DOI: 10.34185/1991-7848.itmm.2023.01.062

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

Bovnegra Liubov[1], Strelbitskyi Viktor[2]
[1]*Odessa Polytechnic National University, Ukraine*
[2]*Odessa National Maritime University, Ukraine*

Abstract. Forklifts are often used in close quarters, and the high load capacity and working intensity put forward high performance requirements for the fork. The structural design of the fork plays a key role in the efficiency and reliability of the forklift. In the work, the pitchfork strength of the forklift structure was studied using package modeling and finite element analysis FreeCAD. Lifting the forks with a load can cause the forks to deform, causing premature cracking and rapid growth and eventual failure. Both numerical calculations and visual analysis showed consistency, and stress concentrations were observed at the fork neck, demonstrating the accuracy of the established finite element model.

Keywords: forklift; forks; stresses; finite element analysis.

Forklifts can be used in many places, such as warehouses, factories, farms, seaports, construction sites, supermarkets, etc. Forklifts usually have two forks on the front for lifting and moving goods. They are the reason the machine is called a forklift and are therefore critical to safe operation [1]. The strength of loader attachment is an important indicator of reliability and is the focus of forklift research. 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 that these lead to multiple cracks due to high stress concentrations.

The traditional stress analysis method mainly adopts static calculation and prototype trial production, and is modified according to the experimental results [3]. Destructive and non-destructive physical testing is often expensive due to the long time and production of prototypes and can impact time to market. Fork stresses can be efficiently analyzed using FEA. FEA does not require physical prototyping and can be used to analyze any part/component of the overall system under specific operating conditions. We used the computational FreeCAD software FreeCAD, a multidisciplinary software application that is the result of a long-term active collaboration between developers and users [4]. Open source scientific hardware promotes open science by facilitating the comparison of scientific experiments. The model was developed in step [5].

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 1 ton loader is used as the object.

The fig.1 shows the solid model of forklift fork. The forks are made of steel structural material is 40Cr steel. According to the actual use of the fork, the constraints and force loads of the finite element analysis are determined.

The fig. 2 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 57,12 MPa, which is does not exceed the allowable.

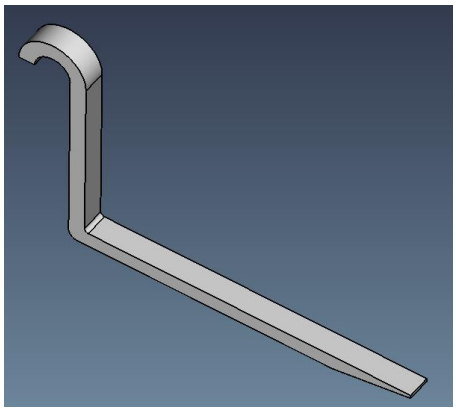

Figure 1 –The 3D model of forklift fork

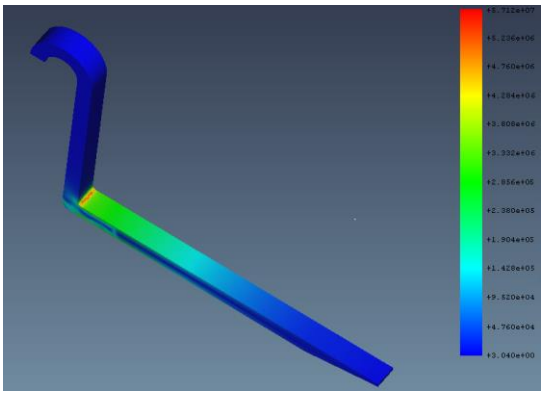

Figure 2- Von Mises Stress on Steel

When the fork is subjected to continuous load, it may cause the crack development and surface to be damaged. 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.

References

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

2. Pantazopoulos G. et al. 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.

3. Jigh B. H. G., Hosseini-Toudeshky H., Farsi M. A. Low cycle fatigue analyses of open-celled aluminum foam under compression–compression loading using experimental and microstructure finite element analysis //Journal of Alloys and Compounds. – 2019. – Т. 797. – Pp. 231-236.

4. 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.

5. Manual:What is FreeCAD [Internet resource]. – Режим доступу: https://wiki.freecad.org/FEM_Workbench .