Structural analysis of a modulated load-bearing structure designed to investigate the interaction between soil and the working parts of agricultural machines

P Cardei, N Constantin, R Sfîru, V Muraru and S L Muraru

National Institute of Research - Development for Machines and Installations Designed to Agriculture and Food Industry - INMA, Ion Ionescu de la Brad Blv. no 6, 013813, Bucharest, Romania

E-mail: sebi_muraru@yahoo.com

Abstract. The article presents how to approach the structural analysis of a modulated load-bearing structure (MLBS) for researching the interaction between the soil and the working parts of agricultural machines. The special approach, in relation to the usual structures, is due to the fact that the supporting structure is a modulated, multifunctional one and, consequently, all the usual modular types of operation must be tested. The structure is analyzed in the hypothesis of the loads that maintain the demands of the materials used in the characteristic range of the linear elasticity, because the structure is expected to work in such conditions. The analyzed modulated structure is obtained by modular design for use in research, in several working variants. The authors also present the difficulties related to the transformations of CAD / CAM models into CAD / CAE models and opinions on the necessity of mixed analysis teams in addressing such problems.

1. Introduction

The purpose and goals of the structure theory in the broad sense is the branch of applied engineering that deals with the methods of analyzing structures of different types and uses subject to arbitrary types of external exposures. The analysis of a structure implies its investigation in terms of its strength, rigidity, stability and vibration [1].

The purpose of analyzing a structure from the point of view of its resistance is to determine the internal forces, which appear on all the components of a structure as a result of the exposure to the external loads. These internal forces produce tension; the resistance of each structure will be ensured if the stresses in each component are lower than the permissible ones [1]. Structural analysis of the supporting structure has nothing unusual, in general, especially in the case of a linear elastic analysis.

This would only be a useful exercise for students who are studying structural engineering. This article contains results about the structural analysis of a load-bearing structure for equipment, obtained through the modular design process, [2-4], consequently modulated and multifunctional. For these reasons, the structural analysis must be carried out in several variants: for each functional variant in part and for the entire structure in operation or in transport, variants of structure in transport, etc.

2. Material and method

The analysis material, whose results are presented in this article, is made up of MLBS.
The modulated load structure (MLBS), analyzed below, and consists of three main component parts (Figure 1):
- the central framework (CF);
- left side frame (wing) (RF);
- right side frame (wing) (LF);

The traction triangle for the wings is visible in Figure 2, for the left side frame, LF (similar for the right side frame RF).

The four possible working variants of the MLBS are:
- Variant of maximum working width, noted LFCFRF (working width, 5 m);
- The LFRF variant, obtained by direct coupling of the wings (elimination of the CF central frame), Figure 2 (working width, about 2 m);
- Variant central frame without wings, CF (working width, about 2.5 m);
- Simple frame variant, LF or RF (working width, approximately 1.25 m).

The complete structural analysis of the MLBS must analyze each of the four variants. Although the car has symmetry with respect to the vertical plane that contains the direction of movement of the aggregate, however the lateral components, LF and RF are not symmetrical but, asymmetrical, but apparently (only the frames). The two substructures they are not even asymmetrical due to the way of mounting the supports of the working parts for functional reasons.

**Figure 1.** Modular load-bearing structure with supports of the working bodies mounted for the distance between rows of 100 mm
Figure 2. The LF component of the MLBS with its own triangle of connection to the traction source

The method of approaching the pre-execution structural analysis of the bearing structure of the MLBS is done according to a methodology drawn up from previous studies of the authors. The stages of this methodology are the following:

- **ASS1** - construction of the CAD / CAM model of the structure;
- **ASS2** - transforming the CAD / CAM model into the CAD / CAE model (eliminating distances or gaps and interferences);
- **ASS3** - construction of structural models (geometry derived from the CAE model together with meshing, defining contacts with the environment or the supports or conditions at the border, including any initial conditions). Usually the definition of materials by components is made from the ASS1 stage, but taking into account the results of the ASS4 stage, some materials can be changed.
- **ASS4** - Structural analysis or calculation and storage and selection of the results necessary for the analysis.
- **ASS5** - Convergence analysis\(^1\), [5-7], if applicable (when dealing with coarse meshing);
- **ASS6** - Analysis of the behaviour of the structure (e.g. resistance checking, safety factor, etc.) and decision making. If the structure meets the requirements, it may decide to move to the execution stage. Otherwise, changes in geometry, support, loading or material are made and the loop described by this methodology is repeated. The loops are repeated until one reaches the conclusion that the structure meets the requirements.

The steps ASS2, ASS3, ASS4, ASS5, have been elaborated and described exactly in [7]. The steps ASS1, ASS2, ASS3 are clearly explained in a huge literature, [8-11], etc. The ASS4 stage is a step that fully respects the methodology required by the Solid Works Simulation calculation and

\(^1\) The convergence analysis was not considered necessary for this stage of post-conception and pre-execution structural analysis. In addition, at this stage of conception and design, the validation cannot be done, because the physical structure, with which the experiments should be performed, is missing. More than that, for all models with solid type (tetrahedral) finite elements, we worked with the finest discretization on which the SolidWorks Simulation programme is set or even finer.
simulation program and is a generally accepted methodology in the field of structural analysis, [12-18], etc. The ASS6 stage is found in any treatise on structural mechanics, material resistance or engineering design. In general, in order to carry out the activities included in this methodology, it is recommended to consult the works [19-24], etc. All interpretations respect the general principles of the materials strength.

3. Results

From the analyzes carried out, results of the static analysis for the lateral frames and the central frame, as well as for the entire MLBS load structure will be presented. The results of calculating the frequencies of the whole MLBS structure in a suspended position when returning in turn are also presented.

3.1. The results of the structural analysis for the RF and LF side frames

Because we performed only the linear-elastic static analysis of the two structures, the important characteristics of the material are reduced to the linear elasticity module $2.1 \times 10^{11}$ N/m$^2$, the shear modulus $7.7 \times 10^{10}$ N/m$^2$, Poisson's ratio 0.29, mass density 7900 kg/m$^3$, flow limit stress (laminating) $3.51571 \times 10^8$ N/m$^2$, breaking stress $4.20507 \times 10^8$ N/m$^2$ (AISI 1020 steel), for all the bars of the RF and LF components of the MLBS, except for the bar-type bars from the main frame, whose material, also of steel type (S355JR), has the following characteristics: linear elasticity module $2.1 \times 10^{11}$ N/m$^2$, shear modulus $7.9 \times 10^{10}$ N/m$^2$, Poisson's ratio 0.28, mass density 7800 kg/m$^3$, flow limit stress (plasticization) $2.75 \times 10^8$ N/m$^2$, break stress $4.5 \times 10^8$ N/m$^2$.

Both side frames were rigidly fixed to the holes of the grip elements at the central frame. The loads were made by means of 130 N forces on each support of the working members in the direction of the final plate on which the working body is attached (chisel type in this case), contrary to the forward direction of the aggregate (Figure 12). Taking into account that the RF and LF components each have 12 supports, the total force applied is 1560 N at RF and 1430 at LF. In the case of the LF structure, on one of the 12 supports of the working parts, the force of resistance to deformation of the soil was not applied (Figure 6, a stress-free support). The reactions that appear in the grip holes in the central frame are given in Table 1.

**Table 1. Reactions for the structural models of the RF and LF components**

|        | forward direction, N | vertical, N  | lateral, N  | Total, N     |
|--------|----------------------|--------------|-------------|--------------|
| RF     | -1103.08             | -1103.04     | 0.00361061  | 1559.96      |
| LF     | 1011.21              | 1011.14      | -0.01084710 | 1430.02      |

Signs differ in reactions because the reference systems in which they were worked were differently oriented, separately for the two structures. It is noted that the reactions are almost equally distributed vertically and horizontally. Vertically, the reaction tends to lift the working parts out of the furrow.
The number of nodes, finite elements and the maximum size of the finite element, for the LF and RF components, are given in Table 2.

**Table 2.** Data on the meshing of the structural models of the RF and LF components of the MLBS

|       | Number of nodes | Number of finite elements | The maximum size of the finite elements, mm |
|-------|----------------|---------------------------|-------------------------------------------|
| **RF** | 2246248        | 1315470                   | 4.94441                                   |
| **LF** | 2245789        | 1314669                   | 4.94759                                   |
Figure 5. MLBS Map of equivalent stress (von Mises) in the RF structure

Figure 6. MLBS Map of equivalent stress (von Mises) in the LF structure
3.2. *The results of the structural analysis for the central CF framework*

For the central CF frame of the MLBS, for the linear-elastic static analysis, a steel type material, coded S275JR, has been chosen, with the following characteristics: the linear elasticity module $2.1 \cdot 10^{11}$ N/m$^2$, the shear module $7.9 \cdot 10^{10}$ N/m$^2$, Poisson's ratio 0.28, mass density 7800 kg/m$^3$, flow limit stress (yield stress) $2.75 \cdot 10^8$ N/m$^2$, breakdown stress $4.1 \cdot 10^8$ N/m$^2$. The geometry of the structure can be seen in Figure 1 and in the following Figures, containing the maps of characteristic request fields.

The upload was done similar to the uploads made to the RF and LF components of the MLBS. The boundary conditions are set by rigid fixing in the coupling holes to the tractor located on the triangle connecting to the tractor (Figure 8).

![Figure 7. MLBS Map of equivalent stress in a section of the structure](image)

![Figure 8. MLBS The supporting locations of the structural model of the CF](image)

The structural model of the CF was meshing with SOLID type elements, resulting in 1811739 nodes and 980310 finite elements, with a maximum size of 6.88555 mm. Under these conditions the horizontal and vertical reactions are -2297.98 N, respectively -2298.13 N.
Figure 9. MLBS Distribution of the resulting relative displacement field on the boundary of the CF structural model, on its deformed form

Figure 10. MLBS Map of equivalent stress (von Mises) on the border of the CF structural model
Figure 11. MLBS The safety factor in the CF structural model with the signalling on this map of the areas of false stress concentrators

The field of the equivalent stress in the CF structure have maximum values with a higher order than the acceptable ones. But the locations where the maximum values are reached are invisible on the map of the equivalent stress distribution throughout the structure. The maximum value visible on the map of the equivalent stress is 120 MPa, which ensures a safety coefficient above value 2 (the recommendations for such machines are around 1.8). A detailed analysis of the equivalent stress state in the CF structure, shows that high stress, over 120 MPa, are located on very small areas and in locations where the distancing have not been totally reduced (for functional reasons, for installation). However, the global contact command ignored (by setting) the distance sufficiently small and thus the structural model was accepted for calculation. Instead, have resulted, what analysts call false stress concentrations, which can be seen in successive details (Figure 10) or on the safety factor map (Figure 11). False results can be obtained also due to the exaggeratedly small meshing with elements of maximum size or in the vicinity of the sharp changes of shape and section without connections, [25], [26-28] and [23]. An article relevant to the occurrence of false stress concentrations is [29]. For analysts with FEA experience, the stress concentrations in the CF structural model are clearly generated by the distancing and sections changes of the CAD model, with the geometry not fully adapted to the CAE requirements.

Obviously, it is absurd to ask for the faithful representation of the welding, which finally makes the definitive connection between elements of the structure. An FEA analysis with such elements should be aware of the mechanical properties of welding and draws attention to the fact that there is often a non-negligible area around welding, for which the material of the welded elements undergoes some modifications, which would greatly complicates the model. Therefore, due to these considerations, the analysis at this stage does not resume on a CAE model which, then, would be even farther from reality by bonding the elements of the CAD model. Problems and solutions related to eliminating interferences and gaps in CAD models can be found in [30] and [31].

As can be seen from Figure 11, since the CF component is the central component in the MLBS, it is expected that the problem of the false stress concentrations will be transmitted also when analyzing the structural MLBS model. The confirmation comes in the results presented in Figures 15 and 16. In Figure 16 are very well highlighted the areas of false stress concentrations.
3.3. The results of the structural analysis for the entire MLBS structure

The results of the static-linear structural analysis of the complete working variant of the MLBS load structure are given in this chapter in addition to the related comments. The geometry of the structural model can be seen in Figure 1. The loading of this structure is similar to that of the RF, LF and CF structures. In this version there are 49 working parts. The support (the boundary conditions or the contact with the external environment) are the same as in the case of the CF structural model, simulating the connection (coupling) to the tractor. The loading and support of the structure are outlined in Figure 12. The material of the structure, (for static-linear analysis, abstraction making the calculation of the safety factor, common characteristics of many steels are sufficient: the modulus of linear elasticity, the shear modulus, Poisson's coefficient, the mass density being necessary for the calculation of its frequencies) it has the same characteristics as for each component.

![Figure 12. Border conditions (A) and force loading (B and detail C) of the structural model of the MLBS](image)

Obviously, the structure is meshing with finite elements of type SOLID, in the number of 3083966, having a shoulder of 5464072 nodes. The maximum size of an element is 6 mm. In these conditions (total force applied $130 \times 49 = 6370$ N) the reactions provided by the calculus program are: on the displacement direction, $-4503.86$ N, on the vertical, $-4503.66$ N, laterally, 0.155 N.

![Figure 13. Distribution of the resulting relative displacement field at the border of the structural model of the MLBS](image)
Figure 14. Map of the vertical component of the relative displacement on the border of the MLBS model

Figure 15. Map of equivalent stress distribution at the border of the structural model of the MLBS

Figure 16. MLBS - Map of equivalent stress distribution in the structure in the area of false stress concentrators
4. Own frequencies. Frequency analysis of the MLBS

The structural model of the MLBS used to estimate the state of deformation and stress in the structure, generated by a request that approximates the load in the field, is used to calculate the frequencies of the structure and its own modes of vibration. The only difference with the model used to simulate a working regime is that the force loads have been cancelled, because the linear analysis program does not calculate its own frequencies for the force loaded structure. Under these conditions, the model simulates the oscillatory behaviour of the MLBS suspended in the tractor clamping system during, for example, the return at the end of the plot. A similar analysis is made for the transport configuration, different from the working configuration, because the lateral frames, for the transport, are being tight. In the transport configuration, the own frequencies are more important than in the case presented in this article, because in transport there are different frequencies of different values depending on the running path and the speed of movement of the aggregate. Among the frequencies that can be generated by the interaction between the aggregate and the rolling path, in transport, it is very likely that some to be very close or even equal ones (the continuous variation of the speed leads to the continuous variation of the spectrum of the excitation frequencies) with some frequencies typical of the MLBS structure and in consequently, can appear vibrations in resonance regime. The resonance may affect the integrity of the structure, but it may also cause stability losses to the moving aggregate (loss of adhesion, traction, steering).

![Figure 17. MLBS - Deformed shape for vibration on its own fundamental frequency: 3.29 Hz](image1)

![Figure 18. MLBS- Deformed shape for vibration on its own second fundamental frequency: 4.39 Hz](image2)

![Figure 19. MLBS - Deformed shape for vibration on its own third fundamental frequency: 4.80 Hz](image3)

![Figure 20. MLBS - Deformed shape for vibration on its own fourth fundamental frequency: 8.96 Hz](image4)
The deformed forms of the vibrating structure on its own first four frequencies are graphically represented in Figures 17, 18, 19 and 20. The participation of the normalized masses in the vibration on the first five modes is given in Table 3.

**Table 3. Participation of normalized masses in vibration on the first five own modes**

| Mode number | Frequency (Hertz) | X direction (lateral) | Y direction (vertical) | Z direction (longitudinal) |
|-------------|-------------------|-----------------------|------------------------|---------------------------|
| 1           | 3.2938            | 1.8349e-06            | 0.73024                | 1.7606e-05                |
| 2           | 4.3912            | 0.017701              | 5.1164e-07             | 1.159e-08                 |
| 3           | 4.8058            | 0.31499               | 2.7552e-06             | 1.9438e-07                |
| 4           | 8.9586            | 3.0502e-06            | 0.065222               | 0.10568                   |
| 5           | 9.8129            | 1.302e-05             | 0.0063616              | 0.76791                   |

Sum X = 0.33271  Sum Y = 0.80182  Sum Z = 0.87361

5. Discussions

The structural analysis presented in this paper is an example of complex structural analysis. The complexity comes first from the complexity of the analyzed structure, from the modular and multifunctional feature. Due to the multifunctionality, each modular component of the MLBS structure is analyzed separately by simulation in working regime. Each module, separately, is the subject of a structural analysis.

Multimodular variants of operation also. If we delve a little deeper into the problem, we find that the real analysis is much more complicated and again due to the modular and multifunctional character. We refer to the fact that between the modules there are interactions. The modules are linked together by connecting elements that have displacements and certain freedom of movement. As a result, in order to carry out a structural analysis of depth, the total rigid contact must be dropped and the joints between modules and between the tractor and the structure or substructure corresponding to the variant in which it works introduced. This leads to a contact problem and more precisely to a problem with many contacts. Contact problems are particularly difficult and require complicated hypotheses of an experimental nature. Therefore, such analyzes are not possible in the pre-execution design stage. In this complex framework, the problem of vibrations and possible dynamic simulation is greatly complicated.

These complications appear to be unimportant. In fact, these delicate aspects do not necessarily imply the failure of structures by irreversible deformations or material breaks. The first disruptive phenomena that occur in such structures, according to the results of the structural analysis, refer to quality losses for the soil work performed. The large amplitude of the structure can have negative effects on the unevenness of the working depth and even on the individual working width of each organ.

Structural analysis shows that there are uneven vertical displacements of the structure. In some working variants (for example, when working separately with one side frame), the relative vertical displacements are tolerable (below 10% of the working depth). When working in modulated variants, however, the relative displacements (and implicitly the unevenness of the working depth), are generally no longer tolerable (up to 20-30% of the required working depth).
6. Conclusions

The structural analysis of complex structures in the pre-execution stage is very useful through the simulations that it can perform on possible working regimes. Through simulations, it is possible to identify a more complete set of parameters of the problem, of some phenomena that are intuitively difficult to anticipate, of consequences and more difficult to predict at a stage when the structure was not physically constructed, therefore neither subject to experiences.

The identification of the weaknesses of the structure before the start of the execution or even after, allows to make some improvements and to correct some errors or design defects. For example, identifying possible unevennesses of the working depth allows taking measures of balancing the edges of the structure away from the triangle of binding to the tractor.

From a technical point of view, one of the most important stages of carrying out the structural analysis of the described type is the transition from the CAD / CAM model of the structure to the structural model, CAD / CAE, on which the usual programs of structural analysis accept them for analysis. The main stage was the elimination of interferences and distances. If the interference is due to tolerance settings of the drawing program or to small human errors, the distances are either functional (displacements) or necessary in the assembly process. The CAD / CAE model, fixes the elements of an assembly in contact and eliminates many small distances between parts, distances with functional role. Thus, in general, the CAD / CAE model is a false model, which cannot actually exist, or if it can still exist, it is not functional.

We have considered scientifically interesting to show results of a CAE model that presents some (few) errors, generating false stress concentrations since the comments on this phenomenon can lead to the generalization of the analyst's experiences in the field of FEA and to useful discussions for improving their activity.

In conclusion, we emphasize the importance and complexity of the structural analysis of a structural model for a complex structure obtained through modular, multifunctional conception and design. Such a structural analysis is performed at least for each functional configuration, separately and, of course, for the complete assembly. It is recommended a structural analysis in the pre-execution phase, able to provide structural models that the simulation of probable situations can lead to noticing new aspects and problems, to solving some or improving them.

Acknowledgement

This work was supported by a grant of the Romanian Education Research Ministry (RRIM), through Programme 1 – Development of the national research-development system, subprogram 1.2 – Institutional performance – Projects for financing excellence in RDI, contract no. 16PF, and was done by “NUCLEU” Programme, developed with the support of the RRIM, project PN19 10 01 02.

References

[1] Karnovsky I A and Lebed O 2010 Advanced Methods of Structural Analysis, Springer
[2] Herlitzius T, Grosa, Henke M and Krzywinski J 2013 Concept Study of a Modular and Scalable Self-propelled Implement System – Limitations of current Tillage and Seeding Tractor – Implement Systems and Vision of Future Technologies, Agricultural Engineering 37-46
[3] Jeevanandam S and Mohan Rao S L 2015 Modularity Techniques in Commercial Vehicle, International Journal of Engineering Research & Technology 4
[4] Chichernea F 2013 Proiectarea modulara a utilajelor industriale, Buletinul AGIR 2
[5] ***Cardei P 2017 Considerations on the convergence of structural analysis results, performed by finite element method, https://www.researchgate.net/publication/312595676_Considerations_on_the_convergence_of_structural_analysis_results_performed_by_finite_element_method
[6] ***Cardei P 2017 The Convergence analysis of the mathematical models of a phenomenon in the structural analysis, https://www.researchgate.net/publication/314113468_The_Convergence_analysis_of_the_m
Mathematical models of a phenomenon in the structural analysis

[7] ***PN 16 24 01 03 – Cercetari avansate informatice si digitale de conceptie si dezvoltare in scopul eficientizarii sistemelor tehnologice inteligente pentru lucrarii agricole, Contractul Nr. 8N/09.03.2016, act aditional nr. 1 / 2016, https://oldsite.inma.ro/Pagina_web_NUCLEU/NUCLEU_nou/PN_16_24_01_03/cercetari_a_vansate_informatice_digitale.htm

[8] Lee K 1999 Principles of CAD/CAM/CAE Systems, Pearson

[9] Rao P N 2002 CAD/CAM Principles and Applications, Second Edition, Tata McGraw-Hill Publishing Company Limited, New Delhi

[10] Zeid I and Sivasubramanian R 2009 CAD/CAM Theory and Practice, Tata McGraw Hill

[11] Coticchia M E, Crawford G W and Preston E J 1993 CAD/CAM/CAE SYSTEMS, Taylor & Francis

[12] Bathe K J 2006 Finite Element Procedures, Prentice Hall, Pearson Education, Inc.

[13] Seshu P 2012 Textbook of Finite Element Analysis, PHI Learning Private Limited, New Delhi

[14] Zienkiewicz O C, Taylor R L and Fox D D 2014 The Finite Element Method For Solid & Structural Mechanics, Elsevier

[15] Maksay S I and Bistrian D A 2008 Introducere in metoda elementelor finite, Editura CERMI, Iasi

[16] Lates M T 2008 Metoda elementelor finite, Editura Universitatii Transilvania Brasov

[17] Posea N, Iordache S, Predescu L and Costache I 2003 Metoda Elementului Finit pentru structuri de rezistenta, Editura Bibliotheca, Targoviste

[18] Blumenfeld M 1995 Introducere in metoda elementelor finite, Editura Tehnica, Bucuresti

[19] Cardei P, Constantin N, Grădinaru V, Marin E, Manea D, Matache M, Muraru V, Muraru C, Pirină I, Sfiru R, Sorica C, Stanciu L and Vladut V 2012 Analiza structurală și materiale noi focalizate pe mecanică, mechatronică, menențană și exploatarea echipamentelor tehnice pentru agricolă și industrii alimentară, Editura Terra Nostra, Iasi

[20] Buzdugan Gh 1964 Rezistenta materialelor, Editura Didactica si Pedagogica

[21] Deutsch I 1979 Rezistenta materialelor, Editura Didactica si Pedagogica

[22] Bansal R K 2010 A text book of Strength of materials, Laxmi Publications (P) LTD

[23] Cardei P, Constantin N, Vladut V and all 2012 Structural analysis and materials focused on mechanics, mechatronics, maintenance and operation of machinery for agriculture and food industry, Terra Nostra, Iasi

[24] Cardei P, Muraru V, Constantin N, Muraru C, Cilan T, Hodre C D and Matache M G 2015 Mathematical modelling and comparative simulation of the vibrations of vibro-cultivators and agricultural cultivators, INMATEH Agricultural Engineering 47(3)

[25] Rades M 2006 Analiza cu elemente finite, UPB

[26] Akin J E 2010 Finite Element Analysis Concepts Via SolidWorks, World Scientific

[27] Murakami Y 2017 Theory of Elasticity and Stress Concentration, Wiley

[28] Deshpande S S, Desai P N, Pandey K P and Pangarkar N P 2016 Detailed Studies on Stress Concentration by Classical and Finite Element Analysis, International Journal of Mechanical Engineering and Technology (IJMET) 7(2) 148-167

[29] Kurowski P 2002 When Good Engineers Deliver Bad FEA https://www.machinedesign.com/3d-printing-cad/fea-and-simulation/article/21831586/when-good-engineers-deliver-bad-fea

[30] Sfiru R, Muraru V, Cardei P and Ticu T Conversion of CAD Models to CAE Models https://oldsite.inma.ro/Pagina_web_NUCLEU/NUCLEU_nou/PN_16_24_01_03/Prezentare_MV.pdf

[31] Muraru V, Cardei P, Muraru S and Sfiru R 2016 Using CAD Models in Finite Element Analysis, International Multidisciplinary Scientific GeoConference SGEM