



International Journal of Intellectual Advancements and Research in Engineering Computations

Design and impact analysis of GO-KART chassis

B.Sakthivel¹, J.Vignesh², K.Vignesh², M.Vijayakumar², R.Vivekanandhan²,

¹Associate Professor, ²UG Students

Department of Mechanical Engineering, Nandha Engineering College, Erode-52,

Tamil Nadu, India.

¹sathivelme1990@gmail.com, ²vigneshjawahar007@gmail.com

Abstract-Go-Kart is also spelled Go-Cart. And Go-Kart vehicle is simple and small four wheel vehicle without suspension and this car normally used for racing purpose. This paper is aimed to perform dynamic analysis for Go-Kart chassis especially for Impact test. In India till there is no manufacturer to manufacture these type of Go-Kart vehicle. And Modeling and analysis are performed in CATIA V5 R20 and ANSYS workbench 14.5. Then chassis built by circular pipe and AISI 1018 is suitable material to fabricate the chassis.

Keywords- Go-Kart, Chassis, Impact test, CATIA V5 R20, ANSYS 14.5

I. INTRODUCTION

The Chassis of Go-Kart is like skeleton of car. It is made by circular pipe sometimes used other shape. So we design the Go-Kart chassis and then we analysis. In the Go-Kart chassis there is no suspension so we use circular profile because its provide enough to flexible. We powered the vehicle by using IC Engine. The drivers in Go-Kart are also not professional. Even children can also drive it. And Go-Kart have 4 wheel and a small engine and single seat vehicle and design the chassis by using CATIA V5 R20. After the solid modeling the design was test by Ansys software.

Dimension are shown below:

All dimension in inch

(Right or Left) Side wheel Base: 44.09

(Right or Left) Overall Length: 62.99

Back side wheel base length: 35.43

Front side base length: 31.49

Some dimension taken from (Overall and wheel length) ISIE INDIA rules book. Because some standard dimension are useful to design & fabricate the product.

II. LITERATURE REVIEW

[1] Sandeep Ramini et al conducted experimental structural analysis of chassis using L bar mild steel and stainless steel material are considered. They conduct impact test for front and rear.

[2] A.P.D. Salman Basha et al conducted test that is front impact analysis and he analysis according with FSAE constrains (Formula society of automobile engineering) and they conclude seat position want to improve.

[3] Abhijit Padhi et al showed how to Increase factor of safety on bush located on Go-kart of the chassis. They increase thickness of the material.

[4] N.A.Z. Abdullah et al conducted the experiment to improve the chassis by using modal analysis to chassis.

[5] P.N.V. Bala Subramanyam et al experiment static analysis for Go-kart chassis. He choosed material ASTM A106 grade B because its affordable cost.

III. METHODOLOGY

In this methodology. Process how we are do the process step by step and they are given below.

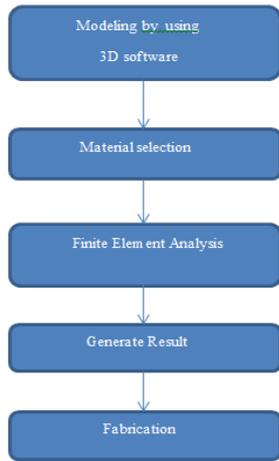


Fig.1 Process flow chart

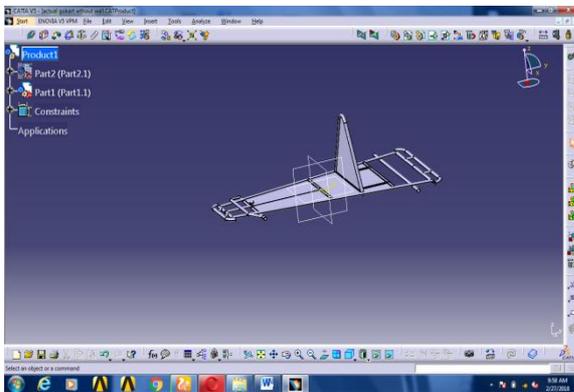


Fig.2 3D model in CATIA image

IV. MATERIAL METHODOLOGY

Chassis material is very important because. The chassis carry all the components. The carbon content or composition in the material is important to determine the properties like hardness, brittleness, strength. Its plays main role because it provide safety and durability.we choose steel AISI1018 because its low cost compare to other material like aluminums alloys and also provide good strength. And we are compare some material standards are given below.

Table.1:Material composition

COMPOSITION	AISI1018	AISI1022	AISI4130
Modulus of elasticity	205	200	210
Carbon Content %	0.15-0.20	0.20-0.23	0.28-0.33
Density (kg/m ³)	7.90x10 ³	7.70x10 ³	7.85x10 ³
Yield strength (Mpa)	370	375	435

V.FINITE ELEMENT ANALYSIS

Finite element analysis (FEA) is a computerized method to find how the material react with given force. In finite element analysis shows whether the breakage is occur and how much deflection are occur we can easily see deviations are collects then finally as a report. The analysis was done by ANSYS Workbench 14.5.

In Ansys, we are analyzing the Impact test in Explicit Dynamics option then we import CATIA 3D FILE into WORKBENCH and then click then enter into model option.Here we want MESH the chassis.MESH is nothing but separate whole component into small piece segments. And the wall also created give concrete properties and also mesh even the wall.

Mainly we analysis that is 1.Impact test, 2.Modal Analysis, 3.Static Structural Analysis.

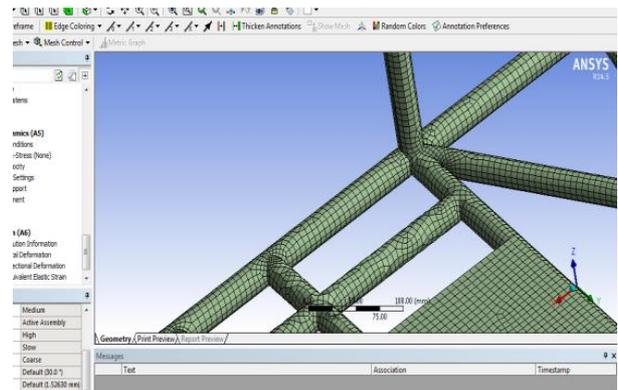


Fig.3 Auto meshed chassis image

That meshed pictures are given below.In the option we also meshed the wall also before that to mesh we want give separatelyconcrete properties to the wall And also steel properties give to the chassis

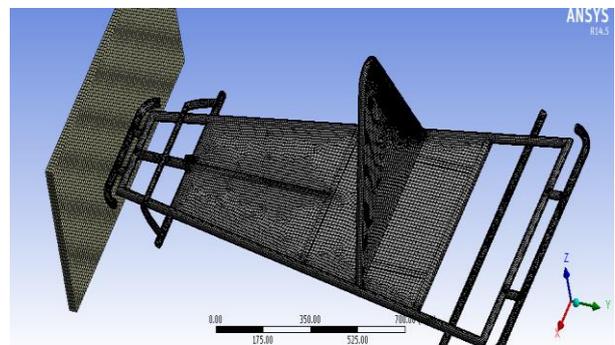


Fig.4 Auto meshed image

VI.BOUNDARY CONDITON

First of all,we analysis in static structural just import the 3D file and give the boundary condition.That is give just load act on the chassis like engine weight and driver weight its shown in fig.5.

Then we do Impact test. We want to fix the wall so select under the option is Explicit dynamics.Then click fixed support and then give velocity of the chassis with give respect timing.Its Impact test so we give nearly 0 seconds to get effective so we give 0.007 seconds.

Before give timing we want give respective loads and force.And the direction of force and load or pressure direction is indicated by arrows.

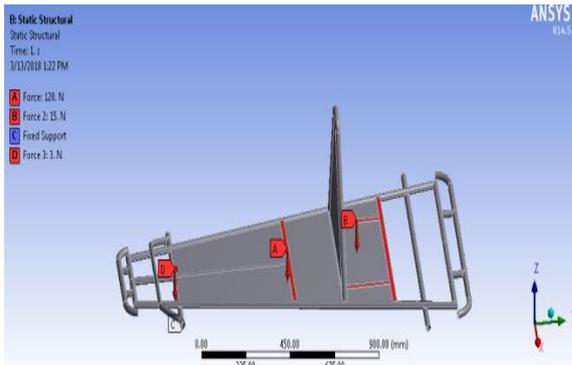


Fig.5 Boundary condition in static image

Table.2: Total Deform

Time [s]	Minimum [mm]	Maximum [mm]
1.1755e-038	0.	0.
3.5015e-005		0.73009
7.0015e-005		1.5078
1.0502e-004		2.1165
1.4002e-004		2.8068
1.7503e-004		3.532
2.1001e-004		4.2307
2.4501e-004		4.905
2.8003e-004		5.6095
3.1502e-004		6.3446
3.5e-004		7.0854
3.8503e-004		7.7954
4.2001e-004		8.5084
4.5503e-004		9.2183
4.9001e-004		9.8923
5.2501e-004		10.549
5.6003e-004		11.257
5.9502e-004	11.978	
6.3002e-004	12.694	
6.6501e-004	13.416	
7.0001e-004	14.142	

VII.RESULT

Finally we get result concluded of AISI 1018 steel material is most economic and better performance.And also easy to produce mass production. Static and explicit dynamic analysis using finite element method was carried out that chassis 3D model to determine the vonmesis stress, total deformation,strain.These are shown below.

1. IMPACT TEST

We done Impact analyse o Go-Kart chassis and we evaluate the result.The Total deformation, stress and strain. We give speed 80KMph but in Ansys we give in velocity 20000mm/s

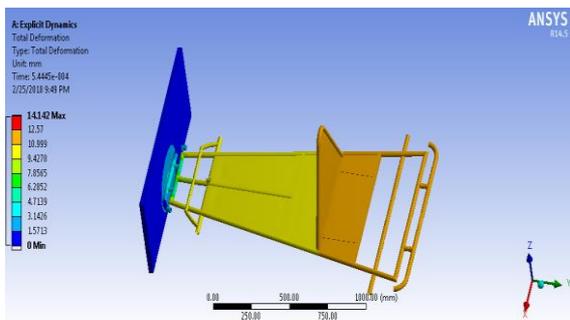


Fig.6 Total deformation image

1.1 EXPLICIT DYNAMICS(Total deformation)

1.2 EXPLICIT DYNAMICS (equivalent stress)

Table.3: von mises stress

Time [s]	Minimum [MPa]	Maximum [MPa]
1.1755e-038	0.	0.
3.5015e-005		1526.3
7.0015e-005		2096.
1.0502e-004		2718.5
1.4002e-004		2411.2
1.7503e-004		3849.4
2.1001e-004		5055.7
2.4501e-004		6112.7
2.8003e-004		6588.2
3.1502e-004		6955.7
3.5e-004		6774.4
3.8503e-004		6869.8
4.2001e-004		6809.5
4.5503e-004		7161.8
4.9001e-004		8018.9
5.2501e-004		9439.5

VII. IC ENGINE DETAIL AND SPEED

In the Go-Kart car is run or powered by IC Engine.We choose ACTIVA 125 cc Engine and its details are given below:

Honda Activa Engine

Displacement : 124.8cc
 Torque : 10.12Nm at 5500rpm
 Mileage : 55 km (apprx)

2.	12.185
3.	19.67
4.	21.685
5.	30.076

By using above information we calculate the speed of the car

We have formula $V = \frac{\pi \times D \times N}{60}$

we Assume Activa wheel Max Speed is V_a

Hence, $N_1 = \frac{V_a \times 60}{\pi D} = \frac{27.8 \times 60 \times 100}{\pi \times 44}$

$N_1 = 1207.2 \text{ rpm.}$

We Assume Go-Kart wheel Max speed

$N_2 = N_a / 0.72 = 1207 / 0.72$

$N_2 = 1676.4 \text{ rpm.}$

Max speed of Go-Kart (appr)

$V_2 = \frac{\pi \times D_2 \times N_2}{60}$

$V_2 = 3.14 \times 0.254 \times 1676.4 / 60$

$V_2 = 22.2 \text{ m/s}$

$V_2 = 22.2 \times 18 / 5$

Finally, $V_2 = 79.9 \text{ Kmph}$

Above calculate velocity of Go-Kart is not achieve 100% because of some losses like frictional loss.

2. MODAL ANALYSIS

We done modal analysis of Go-kart chassis and evaluate the results are shown below.

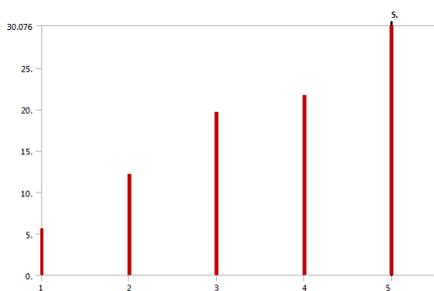


Fig.7 Graph about modal analysis image

Table.3: Frequency with respect to module

Mode	Frequency [Hz]
1.	5.6338

3.STATIC STRUCTURE ANALYSIS

We done static structure analysis of Go-kart chassis and we evaluate stress, strain and deformation and details shown below

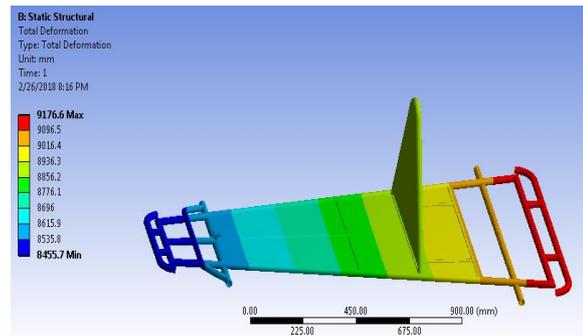


Fig.7 Total deform in static analysis

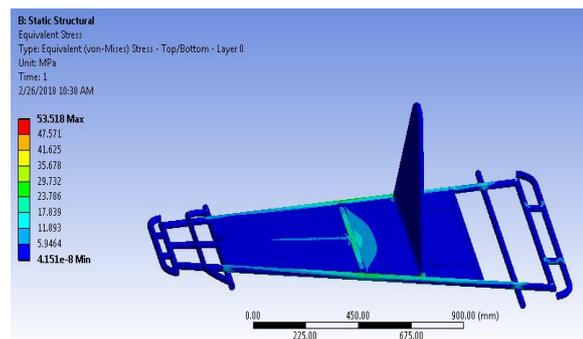


Fig.8 Stress in static analysis

Max deform = 9176.6mm
 Max stress = 53.51Mpa
 Max strain = 0.00028 mm/mm

$FOS = \frac{\text{Yield strength of AISI1018}}{\text{Von mises stree}}$

Finally, $FOS = 370 / 53.51 = 6.1$

VIII. CONCLUSION

We conclude that AISI 1018 material is most economic and got some better performance. Impact analysis using finite element method is successfully done on 3D model and evaluate the result is total deform, stress and strain and factor of safty is above 1 so design is safe.

IX.REFERENCE

- [1] Dr. R. Rajapan, M. Vivekanandhan, “*Static And Modal Analysis Of Chassis Using Solidworks*”, IJE Volume 2 Issue 2 Pages 63-73, 2013.
- [2] Hemant B. Patil, Shrad D. Kachave, Eknath R. Deore, “*Stress Analysis Of Automotive Chassis With Various Thicknesses*”, IOSR-JMCEe- SSN: 2278-1684 Volume 6, Issue 1 (Mar. 0Apr.2013) PP 44-49
- [3] Mr. Girish Mekalke, “*Static Analysis Of Go-Kart Chassis*”, IJMIT, Vol-3 Issue 2, PP: (73-78), ISSN 2348-7593, October 2015 – March 2016
- [4] N.R. Patil et. al “*Static Analysis Of Go-Kart Chassis Frame By Analytical And Solidworks Simulation*”, International Journal Of Scientific Engineering And Technology (ISSN: 2277 1581), Volume No.3 Issue No.5, PP : 661-663 1 May 2014
- [5] Pravin.A. renuke, “*Dynamic Analysis Of Car Chassis*”, International journal of engineering research and Applications, ISSN: 2248-622, Vol. 2, Issue 6, November- December 2012, PP.955-959 955
- [6] Rahul Thavai, Quazi Shahezad, Mirzashahrkh, Mukri Arman, Han Imran, “*Static Analysis of Go-Kart Chassis By Analytical And Solidworks Simulation*”, IJMERE, ISSN: 2249-6645, Vol.5, Issue 4, Apr.2015