ВЕСТНИК
ТУРИНСКОГО ПОЛИТЕХНИЧЕСКОГО УНИВЕРСИТЕТА В ГОРОДЕ ТАШКЕНТЕ

ACTA
OF TURIN POLYTECHNIC UNIVERSITY IN TASHKENT

ВЫПУСК
EDITION 3/2021

CONTENTS

Acta of Turin Polytechnic University in Tashkent, 2021, 30, 30-32
Published Online September 2021 in Acta TTPU (http://www.acta.polito.uz/)

STRESS AND DEFORMATION ANALYSIS OF THE BRAKE PEDAL USING FINITE ELEMENT METHOD.

Seyran Asanov
Department of Mechanical and Aerospace Engineering
Turin Polytechnic University in Tashkent,17, Little Ring Road street, Tashkent, Uzbekistan
Email: seyran.asanov@polito.uz

Abstract– Brake pedal is one of the most crucial components of the braking system of all types of vehicles.During the design phase it is of vital importance to make sure that the pedal is able to withstand the loads exerted by the driver during the deceleration process. Moreover, the pedal must be "ready" to withstand large loads in a hard braking process. This article describes the static stress and deformation analysis of the brake pedal with the help of Finite Elements Method (FEM). The paper summarises that a safe region of stress and deformation is achieved even when a force of 1100 N is applied at the brake pedal.

Key words– brake pedal, FEM, Finite Elements Method, stress analysis, strain analysis

I INTRODUCTION

Currently there are two main factors that encourage the development of already existing braking systems and development of new components and technologies: legislation and consumers. Speaking of automotive components and brake components in particular, one can highlight that their design has to include the considerations of not only safety, but also comfort.However, the scope of the article covers only the safety point of view.Before implementing a component, the analysis of its performance has to be carried out. A proper analysis of brake pedals allows to develop the product not only for real vehicles which will be realised in the global market, but also provides necessary concepts to develop components for brake pedal force simulators. Modern advancements of computer technologies make it possible to perform the analysis of complex systems and components using highly efficient software packages. The implementation of the Finite Elements Method is carried out in Solidworks software (Dassault systems). The Finite Element Method (FEM) is best used when analytical methods are difficult or impossible to use. This method allows the calculation of deformations and loads with high accuracy when the constraints, forces are set correctly[1][2].

TABLE 1: INFORMATION ON THE BRAKE PEDAL

Parameters of the system	Property
Material used	steel AISI 4340 normalized at 870°C
Yield strength $R_{p0.2}$	710 MPa
Ultimate strength R_m	1100 MPa
Young modulus E	205 GPa
Poisson's ratio ν	0.29

TABLE 2: MESH CHARACTERISTICS OF THE PEDAL

Mesh type of the system	Solid Mesh
Mesher Used:	Curvature based mesh
Jacobian points	16
Maximum element size	1mm
Minimum element size	0.33 mm
Mesh quality	High
Remesh parts with incompatible mesh	Off

II METHOD

As a first approximation, an assumption for the pedal statics has to be introduced. The point is that the brake pedal has limited stroke and it reaches the limit condition in a quick manner. Besides, there is a feedback force from the hydraulic circuit of the braking systems which opposes the motion and diminishes inertia effects.The brake pedal will be analyzed in tandem to the idler shaft which is often used in clutch and brake systems(Figure 1). The reason is these parts are the most stressed during the braking operation. Basically the Finite Element Method analysis in Solidworks consists of the following steps [3]:

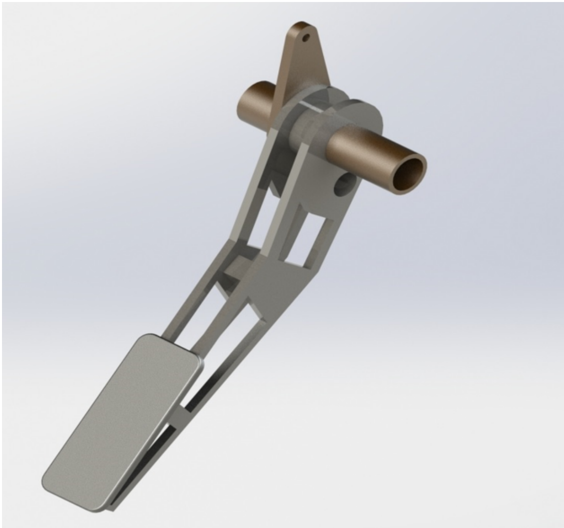

Fig. 1: CAD model of the brake pedal

- Material definition;
- Applying proper connections and boundary conditions;
- Meshing the structure;
- Defining the loads acting on the system;
- Simulation;

The information on material properties is listed in Table 1. The material is supposed to be linearly elastic and homogeneous.

The next step is to apply the proper fixtures and boundary conditions to the system under investigation. .Since the static analysis is to be performed, the shaft has to be fixed on both ends (Figure 2). Due to the complexity of the system, a 3D analysis will be carried out and, therefore, the next step becomes to do discretisation of the system by creating a proper mesh. For the scope of the problem curvature-based mesh will be used and an h-adaptive method for mesh convergence will be introduced. In h-adaptive method the software uses iterative procedures and with each loop it decreases the mesh size in the high stress-areas. As far as loads are concerned, there is only one force applied- the force exerted on the brake pedal. In these studies it is equal to 1100N which is a good representative value of a force during a heavy braking operation.It can be highlighted that in general the braking force is not applied over the whole pedal area, but over a certain region. Finally, the simulation of the system takes place once the previous steps have been done.

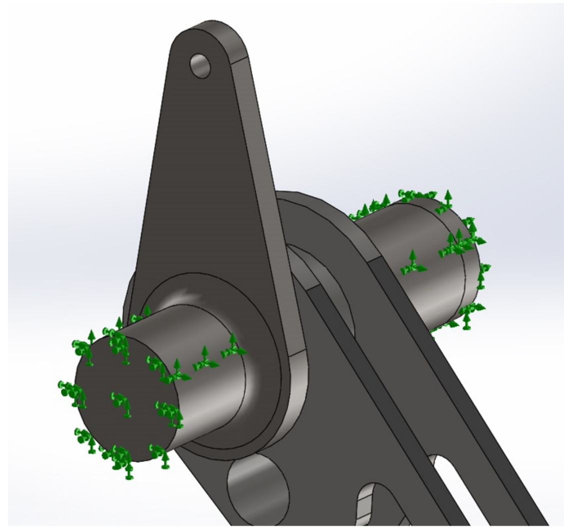

Fig. 2: Fixed geometry

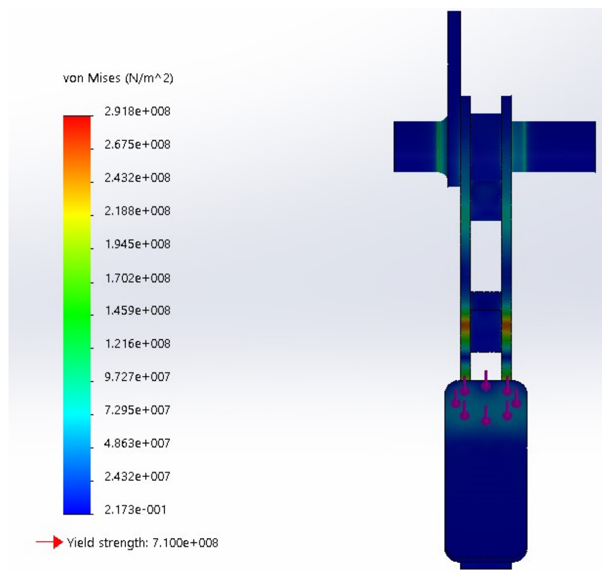

Fig. 3: Von-Mises equivalent stress distribution

III Analysis of the results

Interpreting results starts from understanding if a proper meshing has been done for the system. As can been in the Figure 2, the mesh maximum size is 1mm, the type is curvature based. The mesh quality is high, more than 99 per cent of elements have an aspect ratio lower than 3. The obtained mesh is quite fine.As was mentioned before, an h-adaptive method was used. The characteristics of the method are presented in Table 3. According to the simulation data, one can conclude that the structure consisting of brake pedal and brake pedal idler shaft is able to withstand the maximum load under investigation regardless of the pedal inclination.

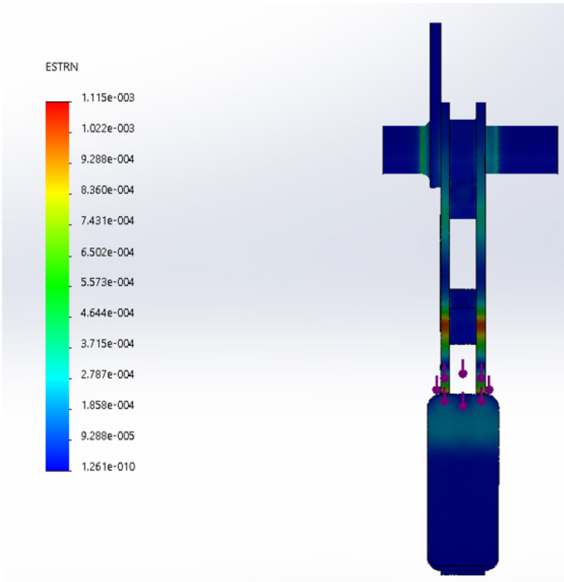

Fig. 4: Strain distribution

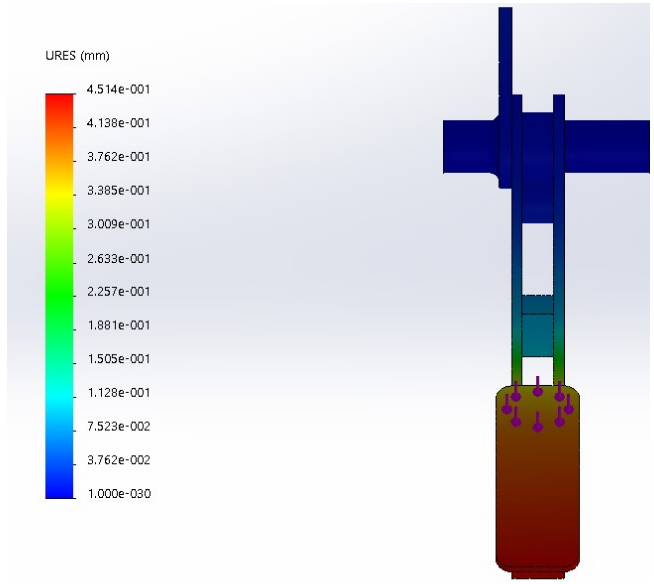

Fig. 5: Transverse displacement distribution of the brake pedal

TABLE 3: MESHING RESULTS

Total nodes	1657958
Total elements	1141705
Maximum Aspect ratio	8.3561
Share of elements with Aspect ratio <3 (percent)	99.8
Share of elements with Aspect ratio >10 (percent)	0
Share of distorted elements (percent)	0
Time to complete mesh (hh:mm:ss)	00:00:51

From the stress plot (Figure 3) showing the equivalent stress according to the Von-Mises failure criterion, it becomes evident that the maximum generated stress does not exceed the value of the yield stress which implies that no yielding will occur under the force of 1100 N. Moreover, basing on the results of the stress distribution it is possible to evaluate the safety factor:

$$F_s = \frac{R_{p0.2}}{\sigma_{eq}^{max}} = \frac{710MPa}{291.8MPa} = 2.43 \tag{1}$$

The obtained value of the safety factor is considered good for ductile materials [4]. According to the strain plot, the maximum value corresponding to the strain is in the region corresponding to the largest stress (Figure 4). This is justified by Hooke's law for linearly elastic and homogeneous materials:

$$[\varepsilon] = \frac{1}{E}[\sigma] \tag{2}$$

As far as the transverse displacement is concerned, the maximum value corresponds to the pedal region (Figure 5). It is quite clear that the amount of displacement is relative small (approximately 0.45mm) to affect the change of geometry of the braking system.

IV CONCLUSION

The results of the article claim that the structure consisting of the brake pedal and the idler shaft is able to withstand an extremely large load without yielding. Moreover, the amount of displacement caused by the load is considered acceptable from engineering point of view. The deformation due to the applied load does not produce significant geometrical changes to the system. The performed foundation serves as a good first approximation of the overall analysis. However, some improvements can be done by taking into account the kinematics and dynamics of the brake pedal during the deceleration process.

REFERENCES

[1] N. Abdukarimov S. Asanov S. Ro'zimov, J. Mavlonov. *SOLIDWORKS dasturida avtomatik loyihalash asoslari*. Tashkent, TTPU, 2021.

[2] S.Ruzimov J.Sodiqov R. Mukhammadaliev. U.Selgren Sh.Andersen, S.Eshkabilov. *UGS I-DEAS NX Series programma komplekslarida avtomatik loyihalash asoslari (in Uzbek)*. Tashkent, Fan, 2006.

[3] www.solidworks.com. 3ds [online].

[4] K. Nisbett R. Budynas. *Shigley's mechanical engineering design, 8th edition.,m.* New York: McGraw-Hill, 2008.