DOI 10.34185/1562-9945-3-146-2023-04
UDK 62-237: 004.94

V.V. Strelbitskyi, L.V. Bovnegra, A.V. Pavlyshko

ANALYSIS OF STRENGTH OF A FORK OF THE FORKLIFT BY NUMERICAL METHODS

Abstract. Forklift is an industrial power truck, commonly called forklifts or lift trucks, then usually have two forks the can be trial power truck used forklifting and transport materials. Forklifts, to this day, the fork strength of a forklift under load still seems to be one of the biggest issues. Fork's related structural design and detail have a significant impact on the overall performance and reliability of a forklift. In this work, static analyses of a fork forklift were performed using Finite Element Method, on the free software FreeCAD, a multidisciplinary software application that is the result of a long-term active collaboration between developers and users. Open science hardware is cheaper, easier to maintain, and more accessible to more people. In this paper, by calculating the maximum normal stress of the section, the force situation and allowable stress of the fork are analyzed. The fork is made of structural material is 40Cr steel. The output values of the static study consist in establishing the distribution of equivalent unitary stresses (stress state) according to the von Mises criterion. The maximum stress of the structure is 128 MPa, which is does not exceed the allowable. Then in order to enhance the structural strength of the weak part, the measures of gradually increasing the fillet radius of the fork root were proposed, and the radius were selected as 35mm and 45mm, the weight of the forklift load was still 0,5t. Numerical experiments have shown that by increasing the radius from 25 to 45 mm, stresses decrease by 1.1...1.15 times, which is consistent with the theory of basic mechanics that stress concentration can easily lead to fatigue failure of the structure. Both numerical calculations and visual analysis showed consistency, with stress concentrations observed at the fork neck, demonstrating the accuracy of the established finite element model.
Keywords: forklift, forks, stresses, finite element analysis, FreeCAD.

Introduction. Because of the varieties, different shapes, different packaging of the goods, loading and unloading has always been a heavy process during transportation. Loading operations directly influences the high level of probability that delivery will be delivered on time, resulting in an impact to customers' service. Due to moment's profitable problems, the demand for universal lifting and transporting vehicles with high effectiveness is high. Thanks to this device, the storage capacity has increased significantly. Forklifts can be used in many places, such as warehouses,

ISSN 1562-9945 (Print)
ISSN 2707-7977 (Online)

factories, farms, seaports, construction sites, supermarkets, etc. Forklift usage can vary from a few hours a day to 24 hours a day, 7 days a week, used indoors and outdoor purposes. In practice, opinions regarding forklifts engagement are left to the storehouse experts. The existing forklift design has rotation restrictions, and the structure has potential safety hazards. The gasoline or propane forklifts are much stronger or faster as compared to electric forklifts, but considered to be difficult to maintain, and it is also fuel efficient. The most important part of a forklift is the fork which irresponsible for carrying the load is forklift fork. It should be noted that while the certification process for each wheel hoist model, the transport and lifting performance of the equipment is tested and specified, and the limits of operating capabilities and conditions of use are measured. The fork is the most popular removable equipment item of the fork lift trucks. Forklifts usually have two forks on the can be moved forward and backward, and they are the reason the machine is called a forklift and are therefore critical to safe operation [1,2].

Literature Review. The strength of loader attachment is an important indicator of reliability and is the focus of forklift research [1-5]. In order to estimate the influence of the applied loads on the material, it becomes necessary to assess the strength of the forks. The fork strength of the forklift under loading still seems to be one of the biggest concerns till date. The traditional methods of stress analysis and fatigue research are mainly carried out through static calculation and prototype trial-manufacture, and modified according to the experimental results [3-5]. However, it takes a lot of cost to repeat the above work.

The development of finite element numerical simulation technology the method of calculating structural strength based on computer aided design (CAD) and finite element method (FEM) has been widely d in engineering field [6-11]. Thus, researchers paper [9] analyzed static stresses in the telescopic masts of forklift, using FEM, and the error of analysis and test was less than 10%. In paper [12] using the FEA method, it was found that, it was found that the maximum stress gradient direction was in agreement with the crack initiating direction. Usually the crack the vertical to the larger of two top stresses. In work [13], a static analysis of a simplified fork design was carried out, the results of which served for the structural design of a forklift fork. Cyclic loading occurs under normal operating conditions on various machine components leading to the role of fatigue failure mechanisms [14-15].

The stresses in the forks depend on the conditions of their fastening, loading and design features, therefore, the **fundamental goal of this work is** analysis of strength of a fork of the forklift by numerical methods.

Research methodology and results. There was a fork breakdown on one of the forklifts, all the others were examined. A detailed study of the operating conditions and structural integrity of all in-service forks made it possible to detect cracks in the forks. Analysis of the work [2] showed the stress concentrations typically serve as fatigue crack initiation sites.

FEA does not require physical prototyping and can be used to analyze any part/component of the overall system under specific operating conditions. The designing dimensioning and modelling of the fork was executed on the free software FreeCAD, a multidisciplinary software application that is the result of a long-term active collaboration between developers and users [7]. Open source scientific hardware promotes open science by facilitating the comparison of scientific experiments.

The model was developed in step [16]:

1.Modeling the geometry: creating the geometry with FreeCAD.

2.Creating an analysis. Adding simulation constraints such as loads and fixed supports to the geometric model.

3.Adding materials to the parts of the geometric model.

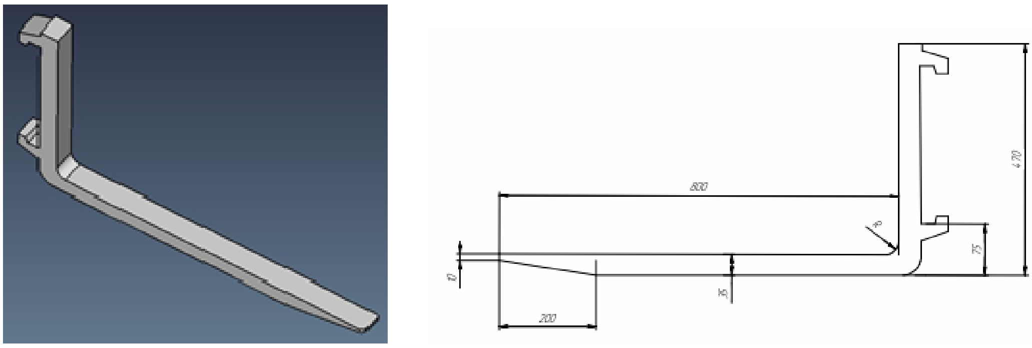
4. Creating a finite element mesh for the geometrical model.

5. Solving: running an external solver from within FreeCAD and postprocessing: visualizing the analysis results from within FreeCAD.

Accurate and reliable computational model is the basis of finite element analysis.

In this paper, the 0,5 t loader is used as the object.

The fork studied in this paper is a widely d hanger type. In fig.1 shows the solid 3d model (a) and dimensions (b) of forklift fork.



a) b)

Figure 1 - 3d model (a) and dimensions (b) of forklift fork

The forks are made of steel structural material is 40Cr steel. Fillet radius of the fork root R=25 mm. According to the actual use of the fork, the constraints and force

ISSN 1562-9945 (Print)
ISSN 2707-7977 (Online)

loads of the finite element analysis are determined (fig.2) and triangular element mesh wads, as shown in fig.3. Most literatures [9,12,14] regard the load as a concentrated load (fig.2).

Figure 2 - Fork support's restricted degrees of freedom

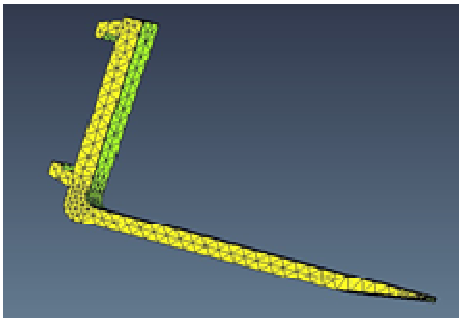


Figure 3 - Mesh model

The output values of the static study consist in establishing the distribution of equivalent unitary stresses (stress state) according to the von Mises criterion. The fig. 4 shows that under concentrated load, the fork has stress concentration problem in the fillet area of the fork root, then are in accordance with the theoretical force analysis, which verifies the accuracy of the model to a certain extent. However, the maximum stress of the structure is 128 MPa, that is greater than the allowed stress 121 MPa.

From Fig. 3 it can be seen that the stress concentration is mainly in the fillet area of the fork root. Therefore, in order to enhance the structural strength of the weak part, the measures of gradually increasing the fillet radius of the fork root were proposed, and the radius were selected as 35mm and 45mm, the weight of the forklift load was still 0,5t. The calculation results are shown in fig.5, the relationship between the maximum structural stress and the fillet radius which is shown in fig. 6.

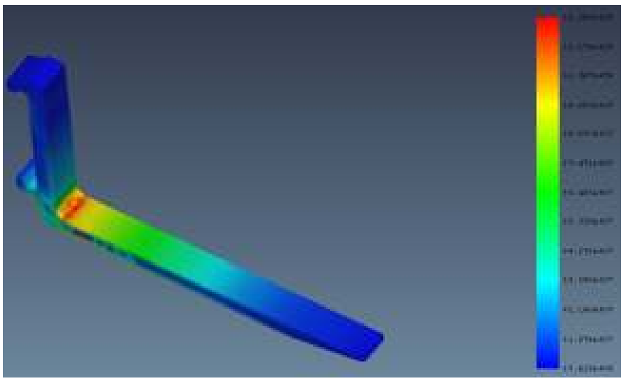
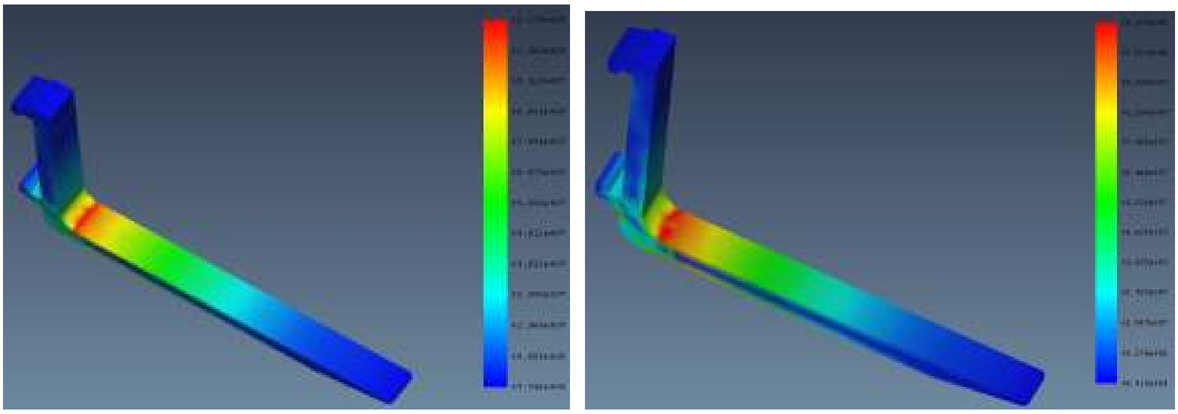


Figure 4 - Result of finite element analysis for R=25 mm



a) b)

Figure 5 - Result of finite element analysis for R=35mm (a) and R=45 mm (b)

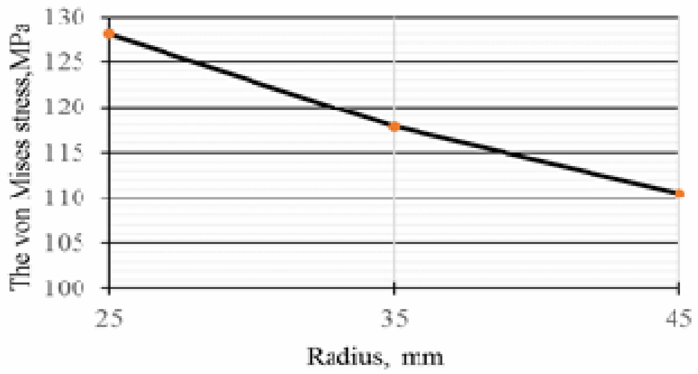


Figure 6 - Dependence of maximum structural stress and fillet radius

From Fig.8 it can be seen that as the radius increases, the stresses decrease, which is consistent with the basic mechanics theory that stress concentration can easily lead to structural fatigue failure, the dependence is nonlinear. In our case, the stress drops in 1,1...1,15 times

The maximum stresses at the fork section can be explained are the result of the contribution of the bending moment and the load being lifted. The analysis of the presented results shows of the fork root, which is a dangerous working condition and the fork is prone to excessive and may ca surface damage or even breakage in the fillet of the fork root, which is a dangerous working condition. the fork is prone to excessive and may ca surface damage or even breakage in the fillet area of the fork root, which is a dangerous working condition. Consequently, it is more appropriate to change the fillet radius to 45mm to meet the strength requirement that the maximum stress is less than the allowable stress.

The remaining areas are relatively stress free.

ISSN 1562-9945 (Print)
ISSN 2707-7977 (Online)

Conclusions. When the fork is subjected to continuous load, it may cause the crack development and surface to be damaged. The fork's structural design plays a key role in the efficiency and reliability of the forklift. According to the actual use of the fork, the constraint conditions and force loads for finite element analysis were determined, calculations were made performed using Finite Element Method in the FreeCAD. After adding geometric model, meshing, setting material parameters, constraints and loads, the finite element calculation model of the original fork was established. Numerical experiments have shown that an increase in radius leads to a decrease, which is consistent with the theory of basic mechanics. As experiments [1] show, the orientation of fatigue zone advocates that crack initiation occurred at the outer fork side, which constitutes the designed compression zone confirms the accuracy of the accuracy of the established finite element model. Calculation results show the radius from 25 to 45 mm, stresses decrease by 1.1...1.15 times. This study will be useful for scientific experts in the forklifts.

REFERENCES

1. Pantazopoulos G. Analysis of abnormal fatigue failure of forklift forks / G. Pantazopoulos, A., Vazdirvanidis, A. Rikos, A. Toulfatzis // Case Studies in Engineering Failure Analysis. – 2014. – Т. 2. – №. 1. – pp. 9-14.

2. Massone J. M., Boeri R. E. Failure of forklift forks / J. M. Massone, R.E. Boeri // Engineering failure analysis. – 2010. – Т. 17. – №. 5. – pp. 1062-1068.

3. Jigh B.H.G. Low cycle fatigue analyses of open-celled aluminum foam under compression–compression loading using experimental and microstructure finite element analysis / B.H.G. Jigh, H. Hosseini-Toudeshky, M.A. Farsi //Journal of Alloys and Compounds. – 2018. – Т. 797. – pp. 231-236.

4. Zhen C. Numerical and Experimental Study on Trimaran Cross-Deck Structure's Fatigue Characteristics Based on the Spectral Fatigue Method / C. Zhen, G. Feng, T. Wang, P. Yu //Journal of Marine Science and Engineering. – 2019. – Т. 7. – №. 3. – p. 62.

5. Deshmukh P. Impact of shifting load centers on the stability of the forklift / P. Deshmukh, G.K. Wadhwa //International Journal of Computer Engineering in Research Trends. – 2018. – Т. 5. – №. 4. – pp. 92-97.

6. Zhou Y. Theoretical and numerical investigation of stress mode shapes in multiaxial random fatigue / Y. Zhou, J. Tao // Mechanical Systems and Signal Processing. – 2019. – Т. 127. – pp. 499-512.

7. Strelbitskyi V., Bovnegra L., Punchenko N. Application of free software FreeCAD for modelling and analysis of generated stresses of Jib crane beam. INFORMATION SYSTEMS AND TECHNOLOGIES IST-2021. Proceedings of the 10-th International Scientific and Technical Conference, Kharkiv - Odesa, Ukraine, September 13-19, 2021, pp.131-135.

8 Florescu V. Sescu-Gal Research on the Parameters Influencing the Numerical Analysis of the Fatigue Behaviour of a Forklift-ARM / V. Florescu, S. Mocanu, A. Neagu C. //Romanian Journal of Transport Infrastructure. – 2021. – Т. 10. – №. 1. – pp. 1-10.
9. Sequeira A.A. Design and Fabrication of Battery Operated Forklift/ A.A. Sequeira, S. Mohammed, A.A. Kumar, M. Sameer, Y.A. Kareem, K.H. Sachidananda //Journal Européen des Systèmes Automatisés. – 2019. – Т. 52. – №. 6. – pp. 569-574.
10. Zhang E. Structural optimization of forklift fork based on numerical simulation and mathematical modeling of stress and fatigue / E. Zhang, J. Zhuo, Z. Liu, T. Guo //Revista Internacional de Métodos Numéricos para Cálculo y Diseño en Ingeniería. – 2020. – Т. 36. – №. 1. – pp.1-12.
11. Akin J. E. Finite element analysis concepts: via SolidWorks. – World Scientific, 2010. - 313 p.
12. Xu X. Fatigue failure of an intermediate transition block in fuel-injection pump fork assembly of a truck diesel engine / X. Xu, Z. Yu, B. Yu //Engineering Failure Analysis. – 2018. – Т. 94. – pp. 13-23.
13. Gu X, Wang W., Lin N., Zhang G., Gu X. The Finite Element Analysis for Fork Based on ANSYS //2015 International Conference on Electrical, Electronics and Mechatronics. – Atlantis Press, 2015. – pp. 152-154.
14 Pantazopoulos G. Fatigue failure of steel links operating as chain components in a heavy duty draw bench / G. Pantazopoulos, A. Vazdirvanidis, A. Toulfatzis, A. Rikos //Engineering Failure Analysis. – 2009. – Т. 16. – №. 7. – pp. 2440-2449.
15. Strelbitskyi V. Eksperymentalne doslidzhennia vplyvu napratsiuvannia ta asymetrii tsyklu na trishchynostiikist stalei portalnykh kraniv / V.V. Strelbitskyi, O.O. Nemchuk // Herald of Khmelnytskyi National University. – 2020. – № 1. – pp. 245–248. (In Ukraine)
16.Manual: What is FreeCAD [Internet resource]. – Режим доступу: https://wiki.freecad.org/FEM_Workbench.
17. Wang Y. Y. Tensile strain limits of girth welds with surface-breaking defects Part II experimental correlation and validation / Y.Y. Wang, D. Horsley, W. Cheng, A. Glover, M. McLamb, J.Zhou, R. Denys //Pipeline Technology, Proceedings of the 4th International Conference on Pipeline Technology. – 2004. – pp. 9-13.
Received 29.03.2023.
Accepted 03.04.2023.

Аналіз міцності вил вилочного навантажувача чисельними методами

Навантажувачі - це промислові силові машини, які називають навантажувачами або вилочними навантажувачами, котрі мають дві вилки. Їх широко використовують на складах, будівельних майданчиках, у морських портах, супермаркетах та ін.. Їх можуть експлуатувати від декількох годин на день до 24 годин на

ISSN 1562-9945 (Print)
ISSN 2707-7977 (Online)

добу, 7 днів на тиждень, як в приміщенні, так і на відкритому повітрі. На продуктивність та надійність навантажувача впливають конструкція та деталі, пов'язані з вилкою, яка відповідає за надійне транспортування вантажу. На сьогодні при дослідженні напруженого стану конструкцій в галузі машинобудування широко використовують чисельні методи розрахунку, а саме моделювання методом скінчених елементів (МСЕ). Аналіз робіт з визначення статичних напружень у конструкціях вилочних навантажувачів з використанням МСЕ показав, що похибка між результатами аналізу і випробувань не перевищила 10%. Напруження у вилах залежать від умов їх кріплення, навантаження і конструкції. У статті виконано аналіз напруженого стану вил вилочного навантажувача за допомогою методу кінцевих елементів у безкоштовному програмному забезпеченні FreeCAD, який є результатом довгострокової активної співпраці між розробниками та користувачами. Безкоштовне програмне забезпечення простіше в обслуговуванні та доступне для більшої кількості людей. Вилка виготовлена з конструкційного матеріалу - сталі 40Х. Було встановлено, що напруження у в області округлення кореня вилки є максимальні та складають 128 МПа. Для зменшення напружень було запропоновано збільшити радіус з 25 м до 35мм та 45 мм. Проведені чисельні експерименти показали, що при збільшенні радіуса з 25 до 45 мм призводить до зменшення напружень у 1,1...1,15 рази, що узгоджується з теорією базової механіки про те, що концентрація напружень може легко призвести до втомного руйнування конструкції.

Стрельбіцький Віктор Васильович – доцент, кандидат технічних наук, кафедра Підйомно-транспортні машини та інжиніринг портового технологічного обладнання, Одеський національний морський університет.

Бовнегра Любов Віталіївна - доцент, кандидат технічних наук, завідувач кафедри інформаційних технологій проєктування та дизайну, Національний університет «Одеська політехніка».

Павлишко Аандрій Володимирович - доцент, кандидат технічних наук, доцент кафедри інформаційних технологій проєктування та дизайну, Національний університет «Одеська політехніка».

Strelbitskyi Viktor Vasylovych – PhD, associated professor, Hoisting and transport machines and engineering of port technological equipment, Odessa National Maritime University.

Bovnegra Liubov Vitaliivna - PhD, associated professor, Head of Department of Design Information Technologies and Design, Odessa Polytechnic National University.

Pavlyshko Andrii Volodymirovich - PhD, associated professor, Odessa Polytechnic National University.