Evaluation of structure stability with the use of remote load

Szymon Molski¹, Paweł Lonkwic², Hubert Ruta¹ and Tomasz Krakowski¹

¹ AGH University of Science and Technology, The Faculty of Mechanical Engineering and Robotics, Department of Machine and Transport Engineering, A. Mickiewicz Av. 30, 30-059 Cracow, Poland
² The State School of Higher Education in Chełm The Institute of Technical Sciences and Aviation, 54, Pocztowa Street, 22-100 Chełm, Poland

molski@agh.edu.pl, plonkwic@pwsz.chelm.pl, hubert.ruta@agh.edu.pl, krakowsk@agh.edu.pl

Abstract. It is a quite common practice to place the load at some distance from the machine restraint. In such a case, it is possible to evaluate the values of stresses and displacements by means of numerical analyses. This article presents the numerical analysis results for the carrying structure of the friction lift power unit with the use of the remote load. The impact of the remote load on the strain of the lift power unit carrying structure is also described. Attention is paid to the places of excessive strain of the structure.

1. Introduction

FEM numerical simulations are used to evaluate the structure in terms of its strain, stiffness, safety, correct geometry and prototyping costs reduction. In the publication [1] the authors described the use of numerical methods to evaluate buckling of a thin-walled profile in the shape of a C-beam subject to axial pressing. In the publication [2] the authors used numerical simulations to evaluate the strain of the safety gear structure in the passenger lift. The results of numerical simulations are compared with the results of physical experiment that has given concurring results. In the article [7] authors suggested a 1D/3D finite element model for static analysis of solid frames with thin-walled rods and open cross-section. This model involves standard discretisation using beam thin-walled elements connected with space discretisation of the frame joints. In the publication [3] the authors described the results obtained in the analysis of the safety gear structure by means of the Finite Element Method and Abaqus software. The authors in the paper [8] present a finite element model for analysis of thin-walled structures. Standard discretisation using beam thin-walled elements is connected with space discretisation of some parts of the frame. Transition elements are used for consistent coupling between shell (3D) and beam (1D) elements. The paper [9] presents calculation models used in the analysis of various building structures and soil-structure interaction. Basic schemes, principles and possibilities, which consider the change in time geometric parameters, material and boundary conditions are described. In the paper [13] authors presented the method of strength and dynamic analysis of agricultural rollers. Selected forms of showing the simulation results, that is, stresses, forces displacements etc., are also presented. The practical method of carrying out computer simulations is shown, as well. The descriptions of these methods are extensive enough to show aspects of costs and time reductions, when carrying out these types of analyses. In the publication [4] the authors discussed the method of modelling a composite profile, which was subject to axial pressing. Numerical simulation results are compared with the results...
received from tensile testing. Authors [10] present a method of strength calculations of truck frames by using integration of MSC Adams software – for dynamics analysis of mechanical systems and NX Nastran/Femap software – for the Finite Element Method analysis. In this study, a method of reduction of freedom degrees has been developed based on the Craig-Bampton approach. The interface has been applied to calculate the strength of the frame of the selected truck running on the test track. The model of the truck can be treated as its virtual prototype useful in the design process. In the publication [5] the authors described the results of numerical simulation conducted to assess some displacements of the machine tool body used for the extruded joints and operated in equal geometric configurations.

This paper [11] presents the study of the vibration characteristics of the truck chassis, that include natural frequencies and mode shapes. The responses of the truck chassis, which include stress distribution and displacement under various load conditions are also observed. In the publication [6] the authors used the Finite Element Method available in Catia program to evaluate strain generated by a carrying frame in a heavy vehicle. Stiffness of the frame elements under load was also evaluated. Applicability of the Finite Element Method is much more extensive in technology, not only for purposes connected with mechanical engineering but also in different fields. Authors in the publication [14] present a problem of the basic concept of using the finite element method in constructing an analytical model for the study of the behaviour of reinforced concrete members. Finite elements chosen to represent concrete, steel reinforcement, and bond links between concrete and steel reinforcement are described. The article [12] presents the calculation model of a structure to drain and disinfect waste containers, which was provided for the fatigue strength analysis. A process of creating the model and discussed techniques, which were used to reflect the kinematics bindings of researched device, are described in detail. In this work [15] the authors focused on solutions for the Euler-Bernoulli and Timoshenko beam theories, in which material behaviour may be elastic or inelastic. Formulation relies on integration of the local constitutive equation over the beam cross section, to develop relations for beam resultants. For this case, axial, bending and shear effects were included. This allows consideration in a direct manner of elastic and inelastic behaviour, with or without shear deformation. The author in the paper [16] reviews basic concepts involved when constructing nonlinear load- deflexion curves of framed structures by the finite element method. The study is limited to the geometrically nonlinear behaviour of elastic structures. Three different procedures are considered: linear incremental method, nonlinear incremental method, and direct method. For each method, governing equations are derived, and the procedure used to plot the load- deflexion curve is outlined. The paper [17] presents a finite element analysis of large deformations in elastoplastic frames. Emphasis is put on the step-by-step analysis of the nonlinear behaviour of such structures. Advantage is taken of the so-called natural formulation in terms of the generalised cross-sectional forces. Limit state functions applied to different loading conditions and frame geometries are implemented into a computer program. This paper [19] presents exploratory stiffness-strength analysis of frame of ARIA sporting car. A conception of innovative car frame and the design assumptions is presented. Moreover, MES models, project modification stages and analysis results are described. The entire paper is summed up by describing reached results and presenting conclusions related with forward progress of construction. The authors presented [18] design of infilled frames to resist lateral loads on buildings in terms of their failure modes, failure loads, and initial stiffness using procedures proposed by previous authors. This verification is made by comparing the results of analytical procedures of previous authors with those of a new finite element model for infilled frames, which are verified using experimental results. To model the interface between the frame and the infill and the mortar joints surrounding the blocks of masonry, a non-associated interface models are formulated using the available test data on masonry joints. This study [20] presents a simple formulation to treat large deflexions by the finite element method. The proposed formulation does not use the displacement concept. It considers position as the main unknown variable of the problem. Strain determination is performed directly from the proposed position concept. Non-dimensional space is created, and relative curvature and fibres length are calculated for both reference and deformed configurations and used to directly compute the strain energy at general points. This paper [21] presents a finite element formulation for the numerical analysis of three-dimensional framed steel,
reinforced concrete or composite steel and concrete structures subjected to fire. Within this context, the purpose of this work is to present the steps taken to extend a previously developed static analysis procedure with beam elements, in order to cope with the thermal and structural analysis of structures under fire action.

All publications mentioned above present the use of the FEM method, assuming that the load is axially applied in relation to the structure. This publication demonstrates the FEM method used to evaluate the strain of the power unit carrying structure in the friction lift. The load is located at some distance from the drive attachment that directly applies load to the carrying structure.

2. Numerical model of the power unit carrying frame in the friction lift

The power unit of the friction lift consists of a reducer, where a friction wheel and an electric motor are installed. Depending on the lift structure, the carrying structures can be distinguished considering the engine room location, see figure 1.

![Figure 1. Types of carrying structures of the friction lift power units due to the engine room location.](image)

The friction lift carrying structure that is installed in the bottom side engine room belongs to those which are more complex. Figure 2 presents a three-dimensional model of such a frame with its main subassemblies and main dimensions.

The carrying frame is made of hot rolled C180 type profiles, and the structure is welded. It is mounted to the structural surfaces of the engine room with the interleaved anchors of Hilti type in the places presented in figure 2. These are sheet metal elements with holes on the external side of the structure. A relation between forces acting on the unit and inertial forces is described by means of the below relation [5] in the form of a matrix:

\[
f(t) = |M|\{\ddot{u}\} + |C|\{\dot{u}\} + |K|\{u\}
\]

where: \{\ddot{u}\}, \{\dot{u}\}, \{u\} – represent vectors of acceleration, speed and displacement respectively, \(|M|, |C|, |K|\) – represent the matrices: mass, rigidity and damping, respectively.
Figure 2. A three-dimensional model of the carrying frame in the friction lift power unit with the bottom side engine room (the authors’ private source): 1 – a reducer (without an electric motor), 2 – a carrying frame, 3 – a friction wheel, 4 – a rope pulley axis, 5 – a bearing block, 6 – a directional rope pulley.

All frame components have been designed with the use of steel with the following mechanical properties [22]:
- Young’s modulus $E = 210 \text{ GPa}$,
- Poisson’s ratio $\nu = 0.3$,
- material density $\rho = 7860 \text{ kg/m}^3$,
- yield point $R_e = 230 \text{ MPa}$

The load system of the modelled unit is presented in figure 3. Loads resulting from the lift operation are generated by the carrying ropes which belt the wheels as presented in the below scheme:
Figure 3. A scheme of the reducer rigging and the carrying ropes load.

The system discretisation process was a primary stage in the FEM analysis, for which the SolidWorks Simulation was used. Contact parameters were based on the mutual interaction of subassemblies in the tangent and regular direction. A total value of 16000 N was taken as the ropes load. The applicable value was a sum of masses: cabin with frame, load capacity and counterweight. A direction of forces generated in the ropes was restrained to the ropes ends (grey rope) and presented in figure 3.

Numerical analysis was conducted for two cases:
Case I – impact of remote load on places where the reducer is mounted;
Case II – impact of remote load on places where the slide bearings are mounted.

In the analysed case, the load impact on the frame is executed as a direct transfer to the place of the reducer and directional rope pulley mounting. Thus, the numerical model of the reducer, the friction wheel and the axis has been omitted in the model discretisation.

As per the 1s\textsuperscript{t} case, the load transfer to the place where the reducer is mounted on the frame is presented in figure 4.

Figure 4. The numerical model boundary conditions of the carrying frame.
A discrete model of the analysed system is presented in figure 5. A high-quality solid mesh with Jacobian determinants located in nodes was defined for discretisation. The size of the assumed mesh was 15 mm with tolerance of successive iterations of 0.75 mm. Such parameters allowed to create a mesh with 198802 nodes and 100873 elements.

Figure 5. The frame discrete model prepared for the 1st case.

The results of stress numerical simulation for the 1st case are presented in figure 6.

Figure 6. Maximum stresses as per Huber-Mises hypothesis.

Basing on the analysis of the above mentioned figure, it is noticed that the places of the reducer installation are strained excessively. This requires additional analysis of the frame, in terms of its stiffening. The maximum stresses generated under the applied load are 760 MPa, which significantly exceed the yield point of 230 MPa for the assumed material.

Maximum displacements present in the place of the highest strain of the frame are presented in figure 7. The results of calculations regarding a safety factor are presented in figure 8. A red field represents the area with the highest values of stresses; thus, the safety factor is below 1.2. As per the 2nd case, the load transfer to the place where the slide bearings of the directional rope pulleys are installed is presented in figure 9. A discrete model of the analysed system is presented in figure 10. Figure 11 presents the results of numerical simulation for the direct load transfer to the place, where the slide bearings are installed generating load of frame vertical beams.
Figure 7. Maximum displacements.

Figure 8. The safety factor of the frame structure.

Figure 9. The boundary conditions of the carrying frame numerical model for the 2nd case.

Figure 10. A discrete model of the frame for the 2nd case.
Figure 1. The maximum stresses as per Huber-Mises hypothesis for the 2nd case.

Having analysed the above figure, it can be stated that the places of the reducer installation are strained excessively. Thus, additional analysis of the frame in terms of its stiffening is required. The maximum stresses generated under the applied load are 107 MPa.

Maximum displacements expected in the place of the highest strain of vertical beams are presented in figure 12.

The value of safety factor assumed for this type of structures is 1.2. The calculation results regarding the safety factor are presented in figure 13. In the 2nd case, there are no places where the strain of the frame elements would give the value of safety factor below 1.2, see figure 13.
3. Conclusions
The numerical analysis of load cases for the friction lift frame structure allows formulation of the following final conclusions:

- The frame structure subject to analysis is used in friction lifts with the bottom side engine room. Due to real way of frame structure loading, the, so called, remote load was assumed for simulation purposes.
- The value of load used in simulation was assumed on the basis of calculation for the friction lift of nominal carrying capacity of 630 kg.
- The assumed frame geometry and the method of its loading were provided in accordance with the design of the friction lift, which is planned to be implemented in the next calendar year.
- Numerical analysis of 1st load case showed that the value of stresses reduced according to the Huber’s hypothesis was exceeded in the place of the reducer body restraint. Therefore, appropriate frame modifications involving reduction of stresses should be provided in those places.
- Numerical analysis of 2nd load case showed that maximum values of reduced stresses are 107 MPa and are lower than the yield point. Therefore, the resultant safety factor is 1.48 and is higher than the recommended factor of 1.2.
- Presented numerical analysis results were discussed with the manufacturer of such structures, who indicated that analysis of mesh sensitivity for greater accuracy of results is not necessary at this stage. Such an approach ensues from the necessity to modify the structure of the analysed frame.
- Having analysed the presented case, it can be concluded that the obtained numerical results represent realistic values. However, simulation results should be treated as rough due to the geometric simplification of the entire structure provided during the simulation.
- As a result of the numerical analysis of the structure, it is justified to formulate a conclusion that the discussed frame structure should be modified in terms of reducing the value of stresses, and then simulation calculations should be re-taken for the purposes of verification.

4. References
[1] Różyło, P., Teter, A., Debski, H., et al. (2017) Experimental and Numerical Study of the Buckling of Composite Profiles with Open Cross Section under Axial Compression Appl Compos Mater 24 pp 1 – 14 https://doi.org/10.1007/s10443-017-9583-y.
[2] Lonkwic, P., Różyło, P., Dębski, H. (2015) Numerical and experimental analysis of the progressive gear body with the use of finite-element method Eksplota icja i Niezawodność – Maintenance and Reliability 17 (4) pp 544–550 http://dx.doi.org/10.17531/ein.2015.4.9.
[3] Lonkwic, P., Różyło, P. (2016) Theoretical and experimental analysys of loading impact from the progressive safety gears on the free fall method Advances in Science and Technology Research Journal 10(30) pp 103-109 https://doi.org/10.12913/22998624/62628.
[4] Wysmulska, P., Dębski, H., Różyło, P., Falkowicz, K. (2016) A study of stability and post-critical behaviour of thin-walled composite profiles under compression Eksplota icja i Niezawodność – Maintenance and Reliability 18 (4) pp 632–637 http://dx.doi.org/10.17531/ein.2016.4.19.
[5] Markowski, T., Mucha, J., Witkowski, W. (2013) FEM analysis of clinching joint machine’s C-frame rigidity Eksplota icja i Niezawodność – Maintenance and Reliability 15 (1) pp 21–57.
[6] Supardjo, A., Tri, W. (2018) Finite element analysis of truck frame by using CATIA V5. AIP Conference Proceedings 030029.
[7] Cichoń, Cz., Koczubiej, S. (2008) Consisten FEM model for thin-walled space frames Czasopismo Techniczne 1-B, pp 1 – 18.
[8] Koczubiej, S., Cichoń, Cz. (2012) Shell –Beam model in stability analysis of thin-walled structures with open cross section Modelowanie Inżynierskie 44, pp 131-138.
[9] Żakowicz, A., Miedziałowski, Cz. (2011) Building structures models taking into account time-dependable geometric and material parameters and statistical schemes Civil and Environmental Engineering 2 pp 197-204.
[10] Przemyk, A., Harlecki, A., Tengler, Sz. (2017) Method of strength calculations of truck frames Autobusy 1 pp 1252 – 1257.

[11] Fui, T. H., Rahman, R. A. (2007) Statics and dynamics structural analysis of 4.5 ton truck chassis, Journal Mekanikal vol. 24 pp 56-67.

[12] Spadoł, M., Ignasiak, Ł., Szczepaniak, J. (2013) Waste containers for the strength analysis Technika Rolnicza, Ogradnicza i Leśna 3 pp 1-7.

[13] Rutkowski, J., Szczepaniak, J. (2005) Simulations and analysis of virtual computer models in application to series of type of agricultural rollers Journal of Research and Applications in Agricultural Engineering Vol. 50 (1) pp 1-9.

[14] Ngo, D., Scordelis, A. C. (2012) Finite Element Analysis of Reinforced Concrete Beams Journal Proceedings 64(3), pp 152-163.

[15] Taylor, R., Filipou, F., Sarotas, A., Auricchio, F. (2003) A mixed finite element method for beam and frame problems Computational Mechanics 31 (1-2) pp 192-203.

[16] Chajes, A., Churchill, J. (2012) Nonlinear Frame Analysis by Finite Element Methods Journal of Structural Engineering 113 (6) pp https://doi.org/10.1061/(ASCE)0733-9445(1987)113:6(1221).

[17] Boni, B., Hindenlang, U., Kleiber, M. (1982) Finite element analysis of two- and three-dimensional elasto-plastic frames—the natural approach Computer Methods in Applied Mechanics and Engineering 35(2), pp 221-248.

[18] Ghosh, A., Made, A. (2002) Finite Element Analysis of Infilled Frames Journal of Structural Engineering 128 (7) https://doi.org/10.1061/(ASCE)0733-9445(2002)128:7(881).

[19] Mańkowski, J., Błaszczyk, B. (2015) Preliminary strength analysis of innovative frame of single seater sporting car Mechanik http://dx.doi.org/10.17814/mechanik.2015.2.44.

[20] Greco, M., Coda, H. (2004) A simple FEM formulation for large deflection 2D frame analysis based on position description Computer Methods in Applied Mechanics and Engineering 193 (33–35) pp 3541–3557.

[21] Caldas, R., Fakury, R. H, Sousa, J. Jr. (2014) Finite element implementation for the analysis of 3D steel and composite frames subjected to fire Latin American Journal of Solids and Structures http://dx.doi.org/10.1590/S1679-78252014000100001

[22] Material tables included in the program SolidWorks 2019.