Explicit Analysis of A Go-Kart Chassis When Collision with Rigid Model

Mr. K. Kiran Kumar[1], Mr. G. Srikanth Reedy[2], Mr. B. Janaiah[3]
[1, 2, 3] Dept of Mechanical Engineering
[1, 2, 3] AVN institute of Engineering & Technology, Hyderabad, TS, India

Abstract- *Vehicle crash is a highly nonlinear transient dynamics phenomenon. The purpose of an explicit analysis is to see how the chassis will behave in a frontal, backward or sideways collision. Crashworthiness simulation is one typical area of application of Finite-Element Analysis (FEA). This is an area in which non-linear Finite Element simulations are particularly effective. The chassis is the main component for any heavy vehicle. The chassis should be able to carry the maximum designed load for all operating conditions. It must also withstand static and dynamic loads without undue deflection or distortion.*

In this project, impacts and collisions involving a go-kart frame model are simulated and analyzed using ANSYS software. The given model is tested under different collision conditions and the resultant deformation and stresses are determined with respect to a time of 70 Mille sec for ramp loading using ANSYS software.

I. INTRODUCTION

The Go-Kart is a vehicle which is simple, lightweight and compact and easy to operate. The go-kart is specially designed for racing and has very low ground clearance when compared to other vehicles. The common parts of go-kart are engine, wheels, steering, tires, axle and chassis. No suspension can be mounted to go-kart due to its low ground clearance and also the driving is trembling. Both two stroke and four stroke engines are used for gokart. Electric motor engines are also used for go kart and are called as ecokart. The go-karting is a variant open wheel motor sport with small, open, four wheeled vehicles. The chassis is independent of suspension to experience thrill.

Because of its simplicity and economical, gokart is the most interested thing for most of the users. The tracks go-kart is similar to F1 racing track. A go-kart is powered by 125cc engine in most of the countries. In some countries, go-karts can be licensed for use on public roads. Typically, there are some restrictions, e.g. in the European nations go-kart on the road should needs of different factors like head light (high/low beam), tail lights, a horn, indicators and a maximum of 20 HP. It is well known that Kart racing is usually a low-cost and relatively safe way to introduce drivers to motor racing.

II. METHODOLOGY

In this project, impacts and collisions involving a go-kart chassis model of a body-on-frame construction model are simulated and analyzed using ANSYS software. The given model is tested under different collision conditions as mentioned below and the resultant deformation and stresses are determined with respect to a time of 70 Mille sec for ramp loading using ANSYS software.

- 60 Kph Front impact.
- 50 kph Side impact.
- 30 kph rear impact.
- 3d model of chassis frame is generated by using CATIA software.
- 3d model of chassis frame is converted into parasolid file.
- The parasolid file is imported to ANSYS software to perform analysis on chassis assembly.
- Transient dynamic analysis performed on the chassis frame for 60 Kph front impact for period of 70 ms.The deflections and stresses are plotted and documented.
- Transient dynamic analysis performed on the chassis frame for 50 Kph side impact for period of 70 ms.The deflections and stresses are plotted and documented.
- Transient dynamic analysis performed on the chassis frame for 30 Kph rear impact for period of 70 ms.The deflections and stresses are plotted and documented.

3D MODELING OF ROTOR SHAFT

The chassis is designed considering the factors like factor of safety - maximum load carrying capacity,

The main component of the frame is divided into two major parts first the front block (cockpit) for steering and seat positions etc. and second rear block (engine compartment) for transmission and brake assembly.

Force absorption capacity, required space for accessories and driver and specific dimensions.

The design of chassis is performed by using cad software CATIA. The load distribution in the chassis should be uniform. The structural design gives the idea about the chassis. Design gives the optimum size and shape of the chassis.
Below are the detailed steps explained about how the chassis is designed

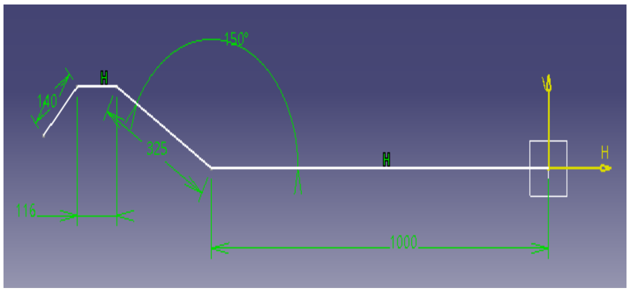


Figure: 2D sketch profile of chassis.

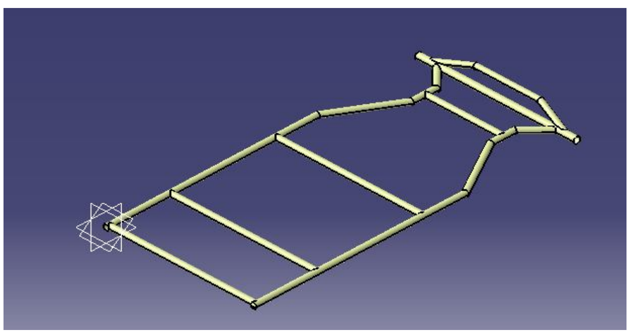


Figure: Isometric view of chassis.

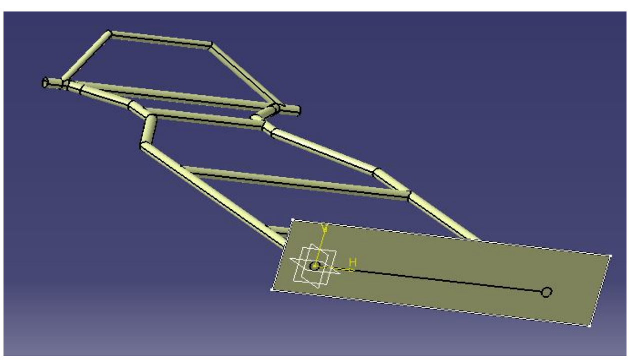


Figure: 3.24 Back impact analysis chassis design.

III. FINITE ELEMENT ANALYSIS OF CHASSIS

The go-kart chassis is like a skeleton frame which made up of pipes and using other materials of different cross-sections. The frame must consist of strengtheners; stability, torsion rigidity, as well as it should also have relatively high degree of flexibility as there is no suspension used for go-kart. It also helps to provide the driver for better strength for operator and other accessories. So, the chassis is design by convenience and safety for operator. The chassis is designed for a safe ride and the load is applied on it must without compromising the structural strength.

Finite element analysis (FEA) is computerized numerical software used for forecasting the results how a product get to the forces applied in real world, vibrations, and other physical effects when the forces are applied. Finite element analysis gives the results like stresses and deflections when the product undergoes structural analysis. The computer software analyses the elements and node & shows us a collective result. The material and structure of chassis was finalized and then FEA was performed on it. It is tested whether the chassis will be able to withstand torsion, impact.

The structural analysis is done to know the stresses and defections developed due to the force applied on the chassis. The impact force testing is performed to know the worse conditions when it is driving mode and also to determine the maximum deformation.

IV. MATERIAL AND METHODOLOGY

The amount of carbon in steel is important to determine the strength, hardness, and machining characteristics. Material selection of the frame plays an important role in providing desired strength, endurance, safety and reliability of the vehicle. The material used for chassis are various grades of steel or aluminium alloys. The main component of steel is carbon which increases the hardness of material of chassis. Aluminium alloy is expensive than steel so mainly steel is used to constructs the chassis.

V. EXPLICIT DYNAMIC ANALYSIS OF CHASSIS FOR FRONTAL IMPACT

Explicit dynamic analysis has been carried out in Ansys work bench for the impact at a velocity of 60 Kph from front side and the fix support is apply from backside. The geometry used for the analysis and the finite element model used are shown in the following figures.

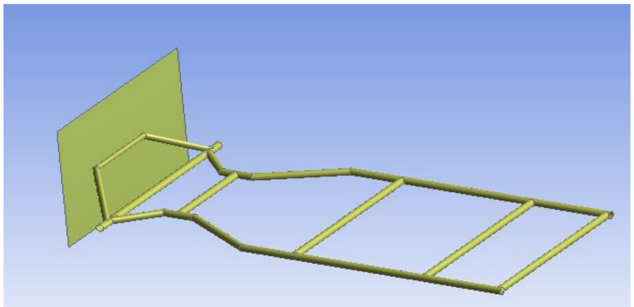


Figure. 3d model of the chassis and the rigid wall

FINITE ELEMENT MODELING

3D model of the chassis was developed in CATIA. The model was then converted into a parasolid to import into ANSYS. A Finite Element model was developed with solid elements. The elements that are used for idealizing the chassis model are described below. A detailed Finite Element model was built with shell elements to idealize all the components of chassis. A total number of 64573 elements were created using shell element type.

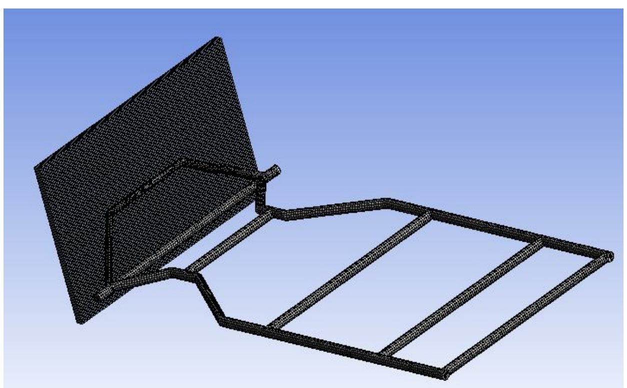


Figure: meshed model of the chassis and the rigid wall

MATERIAL SPECIFICATION

In this project the analysis is carried out for 2 materials. The material properties are given in the below table.

1. The material used for the chassis of the structure is Steel Alloy Heat Treated AISI-4340, with the following properties.

S.No	Parameter	Steel alloy_AISI-4340
1	Young's Modulus(N/mm2)	203000
2	Poisson's ratio	0.33
3	Ultimate Tensile Strength (N/mm2)	1835
4	Yield Stress, бy (N/mm2)	1600

BOUNDARY CONDITIONS

The boundary conditions and loading used for explicit dynamic analysis of the chassis are as follows:

- The velocity of 60 Kph (16680 mm/sec) for frontal impact was applied over the time period of 70 ms on the chassis as shown in the below figures.
- The chassis base constrained in all DOF as shown in the below figures.
- The sides of the wall fixed in all DOF as shown in the below figures.

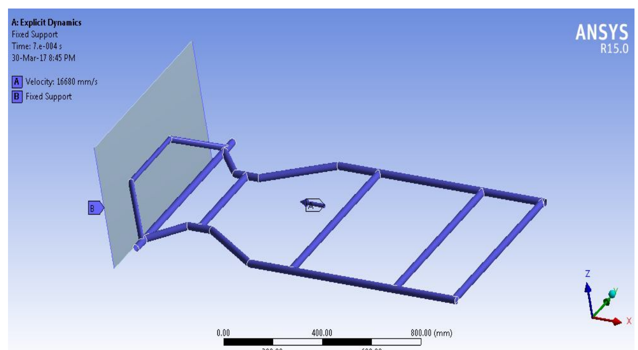
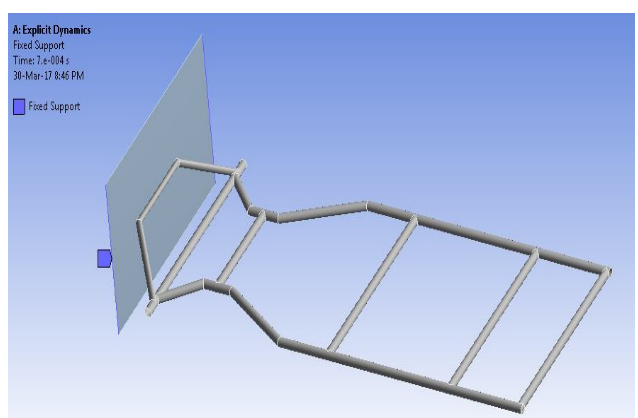
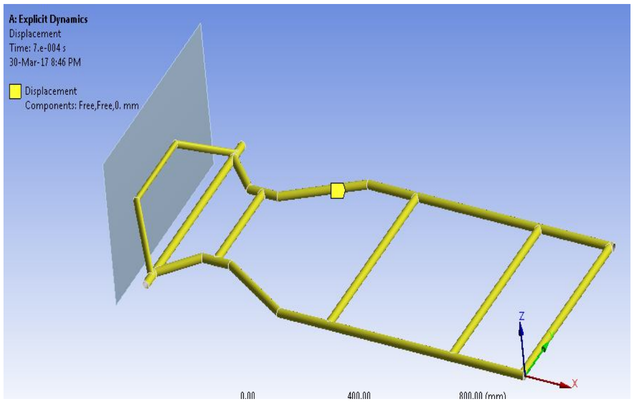


Figure: velocity boundary condition for chassis frontal impact





VI. RESULTS OF THE ANALYSIS

The explicit dynamic analysis was done using Ansys workbench for frontal impact with 60 Kph velocities. The deflections and stresses are plotted as shown in the below figures.

Deflection: From the analysis the maximum deflection of 12.2 mm is seen on the chassis for frontal impact as shown in the below figure.

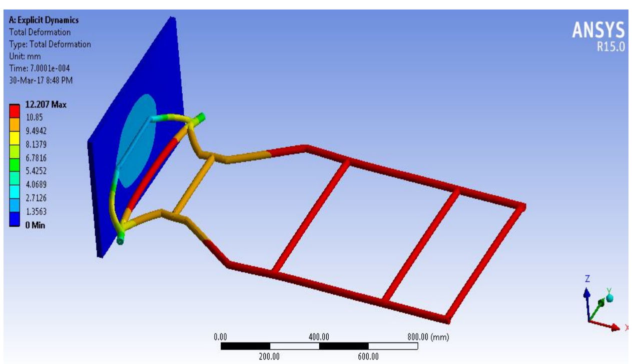


Figure: Deflection plot of chassis and wall for chassis frontal impact

Stresses

A maximum VonMises stress of 421.8 Mpa is seen on the chassis front member for the frontal impact as shown in the below figure.

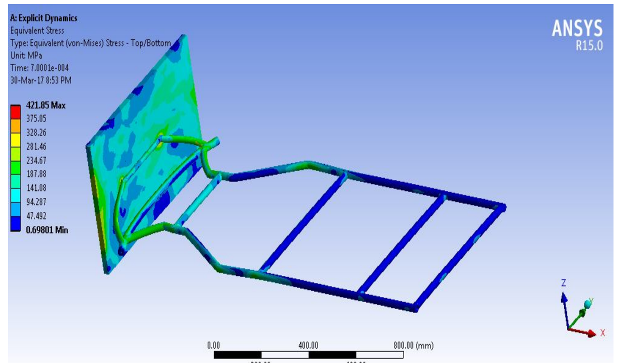


Figure: VonMises stress plot of chassis and wall for chassis frontal impact

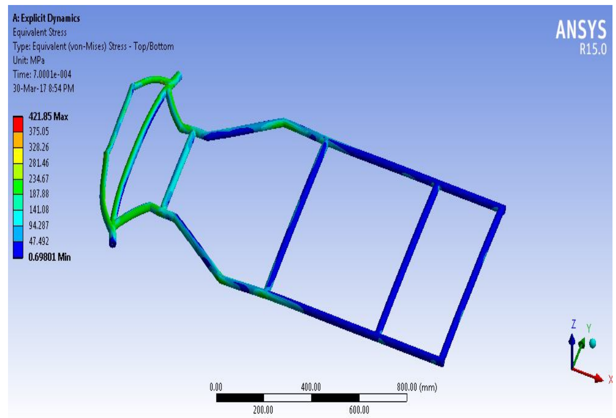
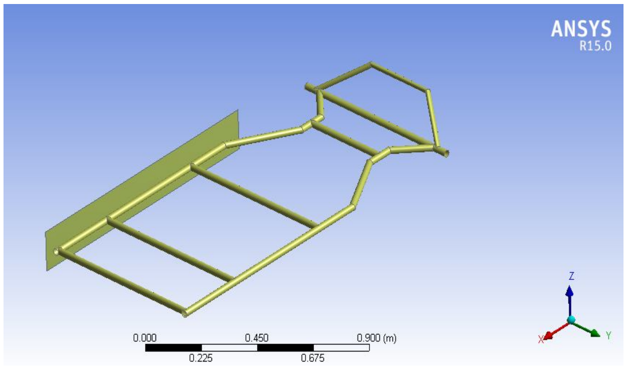


Figure: VonMises stress plot of chassis frontal impact

	Summary of results for Frontal Collision	
S.No	Item	Steel
1	Deflection (mm)	12
2	VonMises Stress (Mpa)	421.85
3	Yield strength (Mpa)	1600
4	FOS	3.79

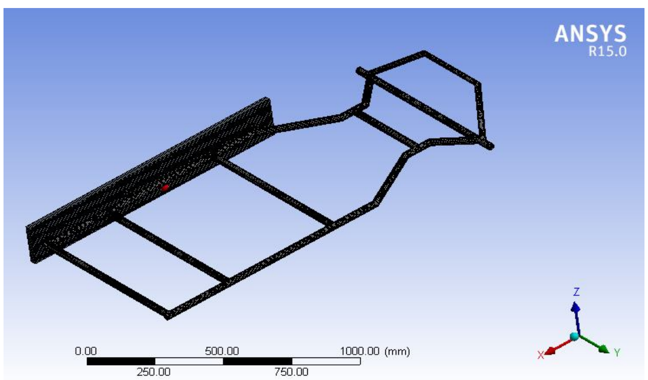
VII. EXPLICIT DYNAMIC ANALYSIS OF CHASSIS FOR SIDE IMPACT

Explicit dynamic analysis has been carried out in Ansys work bench for the side impact at a velocity of 50 Kph. The geometry used for the analysis and the finite element model used are shown in the following figures.



FINITE ELEMENT MODELING

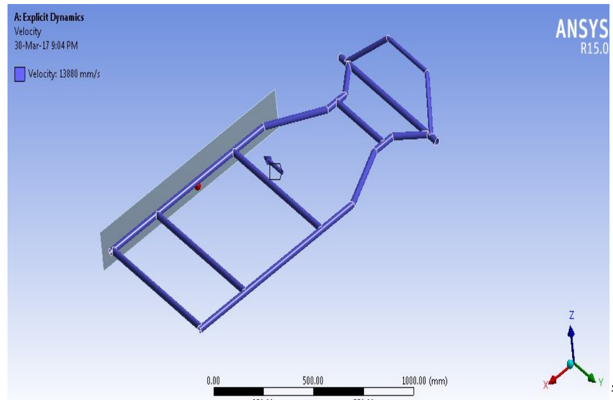
3D model of the chassis was developed in CATIA. The model was then converted into a parasolid to import into ANSYS. A Finite Element model was developed with shell elements. A detailed Finite Element model was built with shell elements to idealize all the components of chassis. A total number of 87573 elements were created using shell element type.



BOUNDARY CONDITIONS

The boundary conditions and loading used for explicit dynamic analysis of the chassis for side impact are as follows:

- The velocity of 50 Kph for side impact was applied over the time period of 70 ms on the chassis as shown in the below figures.
- The chassis base constrained in all DOF as shown in the below figures.
- The sides of the wall fixed in all DOF as shown in the below figures.



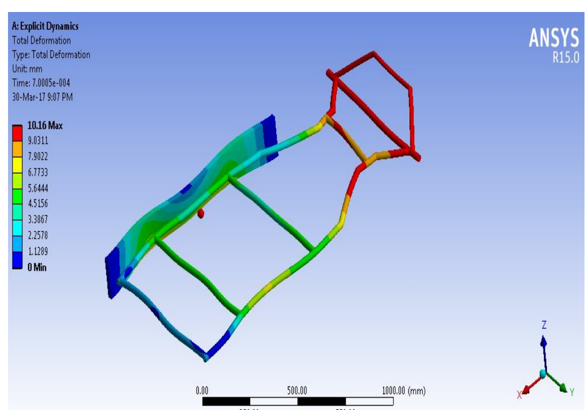
,

Results of the side impact analysis: The explicit dynamic analysis was done using Ansys workbench for side impact with 50 Kph velocities. The deflections and stresses are plotted as shown in the below figures.

	Summary of results for side Collision	
S.No	Item	Steel
1	Deflection mm)	10.16
2	VonMises Stress (Mpa)	1612
5	Yield strength (Mpa)	1600
6	FOS	0.99

Deflection

From the analysis the maximum deflection of 10.16 mm is seen on the chassis for side impact as shown in the below figure.



Stresses

A maximum VonMises stress of 1612 Mpa is seen on the chassis side member for the side impact as shown in the below figure.

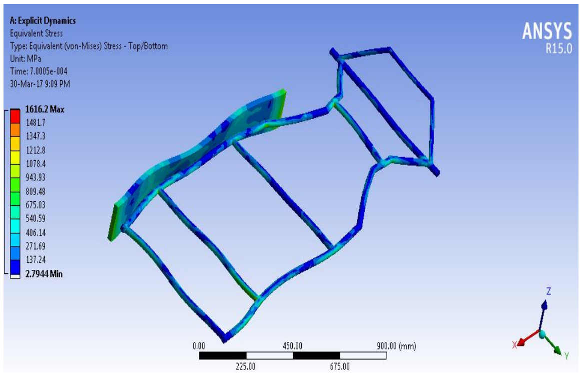


Table: Summary of results for side collision

VIII.CONCLUSION

In this project, impacts and collisions involving a go-kart frame model were simulated and analyzed using ANSYS software. The given model was tested under different collision conditions and the resultant deformation and stresses were determined with respect to a time of 70 Mille sec for ramp loading using ANSYS software. Explicit analysis was done on chassis frame in different directions using ANSYS software. From the above analysis results it is concluded that the go-kart chassis frame is safe for impact loading.

REFERENCES

[1] Mr. Kartik Kelkar1, Mr. Siddhant Gawai2,Mr.Tushar Suryawanshi3, Mr. Shaikh Ubaid4, and Mr. Rajratna Kharat5, "**STATIC ANALYSIS OF GO-KART CHASSIS" ,** International Journal of Research in Advent Technology , Issue: 09th April 2017.

[2] Mr.Virendra.S.Pattanshetti **"DESIGN AND ANALYSIS OF GO KART CHASSIS"** International Journal of Mechanical and Industrial Technology, Vol. 4, Issue 1, Month: April 2016 **-** September 2016.

[3] Sannake Aniket S., Shaikh Sameer R., Khandare Shubham A., Prof. S.A.Nehatrao , "**DESIGN AND ANALYSIS OF GO-KART CHASSIS" ,** International Journal of Advance Research and Innovative