Optimizing Skatboard Truck

1. Intoduction:

The aim of this project is to optimise skatboard truck using Ansys structure workbench, Ansys topology optimisation and Solid Works for the designing and modifying the design. The optimisation is going to minimize the ratio (maximum stress / mass), so we will move in the folowing steps. Firstly; we will design using solid works a skateboard truck, secondly; we will apply normal loads on the skateboard truck using Ansys structure and check the stress, Thirdly; we will check which areas are not stresses and can be deleted using Ansys topology optimization. Finally, we will modify the body as possible using Solid Works to delete the unstressed areas and then check again the stresses using Ansys structure for the same loads.

The boundary conditions will be the same before and after the modifying, and will be the weight of an average person equals to 70 kg, and using a huge safety factor of 2. Thus in Newton the values will be approximated to 1400 [N], and assuming that the maximum mentioned load that can be apply on the skateboard will apply at least on two wheels, so the force on one wheel will be equals to 700 [N], it might be huge, but calculations using it will be safer.

The fixed parts will not be changed before and after the optimization, so the part can be used on the old set. The characteristics of the metal that used to build the skateboard truck is the characteristics of the following Stainless Steel;

Young's modulus of elasticity	200GPa
Poisson's ratio	0.30
Yield strength	280MPa
Tensile strength	460Mpa

2. The Methodology:

2.1 designing the skatboard truck:

As mentioned up, that the design is carried out using Solid Works. The desing is presented in the figure 1. The dwsign and dimentions hand been conducted to be real as possiple. The black parts are for the wheels and we will apply on them the mentioend forces. The openings in the middel of the part is for a secrew that will hold the part with a holder, which mountend under the skatboard. The little part ahead of the truck, has a hole in the holder to get in it, used to prevent the rotations in case the secrew not fully tight and also as director during the instalation.

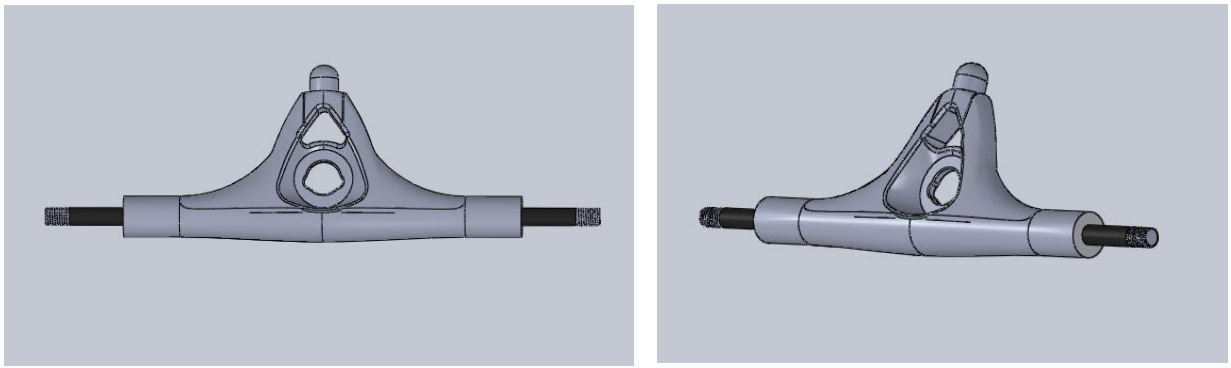

Figure 1 – The designed skatboard in Solid Works

2.2 Meshing the Skatbord in ansys structure:

The meshing took place in Ansys static structure, in Workbench, the settings and the mesh nodes and elements has been explained in the table 1, where used Adaptive size function and the softness in fine, and minimum edge length is 2e-6m. The mesh is presented in the figure 2.

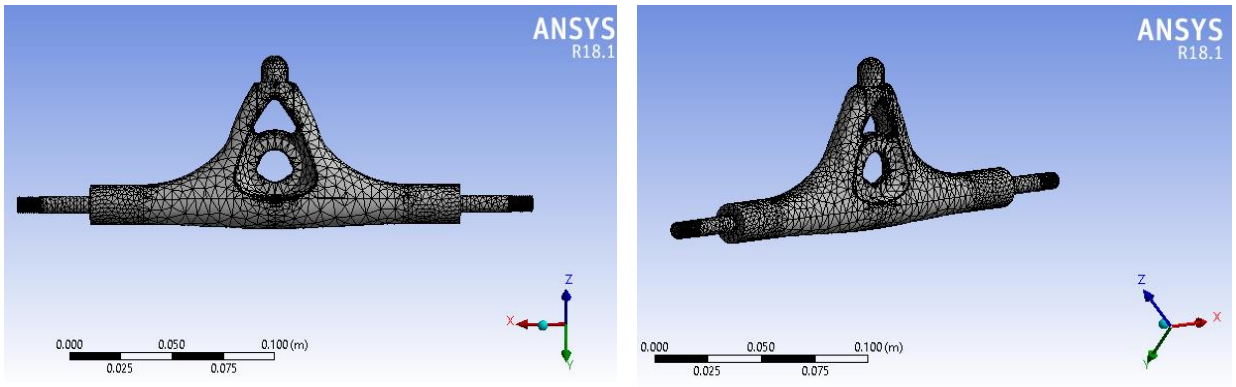

Figure 2 – Meshing the part in Ansys static structure

2.3 Applying the boundary conditions (Fixing and Loading):

We had fixed two places which are A and B in figure 3, and applied two forces each of 700 [N] on the place of the wheels, which are C and D. The fixing is full so the part cannot move in the sex DOF. The forces applying in the negative Y direction, according to the position of the holder.

Table 1 – Mesh settings

Sizing	
Size Function	Adaptive
Relevance Center	Fine
Element Size	Default
Initial Size Seed	Assembly
Transition	Fast
Span Angle Center	Fine
Automatic Mesh Based Defeaturing	On
Defeature Size	Default
Minimum Edge Length	2.1537e-006 m

Statistics	
Nodes	60390
Elements	34263

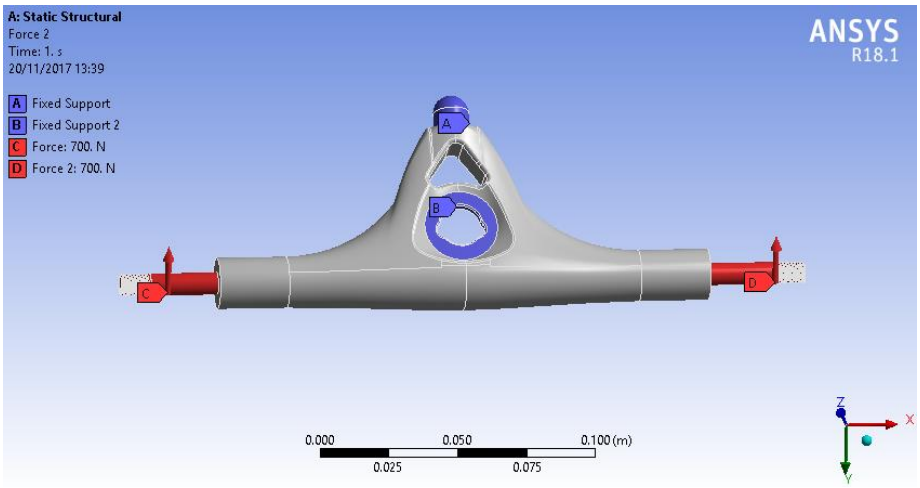

Figure 3 – Applying the boundary conditions

2.4 Solving and finding the stress and strains:

We have choosed that we need to calculate the equivalent Von-Mises stress and equvilant Von-Mises stain, besides of strian energy and total deformation. Then we started solving the problem. The results are shown in figure 4.

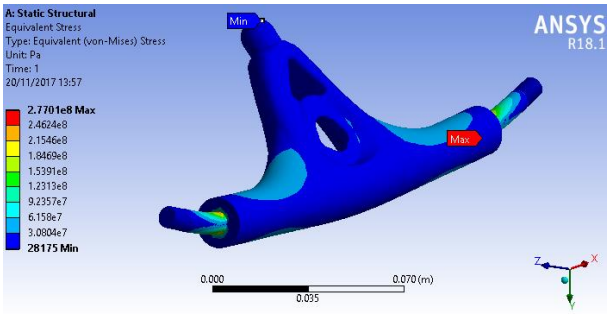

Figure 4.1 – Von-Mises stress

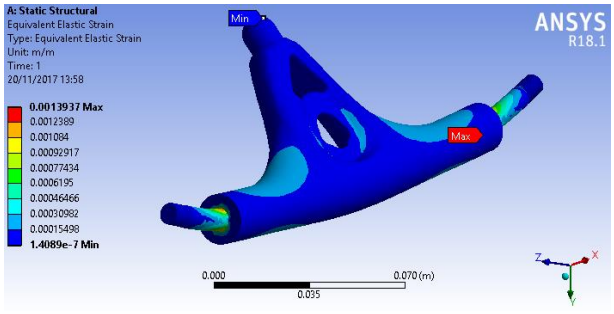

Figure 4.2 – Von-Mises strain

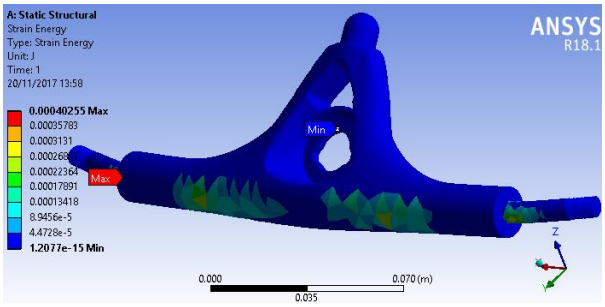

Figure 4.3 – Strain Energy

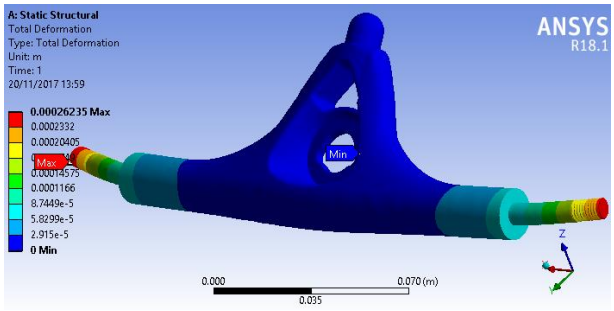

Figure 4.4 – Total deformation

We can see that the maximum stress is 227.09 [Mpa], and the minimum stress is 28.175 [Kpa], which they are in the safe side. The mass of the part can be get from Solid Works, and equals to 648 [Gram], Now we will optimise the part and then delete the unloaded places then redo the same stepes mentioned up and calculate the stress.

3. Optimisation the skatboard truck:

3.1 Optimizing the mass:

Using the topology optimization environment of Ansys, we insert the data in the in the table 2, and after finishing the whole iterations, the program shows us whish places can be deleted. The results showed in the figure 5.

Table 2 – settings of topology optimization

Definition	
Maximum Number Of Iterations	300.
Minimum Normalized Density	1.e-003
Convergence Accuracy	0.1 %
Solver Controls	
Solver Type	Program Controlled
Output Controls	
Store Results At	All Iterations

İn the figure 5, we can see two figures, on the left the part after and befor deleting the unloaded places, on the left, just the the part after deleting the unloaded places. İt seams that it changes the whole form and we cant fully depend on it, but we can take an idea about what places that are not needed, and manage to delete it as it possible. Therefore, we modified the part in Solid Works and the resultant part is presented in the figure 6.

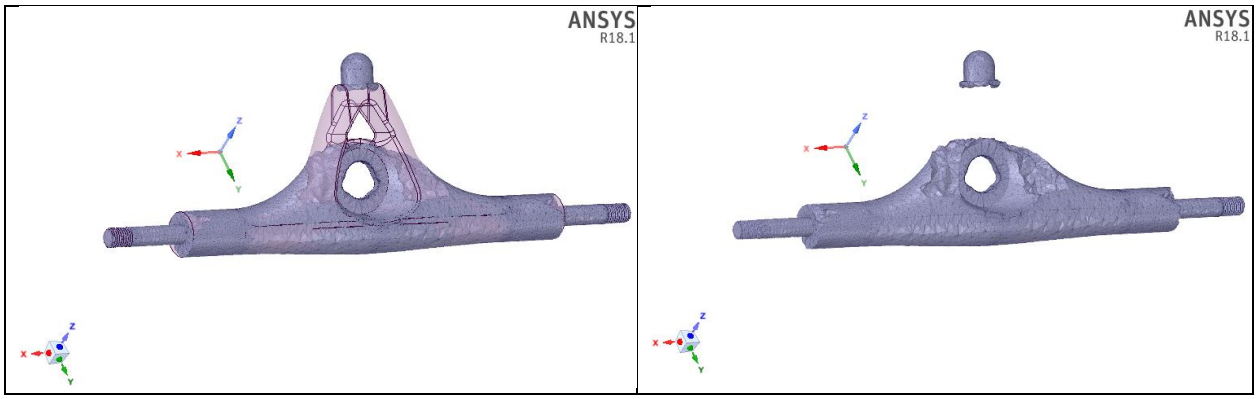

Figure 5 – The part after deleting the unloaded places

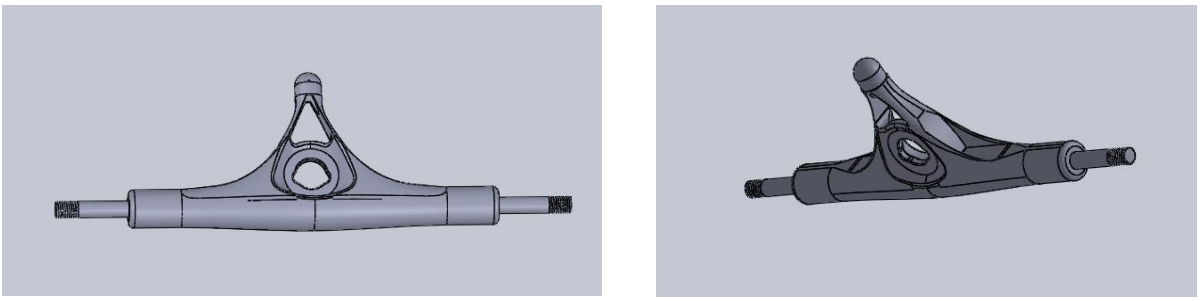

Figure 6 – The Skatboard Truck after modifiying in Solid Works

3.2 Stress of the optimised Skatboard Truck:

Repeating the steps in the section 2.2 to 2.4, we get the results in the figure 7. Form figure 7.1 and 7.2 we can see that there are more working parts, means our deleting of the unloaded places not just decreased the mass, but it also direct the stress to be loaded on the other places that where not in the part befor the optimization. From the figures we can see the maximum stress is 260.64 [Mpa], and the minimum stress is 549.73 [Pa]. The values are in the safe side. From Solid Works we can calculate the mass, and it equals to 540 [Gram]. Comparing with the mass of the first part that was 648 [Gram], we saved 108 [Gram] for a 33 [Pa] increment.

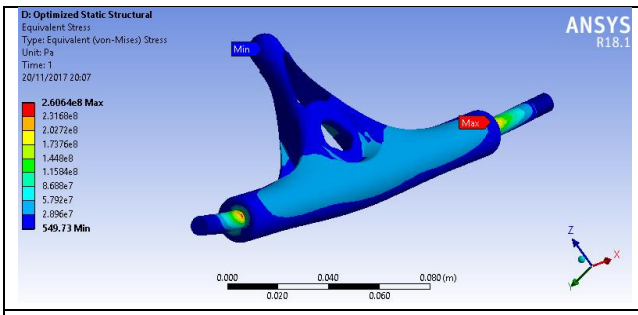

Figure 7.1 – Von-Mises stress

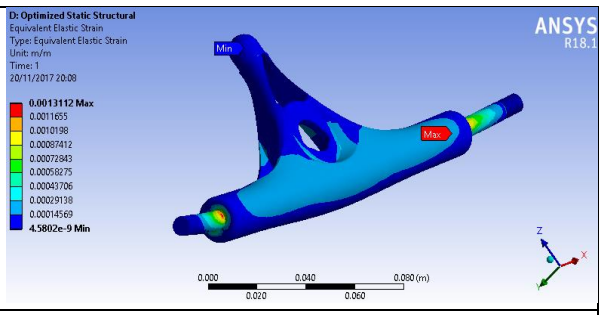

Figure 7.2 – Von-Mises strain

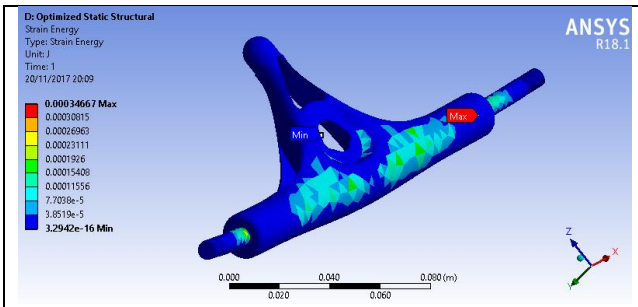

Figure 7.3 – Strain Energy

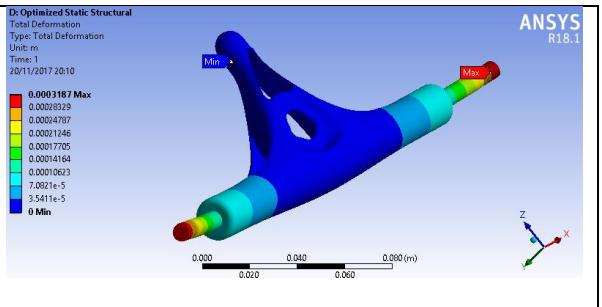

Figure 7.4 – Total Deformation

4. Conclusion:

İn the section 2 we have desinged a skatboard truck, then done a FEM study include meshing and finding the stress and strains. We found that the maximum stress is 227 [Mpa] for a weight thats equals to 648 [Gram]. Then we conducted optimization for the skatboard truck to reduce the mass, using software of Topology optimization in Ansys we defined the unloaded places, and using Solid Works we deleted these places as posible. The redid the study, the results showed that we have a maximum stress of 260 [Mpa] for a new weight of 540 [Gram]. That means we have decreses the mass up to 108 [Gram] for a small increasing in the maximum stress in the safe range 33 [Pa]. The results of the optimized part were resonable, where we can see that in the new part the stress are sepreded on the whole part and not focused in one area as in the part befor the optimization. The figure 8, shows the flow of the study in Workbench.

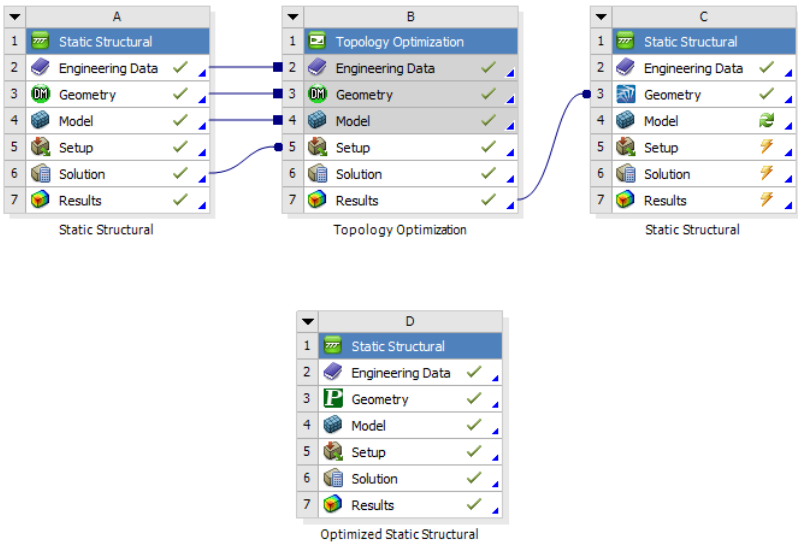

Figure 8- The flow of the Study in Ansys Workbench