FORCE AND STRESS ANALYSIS AT THE LINKAGE POINTS OF THE MINI LIFTING DEVICE BY ADAMS/VIEW AND INVENTOR SOFTWARE

PHÂN TÍCH LỰC VÀ ỨNG SUẤT TẠI CÁC ĐIỂM LIÊN KẾT CỦA THIẾT BỊ NÂNG MINI BẰNG PHẦN MỀM ADAMS/VIEW VÀ INVENTOR

Nguyen Thi Hai Van*, Nguyen Le Van, Nguyen Thai Duong

The University of Danang, University of Technology and Education; haivanbk2010@gmail.com

Abstract - Rotary joints, bearings, rollers, connectors, etc… are details that connect the axes together which are loaded equally or unequally. Building the strength calculation in combination with simulating the distribution of stresses in the use of allowable loads is a necessary and practical element for the active time of machine. Based on the theoretical calculations after model and layout survey, the paper presents the results of calculating the stress concentration at the link points by the method of separation of nodes in terms of conditions and load bearing design allowed. The study also shows the results of calculation after the test and the aggregation of forces in accordance with the links at the elements that need to be considered and given in the problem. These results are compared with those stimulated and calculated by Adams/View and Inventor softwares.

Key words - Mini lifting device; distribution of stresses; link points; method of separation of nodes; strength calculation.

Tóm tắt - Khớp quay, bạc đỡ, con lăn là các chi tiết kết nối các thanh lại với nhau và chịu tác dụng của tải trọng có thể đều hoặc không đều. Việc xây dựng bài toán tính toán sức bền kết hợp với mô phỏng việc phân bố các ứng suất trong việc sử dụng tải trọng cho phép là một yếu tố cần thiết và mang tính thiết thực trong suốt thời gian hoạt động. Trên cơ sở xây dựng lý thuyết tính toán sau khi khảo sát mô hình và cách bố trí, bài báo đã thể ện kết quả tính toán sự tập trung ứng suất tại các điểm liên kết bằng phương pháp tách nút trong điều kiện và phương án thiết kế chịu tải cho phép. Nghiên cứu cũng đã thể hiện rõ kết quả tính toán sau khi kiểm nghiệm và tổng hợp lực phù hợp với các liên kết tại các phần tử cần được xem xét và đưa ra trong bài toán. Các kết quả này được so sánh thông qua mô phỏng và tính toán bằng phần mềm Adams/View và Inventor.

Từ khóa - Thiết bị nâng mini; phân bố ứng suất; các điểm liên kết; phương pháp tách nút; tính toán sức bền.

1. Introduction

Nowadays, lifting devices have been widely progressed with a wide variety of shapes, sizes and designs such as hydraulic lifting, electric lifting, and mechanical lifting. However, they are large structures and produced in some developed countries such as China, Korea, Germany, USA… In Vietnam, we only buy and reuse these old equipment with simple support structure or get a new purchase at much higher price. Recently a number of studies have been conducted to develop and produce ideas for the development of these products on a smaller and moderate structure with reasonable prices in Vietnam condition [1].

In this study, we focus on calculation of stress distribution at the junctions of mini lifting equipment. Initially, we conducted a survey of previous studies related to lifting equipment, transport equipment… In these studies, in order to assess the effect of the damage at the welds, at location concentrated residual stresses, rust holes, or moving loads subjected to dynamic loads. Most of the evaluations were thanked to the forces of action, load, oscillation, working time and temperature environment led to performance, cracking or degrading. In addition to considering each location related to small forklift components called mini under the effect of the largest and smallest dynamic load, its frame structure, the clamping bar, belt and welds may stand the maximum residual stresses that previous studies have not yet mentioned and they just showed simulation results about the locations that have variable size or asynchronous force. For these reasons, scissor lifts with different capacities and elevating heights are increasingly used at many workplaces [2]. Unfortunately, fatal and non-fatal incidents have also happened during scissor lift operations [3]. Many of these incidents were associated with lift tip-over or workers falling within or from the platform.

The article focuses on determining static load conditions when lifting or standing. Then there is the level of dynamic load, with the impact that can be deformed, broken or reduced life after a period time of work. By analyzing the force at the linked locations via the method of separation of nodes, we consider the load-bearing positions warrant permissive stress resistance during the operation process of mini lifting equipment.

2. The model of mini lifting

In static conditions, the model of lifting equipment is shown in Figure 1 which has survey parameters to perform the simulation before machining. Above are some of the specifications of the load frame, load point and locations standing the stresses of the weld shown in figure 2. Initially, the maximum load bearing capacity at the height of the lifting device is 3 m and stand still (largest static load) is 150 kg (including the mass of the person and equipment during work). Then, during the process of control, it can increase the moving speed, braking sudden stops, or when heavy objects on the lifting device fall down, affecting the frame, welding; which are issues to be considered and noted during the allowable working time of this device. In addition, attention should be paid to the level and working conditions at the hinge joints, the fulcrum and the travel ability in allowable journey of the whole frame.

To solve this problem, this study proposes a moving-window concept for forecasting time series. This concept captures the importance of recent data. The most recent data is considered, and the oldest data are neglected. A window is used to select a range of data of interest. New data is added while some old data are dropped from the window as it moves forward in time [4, 9]. The length of the moving-window is kept constant whenever the window is moved. Figure 1 displays the moving-window concept. This method, therefore, limits the volume of data that is used to train the model while retaining the efficiency and general applicability of the model.

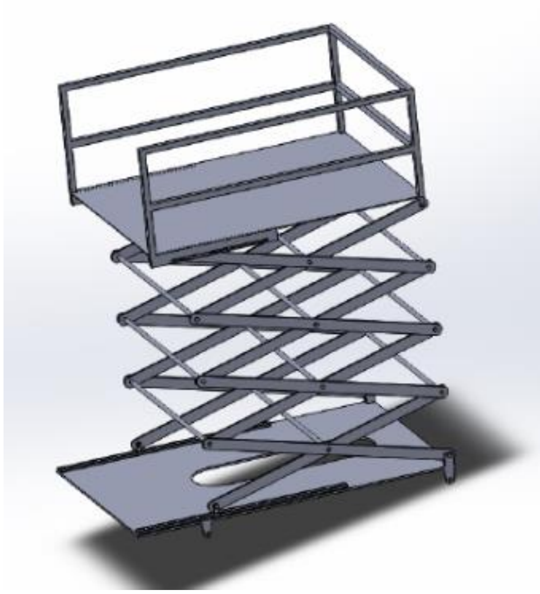

Figure 1. *3D model of mini lifting*

After conducting the model survey with the simulation process of the frame structure. Figure 3 shows the typical location for some links made to test the strength by the method of separation of nodes. The load is distributed on the upper frame where the lifting device and the person with maximum weight 150 kg are considered. The complete CAD assembly model was exported using Autodesk Inventor 2015 format as this format enables the data to be transferred directly between Inventor and other CAE software. Data transfer using the right format will avoid any missing data and eventually eases the meshing of the CAD model when generating the Finite Element Model for analysis [4].

3. Results and Discusses

3.1. Method of separation of nodes

Suppose that in the case of people and things near corner the most, it means that the value b_2=b/2=400mm, a_2=a/2=300mm. By replacing the designed geometry parameters with the maximum lifting capacity Q (N), we get the maximum value effecting on point B. Similarly, we have established the equilibrium force equations at nodes D and E.

Where: N_B=1375 (N), α=51^0, we define the axial force values on each bar.

With Q = 150*1.2 = 180 kg = 1800(N) and $[\sigma]_T$ = 14kN/cm^2 = 140.10^{6}N/m^2

$$N_B = \frac{G}{4}\left(1+\frac{2b_1}{b}\right)+\frac{G}{4}\left(1+2\frac{b_2}{b}+\frac{a_2}{a}\right)$$
$$=\frac{100}{4}\left(1+\frac{2x0}{800}\right)+\frac{1800}{4}\left(1+2\frac{400}{800}+2\frac{300}{600}\right)=1375(N)$$

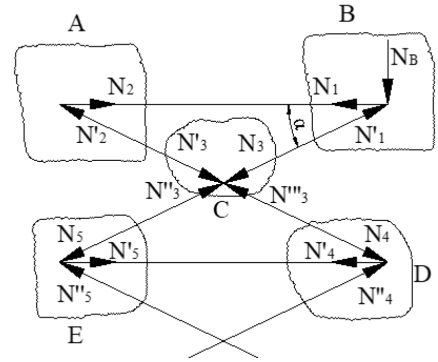

Figure 2. *Force distribution diagram in the case of the load focusing to one side*

After calculation, we have: $N_1 = N_2$ = 1095.4(N); $N'_1 = N'_2 = N_3 = N'_3 = N_4 = N_5$ = 1758.3(N); $N'_4 = N'_5$ = 2190.6(N)

The bars are in the mini lifting equipment with axial compressive force. Test the compressive strength of the bar [8]: $\sigma=\frac{F}{A}\leq[\sigma]$, With: $[\sigma]_T$ = 14kN/cm^2 = 140.10^{6}N/m^2.

On the bar 1:

$$\sigma_1=\frac{F}{A}=\frac{N_1}{216,16.10^{-6}}=\frac{1095,4}{216,16.10^{-6}} \tag{1}$$
$$=5,07.10^6\,N/m^2 \leq 140.10^6\,N/m^2$$

On the bar 2:

$$\sigma_2=\frac{F}{A}=\frac{N_2^{'}}{216.16x10^{-6}}=\frac{1758,3}{216.16x10^{-6}} \tag{2}$$
$$=8,134.10^6\,N/m^2 \leq 140.10^6\,N/m^2$$

The bars 1 and 2 bearing maximum load in this case satisfy the durable condition. Therefore, steel connection of mini lifting equipment is durable enough in the case of dynamic load or shift to a corner.

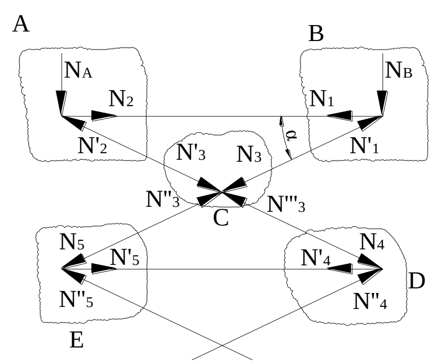

Figure 3. *Force distribution diagram in the case of uniform load*

In the case of people and things in the middle of the upper frame, the load distributes equally for 4 angles.

$$N_A = N_B = N_C = N_D = \frac{1800}{4} = 450(N)$$

Equilibrium equation at nodes: - with $N_A = N_B = 450N$, $\alpha = 51^0$.

We have: N'''3 = N'2 = 575.4(N), N''3 = N'1 = 575,4(N) and N''4 = N5 = N''5 = 575,4(N)

Check the durability of the bars:

Consider bars 1 and 2:

$$\sigma_{1,2} = \frac{F}{A} = \frac{N_1^{'}}{216.16x10^{-6}} = \frac{575,4}{216.16x10^{-6}} \quad (3)$$
$$= 2,66.10^6 N/m^2 \le 140.10^6 N/m^2$$

***Table 1**. Results of stress distribution at the linked points*

Case	N_1(N)	N'_1(N)	N_2(N)	N'_2(N)	N_3(N)
1	1095.4	1758.3	1095.4	1785.3	1785.3
2	358.5	575.4	358.5	575.4	575.4
Case	**N'_3(N)**	**N_4(N)**	**N'_4(N)**	**N_5(N)**	**N'_5(N)**
1	1785.3	1785.3	2190.86	1785.3	2190.86
2	575.4	575.4	717	575.4	717

The bars 3, 4 have a reaction force of N"4 = N"5=575,4N, so the stresses present in the bars are the same as these in bars 1 and 2, and both satisfy the durable conditions.

Consider all load cases of mini lifting equipment. The bars in the mini lifting equipment are compressed at the center, the compressive stress calculated in the bar is much smaller than the critical compressive one. For the case where the load is deflected at an angle, and also bearded the dynamic load, the stress value on the bars is $8,134.10^6 N/m^2$, corresponding to 1/17 times the allowable stress; This value is reduced to 1/52 in case of uniformly distributed load, ie in the middle of the upper frame of mini lifting equipment. Therefore, in all cases of work, the mini lifting device ensures durability and, through this result, the lifting capacity of the lifting device can be increased to a larger value.

3.2. Analyze forces at linkage points by ADAMS/View

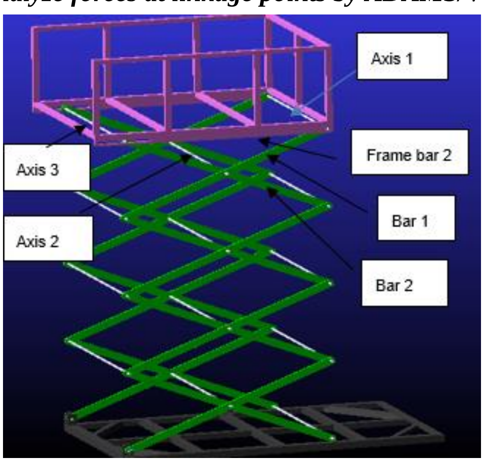

***Figure 4.** Simulation diagram in ADAMS/View*

We use Adam dynamite simulation software to simulate and determine the force on the bars at the assembly points. In this simulation, we also consider the case of eccentric load, it means that the load of 1500 N is set at an angle and in the case of the load placed in the middle of the upper frame and divided equally for the lifting frame. Lift angle is from 0 to 51^0 is the angle corresponding to the lowest and highest position of the lifting frame.

Simulation time: 0.5s, Steps: 5000 - Speed of pulling the shaft (when pulling the bottom shaft) v = 50mm/s.

Initially, because the gap exists, so when the work starts, there will be a collision; so the force will be high, but it will self-stabilize fast as we can see in Figure 5 and Figure 6.

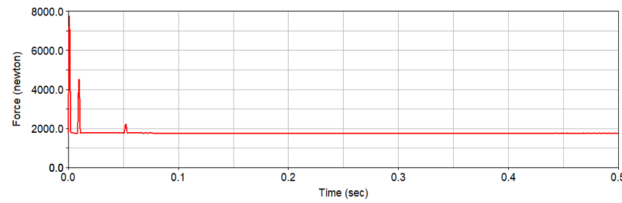

***Figure 5.** Axial force inside bar 1 (Fmax = 1777N)*

In the case of the load placed in the middle of the upper frame

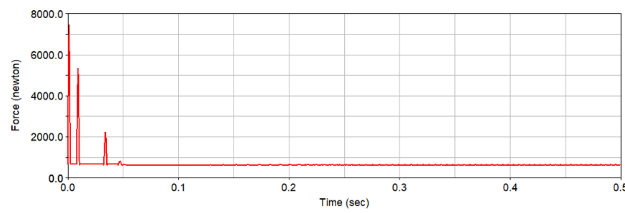

***Figure 6.** Axial force inside bar 1 (Fmax = 680N)*

The results of the ADAMS/View simulations are similar to those from manual analysis and computation, 1758N vs. 1777N and 575N vs. 680N.

Durability test

In the case of the load placed at an angle

On the bar 1:

$$\sigma_1 = \frac{F}{A} = \frac{N_1}{216.16x10^{-6}} = \frac{1777}{216.16x10^{-6}} \quad (4)$$
$$= 8.22x10^6 (N/m^2) < 140x10^6 (N/m^2)$$

On the bar 2:

$$\sigma_2 = \frac{F}{A} = \frac{N_2}{216.16x10^{-6}} = \frac{1263}{216.16x10^{-6}} \quad (5)$$
$$= 5.84x10^6 (N/m^2) < 140x10^6 (N/m^2)$$

In the case of the load placed in the middle of the upper frame

On the bar 1:

$$\sigma_1 = \frac{F}{A} = \frac{N_1}{216.16x10^{-6}} = \frac{680}{216.16x10^{-6}} \quad (6)$$
$$= 3.15x10^6 (N/m^2) < 140x10^6 (N/m^2)$$

On the bar 2:

$$\sigma_2 = \frac{F}{A} = \frac{N_2}{216.16x10^{-6}} = \frac{1263}{216.16x10^{-6}} \quad (7)$$
$$= 5.84x10^6 (N/m^2) < 140x10^6 (N/m^2)$$

3.3. Finite Element Analysis Simulation

In this research, Inventor software is the main CAD solid modelling software used. With its extensive features and powerful modelling tools, it is fully utilized in the

CAD modelling stage. To run the finite element analysis (FEA) simulation by using Autodesk Inventor software, it is necessary to generate the Finite Element Model of the mini lifts structure. Because, much progress has been made finite element method for analysis and today it is viewed as a general procedure of solving discrete problems posed by mathematically defined statements with multiple of numerical experiments that can be carried out [9]. However, only the inner mini lifts is used in this analysis because taking into account that if the inner mini lift can sustain the load exerted on it, the outer mini lift will happen to be safe during operation. The finite element model of the inner mini lift had been meshed. There are a simulations run by the Autodesk software upon the lift finite element model whereby every simulation, the number of meshes of seeds used is decreased to generate a much finer mesh for the finite element model and gives more accurate result (Figure 7).

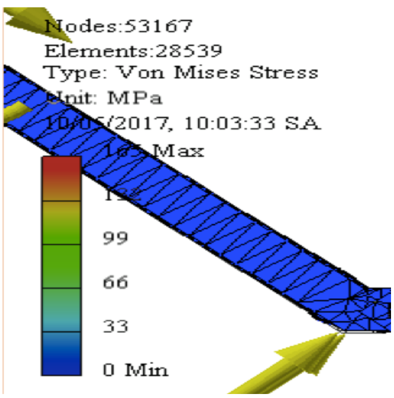

Figure 7. *Number nodes and elements for FEA*

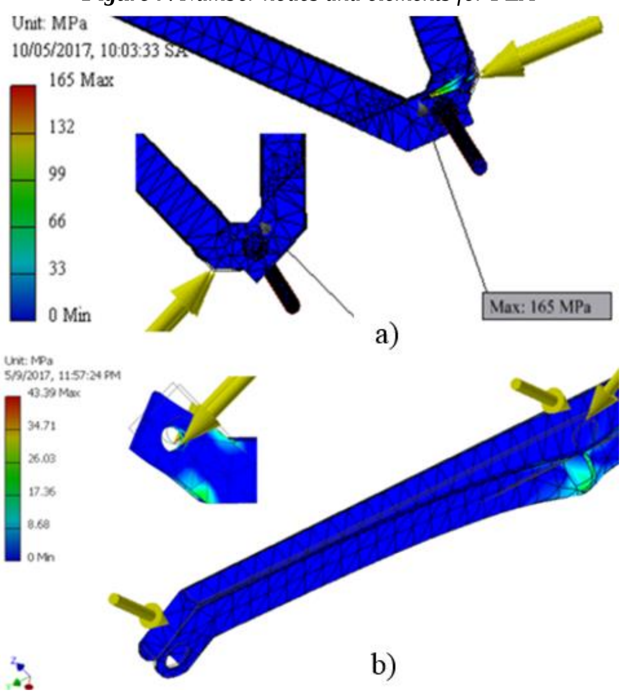

Figure 8. *Maximum Deflection of Inner mini lift*

Figure 8 (a) shows the maximum deflection that occurs on one side of the inner mini lift where the van body is attached to the lift through the FEA simulation using the default size of mesh characteristic. Meanwhile Figure 8 (b) shows the maximum deflection using the finest mesh. These deflections are shown in red color in this figure. Graphically from Figure 8, the deflections from both size of mesh look the same. Because of the different deformation factor scales that are set for each simulation, the legend shows the possible deflections that may occur on the structure. The outer mini is not analyzed because by analyzing the inner mini lifts alone, it is sufficient enough to validate whether the outer lifts maximum deflection is below the calculated maximum deflection or not. Because, the inner lift holds a load of 45% higher than the outer lift, which means that the inner lift will deflect more than the outer lift.

Table 2. *The different result between method of separation of nodes, ADAMS and FEA*

Methods	Case 1		Case 2	
	Bar 1 (MPa)	Bar 2 (MPa)	Bar 1 (MPa)	Bar 2 (MPa)
Method of separation of nodes	8.13	5.07	2.66	2.66
ADAMS/View	8.22	5.84	3.15	3.15
FEA	8.2	5.15	2.61	2.61
Different	0.86%	1.58%	1.91%	1.91%

The table 2 shows the difference of stress between three methods: separation of nodes, finite element analysis method and ADAMS/View. In case 1, the case of people and things near corner the most, the stress in bar 1 is similar with 8.13MPa, 8.22MPa and 8.2MPa, respectively and the different stress about 0.86%. While the stress in bar 2 is smaller than that in bar 1 with stress are near 5MPa. In case 2, the case of people and things in the middle of the upper frame, the highest stress in bar 1 and bar 2 is 3.15MPa, and the different stress of bar 1 and bar 2 are similar with 1.91%. The stress in bar 1 and bar 2 in two cases are significantly smaller than limited stress.

4. Conclusions

After the survey about model, simulations of the bonding locations at the nodes for the calculation of strength and stress analysis at those points are carried out. This article has made a complete analysis and presents methods of determining the force at the metal-structured locations of each link in the whole lifting system. By using compressive stresses and nodal cut methods in cases of dynamic load and static load, we can see that the bearing and distribution of load are unequal at each linked point. With equilibrium method built at nodes A through E, a matrix to compute by computer can be created. Thus, it can decrease the analysis time for determining forces in the metal structure of a lift system which has identified several basic calculation methods. The calculation of compressive stress by the method of separation of nodes in the case of dynamic load allows a scalability in the next study to construct a hardness matrix described as a program for the computer.

The results also carry out the finite element analysis method by using Inventor CAD software to mesmerize and find the allowable stress at the linked positions of the nodes. Moreover, this study also performs force analysis with Adams/view software and demonstrate that the mini lifting is safe.

Acknowledgements

This research is funded by Funds for Science and Technology Development of the University of Danang under project number B2017-ĐN06-08.

REFERENCES

[1] http://hamachiglobal.com.vn/

[2] M. J. Burkart, M. McCann and D. M. Paine. Aerial Work Platforms, in Elevated Work Platforms and Scaf-folding. McGraw-Hill, New York, (2004).

[3] M. McCann. Death in Construction Related to Personnel Lifts, 1992-1999, Journal of Safety Research, Vol. 34, No. 5, (2003), pp.507-514.

[4] J. Kim, M.J. Pratt, R.G. Iyer, and R.D Sriram. Standardized Data Exchange of CAD Models with Design Intent. Computer-Aided Design, 40 (7), (2008), pp.760-777.

[5] Huỳnh Văn Hoàng, Trần Thị Hồng, Kỹ thuật Nâng Chuyển, Nhà Xuất Bản Đại Học Quốc gia TP. Hồ Chí Minh, (2004).

[6] Thái Hoàng Phong, Phan Kỳ Phùng, Giáo trình sức bền vật liệu, Nhà xuất bản Khoa học và Kỹ thuật, Hà Nội, (2006).

[7] В.Г.Ананин, С.Нураков, В.С.Калиниченко, А. Б. Калиев. Определение оптимальных параметров металлоконструкции подъёмника сопряженно-рычажного типа, (2011).

[8] Đỗ Kiến Quốc, Nguyễn Thị Hiền Lương. Sức bền vật liệu. Nhà Xuất Bản Đại Học Quốc gia TP. Hồ Chí Minh, (2004).

[9] S. Mingzhou, G. Qiang and G. Bing. Finite element analysis of steel members under cyclic loading. Finite Elements in Analysis and Design, 39 (1), (2002), pp. 43-54.

(The Board of Editors received the paper on 10/10/2018, its review was completed on 21/12/2018)