International Journal of Scientific Engineering and Technology (ISSN : 2277-1581)
Volume No.3 Issue No.5, pp : 661-663 1 May 2014

Static analysis of Go-Kart Chassis frame by Analytical and SolidWorks Simulation

N. R. Patil, Ravichandra R. Kulkarni, Bhushan R. Mane, Suhil H. Malve

Faculty of Mechanical Engineering, Solapur University, Solapur.
pnitin239@gmail.com, ravichandrak279@gmail.com, manebhushan@live.in. malvesuhil@gmail.com

***Abstract:* This paper aims to model, simulate and perform the static analysis of a go kart chassis consisting of Circular beams. Modeling, simulations and analysis are performed using modeling software i.e. SolidWorks according to the rulebook provided by Indian Society of New Era Engineers (ISNEE) for National Go Kart Championship (NGKC-13).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.**

1. 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.

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
- Improve driver comfort by providing more lateral space in the driver compartment
- Maintain ease of serviceability by ensuring that chassis members do not interfere with other subsystems

This study attempted to analyze 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.

2. Material and methodology:

2.1 Material Selection:The chassis is made up of AISI-1018.This material was selected due to its good Combination of all of the typical traits of Steel - strength, ductility, And comparative ease of machining. The properties of the material are presented in Table. 1

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

Table 1 Material properties

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 Fig. a

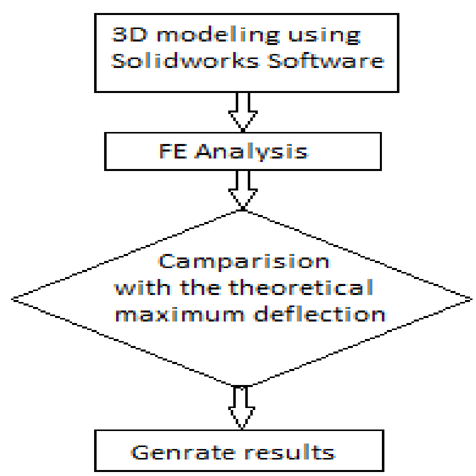

Fig. a Flowchart of paper methodology

2.3 Modeling:3-D modeling was done using SolidWorks software as shown in Fig.1

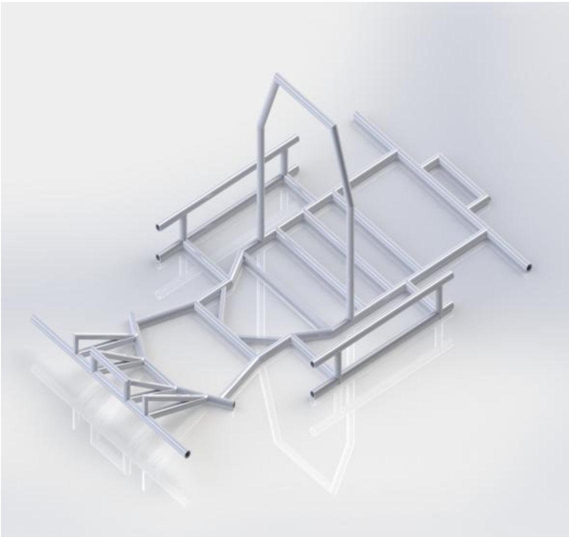

Figure no. 1 CAD model of Frame.

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.

2.5 Meshing:

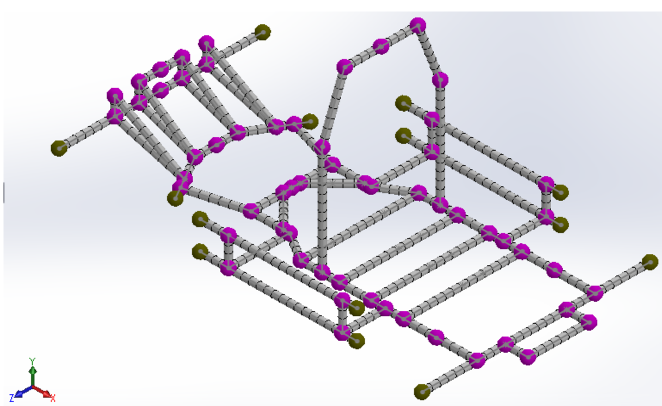

Figure no. 2 Meshing of Frame

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.

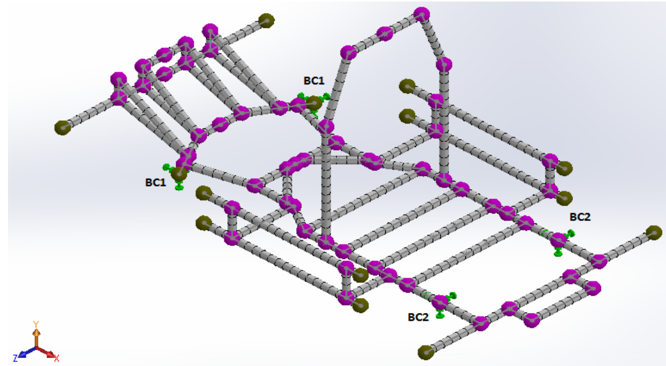

Figure no. 3 Boundary Conditions.

2.7 Loading:

Fig.4 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.

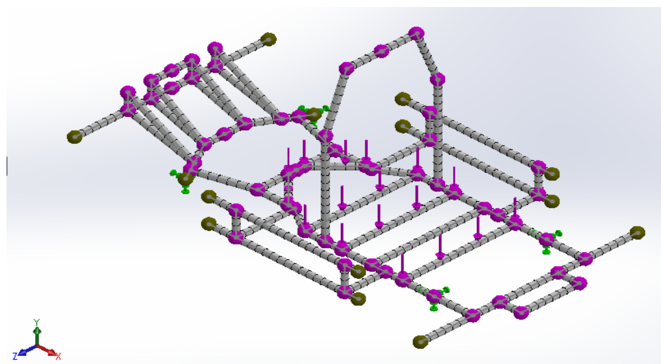

Figure no. 4 Loadings Conditions

3. Results and tables

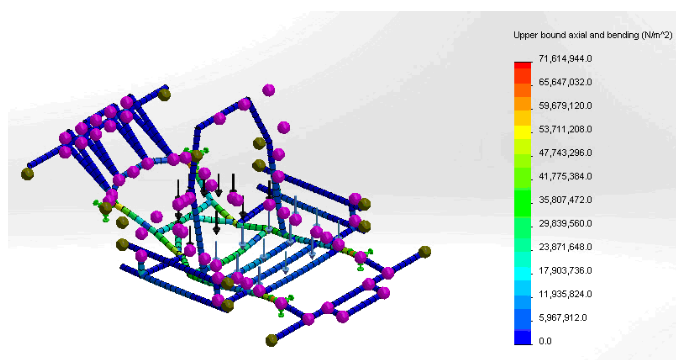

Figure no. 5 Results

Fig.5 shows the deflection of the model. The maximum deflection value is 0.01781mm.

The results of the numerical analysis revealed that the location of maximum deflection agrees well with theoretical location but varies in magnitude aspect.

For analytical calculation, the structure is considered under uniformly distributed load of driver seat and engine compartment. The maximum deflection can be calculated by following equation 1. The maximum deflection is calculated by Moment Area method from Strength of material approach.

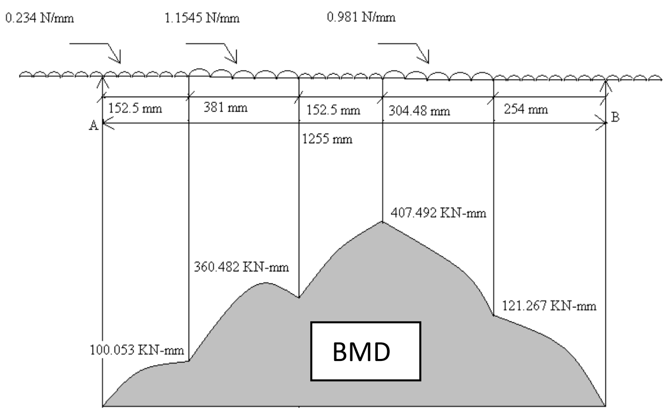

Figure no. 6 Beam with loading conditions and BMD.

$$\delta_{max} = \frac{\text{moment of area of bending moment Diagram.}}{\text{Structural rigidity}} \quad \text{.................equation 1}$$

$$= 5.414\text{mm}$$

There is difference in the value of maximum deflection between numerical simulation and analytical calculation. The value for analytical calculation is higher as compared to numerical simulation. This is possibly due to:-

- Approach of load distribution, load on simulation been distributed to model surface compare distribution of single beam surface for analytical calculation
- The geometry model for simulation is in complex 3D geometry rather than simple 2D theoretical beam approximations for analytical calculation.

4. Conclusion

Static analysis using finite element method was successfully carried out to determine maximum deflection and its location on chassis structure. The results of analysis reveled that the location of maximum deflection agrees well with theoretical maximum location of simple beam. This study found out that there is discrepancy between the theoretical (2-D) and numerical (3-D SOLIDWORKS) results.

Acknowledgment

I express my deep sense of gratitude and Special thanks to Dr. J. B. Dafedar, principal and B. K. Sonage, HOD of Mechanical Department (Solapur University, Solapur) for giving us training and required data for this paper.

References

i. *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*

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

iii. *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*

iv. *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*

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

vi. *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.*

vii. *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*

viii. *Herb Adams " Chassis Engineering"*

ix. *R.K. Rajput "Strength of materials"*

x. *V.B. Bhandari "Design of Machine Elements"*

xi. *SolidWorks Simulation help files.*