OPTIMIZATION OF INSTRUMENTAL PANEL TO REDUCE TRAUMATIC BRAIN INJURY

Shreyansh Vijay Patil[1], Prof. S. A. Pachapuri[2]

[1]Department of Mechanical Engineering (M.Tech in Design Engineering)
KLE Dr. M S Sheshgiri College of Engineering and Technology Udyambag, Belagavi, Karnataka, India 590008
[2]Department of mechanical engineering (professor)
KLE Dr. M S Sheshgiri College of Engineering and Technology Udyambag, Belagavi, Karnataka, India 590008

---***---

Abstract - *There is increase in the fatalities day by day due to vehicle crash which is great challenge for automobile industry. To analyse this situation and to reduce the occupant injury due to impact on interior component such as instrumental panel (IP) a new safety protocol Economic Commission for Europe R-21 Regulation is developed to evaluate head impacted on interior parts to meet these new protocol by maintaining a good design consideration. Attempt is made to study and analyse the head impact on instrumental panel of an automobile and comes with different design iteration suggestion. The impact situation is evaluated by Head Impact Criteria (HIC) of dummy this attempt will helpful to reduce the injury or damage of head, this in turn reduces the total development time of an automobile*

If in case the air bag fails to open properly when vehicle strikes on another vehicle or rigid wall the high frontal impact of driver and passengers take place on instrumental panel. To analyse this concept in this project the instrumental panel and head form is design in catia/solid edge modelling software. The model is imported in to the Hypermesh software and meshing is done. Later impact analysis is done by importing the meshed model in the LS-Dyna software. After the analysis, the resultant acceleration vs time graph is generated. For the safe design point of view the HIC value should not more than 1000. If the HIC value is more than 1000 at that particular point, then there will be head injury will takes place. By reducing the thickness at that point, stiffness will reduce so that more deformation of instrumental panel is takes place at that point and more energy is absorbed and thus the HIC value can be minimized below 1000 value.

Key Words: **Headform, instrumental panel, HIC value, ECE R21 regulation, Hyper-mesh software, LS Dyna software,**

1. INTRODUCTION

There is increment in the fatalities day by day because of the vehicle crash which is very important challenge for an automobile industry, hence for that reason lots of safety features has been developed for example air bags and seat belts, which "reduces" the inhabitant damage in considerable amount, however the current investigations and insights are demonstrating that there is lot of fatalities are accounted for because of impacts on inside components of vehicle. Hence to analyse this situation and to reduce the occupant injury due to impact on interior component, a new safety protocol Economic Commission for Europe R-21 regulation (ECE R21) is developed to evaluation of the head impacts on interior component. An attempt is made to examine, and analyse the head impact on IP (Instrumental Panel) of an automobile and come with different design suggestions. The head impact on panel and conditions is assessed by HIC (Head Impact Criteria) of dummy, these attempts will be helpful to minimize the injury or damage of head and beside numerous design iteration can be reduced. And this HIC value can be calculated through L-S Dyna software through graphs if the HIC value is greater than 1000 value then then design is not safe if less than that then design is safe.

1.1 Problem Statement

"Blunt trauma" is a serious type of head injury which will occur in associate in nursing automobile accident, once a moving head strikes a stationary object just like the windscreen, instrumental panel. To fulfill this requirement the head impact criteria for base line model, to check head impact criteria (HIC) value which is impacted on to the instrumental panel. Do some modifications such that head impact criteria value should be in the given standard range (i.e. below 1000). The impact analysis is carried out in the LS-Dyna.

1.2 Objectives

- Developing a CAD-model in solid works.
- Developing the mesh model using the Altair Hypermesh-14 tool.
- Checking the quality criteria of mesh model such as aspect ratio, jacobian, skewness, warpage etc.
- CAE Analysis is carried out using the LS-Dyna tool and proper cards are specified.
- To achieve the HIC value below the 1000 value regulation according to the ECE R21 which contains the "uniform provisions concerning the approval of vehicles with regards to their interior fittings".

- To optimize the instrumental panel in such way that it can absorb more energy from headform i.e. It should follows the crashworthiness property.
- The design variable are chosen as the thickness of rib and trim, the depth, spacing and number of ribs.

2. ECE-21 REGULATION STANDARD

A free motion headform (FHM) impactor hits the upper interior parts with a velocity of 24 km/h (which is the average velocity of headform, when the vehicle strikes on to another vehicle or rigid wall to the instrumental panel. Free motion head impactor data: Mass of the headform is 4.54 kg, (which is average weight of the adult human head.) Head form according to SAE J921 and J977 including triaxial acceleration sensor.
HIC Calculation:-

The HIC value can be calculated analytically by following formula;

$$HIC = (t_2 - t_1)\left[\frac{1}{(t_2 - t_1)}\int_{t_1}^{t_2} a\,dt\right]^{2.5}$$

HIC > 1000 (serious brain injury) where; t_2-t_1 <36 ms

3. GEOMETRIC AND FINITE ELEMENT MODELLING

To carry out CAE simulation of any component , the solid model of the same is essential . Iti is also called body in white . In this chapter the detailed description of CAD models used in this project are discussed. The CAD modeling software package like SOLID EDGE is used to model the components of this project.

To understand this phenomenon the modelling of the instrumental panel (i/p) and head-form (Head impactor) is done in solid edge software, as the dummy head impactor by considering the biomechanical norms.

3.1 Instrumental Panel and Head Impactor

The diameter of the head impactor is 165mm and the average angle between the head impactor and the instrumental panel i.e. H point is 50^0, the length of the instrumental panel is 1410mm. The CAD model of instrumental panel and head impactor is shown in figure 1 and 2 respectively.

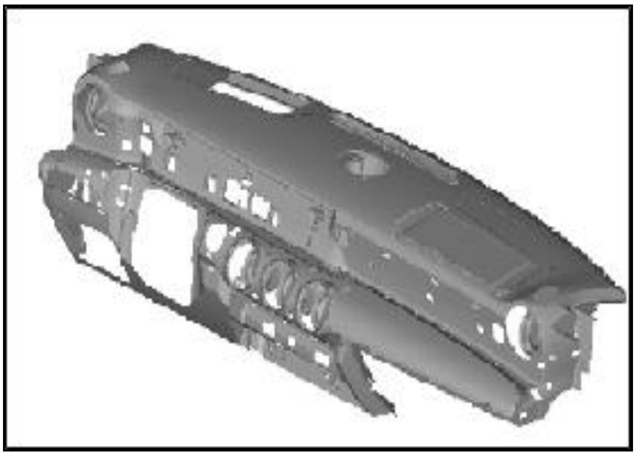

Fig 1: CAD model of instrumental panel

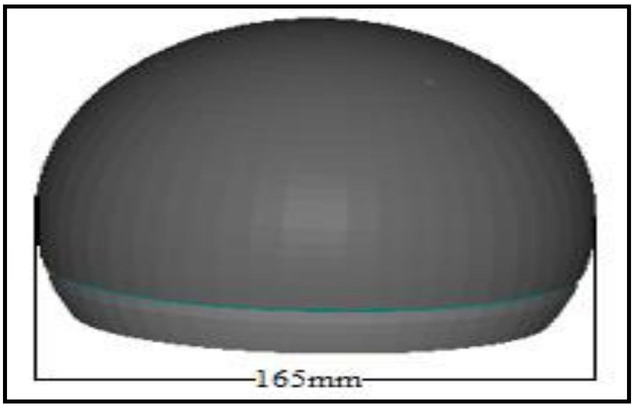

Fig 2: CAD model of head impactor

The meshing is done over the CAD model of instrumental panel and headform (dummy) with element size of 5mm with considering good practises of meshing criteria. The following figure 3 shows the assembly view of instrumental panel with impactor.

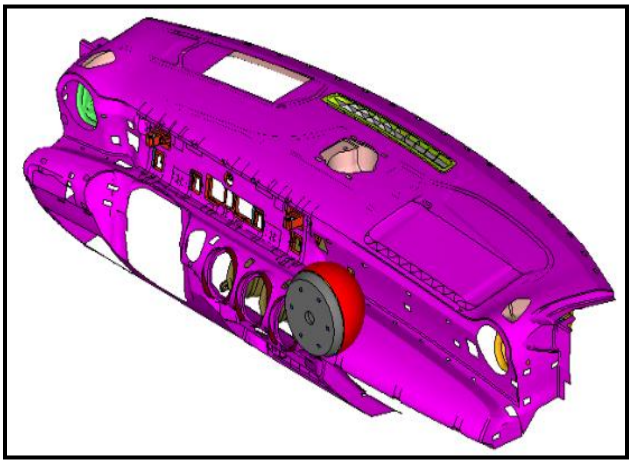

Fig 3: Assembly of instrumental panel with head impactor

3.2 Material and Properties

Most of the instrumental panel is generally made fibre material. In this project the materials used are ABS (Acrylonitrile Butadiene Styrene) And PPE (Polyphenylene Ether), and for headform the material and its properties used are shown in the table 2.

Table -1: Material and properties of instrumental panel

Part	Density mg/mm^3	Young's modulus (E) Mpa	Poisson's ratio
ABS	9.15e-10	1000	0.3
PPE	1.25e-9	1000	0.3

Table -2: Material and properties of headform

Part	Density mg/mm^3	Young's modulus (E) Mpa	Poisson's ratio
Cranium	1.80e-9	15.0e10	0.21
CSF	1.04e-9	12e-12	0.49
Skin	1.20e-9	16.7	0.42
Tentorium	1.14e-9	31.5	0.23
Face	3.0e-9	5.0e3	0.21
Falx	1.14e-9	31.5	0.23

3.3 Boundary Condition

In this project boundary conditions are applied to the instrumental panel. The head form of weight 4.5kg with the velocity of 24km/h is applied in X-direction towards the panel and in the figure 4 shows the boundary condition in which some part of the instrumental panel is fixed (constrain) at all degrees of freedom, as it is constrain all direction As the head is impacted on to the instrumental panel the analysis is carried out in LS- Dyna as explicit dynamics In this explicit dynamics the acceleration is found out.

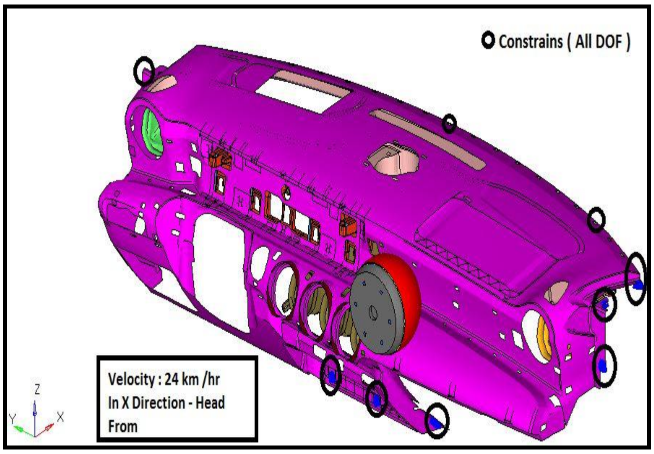

Fig 4: Boundary condition

3.4 Contacts

Accurate Contacts forms an integral a part of several large deformation issues. Correct modelling of contact interfaces between bodies is crucial to the forecast capacity of the finite part simulations. There are different types of contacts are there however in this project the single surface contact is utilizing. This contact is generally utilized as a part of contact option in Dyna, particularly for crashworthiness application. With this sort the slave surface is commonly described as a rundown of part ID's. Regardless surface is characterized. Contact is considered between each one of the parts in the slave list, including self-contact of each part. On the off chance that the model is precisely characterized these contact sorts are extremely accurate and reliable.

4. SIMULATION AND VALIDATION

From the analysis results which we got for the baseline model and the HIC value is generated which is compared with the standard value, i.e. according to the ECE R21 regulation HIC value should be less than 1000. As in such case that if the HIC value is greater than the 1000 then there will be damage of the head takes place. So the acceleration should be minimized, to minimize the acceleration the thickness should be reduce by decrease the material from instrumental panel (I/P), so that the stiffness will reduce and material gets deform more and it absorb more energy, and if thickness is reduce the mass or weight of the i/p also reduce at that particular point and i/p will get weaker at that particular point but the i/p is supported with the fem (front end module) part which is soften in nature and that instrumental panel is deform in fem only so there is no breakage of i/p and whatever deformation is occurring is well within the elastic limit so no question of permanent deformation, if though occurs then also the main thing is to save the life of passenger not the instrumental panel. The baseline model with different shoot points of instrumental panel is shown in Figure 5 below.

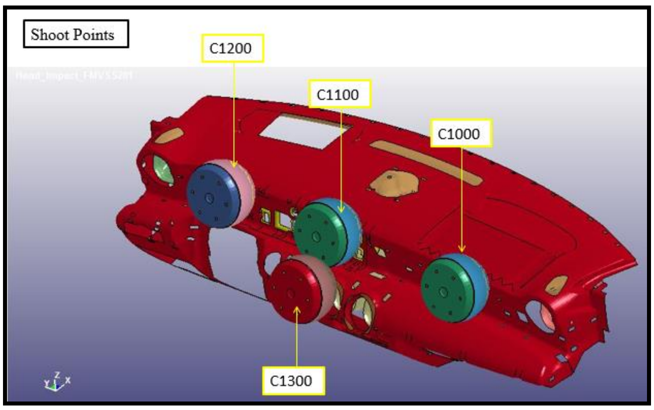

Fig. 5: Shoot points at baseline model

To understand this concept two shoot points are considered here i.e. shoot point C1000 and C1300 which is explained below.

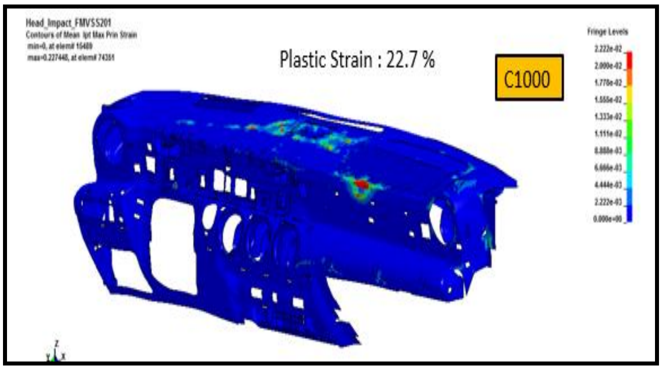

Fig. 6: Plastic strain at baseline point C1000

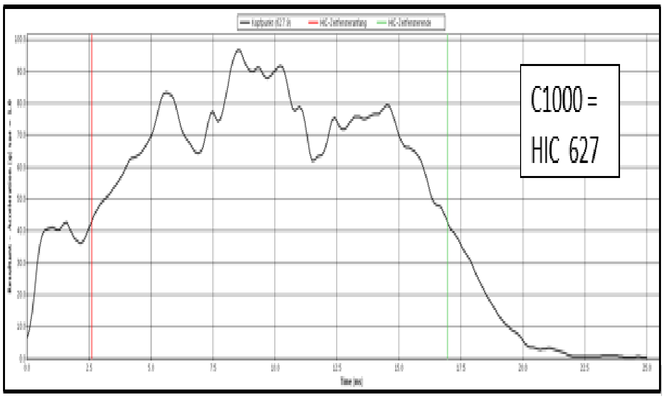

Chart -1: Acceleration vs time graph for shoot point C1000

From figure 6 and chart-1 we observed that for base line point C1000 we got analysis result of maximum principle strain plastic is 22.7% and for that the HIC value is observed through graph which is 627 respectively, and this 627 HIC is lesser than the 1000 value which means this design is safe no need to modify.

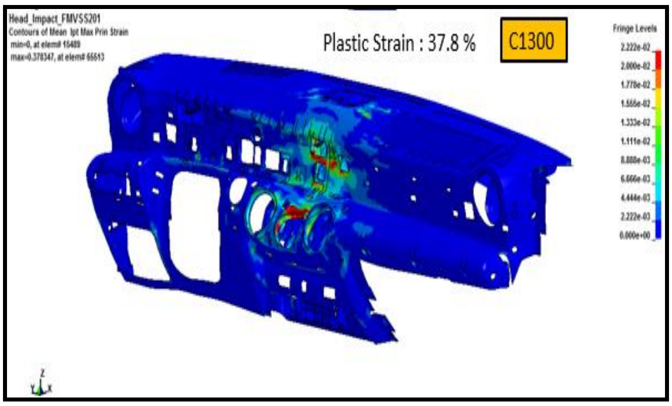

Fig. 7: Plastic strain at baseline point C1300

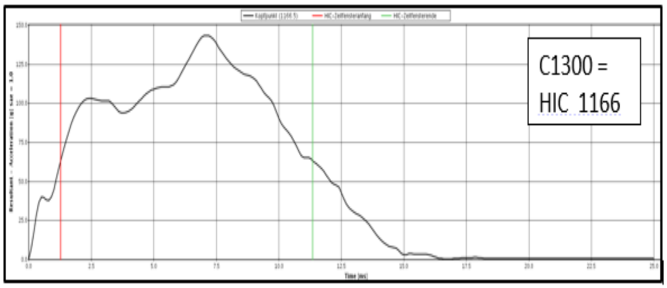

Chart -2: Acceleration vs time graph for shoot point C1300

From figure 7 and chart-2 we observed that for base line point C1300 we got analysis result of maximum principle strain plastic is 37.8% and for that the HIC value is observed through graph which is 1166 respectively, and this 1166 HIC value is greater than the 1000 value which means this design is safe not safe, need to modify.

Table-1: HIC and plastic stain values for different shoot points

Shoot Points	HIC Value	Strain in %
C1000	627	22.7
C1300	1166	37.8

4.1 Modified Model

There is need to modified in the design for the shoot point C1300 because it is exceeding the standard value of HIC ie.1000. Hence for design to be safe HIC value should not be exceed the limit. The rib structure plays a very key role here. Which strengthen the component and stiffness of component is increases. When two surface meets the corner in between these surface take place, and wall thickness up to 1.4 times the nominal wall thickness, due to this differential shrinkage and stress in model and longer cooling time takes place. Therefore at sharp corner risk at failure increases. Here at particular shoot point variable thickness is provided in the region of more stiffness with base of 2.5mm and upper layer with 2mm from 3mm and 2.5mm respectively.

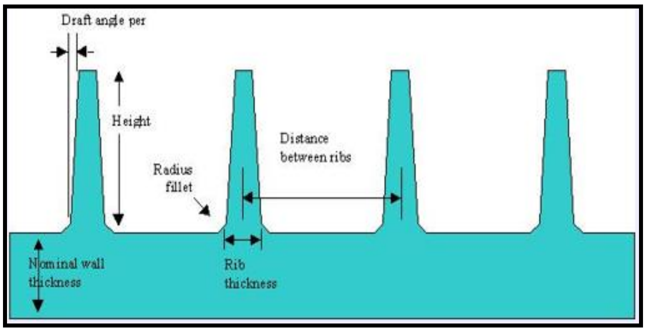

Fig. 8: Rib structural dimension

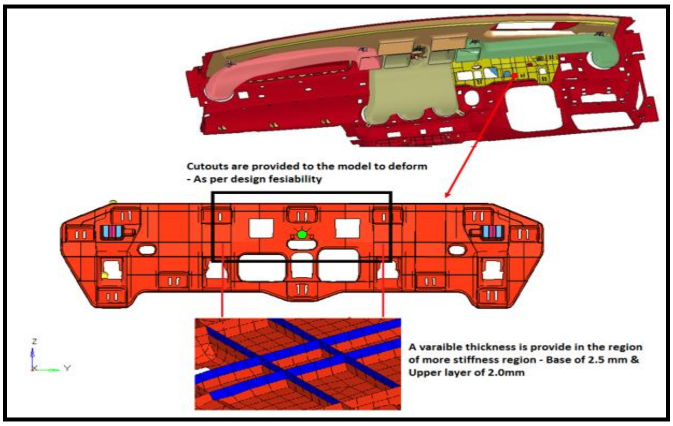

Fig. 9: Modification in design model

The impact analysis is done for the modified baseline design and we got the different result of plastic stain which is shown in the below figure 10. Here we can observed that there is increase in the plastic strain i.e. from 37.8% to 92.5% for shoot point C1300.

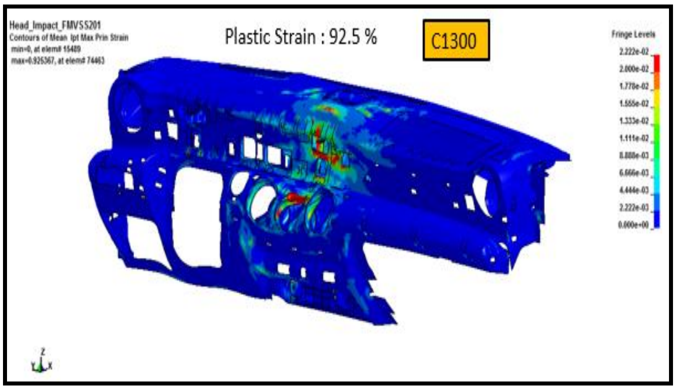

Fig. 10: Plastic strain at baseline point C1300 (Modified)

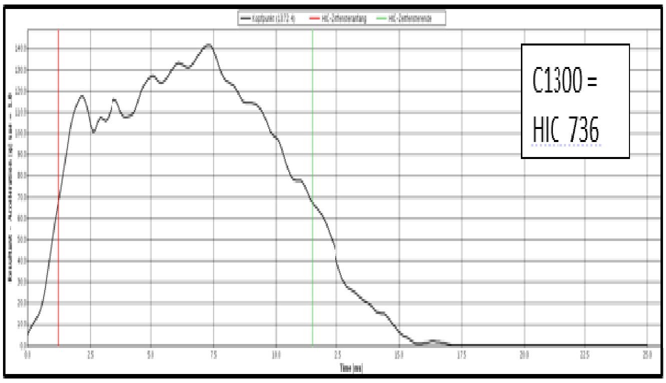

Chart -3: Acceleration vs time graph for shoot point C1300 (Modified)

From the graph which is mentioned as chart-3 we observed that for base line point C1300 we got analysis result of plastic strain is 92.5% and for that the hic value is observed through graph which is 736 and it is lesser than the 1000 value which means this design is safe.

5. RESULTS AND DISCUSIONS

As the force is applied to instrumental panel the HIC value and plastic strain results are obtained from impact simulation along with x axis of direction.

❖ The results are obtained for two different shoot points are different, here we can see the at shoot point C1300. We observe (for the baseline model) which is exceeding the max. Standard HIC value i.e.1000. After modification in the design the HIC value is reduced from 1166 to 736 value. And this design is safe.

❖ After modification in the baseline design there is obtain a different values for plastic strain and HIC values for that shoot points, the HIC values are lesser than the 1000 and the plastic strain correspondingly increasing which is used to absorb more energy from headform and here the design is safe now.

3. CONCLUSIONS

This project mainly concentrate impact analysis scenario which is carried out on to the instrumental panel to reduce the HIC values according to ECE R-21 regulation standard. The following are some of the points which are concluded from below table where comparison of HIC values at different point with standard value.

❖ Practice & standardized CAE process increases productivity, simulations can be used to improve the instrumental panel design to pass the automotive regulation test.

❖ From the below table we observed that shoot point C1000 is found to be in safe condition.

❖ In the base line model at shoot point C1300 the HIC value is 1166 which is above the HIC value (ie.1000). After modification in the model, a variable thickness is provided in the region (i.e. base of 2.5mm from 3mm and upper layer of 2mm from 2.5mm) of more stiffness, after modifying design parameter for shoot point C1300 there is reduction in the HIC value and which is safe in condition, which is shown in the below table.

❖ After modification (the stiffness is reduce) there is increase in the plastic strain value so that the instrumental panel gets more deform after impacting the head, it can absorb more energy, leads to severity of the head.

❖ CAE can also be used to study different test technique with view to saving money in testing.

Shoot point	Baseline design		Modified design (variable thickness is provided in the region i.e. base of 2.5mm and upper layer of 2mm)	
	HIC value	Safe(<1000) /not safe (>1000)	HIC value	Safe(<1000) /Not safe (>1000)
C1000	627	Safe	-----	-----
C1300	1166	Not safe	736	Safe

REFERENCES

[1] S.Srinivasa Ragavan, Dr.A.Siva Kumar and et, al “Investigations into head impact for headliner design without counter measures” International Journal of Innovative Research in Science, Engineering and Technology, ISSN: 2319-8753, Vol. 2, Issue 9, September 2013.

[2] Mr. Uday Thakare and Prof. K. H. Munde “Cockpit Parts Projection Optimization in Head Impact area as per ECE-R21 & IS 15223 Regulation” International Engineering Research Journal Page No 1009-1013 issue 11-14 April 2005.

[3] Mike Keranen, Srikanth Krishnaraj, Kumar Kulkarni, and et, al “Automating Instrument Panel Head Impact Simulation” SAE international, SAE World Congress Detroit, Michigan issue 11-14 April 2005

[4] Erik G. Takhounts, Matthew J. Crai et, al “Development of Brain Injury Criteria (BrIC)” Stapp Car Crash Journal Vol. 57 (pp. 243-266) November 2013.

[5] Z. Ahmad and M.S. Othman “Energy absorption performance of a rain forest vehicle under frontal impact” Journal of Mechanical Engineering and Sciences (JMES) ISSN (Print): 2289-4659; e-ISSN: 2231-8380; Volume 6, pp. 807-817, June 2014

[6] Jinis Sliseris, Kirlis Rocens “Rational Structure of Panel with Curved Plywood Ribs” International Journal of Structural, Civil, Environmental, Construction and Architectural Engineering Vol: 5, No: 3,