OPEN ACCESS

International
Journal
Of Modern Engineering Research (IJMER)

Static Analysis of Go-Kart Chassis by Analytical and Solid Works Simulation

Rahul Thavai[1], Quazi Shahezad[2], Mirza Shahrukh[3], Mukri Arman[4], Khan Imran[5]

[1](Department of Mechanical Engineering, AIKTC/ University of Mumbai, Maharashtra, India)
[2.3.4.5](Department of Mechanical Engineering, AIKTC/ University of Mumbai, Maharashtra, India)

ABSTRACT: *This paper aims to do modelling, simulation and performing the static analysis of a go kart chassis consisting of Circular beams. Modelling, simulations and analysis are performed using 3-D modelling software i.e. Solid Works and ANSYS according to the rulebook provided by Indian Society of New Era Engineers (ISNEE) for National Go Kart Championship (NGKC-14).The maximum deflection is determined by performing static analysis. Computed results are then compared to analytical calculation, where it is found that the location of maximum deflection agrees well with theoretical approximation but varies on magnitude aspect.*

Keywords: *3D-Cad Model, ANSYS, Chassis, FEA, Go-Kart, SolidWorks.*

I. Introduction

The chassis takes a load of the operator, engine, brake system, fuel system and steering mechanism, so chassis should have adequate strength to protect the operator in the event of an impact. The driver cabin must have the capacity to resist all the forces exerted upon it. This can be achieved either by using high strength material or better cross sections against the applied load. But the most feasible way to balance the dry mass of chassis with the optimum number of longitudinal and lateral members. The chassis must be constructed of steel tubing with minimum dimensional and strength requirements dictated by ISNEE. The NGKC vehicle development manual also restricts us about the vehicle weight, shape, size and dimensions. Circular cross-section is employed for the chassis development as it helps to overcome difficulties as increment in dimension, rise in the overall weight and decrease in performance due to reduction in acceleration[1] .

Circular section is always a preferred over other cross section become it resist the twisting effects. Circular section is selected for torsional rigidity. Design objectives of chassis are:-

- Provide full protection of the driver, by obtaining required strength and torsional rigidity, while reducing weight through diligent tubing selection
- Design for manufacturability, as well as cost reduction, to ensure both material and manufacturing costs are competitive with other Go Karts
- Maintain ease of serviceability by ensuring that chassis members do not interfere with other subsystems

This study attempted to analyse stress on the chassis design using finite element analysis (SOLIDWORKS). This is important because the simulation data are useful for further design improvement and subsequently leads to cost effectiveness.

II. Material And Methodology

2.1 Material Selection:

The chassis is made up of AISI-1018 which is a medium carbon steel. This material was selected due to its good Combination of all of the typical traits of Steel – high tensile strength, ductility, light weight, better weldability and comparative ease of machining[2][8].
The properties of the material are presented in Table. 1

Table I: Material properties

Modulus of elasticity (MPa)	200
Density	7.7 to 8.03
Poisson ratio	0.285
Yield strength (MPa)	386
Tensile strength (MPa)	634

2.2 Methodology:

The main objective of the study is to obtain a maximum deflection of chassis under static condition. The overall study flow chart is as in Figure. 1[7]

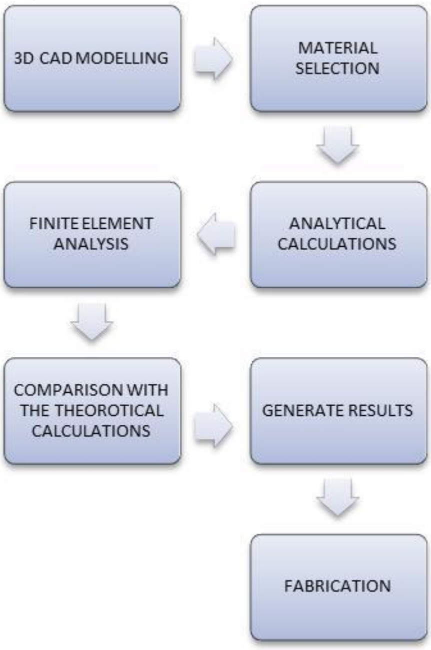

Figure.1: Flowchart of paper methodology

2.3 Modelling:

3-D modelling was done using Solid Works software as shown in Figure.2 [4]

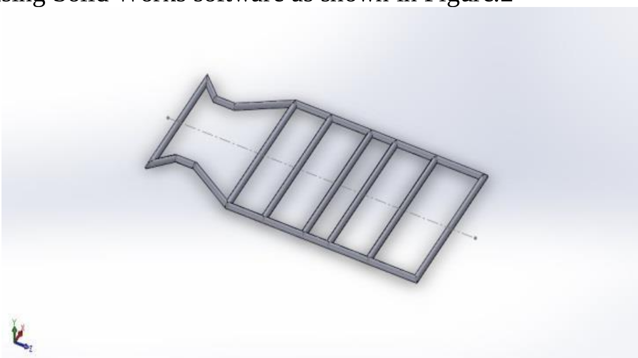

Figure.2: 3-D CAD Model

2.4 Finite element analysis:

The safety and the strength of chassis are important issues for its structure. To meet these requirements, it is essential to perform a static analysis on the chassis. Static analysis was done using finite element method as it is an effective and efficient approach. SolidWorks software was used for finite element analysis. [5]

2.5 Meshing:

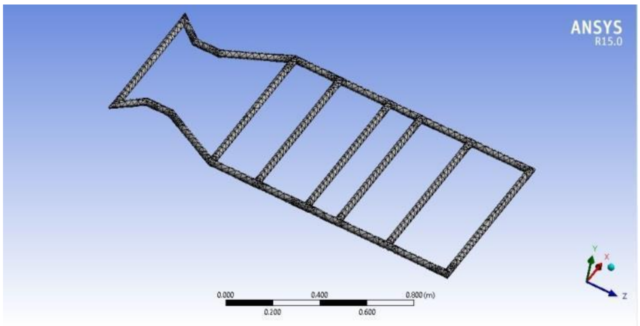

Figure.3: Meshing of Frame

2.6 Boundary conditions:

Boundary conditions selected were two area of fixed point, in which one is steering knuckle joint and another is bearing on rear axle. [5]

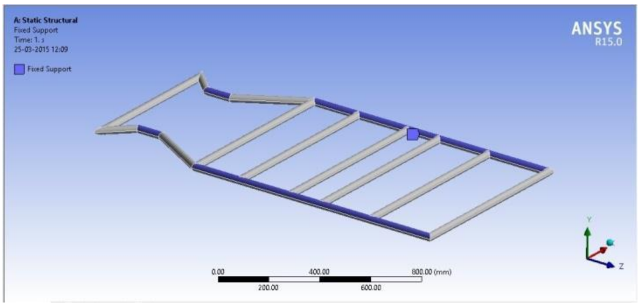

Figure.4: Boundary Conditions.

2.7 Loading:

Figure.5 below shows the forces that have been imposed downward to the structural model. The load is distributed uniformly on member below of driver's seat and engine compartment.[6]

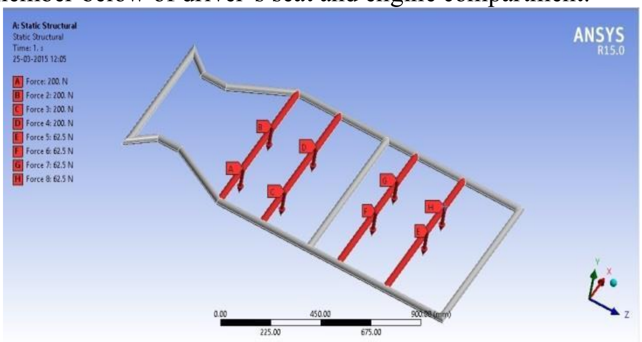

Figure.4: Loading Conditions

III. Results And Tables

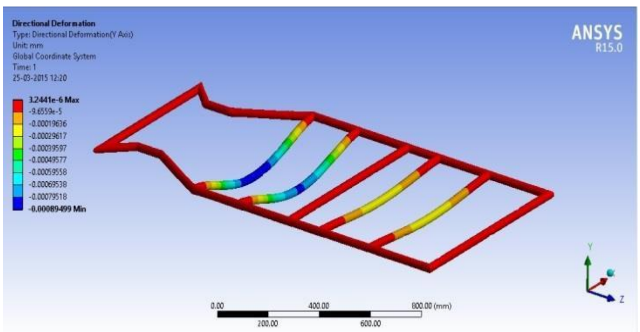

Figure.5: Results

Figure.5 shows the deflection of the model. The maximum deflection value is 3.2441×10^{-6}mm.
The result shows, that the location of maximum deflection goes well with theoretical location but varies in magnitude aspects, from the numerical analysis.

The structure is considered under uniformly distributed load of driver seat & engine compartment for analytical calculations. The below equation 1 calculates the maximum deflection which is calculated by moment area method from strength of material approach. [2][3]

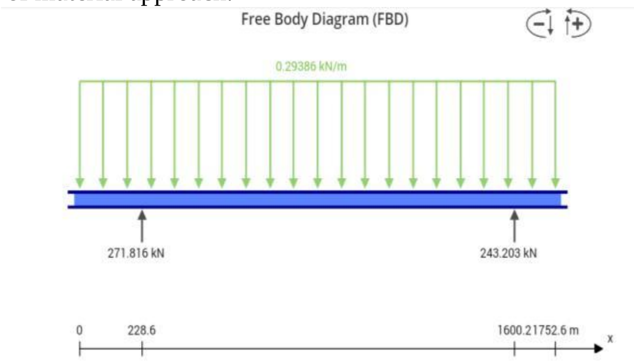

Figure.6: Free Body Diagram

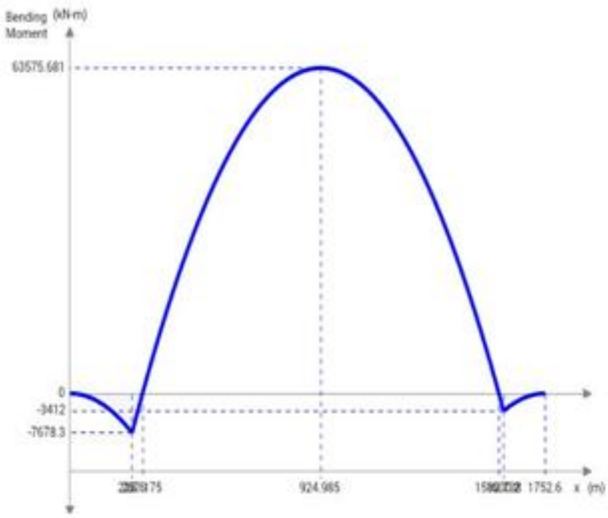

Figure.7: Bending Moment Diagram

$$\delta \max = \frac{moment\ of\ area\ of\ bending\ moment\ diagram}{structural\ rigidity}$$

= 2.092 mm

As we can observe there is difference in the value of maximum deflection between numerical simulation and analytical calculation, former (numerical simulation) being greater than later (analytical calculation).

IV. Conclusion

Using ANSYS software, static analysis was successfully carried out to determine maximum deflection. To countercheck these results, analytical calculations were carried out. The results of analysis show that the location of maximum deflection agrees well with theoretical maximum location of simple beam.

Acknowledgements

We express our deep gratitude towards our faculty in charge Prof. Rahul Thavai and Prof. Zakir Ansari (HOD of mechanical department) for his tremendous support and for providing us the technical guidance required to construct this piece of work.

REFERENCES

[1] N. R. Patil, Ravichandra R. Kulkarni, Bhushan R. Mane, Suhil H. Malve "Static analysis of Go-Kart Chassis frame by Analytical and SolidWorks Simulation" IJSET MAY 2014, Volume No.3 Issue No.5, pp : 661-663

[2] R.K. Rajput "Strength of materials"

[3] Vijaykumar V Patel and R.I.Patel "Structural Analysis OF Ladder Chassis Frame"WJST 2012

[4] SolidWorks Simulation help files.

[5] J.N.Reddy "An Introduction To Finite Element Method

[6] Kirpal R Singh "Automobile Engineering"

[7] V.B. Bhandari "Design of Machine Elements"

[8] "Design Data Book Of Engineering" by PSG .

[9] Mohd. Azizi Muhammad Nor, Helmi Rashid Wan Mohd Faizul , Wan Mohyuddin , Mohd Azuan Mohd Azlan , Jamaluddin Mahmud "Stress analysis of low loader Chassis"IRIS 2012 page no.995-1001.

[10] Ms. Kshitija A. Bhat1, Prof. Harish V. Katore " The FailureAnalysis of Tractor Trolley Chassis An Approach using Finite Element Method - A Review" IOSR-JMCE e-ISSN page no. 2278-1684.

[11] Hemant B.Patil, Sharad D.Kachave, Eknath R.Deore "Stress Analysis of Automotive Chassis with Various Thicknesses" IOSR-JMCE e-ISSN: 2278-1684 Volume 6, Issue 1 (Mar. - Apr. 2013), PP 44-49

[12] N.K.Ingole, D.V. Bhope "Stress analysis of tractor trailer chassis for self weight reduction" International Journal of Engineering Science and Technology (IJEST), ISSN: 0975-5462 Vol. 3 No. 9 September 2011

[13] Dr.R.Rajappan, M.Vivekanandhan Static and Modal Analysis of Chassis by Using SolidWorks IJES Volume 2 Issue 2 Pages 63-73 2013

[14] Beam formula with shear and moment diagram, American forest and paper association, Inc, American Wood Council, 1111 19th St., NW. Suite 800, Washington. DC 20036, 202 – 463 – 4713. Sane, S. S., Jadhav, G., Anandraj, H., 1955, " Stress Analysis of Light Commercial Vehicle Chassis by FEM",Piaggio Vehicle Pte.Ltd pune.Stress Analysis of Heavy Duty Truck Chassis using Finite Element Method," Phil. Trans. Roy. Soc. London, vol. A247, pp. 529–551

[15] Herb Adams " Chassis Engineering"

[16] V.B. Bhandari "Design of Machine Elements".