Janusz RUTKOWSKI
Przemysłowy Instytut Maszyn Rolniczych, Poznań, Poland
e-mail: office@pimr.poznan.pl

Received: 2018-08-07 ; Accepted: 2018-11-27

THE IMPACT OF THE MODELLING METHOD OF THE FRONT LOADER ON THE ACCURACY OF THE FEM CALCULATIONS RESULTS

Summary

The article discusses selected issues of front loader modelling for the needs of FEM calculations. Particular attention was paid to the necessity of correct modelling clevis pin connections and hydraulic cylinders. In the case of hydraulic actuators, the effects of a different approach in modelling the actuators on the quality of the structure under operating load were demonstrated. Differences in the modelling cylinders equipped with a hydraulic lock and cylinders connected in parallel without locks were taken into account. The results of calculations obtained on the examples are discussed and the methodology of self-control of the results of FEM computations is presented. Some examples of various machines and possible errors during strength analyzes were also presented.

***Key words**: FEM computations, finite elements method, strength computations of agricultural machines*

WPŁYW SPOSOBU MODELOWANIA KONSTRUKCJI NOŚNEJ ŁADOWACZA CZOŁOWEGO NA DOKŁADNOŚĆ WYNIKÓW OBLICZEŃ MES

Streszczenie

W artykule omówiono wybrane zagadnienia modelowania ładowacza czołowego dla potrzeb realizacji obliczeń MES. Zwrócono szczególną uwagę na konieczność prawidłowego modelowania połączeń sworzniowych oraz siłowników hydraulicznych. W przypadku siłowników hydraulicznych wykazano skutki różnego podejścia w modelowaniu siłowników na jakość pracy konstrukcji pod obciążeniem eksploatacyjnym. Uwzględniono różnice w modelowaniu siłowników, wyposażonych w zamek hydrauliczny i siłowników połączonych równolegle bez zamków. Omówiono wyniki uzyskanych obliczeń na przykładach oraz przedstawiono metodykę samokontroli wyników obliczeń MES. Przedstawiono także kilka przykładów różnych maszyn i możliwych do popełnienia błędów podczas analiz wytrzymałościowych.

***Słowa kluczowe**: obliczenia MES, metoda elementów skończonych, obliczenia wytrzymałościowe maszyn rolniczych*

1. Introduction

In the finite element computations (FEM), various issues are relevant regarding the way of modelling and mapping real conditions. The paper is limited to considerations involving linear static analysis, it mean a typical analysis of the strength of machines. In this case, the calculation model contains information about the shape of the structure, dimensions, wall thickness of the beams, interaction of welded, bolted, riveted and pin jointed, further, on the support method and static load or static equivalent of the dynamic interaction (replacing the strength of the variable with one instantaneous value). The quality of the model depends to a large extent on the quality and correctness of creating a computational model, i.e. a model consisting of a finite element mesh. Important factors here are local geometry simplifications, especially in places of shape changes or cross-sections, the size and type of finite elements used to build a model and how they are connected to each other in motion nodes. Information on this subject can be found in the specialized literature, in the help files for FEM programs and scientific publications (e.g. [1, 13]). However, the experience of the person preparing the calculation model is the most important for the accuracy. In the real, the knowledge and experience of a man guarantees achieving true and reliable results. The paper shows which factors and how and to what extent they are effect of calculation errors. An attempt was also made to estimate exactness of calculations, although, giving into consideration nature of some tests, the given error values should be taken for orientation purposes only. All calculations were made in the system I-DEAS NX 6.5.

2. Research subject

The subject of done strength analyses was the innovative front loader of the Polish manufacturer (Fig. 1).

Source: own study / Źródło: opracowanie własne

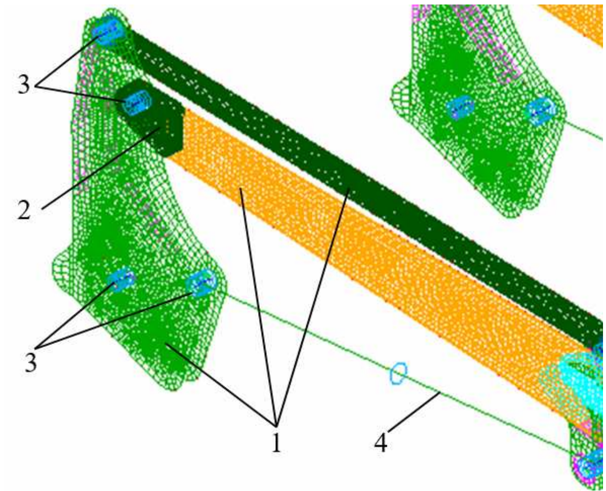
Fig. 1. 3D model of front loader
Rys. 1. Model 3D ładowacza czołowego

The loader is designed for large tractors with 140-200 kW engine power. It was a typical agricultural loader. Its main arm is rigid, composed of two stringers connected by a pipe beam. Hydraulic cylinders, work in pairs and cause the rotation of the arm and the rotation of the bucket (work tool), installed on the frame. Fixing of horizontal position of the work tool while lifting the load ensures the straight-line mechanism. The front loader is attached to the tractor on two pillars.

The FEM analyzes were conducted to check the distribution and values of stresses in the loader structure and to determine the forces in the actuators. The article, however, focuses on the issues of the exactness of running FEM simulations, not on the results themselves, as the result of the loader's computational analysis. The loader served only as an example in this regard.

3. Computational model details

The model showed a large variation in the thickness of metal sheets, from 4 to 45 mm, and in some places there were made double-layered overlays. The largest sheet thicknesses were formed in mono-blocks with holes for clevis pins and they were located on the end of the arms. Pin joints were embedded in sleeves with sliding inserts. Such a specificity of the structure, in which there is a large variation in thickness and there are moving parts, requires the building of a hybrid computational model, i.e. containing various types of finite elements (Fig. 2).



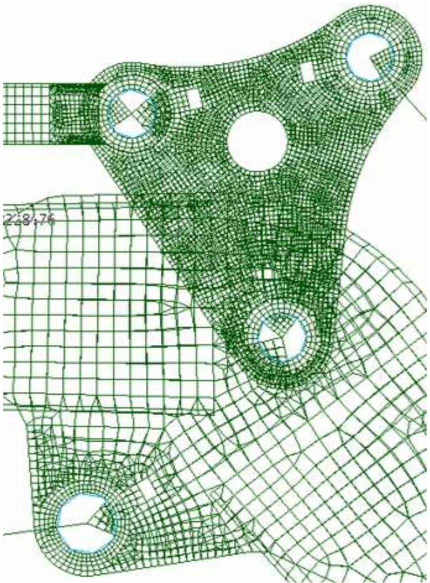
Source: own study / Źródło: opracowanie własne

Fig. 2. Finite elements types used in FEM model of front loader (markers description in the text)

Rys. 2. Rodzaje elementów skończonych w modelu MES ładowacza (opis oznaczeń w tekście)

The main structure has been reproduced with flat, two-dimensional plate and shell elements (1), which are best suited for modelling thin-walled structures. Fragments with thick mono-blocs were left as solids, which were modelled by solid-type finite elements (2). For the compatibility of the grid, all elements were rectilinear. No curved elements were used. This choice is justified by tests on primitive examples, about which will be said in the paper later. Beam-type finite elements were applied to modelling clewis pins connected with flat mesh in pin holes (3). This combination of flat mesh and beams has a low torsional stiffness and it allows to reproduce the freedom of rotation of the pin. The holes in the lugs were not fully filled, but the grid was filling them with about ¼ of circumference. This allowed to transfer the pin force to the lug hole only on the fragment of the circumference (Fig. 3). The hydraulic cylinders, whereas, are represented by rod-type finite elements that carry only the longitudinal force (4). More on this approach of modelling of pin connections is in [9].

The density of the finite element mesh was adjusted to the local conditions, and so around the holes of the pin bolt the mesh was more dense, the beam of the beam had a medium density, and the largest eyes of the mesh were on the bucket, which served only to inflict loads (Fig. 3). Problems of mesh density are also discussed in numerous literature items, e.g. in [2].



Source: own study / Źródło: opracowanie własne

Fig. 3. The way of lug modelling

Rys. 3. Sposób modelowania uch sworzni

The final stage of the preparation of the calculation model included the implementation of the model restraints (removal of global degrees of freedom) and the loads from exploitation forces as well as gravitational acceleration, which, acting on the modelled elements, generates the load come from the own weight of the structure.

The material assigned in the properties of finite elements was the same for the entire computational model and corresponded to the general properties of steel (Young module E = 2,068e+11 Pa, Poisson ratio ν = 0,29, Kirchoff module G = 8,0155e+10 Pa, density ρ = 7 820 kg·m^{-3}).

4. Loads and load cases

The load cases included different front loader positions, different load distribution and values, and different situations of hydraulic cylinders modelling, which will be discussed later in the paper. So, the loader had three calculation positions:

- transport position, when the bucket with the load is turned upwards and it is at the height of the tractor front,
- position of pushing and picking up the load when the bucket is at its lowest position and there exists both load from pushing and from picking up the load on the bucket,
- position when the bucket is raised up with the load in it.

The loads corresponded to the loader work positions and contained accordingly:

- nominal gravity force of load 27 500 N,
- come from dynamic overloads during transport, when the tractor moves over road unevenness (dynamic surplus of 50% of nominal load was assumed),
- pushing force of the bucket in two variants of 55 000 N and 103 000 N, which resulted from the adhesion of the tractor wheels to the ground,
- the inertia of the raised load when the tractor brakes, 13 700 N.

In addition, for the case of load picking the load distribution was varied in two variants: the entire bucket load was symmetrical or oriented only on the bucket half, causing load asymmetry. Details of the whole calculation cases and results are available in the attachment to the report on the research report [5].

Restraints of the model were carried out in the same way for all models and they were in the places where the front loader is attached to the agricultural tractor. In the further part of the paper, only selected results from these calculations will be presented regarding the analysis of computational accuracy.

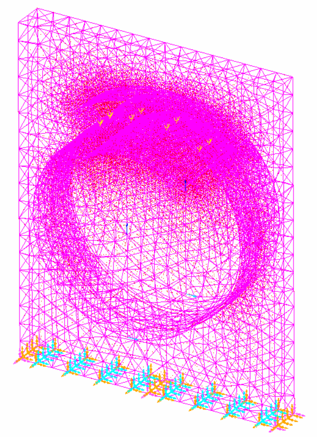
5. Study of the impact of the lug modelling on the accuracy of computations

The method of modelling the lugs has a significant impact on the quality and exactness of the obtained calculation results. The more accurate calculation model is, the more accurate and more reliable the results are. For comparison, in [11] the author discusses the issues of different modelling of screw joints, paying attention to the suitability of calculations. In [2], in turn, we have a discussion on the impact of mesh density in places of notches on the type of used finite elements.

In the case of pins modelling, the most accurate calculation results can be obtained when lugs with pins are calculated taking into account the phenomenon of contact interaction. Although this is also not the rule. Such calculations require the special preparation of a computational model in which contact finite elements are generated, preferably with dense mesh. Also the solver changes to the iterative one, i.e. which computes interaction on the contact surface in this way, that in the next steps it will lead to a balance between the contact forces and the forces inside material dependent on the deformation of the structure. This are long-lasting and time-consuming computations. They can be made for simple machine components, e.g. one pin joint, but the conversion of the entire machine, where such lugs are is virtually unjustified (if it is doable at all). Similar problem was solved in [10]. There the computation concerned contact of cistern cover to the cistern body. The way of modelling has been shown there.

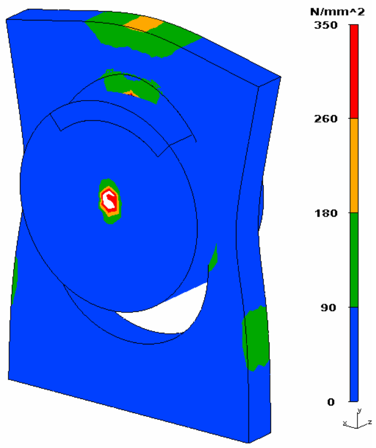
Fig. 4 shows an example of a simple lug model with a pin that has been modelled with the help of contact elements. In the model there are separate surfaces of the pin and hole, and between them the contact is calculated. In the place where the contact is expected to occur, the mesh is compacted. The elements are solid, curved (hyperbolic). A clearance of 0.5 mm between the pin and the hole is also taken into account. This model will serve as a reference for further consideration. Reader can find more about research on differences between element types (rectilinear, hyperbolic), for example, in [4].

Fig. 5 presents the result of calculations in the form of a distribution of reduced stresses. Whenever the paper will talk about reduced stresses, it will be about reduced stresses calculated according to the hypothesis of Huber-Misses. Stress values will be presented in the unit [MPa] or the corresponding [N·mm^{-2}], which depends on the capabilities of the FEM calculation system.



Source: own study / Źródło: opracowanie własne

Fig. 4. Reference model of lug with modelled contact
Rys. 4. Model referencyjny ucha z zamodelowanym kontaktem



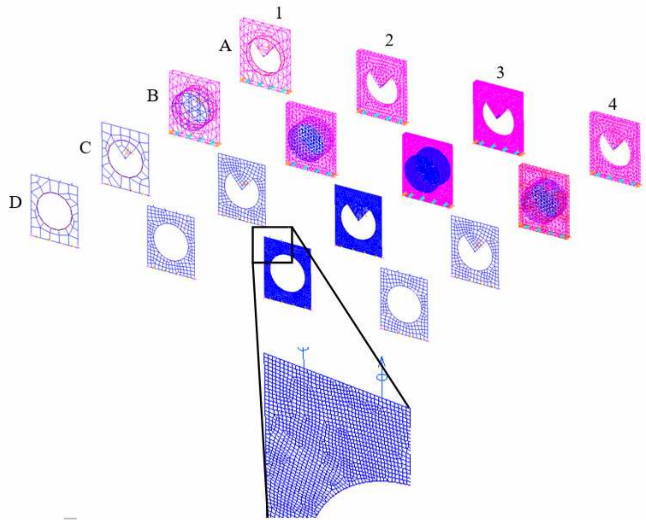
Source: own study / Źródło: opracowanie własne

Fig. 5. Reduced stress for reference model
Rys. 5. Naprężenia zredukowane w modelu referencyjnym

Such accurate modelling of the pins in the calculated loader is not doable. Therefore, needed simplification was applied, which was discussed in the chapter "Computational model details". This simplification makes the FEM model be continuous, so there is no place to calculate contact interactions. It allows to maintain the freedom of rotation of the pin, transfers the forces of the pin only to a part of the circumference of the lug holes and allows to determine the forces and stresses in the pin. However, what is the impact of the adopted simplification on the quality of FEM computations and on the value of stresses? To answer this question, 16 different simplified models were made (Fig. 6).

The specifications of individual models are as follows:

- row A – there are solid models, but the pin is "glue-sticked" to the hole and modelled in a fragmentary way,
- row B – models also are made in solid, but the pin is full-filled and contact behaviours are computed,
- row C – contains surface models, with a pin (the same way are modelled in the loader calculation),



Source: own study / Źródło: opracowanie własne

Fig. 6. Simplified lug models for comparison tests
Rys. 6. Uproszczone modele porównawcze uch

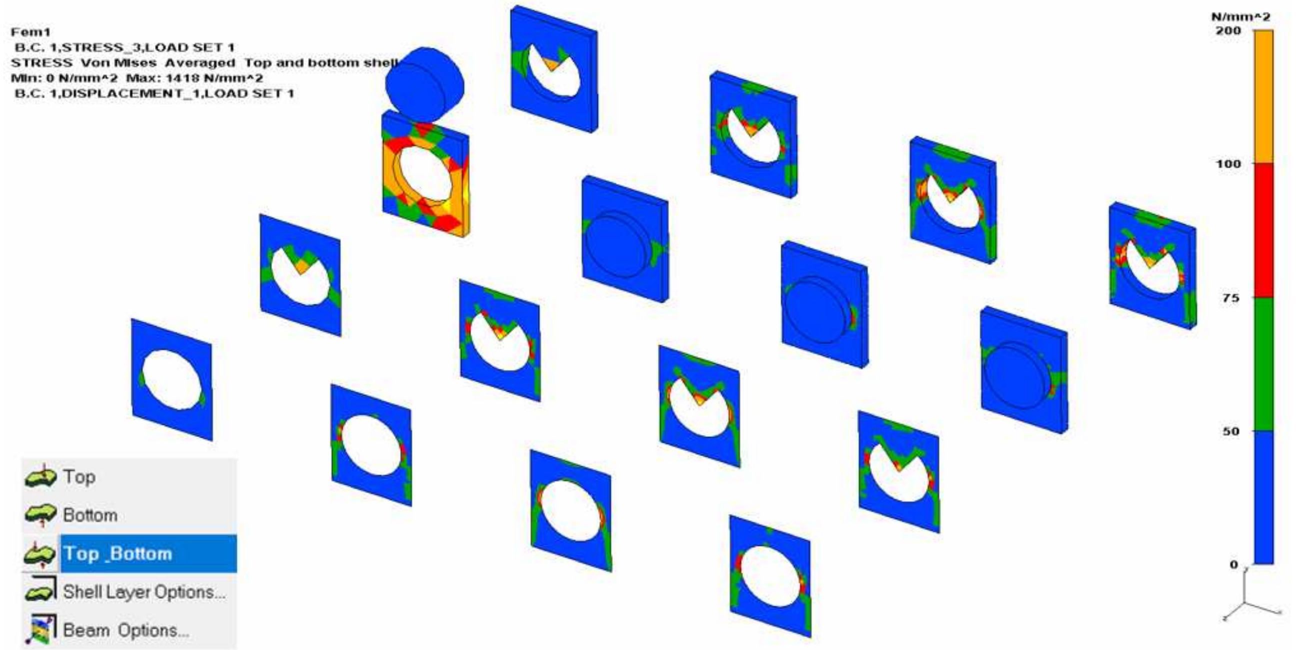
- row D - contains only the lug without pin and the load is implemented on the upper half of the circumference of the hole,
- row 1 - contains the most coarse rectilinear elements,
- row 2 - the grid has a medium size mesh (optimal), i.e. that one, which would be ideal for computations of the loader,
- row 3 - contains the smallest mesh with rectilinear elements,
- row 4 - is prepared like row 2, but in this row the elements are curvilinear (hyperbolic).

The bottom edge of the rectangle (lug) is fixed and the tensile force is applied to the centre of the pin (except for the order of row D). The magnitude of the force load applied to the lug was the same for all models. The size of lug as well as value of force is irrelevant to comparative considerations.

Reduced stresses were calculated for such prepared models. The results were presented for all models in one scale, set at a maximum of 200 MPa. The local stress values were read for individual elements. As can be seen in Fig. 7, there are clear differences in the obtained results. There is also a marked difference with the reference model. Although, in some places one can speak of a certain similarity, at least in terms of the magnitude of the stress. So the modelling method itself is already the source of some computational errors.

The method of preparing the model is not sufficient. At this stage, the very art of presentation and display of results is important for the adequacy of the calculation results. For comparison, in [4] is presented the effect of averaging the results of calculations too. The results visualization program offers various options for displaying the stress maps (Top, Bottom, Top & Bottom) and various averaging of results (or not) with options such as node averaging and averaging within an element, including: Contour, Maximum, Average, Centroid, Top & Bottom. Each of these options affects the change in the read values of stresses. Basically, one should use the node averaging option and the Average option for an element. For this association, the best convergence between simplified and reference models comes out, although it also depends on the type of elements. Basically, solid elements are not useful for selected options, but flat elements give the most similar results after choosing these two options. Thus, lack of knowledge in presenting result by choosing the wrong option will bring about the incorrect interpretation of results. It is a source of errors lying on the side of reading the results of calculations.

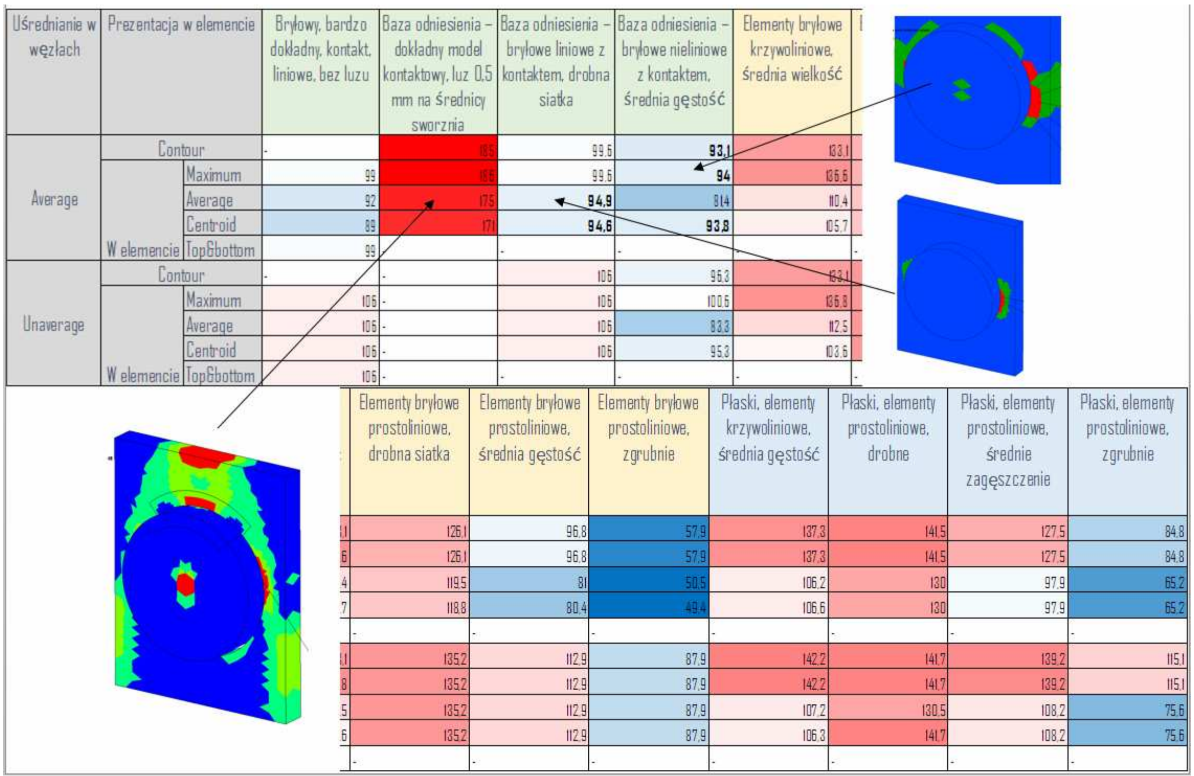
As the table in Fig. 8 shows, the stress values depend on both the type of model and the way in which the results are displayed. In the columns, specific simplified models are distinguished, but in the rows there are distinguished differences in presenting the results according to the display option in the computational system.



```
Fem1
 B.C. 1,STRESS_3,LOAD SET 1
STRESS  Von MIses  Averaged  Top and bottom shell
MIn: 0 N/mm^2  Max: 1418 N/mm^2
 B.C. 1,DISPLACEMENT_1,LOAD SET 1
```

Source: own study / Źródło: opracowanie własne

Fig. 7. Reduced stress in simplified models
Rys. 7. Naprężenia zredukowane w modelach uproszczonych



Uśrednianie w węzłach	Prezentacja w elemencie		Bryłowy, bardzo dokładny, kontakt, liniowe, bez luzu	Baza odniesienia – dokładny model kontaktowy, luz 0,5 mm na średnicy sworznia	Baza odniesienia – bryłowe liniowe z kontaktem, drobna siatka	Baza odniesienia – bryłowe nieliniowe z kontaktem, średnia gęstość	Elementy bryłowe krzywoliniowe, średnia wielkość	Elementy bryłowe prostoliniowe, drobna siatka	Elementy bryłowe prostoliniowe, średnia gęstość	Elementy bryłowe prostoliniowe, zgrubnie	Płaski, elementy krzywoliniowe, średnia gęstość	Płaski, elementy prostoliniowe, drobne	Płaski, elementy prostoliniowe, średnie zagęszczenie	Płaski, elementy prostoliniowe, zgrubnie
Average	Contour		-	185	99,6	**93,1**	133,1	126,1	96,8	57,9	137,3	141,5	127,5	84,8
	W elemencie	Maximum	99	186	99,6	**94**	136,6	126,1	96,8	57,9	137,3	141,5	127,5	84,8
		Average	92	175	**94,9**	81,4	110,4	119,5	81	50,5	106,2	130	97,9	65,2
		Centroid	89	171	**94,6**	**93,8**	105,7	118,8	80,4	49,4	106,6	130	97,9	65,2
		Top&bottom	99		-	-	-	-	-	-	-	-	-	-
Unaverage	Contour		-	-	106	95,3	133,1	135,2	112,9	87,9	142,2	141,7	139,2	115,1
	W elemencie	Maximum	106	-	106	100,6	136,8	135,2	112,9	87,9	142,2	141,7	139,2	115,1
		Average	106	-	106	83,3	112,5	135,2	112,9	87,9	107,2	130,5	108,2	75,6
		Centroid	106	-	106	95,3	103,6	135,2	112,9	87,9	106,3	141,7	108,2	75,6
		Top&bottom	106	-	-	-	-	-	-	-	-	-	-	-

Source: own study / Źródło: opracowanie własne

Fig. 8. Reduced stresses in simplified models in a tabular table (fragment)

Rys. 8. Naprężenia zredukowane w modelach uproszczonych w zestawieniu tabelarycznym (fragment)

Stress samples were taken at the narrow side of the lug, because there the interference from the modelling of the pin is smaller than at the top of the hole and the models can be compared better. The greatest value of stresses was taken there. The maximum value of reduced stresses was 186 MPa and the smallest value was 49.4 MPa. In the reference model, this value was about 100 MPa. The C version of the lug model produced satisfactorily accurate reproduction of the stress distribution and enabled the creation of easy-to-make pin connections in the loader.

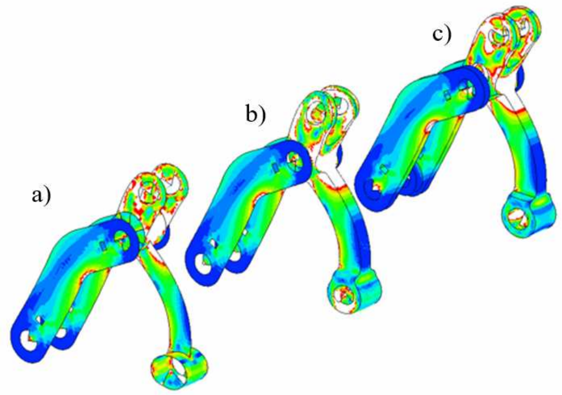
6. Hybrid modelling and exactness of FEM simulations

In the FEM calculation model, there are a situations where thick and thin parts are in contact. Then it is necessary to use finite elements with solid elements. Flat elements are able to replace even solid elements to a large extent. In the loader this situation concerned, among others the linkage system for rotation the bucket. Examining the effects of a different method of modelling the linkage system on the quality and exactness of FEM computations, three comparative models have been built: solid, surface and hybrid (Fig. 9).

The pin connection was modelled as the beam-type in the all three cases. The right linkage was the one which, due to the heterogeneous shape, was difficult to surface modelling. However, the results of the calculations show that, regardless of the modelling method, quite similar results can be obtained in all three cases.

7. The impact of the modelling method of hydraulic cylinders on the distribution and magnitude of stresses in the loader

In the case of modelling hydraulic cylinders in machines, it is important to consider whether or not the existence of hydraulic locks.



Source: own study / Źródło: opracowanie własne

Fig. 9. Comparison of stresses for three different approaches in modelling the bucket rotation linkage system: a) surface model, b) hybrid model, c) solid model

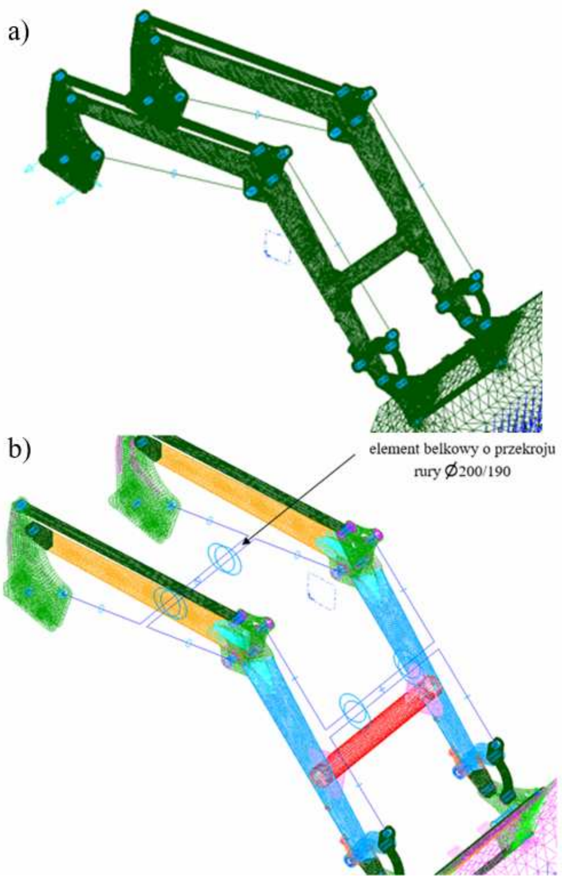
Rys. 9. Porównanie naprężeń dla trzech różnych podejść w modelowaniu cięgien obrotu czerpaka: a) model powierzchniowy, b) model hybrydowy, c) model bryłowy

This is not important when the actuator is working alone, such as in the frame of an excavator. However, when the actuators work in pairs, it has the significance of how they cooperate with each other, i.e. whether between the actuators during their work the oil can flow freely and the pressure can equalize or not.

The first and second variant causes equalizing of forces in the actuators because the hydraulic oil can flow freely between them. The third variant causes that both actuators work independently and it may be the case that one of them transfers predominant amount of the load by itself, and the other one only takes a bit.

From the point of view of loader calculations, all variants were possible due to assembly options. Therefore, two separate calculation models were constructed, in which the effects of both variants on stress distribution were compared (Fig. 10).

The simplest way of modelling the actuators in Fig. 10-a) concerns the situation with hydraulic locks and is also the simplest to do. The situation of parallel operation, when the forces in the actuators are equalized, is more difficult to reproduce in the FEM model. It is necessary to build a special swing made of beam elements, which results in the equalization of forces in hydraulic cylinders on both sides of the machine as in example in Fig. 10-b).



Source: own study / Źródło: opracowanie własne

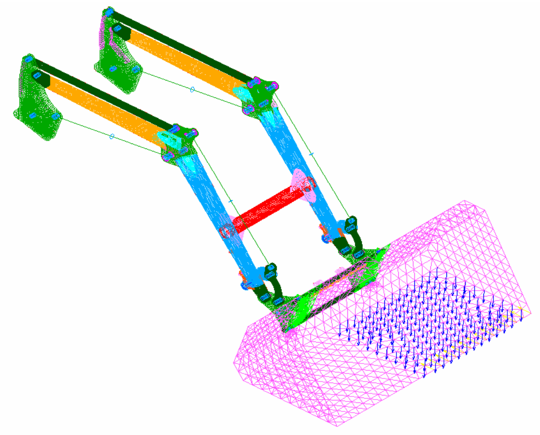
Fig. 10. Modelling hydraulic cylinders: a) with hydraulic locks, b) without hydraulic locks
Rys. 10. Sposób modelowania siłowników hydraulicznych: a) z zamkami hydraulicznymi, b) bez zamków hydraulicznych

However, the modelling the actuators does not matter when the loader is loaded symmetrically. But when load asymmetry appear, then the loader frame is twisted. Such asymmetry may occur, for example, when a rock is loaded and the is placed on the side of the bucket (Fig. 11).

The following variants of the hydraulic cylinders are possible:

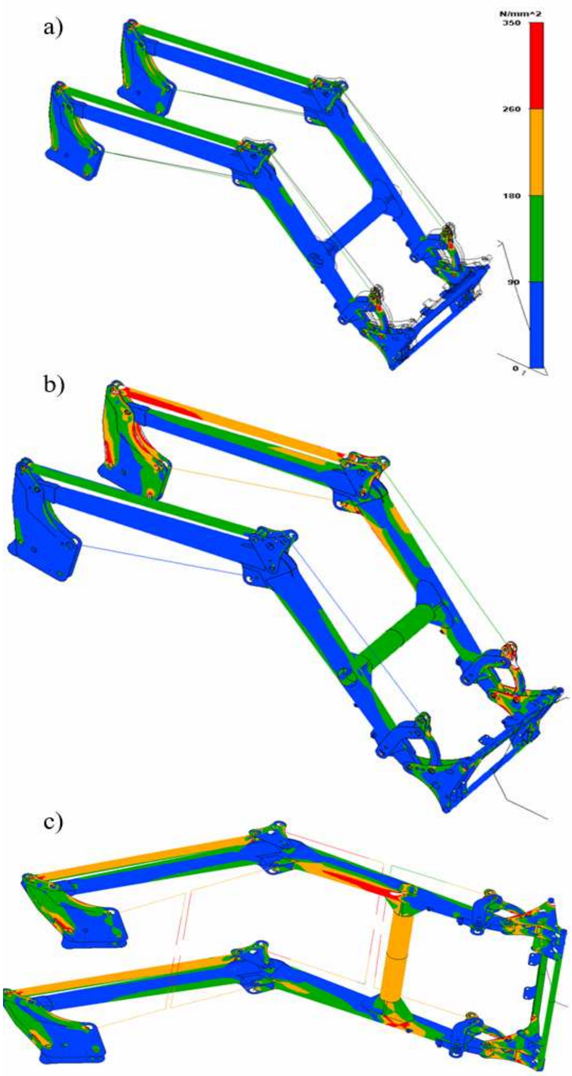
- the cylinders are connected with each other by T-pieces on hydraulic hoses and there are no hydraulic locks,
- as above, but before the T-piece there is a hydraulic lock, which simultaneously cuts off both cylinders,
- each cylinder has a separate hydraulic lock.

The results show the importance of including in FEM computations both load asymmetry and different approaches in modelling hydraulic cylinders (Fig. 12).



Source: own study / Źródło: opracowanie własne

Fig. 11. The loader calculation model with unsymmetrical bucket load
Rys. 11. Model obliczeniowy ładowacza z niesymetrycznym obciążeniem łyżki



Source: own study / Źródło: opracowanie własne

Fig. 12. Differences in results for: a) load symmetry, b) load asymmetry and actuators with hydraulic locks, c) load asymmetry and actuators connected in parallel
Rys. 12. Różnice w wynikach dla sytuacji: a) symetrii obciążenia, b) asymetrii obciążenia i siłownikami z zamkami hydraulicznymi, c) asymetrii obciążenia i siłownikami połączonymi równolegle

Example a) shows how small the effort of the structure is when the model is symmetrical. In the example b), the actuators work in the same way as with the hydraulic locks and there is clearly a difference in the load between the right and the left cylinder and the difference in the effort of the frame between the right and left stringers. When force equalization is applied in the actuators, as in example c), the uniformity of stresses on both sides of the frame is to be seen. There were also new places of high stresses in the connecting pipe and in the vicinity of its fastening. Only such an example is able to justify the necessity of this pipe and its dimensions.

The presented results show the existence of two major sources of potential FEM calculation errors resulting from omission of such machine operating phenomena as load asymmetry as well as the way of the connection of hydraulic cylinders. But this can also take place in other situations, when there are elements working in parallel, and this parallelism is not reflected in the computational model.

Interesting work concerned modelling hydraulic cylinders in FEM computation is presented in [12]. There the cylinders change of positions influenced changes of loads distribution and the differences of simulation cases depended on.

8. Other examples of error sources in the FEM computations of agricultural machines

The described sources of errors in the FEM calculations concerned just calculations of the front loader. However, in many other machines these errors can occur from various other causes. They will be presented on selected examples.

Fig. 13 shows the results of the calculation of the lug mounted on the shaft which is made of two steel pieces welded together.

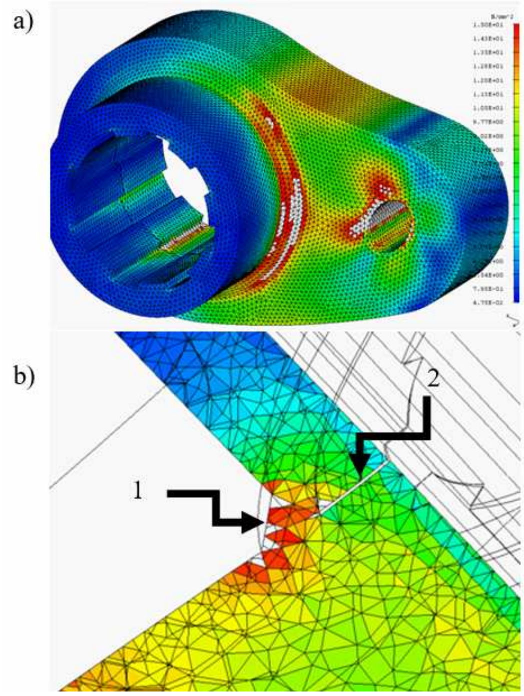


Fig. 13. Welded lug: a) general view of the distribution of reduced stresses in the weld, b) model details showing the implementation of the slit, 1) concentration of stresses in the weld, 2) separation slit [6]

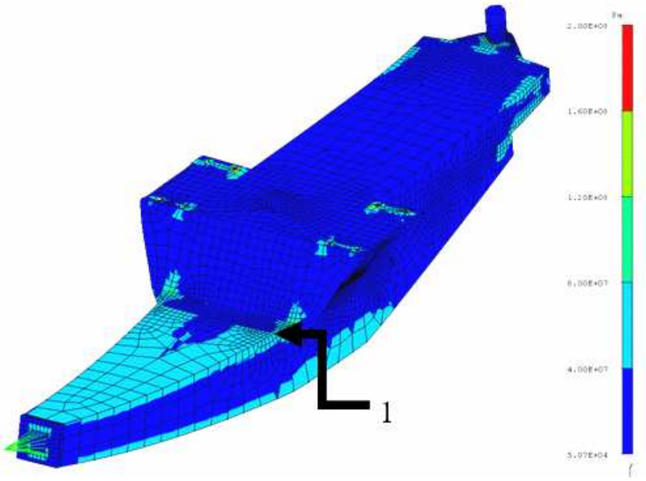
Rys. 13. Ucho spawane: a) widok ogólny rozkładu naprężeń zredukowanych, b) szczegół dotyczący wykonania szczeliny rozdzielającej części, 1) koncentracja naprężeń w spoinie, 2) szczelina separacyjna [6]

The stress concentration in the weld is visible, which was obtained by the modelling of slit between the connected parts. The lug computed without this slit did not showed any strength problems. The failure occurred during the work of the prototype machine only. Such a precisely made model with a slit which mapped the real range of the weld allowed to know the cause of the crack. In this case, the omission in model of material discontinuity was the source of FEM strength analysis error. Also [8] dealt with this type of computational problems.

Fig. 14 shows the drawbar of a slurry tanker. In the place shown by the arrow, low-cycle fatigue cracks appeared after about 2 years of operation of the tanker. The original FEM analysis did not show such a hazard, because the level of stress in this place was low, on the level of 50-80 MPa. It indicated sufficient fatigue strength. However, later analysis of the problem showed, that the load assumed in the calculations was about four times too low from that occurring in reality. The reason for this was not to take into account a certain operational case, which proved to be important for the durability of the structure.

The calculations of the slurry tanker were carried out with the assumption of an equal load distribution between all wheels and drawbar. This is proper when the road is flat and the slurry tanker suspension is able to deal with small unevenness of the road. In the real conditions, however, it turned out that the unevenness of the ground could be much larger when the agricultural tractor leaves the agricultural field going on the road. The differences between the level of field and road could be even 0.5 m. In this situation, the six-wheels slurry tanker is supported only on two rear wheels and on the drawbar. The drawbar reaction is then about 50% of the weight of the entire tanker. So, the lack of prediction of important computational case load was the source of the computational error in this case.

The last example shows the important role of taking into account the fact of uneven ground and the resulting diversification of reactions on the supports of the machine.

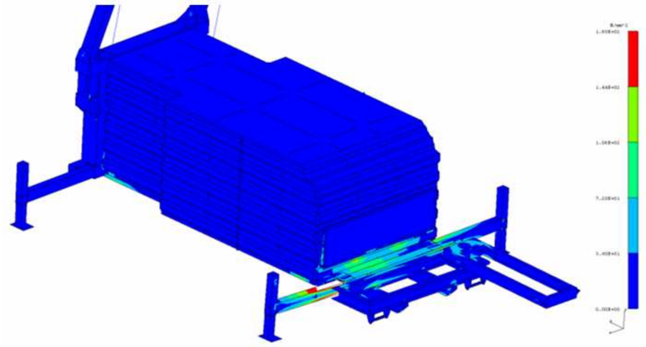


Source: own study / Źródło: opracowanie własne

Fig. 14. The results of the slurry tanker calculation (1 – the breakage point of the drawbar)

Rys. 14. Wyniki obliczeń dyszla wozu asenizacyjnego (1 – miejsce pękania dyszla)

In Fig. 15 there has been shown the results of the water tank analysis with four legs.



Source: own study / Źródło: opracowanie własne
Fig. 15. Differentiation of stresses in tank legs caused by uneven ground
Rys. 15. Zróżnicowanie naprężeń w nogach zbiornika spowodowane nierównym podłożem

In the extreme situation of the uneven ground, the tank is supported practically on two legs lying diagonally only, and the one of the other legs carries a residual load to maintain the balance. This means that the load on a single leg is about 50% of the weight of the whole machine and thus it is twice as much as during the support on ideally even ground. In addition, the shape of the frame is also twisted. In such a case, the size of deflection of the non-supported leg must be checked and compared with the ground unevenness conditions of the machine (such as machine support recommendations or anchoring in the ground). The leg deflection for the tanker was about 70 mm and was real due to putting the tank on the soil. So, the source of calculation errors can be performing computations only for ideal machine support on all legs.

How the supporting is important for obtaining the proper computation results is shown in [3]. There the problem concerned modelling of chassis of trailer lorry. It was especially important because of reactions distribution and proper main frame deflection.

9. Conclusions

The analyses presented in the paper allow to draw the following conclusions:

1. From a practical point of view, the most important impact on the exactness of FEM computations is the skill of the person in such areas: building a computational model, presentation of calculation results, way of modelling operating conditions in calculation model, the way of including specific characteristic of the machine structure and others.
2. The sources of errors in calculations depending on complication of the computational model are: geometry mapping method, contact mapping, mesh density, type of finite elements used, load symmetry and asymmetry, load distribution between inner parts of the machine, like exemplary hydraulic cylinders, omission of important case load and others.
3. It was estimated, based on the collected data from the examples described in the paper, that the FEM specialist can commit the following order magnitude of errors:

- 35% - due to load symmetry/asymmetry (case observed in front loader analysis),
- 40% - due to structural symmetry/asymmetry (in front loader analysis),
- 43% - due to used settings during results reading in post processing (in the lug analysis),
- 47% - due to incorrect mesh density (lug analysis),
- 50% - due to support symmetry/asymmetry (case of tank legs),
- 67% - due to incorrect modelling of material continuity (case of weld in the lug),
- 75% - due to omission of important load case (slurry tanker drawbar analysis),
- 96% - due to including or not the contact modelling (lug hole – pin contact analysis),
- 100% - due to the lack of analysis for significant case (in general).

4. The exactness referring to point 3 above have nothing common with the accuracy described in books devoted to numerical calculations, resulting from the very theory of finite elements method. This are errors that depend on the skills of the FEM specialist and the practical knowledge he has acquired. However, there are also errors that cannot be avoided due to the exactness of the mapping the structure or operating conditions in the calculation model. Large or complicated machines requires far-reaching simplifications, so that their calculation becomes possible at all. In this cases, the general state of effort of the structure is considered only, but the fact of stress disturbances at the places of welds and notches is omitted. Of course, it is possible to increase the exactness of calculations of such nodes by modelling them more accurate fragmentary, but it increases the time and cost of calculations and in many of cases it is not necessary. The approximate state of stress is often enough for designers who have experience in exploiting similar machines.
5. For the front loader, the estimated, practical inaccuracy of the calculation of the reduced stress values for the FEM model developed in this way is ± 25%. The calculated stress values at the concentration points can be underestimated for solid finite elements, and overstated for plate-shell elements.
6. Load asymmetry must always be included in the calculation. In doing so, consideration should be given to the appropriate criteria for the admissibility of stress heights, paying attention to which calculation case is to be regarded as ad hoc and which as fatigue.

10. References

[1] Simulation: Finite Element Modelling User's Guide. Help files for I-DEAS NX system. Siemens Product Lifecycle Management Software Inc. 2016.

[2] Without authorship. Does FEM mesh quality really matter?. Internet feuilleton. VizionZ Engineering BV. Available on-line: https://www.vizionz. nl/articles/12-fem. Access: 10.06.2018.

[3] Marcinkiewicz J., Bieńczak A., Dembicki D., Dudziński P., Mac J., Szczepaniak J.: Strength analysis of insulated body with the use of FEM. Journal of Researches and Applications in Agricultural Engineering, 2015, 60(1), 44-49.

[4] Jovanović M., Milić P., Janošević D., Petrović G.. Accuracy of the fem analyses in the function of the finite element type selection. Facta Universitatis, 2010, Vol. 8, 1, 1-8. Available: http://facta.junis.ni.ac.rs/me/ me201001/me201001-01.pdf.

[5] Pawłowski T.: Ładowacz czołowy Ł-110 do ciągników o mocy powyżej 140 kW – opracowanie innowacyjnej konstrukcji i obliczenia (not published inner research report). Przemysłowy Instytut Maszyn Rolniczych, Poznań, 2017.

[6] Pawłowski T., Szczepaniak J.: Techniki i technologie informacyjne w projektowaniu i badaniach maszyn rolniczych w PIMR. Jubilee conference: Inżynieria rolnicza w dobie innowacyjnej gospodarki. Wydział Inżynierii Produkcji i Energetyki. Kraków. 26.09.2012.

[7] Pointer J.: Understanding Accuracy and Discretization Error in an FEA Model. Woodward Governor Company. Available on-line: https://www.ansys.com/-/media/ansys/ corporate/resourcelibrary/conference-paper/2004-int-ansys-conf-54.pdf 2004.

[8] Ramamoorthy N.: How Accurate is Your Finite Element Analysis – Part I. Dessault Systemes. Available on-line: http://blogs.solidworks.com/tech/2015/07/accurate-finite-element-analysis-part.html. 2015.

[9] Rutkowski J., Szczepaniak J.: Modelowanie węzłów kinematycznych w wytrzymałościowej analizie dynamicznej konstrukcji wybranej maszyny rolniczej. Journal of Researches and Applications in Agricultural Engineering, 2003, 4, 60-64.

[10] Spadło M., Dembicki D., Woźniak P., Szczepaniak J.: Strength analysis by method of the finite elements in construction of spray sterilizer with restrictions of standard EN 13445-3. Journal of Researches and Applications in Agricultural Engineering, 2016, 61(2), 97-103.

[11] Visser G.: How accurate is FEA? Esteq. Available on-line: https://esteq.co.za/2014/10/21/accurate-fea/. 2014 r.

[12] Zembrowski K., Sobocki S., Rakowicz A., Siczyński Ł.:. Strength analysis of the supporting structure of the ABT vehicle during container reloading. Journal of Researches and Applications in Agricultural Engineering, 2016, 61(2), 134-139.

[13] Zienkiewicz O.: Metoda elementów skończonych. Arkady, Warszawa 1972.

***Acknowledgement**:*

External order date 01.03.2017 r.; the ordering party: Zakład Metalowy AGROMASZ Gruszczewski Janusz, Mrągowo, project no. 30/2017/LB/ZW/N.