TECHNICALUNIVERSITY OF CLUJ-NAPOCA
ACTA TECHNICA NAPOCENSIS
Series: Applied Mathematics, Mechanics, and Engineering
Vol. 65, Issue II, June, 2022

DISTORTION DETERMINATION BY FINITE ELEMENT THEORY AND COMPUTER PROGRAMM FOR THE LIFTING ARM OF THE FRONTAL LOADER

Sorin-Dragos CRUDU

Abstract: *This article is about/aims at determining the distortion values for the lifting arm of the frontal loader equipment by the finite- element method and computer programm. A general constructive model for the lifting arm is considered, with the biggest force acting on his bucket's joint . A reduced-scale model for the lifting arm of the frontal loader is presented as a case study.*

Key words: *frontal loader, lifting arm, finite element method.*

1. INTRODUCTION

.

The working equipment of the frontal loader has different structures and the existing forces are stressing them, which causes distortions valid for the design process. For an accurate design to satisfy the required conditions about minimal required distor-tions, calculations and a computerized si-mulation are recommended, using a finite- element theory and/or program. All these shorten the time to effectively manufacture such a huge component as the lifting arm and ensure its good functioning, in the working process of the frontal loader, without any damage.

.

2.THE FINITE-ELEMENT METHOD

2.1. Basics

The finite-element method is a numerical technique for finding approximated solutions to problems in science and engineering. These problems are governed by three components: differential equations, boundary conditions and geometries. The method starts by dividing the problem domain or geometry into a number of small elements. These ele-ments are connected via nodes, where the unknowns are to be determined. The finite-element equations for each element are derived from the governing differential equations describing the physics. These finite-element equations are assembled into a large set of algebraic equations. The boun-dary conditions are then imposed to the set of algebraic equations in order to get solutions at each node. The finite-element method pro-cedure generally consists of 7 steps [4]:

Step 1: Modelling. The part is modelled avoiding complicated geometrical features. This is the first and the most crucial step in any analysis. If one models the domain correctly, a lot of time can be saved. So, attention to geometry is crucial, as is understanding the reason why the simulation of this component is being done. This will provide the insights to remove insignificant features from the geometry, eventually saving some computational time and unwanted complexity. .

Step 2: Choice of material. The material properties are defined in this step. These material properties depend on the type of analysis that needs to be carried out. Literature survey for similar problems could help significantly in tackling this step. If the material you want to use in your simulation exists in the material library of your FEM tool, then your job is almost done. One just has to select and import the material properties from the library.

Step 3: Definition of loads. This step is about the definition of external forces acting on

---

Received: 29.04.22; **Similarities**: 18.05.22: **Reviewed**: 01.06./27.05.22: **Accepted**:21.06.22.

the part or the weight force by virtue of the weight of the component. One should be careful about identifying the force type, in order to avoid such problems as singularities.

Step 4: Boundary conditions. This step is mainly done to reduce the complexity of the problem from an engineering perspective. For example, in some problems, the user might know some initial conditions (like displacement of a point in the geometry) before starting an analysis. Such cases fall into the "Initial value problem" category.

Step 5: Meshing. Your geometry is divided into smaller and simpler shapes called finite elements. Any standard FEM textbook will have the information about the different types of elements and their applications. Today, many programms tend to automate this process, avoiding the trouble for the end user.

Step 6: Solution. This step happens in the backend after the definition of the simulation properties. By discretization the partial differential equations are converted into algebraic equations. Doing this helps the code to represent equations in terms of matrices. Matrices of individual elements are assembled into global matrices for the entire geometry, which is then solved by solvers for unknown variables.

Step7: Post-processing. Any FEM software will have some form of an in-dicator to show the user if the solution has been completed successfully. Once the solver gives this message, the variables (Displacement, von Mises Stress etc.) which have been calculated are presented in terms of contour plots or graphs. A proper physical understanding of the phenomenon one is simulating is very important to validate the simulation results. Cross checking the results and the approach to the solution with literature or experiments is highly recommended.

2.2.A general model of the lifting arm

The upside down Z shape (fig.2) from the MMT45 frontal loader (fig.1) [3] can be chosen as a general model of the lifting arm. Any other model (straight line arm, arm with vertical sector), is a particular cases of this general model. The general model of this arm is considered from the MMT 45 frontal loader. The function parameters are also valid for simulating the stress during working process, as the least favourable case (fig. 3) [1] & [3]. The steps are those listed above, but the virtual model will be considered a half-scale one for eventually seeing how to reduce the simulation force in order to establish a similarity with the case of real-life size virtual model.

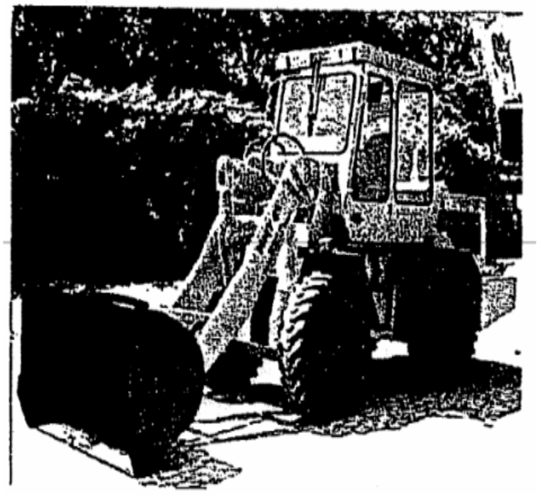

Fig.1. The MMT45 frontal loader

The technical parametersof MMT45 frontal loader, for FEM simulation-traction force are:

....3000 daN
moving speed.............................0 –22 km/h
lifting height.................................3100 mm
weight...4600 kg

The forces are acting on the bucket in the axis of the lateral extreme tooth, considering:

Rz = T, where

T is the maximal traction force in good adherence conditions, considering the slipping coefficient δ_p=0,07

The simulation stress is as shown in fig. 3 for the worst conditions $R_y = 0{,}75 * N_S$ [1] & [3], where Ns is the snatching force, determined by the stability conditions (the raising of the back wheels from the ground). The scale model of the lifting arm is as shown in the fig.4.

The next steps for FEM are to be followed as mentioned in the prior chapter, the effects showing as follows:

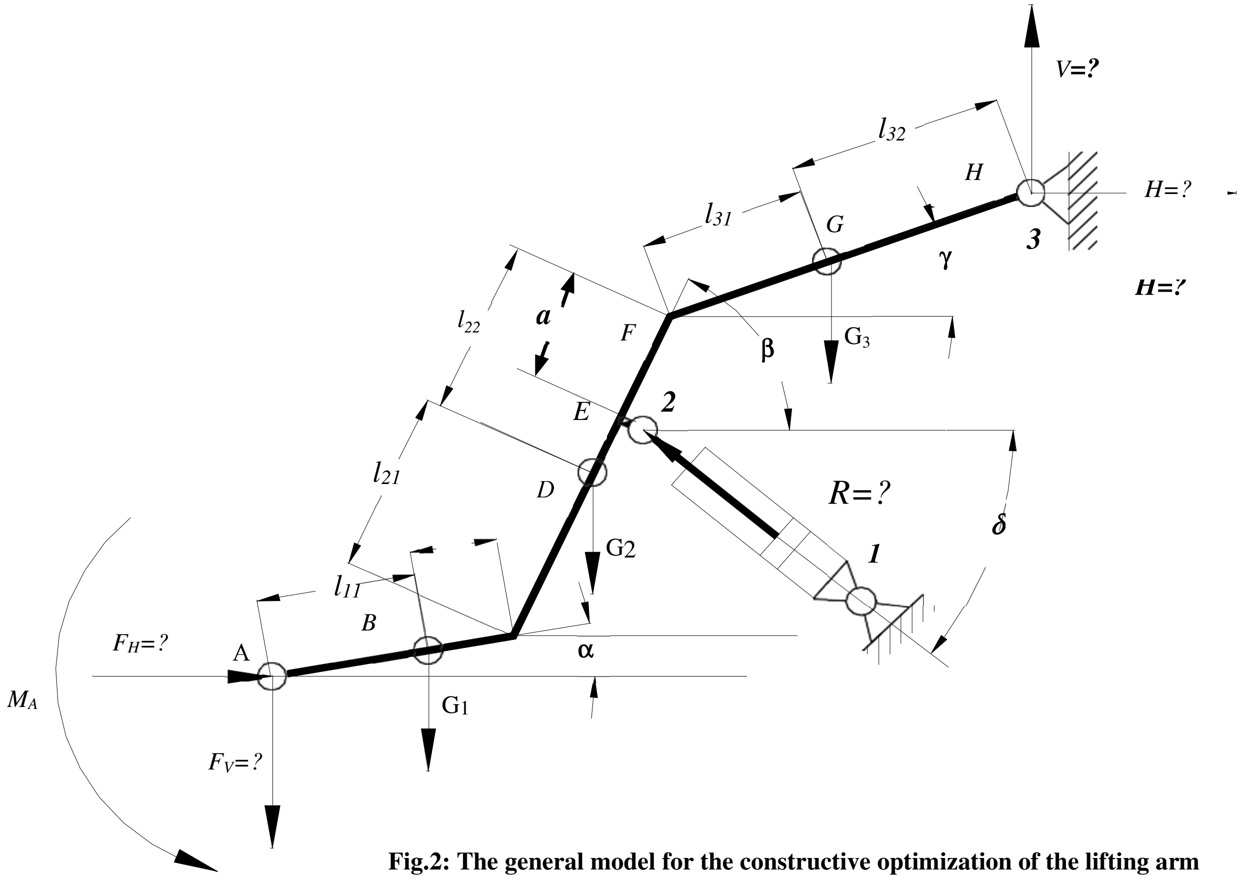

Fig.2: The general model for the constructive optimization of the lifting arm

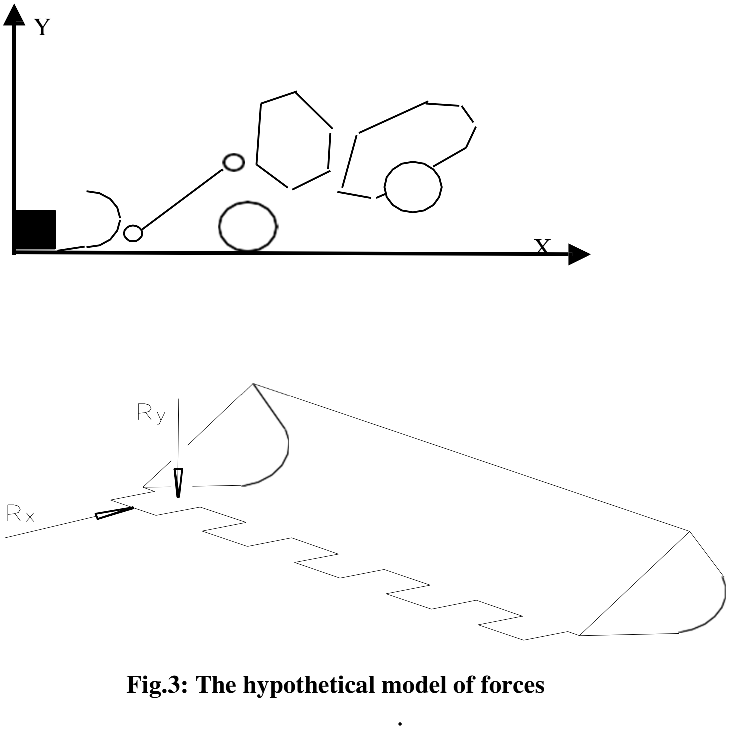

Fig.3: The hypothetical model of forces

.

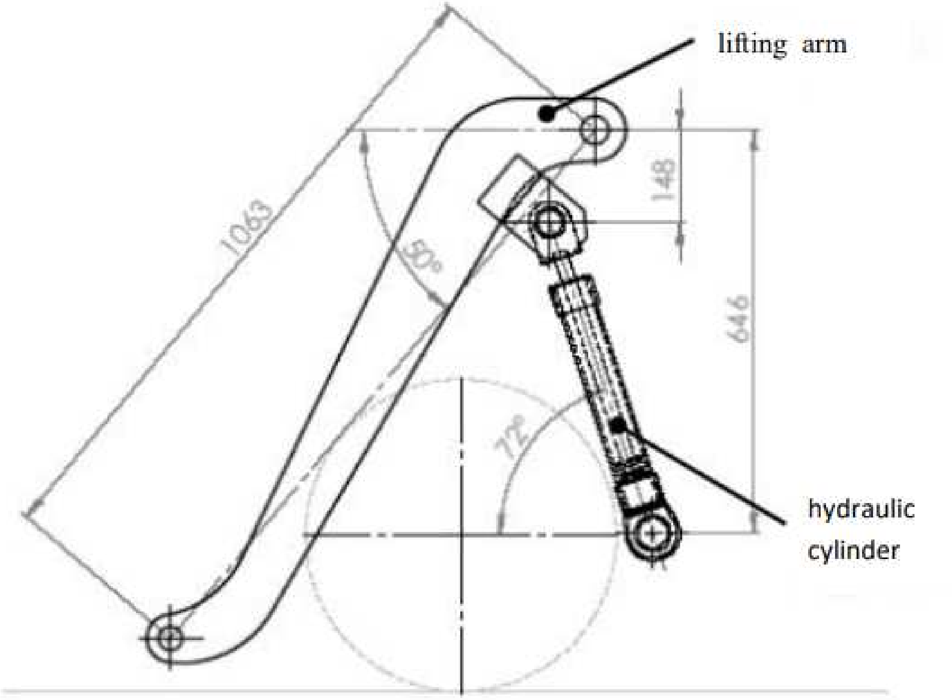

Fig.4: Scaled model for lifting arm and hydraulic cylinder

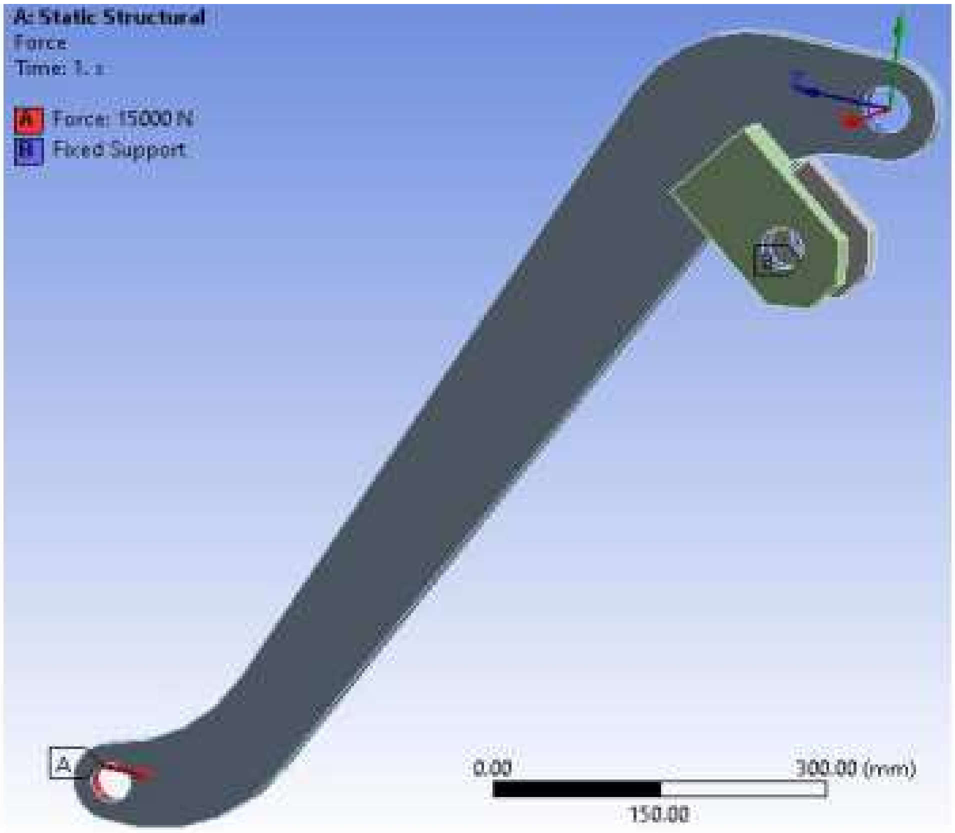

Fig.5: FEM model for applied 1500 daN Force

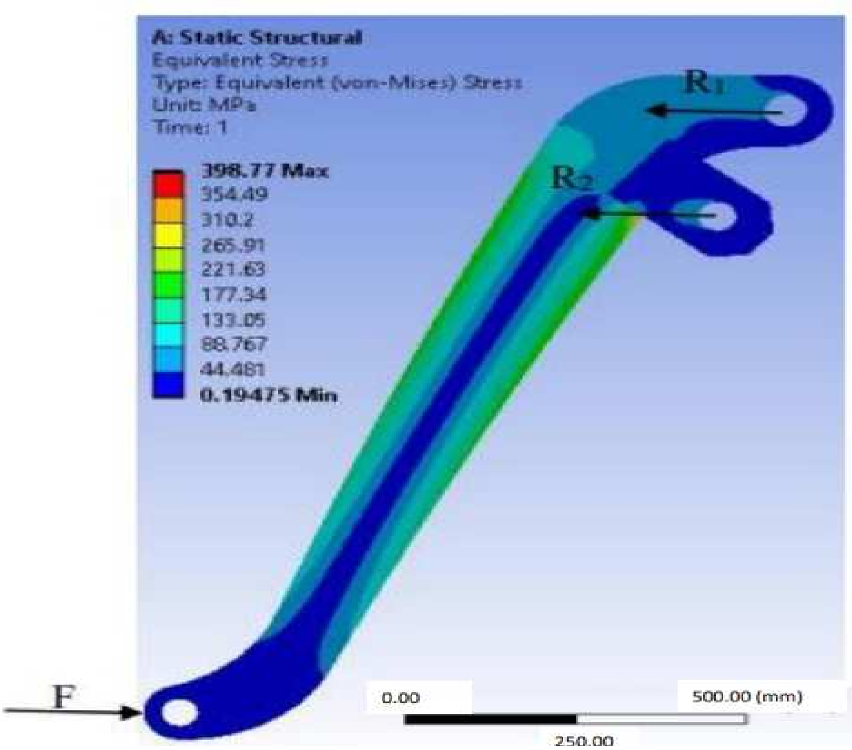

Fig.6: Stress in every point of the arm for F=1500 daN force

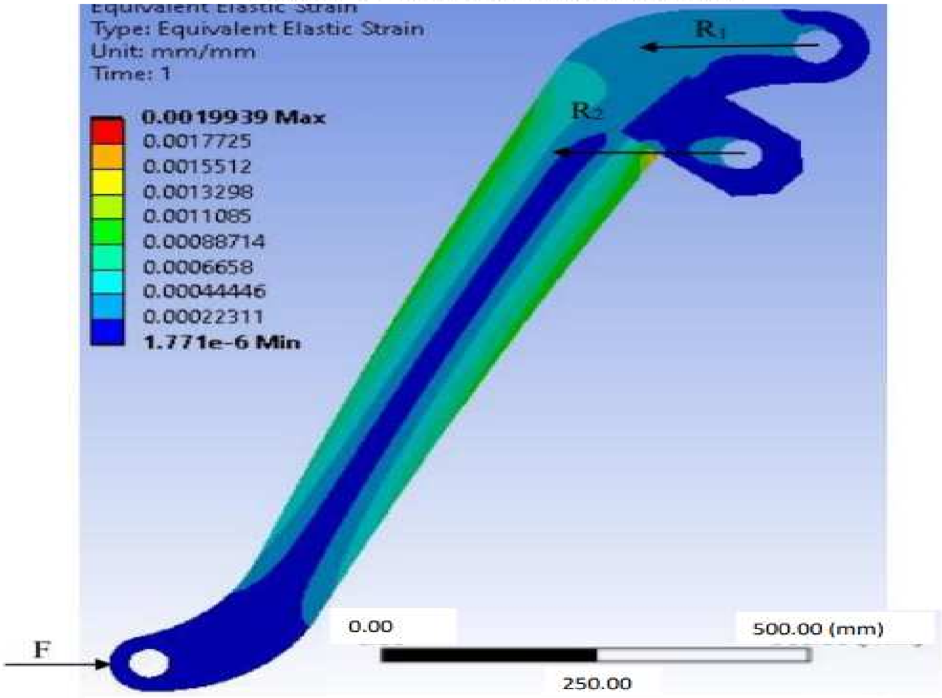

Fig.7: Distortion caused by the F=1500 daN force

The next steps for FEM are to be followed as mentioned in the prior chapter. The results will show as it follows, considering the impact force Rx=1500daN. As can be seen, the force F=1500daN produces reactions R1 and R2 and distortions in the elasticity limits. All these could be used for the constructive optimization of the lifting arm, both as shape and as weight reducing, by changing the form of the arm in order to avoiding over dimensioning. The lifting arm will be considered to work normally as long as the elasticity limit along the working cycle has not been exceeded, as in the case discussed .

3.CONCLUSIONS

The finite-element method (FEM) could be used for the constructive optimization of the working mechanism of the frontal loader. This can be easily done by getting many arm shapes of various dimensions, which have to satisfy the conditions of elasticity and by choosing the most convenient one of them. The best arm could also be the one which has the weight. This means a smaller lifting force in the cylinder and less fuel consumption (which is a long-term investment) along an entire working period.

4. REFERENCES

[1].Bratu P. and others1991. Phase I. Study for determining of geometrical and functioning parameters. Optimizing the calculation and sizing of loader type earth moving machines. INCERC Bucharest—Internal specification

[2].Bratu P. and others1991. Phase III. Methodology of resistance calculating for resistance of the working equipment and machine stability. Optimizing calculating and dimensioning for loader type earth moving machines. INCERC Bucharest—Internal specification.

[3].Crudu SD - PhD thesis- Technical University „Gheorghe Asachi” Iasi. The perfectioning of the design process for the main working equipment from the frontal loaders–Faculty of Mechanics 2010 PAG. 64-70.

[4].Pramote Dechaumphai, Sedthawat Suchariptpwaskul- Finite Element Analysis with ANSYS Workbench- AlphaScience International Ltd. Oxford, U.K..pag.3-6, 11-12

https://caendkoelsch.wordpress.com/2018/05/19/7-general-steps-in-any-femsimulation.

DETERMINAREA DEFORMATIILOR CU METODA ELEMENTULUI FINIT SI PROGRAM DE CALCULATOR, PENTRU BRAŢUL DE RIDICARE AL ÎNCĂRCĂTORULUI FRONTAL

Rezumat Articolul are ca tema determinarea valorilor deformaţiilor braţului de ridicare a echipamentului de lucru de la încărcătorul frontal, prin metoda elementului finit şi pro-gram de calculator. S-a considerat un model constructiv general pentru braţul de ridicare, cu forţa cea mai mare acţionând în punctul de prindere a cupei. Ca şi caz demonstrativ, este considerat un model redus la scară pentru braţul de lucru al încărcătorului frontal.

CRUDU Sorin-Dragos, dr. ing. ,S.C. GANETEVIC SRL, administrator, ganetevic@yahoo.com,
tel +40749705200 Home Address; Romania, Iasi, Str. V. Babes nr.8, sc.B, et.2, ap.5,
Iasi district