COMETa 2022
6th INTERNATIONAL SCIENTIFIC CONFERENCE
17th - 19th November 2022
Jahorina, B&H, Republic of Srpska
University of East Sarajevo
Faculty of Mechanical Engineering
Conference on Mechanical Engineering Technologies and Applications

IMPROVED PROCEDURE FOR NUMERICAL ANALYSIS OF VEHICLE TRANSPORT PLATFORM

Milan Bojović[1], Miloš Pešić[2], Nikola Jović[3], Aleksandar Bodić[4], Vladimir Milovanović[5]

Abstract: The procedure for numerical analysis of platform for vehicles transport is presented in this paper. The structural analysis is performed using a finite element method. Model is prepared in software Femap and calculation performed in NX Nastran. Platform structural analysis requires consideration of various loading cases for different vehicles. As a result, it is necessary to assign a large number of loads manually, which takes a lot of engineering time. An API (Application Programming Interface) script has been created to automatize load assignment. Based on the starting position on the platform, vehicle wheelbase and wheels tread surface the API calculates pressure values and creates load sets. In this way, multiple load sets that simulate different vehicle positions on the platform are created automatically. Based on the presented procedure, it can be concluded that significant saving of engineering time is achieved.

Key words: API, FEM Analysis, Vehicle Transport Platform

1 INTRODUCTION

The needs of the human community for the transportation of people and goods with an increasing number of motor vehicles in the world require an increase in the safety of the vehicles themselves, as well as a reduction in the toxic emission of exhaust gases. For this reason, there is a trend toward greater use of vehicles with electric drive. In order to satisfy the market demand for these vehicles, it is necessary to transport them. The aim of this paper is to examine whether the existing platforms for transporting vehicles with conventional drive can transport significantly heavier vehicles with electric

[1] Milan Bojović, Faculty of Engineering University of Kragujevac, Kragujevac, Serbia, milan.bojovic@uni.kg.ac.rs
[2] Miloš Pešić, Institute for Information Technologies University of Kragujevac, Kragujevac, Serbia, milospesic@uni.kg.ac.rs
[3] Nikola Jović, Faculty of Engineering University of Kragujevac, Kragujevac, Serbia, njovic1995@gmail.com
[4] Aleksandar Bodić, Faculty of Engineering University of Kragujevac, Kragujevac, Serbia, abodic@uni.kg.ac.rs
[5] PhD Vladimir Milovanović, Faculty of Engineering University of Kragujevac, Kragujevac, Serbia, vladicka@kg.ac.rs

drive.

Numerical analysis of a vehicle transport platform requires consideration of different load cases for different transporting vehicles. For this reason, it is necessary to set manually a large number of loads. This way of assigning loads requires a lot of engineering time, so it is essential to automatize the load assignment process. For this reason, an API script is created. API has a wide application in numerical analysis. Some of the application examples are: for setting the load caused by the wind [1], the API program has been written for the calculation of the load from bulk materials onto the sides and the bottom of the wagon [2]. In paper [3], API was used for connecting FEMAP with MS Word, and generating images for the analysis report. In this paper, API was used for assigning loads. API takes into account the vehicle's initial position on the platform, the wheelbase, and the wheel tread as input parameters. Based on these input parameters, API assigns loads on the platform and creates multiple load sets.

In this paper, the procedure of numerical analysis of the vehicle transport platform using the finite element method is presented. The model is created in the pre- and post-processing software Femap [4]. Using the created API script, different load cases corresponding to vehicle positions on the platform were specified. The calculation is performed in NX Nastran software.

In the second chapter, the basics of the API and its application within the Femap software, are briefly explained.

In the third chapter, the FEM model, loads, and boundary conditions are presented. Also, a detailed description of the API script for assigning loads is given.

At the end of the paper, a presentation and discussion of the obtained results are given.

2 APPLICATION PROGRAMMING INTERFACE (API)

An Application Programming Interface (API) represents user interface to implemented functionality that programmers can use to perform various operations. Using an API, the data or services of a software application can be exposed as a set of predefined resources, such as methods, objects, or URIs. In this way, data or services can be accessed by other applications without the need to implement basic objects and procedures. Today, APIs are at the center of many modern software architectures [5, 6].

The FEMAP API is an object-oriented system. This means that all data is represented as discrete objects that the user and other objects can interact with. There are two levels of objects in the API: the application object, and other objects. The first level includes one object, which is the Femap application object. For each application that is written, it is necessary to define and reference a Femap application object that defines the connection between the application and the Femap software. The application object includes data and functionality of a global nature. Application object properties include all global settings of the Femap session, while almost all functions provided by the Femap software menus are available as methods within application object [7].

Detailed description about the API script for load assignment on the vehicle transport platform is given in chapter 3.2.

3 NUMERICAL ANALYSIS OF VEHICLE TRANSPORT PLATFORM

3.1 Finite element model

FE model of vehicle transport platform (Figure 1) is created in Femap software using 2D plate 4-noded elements and 3D hexahedral 8-noded elements. Due to the symmetry of the vehicle transport platform model, only one half is created. FE model consists of 81633 finite elements and 94943 nodes.

Since only half of the platform is modelled, symmetry boundary conditions are set in the model symmetry plane. Loads that are set on model are self-weight and the load from the vehicles. Loads of transported vehicle are assigned using an API script. This script is created so that the pressures are set on the platform based on the vehicle wheelbase, wheel tread area and the initial position of the vehicle on the platform. In this way, it is possible to simulate the transport of different types of vehicles (different wheelbase), as well as different vehicle positions on the platform.

Figure 1. *FE model of vehicle transport platform*

3.2 API

To automatize load assignment, an API (Application Programming Interface) script has been created. The algorithm based on which the API was developed is shown in Figure 2.

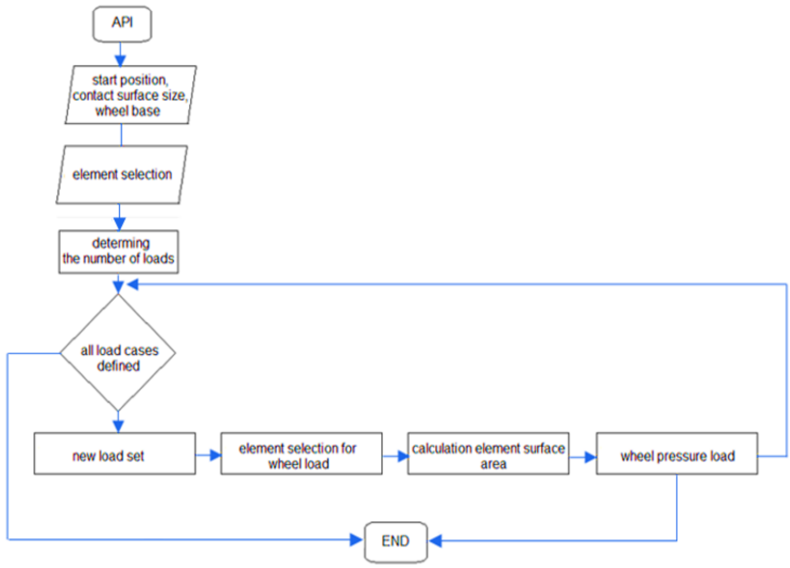

Figure 2. *API algoritam for load definition*

Defining input data is the first step (dimensions of tread surface, initial position, wheelbase, load). Second step is the selection of all tread elements which is done through face element selection. In the first two steps, initial data is entered manually by the user. Based on the entered initial data by user, the API determines the center of mass of all selected elements. After that, API determines the number of possible vehicle positions on the platform based on wheel tread surface and selected elements (step 2). Next step is creation of new load sets, and setting the self-weight of the platform in all load sets. After that, based on the center of mass of the elements (step 3) and the wheel tread surface dimensions, the elements included in the tread surface for each load case are defined. The area of the elements that define the tread surface is calculated in the next step. Finally, based on the calculated element area and the force exerted by the wheel on the platform, the pressure value is calculated and set in the appropriate load set.

3.3 Load position description of the platform

Vertical loads on the platform come from the wheels (weight of 0.50 t per wheel is multiplied by a dynamic factor of 1.30 and equals 0.65 t per wheel) and the load from each wheel is given as the pressure on the load-carrying surface (cca140 x 160 mm). The initial distance between wheel axles is determined in order to define load cases. The scheme of the position of the car with the distance between wheel axles is shown in Fig. 3.

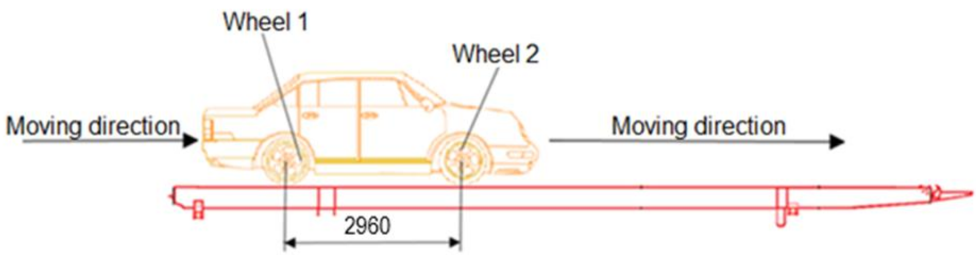

Figure 3. *Scheme of position of the cars*

Wheels tread surface dimensions are 420x15040 mm and it is divided into three strips: A (red color), B (blue color), and C (green color) with dimensions cca140x15040 mm each. Strips A, B, and C are divided into 76 positions in the direction of the longitudinal axis. For each strip (A, B, and C), 76 load cases are defined. The total number of load cases is 228. Fig. 4 shows the load sets created using API for different vehicle positions for all three strips.

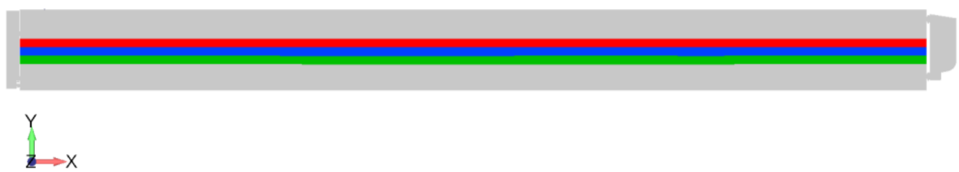

Figure 4. *Load assignment using API*

With this procedure, it is possible to significantly save engineering time. Otherwise, a large number of load sets would be assigned manually by the user.

3.4 Results

MultiSet linear analysis is performed in NX Nastran software. Each calculation set corresponds to a different vehicle position on the platform. **Error! Reference source not found.** show maximal vertical displacements of the central node on the platform for all load sets.

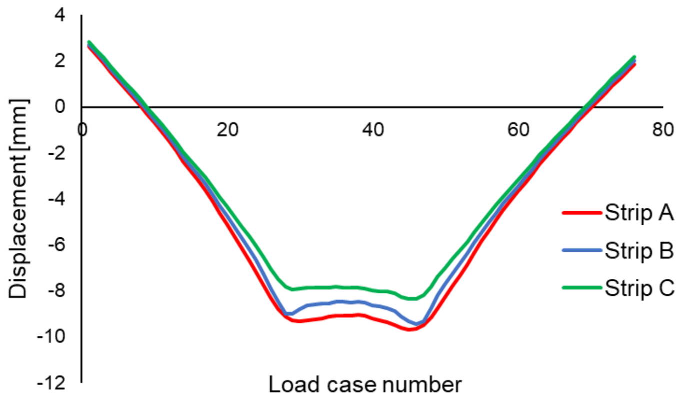

Figure 5. *Vertical displacement of the central node*

Based on the results from the Fig. 5, it can be concluded that the maximal vertical displacement of the central node on the platform is dependent on the wheel position. When the vehicle arrives at the platform, the vertical displacement of the central node of the platform is the smallest. As the vehicle position moves toward the center of the platform, the vertical displacement of the center node increases. The highest value of vertical displacement is when the vehicle is in the middle of the platform, which corresponds to load case 45 in strip A. The value of maximal vertical displacement for load case 45 in strip A is 9.68 mm.

The most unfavourable load case for the entire platform is load case 71 in strip A. The maximum value of von Mises equivalent stress for presented load case is 168.48 MPa.

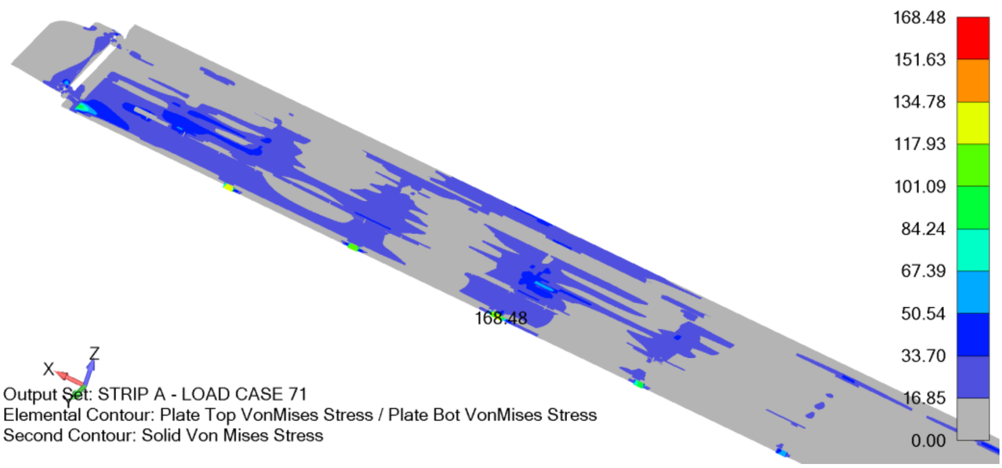

Figure 6. *Von Mises equivalent stress field*

4 CONCLUSION

The procedure for the numerical analysis of the vehicle transport platform using the finite element method is given in this paper. The presented analysis of the platform was carried out in order to examine the possibility of transporting vehicles with electric drive. The original purpose of the platform was to transport vehicles with a conventional drive.

In order to avoid manually defining a large number of load cases, an API script was created for assigning loads. Based on the vehicle's initial position, the wheelbase and the wheel tread API assign loads on the platform and create multiple load sets.

The finite element model of the vehicle transport platform is created in Femap software. MultiStep linear analysis is performed in NX Nastran software. Each calculation step corresponds to a different position of the vehicle on the platform. Results are shown in form of vertical displacement for all load cases of the platform and the maximum value of von Mises equivalent stress for the most unfavourable load case. The presented results show that significantly loaded zones on the platform correspond to vehicle position for the considered load case. Based on the obtained results, it can be concluded that the procedure given in this paper for the numerical analysis of the vehicle transport platform is effective.

ACKNOWLEDGEMENT

This research is partly supported by the Ministry of Education and Science, Republic of Serbia, Grant 451-03-68/2022-14/ 200378, and Grant TR32036.

REFERENCES

[1] S. Vulović, R. Vujanac, M. Živković, M. Topalović and A. Dišić (2020), *FEM modelling of wind load on industrial filter;* 5th International Scientific Conference, Cometa 2020 "Conference on Mechanical Engineering Technologies and Applications", East Sarajevo, RS, B&H, pp. 138-145, ISBN 978-99976-719-8-1.
[2] S. Vulovć, D. Pavlović, M. Živković, R. Vujanac and M. Topalović (2021), *Analysis of freight wagons for transporting of bulk materials;* 5th Serbian-Greek Symposium on Advanced Mechanics, Kragujevac, Serbia.
[3] S. Vulović, M. Bojović and M. Topalović (2020), *Automation of FEM Analysis Report Generation using Visual Basic FEMAP API*. In: Zdravković, M., Konjović, Z., Trajanović, M. (Eds.) ICIST 2020 Proceedings, pp.10-15.
[4] Siemens (2015), FEMAP User Guide Version 11.2, Siemens PLM Software Inc.
[5] M. P. Robillard (2009), "What Makes APIs Hard to Learn? Answers from Developers," IEEE Software, 26/6, p.p. 27-34
[6] M. Meng, S. Steinhardt and A. Schubert (2017), "Application Programming Interface Documentation: What Do Software Developers Want?" Journal of Technical Writing and Communication, 48/3, p.p. 1-36
[7] Siemens (2021), FEMAP API Reference, Siemens Digital Industries Software Inc