ISSN 2395-1621

Developing a methodology to Simulate cracking of a glass during conditions of crash

[#1]Pavan Gorde, [#2]N. Gangawate ,[#3] Kiran More

[1]pavan.gorde1@gmail.com

[2] nipun.g.90@gmail.com

[3]kiran.imagine67@gmail.com

[#12]Mechanical Design Engineering, Savitribai Phule Pune University
Pune

ABSTRACT

Rollover simulation has become an integral part of a vehicle manufacturer, since the proportion of deaths caused by rollover accidents are high on road. The rollover test and simulation are done based on the European standard regulations. The main aim of the regulation is that the bus super structures should not penetrate into the survival space during testing. AT Technologies being a global leader in crashworthiness is conducting research on laminated glass for accurate crash analysis. This project deals only with the behavior of laminated glass, its simulation and evaluation of its strength.

The objective of this project is to develop a methodology for determining the behaviour of the laminated glass under various conditions of load application. In this project experiments are performed on laminated glass sheets. A simulation test is executed by applying the same loading conditions using hypermesh and LS Dyna. The experimental test results are used to code the material properties of the glass in LS dyna. By simulating the experiments using various possible modelling techniques and codes a methodology is developed by the use of which the simulation results match exactly with the experimental results. The methodology in this project will act as a guide to the use of laminated glass in windows of the vehicles introduced in future, help to build the full vehicle model and analyse it for crash analysis.

***Keywords*— laminated glass, ls-dyna, windshield, crash analysis of windshield**

ARTICLE INFO

Article History

Received :18th November 2015

Received in revised form :

19th November 2015

Accepted : 21st November , 2015

Published online :

22nd November 2015

I. Introduction

Impact against safety glass is of interest for example to civil engineers and car makers. The motivation for the work presented in this paper results from pedestrian protection in accidents involving a car. One scenario is that the pedestrian impacts the windshield with his head. In order to save the life of the pedestrian the acceleration of the head should not exceed a certain value. In addition it is important that the head does not touch any part inside the car (for example the steering wheel). Today the development process for cars is based on numerical simulations. For simulations regarding pedestrian protection explicit finite element solvers are used. The correct calculation of impactor acceleration and deformation of the glass is still a challenge. A simulation model must be able to predict the initial failure of the glass and the crack propagation. Several experimental and numerical studies have been carried out in the past to investigate the behaviour of laminated glass under dynamic loading conditions.

The first successful frontal full car crash simulation: a Volkswagen Polo collided with a rigid concrete barrier at 50 km/h (ESI 1986). In the following years, German car makers produced more complex crash simulation studies, simulating the crash behaviour of individual car body components, component assemblies, and quarter and half car bodies in white (BIW).

Fig. 1. Laminated glass windshiel

These experiments culminated in a joint project by the Forschungs gemein schaft Automobil-
Technik (FAT), a conglomeration of all seven German car makers (Audi, BMW, Ford, Mercedes-Benz, Opel, Porsche, and Volkswagen), which tested the applicability of two emerging commercial crash simulation codes. These simulation codes recreated a frontal impact of a full passenger car structure (Haug 1986) and they ran to completion on a computer overnight. Now that turn-around time between two consecutive job-submissions (computer runs) did not exceed one day, engineers were able to make efficient and progressive improvements of the crash behavior of the analyzed car body structure.

II. EXPERIMENTAL SETUP

Total three series of experiments were performed on the laminated glass windshield. Brand new windshields of Maruti Omni van were used in each set of experiments. The first set of experiment consisted of a drop test, the second test was a quasi static loading test and the final test was the buckling test.

In the first set of experiment an ms rod was dropped on the windshield from a certain height. The objective of the test was to record the height of the drop object from which the crack generation starts in the windshield after impact. An ms rod 27 inches long and 28 mm diameter. The recorded weight of the rod was 3.4 kgs.

The drop height of the rod was being increased gradually until the crack generation was observed in the windshield. Immense precautions were taken so
that the rod axis remains parallel to the ground surface during impact. It was observed that the crack generation starts when the rod is dropped from a height of 21 inches from the surface of glass.

Fig. 2. Measuring drop height

The second set of eqperiment was performed was of quasistatic nature. The windshield was placed on the plane surface of the floor and it was gradually loaded. To ensure quasistatic loading a 60 lit capacity can was first placed on the windshield and it was filled with water at a constant flow rate. It was observed that the windshield started failing as the can was filled over 52 liters.

The final test performed was the buckling test. A fixture was

designed in such a manner that it holds the windshield and applies load vertically over it.

Fig. 3. Cracks after drop test.

The fixture consisted of two grooved wooden logs, an ms c-channel and two ms pipes. The pipes were welded to the c-channel such that they are perpendicular to the channel surface. Both the wooden logs were drilled on both the ends in such a manner that they could pass easily through the vertically welded ms pipes in upward and downward direction. Four metal strips were welded on the bottom of the c-channel to provide support to the fixture during loading.

Fig. 4. Buckling test fixture.

The first wooden log was slid down through the vertically welded pipes such that its groove is exposed. After proper placement of the log, the smaller edge of the laminated

windshield is placed in the groove. The second wooden log is then slid through the vertical
pipes and placed over the windshield such that the upper edge of the windshield is placed exactly into the groove of the log. Arrangements are made that the windshield is properly held between the two logs

Fig. 5. Loading during Buckling test

After the complete fixture setup is ready, the upper wooden log is loaded in steps by adding weights as shown in figure. The vertical deflection of the upper log is noted as the loading is increased. Failure occurs in the windshield at around 110 kg loading. The table shows the load vs deflection in the two windshields that were used in the buckling test.

Table 1. Load v/s. deflection.

Load (kg)	Deflections (mm)	
	Windshield 1	Windshield 2
39.05	5	5
17.86	8	7
14.18	9	9
13.35	1	11
5.4	11	12
5.2	13	14
5.1	15	14
3	15	15
2.25	16	17
2.15	16	17
1.65	18	19
Total=109.19		

Fig. 6. Side view of buckling test. Simulation

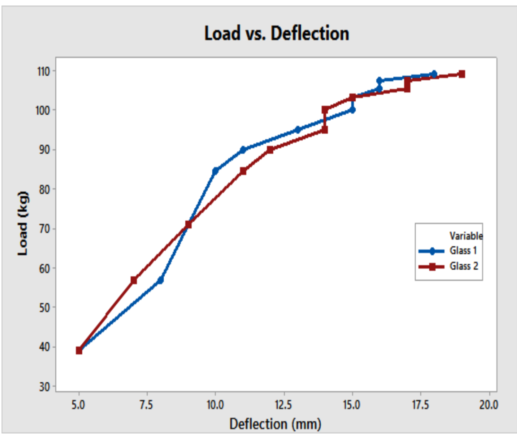


Fig. 7. Load Vs Deflection

III. SIMULATION RESULT

For simulating the experiment we first have to develop exact models of the components used in the testing. These models are then meshed to create finite elements over them for the analysis. After the meshing process loading conditions and degree of freedom constraints are applied to the model. Later the models are loaded in a solver software for the simulation process.

Siemens Unigraphics 9 or NX-9 was the tool used for basic modelling of the components. For meshing and application of loading condition Altair Hypermesh 12.0 was used. A high end analytical simulation software LS-DYNA is used for the process of simulation.

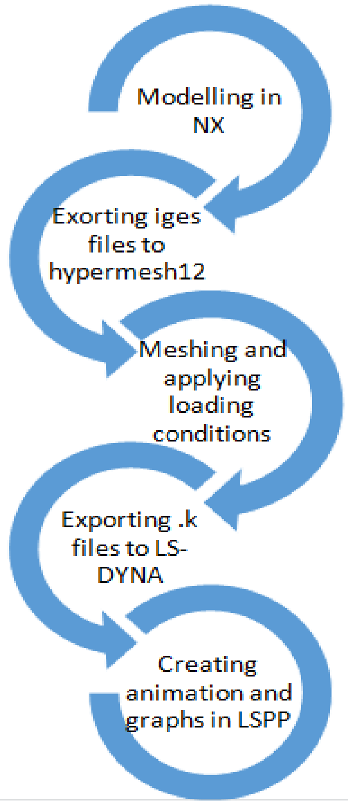


Fig. 8 Simulation methodology

Drop Test:

In the beginning, the basic element of the project, the windshield is modelled in the NX-9 software. The exact dimensions of the windshield are noted by actual measuring the sheet after placing it on the ground. After developing the model, the nx file is converted into iges (*Initial Graphics Exchange Specification). The iges format files are accepted by almost all the new modelling as well as analysis softwares.*

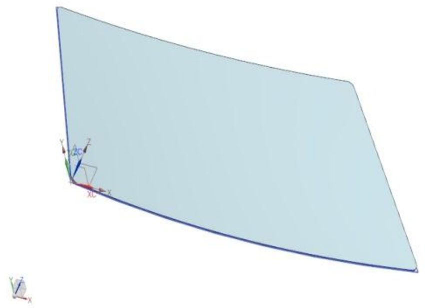


Fig. 9 Windshield model

After creating the model in nx-9 it is saved as an iges file. This iges file is then loaded in the hypermesh 12.0 software. The rod is designed in the hypermesh software itself. The glass model is meshed in 2d option i.e. after meshing the glass only shell elements are obtained on the surface.

In case of the rod, first the cross section of the rod is created and meshed using shell elements. This profile of the rod is then dragged along the length of the rod which is 27 inches long for creating the model.

For assigning the material properties the glass is assigned MAT_L_32 card and the rod is assigned MAT_L_20 card. The MAT_L_20 card is defined as a rigid material. The properties like density, Young's modulus, Poisson's ratio can be assigned manually to the applied card.

The velocity of the rod can be calculated by the formula:

$v^2 = u^2 + 2as$

Where,

v: final velocity

u: initial velocity

a: acceleration due to gravity

s: displacement

Thus we get the value of v as

$$v = \sqrt{(2as)}. = 3.22 \text{ m/s}$$

And the kinetic energy will be calculated as

k.e. = $0.5*m*v^2$ = 17.627 joules.

V= = where m is the mass of the rod (3.4 kg).

After assigning the constraints and the loading values to the glass and the rod, the model is exported into the LS-DYNA format (.k file). The high end solver LS-DYNA is opened and the converted file is loaded into the solver. The process of simulation is started. The solver generates output files and stores in the folder created by the name of the model.

LS-DYNA has a preprocessor and a post processor called as the LS PREPOST (LSPP). The preprocessor is used for modelling but is not generally preferred because of its complexities. To view the animation of the experiment the d3plot file generated by the solver is opened in the LSPP. As the file opens in LSPP, the animation toolbar opens where we can set the fps and the animation speed.

The screenshot of the animation can be seen in Fig 10

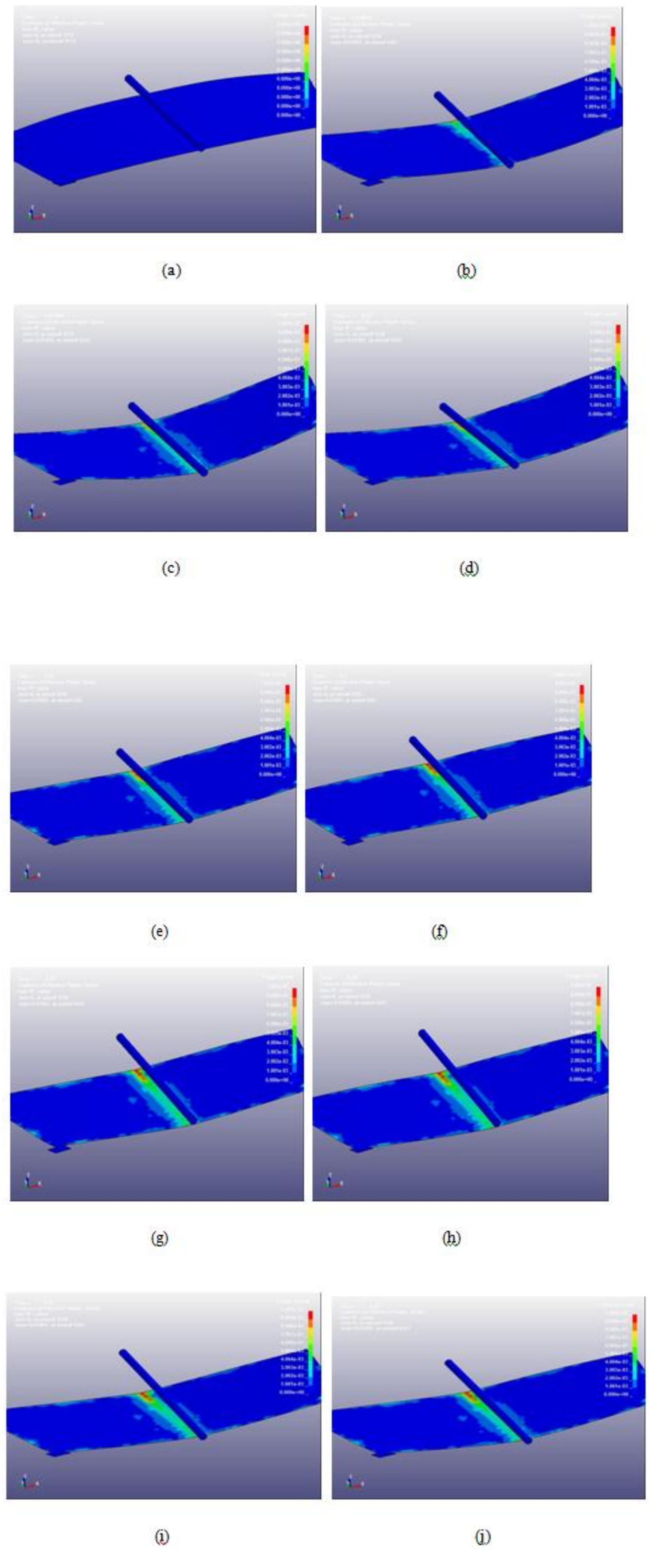


Fig. 10 Screenshot of drop test animation showing plastic strain

The card only defines the material behavior. A number of iterations are done by changing the properties of the card allotted to the elements until the simulation matches to the experimental results accurately.

In the drop test, the first crack was generated when the rod was dropped from a height of 21 inches (53 cm). In the simulation process the rod is not dropped from the height, instead the kinetic energy that the rod gains just before the collision is assigned to the rod. For calculating the kinetic energy we need the velocity of the rod just before coming into contact with the windshield.

Buckle Test:

In the buckle test simulation only two main elements are required i.e. the windshield and the wooden logs. As mentioned in section 2.3.3 grooves are created on the wooden log and a chamfer is provided on one of the edge of the groove. Considering the same, a cad model is developed in NX-9 using the available tools.

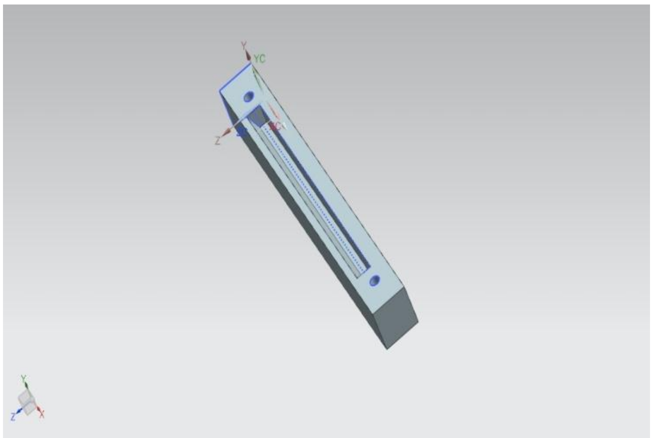
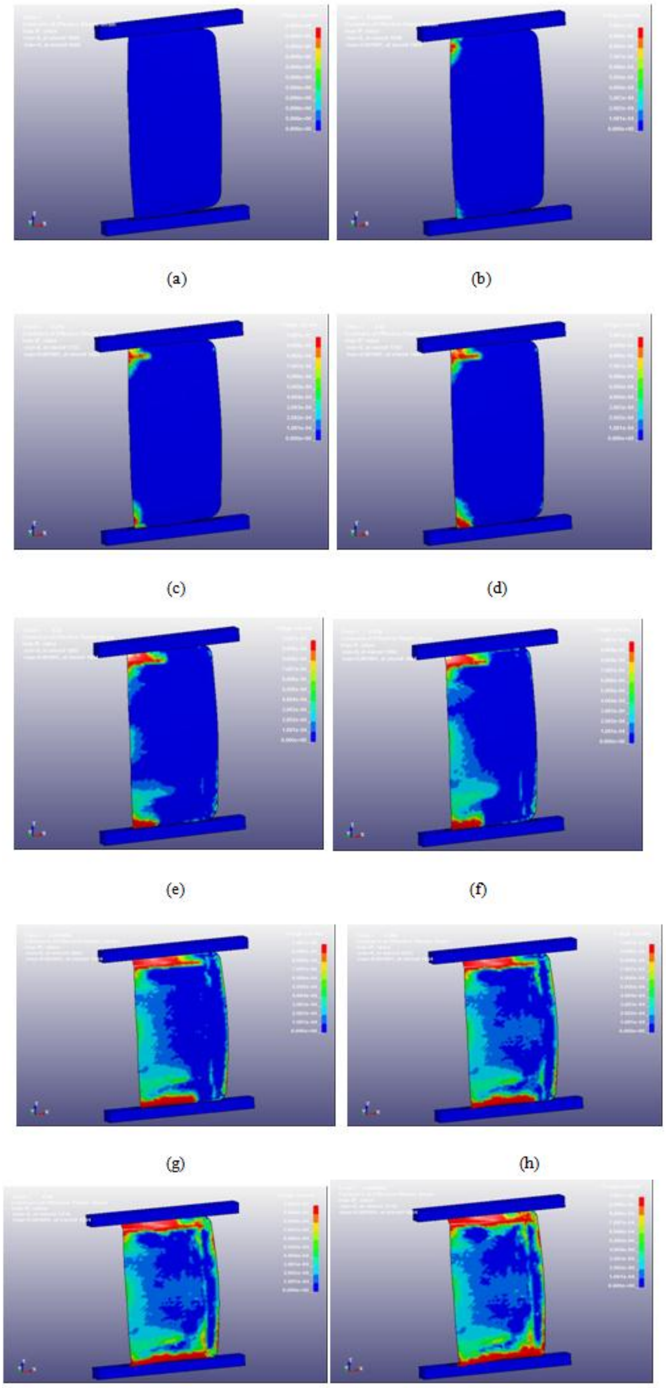


Fig 11 View showing chamfer on the groove edge

Below is shown the simulation results of buckling test.
The same procedure is followed here. The fixture assembly is loaded in the Altair hypermesh 12 software. Glass is meshed using shell elements. The loading conditions are applied on the upper wooden log. The bottom wooden log is to be kept fixed.



(a) (b) (c) (d) (e) (f) (g) (h)

(i) (j)

Fig. 12 Screenshots of Buckling test simulation showing plastic strain.

A graph of load v/s deflection is plotted with the help of excel files generated by the LSPP

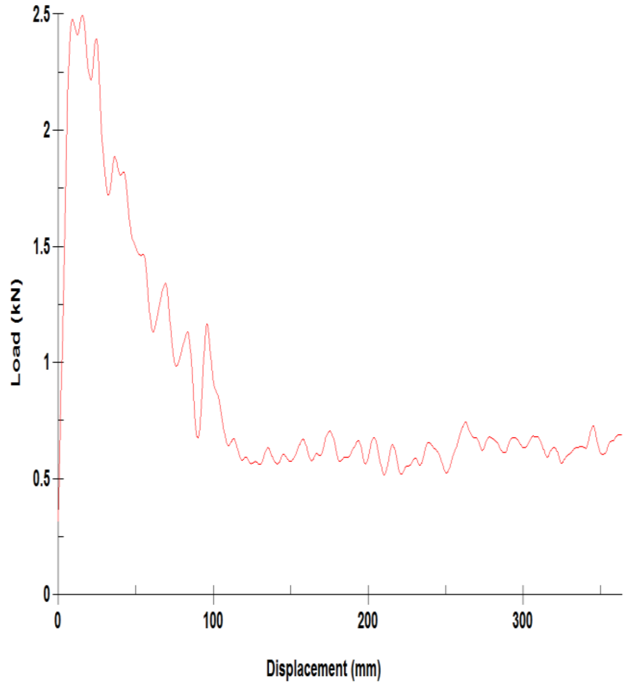


Fig. 13 Load vs Deflection graph

IV. OBSERVATION

Comparing the experimental data and the simulation results, it can be seen that the behavior of glass matches in both the cases. A good correlation is observed between the simulation and experiments. The simulation and experimental graphs of buckling test shows same deflections before breakage.

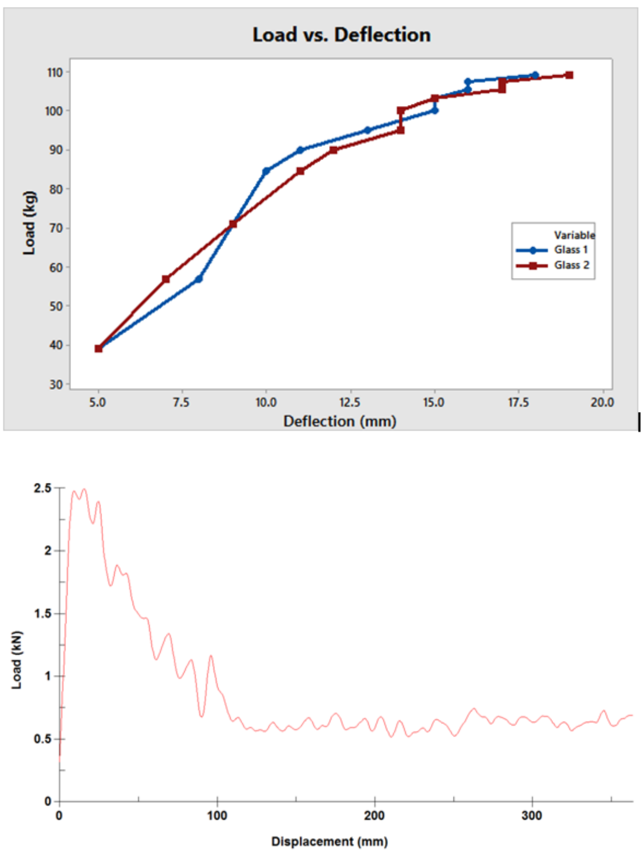


Fig. 14 Comparison between experiment and simulation.

V. CONCLUSION

A simple windshield investigation and material model has been developed for crash analysis of the windshield. Experimental tests were conducted for Saint Gobain Laminated windshield for Maruti Omni van to determine the plastic strain responses under static and dynamic loading conditions. The test data was used to identify the material properties used to model the adhesive bond layer. The methodology has been validated by simulating the same tests with proper boundary conditions in Hypermesh and LS-DYNA solver. The CAE simulation correlated well with the test data.

To include failure of laminated windshield into FE models will significantly improve the accuracy of the results in crash simulations and the correlation to the actual performance. The improvement of CAE tools for laminated glass in automotive design and its continuous optimization will help to extend the use of glazing in the future..

ACKNOWLEDGEMENT

Any achievement, be it scholastic or otherwise does not depend solely on the individual efforts but on the guidance, encouragement and cooperation of intellectuals, elders and friends. A number of personalities, in their own capacities have helped me in carrying out this project work. I would like to take this opportunity to thank them.

I would like to thank my P.G. guide, Mr Kiran More, Professor, Department of Mechanical Engineering, FIT, Pune and my company guide Mr. A. Bhore. With their enthusiasm, inspiration and their great efforts to explain things clearly and simply, they enabled me to develop an understanding of the subject. Throughout my project work, they provided encouragement,, sound advice, good teaching and lots of
good ideas.

REFERENCES

[1] http://soar.wichita.edu/handle/10057/7036

[2] T. Pyttel, H. Liebertz , J. Cai "Failure criterion for laminated glass under impact loading and its application infinite element simulation", University of Applied Science Giessen-Friedberg, Wilhelm-Leuschner-Str. 13, 61169 Friedberg, Germany; Volkswagen AG, 38436 Wolfsburg, Germany; ESI GmbH, Mergenthaleralle 15-21, 65760 Eschborn, Germany

[3] Naman Gupta, "Head impact evaluation on to conventional tempered glass and laminated glass car windows in side impact and rollover accidents" , Wichita State University M.S. thesis paper, 2013

[4] Yong Peng, Jikuang Yang, Caroline Deck , Remy Willinger, "Finite element modelling of crash test behavior for windshield laminated glass", International Journal of Impact Engineering 57 (2013) 27e35