Numerical Simulation of an Electric Lastmile's Delivery Vehicle

A Ramaraju[1]| M Manikanta Mahesh[1]| P Yahoshuva[1]| P Jagadesh[1]| T V Dharmaraju[2]

[1]UG Students, Department of Automobile Engineering, Godavari Institute of Engineering and Technology (A), Rajahmundry, Andhra Pradesh, India
[2]Assistant Professor, Department of Automobile Engineering, Godavari Institute of Engineering and Technology (A), Rajahmundry, Andhra Pradesh, India

Abstract: An automobile is made of subsystems like the body, the steering, the braking system, cited above the frame served as the base and support for all the other components that they are all attached to the frame.
So, the frame should be design accordingly, in order to support all loading conditions and ensure durability. The automobile industries are designing the chassis frames to increase stiffness and durability, the chassis frame of an electric vehicle is designed in CATIA software in order to check their loading capacity at the maximum point of the different material properties through the simulation of the model imported from the ANSYS software.The objective of this project is to provide strength and stiffness in order to support all loading conditions and ensure durability.
In our work, we analyze a ladder frame in ANSYS using different materials

KEYWORDS: Weight Reduction, Materials Properties used, CATIA, ANSYS

Check for updates

DOI of the Article: https://doi.org/10.46501/GIETAE05

Available online at: http://ijmtst.com/vol7si04.html

As per UGC guidelines an electronic bar code is provided to seure your paper

To Cite this Article: A Ramaraju; M Manikanta Mahesh; P Yahoshuva; P Jagadesh and T V Dharmaraju. Numerical Simulation of an Electric Lastmile's Delivery Vehicle. *International Journal for Modern Trends in Science and Technology* 2021, *7*, pp. 28-33. https://doi.org/10.46501/GIETAE05

Article Info. Received: 23 June 2021; Accepted: 2 July 2021; Published: 8 July 2021

I.INTRODUCTION

Numerical simulation of an electric last mile's delivery vehicle is also considered as the light weight commercial vehicle is design for the purpose of reduce of the fuel efficiency and it may be converted the IC engines to an electrical vehicle which can transport stock market goods very easily and also saving time and reaches short distance for delivers the goods to the different places.

Designing of an electrical vehicle is a very complex since it involves a lot of attributes like interior space, dynamic performance and also safety connectivity etc. The current article of an electric vehicle consists of energy consumption, battery capacity, electric motor which can capable of carrying the vehicle. Mode management, vehicle system and components simulation model simulation results and evaluation. The entry point in the electric vehicle design is the chassis frame with performance parameters such as top speed 0-100 kmph acceleration time and range of distance cover for one charge of point.

The chassis frame structural geometric analysis preparing the following steps are included for connecting and modifying beam profiles by using tools and features the beam structural profiles by imported the model of mechanical ANSYS software.

FEA software the chassis frame structure is meshed then find out simulation applying the boundary conditions for different loading cases and looks the stress and deformation contour. An effort is to study the function of the chassis frame is to provide strength and stiffness because it's the skeletal frame which can carry the loads placed on it. The maximum shear stress and the deformation under the maximum the maximum loads.

II. DESIGN AND ANALYSISWORK

The chassis frame is static structural using different materials properties in analysis software for which can more suitable to find out the rectified the deformation in construction. This model is saved IGES format which can be directly imported ANSYS workbench. The chassis frame is the example of rectangular box model cross section member imported to ANSYS workbench.

In these mechanical static structural properties, we can use the ANSYS software for find out the loads act which can withstand for the shock, twist vibrations and other stress according to the road conditions. Chassis strength is important aspect that needs to be considered in the study of a road vehicle design.

Chassis frames are the skeletal structure for road vehicle design. Since the strength of chassis can be affected on stability and safety of vehicle. Because most of load are fully distribute along the chassis. Its will be have some critical point due to section that have more load over the beam section especially driver weight.

Fig 1: PREVIEW OF THE CHASSIS FRAME

1)SIMULATION 3D MODEL OF THE CHASSISFRAME:

This model is shown the geometric views of coordinate system when we can create 3D model of an electrical vehicle chassis frame in Catia software it shows in all directions. Finite element analyzing model and time as follows three dimensional directions. Modelling of the chassis frame structure consists of three types i) C type cross section chassis frame ii) I type cross section chassis frame iii) Rectangular box type cross section chassis frame is used. In this model rectangular box type cross section is used for lightweight and load carrying capacity in all equaldirections.

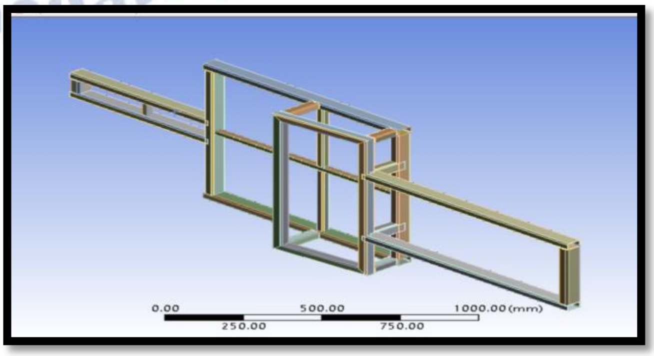

Fig 2: GEOMETRIC VIEW OF 3D MODEL IN CAD

2) MESHING OF CHASSIS FRAME:

Finite element analysis software typically used a cad represents the physically model and breaks it down into small pieces called finite element. The mathematical represents the higher quantity of the mesh in the physical model shown enhanced the meshing part is completed on the model using hex mesh in the static structure in the Ansys software.

From the analysis software the chassis frame Ansys is used which can includes the important three steps i) pre-process ii) solution iii) post process which can give exact results and using different types of material properties. Through this numerical simulation model, we can apply boundary condition six different types of loads which can apply the stress, stiffness and the deformation of the chassis frame in the period of lifespan such as i) Frontal impact test ii) Side impacttest iii) Roll over impact test iv) Rear impact test v) Torsion impact test and iv) Frontal bump test based on the meshing we can analyzing the material properties in the Ansys R19.2 software.

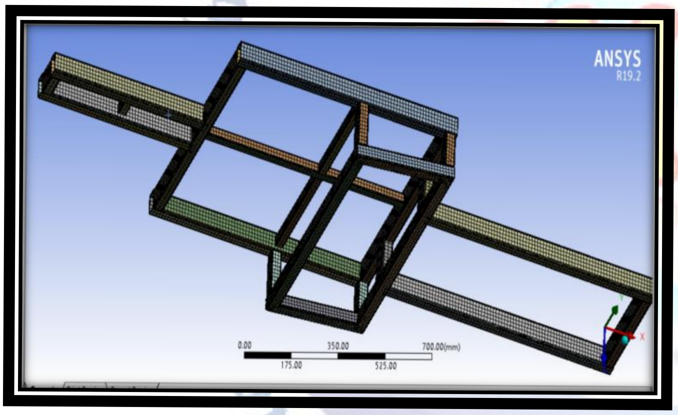

Fig 3: MESHING OF THE CHASSIS FRAME

3) STANDARD EARTH GRAVITY:

The standard acceleration due to gravity, sometimes abbreviated as standard gravity, usually denoted by g_0 or g□.

The nominal gravitational acceleration of an object in a vacuum near the surface of the Earth. It is defined by standard as 9.80665 m/s².

Table-1 Standard earthGravity:

Details of "Standard Earth Gravity"	
Scope	
Geometry	All Bodies
Definition	
Coordinate System	Global Coordinate System
X Component	0. mm/s² (ramped)
Y Component	0. mm/s² (ramped)
Z Component	-9806.6 mm/s² (ramped)
Suppressed	No
Direction	-Z Direction

4) FIXED SUPPORT:

In the ANSYS software the geometry selection it is fixed at the front end and rear ends are two fixed supports.

Table-2 Fixed support

Details of "Fixed Support 2"	
Scope	
Scoping Method	Geometry Selection
Geometry	13 Faces
Definition	
Type	Fixed Support
Suppressed	No

5) FORCE ACTING:

After the fixed support then we can calculatethe displacement condition of the structure is considered as the bounday and the boundary conditions around the structure as dislacement because of the force acting from the magnetude direction towards the downward as shown -9810.00 N results will be done in mechanical ansys software.

Table-3 Force

Details of "Force"	
Scope	
Scoping Method	Geometry Selection
Geometry	13 Faces
Definition	
Type	Force
Define By	Components
Coordinate System	Global Coordinate System
X Component	0. N (ramped)
Y Component	0. N (ramped)
Z Component	-9810. N (ramped)
Suppressed	No

6)ANALYSIS SETTING:

Here we are applying the engineering data for material properties and also considering the static structure analysis in ANSYS software imported from chassis frame in Catia software. The following steps are involved in above.

Table -4 Analysis setting

Details of "Analysis Settings"	
Step Controls	
Number Of Steps	1.
Current Step Number	1.
Step End Time	1. s
Auto Time Stepping	Program Controlled
Solver Controls	
Solver Type	Program Controlled
Weak Springs	Off
Solver Pivot Checking	Program Controlled
Large Deflection	Off
Inertia Relief	Off
Rotordynamics Controls	
Restart Controls	
Nonlinear Controls	
Output Controls	
Analysis Data Management	
Visibility	

III. METHODOLOGY

MATERIALS USED IN CHASSIS FRAME:

Most cars on the street are using steel as the material of choice to build the chassis, this include monocoque and traditional ladder frame chassis. Steel found its way to cars that demands high durability and reliability such as passenger cars, vans, trucks, and SUVs. High performance cars use aluminium in least structural parts of the chassis like middle roof brace and front and rear hangover section this doesn't applicable to older high-performance cars. Materials used for chassis frame for find out the stress and deformation properties such as Aluminium alloy, grey cast-iron, structural steel, stainless steel, titanium alloy.

The following material properties shown in figure for Deformation:

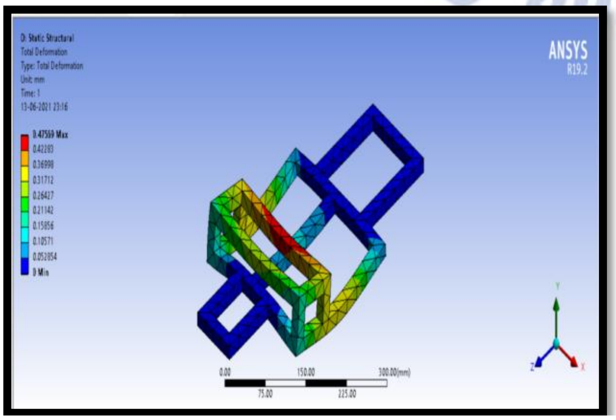

Fig 4: Aluminum alloy, it shows the maximum deformation of the chassis frame is 0.47569 Mpa.

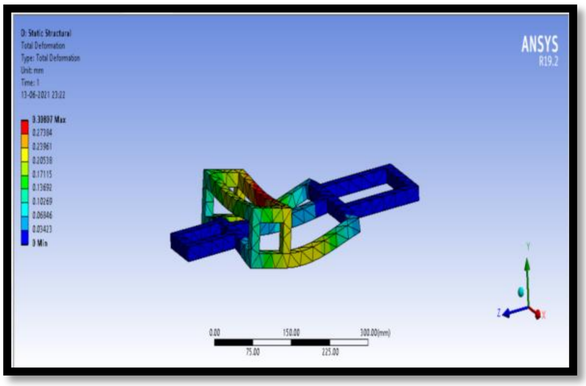

Fig 5: Grey cast iron,

It shows the maximum deformation of the chassis frame is 0.30807 Mpa.

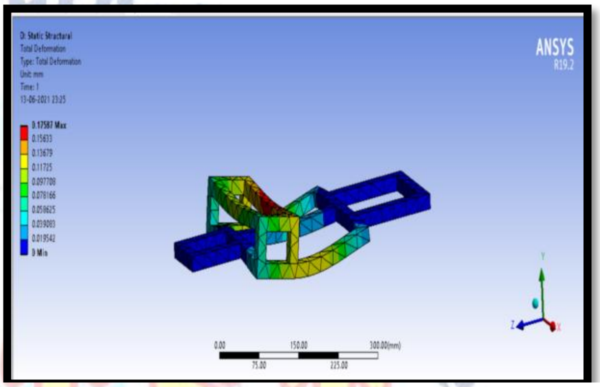

Fig 6: Stainless steel,

It shows the maximum deformation of the chassis frame is 0.18787 Mpa.

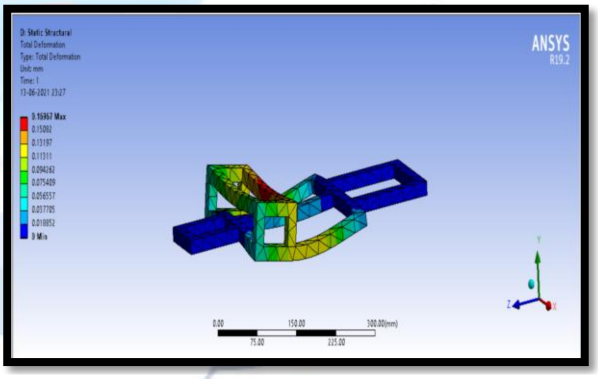

Fig 7: Structural steel,

It shows the maximum deformation of the chassis frame is 0.16967Mpa.

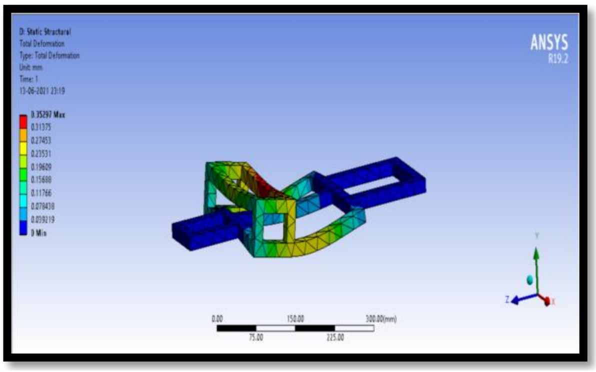

Fig8: Titanium alloy, it shows the maximum deformation of the chassis frame is 0.3529Mpa.

Table-5 Materials properties Maximum Deformation

S.no	Material properties	Maximum deformation
1	Aluminum alloy	0.47569Mpa
2	Grey cast iron	0.30807Mpa
3	Stainless steel	0.18787Mpa
4	Structural steel	0.16967Mpa
5	Titanium alloy	0.3529Mpa

The following material properties shown in figure for Stress:

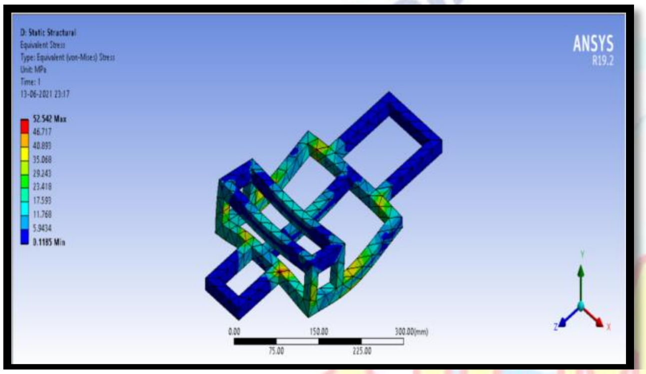

Fig 9: Aluminum alloy, it shows the maximum stress of the chassis frame is 52.542 Mpa.

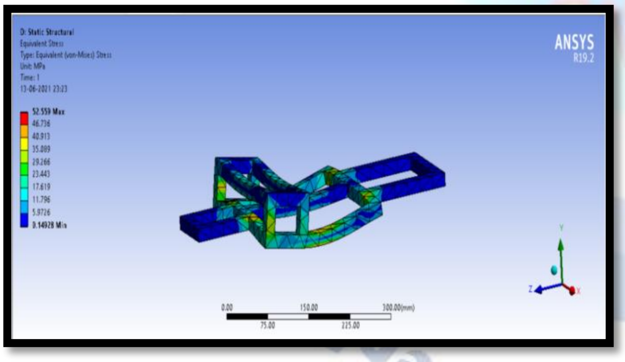

Fig10: Grey cast iron, it shows the maximum stress of the chassis frame is 52.559 Mpa.

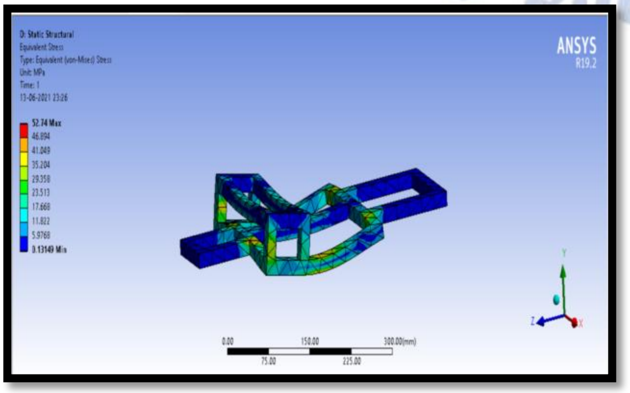

Fig 11: Stainless steel, it shows the maximum stress of the chassis frame is 52.74Mpa.

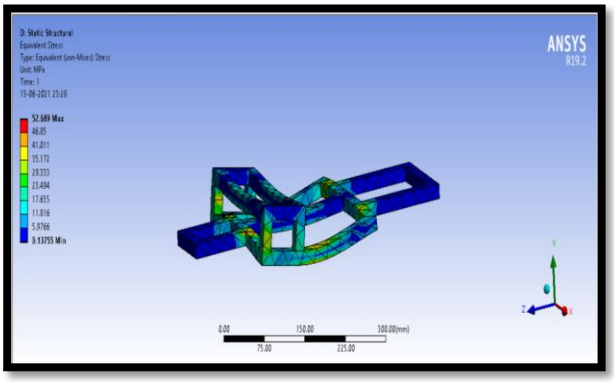

Fig 12: Structural steel, it shows the maximum stress of the chassis frame is 52.89 Mpa.

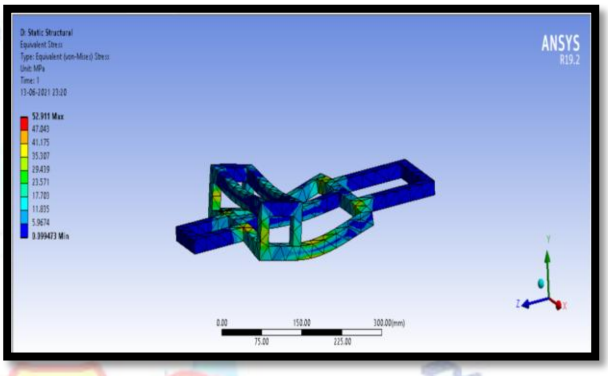

Fig13: Titanium alloy, it shows the maximum stress of the chassis frame is 52.911 Mpa.

Table -6Materials properties Maximum Stress

S.no	Material properties	Maximum stress
1	Aluminum alloy	52.542Mpa
2	Grey cast iron	52.559Mpa
3	Stainless steel	52.74Mpa
4	Structural steel	52.89Mpa
5	Titanium alloy	52.911Mpa

VI. CONCLUSION

In this paper the chassis frame is design in Catia software and imported into finite element analysis to find out the chassis loads acting on the maximum stress is Titanium 52.911MPa and maximum deformation is 0.3529MPa as to considered the material cost is highly cheaper so far better to take structural steel is 52.89MPa and deformation is0.16967Mpa also cost is less as compare to remaining materials.

REFERENCES

1. G V Arun "Structural Analysis of Chassis using AISI 4130 and AA 7068". 2021. *IOP Conf. Ser.: Mater. Sci. Eng.* **1059** 012034.

2. Yagyansh Mishra, "Design & Analysis of Ladder Frame Chassis" e-ISSN: 2395-0056 Volume: 07 Issue: 08 | Aug 2020 www.irjet.net p-ISSN: 2395-0072
3. ShwetankAvika, "Design and fatigue analysis of front axle beam of a heavy-duty truck using ANSYS". 14 February 2020,doi.org/10.1016/j.matpr.2020.02.901
4. S. Satish, "Design and analysis of mild steel mini truck body for increasing the payload capacity" 23 June 2020, doi.org/10.1016/j.matpr.2020.06.459
5. Hongshen Zhang, Gan Huang and Dali Yu, "Numerical modelling for the frame structure of light van-type electric truck",2020,Vol.103(2)1–27.DOI:10.1177/0036850420927204
6. Yuuji Hanaoka"Numerical Simulation of Door FrameDeformation due to Aerodynamic SuctionForce". Downloaded from SAE International by North Carolina State Univ, Tuesday, September 18, 2018.
7. Monika S. Agrawal, "Finite Element Analysis of Truck Chassis Frame"Volume: 02 Issue: 03, June-2015,e-ISSN: 2395-0056.