INTERNATIONAL JOURNAL OF ADVANCE RESEARCH, IDEAS AND INNOVATIONS IN TECHNOLOGY

ISSN: 2454-132X
Impact Factor: 6.078
(Volume 7, Issue 2 - V7I2-1358)
Available online at: https://www.ijariit.com

Static structural analysis of a brake pedal

Aayush Shah
aayush.21810475@viit.ac.in
Vishwakarma Institute of Information Technology,
Pune, Maharashtra

Atul Kulkarni
atul.kulkarni@viit.ac.in
Vishwakarma Institute of Information Technology,
Pune, Maharashtra

ABSTRACT

This work contains the design and analysis of brake pedal for a Formula 3 Vehicle. different 'Engineering Software' like SOLIDWORK and ANSYS are utilized for the analysis of brake pedal, considering the structural aspects. s. In this work, every progression of design was assessed and all the results of analysis have been interpreted. Finite element analysis has provided a lot of new opportunities for the development of new components and it saves a lot of time and expenditure on the proof of concept. Popular finite element softwares are ANSYS, ABACUS, etc. These softwares are proved to give an almost accurate solution. However, these results are considered to be approximate results. In current research paper, a Static structural analysis of a brake pedal is presented using ANSYS. It is specially designed and used for a F3 Vehicle. It has features like weight optimization and sufficient stiffness to absorb all the forces acing on it. The result has been compared with different meshed models like tetrahedron, hexahedron and the combination of both and the comparative analysis has been reported. It is observed that there is a significant difference in stress, deformation based on the mesh, however the stress and deformation are within the limits. Hence, this research paper will be helpful in future for predictive analysis.

Keywords: *Brake Pedal, Meshing.*

1. INTRODUCTION

Braking system is an essential part of any automobile. Without the ability to slow and stop our vehicles, we cannot hold control on it and ultimately accidents would occur. So each and every motor vehicle requires a reliable braking system. Finite Element Analysis or FEA is the simulation of a physical phenomenon using a numerical mathematical technique Finite Element Method, or FEM. This process is at the core of mechanical engineering, as well as a variety of other disciplines. It also is one of the key principles used in the development of simulation software. Engineers can use these FEM to reduce the number of physical prototypes and run virtual experiments to optimize their designs.

2. LITERATURE REVIEW

Bhagyashri Kurkure: Now-a-days, industries are replacing accelerator and clutch pedal by lightweight materials such as polymer plastic, composites, aluminum and its alloys, etc. The purpose is to reduce weight, cost, and improvement in corrosion resistance without change in material reduction. a commercial vehicle casted brake pedal lever. The FEM and analysis of a brake pedal lever has been carried out. The FE model was generated in CATIA or Pro-E and imported in ANSYS for stress analysis and then optimizing it with the help of Optistruct software. A comparison of baseline and optimized model FEA results have been done to conclude.

Dr. Hossein Saidpour: The vehicle component of materials is dependent on a supply and demand process, subject to requirements. Metals i.e. steel, aluminum and magnesium are used for elements of the body structure and panels and Plastics are used for exterior attachments to the body. Cars consist of steel and iron but due to the impending use of multi material constructions, it is expected that the amount of steel and iron used is reduced. the steel unibodies are multi material unibodies and aluminum space frames. Magnesium and steel space frame concepts for volume applications are still under development. The materials are replaced by durability and specific strength/ stiffness of high performance carbon fiber composites

Pankaj Chhabra: To determine design concept, Concurrent Engineering (CE) approach is used and material of the composite accelerator pedal at conceptual design stage. using the Morphological approach, the Various design concepts are generated. at design stage, CATIA is used and ANSYS is used for analysis. on the basis of past research & specifications, material selection is done. The pedal arm profile on the basis of stress, mass & volume, and deformation results achieved on ANSYS. Analyzed and optimized the accelerator pedal for safety parameters and finally prototyped using Selective Laser Sintering. the feasibility of composite accelerator pedal with glass filled polyamide providing saving better properties and substantial weight than existing metallic pedal

3. CALCULATIONS

The pedal ratio according to the design is 1:4. The force on the

pedal is taken as 2kN.
Pedal force = half of the weight of driver * gravitational force
= 35*10 =350N

4. METHODOLOGY
The process started with designing of the brake pedal. While designing weight reduction of the pedal was done. After the design, analysis was done keeping in mind the material selection. Different meshing models like Tetrahedron, Hexahedron and a combination was analyzed and the results are shown.

5. BRAKE PEDAL DESIGN
The brake pedal is a device used to actuate the brakes of any automobile vehicle. Hence it is expected that brake pedal must be durable and must have low weight

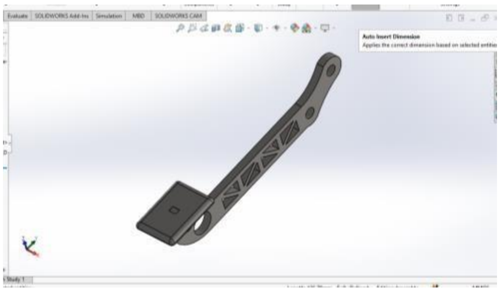

Fig. 1: Design of Brake Pedal

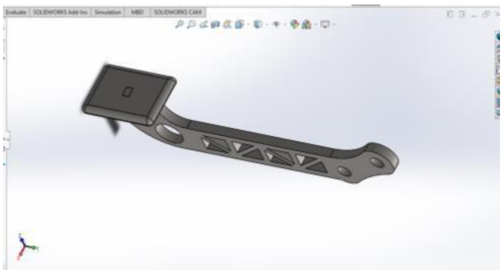

Fig. 2: Design of Brake Pedal

Here we can observe the design of the brake pedal and the material has been removed from it for the purpose of weight optimization.

6. MATERIAL
ALUMINIUM 6061-T6
Based on the type of loading, the pedal box would be subjected to it was modelled as beam in bending andwas decided that the desired material needed have the following material properties:
- High strength
- Good toughness
- Light weight
- Good machinability
- Corrosion resistance
- Wieldable

7. MESHED MODEL & COMPARISON
The finite element mesh is used to subdivide the CAD model into smaller domains called elements, over which a set of partial differential equations are solved. These equations approximately represent the governing equation of interest via a set of polynomial functions defined over each element. As these elements are made smaller and smaller, as the mesh is refined, the computed solution will approach the true solution. The selected element is a three-node beam element in 3-D. With default settings, six degrees of freedom occur at each node; these include translations in the x, y, and z directions and rotations about the x, y, and z directions.

Here, we make use of unstructured tetrahedron mesh since there are a lot of curvatures and uniform mesh won't be helpful in this case. Tetrahedron is best for discretizing complex geometries, although it takes more computational time.

No of Nodes – 26,976

No of Elements – 1,03,878

Mesh Type – Tetrahedron and Hexahedron

Fig. 3: Meshed Model

The mesh size and type has significant influence on the analysis. Hexahedron type of meshing is more accurate. But the skewness is better in hexahedron meshing only in plane geometries. Tetrahedron type of meshing is done at the curved surfaces. The element size is kept less to have accuracy at the curvatures.

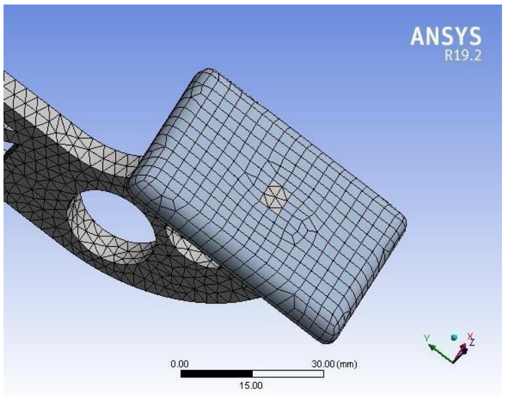

Fig. 4: Detailed View

Comparing the meshed model with the models meshed in only tetrahedron and hexahedron separately the results are as follows:

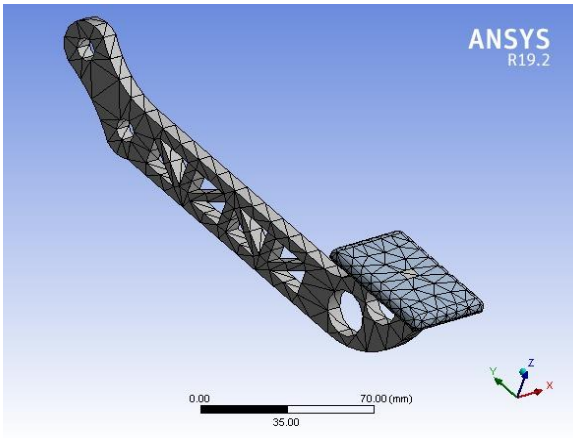

Fig. 5: Tetrahedron meshed model

Full Part Tetrahedron Mesh:
Nodes - 2970
Elements – 1300

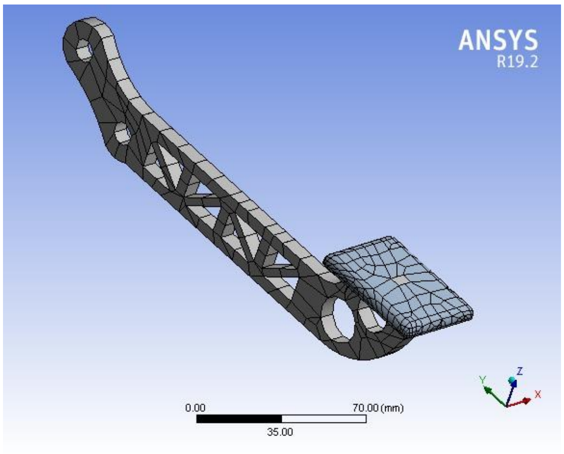

Fig. 6: Hexahedron meshed model

Full Body Hexahedron Mesh
Nodes - 5173
Elements – 1578
Hence from the models we can observe that the meshing we used in the first model, a combination of Tetrahedron at the nonlinear part and hexahedron at the linear part is much more accurate and better as compared to full body Tetrahedron or full body Hexahedron mesh.

7. ANALYSIS

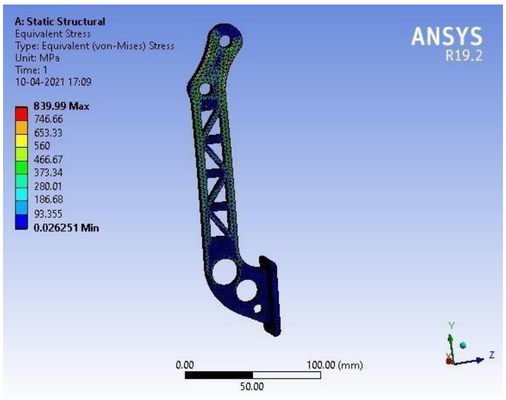

Fig. 7: Total Deformation of Brake Pedal

The total elastic deformation of the brake pedal is 4.89mm for a normal load of 2kN.

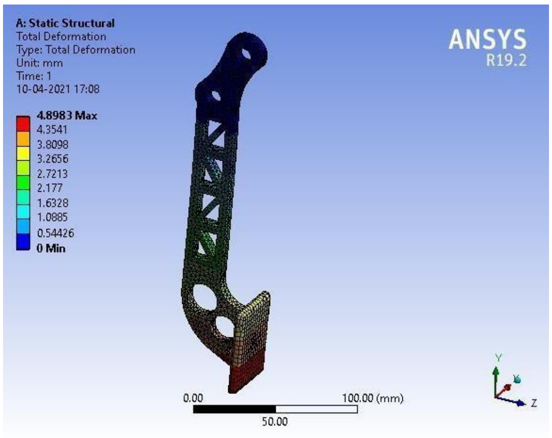

Fig. 8: Equivalent Stress of Brake Pedal

The Stress in the brake pedal under the load of 2kN ismaximum of 39.99 MPa & minimum of 0.026 MPa. We have done weight optimization in the pedal to decrease the weight.

8. CONCLUSION

Hence the FEA of brake pedal is done and is proved as safe since none of the deformations exceed the given safety values. Also a combination of hexahedron at linear surfaces and tetrahedron at nonlinear was found more accurate as compared to the single type meshed models.

9. REFERENCES

[1] Dr. Hossein Saidpour, “Lightweight High Performance Materials for Car Body Structures”, NTI Technology Conference, CEME, Ford Motor Company, 16 June 2004.
[2] Bhagyashri Kurkure1, Mr. D. B. Sadaphale, “Optimization of Brake Pedal”, International Journal of Mechanical and Industrial Technology ISSN 2348-7593 (Online), Vol. 3, Issue 1, Month: April 2015 - September 2015.
[3] Pankaj Chhabra, “Concurrent Design and Prototyping of Composite Accelerator Pedal”, International Journal of Advancements in Technology, ISSN 0976-4860, Vol. 2 No. 4 October 2011
[4] International Research Journal of Engineering and Technology (IRJET) Volume: 04 Issue: 05 May -2017

