*Symulacja w Badaniach i Rozwoju*
Vol. 8, No. 3-4/2017

**Adam MYSŁEK**
Wydział Mechaniczny Politechniki Krakowskiej
E-mail: adam.myslek@poczta.onet.pl

**Henryk SANECKI**
Zakład Niezawodności i Eksploatacji Technicznej, Instytut Pojazdów
Szynowych, Politechnika Krakowska
E-mail: hsa@mech.pk.edu.pl

# Optimization of hydraulic crane prototype

## 1 Introduction

One of the advantages of Computer Aided Design (CAD) and Computer Aided Engineering (CAE) is possibility of anticipating how the structure can behave in current stated condition and circumstances (loads, material type etc.). At the beginning of the design process the designer can verify used solutions with certain accuracy without building often more expensive prototypes. Nowadays, designers have begun to use 3D modeling and numerical simulations alone as acceptable means of validation. So called "virtual prototyping" employing numerical simulation tools based on finite element method (FEM) replaced traditional physical model tests, [1], [2].

This work describes an example of using computer aided design and simulation with use of finite element method. The aim was to determine stress and strain state of hydraulic crane at certain working conditions. Such machines are widely used on heavy duty vehicles for manipulating of very heavy cargo. 3D model and linear FEM simulation combined with optimization has been performed in Creo Parametric system.

## 2 Numerical tools applied to solve the problem

Application of the finite element method starts with dividing a geometrical model of the analyzed structure into very small and simple pieces (the finite elements) connected at their nodes. Behaviour of the elements is determined by a corresponding physical law. So called "shape functions" are used to approximate displacements or other unknowns over the element. The role of the shape functions fulfil specially defined polynomials which are interpolated over the element from the nodal values. The calculation leads to a set of simultaneous algebraic equations with the nodal values being unknown. During the solution process the nodal values are being found. The number of unknowns depends on DOF (degrees of freedom) of the model. Then all interesting quantities (strains, stresses) are calculated within the elements.

There are several types of element definitions. In classic h element type (h refers to an element size) the convergence is obtained by use of the mesh refinement and is related to a step size used in the solution. To get more accurate estimation of the stress state using the first order elements it is necessary to use rather small elements. This leads often to modification of the entire mesh. This situation is described in figure 2(a) and (b), [4].

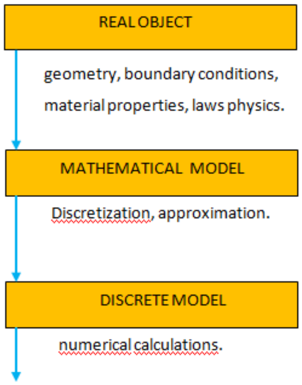
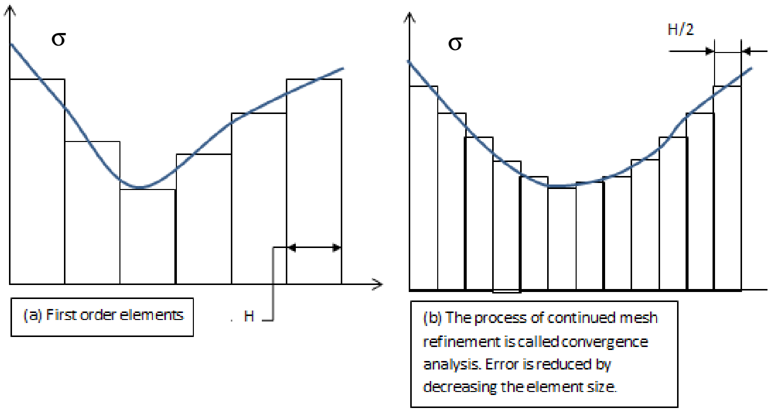


*Fig. 1. FEM algorithm*

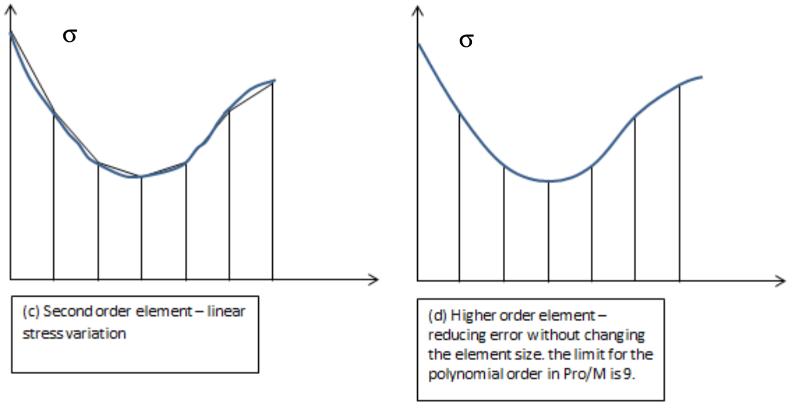
Another way is to use polynomial finite elements - the Creo approach. So called p -elements have shape functions with a variable polynomial order. The convergence is achieved by increasing the polynomial order for the given mesh (on each element) instead of constantly refining and recreating finer and finer meshes (like in classic h finite element types). The mesh stays the same for every iteration called a p-loop pass. This situation is depicted in fig. 3 (c), (d). The system can monitor the errors and automatically increase the polynomial order only on those elements where it is necessary.



(a) First order elements

(b) The process of continued mesh refinement is called convergence analysis. Error is reduced by decreasing the element size.

*Fig. 2. Approximation of stress function in a model with h-elements*



(c) Second order element – linear stress variation

(d) Higher order element – reducing error without changing the element size. the limit for the polynomial order in Pro/M is 9.

*Fig. 3. Approximation of stress function in a model with p-elements*

Process of creation of equations in the FEM is mathematically complex. The frequently used case is four node tetrahedron element model. It is used here but is not presented in details. Characteristic features of the process are described in [5]. Some of them are as follow: the displacement field is defined by the three components $u_x, u_y, u_z$; the shape functions use the tetrahedral coordinates $N_i = \zeta_i$, $i = 1, 2, 3, 4$ where $\zeta$ is dimensionless tetrahedral coordinate.

After performing the calculations by the solver of the FEM system data concerning the strain and stress fields are received. They are visualized in linear range by the postprocessor as Von Mises stress distributions (according to Huber-Mises-Hencky hypothesis [3]):

$$\sigma_{red} = \frac{\sqrt{2}}{2}\sqrt{(\sigma_x - \sigma_y)^2 + (\sigma_y - \sigma_z)^2 + (\sigma_z - \sigma_x)^2 + 6(\tau_{xy}^2 + \tau_{yz}^2 + \tau_{zx}^2)} \qquad (1)$$

## 3 Hydraulic crane design

The first step of the project was to create a 3D model of the hydraulic crane prototype. An initial version of the crane structure had weight of 1026 kg, wall thickness 10 mm in each segments and was specified in a position which carries the highest stress during work (fig. 4).

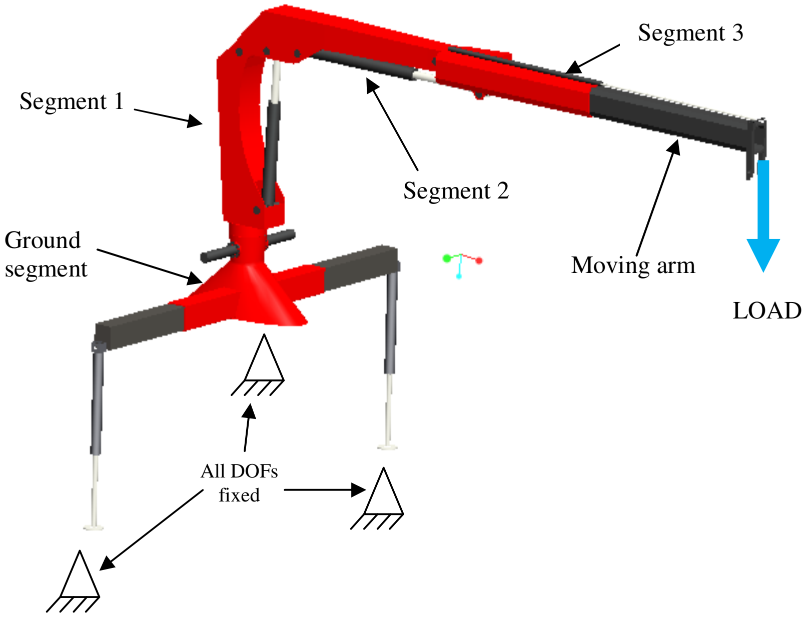


*Fig. 4. Main parts of hydraulic crane prototype*

An important step of the project was to create a kind of idealization of joints (bolts) between the segments in order to achieve more accurate results in a reasonable time of the calculations. In order to obtain this effect so called "rigid links" were used. The rigid link is a type of connection that joins geometric entities such as surfaces, curves and points so that they remain rigidly connected during the numerical simulation. The rigidly linked components are free to move in any manner but will maintain their relative positions. Also touching surfaces between the moving arm and the last segment of the crane (fig. 4) had been connected with the method described above using the defined surfaces and edges. Such assumptions were necessary because those two elements practically could not fit precisely one in another. Examples of other places with such type of connection are shown in fig. 5.

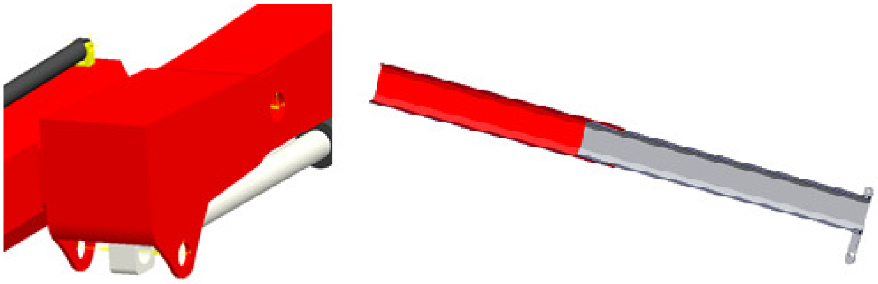


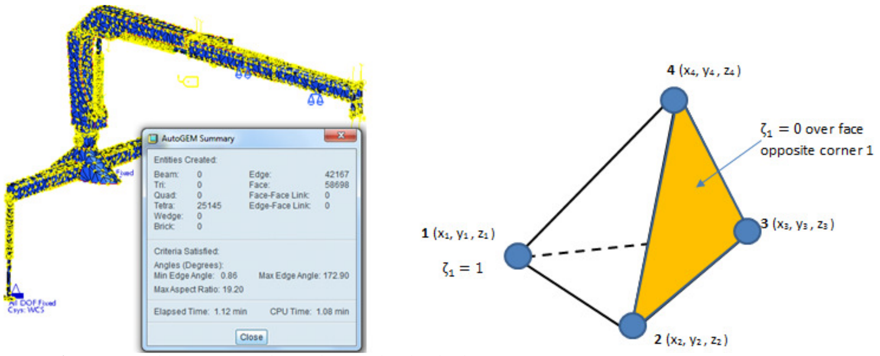
*Fig. 5. Examples of rigid links in crane structure*

In order to properly model the working operation of the crane appropriate boundary conditions were defined. All degrees of freedom on the bottom surface of the ground segment as well as the lower parts (footings) of the legs were fixed. Table 1 contains the main data necessary to perform the analysis.

*Tab. 1. Input data*

| Load [N] | Material | Young Modulus MPa | Poisson Ratio |
|---|---|---|---|
| 20000 | Isotropic | 199948 | 0,27 |

After the preparations described above the model of the crane is ready for a static analysis in a linear range. This is a standard design study, which is necessary at a preliminary stage of the project. The FEM model with a finite element mesh (p-elements) which was generated with the preprocessor was presented in fig. 6. The generated element type was the four node tetrahedron. The positions of the boundary conditions were shown here, too.



```
Entities Created:
Beam:    0          Edge:             42167
Tri:     0          Face:             58698
Quad:    0          Face-Face Link:   0
Tetra:   25145      Edge-Face Link:   0
Wedge:   0
Brick:   0

Criteria Satisfied:
Angles (Degrees):
Min Edge Angle:   0.86      Max Edge Angle: 172.90
Max Aspect Ratio: 19.20

Elapsed Time:  1.12 min     CPU Time:  1.08 min
```

*Fig. 6. Finite element mesh and tetrahedral element*

The equivalent von Mises stress distribution and the displacement distribution were shown in fig. 7. As can be seen the highest stress value is 510 MPa and the highest displacement is 80 mm.

Knowing these results it is now possible to assign to the members of the structure a proper type of material. It seems that in this case S460Q steel may be applied. Due to the safety factor of the design the Von Mises stress for this material may not reach 355 MPa. To fulfil the condition that the stresses are less than the permissible value an optimization processes is needed (described in the next chapter).

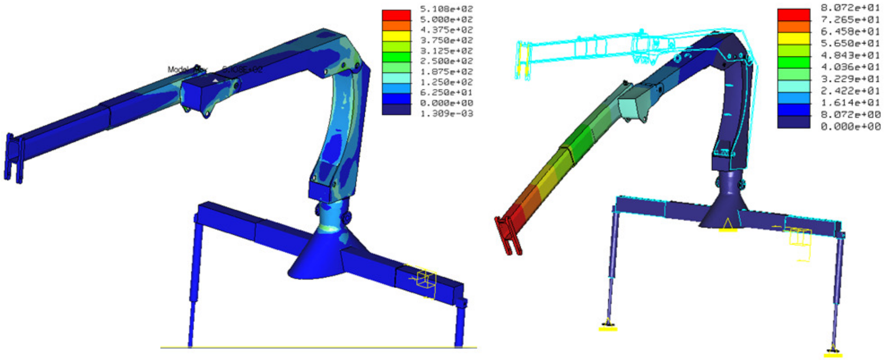


*Fig. 7. Equivalent von Mises stresses and displacements*

## 4 Optimization of crane design

First, some optimization basics have to be established. A number of selected geometric dimensions of the structure was designated as the design variables. Next step was to find a function of these design variables that can minimize an objective function (like the total mass of the model) and to subject these variables to some design constraints (like the allowed maximum stress and/or deflection). The optimum set is picked from the specified ranges of the design variables and thus it is the solution of the problem.

In the optimization study concerning the crane prototype the criterion function is the minimal mass in each segment according to the specified parameters. The solution have to fulfill the stress limitation. The limitation is $\sigma_{max} \leq 355$ MPa. The wall thicknesses of the members of each segment of the structure were the optimization variables. Their values should be in a specified range determined by the designer.

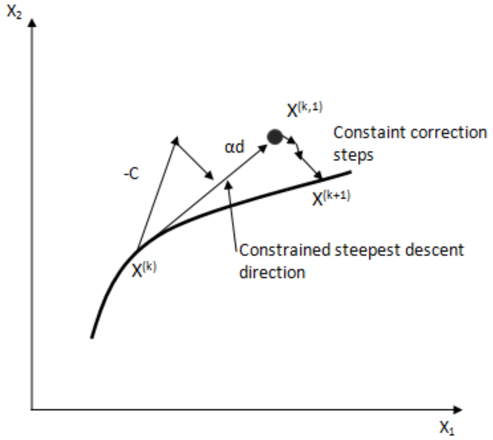


*Fig. 8. Steps in gradient projection method; C - the steepest-descent direction, d - negative projected gradient, point X(k) is taken to X(k,1) and the feasible point X(k+1) is reached by executing the constraint correction steps.*

Generally, there are several methods, which could be applied to solve the optimization problems, [6]. Two such algorithms: GDP (Gradient Projection Programming) and SQP (Sequential Quadratic Programming) are available in the system [4], which is used here. They could be chosen automatically during the process. A short descriptions of the SQP and GDP optimization algorithms are presented below.

Gradient projection method (GDP) was developed in order to calculate the direction vector - to derive an explicit expression for the search direction. In this algorithm, if the initial point is inside the feasible set, the direction of the steepest descent for the objective function is used until a constraint boundary is met. If the starting point is infeasible, then the constraint correction step is used to reach the feasible set. When the point is located on boundary the direction, which is tangent to the constraint surface, is calculated and is used to change the design. The direction is calculated by projecting the steepest-descent direction for the objective function onto the tangent hyperplane. The iterative process of this method is shown in fig. 8.

Sequential quadratic programming (SQP, [7]) basically accomplishes the iterative concepts:

- vector form: $x^{k+1} = x^k + \Delta x^k$; $k$ = 0, 1, 2,...

- component form: $x_I^{(k+1)} = x_i^k + \Delta x_i^k$; $k$ = 0, 1, 2,…; $i$ = 1…$n$; $k$ represents the iteration or design cycle number, $i$ refers to the $i$-th design variable, the change in design $\Delta x_i^k$ is decomposed as $\Delta x_i^k = \alpha d^k$, where α is step size in the direction $d$

- following must be realized: 1) a search direction in the design space has to be calculated by utilizing the values and the gradients of the problem functions, a quadratic programming sub-problem should be defined and solved, 2) a step size along the search direction must be calculated to minimize a descent function; a step size calculation sub-problem has to be defined and solved. In the SQP method the linearized objective function is modified by adding a second order term.

For more information concerning the algorithms please refer to [7].

The process of optimization was performed on the model of the hydraulic crane prototype. The obtained results of optimization were filled in table 2 (and compared with the results obtained after standard design in chapter 3) and shown in fig. 9. As can be noticed the maximum stress value is no longer higher than the defined one. Also the maximum displacement has been decreased. The total weight of the crane model has increased only in 26 kg.

*Tab. 2. Results of optimized design one*

| | Max. stress von Mises, MPa | Max. displacement mm | Total weight, kg |
|---|---|---|---|
| Standard analysis | 510 | 80 | 1026 |
| Optimization | 352 | 68 | 1052 |

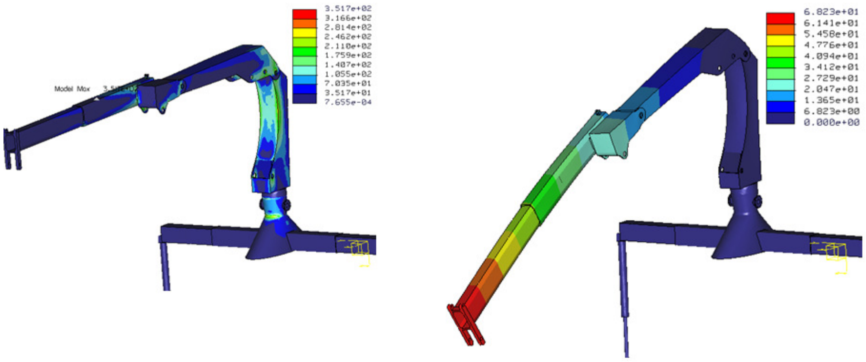


*Fig. 9. Equivalent von Mises stress and displacement distributions after optimization*

Fig. 10 presents an example of the design study performed on the segment 2. The wall thickness range is specified from 9 mm to 20 mm, stress $\sigma_{max}$ should not be greater than 355 MPa. The system is choosing an appropriate value from the given range and assigning it to the model. The calculated value for the wall thickness of the segment 2 which fulfill the stress limitation is 11.550 mm. For the ground segment the variable (thickness) is specified in range 5-20 mm and the calculated value is 8.351 mm (wall thickness was reduced in comparison with the initial value 10 mm).

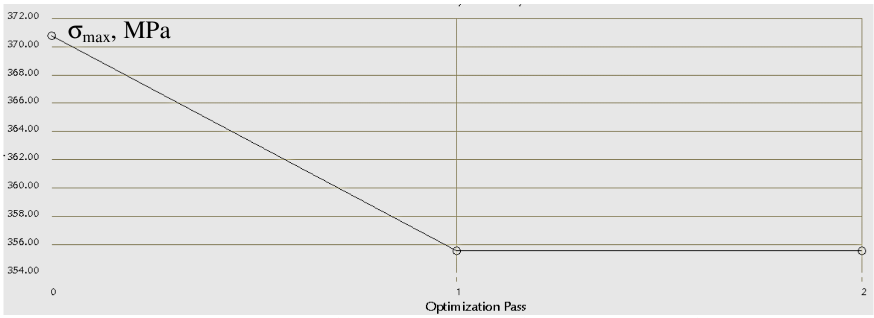


*Fig. 10. Design optimization study for segment 2; changes in stress during 2 passes*

## 5 Conclusions

The paper presented briefly the method of designing the mechanical structure using combination of CAD and CAE tools. Namely, it was the optimization study of the design by using the finite element method. The presented results showed the possibility of improvement of the solution according to the specified criteria instead of the multiple analysis made by “manually” checking the variables.

Final results showed that the maximum stress value did not exceed the material's yield and the permissible stress. Another requirements were also fulfilled entirely. The finite element method aided by the other numerical tools has great significance for the production processes because it allows to simulate easily the working conditions. Due to this the results are more realistic and diminish the cost of research and development.

## References

1. Kindmann R., Kraus M.: *Steel structures: design using FEM*, Berlin 2011
2. Rusiński E.: *Computer-aided analysis of frames and bodies of vehicles and heavy machines* (in Polish). WKŁ Warszawa 1990
3. Bąk R., Burczyński T.: *Strength of materials with computer-aided elements* (in Polish). WNT, Warszawa 2013
4. *Pro/Mechanica Structure*. Release 2001, Student Edition Tutorial
5. *The linear tetrahedron*, University of Colorado
6. Reklaitis G.V., Ravindran A., Ragsdell K.M.: *Engineering optimization. Methods and Applications.* J. WILEY&SONS, New York
7. Jasbir S. Arora: *Introduction to optimum design*. Third edition 2012

## Summary

The paper presents an application of research methodology leading to structural optimization of a crane model containing hydraulic cylinders. CAD and CAE systems were used to obtain a three-dimensional model of the crane and state of stresses and displacements, as well as to carry out appropriate optimization procedures. This issue is important from the point of view of full strength verification of modern transport equipment, particularly at the design stage, while detailed experimental research is not yet conducted.

**Keywords**: hydraulic crane, CAD, FEM, structural optimization, GDP and SQP algorithms

# Optymalizacja prototypu żurawia hydraulicznego

## Streszczenie

Praca prezentuje metodykę badań prowadzącą do optymalizacji strukturalnej żurawia zawierającego siłowniki hydrauliczne. Zastosowano systemy CAD i CAE do uzyskania trójwymiarowego modelu żurawia oraz stanu naprężeń i przemieszczeń, a także do przeprowadzenia odpowiednich operacji optymalizacyjnych. Zagadnienie jest istotne z punktu widzenia pełnej weryfikacji wytrzymałościowej nowoczesnych urządzeń transportu bliskiego zwłaszcza na etapie projektowania, gdy jeszcze nie prowadzi się szczegółowych badań doświadczalnych.

**Słowa kluczowe**: żuraw hydrauliczny, CAD, MES, optymalizacja strukturalna, algorytmy GDP i SQP