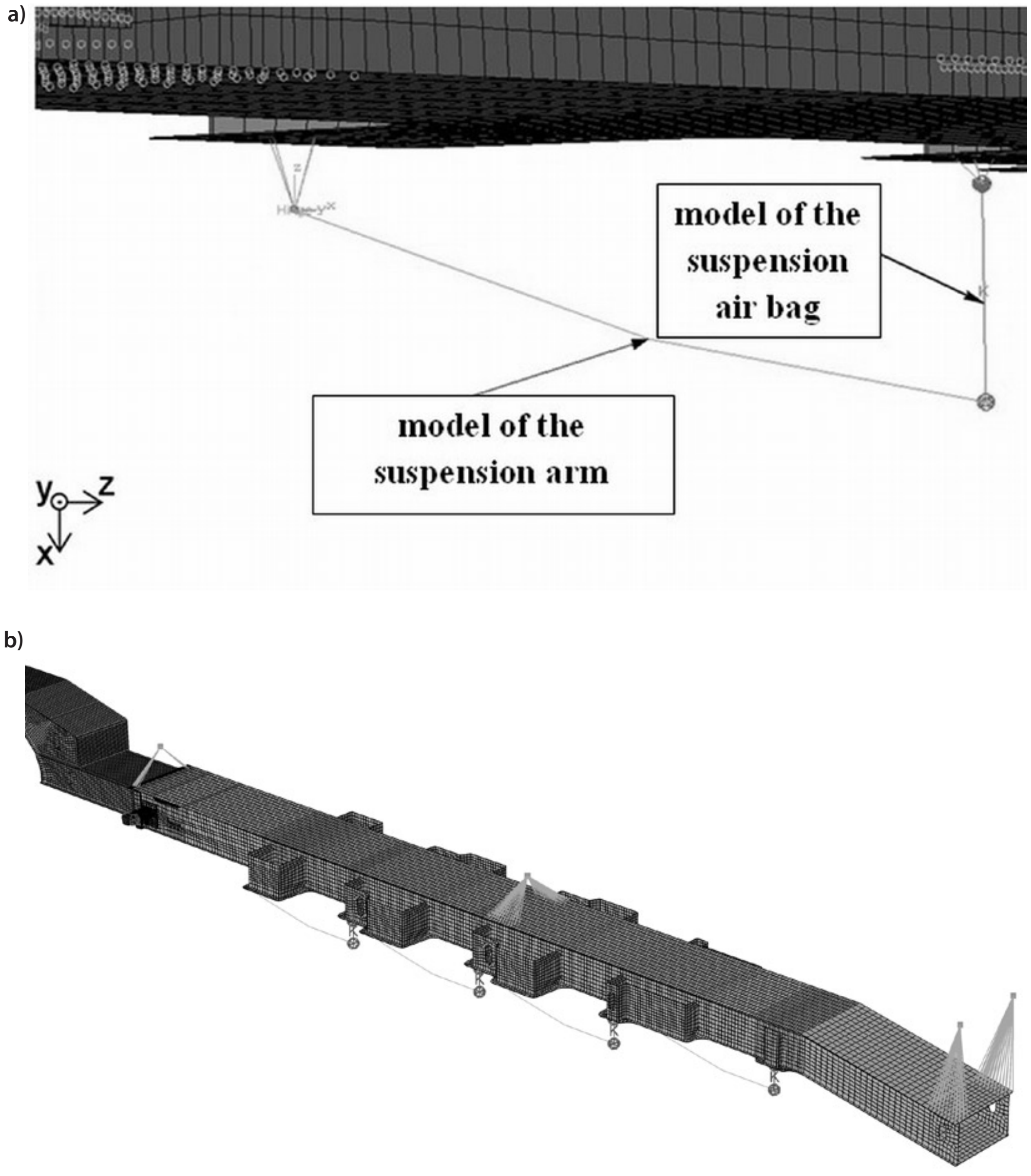

Fig. 3. Boundary conditions of the FEM model: a) model of the wheel suspension, b) support of the frame's kingpin and the model's axial symmetry conditions

6. Conclusions

The numerical analysis of the structure of the support frame of a trailer conducted in this study enables evaluation of the adequacy of the newly developed design solution, allowing identification of the trouble areas determining the strength of the entire structure. This is extremely important in cases where, in the process of looking for new design solutions, too many unknown design parameters occur at the stage of designing complex load-bearing members. The knowledge of stress distribution in the critical elements is then an issue of primary importance, and the application of the finite element method allows analysis of the effort of a structure as early as the design stage [6, 8, 9].

Use of numerical FEM calculations in the design process enables fast and effective introduction of indispensable modifications of structural details, leading to the creation of successive, ever more adequate design variants which, ultimately, make it possible to develop an optimum solution.

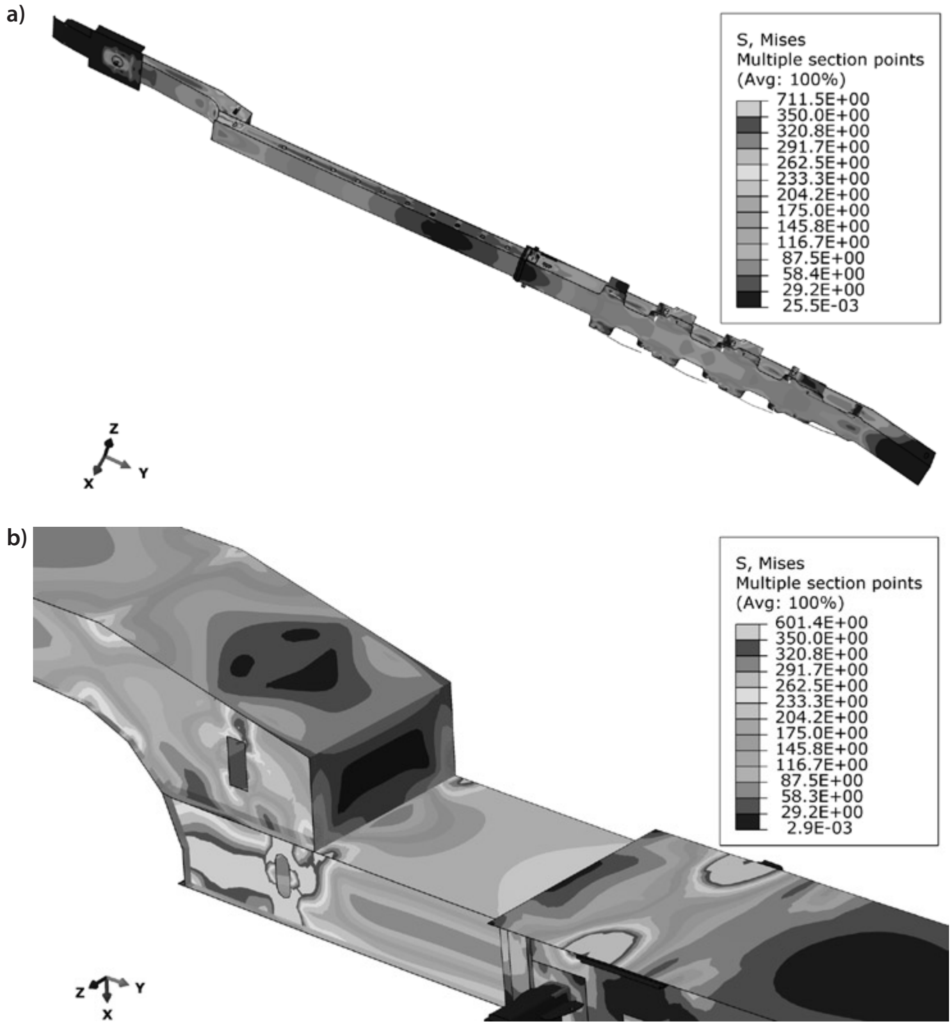

Fig. 4. H-M-H stress distribution in the model of the frame: a) retracted frame – general view, b) extended frame – detail of the joining zone

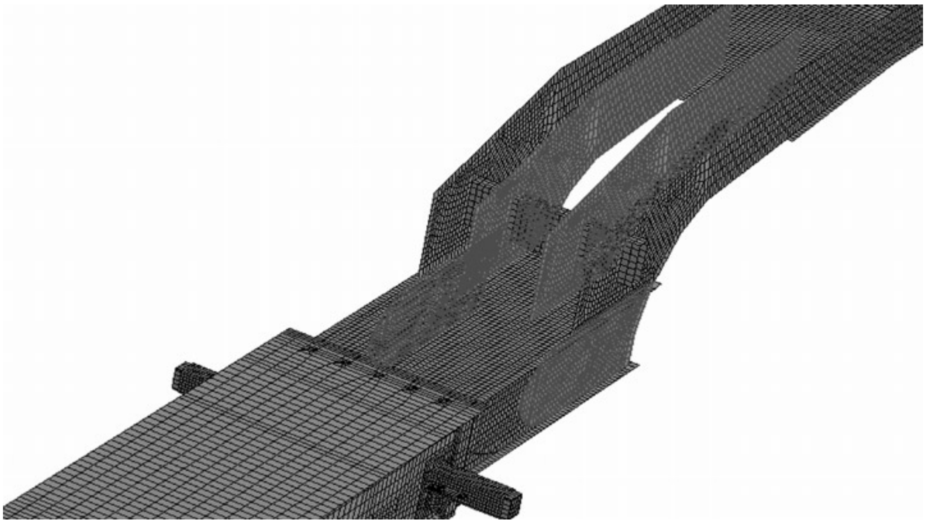

Fig. 5. Cover plates and ribs reinforcing the front part of the frame

Fig. 6. H-M-H stress distribution in the modified model of the frame: a) general view b) detail of the front part of the frame

References

1. Abaqus HTML Documentation, 2010.
2. Bathe K J. Finite element procedures. Prentice Hall, 1996.
3. Dacko M, Borkowski W, Dobrociński S, Niezgoda T, Wieczorek M. Metoda elementów skończonych w mechanice konstrukcji. Arkady, Warszawa 1994.
4. Grądzki R. Wprowadzenie do metody elementów skończonych. Politechnika Łódzka, Łódź 2002.
5. Koszałka G., Niewczas A., Dębski H., Golec M., Kaczor M., Taratuta L. FEM analysis in design of extendable central beam for a semi-trailer. Machine Design, 2012(3);1: 47–50.
6. Osiński J. Obliczenia wytrzymałościowe elementów maszyn z zastosowaniem metody elementów skończonych. Oficyna Wydawnicza PW, Warszawa 1997.
7. Rakowski G, Kacprzyk Z. Metoda Elementów Skończonych w mechanice konstrukcji. Oficyna Wydawnicza PW, Warszawa 2005.
8. Rudawska A., Dębski H. Experimental and numerical analysis of adhesively bonded aluminum alloy sheets joints. Eksploatacja i Niezawodnosc – Maintenance and Reliability, 2011; 1: 4–10.
9. Rusiński E, Czmochowski J, Smolnicki T. Zaawansowana metoda elementów skończonych w konstrukcjach nośnych. Oficyna Wydawnicza Politechniki Wrocławskiej, Wrocław 2000.

10. Rusiński E, Kowalczyk M. Zasady przedłużania ram samochodów ciężarowych. Samochody Specjalne, 2003; 6: 78–81.
11. Rusiński E, Porada D. Komputerowe wspomaganie projektowania ram samochodów ciężarowych. Samochody Specjalne, 2000; 6: 72–73.
12. Tarnowski W. Podstawy projektowania technicznego. WNT, Warszawa 1997.
13. Winkler T. Komputerowy zapis konstrukcji. WNT, Warszawa 1989.

Hubert DĘBSKI, PhD (Eng.)
Grzegorz KOSZAŁKA, PhD (Eng.)
Mirosław FERDYNUS, PhD (Eng.)
Faculty of Mechanical Engineering
Lublin University of Technology
Nadbystrzycka 36, 20-816 Lublin, Poland
E-mail: h.debski@pollub.pl
E-mail: g.koszalka@pollub.pl
E-mail: m.ferdynus@pollub.pl