# Scissor Lift Structural Analysis

M. Dub[1,*], M. Havlíček[1], F. Lopot[1]

[1] *Department of Designing and Machine Components, Faculty of Mechanical Engineering, Czech Technical University in Prague, Technická 4, 160 00 Prague 6, Czech Republic*

[*] *martin.dub@fs.cvut.cz*

**Abstract:** The contribution deals with the structure analysis of the damaged scissor lift steel structure. Scoped scissor lift has two parallel triple-scissor mechanisms connected with several lateral beams. The motion is realised by means of two hydraulic cylinders. The FEM analysis is carried out in software RFEM 5 for several load cases, that represent the possible loadings during the lift operation. The structure is modelled predominantly by means of beam elements. Non-linear static analysis is carried out. Results reveal that the structure is insufficiently dimensioned and critical areas correspond to the damaged parts of real structure.

**Keywords:** scissor lift; structural analysis; FEM; structure failure.

## 1 Introduction

The contribution deals with the structural analysis of the scissor lift. This scissor lift was design according to customer requirements and it is not a serial solution. The structural analysis is carried out due to the frequent and large damages, which occurred on the structure during common operation of the machine. The main goal is to identify the causes of these damages.

Scissor lift structure failure is inadmissible state and it can also cause serious injury to the operator. This is the motivation for analysis of fall arrest forces on the scissor lift structure in [1]. Design and finite element analysis of the scissor lift structure is carried out also in [2], [3] and [4]. From these sources follows that safety design of scissor is an actual topic. Also the development of the detailed FE model is the important topic to support and verify theoretical computations. FE models can be also used for the structure optimization and comparison of various design solutions, [6]. These computations can be supported by means of measurement by means of strain gauges. This measurement will provide the information about the deformations of the steel structure during various load cases.

Scissor lift is the lifting device and the structural computations should be in accordance with the appropriate standards. The assessment of this design is carried out with respect to the standard ČSN EN 1493+A1 Vehicle Lifts [5]. The loading of the scissor lift is realised in several load cases and also with respect to the dynamic loads during departure and breaking.

## 2 Problem description

The scissor lift consists of two parallel triple-scissor mechanisms connected with several lateral beams. The motion is realised by means of two hydraulic cylinders driven by means of 5.5 kW electric motor. Those are connected to the scissor structure by means of welded consoles with a considerable eccentricity. The spatial beam model of the structure appears in Fig. 1.

The scissor lift has the stroke 1800 mm and load capacity 1500 kg. The load area dimension is 1300 x 1200 mm (length x width). The structure of the scissors is realised by means of standard cross-sections 100x50x5, lateral beams are made of 100x100x5 profile, both made of steel S355. The beams of scissors are coupled by the pins with diameter 30 mm.

Damage occurred during common operation in quite short time and repeatedly after several reparations. The critical areas revealed in the pin joint surrounding (Fig. 3) and near the welds of the console and the lateral beam (Fig. 2). As the scissor lift was not overloaded, there was a suspicion of insufficient dimensioning of the steel structure. In further paragraphs the detail structural analysis is described.

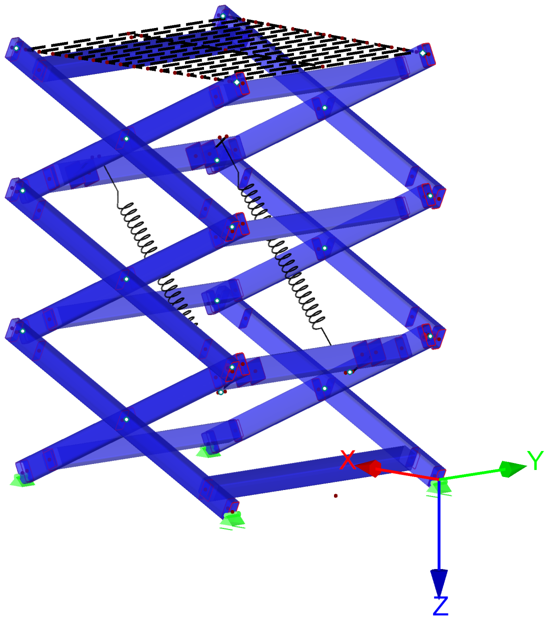


Fig. 1: The 3D beam model of scissor lift, hydraulic cylinders are modelled by means of spring members.

Fig. 2: Damage of the console for the hydraulic cylinders.

Fig. 3: Damage of the pin joint.

# 3 Structural analysis

## 3.1 Model description

Structural analysis is carried out by means of finite element analysis in RFEM 5, [7]. The structure is modelled by means of member elements. Linear elastic material model is used for construction steel S355 ($E = 210\,000$ MPa, $\nu = 0.298$, $\sigma_y = 350$ MPa). The structure is modelled by means of 190 beam elements with higher order definition. The pin joints of the beams are modelled by means of hinges with appropriate degree of freedom.

Hydraulic cylinders are modelled by means of non-linear springs to reach arbitrary position of the scissor lift in whole range. By means of these special members it is also possible to simulate asymmetric shift of the cylinders. This load case may occur due to the asymmetric placement of the load and also due to the design of the hydraulic circuit. Due to the usage of these members, the non-linear analysis has to be carried out.

The pin joint is then modelled more detailed by means of shell elements to get stress field around the hole for the pin. Internal forces obtained from the member elements are used for loading of the detailed shell model of the pin joint. In case of shell model of the pin joint, the basic elastic-plastic material model for steel S355 is used. Only a part of the profile is modelled. Total 11289 shell elements are used in RFEM 5. Global element size is set to 5 mm. Mesh refinement to 1 mm is used around the hole for the pin. Both beam and shell model simulations are computed on basic laptop with negligible computational time.

### 3.2 Load cases

The top platform is modelled as rigid and it is used for loading. The structure is loaded by means of self-weight and in addition by means concentrated force which represents the maximum weight of the load 15000 N. Within three load cases, there are simulated three different load positions of the force, Fig. 4. These load cases are based on the possible positions of the palette. Load case 1 (LC1) represents the ideal middle position of the palette. LC2 simulates the limit position of the palette, which is positioned to the edge of the top platform. LC3 simulates the limit position of the rotated palette. Furthermore, the special load cases are designed to simulate asymmetric shift of the cylinders. These load cases simulate alternative to the LC1 - LC3. This state is simulated by means of non-linear spring members. The shift difference is set to 10 mm. Together six load case variants are simulated. The worst assumed load case supposed to be LC3 with asymmetric shift of cylinders.

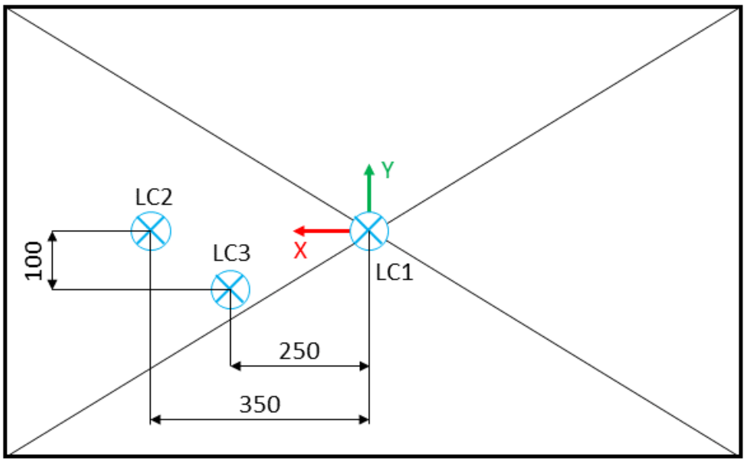


Fig. 4: Schema of the external loads on the top platform – three load cases.

## 4 Results

### 4.1 FEA results

As was mentioned above, results are calculated for three different load cases and of two configurations of hydraulic cylinders, total six simulated situations. The von Mises stress is evaluated both in member and shell model. The internal forces from member models are used for loading of the corresponding shell models.

In Fig. 5 and 6 there is the von Mises stress field on the whole structure (member model) and corresponding stress field around the hole for the pin (shell model) for load case LC1 (loading in the center of gravity) with asymmetric shift of the cylinders. During this load case the yield stress of the material is reached although no effect of dynamics is considered. The maximum stress occurs near the welded joint of the scissor beam and lateral beam. In the results from shell model, there can be seen very good agreement of the area in plastic state with the damaged area around the pin joint.

### 4.2 Evaluation

In case of the scissor lift structure, the limits given in standard [5] should be taken as a recommendation. This document can be used to determine the allowable stress $\sigma_A$. For the steel S355 the allowable stress is $\sigma_A = 237$ MPa for the basic material. This stress should be decreased due to the dynamic effects by means dynamic coefficient $\phi$.

$$\phi = 1.1 + 0.34v, \tag{1}$$

where $v$ is the speed of motion, which is for this scissor lift 85.7 mm/s. The final allowable stress $\sigma_{Af}$ is then

$$\sigma_{Af} = \frac{\sigma_A}{\phi} = 210 \text{ MPa}. \tag{2}$$

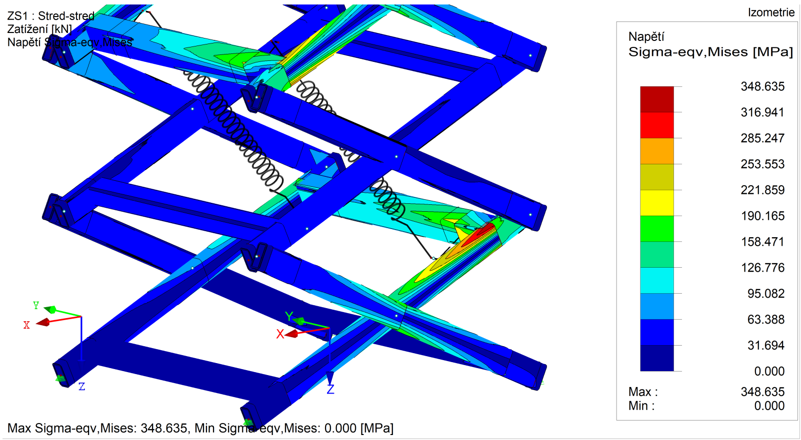


Fig. 5: Von Mises stress of the beam structure.

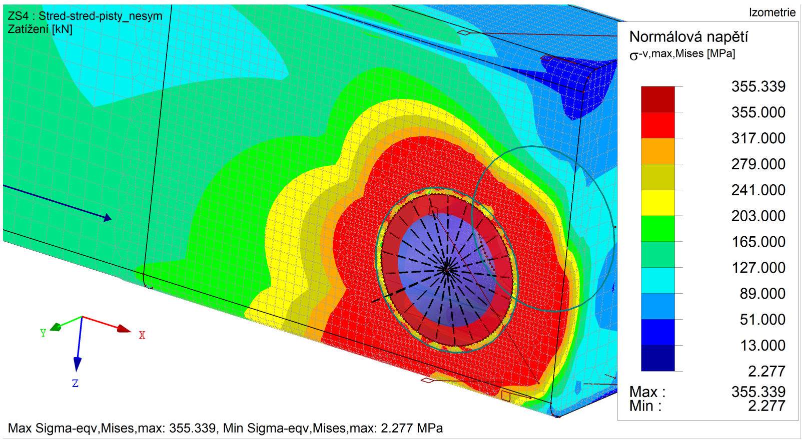


Fig. 6: Von Mises stress around the hole for the pin.

The maximum von Mises stress obtained from member model summarizes the Tab. 1.

Tab. 1: Maximum von Mises stress in the beam structure for all the load cases.

| von Mises stress [MPa] | Symmetric shift of cylinders | Asymmetric shift of cylinders |
|---|---|---|
| LC 1 | 252 | 348 |
| LC 2 | 262 | 357 |
| LC 3 | 278 | 359 |

If the values of von Mises stress calculated in member model are compared with the value of the allowable stress, it is evident that at the point of connection of the lateral reinforcement carrying hydraulic motors does not meet the allowable stress condition of the structure in either load case, even in the case of symmetrical shift of both pistons. In the case of asymmetric shift of the hydraulic motors by 10 mm, the stress value oscillates around the yield strength of material S355.

The maximum von Mises stress obtained from shell model of the pin joint summarizes the Tab. 2.

Tab. 2: Maximum von Mises stress in the shell model of the pin joint for all the load cases.

| von Mises stress [MPa] | Symmetric shift of cylinders | Asymmetric shift of cylinders |
|---|---|---|
| LC 1 | 194 | 355 |
| LC 2 | 205 | 355 |
| LC 3 | 224 | 355 |

The shell model results then shows the exceeding of allowable stress in load case LC3 (for the same shift of hydraulic cylinders). In case of asymmetric loading the plasticization of large areas around the hole for the pin occurs. In this case the material anisotropy of the welded joints is not taken into account. This will cause further decrease of the allowable stress.

Reached results proved the assumption that the worse load case is LC 3. But it can be stated that the structure does not fulfil the requirements for the scissor lift in the none of simulated load cases. Simulation of asymmetric cylinders shift leads to the yield stress reach.

## 5 Conclusions

Within the structural analysis three load cases and two configurations of the hydraulic cylinders were modelled and tested. The scissor lift was modelled at first by means of member elements then more detailed analysis of the pin joint was carried out using shell model. Performed structural analysis proved the underestimation of the scissor lift structure. The areas with maximum stress correspond well to the places of damage. The values of the von Mises stress were compared with the maximum allowable stress according standard for vehicle lifts [5]. Detailed analysis of the design calculations of the producer showed excessive simplification of the computational model. This simplification led to the wrong results and their misinterpretation. The simplification of the mechanical model is possible but it has to be taken into account during determination of the safety factor respectively the maximum allowable stress.

## References

[1] J. Harris, et al., Fall arrest characteristics of a scissor lift, Journal of Safety Research, 41, (2010), 213-220, doi: 10.1016/j.jsr.2010.01.004.

[2] S.A. Manoharrao, R. S. Jamgekar, Design and Analysis of Hydraulic Scissor lift by FEA, International Research Journal of Engineering and Technology, 3.10 (2016).

[3] R. Helmi, et al., Design Review of Scissors Lifts Structure for Commercial Aircraft Ground Support Equipment using Finite Element Analysis, Procedia Engineering, 41, (2012), 1696-1701, doi: 10.1016/j.proeng.2012.07.370.

[4] C. Kiran, J. Sruthi, Design and Structural Analysis of Scissor Jack using ANSYS Workbench, CVR Journal of Science and Technology, 15 (2018), 101-105.

[5] ČSN EN 1493 + A1. Vehicle Lifts. Prague: Czech Office for Standards, 2010.

[6] F. Starý, et al., Comparison of Different Construction Hoist Masts by FEM Analysis, in proc.: Proceedings of 58th International Conference of Machine Design Departments – ICMD 2017 (2017), Česká zemědělská univerzita v Praze. Technická fakulta, Praha-Horoměřice, 344-347.

[7] Dlubal Software GmbH, RFEM 5.26.02 (2020)