Topology Optimization and Structural Analysis of Brake Rotor and Bell Crank of a Formula Student Car

Prayag Raj Mishra
Veer Surendra Sai University of Technology,
Sambalpur, India

***Abstract*- Over the years various optimization techniques were used to find out the optimal shape and size of engineering structures, under various constraints. So, topology optimization is a mathematical method to optimize the material layout within the design range. So, this study mostly aims on designing an optimal brake rotor and bell crank for the formula student car and performing the structural analysis on both.**

Keywords—Solidworks,ANSYS, Topology Optimization, Total deformation, Equivalent Stress, Brake Rotor, Bell Crank.

I. INTRODUCTION

Topology Optimization is a tool uses mathematical method helps in generating the optimal shape of mechanical structure within a design range. Topology Optimization has a varied range of applications in aerospace, automotive, machine and civil engineering. Currently, engineers mostly use topology optimization at the concept level of a design process.

Formula Student Vehicles are mostly formula 3 prototype that consist of different automotive parts. In this study we discuss about the brake rotor and bell crank, we will see mainly the total deformation and equivalent stress acing on both in different boundary condition and try to optimize and get an efficient design by using Topology Optimization using ANSYS 18.1. Our main aim of using Topology Optimization will be to minimize the weight of the part subjected to given conditions. The idea of topology optimization is the removal of material which is less efficient in the structure. To find the efficient structure one should increase the number of iterations in the analysis, when number of iteration and element increases accurate solution of optimization achieved.

Static Structural Analysis is done using ANSYS where FEA is used to determine the Equivalent stress and Total displacement of the structure. FEM is the numerical method used to perform FEA seeking an approximate solution of displacement in stress analysis. It is basically done by splitting the problem domain into small (finite) bodies or units formed by nodes. FEM calculates at node points and then interpolates the result for the entire domain.

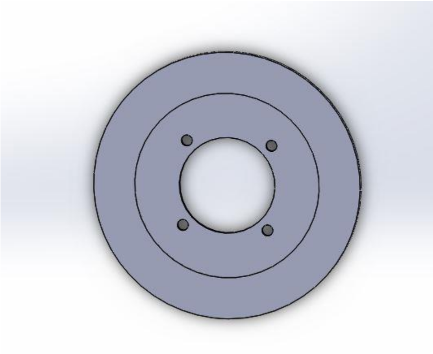
II. METHODOLOGY

Topology Optimization is a tool to find the optimal design of a structure. Many such investigation and research are already done in order to reduce the weight of the structure and to get a suitable design. This research mainly focuses on the objective of achieving a suitable design for the brake rotor and bell crank for the formula student prototype by reducing the weight. The model of the brake rotor and bell crank is designed with the help of Solidworks with considering proper dimensioning. Then it is converted to step file and imported to ANSYS.

- **Brake Rotor –** Once the step file of the rotor imported to static structural, the holes were given as the fixed geometry. Pressure were applied on both faces of rotor and a rotational velocity given to the rotor.
  The pressure that is applied on rotor are the pressure exerted by the brake pads to the rotor while the rotor had a rotational velocity. After this we got the total displacement and Von-mices equivalent stress. Solution was then transfer to the setup file of Topology Optimization.
  In topology optimization we set the iteration to 500 as default keeping the response type as compliance and minimize the mass in a static structural environment, keeping the retain percent to 50.

Geometry of Rotor.

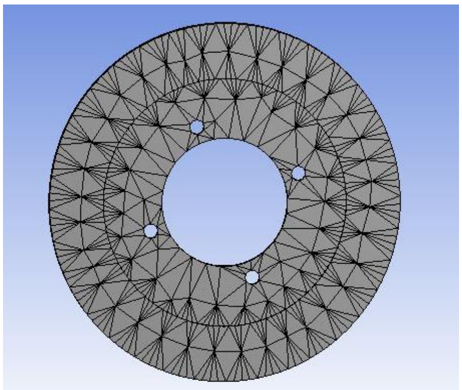
Overall Diameter of Rotor is dimensioned to 200mm, and a hole of diameter 75 mm cut extrude from it for the insertion of Hub diameter and for the bolting the we cut extrude 4 holes at a PCD of 115mm of dimension 8mm.
Material- Structural Steel.



Design in Solidworks

Meshing is an integral part of the engineering simulation process where complex geometries are divided into simple elements that can be used as discrete local approximations of the larger domain.

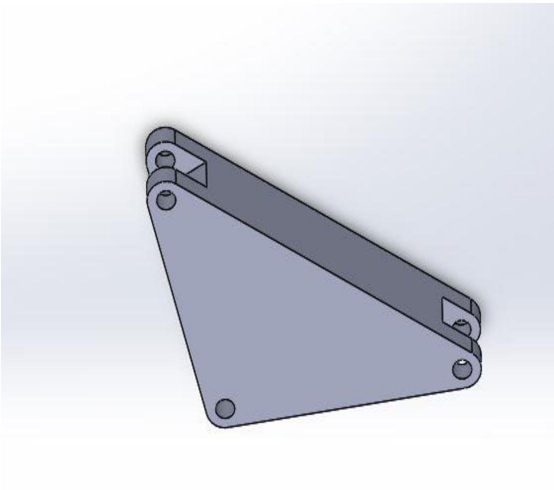
International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 9 Issue 11, November-2020



Nodes -10325, Elements -5219

Bell Crank - Once the step file of the bell crank imported to static structural in analysis, required boundary conditions are given. On the rocker side cylindrical support is provided and on other holes required push rod and damper forces are given. After this we got the total deformation and equivalent stress. Solution was then transfer to the setup file of Topology Optimization.

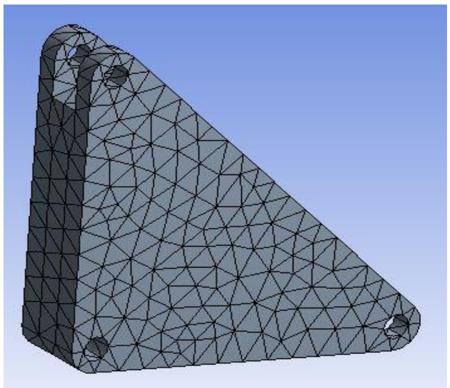
In topology optimization we set the iteration to 500 as default keeping the response type as compliance and minimize the mass in a static structural environment, keeping the retain percent to 50.



Design in Solidworks

Mesh of Bell Crank

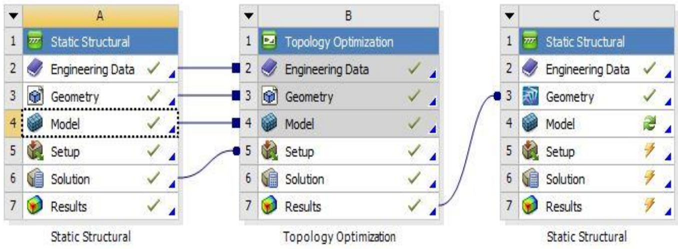
Nodes – 3705
Elements -1981



Geometry of Bell Crank

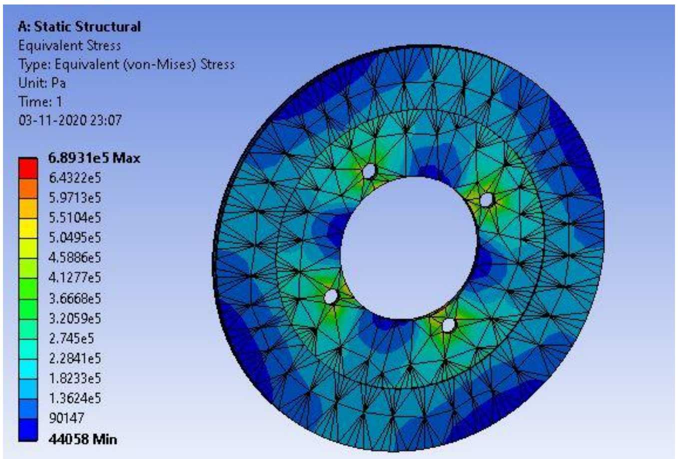
Basically, the geometry of the bell crank is decided in the lotus software where the rocker axis points were found according to suspension geometry. Diameter of holes are 8mm and thickness of the bell crank was kept 30mm.
Material- Aluminium 6061

ANSYS Process for both Rotor and Bell Crank



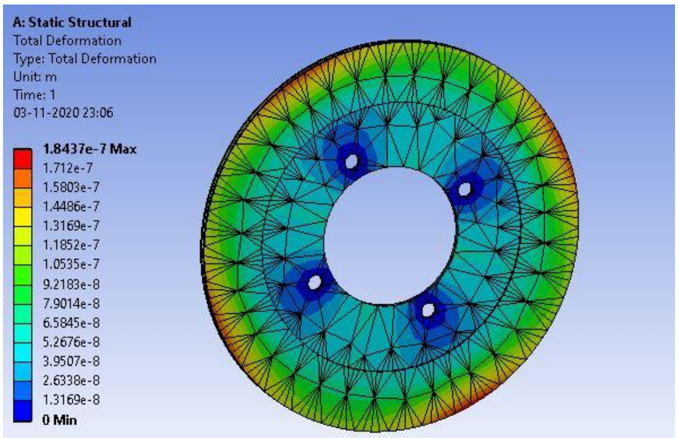
III. RESULT AND DISCUSSION

The results after the static structural analysis of Brake Rotor are obtained. The Total Deformation and Equivalent Stress:



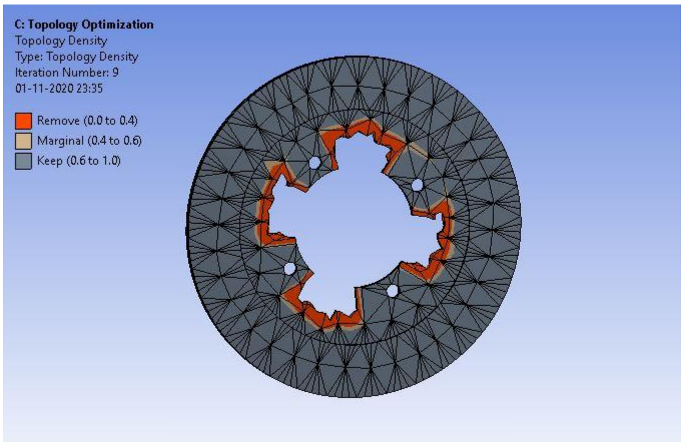
Equivalent Stress in Brake Rotor.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 9 Issue 11, November-2020

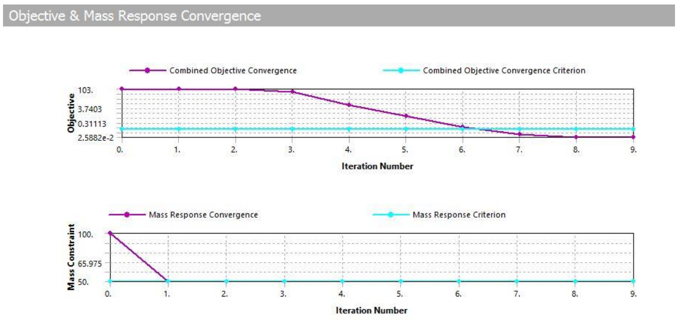


Total Deformation of Brake Rotor

This Result were then transferred to the TOPOPLOGY OPTIMIZATION PROCESS to get an optimal structure for the brake rotor.



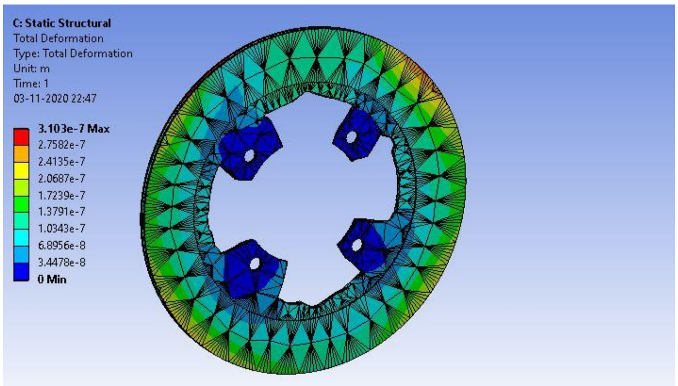
So basically, the unnecessary material is removed after 500 iteration and retain 50% mass. The red part gives us an idea of the removal area of the material.



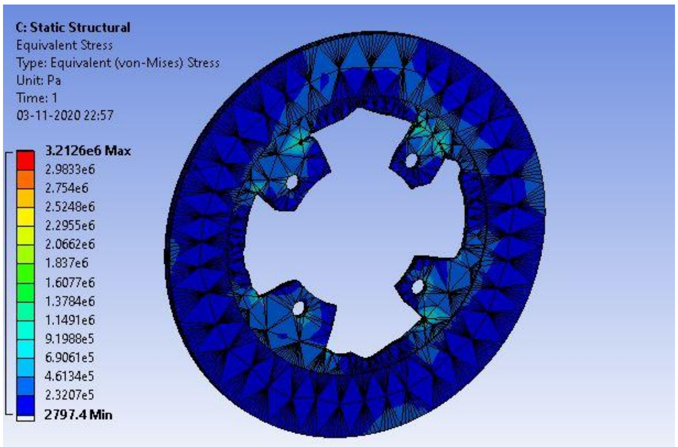
Mass Convergence Graph

Static Structural Analysis of the New Model:

So Basically, the constraints remain the same.



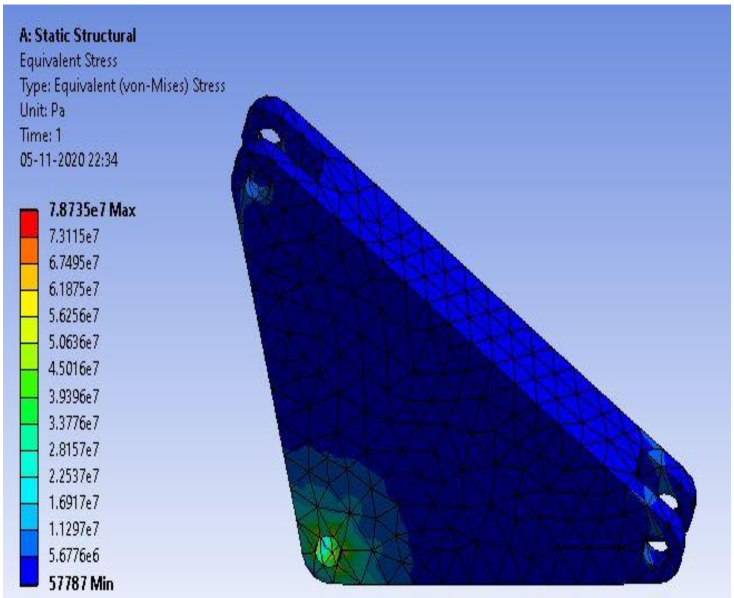
Total Deformation of the New optimized model.



Equivalent Stress of the New optimized model.

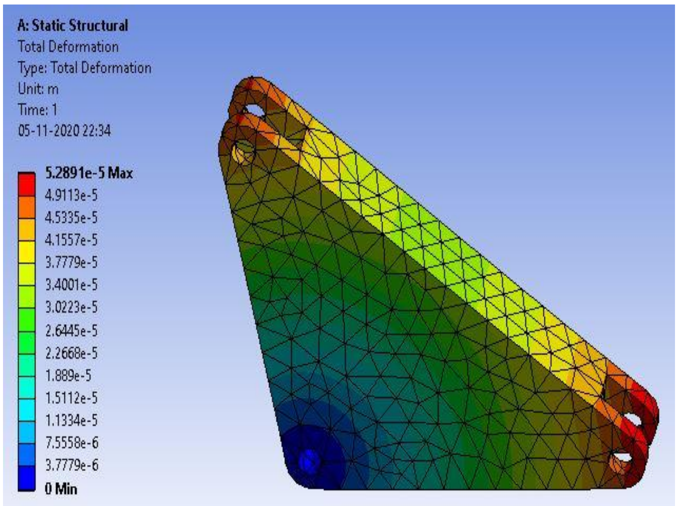
Now the result of bell crank

Result of total deformation and equivalent stress of Bell Crank are obtained.



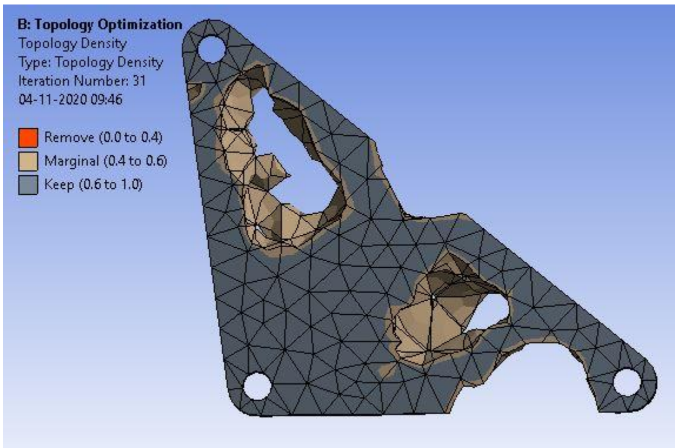
Equivalent Stress of the Bell Crank under the Boundary Conditions.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 9 Issue 11, November-2020



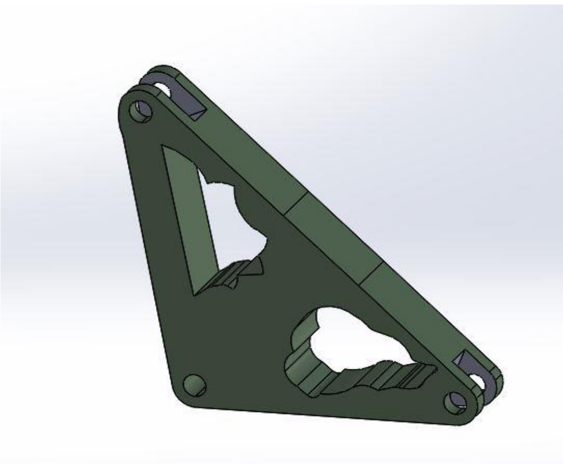
Total Deformation of Bell Crank.

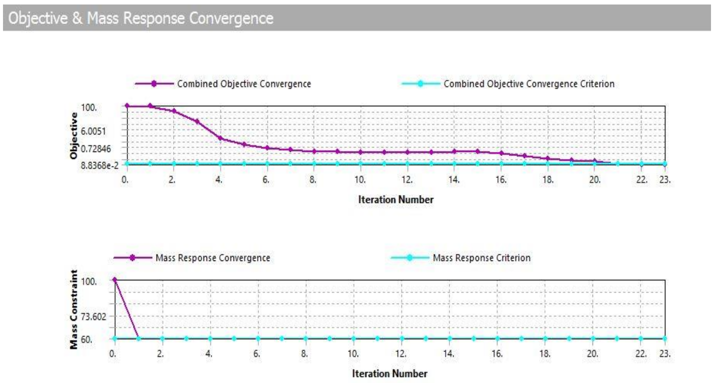
This Result were then transferred to the TOPOPLOGY OPTIMIZATION PROCESS to get an optimal structure for the bell Crank.



So basically, the unnecessary material is removed after 500 iteration and retain 50% mass. The red part gives us an idea of the removal area of the material.

So, after this the optimized model is remodeled again in ANSYS Spaceclaim for better result.

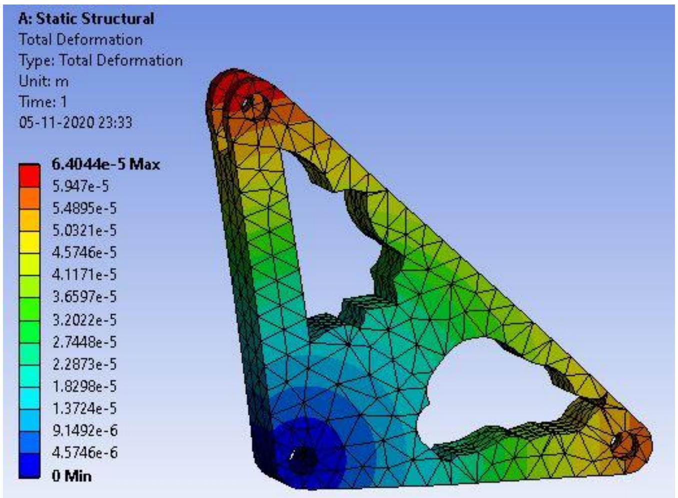




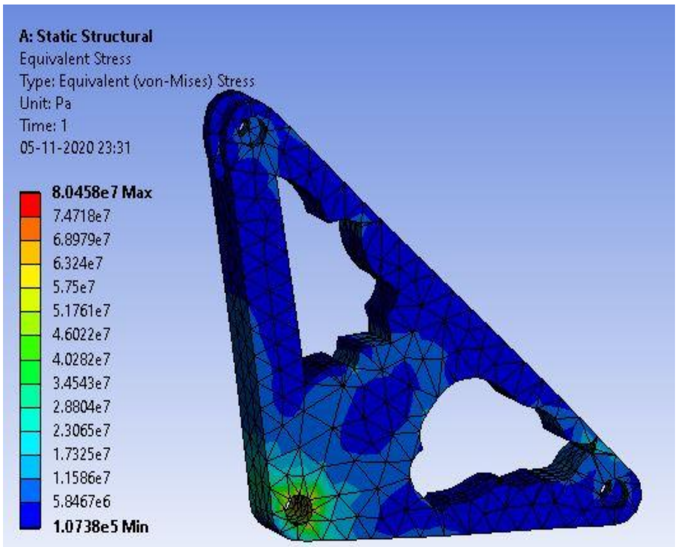
Mass Convergence Graph

Static Structural Analysis of New Model:

The Constraints remains the same.



Total Deformation of the New optimized model.



Equivalent Stress of the New optimized model.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 9 Issue 11, November-2020

IV. CONCLUSION

The Research study shows the total deformation and Equivalent stress of initial model and optimized model.

The following observation were:

1. No major change occurs in the total deformation of both the models whereas the optimized part is less in weight and deformation is close to the initial model.
2. Equivalent Stress on the initial model is less compare to the optimized model but the new model could easily sustain more stress.

The study that was perform can be expanded further according to the condition and we can achieve a more optimized design in future.

REFERENCES

[1] DheerajGunwant 2012, “Topology optimization of continuum structures using optimality criterion approach in ANSYS”, Thesis, G.B. Pant university of agriculture and technology.

[2] Huayang Xu, Liwen Guann, Xiang Chen, Liping Wang; Guide-Weight method for topology optimization of continuum structures including body forces.

[3] Prashant Kumar Srivastava, Simant, Sanjay Shukla,2017. Structural Optimization Methods: A General Review, International Journal of Innovative Research in Science, Engineering and Technology, Vol. 6, Special Issue 9

[4] TOPOLOGY OPTIMIZATION IN STRUCTURAL MECHANICS (G.I.N. ROZVANY ESSEN UNIVERSITY)