International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 08 Issue: 07 | July 2021 www.irjet.net p-ISSN: 2395-0072

Design and Analysis of Four Wheeler Vehicles Frame

Aman Singh[1], Alok Yadav[2], Abhishek Kumar Singh[3], Akash Mishra[4], Arjun Singh[5], Professor Kamaleshwar Dhar Dwivedi[6]

[1-5]Dept. of Mechanical Engineering, BBDITM, Lucknow, India

Abstract: *Future coming generation stepping towards future development then we have to work on improvement of technology. The leading step of development is more important in automotive sector. This project is used to determine the static stress and strain concentrated on the frame structure.*

SolidWorks software (3-D) use for presenting and fabricating basic chassis frame structure. The designing of part are done at SolidWorks and static loading test is conducted in the environment of ANSYS.

The developed chassis frame is tested at the loading of 150 kN as point load at different joints. The structure of chassis is fabricated by structural steel material which has a definite structure. The chassis structure is modified to a good chassis structure. The design of chassis with satisfying strength is the ambition of paper.

INTRODUCTION

Chassis act as the main frame of the body structure of the vehicle. It is important part of the vehicle considered when designing a vehicle. Chassis is lower part of the vehicle which include frame, engine, drive line and suspension. Generally we use three type of chassis structure ladder chassis frame, space chassis frame and monocoque chassis frame.

Things that always considered during the design of the chassis structure is such as strength and stiffness of the chassis. Chassis structure should be strong as it tends to be subjected to static stress and strain and also vibration due to various dynamic exitation. This things are very important as static analysis is important in determining the safety of the structure. Also vibration on chassis can effect the vehicle in various aspect like during ride, stability of the vehicle and safety of vehicle. It can also caused high stress concentration at various point fatigue on the structure and loose in the joint of structure. Thereforethe vibration analysis is vital to be done on chassis, ensuring the safety and stability of vehicle.

This whole project is about developing four wheeler vehicle chassis and for performing vibration analysis and static analysis on the chassis develop or design of chassis will utilized the space frame chassis type and material chosen is structural steel. Before the process of developing, a rapid prototyping is drawn using solid work software in order to perform the FEA on design chassis structure. Material used is structure steel and we have to assigned it in the software to increase accuracy of the results. Static analysis is done on modeling of chassis using simulation software to determine the distribution of static stress on the structure after the load force in to the simulation.

The safety factor for chassis can be calculate the analysis.
The objective of this project are started :

1. To develop a chassis structure with reduced vibration.
2. To identity and analysed the static stress.

Chassis

As we know about the important of chassis in any vehicle. It is the whole support system. All load directly or indirectly action it. So designing the chassis one of the most important aspect for Designing Engineers. The passage of time. We have various changes and improvement in chassis structure. The chassis is appeared to be most important component of vehicle. It gives strength and stability to vehicle under different condition. Our chassis has been design keeping in mind the load of 150 KN. It has been structural steel used as its material. Structural steel is special category of steel which is used to make different shapes and it can be elongated easily in a beam for particularcross section.

It has carbon contain of up to 2.1 % of weight increasing the amount of carbon in composition of steel result in increasing strength and decreasing low ductility. Just for giving and over view of structure steel , hereis different profile formed with specific cross section.

1. Beam :- An I shaped cross section.

2. Shape :- Half a flung going in opposite direction as of in Z.
3. Rod :- A round or square long piece of steel.
4. Angle :- L shaped structure.

As the ductility of structure Steel different shapes , thickness and size can be costumed out of it to meet the customer demand.

Impart of Model and Meshing

In Ansys software first step in preprocessingis to import chassis structure and input of material is given of structural steel a mesh generation is created. After the boundary condition are applied on chassis structure. Our job is to conduct stress analysis and for that mesh was already developed with several nodes. Various figures related to this is given below.

Material Selection

Material which is selected for this project is widely used in manufacturing and it is structuralsteel.

Structural Steel

It is generally used for mapping construction material in a Variety of shape and structure.

Table 1

Material	
Assignment	Structural Steel
Nonlinear Effects	Yes
Thermal Strain Effects	Yes

Properties										
Volume	6.8901e-008 m³		8.4673e-008 m³		9.8808e-008 m³					7.0539e-008 m³
Mass	5.4088e-004 kg		6.6469e-004 kg		7.7564e-004 kg					5.5373e-004 kg
Centroid X	1.9935e-002 m	-4.6065e-002 m	4.4351e-003 m	-3.0565e-002 m	-3.1565e-002 m	-5.2565e-002 m	2.6435e-002 m		5.4351e-003 m	-5.2565e-002 m
Centroid Y	3.5798e-003 m		-4.5776e-003 m		-1.2816e-002 m	-1.7586e-002 m	-9.5856e-003 m	-1.7586e-002 m	-1.2816e-002 m	-9.5856e-003 m	-1.8384e-002 m
Centroid Z	-3.1626e-002 m	-5.6259e-003 m	-4.4626e-002 m		-0.39817 m	1.3423e-002 m			-0.39817 m	1.3423e-002 m	-0.15913 m
Moment of Inertia Ip1	1.1295e-009 kg·m²		1.2166e-009 kg·m²		1.3391e-009 kg·m²					1.0952e-009 kg·m²
Moment of Inertia Ip2	3.2003e-009 kg·m²		6.877e-009 kg·m²		1.1215e-008 kg·m²					3.8548e-009 kg·m²
Moment of Inertia Ip3	3.2e-009 kg·m²		6.8762e-009 kg·m²		1.1214e-008 kg·m²					3.8542e-009 kg·m²

Table 2

Design

In this section we like to describe about the design of our vehicle chassis frame.

We design components of chassis frame, it has various different components:

1. Chassis Battery Plate studs
2. Chassis Battery Holder Plate
3. Battery Holder Plate
4. Chassis Front Bumper Mount
5. Chassis Front Bumper
6. Chassis Front skid Plate
7. Chassis Mid Brace
8. Chassis rear bumper
9. Chassis servo box Mount Plate
10. Chassis servo box
11. Chassis servo mount front
12. Chassis servo mount rear
13. Chassis shock mounts
14. Chassis side extensions
15. Chassis Sides
16. Socket Button head cap screw
17. Socket countersunk cap screw.

These are design in right dimension with certain limit.

FINITE ELEMENT ANALYSIS

Finite element analysis is computer based numerical techniquefor calculating the energy and behavior of

engineering structure. It is used to calculate deflection, pressure vibration, buckling behavior and other phenomenon.

MESHING

The basic idea of meshing FEA is to make calculation of finite number of element and then interpolate that result for entire domain.

As we already knows that any continuous surface has infinite degree of freedom and its not possible to solve that problem.

So, we create finite element method to reduce the complexity with the help of discretization or meshing. We use analysis Ansys software for that.

FRAME DESIGN STATIC ANALYSIS

After designing the model when we choose to go for analysis we need to go under process:

Step1: Discretization of structural domain.

Step2: Selection of proper interpolation mode.

Step3: Derivation of element stiffness matrices and load vector.

Step4: Assemblage of element equation to obtain overall equilibrium equation.

Step5: Solution of system equation to find nodal value of displacement.

Step6: Computation of element of strain and stress from the known model displacement.

The design of model with applied load are shown below:

In this chassis model all applied load are point load and applied at the different position of the frame. There are seven point in model where loads are applied and points are named as

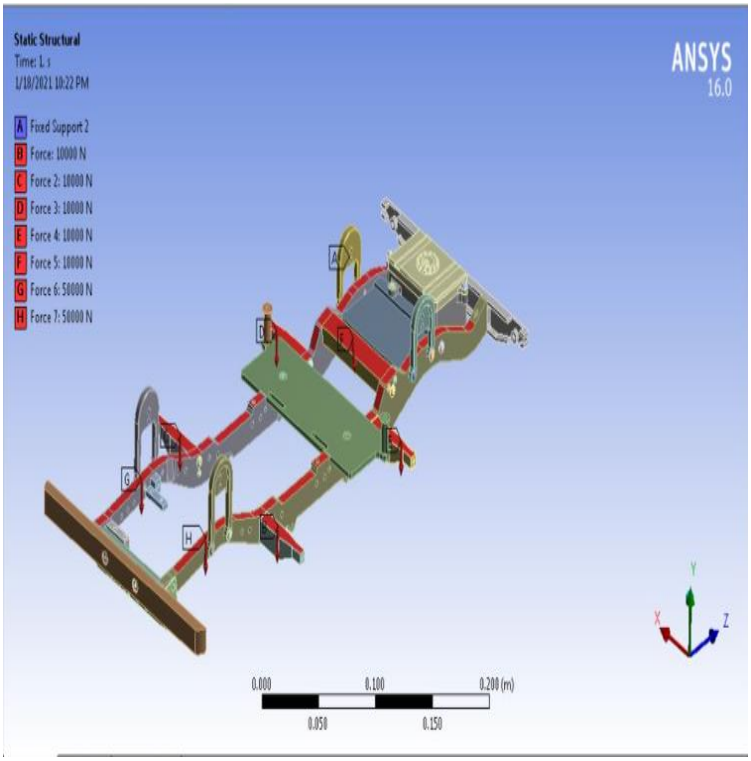

Figure 2

B,C,D,E,F, G, H. "A" point is a fixed support point of the frame model. At point 'B', 'C', 'D', 'E' and 'F' the applied load on each point is 10000 N. At point 'G' and 'H' applied load on each is 50000 N. The total amount of applied is 150kN. The frame model is analyzing under the static loading condition.

RESULT

The table shows result of model are as:

Object Name	Total Deformation	Normal Stress	Equivalent Elastic Strain	Strain Energy	Equivalent Stress	Equivalent Elastic Strain 2	Maximum Shear Elastic Strain
State	Solved						
Scope							
Scoping Method	Geometry Selection						
Geometry	All Bodies						
Definition							
Type	Total Deformation	Normal Stress	Equivalent Elastic Strain	Strain Energy	Equivalent (von-Mises) Stress	Equivalent Elastic Strain	Maximum Shear Elastic Strain
By	Time						
Display Time	Last						
Calculate Time History	Yes						
Identifier							
Suppressed	No						
Orientation		X Axis					
Coordinate System		Global Coordinate System					
Results							
Minimum	0. m	-8.3923e+009 Pa	2.3335e-011 m/m	2.9801e-022 J	4.2708 Pa	2.3335e-011 m/m	3.197e-011 m/m
Maximum	1.2207e-002 m	1.3193e+010 Pa	0.11542 m/m	0.53317 J	2.0826e+010 Pa	0.11542 m/m	0.15525 m/m

Table 3

CONCLUSION

In this frame model, the following factors are analyzed names are as normal stress, total deformation, elastic strain, strain energy, equivalent stress, maximum shear elastic strain. The maximum total deformation is 1.2207 e-002m at part 55 and minimum is 0m at part45 of discrete. The maximum normal stress occur at part 54 is 1.3193 e+010 Pa and minimum at part 41 is 8.3923 e+009 Pa. This model which suitable to bear the 150kN easily, deformation factors are very low in this model after applied load.

REFERENCES

- Rajappan, R., &Vivekanandhan, M. (2013). Static and model analysis of frame by using FEA. The International Journal of Engineering (IJES), 2(2), 63-73.
- Solasa, G., Saradeep, N.S.D. Prasad, T.R.K., &Babu, G.S. (2013). Model Analysis of chassis. International Journal of Engineering and advanced Technology (IJEAT), 2(4), 789- 795.
- William B. Riley and Albert R. George, "Design, analysisand testing of a formula SAE Car chassis", SAE Technical paper series 2002-01-3300, 2 December 2002.
- W.Schutz (1996). "A history of fatigue. Engineering Fracture Mechanics" 54: 263- 300.
- W.J.M. Rankine (1842). "On the causes of the unexpected breakage of the journals of railway axles, and on the means of preventing such accidents by observing the law of continuity in their construction". Institution of civil Engineers, Minutes of Proceedings, 105-108.