OPEN ACCESS INTERNATIONAL JOURNAL OF SCIENCE & ENGINEERING

MODELLING AND STRUCTURAL ANALYSIS OF RACE CAR CHASSIS USING DIFFERENT MATERIALS WITH FEM

[1]TUMMETI KRISHNA ,[2]GADDE ANIL KUMAR

[1]*M.Tech student, Department of Mechanical Engineering, G.V.R & S college of engineering and Technology , Guntur, Andhra Pradesh*

[2]*Assistant Professor, Department of Mechanical Engineering, G.V.R & S college of engineering and Technology, Guntur, Andhra Pradesh*

[1]krishnatummeti@gmail.com, [2]gaddeanilkumar302@gmail.com

Abstract: Chassis is basis foundation of any Automobile locomotive. Go-Kart is primary form of racing car where engineers can make use of theoretical knowledge with practical existence. A Go-Kart is a small four wheeled vehicles without suspension or differential. It is a light powered vehicle which is generally used for racing. There are various kinds of chassis out of which Tubular frame chassis are suitable for small vehicle. Design of chassis is on the basis of rigidity, strength and safety of driver by considering car that is durable as well as reliable whose material easily available in India. The kart has been designed using sound design principles .Go-kart is small four wheeler vehicle without suspension light powered vehicle used for racing. Basically CATIA, AUTO-CAD software are used for designing of Chassis whereas ANSYS- BENCHWORK software is used for Analysis of Chassis. Chassis is made up of joining various small links by using welding with limited number of joints so to avoid increase of weight and make the chassis strong enough to withstand high load. Finally select which is the suitable material(AISI1018, AISI 1022, AISI 4130) is best to be used for the go-kart chassis because of High Strength to weight ratio. Rigidity, Corrosion resistance, based on stresses, strain and deformation in static analysis and Total deformations in different frequncies in modal analysis & stress, strain, deformation in harmonic analysis values concluded the suitable material for gokart chassis

Keyword: *Chassis, Go-Kart, AISI1018, AISI 1022, AISI 4130 catia ANSYS 14.5*

I INTRODUCTION

1.1 INTRODUCTION TO GO KARTS

Kart racing or karting is a variant of open-wheel motorsport with small, open, four-wheeled vehicles called karts, go-karts, or shifter karts depending on the design. They are usually raced on scaled-down circuits. Karting is commonly perceived as the stepping stone to the higher ranks of motorsports.

The Go-Kart is a small powered single/double occupancy racing vehicle, having a similar functioning as of a F1 vehicle but specifically meant for low powered engines. The Go-kart tracks are smaller when compared to F1 tracks but the door to F1 opens after being part of International Go-Kart Championships. The Go Kart is very volatile as similar to F1 car chassis

The first go-kart was made in 1956 by the man named Art Ingels also called the father of go-karts. It was made from scrap metal and a lawn mower engine.

Go-karting is a big craze to Americans and Europeans. It is initially created in United states in 1950's and used for recreational purpose. Gradually it became a good hobby and other countries followed it. In India go karting is getting ready to make waves. A racing track is ready in Nagpur for Go karting and chennai is also trying to make one.

Indian companies like MRF, Indus motors are also producing Go karts. Go karts help to unleash the budding talents of engineers and emerging drivers for formula one.

1.2 ROLE OF CHASSIS IN AUTOMOTIVES

Every vehicle body consists of two parts; chassis and bodywork or superstructure. The chassis is the framework of any vehicle. Its principal function is to safely carry the maximum load for all designed operating conditions. It must also absorb engine and driveline torque, endure shock loading and accommodate twisting on uneven road surfaces. The chassis receives the reaction forces of the wheels during acceleration and braking and also absorbs aerodynamic wind forces and road shocks through the suspension. So the chassis should be engineered and built to maximize payload capability and to provide versatility, durability as well as adequate performance. To achieve a satisfactory performance, the construction of a heavy vehicle chassis is the result of careful design and rigorous testing.

It should be noted that this 'ladder' type of frame construction is designed to offer good downward support for the body and payload and at the same time provide torsional flexibility, mainly in the region between the gearbox cross member and the cross member ahead of the rear suspension. This chassis flexing is necessary because a rigid frame is more likely to fail than a flexible one that can 'weave' when the vehicle is exposed to arduous conditions. A torsionally flexible frame also has the advantage of decreasing the suspension loading when the vehicle is on uneven surfaces.

1.3 FRAME CONSTRUCTION

The frame or unibody is the main structural piece of the vehicle. Though they provide similar functions, they have different designs and purposes. When combined with all the braking, steering and suspension systems, this is commonly referred to as the chassis, or undercarriage, of the vehicle. Frame construction usually consists of channel-shaped steel beams welded and/or fastened together. The frame of the vehicle supports all the "running gear" of the vehicle, including the engine, transmission, rear axle assembly and all suspension components. This frame construction, referred to as full frame, is so complete that most vehicles can usually be driven without the body. Most trucks and larger rear-wheel drive cars use a full frame.

1.4 CROSS MEMBERS

section and consists of two trough The cross members control axial rotation and longitudinal motion of the main frame and reduce torsional stress transmitted from one mainframe to the other [26]. All cross members are designed to provide rigidity and strength, along with sufficient flexibility to withstand twisting and bending stresses encountered when operating on uneven terrain. Gussets are used to connect the cross members and the main frame. Gussets are generally welded or bolted to the main frame web. Cross-members may be rectangle beam, Circular-section, Tubular, boxed, or other shapes, and may be bolted or welded together. In case of bolted joints, the majority of the load is transferred by the frictional force or clamping force between the members of the joint. The bolts must be properly tightened to develop and maintain the desired clamping force.

II.LITERATURE REVIEW

2.1 LITERATURE REVIEW

Lonny L.Thompson, et al [15] determined that a high sensitivity value indicates a strong influence on the torsional stiffness of the overall chassis. Results from the sensitivity analysis are used as a guide to modify the baseline chassis with the goal of increased torsional stiffness with minimum increase in weight and low center-of-gravity placement. The torsional stiffness of the chassis with various combinations of added members in the front clip area, engine bay, roof area, front window and the area behind the roll cage was predicted using finite element analysis. They concluded that with strategic placement of structural members to a baseline chassis, the torsional stiffness can be more than tripled with only a 180 N increase in weight.

Kim, H.S et al. [12] have presented a method for dynamic stress analysis of structural components of bus systems. They have used the hybrid superposition method that combines the finite element static and eigen value analysis with flexible multibody dynamic analysis. In the stress recovery, dynamic stresses are estimated as a sum of pseudostatic stresses and modal acceleration stresses, which are obtained by applying the principle of linear superposition to the modal acceleration method.

A method for vehicle analysis based on finite element technique has been proposed by Johansson, I., and Gustavsson, M., [11]. Vehicle dynamics and durability have been taken into account in their work and an in-house developed pre and post processor is used to achieve effectiveness. Oijer, F., [16] has proposed a method for force and stress calculation using complete vehicle models in MSC.Nastran, where variables such as road profile and curve radii are used as input. This, in combination with modal superelement reduction, will result in faster design studies. Accurate calculations of force histories are of utmost importance for reliable fatigue life estimates. The forces are often calculated by the use of multi-body software (MBS) and used as input for stress analysis in a finite element package. A drawback is that the MBS calculations are very time consuming, especially if flexible bodies are included, and are thus, not well suited for fast parameter studies. This literature survey reveals that there is a strong need to predict the transient response of truck chassis when subjected to dynamic loads while it encounters a bump with different speeds of vehicle.

The stress analysis of truck chassis using riveted joints has been performed by Cicek Karaoglu et al [4], in order to achieve a reduction in the magnitude of stress near the riveted joint of the chassis frame. Side member thickness, connection plate thickness and connection plate length were varied. Numerical results showed that stresses on the side member can be reduced by increasing the side member thickness locally. If the thickness change is not possible, increasing the connection plate length may be a good alternative.

In order to investigate the transient response of a vehicle–structure interaction system in time domain, Tso-Chien Pan et al [20] developed a dynamic vehicle element (DVE) method. The DVE method treats the vehicle as a moving part of the entire system, which considers the vehicle influence at the element level by incorporating the detailed interaction between multiple vehicles and the structure induced by irregular road profiles. In addition, a simplified decoupled dynamic nodal loading (DNL) method is proposed. The DNL method generates a time series of concentrated nodal loading which represents the vehicle reaction force on the structure. The DNL method therefore, accounts for the road irregularities and vehicle inertia effect, but neglects the interaction between the two subsystems. Parametric studies for the effects of road roughness, speed parameter, mass ratio, and frequency ratio on the dynamic vehicle–structure interaction are then carried out using the DVE and DNL methods.

A.V. Pesterev et al [18] determine the dynamic amplification factor function for an irregularity represented as a superposition of simpler ones. Another purpose of this paper is to demonstrate the application of the pothole dynamic amplification factor (DAF) functions technique to finding a priori estimates of the effect of irregularities with a repeated structure. Specifically, the problem can be solved by finding the conditions under which the dynamic effect of two identical potholes located one after another is greater than that due to the single pothole. We also find the estimate for the number of periods of a periodic irregularity that are sufficient in order to consider the oscillator response as steady state.

III .DESIGN OBJECTIVES

3.1 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 Deciding the cost efficiency of such in terms of large scale manufacturing. Calculation of stresses acting on the chassis of the vehicle under different loading conditions. The product can prove to be very efficient in all the aspects such as cost, drivability, maintenance, easy usage, safety etc.

1)To get knowledge of Go-Kart Chassis Design for beginner in stepwise manner so to avoid unnecessary thing and focuses on competition.

2) Focusing area of analyzing Software to get desirable result.

3) To make use of welding alternative of hydraulic press.

4)Use of welding Principle effectively without increasing weight of chassis and make it simple.

SCOPE OF PROJECT:

1. Go- kart is gaining wide popularity as it is suitable for most of people to make their own playing car as well as working car by using their knowledge and competition with other in racing.

2. Making advance engineering Knowledge to use at chassis for making it better and light weight.

3. Using variety of material to make better chassis for various applications and mostly for racing purpose.

4. Practice Engineering knowledge with budgeting for making Cost effective chassis.

3.2 METHODOLOGY

DESIGN METHODOLOGY:

Design of any component is consists of three major principles:

1. Optimization
2. Safety
3. Comfort

Step 1: Collecting information and data related to gokart chassis

Step 2: A fully parametric model of the gokart chassis is created in catia software.

Step 3: Model obtained in igs is analyzed using ANSYS 14.5 (work bench) to obtain stresses, strain, deformation and in modal analysis, harmonic analysis find out the deformations,stresses in different frequencies,

Step 4: Front and side impact manual calculations are done

Step 5: Finally, we compare the results obtained from ANSYS **(SATIC, MODAL, HARMONIC)** and compared different materials. (1018 and AISI1018, AISI 1022, AISI 4130),

3.4 MATERIAL PROPERTIES

3.4.1 AISI 1018

AISI 1018 mild/low carbon steel has excellent weldability and produces a uniform and harder case and it is considered as the best steel for carburized parts. AISI 1018 mild/low carbon steel offers a good balance of toughness, strength and ductility. Provided with higher mechanical properties, AISI 1018 hot rolled steel also includes improved machining characteristics and Brinell hardness. Specific manufacturing controls are used for surface preparation, chemical composition, rolling and heating processes. All these processes develop a supreme quality product that are suited to fabrication processes such as welding, forging, drilling, machining, cold drawing and heat treating.

3.4.2AISI1022:

Hot rolled AISI 1022 steel is a low hardenability carbon steel with low tensile strength ranging between 360-560 MPa. AISI 1022 low carbon steel has Brinell hardness ranging between 100 – 170 and it is mostly supplied in the black hot rolled condition. Compared to AISI 1020 steel, AISI 1022 steel has a higher content of manganese and carbon.AISI 1022 steel is usually used in the rolled condition. AISI 1022 can be carburized achieving case hardness more than Rc65 with smaller sections. The case hardness reduces with an increase in section size. Hot rolled AISI 1022 low carbon steel has excellent machinability and ductility with reasonable strength

3.4.3 AISI4130

Alloy steels are designated by AISI four-digit numbers. They are more responsive to mechanical and heat treatments than carbon steels. They comprise different types of steels with compositions which exceed the limitations of B, C, Mn, Mo, Ni, Si, Cr, and Va in the carbon steels.

Properties of Material	AISI 1018	AISI 1022	AISI 4130
Modulus of elasticity (GPa)	205	200	210
Carbon content %	0.15-20	0.20-23	0.28-33
Yield strength (MPa)	370	375	435
Ultimate strength (MPa)	440	400	560
Density (Kg/m^3)	7.87×10^3	7.70×10^3	7.85×10^3

TABLE 1:MATERIAL PROPERTIES

AISI 4130 alloy steel contains chromium and molybdenum as strengthening agents. It has low carbon content, and hence it can be welded easily. The datasheet given below provides more details about AISI 4130 alloy steel.

IV.INTRODUCTION TO CATIA V5R20

Welcome to **CATIA (Computer Aided Three Dimensional Interactive Application)**. As anew user of this software package, you will join hands with thousands of users of this high-end CAD/CAM/CAE tool worldwide. If you are already familiar with the previous releases, you can upgrade your designing skills with the tremendous improvement in this latest release.

CATIA V5, developed by Dasssault Systems, France, is a completely re-engineered, Next-generation family of CAD/CAM/CAE software solutions for Product Lifecycle Management. Through its exceptionally easy-to-use and state-of-the-art user interface, CATIA V5 delivers innovative technologies for maximum productivity and creativity, from the inception concept to the final product. CATIA V5 reduces the learning curve, as it allows the flexibility of using feature-based and parametric designs.

CATIA V5 provides three basic platforms: P1, P2, and P3. P1 is for small and medium-sized process-oriented companies that wish to grow toward the large scale digitized product definition.

P2 is for the advanced design engineering companies that require product, process, and resource modeling. P3 is for the high-end design applications and is basically for Automotive and Aerospace Industry, where high quality surfacing or Class-A surfacing is used. The subject of interpretability offered by CATIA V5 includes receiving legacy data from the other CAD systems and even between its own product data management modules. The real benefit t is that the links remain associative. As a result, any change made to this external data gets notified and the model can be updated quickly.

4.1DESIGN PROCEDUE IN CATIA :

Create the rectangular profile in sketcher workbench as per the below dimensions after go to part design create plane now select plan go to sketcher now create two hollow circles at the end of the rectangle profile again go to part design apply rib option now converted into the solid body.

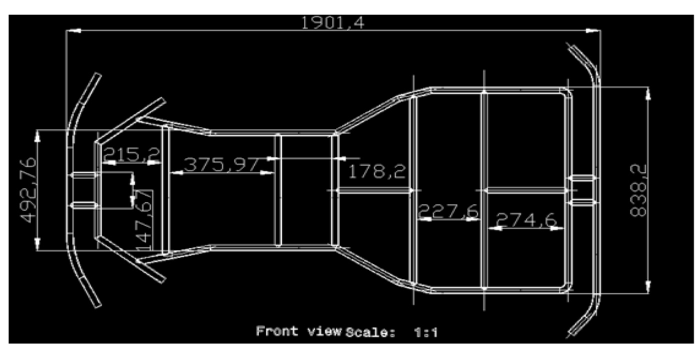

Figure 1: GOKART 2D DRAFTING IMAGE

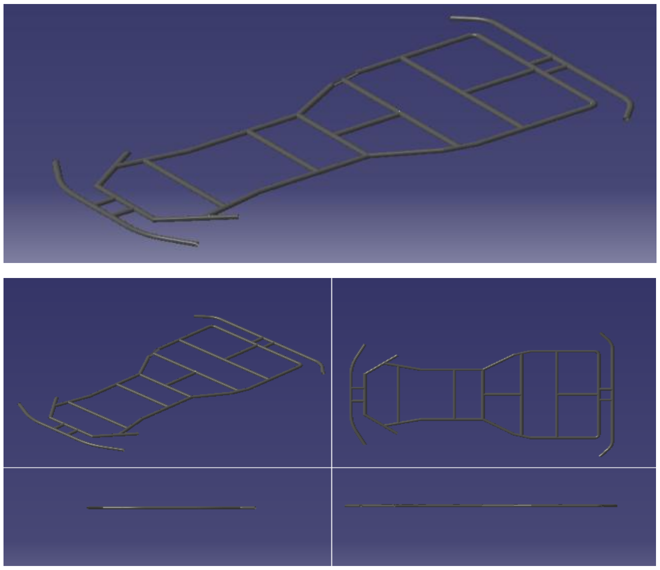

Figure 2:CHASSIS FRAME IN CATIA WORKBENCH

V.INTRODUCTION TO ANSYS

ANSYS is a large-scale multipurpose finite element program developed and maintained by ANSYS Inc. to analyze a wide spectrum of problems encountered in engineering mechanics.

5.1 PROGRAM ORGANIZATION:

The ANSYS program is organized into two basic levels:

- Begin level
- Processor (or Routine) level

The Begin level acts as a gateway into and out of the ANSYS program. It is also used for certain global program controls such as changing the job name, clearing (zeroing out) the database, and copying binary files. When you first enter the program, you are at the Begin level.

At the Processor level, several processors are available. Each processor is a set of functions that perform a specific analysis task. For example, the general pre-processor (PREP7) is where you build the model, the solution processor (SOLUTION) is where you apply loads and obtain the solution, and the general postprocessor (POST1) is where you evaluate the results of a solution. An additional postprocessor, POST26, enables you to evaluate solution results at specific points in the model as a function of time.

5.1.1Material Models:

ANSYS allows several different material models like:

- Linear elastic material models (isotropic, orthotropic, and anisotropic).
- Non-linear material models (hyper elastic, multi linear elastic, inelastic and

Visco elastic)

- Heat transfer material models (isotropic anorthotropic)
- Temperature dependent material properties andCreep material models.

5.1.2Loads:

The word loads in ANSYS terminology includes boundary conditions and externally or internally applied forcing functions, as illustrated in Loads. Examples of loads in different disciplines are:

5.2 ANALYSIS TYPES:

The following types of analysis are possible using ANSYS

· Structural Analysis: Static Analysis, Modal Analysis, Harmonic Analysis,

Transient Dynamic Analysis, Spectrum Analysis, Buckling Analysis, Explicit

Dynamic Analysis, Fracture mechanics, and Beam Analysis.

· Thermal Analysis: Steady-state thermal analysis, transient thermal analysis.

· CFD (Computational Fluid Dynamics) Analysis: Laminar or turbulent, Thermal or adiabatic, Free surface, Compressible or incompressible, Newtonian or Non-Newtonian, Multiple species transport.

· Several types of Electromagnetic field analysis and Coupled field analysis.

5.3 POST PROCESSING:

Post processing means reviewing the results of an analysis. It is probably the most important step in the analysis, because you are trying to understand how the applied loads effect your design, how good your finite element mesh is, and so on Two postprocessors are available to review your results: POST1, the general Postprocessor, and POST26, the time-history postprocessor. POST1 allows you to review the results over the entire model at specific load steps and sub steps (or at specific time points or frequencies).

POST26 allows you to review the variation of a particular result item at specific points in the model with respect to time, frequency, or some other result item. In a transient magnetic analysis, for instance, you can graph the eddy current in a particular element versus time. Or, in a nonlinear structural analysis, you can graph the force at a particular node versus its deflection.

5.4 STATIC ANALYSIS:

A static analysis calculates the effects of steady loading conditions on a structure, while ignoring inertia and damping effects, such as those caused by time varying loads. A static analysis can, however, include steady inertia loads (such as gravity and rotational velocity), and time-varying loads that can be approximated as static equivalent loads (such as the static equivalent wind and seismic loads commonly defined in many building codes).

VI.FINITE ELEMENT METHOD

6.1 INTRODUCTION

The Basic concept in FEA is that the body or structure may be divided into smaller elements of finite dimensions called “Finite Elements”. The original body or the structure is then considered as an assemblage of these elements connected at a finite number of joints called “Nodes” or “Nodal Points”. Simple functions are chosen to approximate the displacements over each finite element. Such assumed functions are called “shape functions”. This will represent the displacement with in the element in terms of the displacement at the nodes of the element.

The Finite Element Method is a mathematical tool for solving ordinary and partial differential equations. Because it is a numerical tool, it has the ability to solve the complex problems that can be represented in differential equations form. The applications of FEM are limitless as regards the solution of practical design problems.

6.2 THE BASIC STEPS INVOLVED IN FEA

Mathematically, the structure to be analyzed is subdivided into a mesh of finite sized elements of simple shape. Within each element, the variation of displacement is assumed to be determined by simple polynomial shape functions and nodal displacements. Equations for the strains and stresses are developed in terms of the unknown nodal displacements. From this, the equations of equilibrium are assembled in a matrix form which can be easily be programmed and solved on a computer. After applying the appropriate boundary conditions, the nodal displacements are found by solving the matrix stiffness equation. Once the nodal displacements are known, element stresses and strains can be calculated.

Basic Steps in FEA

- Discretization of the domain
- Application of Boundary conditions
- Assembling the system equations
- Solution for system equations
- Post processing the results.

6.3 ANALYSIS PROCEDURE IN ANSYS:

Designed component in catia workbench after imported into ansys workbench now select the steady state thermal analysis .1.ENGINEEERING MATERIALS (MATERIAL PROPERTIES).

2.CREATE OR IMPORT GEOMENTRY.

3.MODEL(APPLY MESHING).

4.SET UP(BOUNDARY CONDITIONS)

5.SOLUTION

6.RESULTS

6.4 STATIC STRUCTURAL ANALYSIS

The static structural analysis calculates the stresses, displacements, strains, and forces in structures caused by a load that does not induce significant inertia and damping effects. Steady loading and response conditions are assumed; that the loads and the structure’s response are assumed to change slowly with respect to time. A static structural load can be performed using the ANSYS WORKBENCH solver. The types of loading that can be applied in a static analysis include:

6.5Meshing the two models are imported into the Ansys work bench and generate mesh nodes and elements are created

6.6 BOUNDARY CONDITIONS :

Load of Driver, Driver Seat and Engine were taken into consideration while load of steering system, fuel tank, etc. is low as compared to above components hence it can be neglected. Also, as chain drive transmission system is used load of transmission system can also be neglected. Finally we taking load Front Impact:3328.6N, Side impact:2487. Taking materials are (AISI1018, AISI 1022, AISI 4130)

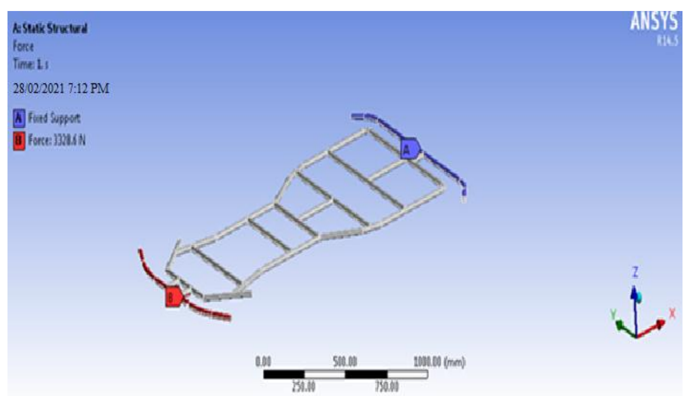

Figure 3: FRONT IMPACT BOUNDARYCONDITION

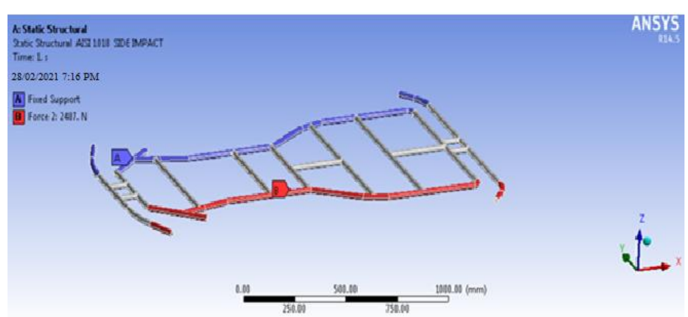

Figure 4: SIDE IMPACT BOUNDARY CONDITION

VII RESULTS AND DISCUSSION

7.1 STATIC ANALYSIS RESULTS:

This static,modal and harmonic analysis is performed to find Structural parameters such as Stresses, Strains, Deformation, Here we observed results on Three materials namely (AISI1018, AISI 1022, AISI 4130).

7.1 aisi 1018 material

7.1.1 Front impact :

The Front Impact Analysis has been carried out on the Ansys 14.5 while constructing a perfect space frame tubular chassis on catia and then it was imported to Ansys 14.5 have been applied on the regions A force of Front Impact:3328.6N was applied to the front ends constraining the body panel rods and the observed deformation, Stress and strain as shown in below figures.

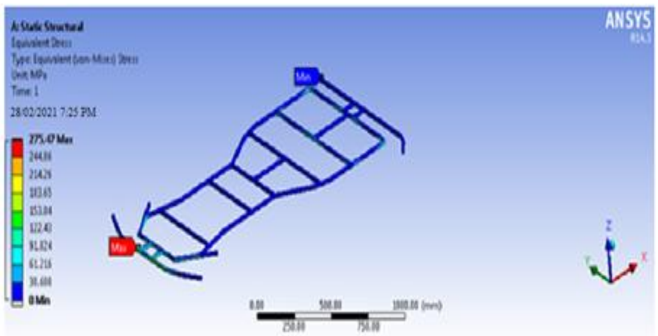

Figure 5 STRESS ON AISI1018

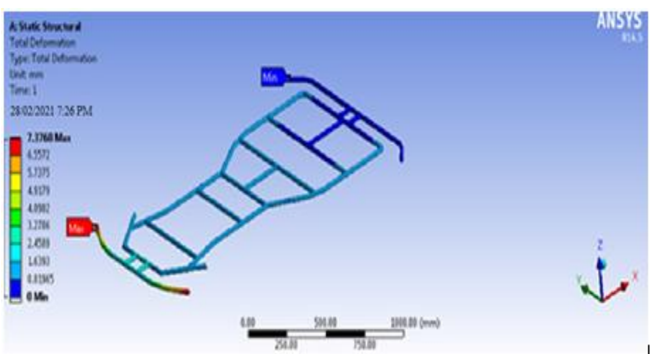

Figure 6:TOTAL DEFORMATION ON AISI 1018

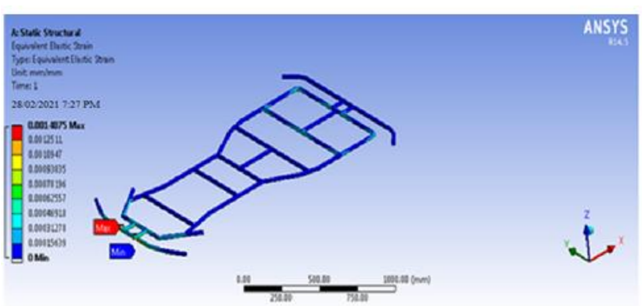

Figure 7STRAIN ON AISI 1018

7.1.2 Side impact:

The Side Impact Analysis has been carried out on the Ansys 14.5.while constructing a perfect space frame tubular chassis catia and then it was imported to Ansys 14.5 with a Force with respect to the 2G criteria A force of Side impact:2487 has been applied and the observed deformation, Stress and strain as shown in below figures

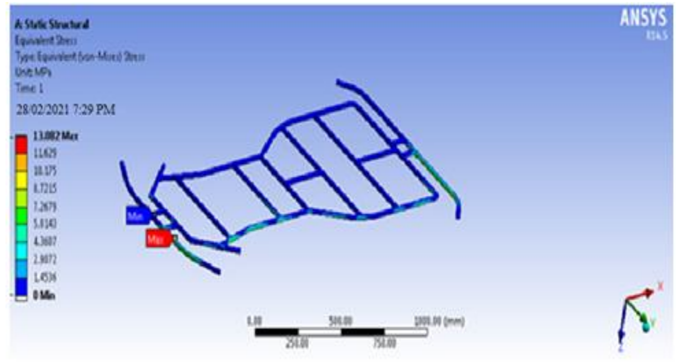

Figure8 STRESS ON AISI 1018

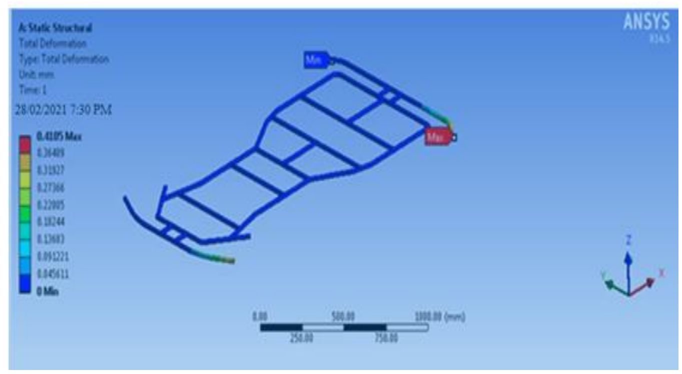

Figure 9 TOTAL DEFORMATION ON AISI 1018

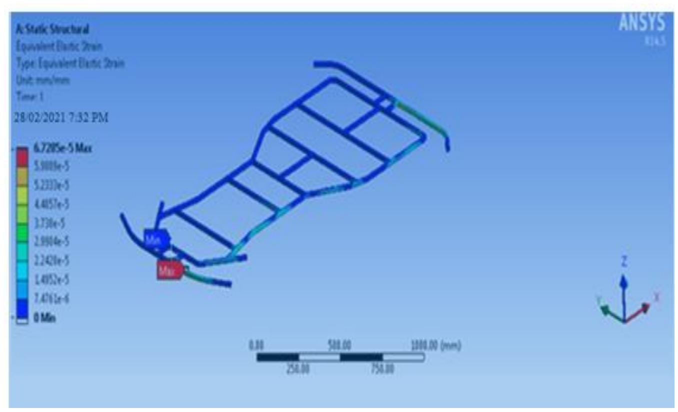

Figure 10 STRAIN ON AISI 1018

7.2 AISI 1022 SIDE IMPACT:

The Side Impact Analysis has been carried out on the Ansys 14.5 while constructing a perfect space frame tubular chassis catia and then it was imported to Ansys 14.5 with a Force with respect to the 2G criteria A force of Side impact:2487 has been applied and the observed deformation, Stress and strain as shown in below figures

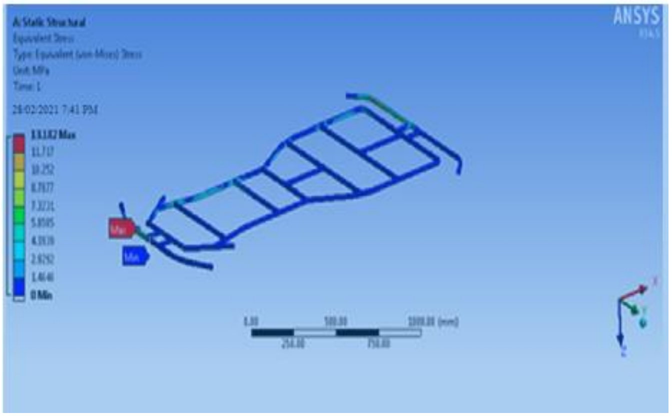

Figure 11 STRESS ON AISI 1022 MATERIAL

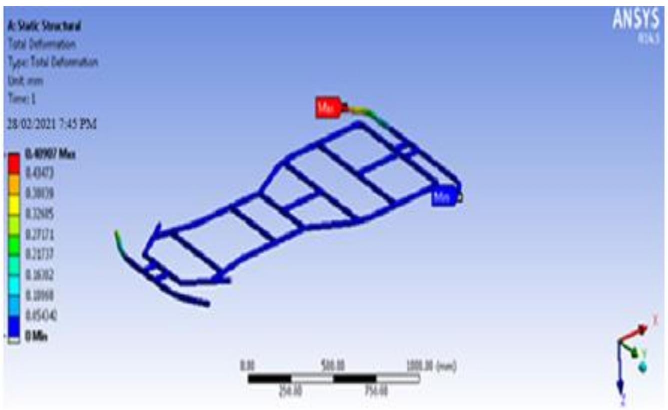

Figure 12 TOTAL DEFORMATION ON AISI 1022

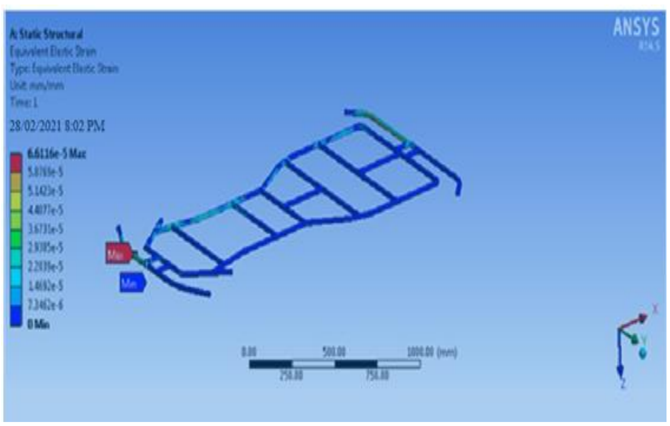

Figure 13 STRAIN ON AISI 4130

7.3 GRAPHS:

7.3.1 VON-MISSES STRESS FRONT IMPACT:

From below figure, we can observe that in case of equivalent (von-mises) stress, gokart chassis when apply front impact chassis is made up of (AISI1018, AISI 1022, AISI 4130) AISI 1018 is Observed to have least stress of 275.47 in comparison with remaining materials including the present material is 4130 material as shown below figure.

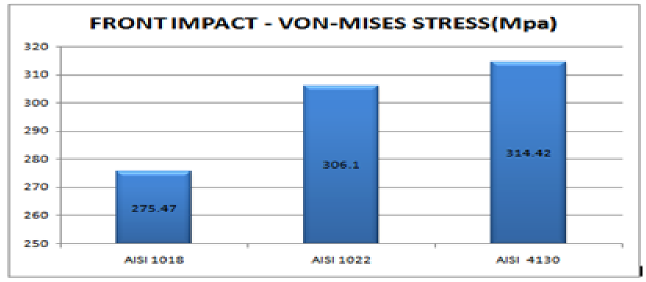

Figure 14 FRONT IMPACT OF VONMISSES STRESS

7.3.2 TOTAL DEFORMATION OF FRONT IMPACT:

From below figure, we can observe that in case of Total deformation, gokart chassis when apply front impact chassis is made up of (AISI1018, AISI 1022, AISI 4130) AISI 1018 is Observed to have least Total deformation of 7.3 in comparison with remaining materials including the present material is 4130 material as shown below figure.

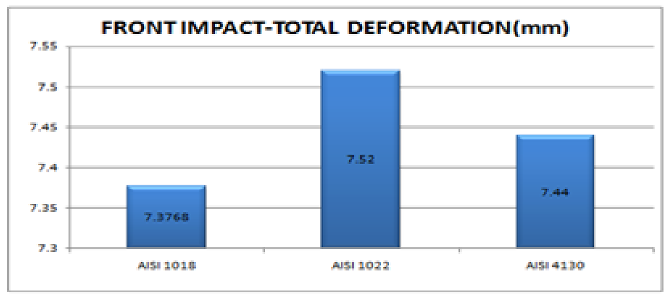

Figure 15 FRONT IMPACT OF TOTAL DEFORMATION

7.3.3 STRAIN OF FRONT IMPACT:

From below figure, we can observe that in case of STRAIN, gokart chassis when apply front impact chassis is made up of (AISI1018, AISI 1022, AISI 4130) AISI 1018 is Observed to have least STRAIN of 7.3 in comparison with remaining materials including the present material is 4130 material as shown below figure.

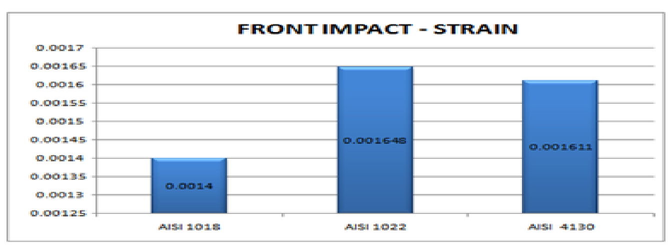

Figure16. FRONT IMPACT OF STRAIN

VIII .CONCLUSION

To achieve the set goals, we used the finite element for the evaluation, creation and modification of the best vehicle design. Our prior aim was to build a go kart with minimum cost without compromising the safety and performance of the vehicle. The final result is a desired Go Kart design meeting all the above factors.circular section go-kart design and analysis are done with AISI1018,AISI 1022, AISI 4130 materials finally Result concluded that the AISI1018 material is gives better performance compared to the Remaing material, AISI 1018 is the suitable material to be used for the go-kart chassis because of High Strength to weight ratio., Rigidity, Corrosion resistance, , Fatigue Resistance. we are concluded based on stresses,strain and deformation values the suitable material for gokart. Static ,Modal analysis & Harmonic analysis is performed using finite element method was successfully carried out on chassis catia model to determine in Ansys equivalent stresses, maximum deformations, Hence the chassis design is safe with AISI 1018 material compared to remaining material than proceed the manufacturing process

REFERENCES

[1] HERB ADAMS," CHASSIS ENGINEERING".

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

[3] V.B. Bhandari, "Design of Machine Elements".

Journal Paper,

[4] P.K. Sharma ,Nilesh J. Parekh, DarshitNaik, 2014, "Optimization and Stress Analysis in Chassis in TATA turbo truck SE1613" IJEAT 2014 page no.181-187.

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

[6] Mohd. Azizi Muhammad Nor, Helmi Rashid Wan MohdFaizul , Wan Mohyuddin , MohdAzuanMohdAzlan , Jamaluddin Mahmud,2012,"Stress analysis of low loaderChassis"IRIS 2012 page no.995-1001

[7] 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 pageno. 2278-1684

[8] HemantB.Patil, SharadD.Kachave, Eknath R.Deore,"Stress Analysis of Automotive Chassiswith Various Thicknesses" IOSR-JMCE e-ISSN: 2278-1684 Volume 6, Issue 1 (Mar. - Apr.2013), PP 44-49

[9] N.K.Ingole, D.V. Bhope,2011,"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

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

[11] "Beam formula with shear and moment diagram", American forest and paper association, Inc, American Wood Council, 1111 19th St., NW. Suite 800, Washington. DC20036, 202 – 463 – 4713.

[12] Sane, S. S., Jadhav, G., Anandraj, H,1955,"Stress Analysis of Light CommercialVehicle Chassis by FEM",Piaggio Vehicle Pte.Ltd,pune.

[13] "Stress Analysis of Heavy Duty Truck Chassis using Finite Element Method," Phil.Trans. Roy. Soc. London, vol. A247, pp. 529–551