Stress analysis of lifting table using finite element method

N Rašović¹, A Vučina¹ and M Obad¹
¹University of Mostar, Faculty of Mechanical Engineering, Computing and Electrical Engineering, Matice hrvatske bb, 88000 Mostar, Bosnia and Herzegovina
E-mail: nebojsa.rasovic@fsre.sum.ba

Abstract. Lifting table has been designed and developed through the concept of Learning Factory (LF) at the University of Mostar. The idea for lifting table design has come from the local industry needs for a lifting platform that should lift a man and/or load at a certain height. For safety reasons, design is checked under the loading using a method of finite element analysis. The paper predicts and explains methodology for structural analysis used in presented case study. Results of FEM analysis are basis for making ways and guidelines to optimize current design in order to get optimal parameters for weight, stability, capacity, mobility and layout of the lifting table.

1. Introduction
From the previous work [1], design of lifting platform has been realized through learning factory concept at Department of mechanical engineering University of Mostar. It is about mechanical lifting mechanism that is able to carry great loads by vertically hand moving. A need to use a lifting mechanism is very widespread across labs, workshops, departments etc. The goal of developed design is to lift some objects or man at a certain height in order to perform some operations. The required highest level of lifting was up to 550 mm. The technical requirements for design have been defined by previous work and lifting table has been achieved as represented (Table 1).

| Technical Requirements, Constraints |
|-------------------------------------|
| Table dimensions: 600×300 mm        | Max. costs: 150 €                  |
| Max. lifting height: 550 mm         | Easy assembly of parts: ✔          |
| Min. height of design: 200 mm       | Easy to handle: ✔                  |
| Max. load carrying: 300 kg          | Easy dismantling: ✔                |
| Total structure mass: 30 kg         | Adopt to available technology at LF: ✔ |

Corrado et al. [2] has been focused on design and analysis of belt drive of lifting table scissor in only two characteristic positions, the highest and the lowest, where the stress was much lower than the yield stress of material (S275 and load carrying of 1000 kg). Dengiz et al. [3] has designed two-level scissor lifting system with hydraulic drive, where the stress was in the allowable limits (for St37, St50 and load carrying of 500 kg). Rani et al. [4] introduced even two hydraulic cylinders at two-level scissor for better operating and controlling. Besides stress analysis, some papers [5-8] have been dealing with design improvement by selecting some optimal criteria of topology optimization.
following sections, methodology of stress analysis and optimization of lifting table structure has been presented.

2. Objective and scope
Stress and strain analysis for the entire structure of lifting table represents one of the main tasks in the phase of embodiment design. This is the most important step because calculation results need to confirm design correctness. Correctness of design implies capability to carry out full predictable load, easy mechanical handling, safe lifting and smooth motion of the mechanism. Because of very low working speed, it will be observed three characteristic positions of the lifting table mechanism: the highest, the lowest and middle position under acting of maximal load. The objective of this paper is to check the stress and strain for all three characteristic positions and on that basis make ways and guidelines to optimize current design.

In accordance with previous mentioned, material selection has to be appropriate depending on component type and function it performs. The most of the components are made of structural steel (S275) like as two frames (upper and lower), four legs, longitudinally stiffener, transversely stiffener and lever. The four wheels are made of polyethylene. Slider and spindle screw nut are made of bronze, while spindle is made of stainless steel.

3. Methodology for structural analysis
Technical requirements (Table 1) will play main role in defining boundary conditions of parametrized model of lifting table. The Figure 1 shows workflow algorithm for structural analysis for previous defined requirements. Appropriate geometric mesh model that ensures low approximation errors and validity of obtained results has been created. This task can be done in few different derived mesh iterations in order to get congruence or confirmation of stress results.

Figure 1. Workflow algorithm of stress analysis using FEM.
To achieve full validity of results, all relationships between parts in their hierarchal order within assembly have to be established. For purpose of relationships defining, Ansys® SpaceClaim has been used. The most important, all connections (joints) have to be defined from the motion type aspect and all of them are constrained in certain axes directions. From simplification and calculation aspect, the spindle has replaced by circular beam like cylindrical rigid body with same diameter and material. Friction coefficient has been added between polyethylene wheels and structural steel frames, which is 0.2. Support is defined as fixed type and it is located on the lower frame at the bottom face. Force value is 3000 N and it is applied on the upper frame at the top face.

4. Case study
For structural analysis implementation, Ansys® Workbench has been used [9, 10]. All three positions are analysed under full load of 3000 N, which is applied on top face at the upper frame. Entire configuration of the lifting table scissor is considered as a rigid structure, not as a mechanism. That means the components in their contacts have been defined as bonded connections. Equivalent (von-Mises) stress and total deformation are obtained by structural analysis and the criterion for decision is value of yield stress for structural steel (275 MPa) as maximal limit in defining stress condition of the structure (Figure 2).

![Figure 2. Structural analysis for general mesh in the three characteristic positions under full load.](image)

The maximal reached value of stress has been 224.11 MPa, related to low position and it is located at welded joint of mounting bracket and lower frame (Figure 2, c). In the middle position, maximal stress value has been 219.51 MPa and it is located at spindle (Figure 2, b). The minimal value of stress has been 121.49 MPa, related to high position and it is located at lever (Figure 2, a). These results represent spotted values that become a base for further analysis. The greatest value of total deformations for all mechanism positions does not exceed 1.7 mm (Figure 2), so strain analysis is not considered in further procedure.
Taking into account workflow algorithm (Figure 1), a few different meshes are created to confirm obtained results from the first mesh iteration (General Mesh\(^{(1)}\)). Each mesh has to be analysed, since each following mesh is generated with respect to the results derived from the previous one. The goal is to confirm results from previous analysis (Figure 2), and the components with the greatest stress values have been taken to additional observing.

According to structural analysis (Derived Detailed Mesh\(^{(2)}\), Table 2), three components had maximal stress values: lever (in high position), spindle (in middle position) and mounting bracket at welded joint (in low position). These components are taken to further analysis and mesh details are listed in the Table 2.

**Table 2.** Mesh details for multiple iterations of structural analysis.

| Characteristics of Mesh | General Mesh\(^{(1)}\) | Derived Detailed Mesh\(^{(2)}\) | Derived Detailed Mesh\(^{(3)}\) |
|-------------------------|------------------------|-----------------------------|-----------------------------|
| Mesh Method             | Automatic              | Hex Dominant                | Hex Dominant                |
| Element Size [\(\text{mm}\)] | 6                      | 3                           | 1                           |
| Size Function           | Adaptive               | Curvature                   | Proximity and Curvature     |
| Mesh Quality            | Medium                 | Fine                        | Fine                        |
| Transition Ratio        | 0,272                  | 0,272                       | 0,272                       |

Using results derived from previous mesh analysis, three component with the highest equivalent (von Missis) stresses are excluded into individual observing (Derived Detailed Mesh\(^{(3)}\), Table 2) in order to get more precisely values. In this case, it should take into account that very small surface of elements could lead to enormously great and unreliable values of stresses. After the last iteration results have been represented (Figure 3).

**Figure 3.** Derived detailed results of previous parameterization for excluded components.
As the Figure 3 shows, the maximal stress values of the excluded components have been increased for 10% approximately. This increasing has been related to the case of derived detailed mesh and these stress values represent purified results through few mesh iterations. It can be noted that obtained results are compromise in numerical evaluation of multiple model parameters. Stress results obtained in this section will serve in creating guidelines for design improvement.

5. Guidelines for design improvement

Improvement of lifting table design using method of topology optimization is presented in this section. The procedure of topology optimization is based on equivalent (von-Mises) stress results in order to generate optimal shape of part or structure. For a part subjected to a given constraint (equivalent stress) the intention was to increase its stiffness on the one hand and to reduce the weight on the other. Topology optimization results have to offer a potential geometry for removal of less efficient material from total part volume. To find the most precise geometry for topology optimization, the number of iterations have to be increased as much as possible, as well as number of elements. It means that procedure of FE analysis has to find out and suggest as much as possible of passive elements in each iteration step. To obtain more accurate and precise results of topology optimization process, it has been used parameters for fine quality of meshing (Derived Detailed Mesh (3), Table 2). To conduct topology optimization process, the objective goal is compliance minimization as specific criterion under static structural environment. The term compliance in topological terminology is opposite to term stiffness. Therefore, minimizing the compliance is equivalent to maximizing the global structural static stiffness.

From FE analysis, three characteristic positions of the scissor mechanism have given three components with the greatest stress values. Criterion of response constraint is volume reduction, and for all structure parts is 85% to retain. Thus, process of topology analysis has shown potential cutting contour of excess material (Figure 4).

Changes in geometry have been made according to proposed pattern in topology analysis (Figure 4, a), the shape of pattern is mostly retained and new feature has been created through SpaceClaim modeller (Figure 4, b). Renew structural analysis has confirmed previous results in stress, so validity of topology optimization is proven (Figure 4, c). Based on maximal stresses occurrence, some guidelines for potential topology optimization of the other two components, ordered by priority would be:

For spindle:
1. Add radius fillet at two transition edges of the spindle.

For welded joint of mounting bracket and lower frame:
1. Increase the thickness of mounting bracket for greater strength;
2. Add radius fillet at welded mounting bracket with lower frame;
3. Place mounting bracket on steel plate and make welded joint between plate and frame;
4. Strengthen mounting bracket with additional welded seam at frame;
5. Improve quality of welded joints.

These guidelines have been ordered by degree of priority, and they will be subject of further stress and topology analyses in order to further improve lifting table design. Note that each change in topology or design must be within constraints listed in the Table 1.

6. Conclusion
In this paper, the methodology for stress and topology analysis of lifting table has been presented. This methodology has given a way to calculate values of stresses and deformations taking into account all mentioned constraints. Results obtained by FEM analysis has shown that stress does not occur always at the same place. Its distribution is depending of scissor position and for three different positions, there is transiting over three different components. FEM results have confirmed that the maximum stress value does not exceed value of yield stress. It means that static principles have satisfied. This type of structural analysis has served as introduction to topology analysis based on criterion of volume reduction. The goal was to retain rigidity while reducing mass in lower stress area, dominant blue colour in stress analysis. Initial proposed pattern of boundary shape has been adjusted and applied in feature creation. Further structural analysis confirmed previous stress results. Maximal stress appeared at same place as previous, and topology optimization has been proven.

References
[1] Rašović N, Vučina A and Dedić R 2018 Lifting Table Design in a Learning Factory at University of Mostar, Proceedings of the 4th International Scientific Conference on Mechanical Engineering Technologies and Applications COMETA 2018, 374–379
[2] Corrado A, Polini W, Canale L and Cavaliere C 2016 To Design a Belt Drive Scissor Lifting Table, International Journal of Engineering and Technology 8(1) 515–525
[3] Dengiz C, Şenel M, Yıldızlı K and Koç E 2018 Design and Analysis of Scissor Lifting System by Using Finite Elements Method, Universal Journal of Materials Science 6(2) 58–63, https://doi.org/10.13189/ujms.2018.060202
[4] Rani D, Agarwal N and Tirth V 2015 Design and Fabrication of Hydraulic Scissor Lift, MIT International Journal of Mechanical Engineering 5(2) 81–87
[5] Bankoti S, Jain N and Misra A 2015 Topological Optimization of 3D Structures by Optimality Criteria using ANSYS, International Journal for Research in Emerging Science and Technology 2(2) 30–35
[6] Li D and Kim I Y 2018 Multi-material Topology Optimization for Practical Lightweight Design, Struct. Multidiscip. Optim. 58(3) 1081–1094, https://doi.org/10.1007/s00158-018-1953-z
[7] Beňo P, Kozak D and Konjatić P 2014 Optimization of Thin-walled Constructions in CAE System ANSYS, Tehnički vjesnik 21(5) 1051–1055
[8] Picelli R, Townsend S, Brampton C, Norato J and Kim H A 2018 Stress-based Shape and Topology Optimization with the Level Set Method, Computer Methods in Applied Mechanics and Engineering 329 1–23, https://doi.org/10.1016/J.CMA.2017.09.001
[9] Rašović N, Vučina A and Dedić R 2019 Design and Analysis of Steel Reel Shaft by Using FEA, Tehnički vjesnik 26(2) 527–532, https://doi.org/https://doi.org/10.17559/TV-20180116103950
[10] Dizdar S, Vucina A, Rasovic N and Tomovic R 2018 Application of Pipestress and Ansys in Stress Analysis of Nuclear Pipe Support – Case Study, IOP Conference Series: Materials Science and Engineering 393(1) 012021, https://doi.org/10.1088/1757-899X/393/1/012021