International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 8 Issue 06, June-2019

Topology Optimization of an All-Terrain Vehicle Brake Pedal

Pritesh Muralidhar Ingale
Department of Mechanical Engineering
Lokmanya Tilak College of Engineering, Koparkhairane
Navi Mumbai, India

***Abstract*—This paper studies the topological optimization of All Terrain Vehicle (ATV) brake pedal for minimum mass without compromising Factor of Safety. The three dimensional solid model of a pedal is created using Solidworks 2019 and static structural analysis and Topology optimization, geometry modification is done using Ansys 18.1. The main objective of study is to minimize the weight without compromising the Factor of Safety.**

***Keywords*—Topology, factor of safety, static analysis, All terrain vehicle**

I. INTRODUCTION

Topology optimization (TO) is a mathematical method that optimizes material layout within a given design space, for a given set of loads, boundary conditions and constraints with the goal of maximizing the performance of the system. Topology Optimization is different from shape optimization and sizing optimization in the sense that the design can attain any shape within the design space, instead of dealing with predefined configurations.

The conventional Topology Optimization formulation uses a finite element method (FEM) to evaluate the design performance. The design is optimized using either gradient-based mathematical programming techniques such as the optimality criteria algorithm and the method of moving asymptotes or non-gradient-based algorithms such as genetic algorithms.

Topology Optimization has a wide range of applications in aerospace, mechanical, bio-chemical and civil engineering. Currently, engineers mostly use TO at the concept level of a design process. Due to the free forms that naturally occur, the result is often difficult to manufacture. For that reason, the result emerging from Topology Optimization.

II. DESIGN CONSIDERATION

The design of brake pedal plays a very crucial role as it is the deciding factor for the force available at master cylinder end. Since there is space constraint for driver while designing an ATV, the resting position of leg on brake pedal is such that it is quite difficult for the driver to produce exact amount of pedal force as compared to force produce by the leg in a regular vehicle. Based on the space constraint and resting position of the leg of the driver the brake pedal is needed to be designed so that the force produced is able to push in the master cylinder completely to its full-length stroke. The maximum force applied while sitting in an ATV can be supposed from human performance capabilities

A. Design

To design a brake pedal having brake pedal ratio of more than 5:1, as it is suggested that manual braking system (with no power booster) should have a ratio greater than 5. The length of brake pedal should not be too short or too long that hinders the driving position of the driver's leg. The brake pedal ratio should be such that the driver would not feel fatigue while braking.

The brake pedal was designed using SolidWorks considering the design constraints and parameters. Figure 1 shows the 2D sketch of the pedal. The length of brake pedal was decided by calculating the brake pedal ratio. Brake pedal ratio is the ratio of the perpendicular distance from pivot point center to lower end of the brake pedal by the distance from pivot point center to pushrod mounting point center.

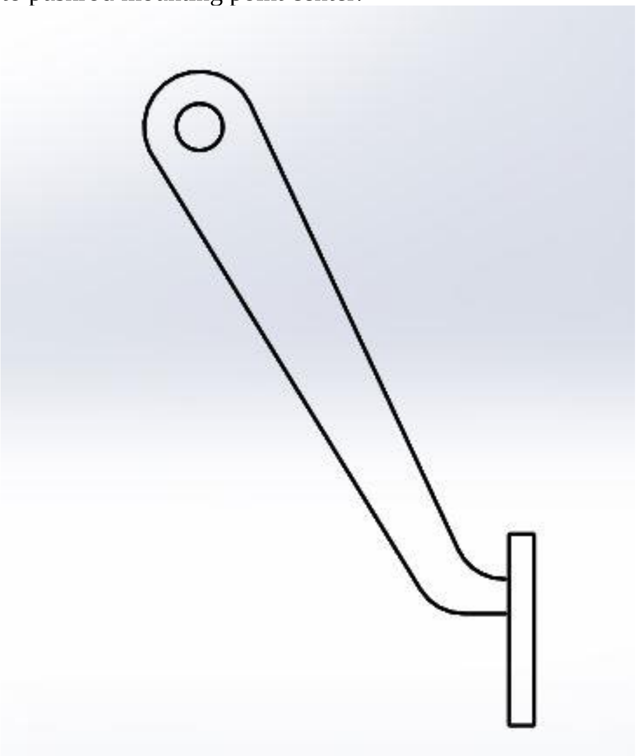

Figure 1: 2D Sketch of brake pedal

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 8 Issue 06, June-2019

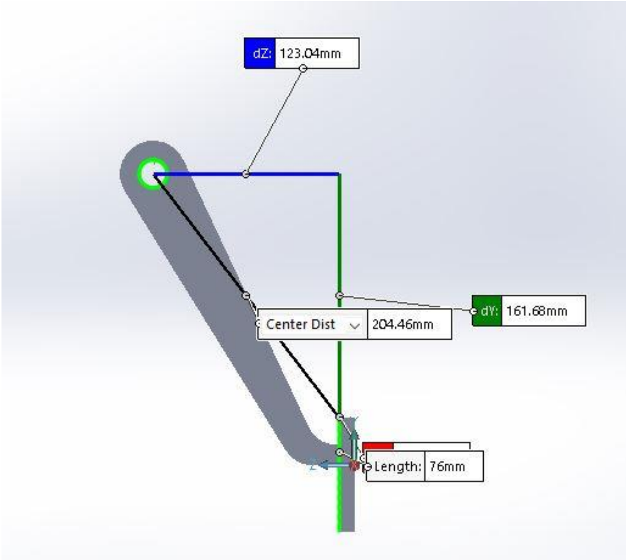

Figure 2: Dimensions of Brake Pedal

B. Material Selection for Pedal

Al-7075 and Al-6061 was chosen for comparison and various comparable properties are shown in Table1. Comparison of material is based on following factors

1. Strength to weight ratio
2. Cost
3. Physical and mechanical properties
4. Availability

Table1: Material properties

Properties/ Material	Al-7075	Al-6061
Yield Strength (MPa)	475.73	276
Elongation (%)	7.9	12
Density (g/cc)	2.81	2.7
Fatigue Strength (MPa)	159	96.5

III. STATIC ANALYSIS IN ANSYS WORKBENCH

Table2: Mesh Information

Mesh type	Solid mesh
Element Size	2.00 mm
Total Nodes	260423
Total Elements	127651
Maximum Aspect Ratio	7

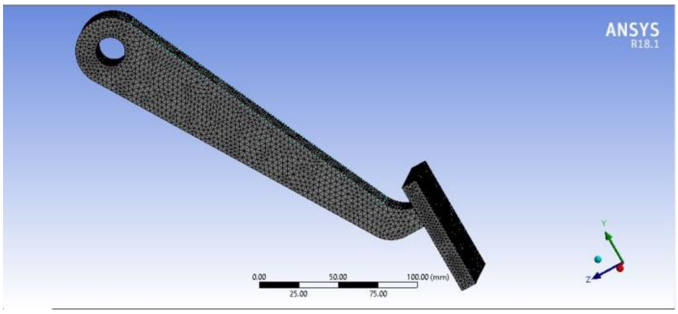

Figure 3: Meshed view of Brake Pedal

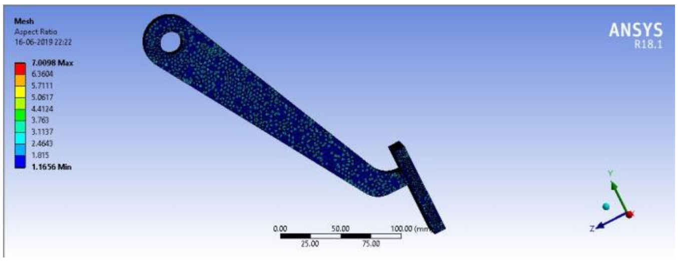

Figure 4: Mesh check Aspect Ratio

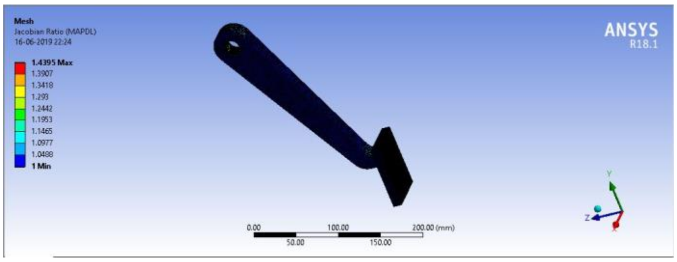

Figure 5: Mesh check Jacobian Ratio

Load and Fixture for the Analysis

Fixture Name: Fixed Support

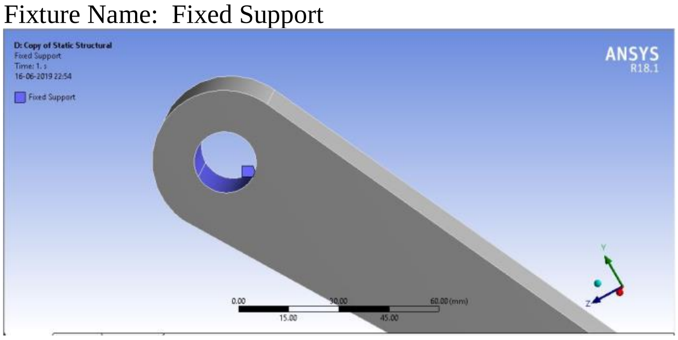

Figure 6: Fixed support at mounting point

Fixture details:

Entities: 1 face(s)
Type: Fixed Geometry

Load Name: Force

Load Direction: Perpendicular to the face

Load Quantity: 500N

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 8 Issue 06, June-2019

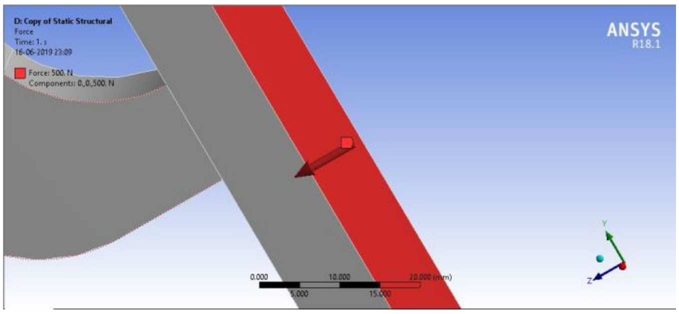

Figure 7: Load acting on pedal surface

Load Details
Force-1
Entities: 1 face(s)
Type: Apply normal force
Value: 500 N

Analysis Results:

Stress Generated:
Max: 35.47 MPa
Min: 0 MPa

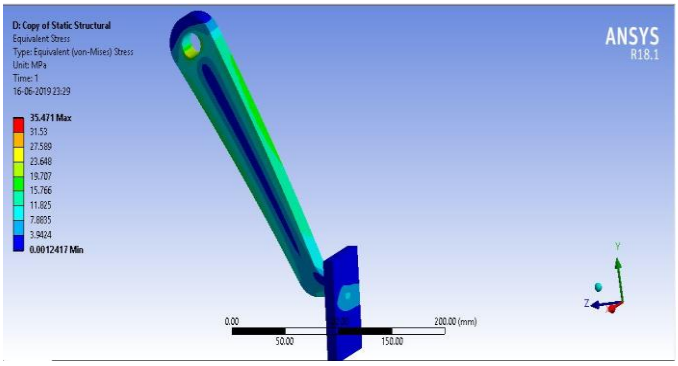

Figure 8: Stress contour in Pedal

Deformation Generated:
Max: 0.52mm
Min: 0 mm

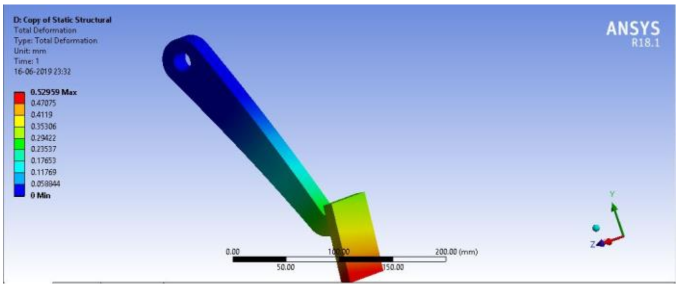

Figure 9: Deformation contour in Pedal

Hence the Static Structural Analysis is done in Ansys

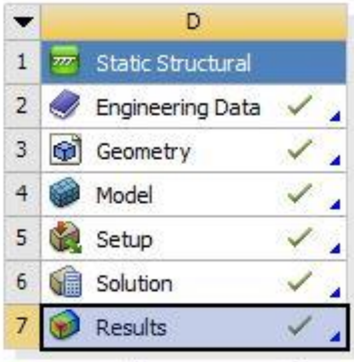

Figure 10: Static Structural Layout

As the stress generated is lower than permissible limits. To minimize the weight by keeping the stiffness of model maximum We will perform Topology Optimization.

IV. TOPOLOGY OPTIMIZATION

To perform Topology Optimization on the Static structural analysis. We will connect Engineering Data, Geometry and Model to the Topology Optimization Setup.

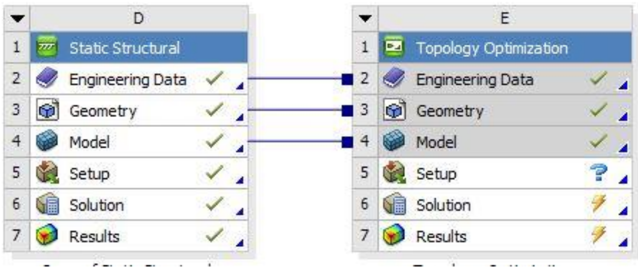

Figure 11: Static Structural Layout connected to Topology Optimization setup

Upload the Solution of Static Structural Analysis to Setup

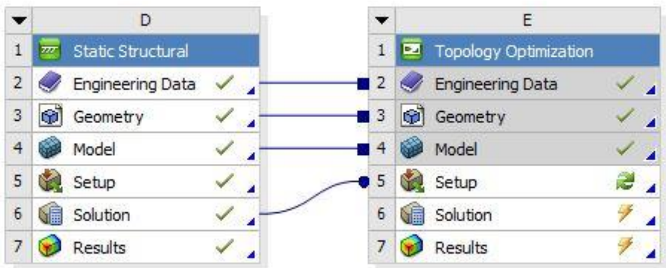

Figure 12: Static Structural Solution connected to Topology Optimization setup

Design Region in Topology setup is the region where the mass will be reduced.
Exclusion region is the region which will not be change after the optimization.

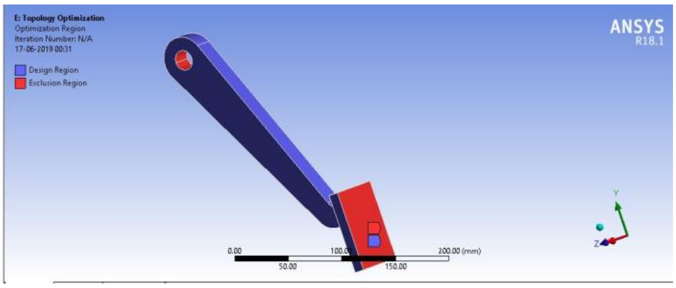

Figure 13: Design and Exclusion Region

Run the solution the software will perform iterations on the Design region for maximum stiffness and minimum weight.

RESULT OF TOPOLOGY OPTIMIZATION:

The result of the optimization is geometry with minimum weight and the region with red color in result is removed material and grey color represents the marginal material as shown in Figure 14.

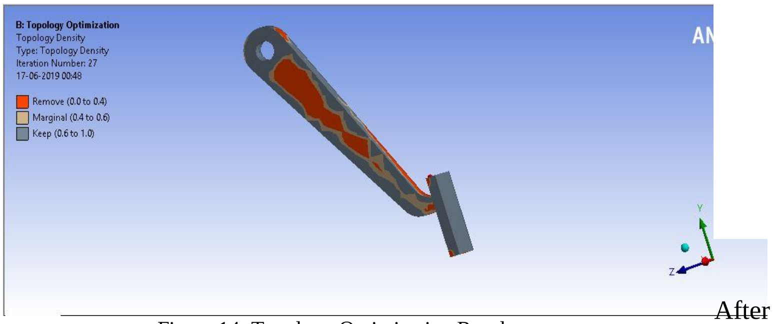

Figure 14: Topology Optimization Result

Figure 15: Topology Optimization Final geometry

V. POST PROCESSING:

The result we obtain from the Topological Optimization is not preferable for manufacturing. We need to perform post processing on the geometry to make it suitable for Manufacturing.
Static structural analysis on optimized model is also necessary.

Therefore we can transfer the Optimization result to new static structural study by Transfer to Design Validation System

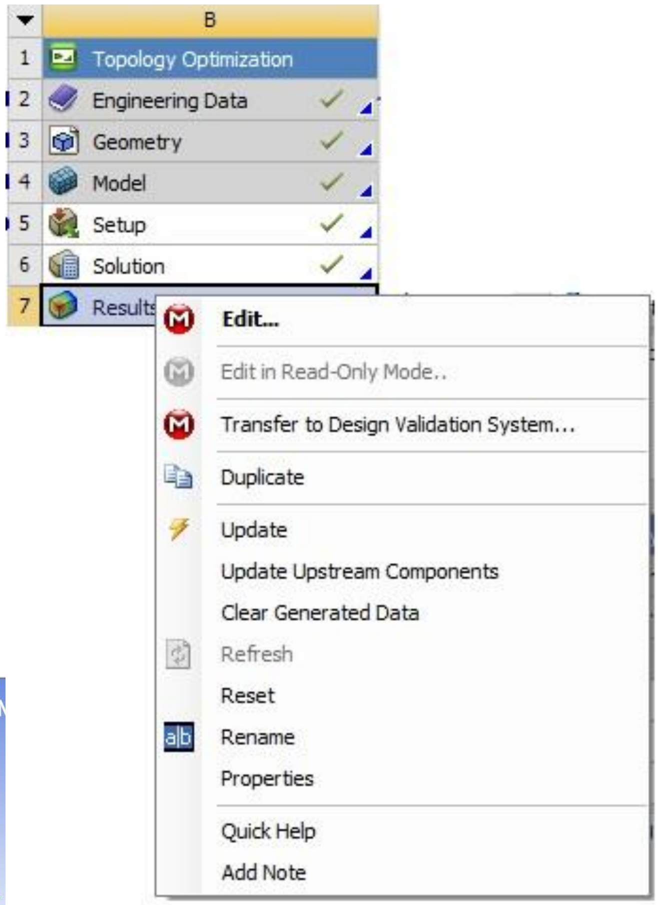

Figure 16: Design Validation system

After transferring results to the design Validation System we have to clean the geometry in space claim to get proper manufactural geometry.

The geometry cleaned in ansys space claim is given in figure 17.

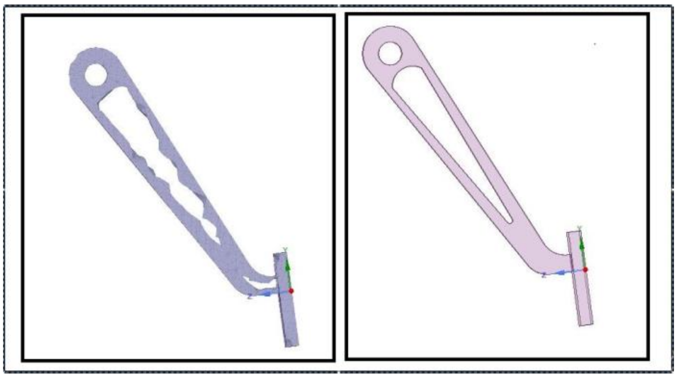

Figure 17: Geometry obtained in results and modified geometry

Static Structural Analysis on modified geometry:

For Material Al-6061

Mesh Attributes has been already discussed above.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 8 Issue 06, June-2019

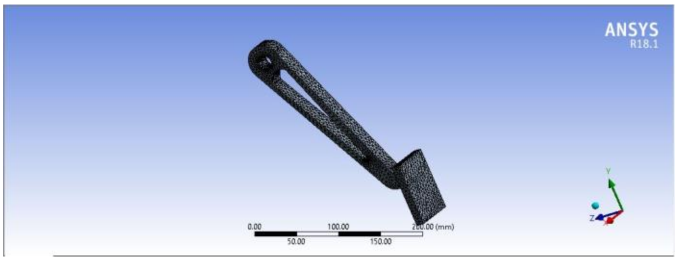

Figure 18: Mesh generated for new geometry

Load and Fixture for the Analysis

Load and fixed support conditions are same as the previous model.

Results of the Analysis:

Stress Generated:
Max: 47.77 MPa
Min: 0 MPa

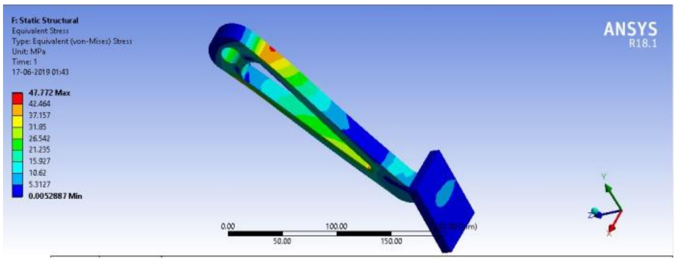

Figure 19: Stress generated in new geometry

Deformation Generated:
Max:0.77mm
Min:0mm

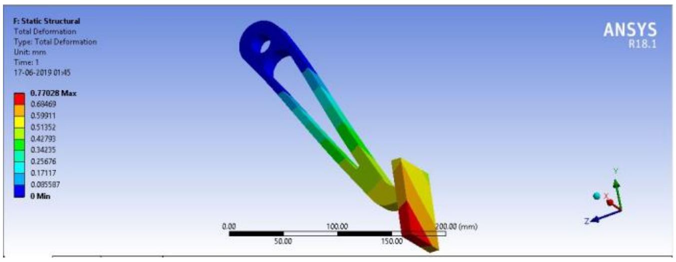

Figure 20: Deformation generated in new geometry

For Material Al-7075

Mesh Attributes has been already discussed above.

Load and Fixture for the Analysis

Load and fixed support conditions are same as the previous model.

Results of the Analysis:

Stress Generated:
Max: 47.77 MPa
Min: 0 MPa

Deformation Generated:
Max:1.20mm
Min:0mm

Comparison of both the results before optimization and after optimization:

Quantity	Before Optimization	After Optimization
Stress (MPa)	35.47	47.77
Deformation (mm)	0.52 (Al-6061) 0.89 (Al-7075)	0.77 (Al-6061) 1.20 (Al-7075)
FOS	6.99 (Al-6061) 12 (Al-7075)	5.19 (Al-6061) 9.29 (Al-7075)
Weight (Kg)	0.521 (Al-6061) 0.562 (Al-7075)	0.384 (Al-6061) 0.370 (Al-7075)

VI. CONCLUSION:

Stress generated in optimized design is close to previous design. Weight after Topology Optimization is approximately half of the original model.The larger factor of safety insures pedal will withstand higher force. Larger FOS is required due to uncertainity in the force.

REFERANCES

[1] Brake technology Handbook
[2] FMVSS135 section 7.11
[3] Design and analysis of brake pedal : an ergonomic approach.