ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

Mechanical Design and Analysis of Car Hit Simulator

I. Murali Mohan
MTech Student
Department of Mechanical Engineering,
Brilliant Grammar School Educational Society's Group Of Institutions,
Abdullapurmet(V), Hayathnagar(M), Ranga Reddy (Dist), India.

K.Sudhakar Reddy
Associate Professor
Department of Mechanical Engineering,
Brilliant Grammar School Educational Society's Group Of Institutions,
Abdullapurmet(V), Hayathnagar(M), Ranga Reddy (Dist), India.

V. Srikanth
Assistant Professor
Department of Mechanical Engineering,
Brilliant Grammer School Educational Society's Group Of Institutions,
Abdullapurmet(V), Hayathnagar(M), Ranga Reddy (Dist), India.

Abstract: ***Ensuring passenger safety is order of the day for high speed cars. Today's cars are coming up into automotive market with inclusion of high end passenger safety system. Regulatory norms insist for functional demonstration of such systems before a particular car is launched in market. The two vital requirements that needs to be met during such demonstration are spontaneous reaction of sensors in actuating the said safety system with in no time and no physical damage to the dummy passenger mounted on seat. This project proposes design of a simulator which can provide shock that will be experienced during car hit. This simulator demands for door of the car along with its safety system which needs to be evaluated. A mechanism will be designed as part of this project for driving an impactor against the door besides which seat will be mounted which in turn houses the passenger safety system. This mechanism comprises of a special type of cam known to be snail cam, spring, bearing and other mechanical elements. It also incorporates a piston cylinder arrangement so as to simulate the friction characteristics of car with its mounted track. Further it also accommodates guide wheels, which provides movement to the car simulator upon receiving the impact similar to that of how it happens in actual scenario. To begin with sizing will be done through which dimensional model of all subsystems will be arrived at using design calculations. Structural analysis will be carried out using Finite Element Method (FEM) in order to ascertain design adequacy. The outcome of the project would be design of car hit simulator using which any kind of passenger protection system can be evaluated for its intended performance.***

Keywords: ***Passenger Safety, Cars, Impact, Friction, Structural analysis.***

Introduction: The automotive industry in India is one of the largest automotive markets in the world. It was previously one of the fastest growing markets globally, but it is currently experiencing flat or negative growth rates. In 2009, India emerged as Asia's fourth largest exporter of passenger cars, behind Japan, South Korea, and Thailand, overtaking Thailand to become third in 2010. As of 2010, India was home to 40 million passenger vehicles. More than 3.7 million automotive vehicles were produced in India in 2010 (an increase of 33.9%), making India the second fastest growing automobile market in the world (after China). India's passenger car and commercial vehicle manufacturing industry recently overtook Brazil to become the sixth largest in the world, with an annual production of more than 3.9 million units in 2011.

While Indian government is still planning to bring in safety reforms with regard to braking and supplemental restrain system (SRS) airbags, which are already a standard in developed markets, the safety rating is expected to force car manufacturers to go for higher safety devices. In Europe, cars with higher NCAP ratings attract much lower premium and are preferred by customers. In India, currently most of the cars do not come with critical safety features like airbags or anti-braking system as standard equipment.

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

DESIGN SPECIFICATIONS

The objective of this project is to design a car hit simulator for evaluating the performance of safety system installed in a car in hit scenario. Specifications (Requirements) form the starting point for the design. In order to frame the specifications hit potential measured in terms of acceleration experimentally [R] is shown in figure below.

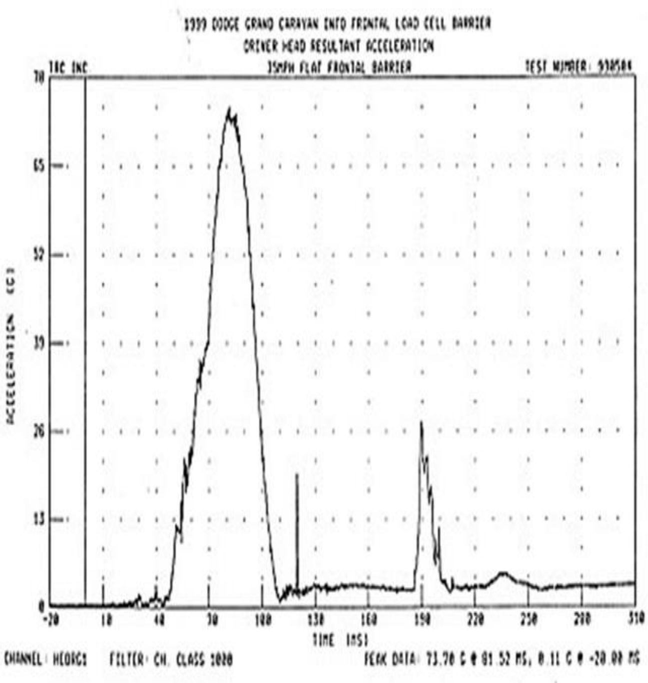


Figure: Acceleration time history measured during hit

Y axis in the above plot represents the magnitude of acceleration and the X axis represents the time duration. From this plot it is evident that magnitude of the acceleration is of the order of 75 g.

DESIGN PHILOSOPHY

The proposed simulator should simulate the above shown acceleration up on its use. The velocity corresponding to the acceleration is calculated as following.

Velocity = Acceleration x time = 75 x 9.81 x 0.0025 = 1.84 m/sec

(Time taken to attain acceleration is assumed to be 2.5 millisecond)

This velocity has to be attained by linearly. In order to achieve such high velocity means capable of generating instantaneous (Sudden) value of high displacement is to be employed.

Displacement corresponding to the velocity is calculated as following.

Displacement = Velocity x time = 1.84 x 0.05 = 0.092 m = 92 mm (Rounded to be 100 mm)

(Time taken to attain acceleration is assumed to be 50 millisecond)

It is planned to use snail cam to generate so high instantaneous displacement. A complex requirement associated with this project is to simulate impact kind of excitation which demands for withdrawal of force immediately after application of force within no time. Moreover displacement produced by snail cam needs to be transformed to impactor. These two requirements are planned to be met by close coiled helical spring. However a plate like structure (Here after it will be referred as spring plate) needs to couple cam follower and spring is required, design of which is complex as it has to resist the impact load that is generated during functional operation of the proposed simulator. In order to derive the expected gain from spring one end should be floating and other end should be rigid.

Floating end has to be connected to the spring plate. In order to maintain the other end of the spring as rigid a rigid structure, integral to the main base of the simulator. In order to further couple the motion generated due to the combination action of cam and spring, another plate (Here after it will be referred as impactor plate) is required to be provided in the design which should act as an interface between spring plate and impactor. A cage (Here after it will be referred as target cage) like structure is needed for supporting the seat along with passenger (Dummy) and safety system. However door which needs to receive impact from impactor is also has to be attached to the front potion (Towards impactor) of target cage. This target cage should be like skeleton structure so that it enables easy positioning and withdrawal of test dummy (Door, seat

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

and safety system). During simulation of hit environment both impactor and target will be set into motion. In order to enable smooth motion of these two systems, a mechanism comprising of guides along with wheels is needed. In order to absorb the impact energy finally an energy absorber is required. This energy absorber is required to be provided on other end (Opposite end of impact) of target cage.

Outline sketch of the proposed design based on above design philosophy is shown in figure below.

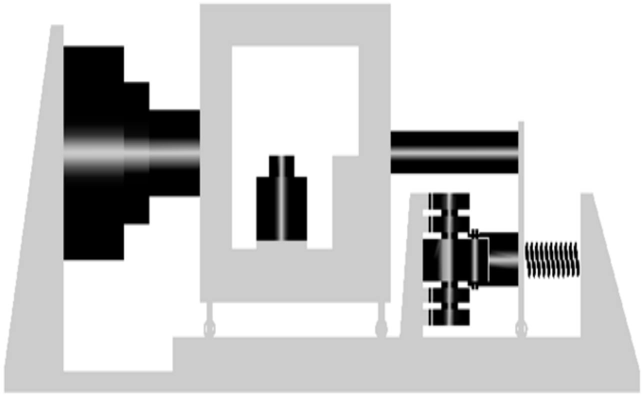
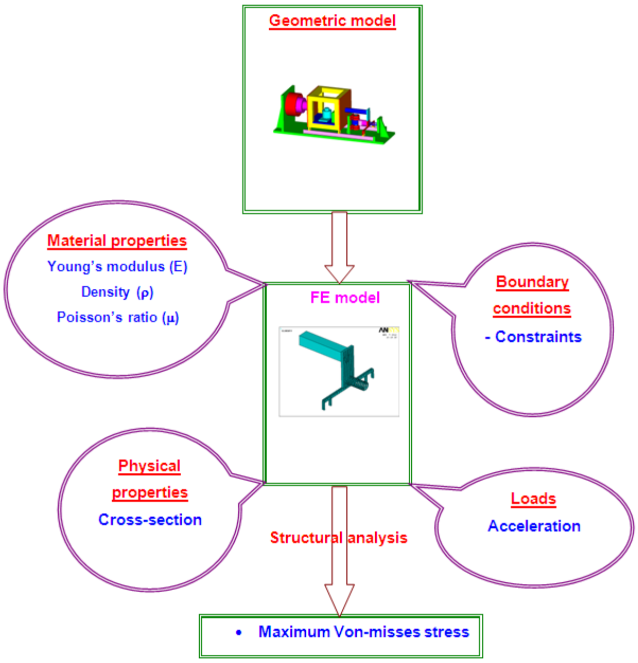


Figure : Outline sketch of the proposed design

STRUCTURAL ANALYSIS OF CAR HIT SIMULATOR USING FINITE ELEMENT METHOD (FEM)

Structural analysis of simulator is carried out using Finite Element Method (FEM) against the functional loads in order to assess the design adequacy. To begin with static analysis is carried out. Apart from this modal analysis is also carried out to identify the first natural frequency of the simulator. Then the analysis is extended to harmonic analysis. Finite element analysis gives out the maximum stress developed in all the subsystems. The outcome of the analysis would be useful in ascertaining that the design is safe.

For carrying out the analysis all the load bearing members of the intended system are considered. The basic approach followed in finite element analysis is represented in form of flow chart below.



FE modeling

All the subsystems are built up separately and then assembled. Geometric model of the assembly is shown in figure below.

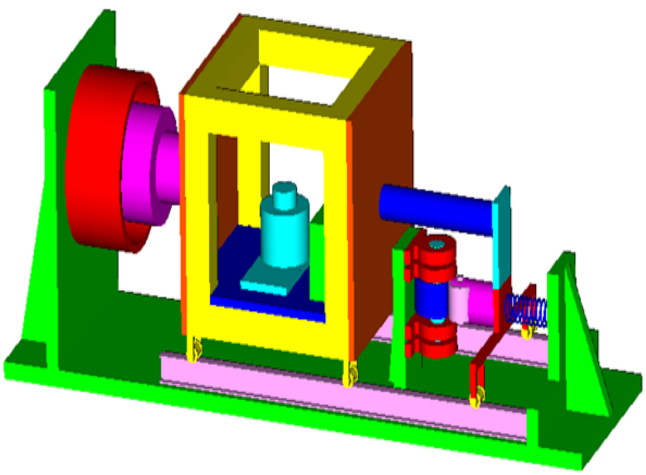


Figure: Geometric model of the simulator

Geometry model is converted into FE model by discretizing the each individual subsystem. Different types of elements used for each subsystem is given in Table below.

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

Sl. No.	Sub system	Type of element
1.	Impactor	Beam elements (BEAM 4)
2.	Spring	
3.	Spring plate	Shell elements (SHELL 63)
4.	Impactor plate	
8.	Energy absorber	Considered as damping (5%) during harmonic analysis

Table: Elements used for subsystems

Results of different analyses carried out using FEM are summarized in Table below.

Sl. No.	Analysis	Maximum Value	Allowable value	Factor of safety
Static				
1.	Von Misses stress	94.3 MPa	980 MPa	10
Modal				
1.	Fundamental frequency	12.5 Hz	---	---
Harmonic				
1.	Dynamic bending stress	346 MPa	980 MPa	2.8

Table: Summary of FEA results

VALIDATION OF RESULTS

The objective of this exercise is to validate the design calculations with that of FEA there by establish the confidence in the design procedure adopted. However validation is limited to maximum Von Misses stress corresponding to static analysis only as the theoretical formulation available is limited to static loading only as the present configuration is complex in nature. Outcome of some of design calculations are compared in Table below.

Sl. No.	Parameter	Design calculation	FEA
1.	Von Misses stress – Static analysis (Spring plate)	96 MPa	94.3 MPa
2.	Von Misses stress – Static analysis (Impactor)	0.06 MPa	0.068 MPa

Table: Comparison of results

OBSERVATIONS

- Minimum available factor of safety in case of static loading is 3 which is more than desired value i.e. 1.5 hence the design is safe again static loading.
- Available factor of safety in case of dynamic loading is 2.8 which is more than desired value i.e. 1.5 hence the design is safe against dynamic loading also.
- Available factor of safety in case of dynamic loading is less than that of static loading.
- First natural frequency i.e. fundamental frequency of the intended system is found to be 12.5 Hz.
- Result obtained using design calculation is matching well with that of FEA which indicates the degree of confidence associated with the design procedure adopted.

CONCLUSIONS

- These days different cars are coming up with different types of passenger safety systems which needs to be evaluated before they are installed in cars and put in use.

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

- This project aims at design of such system which evaluated the ensure functioning of the passenger safety system.
- The intended system seeks the particular passenger safety system and then it simulates the car hit environment on the safety system during which its functionality can be checked.
- This design also consists of seat over which dummy simulating passenger will be positioned. As part of the project a platform is designed over which passenger safety system is positioned.
- Initially all the elements are designed and dimensions are framed.
- These dimensional models are transformed in form of assembly of the total system.
- Then the structural analysis is carried out to check the design adequacy of the intended system.
- The outcome of the project is design of test rig using which functional evaluation of any kind of passenger safety system can be done.
- The same simulator can be extended for any type of automobile vehicle.
- Available factor of safety in case of both static and dynamic loading is more than desired value and hence the design is safe against both static and dynamic loading environments.
- Available factor of safety in case of dynamic loading is less than that of static loading.
- Result obtained using design calculation is matching well with that of FEA which indicates the degree of confidence associated with the design procedure adopted.

FUTURE WORK

A working model will be realized as a part of extended future work in order to demonstrate the functioning as anticipated during design. Once after gaining practical confidence concept can be extended for full scale. Enormous scope of adopting other types of loads will be the immediate future work.

REFERENCES

1.Motoaki Deguchi," Simulation Of Motorcycle-Car Collision", 05-0041, Technical paper.

2.Stephan Eichler, "Simulation of Car-to-Car Messaging: Analyzing the Impact on Road Traffic", Technical paper.

3.S. Heimbs, "Crash simulation of an F1 racing car front impact structure", 7th European LS-DYna conference, 2009.

4.E. Nassiopoulos, "Finite element dynamic simulation of whole rallying car structure :Towards better understanding of structural dynamics during side impacts", 8th European LS-DYNA Users Conference, Strasbourg - May 2011.

5.Gary R. Consolazio, "Impact simulation and full scale crash testing of a low profile concrete work zone barrier", Computers and Structures 81,1359–1374, 2003.

6.Rob Moncur, "F1 Pace Car: Crash Analysis", Technical report, 2010.

7.A. Calienciug, "Design and FEA crash simulation for a composite car bumper", Bulletin of the Transilvania University of Brasov Series I: Engineering Sciences • Vol. 5 (54) No. 1 – 2012.

8.Yucheng Liu, "Impact Experimental Analysis and Computer Simulation", Department of Mechanical Engineering, University of Louisville.

9.Satyan Chandra1, "Crash Simulation of PACE Formula-1 Race Car", Computer-Aided Design & Applications, PACE (1), , 31-46, 2011.

10.Boria, "Crash analysis of an impact attenuator for racing car in sandwich material", Department of Mechanical, Nuclear and Production Engineering, University of Pisa, Italy.

Author Details:

I. Murali Mohan