LIGHT WEIGHTING THE TRIM METAL BRACKETS BY REPLACING THEM WITH PLASTIC BRACKETS

Satindar Singh Dua
Technical Lead, CAE
TATA Technologies ltd
Blue Ridge, SEZ Building
6th Floor, IT-3
Hinjewadi, Pune-411057

Surinder Matharu
Team Leader,
CAE Body Durability - Interiors
GDEC, Gaydon,
Banbury Road, Warwick,
CV35 ORR, UK

Pankaj Bhirud
Technical Specialist, CAE
TATA Technologies ltd
Blue Ridge, SEZ Building
6th Floor, IT-3
Hinjewadi, Pune-411057

Ajay Virmalwar
Sr. Project Manager, CAE
TATA Technologies ltd
Blue Ridge, SEZ Building
6th Floor, IT-3
Hinjewadi, Pune-411057

Abstract: The baseline load floor design was analyzed under standard loading conditions using Finite Element Analysis. It was observed that there was an opportunity for light weighting trims brackets supporting the load floor. The load floor support brackets were designed to be made of metal in the initial design phases. Design studies were carried out to design a plastic bracket which can replace this metal bracket. Optimum results were achieved by the new plastic bracket with the help of this study. Part mass reduction by 20% and process evolution were achieved by as a result of this study. Due to volume of vehicles produced, the annual savings will be of a competitive amount. The final design of plastic bracket is being used in the vehicle currently and has passed all the physical tests.

Keywords: Load floor, Lightweight, Support bracket, Polypropylene, Plastics

1. Introduction - Load floor:

Load floor is a part in the trunk area of a car which sits just above the spare wheel. Besides this it's parallel function is to carry the luggage and other items above it. Load floor is made of wood ply, hardboards etc. and is covered with thin carpet from all sides. In luxury cars, the load floor is tested for its functional strength under standard use and under abuse. Two load cases are often conducted to verify the integrity of the load floor, these are -

1. 1kN load applied normal to the load floor using a 75 mm diameter disc applicator
2. 36 kg mass drop from a certain height.

The performance of the load floor is judged on the basis of deflection, permanent set and plastic strains. For the first load case deflection should not be more than 10mm, permanent set should not be more than 5mm and plastic strain in the model should not be more than 1%. For the second load case plastic strain less than 1% is the main criteria as this is a dynamic load case. A load floor is supported by trunk scuff (Plastic material), side trims (Plastic-carpet composite) and support brackets (Metal material) or support rails (Metal material). Figure1 shows the load floor and its adjoining parts in the study under consideration. The load carrying members under loading of load floor are scuff plate, side trims and support brackets. Thus main focus in design and development of a load floor assembly is on its supporting parts.

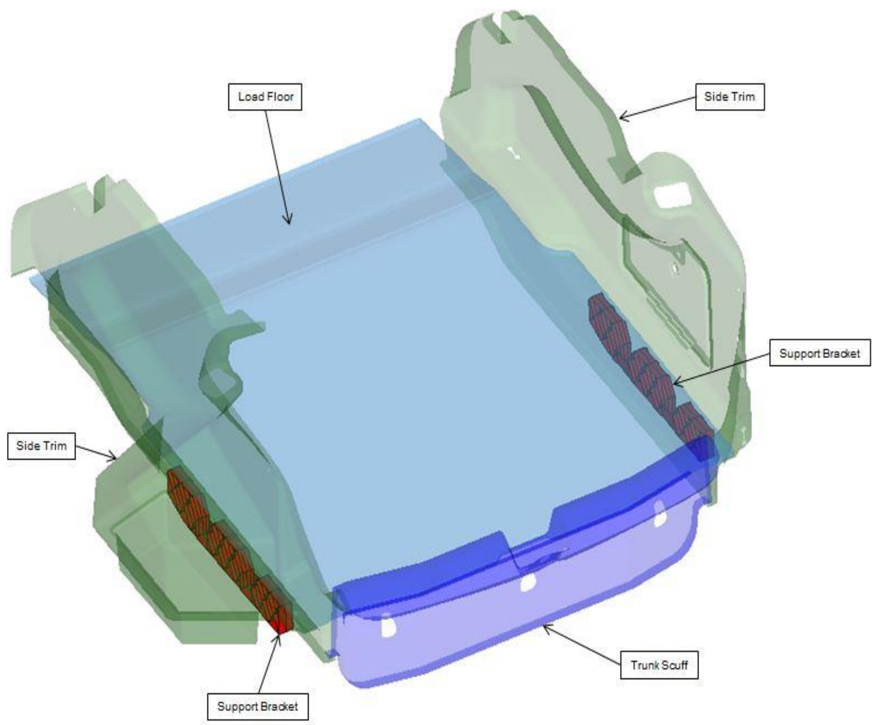


Figure 1.Typical Trunk Load floor assembly

2. Background of Study:

Load floor is one of the most important parts in trunk area of a car to be analysed using Finite Element Analysis (FEA) methods. The load floor goes through a variety of loading throughout its life. The owner of the car may use the trunk for storing his luggage, tools, cartons, etc. The FEA analyst needs to analyse the trunk to cater to all such requirements of the customers. There is one such situation where load floor is loaded by the knee of a person while accessing the load floor. In order to simulate this load case a 1kN load is applied on load floor at various locations using a 75mm diameter disc applicator to represent the knee. Under this load-case deflection of the load floor and strain on the adjacent parts are checked to ensure the strength of the load floor assembly.

In the vehicle program considered in this paper, the load floor was supported by Side trims and support brackets on two sides and trunk scuff plate on the rear side. The baseline load floor assembly was not showing any risk of damage in the knee load test. Rather it was indicating that the supporting parts for the load floor were overdesigned. Thus there was a scope to reduce some mass from the load floor supporting parts.

3. Design Development Methodology

3.1. Baseline Analysis Details:

3.1.1. Finite Element Model:

Load floor, its adjacent parts and Body in White components were meshed and connected together. Non-linear material properties were given to all the components. Body in white was cut down to remove unnecessary parts from the final model to save solution time. The model was constrained at the suspension location and at the body in white cut boundary. Load floor was meshed using 3-D Hexagonal elements and rest of the parts were meshed as 2-D shell elements. Metal support brackets were connected with Body in white using 1D rigid element. Figure 2 shows the FEA model for load floor analysis.

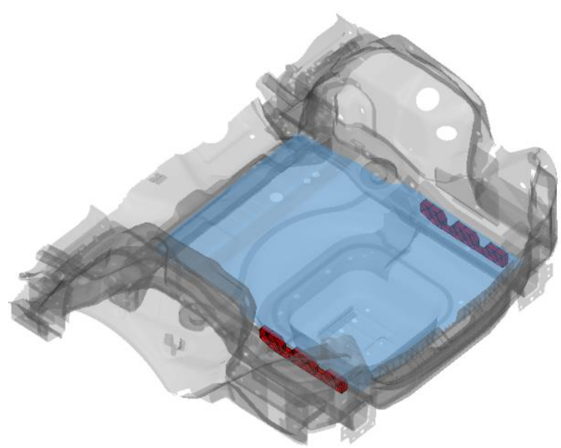


Figure 2. Load Floor Analysis FEA Model

3.1.2. Finite Element Analysis:

The Load floor was applied with 1kN load using a 75mm diameter standard disc applicator at 4 positions. The applicator was positioned above the load floor such that it just touches the floor top and is parallel to it. The load was applied through a local coordinate system so that the load applied is in normal direction to the floor. Refer Figure3 for the 4 positions load floor was analysed for knee load case. The deflection of the load floor node just below the centre of the applicator was requested in the solver deck. This analysis was carried in Abaqus solver which is very efficient in handling such type of complex non-linear contact models.

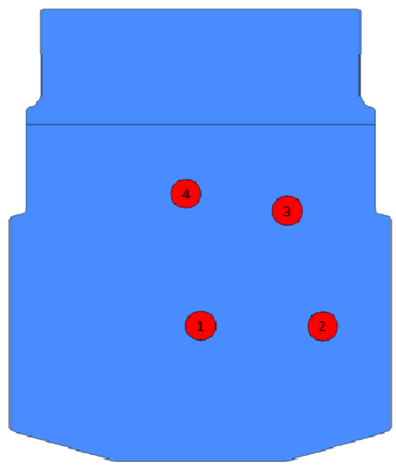


Figure 3. Load Positions

3.1.3. Results discussion:

The output file was reviewed to verify the behaviour of the load floor under the load. Force versus deflection curve was plotted for the load floor node just below the applicator centre. Out of the 4 load positions, position 1 was showing the deflection of 11mm, which is also acceptable. Permanent set of the load floor was less than 5mm for all positions. The two metal support brackets were showing negligible plastic strain and the body in white was showing zero plastic strain for all positions. Figure 4 shows how the applied load in this load case gets transferred to the body in white. Black colour represents body in white, green colour represents support brackets, blue colour represents load floor and yellow colour represents the load applicator. The load applied to the load floor transfers to support brackets and in turn passes on to body in white.

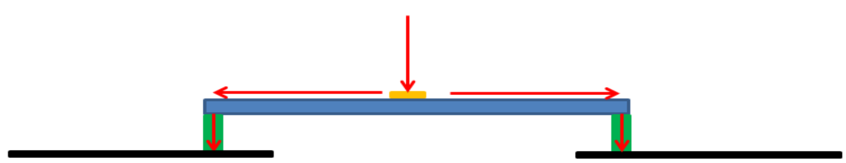


Figure 4. Load Transfer Path

3.2. Light weighting strategy:

As it was observed that the results were quite satisfactory in the baseline design analysis and only the load position 1 was showing a bit high deflection than the target. Hence this position was natural selection for the light weighting study to reduce some mass from the load floor adjoining parts. As the support brackets were showing negligible plastic strain in all the load positions, and were the main load carrying members between the load floor and the body in white, it was selected for light weighting study.

Figure 6-A, shows the baseline design of the metal bracket. At first, a study was carried out by changing the material property of the metal bracket to that of generic Polypropylene plastic material. The results were not complying with the deflection requirements but were indicating that plastics can be used in place of metal in support brackets. A new design of plastic support bracket was created from scratch. This design is shown in Figure 6-B. Metal brackets were replaced by this first concept design of plastic bracket in the same baseline load case model for position 1.The results for deflection and permanent set criteria were meeting the target. The maximum plastic strain on the plastic support brackets was 0.1% in this design. Thus this was an overdesign. The next step was to modify the design at locations of maximum strain. The first design was modified by removing material from high strain locations and the design as shown in Figure 6-C was created. The results of this design showed plastic strain of 0.2%. Third design was created as shown in Figure 6-D by modifying the second design at locations of high strains. This design when studied showed plastic strain of 0.4%. The deflection and permanent set were meeting the criteria of 10mm and 5mm respectively. Any further modifications in this design neither met deflection criteria nor the plastic strain criteria. This model with finalised bracket design was also tested under dynamic loading of 36 kg. The results were meeting the plastic strain criteria of 1%. Thus this design was taken as the final result of our study. The finalised design was also shared with mould flow team for their input and they confirmed the design from tooling feasibility and manufacturing point of view. Finally this design was forwarded to design leaders and their design team for creating 3-D geometry.

Figure 7 shows Force versus Displacement curve for load floor knee load case analysis under consideration with baseline metal bracket and finalised plastic bracket. Non linearity in the graph is due to the non-linear material characteristics, non-linear bending of the floor and continuously changing boundary conditions which is captured by Abaqus using complex contact algorithms. It can also be observed that stiffness characteristics of the plastic bracket and metal bracket are in line with the test requirements. Table 1 shows the results comparison between the initial metal brackets design and the finalised plastic bracket design for the knee load case for position 1. It shows that the final design of plastic bracket is 20% lighter than the metal bracket.

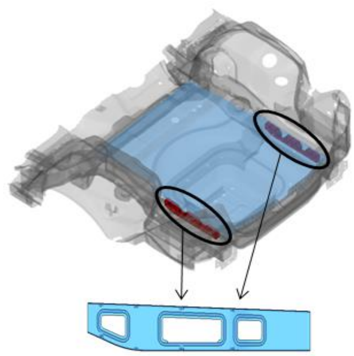
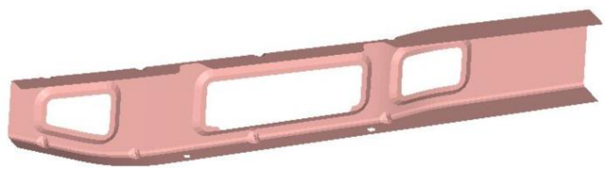
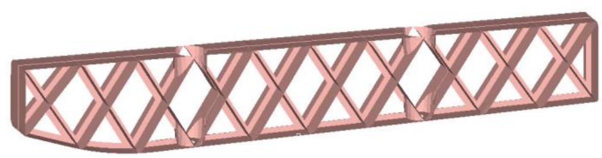


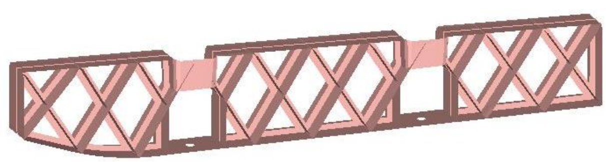
Figure 5.LoadFloor Model Baseline Metal Bracket



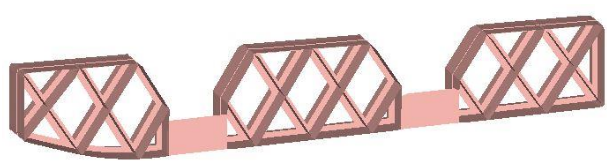
A



B



C



D

Figure 6.Development cycle of Metal bracket to Plastic bracket

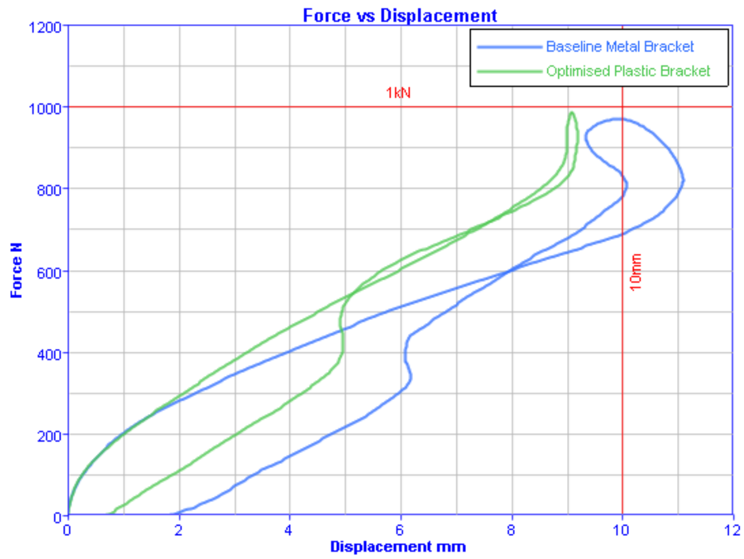


Figure 7.Force vs. Deflection Curve

Model	Bracket Material	Bracket Mass	Deflection	Plastic Strain
Baseline Metal Bracket	Metal	100%	Minor Risk	0.13%
Plastic Bracket First Concept	Polypropylene	90%	Pass	0.10%
Plastic Bracket First Iteration	Polypropylene	85%	Pass	0.20%
Plastic Bracket Final Design	Polypropylene	80%	Pass	0.40%

Table1.Plastic vs. Metal Bracket Comparison

4. Advantages of Plastics over metals:

Material science research and development is being carried out all over the globe for technology enhancements at a higher pace in recent times. Advantages of plastics over metals are mentioned below:

- Mass saving is the major advantage of using plastics in place of metals; in turn it also improves fuel economy of the vehicle.
- Recycled raw material can be used effectively in plastic parts.
- Multiple parts can be moulded in single mould using two/four cavity moulding method.
- Plastics are cheap and easy to transport and handle as many more plastic parts can be transported in single freight than metal parts.
- Less carbon-dioxide production during part production cycle (raw material-manufacturing-transport-assembly).
- Complex shapes can be made with similar cost as simple designs.
- Multi-functionality can be induced easily in plastics than metals.
- Plastics are corrosion free and chemical resistant, whereas metals need expensive paints for this feature. Plastics are electrically non-conductive and thermal insulators.
- Plastics can be made of various colours, whereas metals need external painting for this.
- Plastics do not need finishing operations like machining is needed for metals.
- Longer tool life for plastic products than metals products.
- Plastic part manufacturing is highly repeatable in processing with less scrap
- Plastics are far better than metals in NVH conditions.

5. Future scope:

In the baseline model, metal brackets were connected by screws with the body in white panels. The plastic bracket also uses the same screw connections. But further design enhancement can be attained by totally eliminating the screw connections between the brackets and the body in white panels. Instead, the bracket could have bosses which would push fit into the body panels, to provide a secure connection. This simplification would eliminate one or two assembly operations.

6. Conclusion:

The new plastic bracket design developed from this study performs well within the test requirements. In fact it serves better than the initial metal design. Stiffness achieved with plastic design is in line with metal design. Strains in plastic bracket are well within the acceptable limit. The major outcomes of this study are weight saving and the process evolved. With this material conversion, approximately 20% of the part mass was reduced in turn providing highly flexible manufacturing process. The use of recyclable plastic material ensures green design goals. Second major outcome of the study, the process developed, is being utilised in on-going and future programs. Abaqus solver played an outstanding role in this complete product development cycle. The results from Abaqus were quick and reliable. This resulted in feasible design solution which has proved itself in prototype as well as production models.

7. Acknowledgements:

The author would like to thank the following for their valuable technical inputs

Mr. Matthew Beadle, UK

Mr. Vinayak Gapchup, Tata Technologies Ltd., India

Mr. Nitin Hadpe, Tata Technologies Ltd, India

Mr. Shreyas Shingavi, Tata Technologies Ltd, India

The authors are extremely grateful to the Body Durability group at Tata Technologies Ltd.

8. References:

[1] Abaqus Theory manual

[2] R J Crawford, "Plastics Engineering", Butterworth Heinemann

[3] Robert A. Malloy, "Plastic Part Design for Injection Molding", Hanser Publishers

[4] S. S. Rao, "Optimization: Theory and applications", Halsted Press

[5] Brian Cantor, P. Grant, C. Johnston, "Automotive Engineering: Lightweight, Functional, and Novel Materials", Taylor & Francis Group, CRC Press