A XIX-a Conferinţă internaţională – multidisciplinară
„Profesorul Dorin PAVEL – fondatorul hidroenergeticii româneşti",
CLUJ NAPOCA, 2019

ANALIZA STRUCTURALĂ ȘI MODALĂ ASUPRA UNUI LIMITATOR DE VITEZĂ INTELIGENT PENTRU ALEGEREA MATERIALULUI OPTIM

Mihai REBREANU

THE STRUCTURAL AND MODAL ANALYSIS OF AN INTELLIGENT SPEED BUMP FOR CHOOSING THE SUITABLE MATERIAL

This study has the purpose of giving information related to the structural resistence of the speed bump elements while being tested in a virtual finite element analysis simulation with real world data scenario. The tests consists of two types: stress and modal frequency calculation. The test subjects were the two main elements of the assembly: the fixed central part; the sliding element. There are two materails that have been considered for this test, namely: Peek plastic and steel.

Keywords: finite element analysis, simulation, intelligent speed bump, innovative, traffic, public road, testing, material, Peek plastics, Internet of Things
Cuvinte cheie: analiza cu element finit, simulare, limitator de viteza inteligent, inovativ,trafic, drum public, testare, material, plastic Peek, Internet of Things

1. Introduction

The traffic's condition in many of the old cities of Europe, but not only has gotten into a point where drastic changes have to be brought up in order to ease the daily routine of the growing population that keeps those cities in a state of economic blossomness. The importance of day to day traffic is high enough to be considered the factor that can make or break a city and thus the managing system that conducts and monitors this aspect has to be up to its highest standard in order to allow the economic status to expand.

The solution to this complex problem must improve the traffic flow, monitor the behavior of the individual or group transportation means, ensure that the imposed traffic rules are obeyed and, ideally, be as simple as possible in order to allow it to be easily implemented into the existing infrastructure. There are more types of devices that reduce the speed limit of a vehicle. These models can be categorized as passive or intelligent speed controlling devices also known as speed bumps. The reliability of those devices is crucial in their operation and tests have to be conducted to prove that they can perform no matter the engineering approach that has been used. Out of all the existing models, this study focuses upon the FEA simulation of an intelligent speed bump that is mountable on the tarmac surface (figure 1).

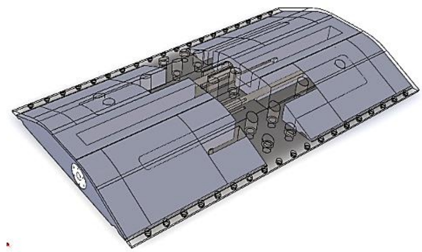


Fig. 1 View of the intelligent speed bump with transparent body

2. Approach method

The FEA simulation tests have to prove that the device can handle the stresses that act upon it. Those stresses have been considered in such a manner that they would mimic the ones of a large personal vehicle that slows down in an urgent manner from 80 km/h to 15 km/h at the moment of contact with the intelligent speed bump's body. The resulting force is discretized and applied to the desired contact patch. In this simulation, there are two contact patches that are tested, one on the extending element of the speed bump and the other on the central body. After the two contact tests have been calculated, the main assembly parts were tested for fatigue in order to have an estimation of their life span according to its own resonance frequency.

The input data for the calculations was taken from a theoretical vehicle mass of 2000 [kg] proportionally distributed between the two axes and wheels so that each wheel holds a constant mass of 500 [kg]. The of the calculations were deduced from the total energy formula.

The calculations for the contact patch collision test has been done with the formula for the conservation of the kinetical energy of an object that is slowing down as shown in (figure 2).

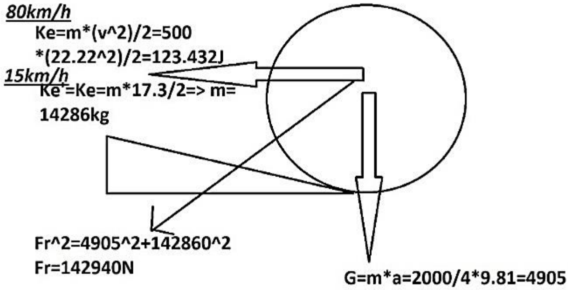
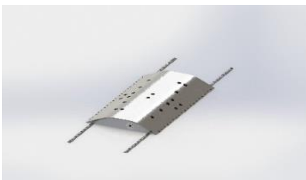


Fig. 2 Calculus for the contact force

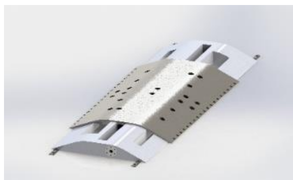
3. The model creation

The CAD model for both the mobile/ extending element as well as for the central body have been set up with a discretized mesh shape to allow it to be numerically calculated by the FEA simulation software.

As a software tool for both the CAD modelling and FEA simulation the program SolidWorks from Dassault Systems was used. The assembly contains the main/central body of the speed limiting device, the two extending elements on the sides, the motors, the rod caps and the optional guide lines. The two phases in which the speed limiting device can be found in are illustrated in the pictures below:



a.



b.

Fig 3 The two states of the intelligent speed bump

4. Experimental analysis

The analysis consists of the 3 tests that were conducted in the virtual environment for two different materials that were considered for the mobile elements, namely the innovative Peek plastic and JIS S20C steel. An important point is that the tests will run exclusively on the mobile elements of the assembly, given the fact that they are at the highest risk of being damaged during its life span.

The first test was conducted with the aid of the calculus of the resulting force between the gravitational and inertia force of the vehicle slowing down. The resulting force was applied on a contact patch on the body of the mobile part, where the car’s wheel theoretically makes contact with the element (figure 4.1).

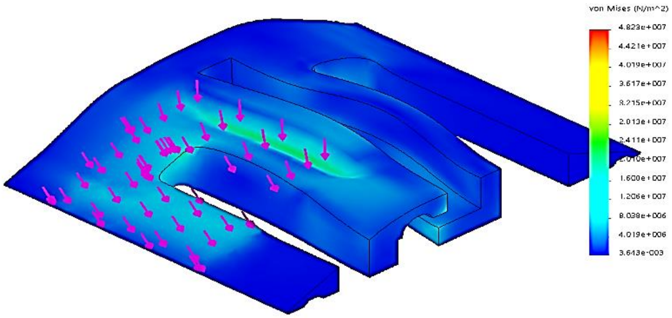


Fig 4.1: Result of stress applied to contact patch; Peek plastics

The results of the test show that the stress has manifested as a shearing force on the upper side of the structure because of the hollow underside of the affected part of the element. Comparing the two visual results, we can observe that the maximum stresses of 48 MPa are close to identical and thus we can notice that the two materials have similar performance.

The visual results indicate that the displacement caused by the pressure applied by the wheel tends to shear a part of the upper structure of the mobile element. The amplitude of the deformation that occurs in that area of maximum stresses has a value of 0.65 [mm] for the Peek plastic model and 0.13 [mm] for the steel model.

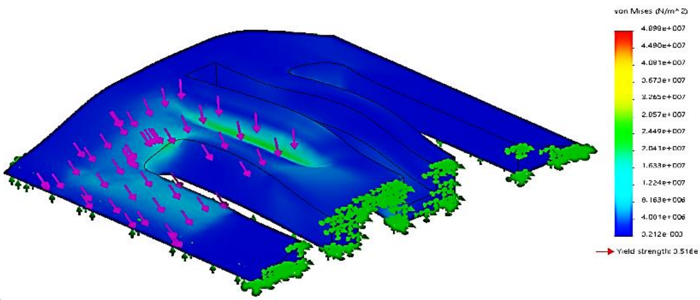


Fig 4.2 Result of stress applied to contact patch; JIS S20C steel

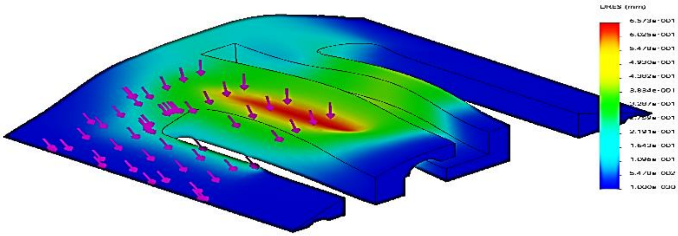


Fig 4.3 Result of displacement of the mobile element; Peek plastics

These results show that there is a considerable displacement difference between the two construction solutions, but considering the elastic properties of the Peek plastic, the solution is still viable.

The modal test was executed on the Peek plastic model for better understanding the way in which the material reacts to the external factors. The test ran for approximatively 1,000,000 cycles before affecting the structural integrity of the model.

According to the fatigue results, the model deals with the external vibrations well and does not display many signs of stress or damage.

5. Conclusions

■ Following the results of the tests we can conclude of the Peek plastic model solution can withstand the harsh conditions that the daily traffic implies. Even considering the performance of the steel structure, the revolutionary Peek plastic material is more interesting for this specific task since it has a lower mass density.

■ The Peek material is a rarer one and has to pass the financial tests to be labeled viable for the usage in such a device. On the other hand the steel option is a more accessible one because of its wide spread usage.

■ The first test that resulted in the stress and displacement charts was conducted in both material conditions and the results are plausible and return values in an accepted range.

■ The fatigue test returned good values and deems it usable for certain road and traffic condition. However, the conditions in which it ran are not necessarily relevant to the conditions in which it's supposed to work.

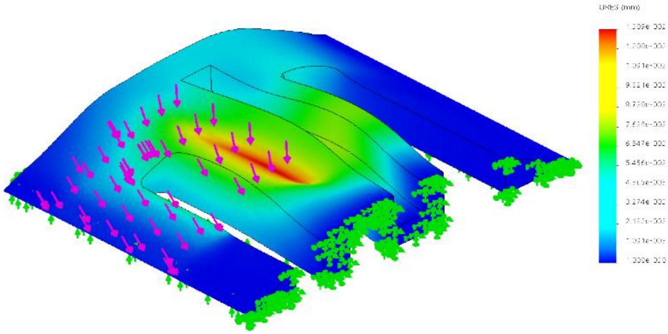


Figure 4.4 Results of displacement of mobile element; S20C steel

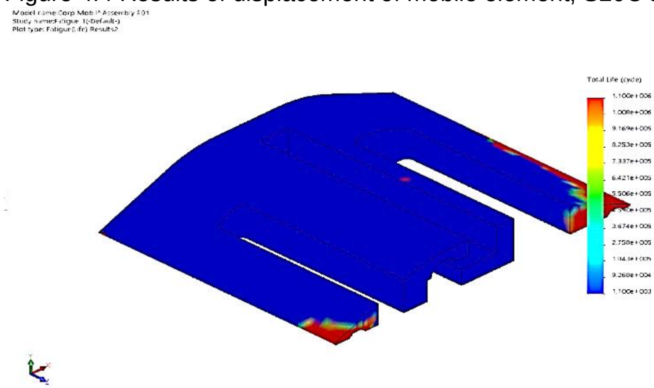


Figure 4.5 Fatigue results for the Peek plastic model

REFERENCES

[1] * * * AZO Materials https://www.azom.com/properties.aspx?ArticleID=1882
[2] * * * Dielectric Corporation http://www.dielectriccorp.com/downloads /thermoplastics/peek.pdf
[3] Dan Sorin Comșa, *Metoda elementelor finite: curs introductiv*, Editura U.T. Pres, Cluj-Napoca, 2007.

Ing. Mihai REBREANU
Student la specializarea de masterat TAIA,
Universitatea Tehnică Cluj Napoca,
e-mail: mihairebreanu95@gmail.com
telefon: 0733 613 246