Size and Shape Optimization of a Polymeric Impact Energy Absorber by Simulation

IULIAN LUPEA[1]*, JOEL CORMIER[2]

Technical University of Cluj-Napoca, 15 Daicoviciu Str., 400020, Cluj-Napoca, Romania

The Oakwood Group, 1100 Oakwood Blvd, Dearborn, MI, USA

In this paper, aspects related to the energy absorption optimization of SafetyPlastic are presented. SafetyPlastic® is a patented impact energy absorbing technology currently used in the automotive industry as a countermeasure for occupant protection on head and side impact events. The optimization process is focused on properly changing the energy absorber wall thickness and shape, in order to promote an efficient buckling mode on the side walls. By using optimal thickness values along the height for different layers in which the particular absorber was divided, an outstanding improvement in Head Injury Criterion has been recorded. The results and the optimization procedure can be extended to various recesses shapes pertaining to the energy absorber.

Keywords: optimization, polymeric energy absorber, buckling, finite element analysis

In this paper, aspects related to the energy absorption optimization of SafetyPlastic® are presented. The mentioned product is a patented impact energy absorbing technology currently used in the automotive industry as a countermeasure for occupant protection in head and side impact events [2, 6, 7, 15].

SafetyPlastic® is characterized by a connected plurality of structural recesses that repeatedly give resistance, and then buckle when impacted. The recesses often take the form of truncated cones. SafetyPlastic® offers performance, cost, and mass competitive energy management solution, and has been embraced by the automotive industry in both head and side impact occupant protection applications, following the United States' Federal Motor Vehicle Safety Standard FMVSS 201u and FMVSS 214. In either case, the manufactured countermeasure is mounted between the interior trim and the body in white (BIW) structure where space is usually limited. Designs can and do vary greatly when customizing them to fit and perform within the geometrical environments into which they are packaged.

Generally, a mechanical energy absorber is best assessed by observing its force versus displacement response when impacted. With the area under the curve equating to the work done, a constant load maintained through the available space for intrusion is considered most efficient. However, the standards by which automotive occupant protection is measured are based on probability

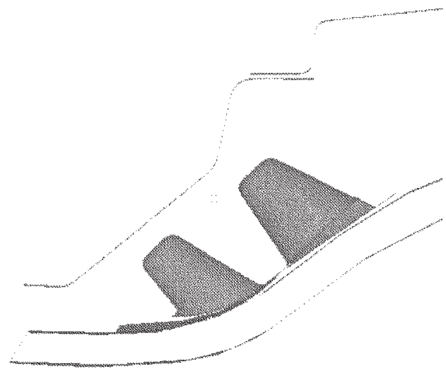

Fig. 1 The countermeasure lies on the headliner and buckle against the BIW

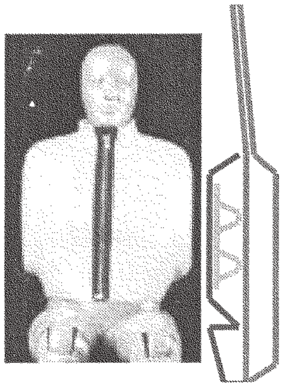

Fig. 2 Countermeasure for side impact

of injury. Thus, the measurements presented are mostly related to FMVSS 201u for upper interior head impact protection, which is evaluated via the Head Injury Criterion (HIC).

Optimization procedure and experiments

Plastic material modeling

The energy absorbers under observation are made from polymers. For various reasons, the preferred (but not exclusive) method of manufacture of SafetyPlastic® is thermoforming. Balancing, cost, performance, and formability a selection of polypropylene (PP) and acrylonitrile butadiene styrene (ABS) material grades are

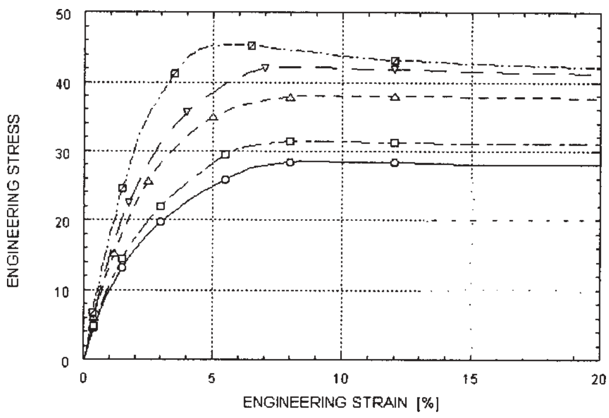

Fig. 3. Strain rate dependence

* i_lupea@yahoo.com: Tel: 264 582464

Table 1
Ls-DYNA MATERIAL MODELS FOR PLASTIC MATERIALS

Material title and number	Strain Rate Dependence			
	Yield stress	Young modulus	Failure Strain	Failure Stress
Strain rate dependent plasticity MAT19	•	•		•
Piecewise linear plasticity MAT24	•			
Plasticity with damage MAT81	•			
Plasticity polymer MAT89	•		•	

employed. Specifically, these include Basell Polyolefins Pro-fax SV152 Polypropylene Copolymer, BP Petrochemicals Homopolymer 6015 Sheet Extrusion Resin (6015), and General Electric Cycolac EX75 (EX75).

Finite element modeling of SafetyPlastic® needs to account for material behaviour at high and various rates of strain. Strain rates experienced during the crash, can be from quasi-static deformation up to 600 s^{-1} at velocities up to 6.7 m/s. Plastics experience stronger strain rate effects comparing with metals. In order to catch these effects, the material model used for the finite element modeling, comprises up to five true stress versus true plastic strains curves each one at a different strain rate ε. In figure 3, the elastic portion and the first plastic zone of the engineering or nominal stress (δ_N) versus engineering or nominal strain (ε_N) are shown, in case of a high impact polypropylene copolymer. The strain rates associated to the curves are 0.5/s, 5/s, 50/s, 200/s and 450/s. These nominal curves are further transformed [11] by considering the necking effect into true stress (δ_T) versus true strain (ε_T) curves (1).

$$\delta_T = \delta_N (1+\varepsilon_N)\ , \qquad \varepsilon_T = \ln(1+\varepsilon_N) \qquad (1)$$

In the next stage the elastic behavior is eliminated (2) from the (total) true strain, yielding true stress versus true plastic strain (ε_{PL}) curves. These curves were needed with the material model 24 of Ls-Dyna, which was employed on simulation.

$$\varepsilon_{PL} = \varepsilon_T - \delta_T / E \qquad (2)$$

The strain rate effects can be modeled by using Ls-Dyna elasto-plastic material models MAT19, MAT24, MAT81 and the model MAT89 later developed and dedicated to plastics. The mentioned models offer a range of rate dependence modeling capabilities being able to observe the effect of rate dependence of some material parameter like yield strength, Young's modulus, strain and stress failure [table 1].

Mesh size and other modeling aspects

Because the cones can buckle multiple times (two or three levels), attention to mesh density relative to the cone height (fig. 4) is required to capture the deformation specific to multiple folding adequately. Therefore, an improper (larger) mesh size has a bad influence on the parameters under observation [7] like HIC(d) and Peak acceleration (fig. 5). On the other side a small mesh size will decrease the integration time step performed by Ls-Dyna solver.

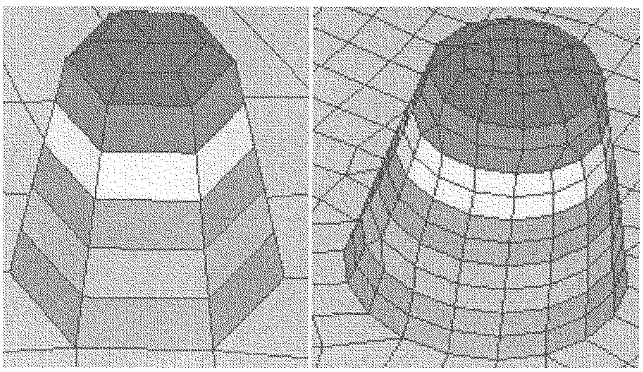

Fig. 4. Coarse versus fine mesh

The time step (Δt) size for shell elements (3) is dictated by a single element (the smallest) of the finite element mesh [4].

$$\Delta t = L / c \quad and \quad c = \sqrt{E \bullet (\rho - v^2)^{-1}} \qquad (3)$$

where c is the sound speed in the polymeric material under observation, L is the characteristic length of the element calculated in function of the shell element geometry, E is the Young modulus and ρ is the material density.

During the solution calculation, each new integration time step derivation is considering the minimum value over all finite elements (4).

$$\Delta t_{n+1} = TSSFAC \bullet \min(\Delta t_1, ..., \Delta t_N) \qquad (4)$$

where *TSSFAC* is a scale factor (by default=0.9) present in the Ls-Dyna *Control_timestep card, N is the number of finite elements in the model.

The thickness variation after thermoforming and residual stresses embedded in the structure after cooling are important inputs for the finite element model.

Because membrane straining during side wall folding (buckling) affects the thickness into shell elements, a special variable called Istupd of the *Control_shell card has been activated in order to account for this effect.

In defining the contact during the crash, the Shlthk variable of the *Control_contact card is activated in order to assure the thickness offset in an automatic surface to surface contact type.

Experimental part

The physical FMVSS 201u test is performed by launching a 4.5 kg (10 lb) modified anthropomorphic Hybrid III Free Motion Headform (FMH) at 4.7 m/s (15 mph) towards a target within a vehicle. The FMH is equipped with a tri-axial accelerometer at its centre of mass. For the calculation of the HIC, the relation (5) is used, where *a* is the resultant acceleration expressed as multiples of gravitational acceleration *g*, and t_1 and t_2 are any two successive instants during the contact between the head and the target.

$$HIC = \left[\frac{1}{t_2 - t_1} \int_{t_1}^{t_2} a dt \right]^{2.5} (t_2 - t_1) \qquad (5)$$

To account for the absence of the entire body representation in the test, relation (6) is used.

$$HIC(d) = HIC * 0.75446 + 166.4 \qquad (6)$$

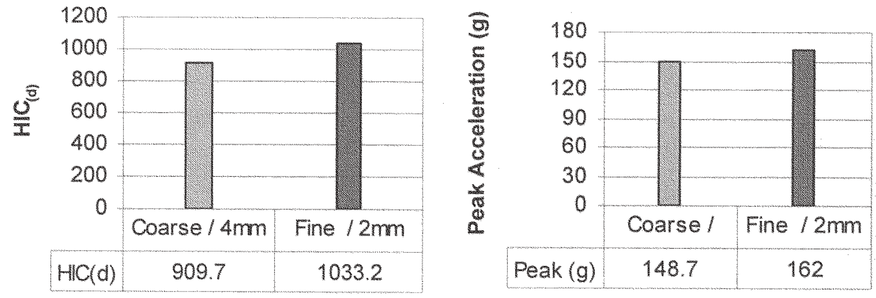

Fig. 5. Mesh size influence upon the HICd and the peak acceleration

HIC(d) of less than 1000 is deemed as compliant; though automotive manufacturers typically aim for HIC(d) values less than 800. Chest injury in FMVSS 214 and Lateral Impact New Car Assessment Program (LINCAP) tests for side impact protection are represented by the dummy Thoracic Trauma Index (TTI(d)). TTI is given by the relation TTI = $(a_{R(max)} + a_{LS(max)})/2$ where $a_{R(max)}$ is the peak rib acceleration and $a_{LS(max)}$ is the peak lower spine acceleration (T12 vertebra), and both are expressed in g's or multiples of gravitational acceleration.

Sample parts containing crashed truncated cones resulted after the crash test procedure are shown in figure 6. One can observe two and three sided buckling modes, and two or three levels of buckling.

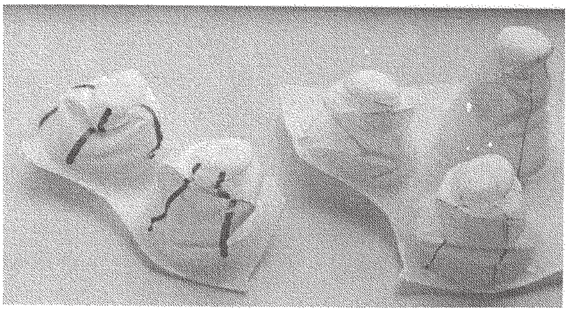

Fig. 6. Real buckling modes before optimization

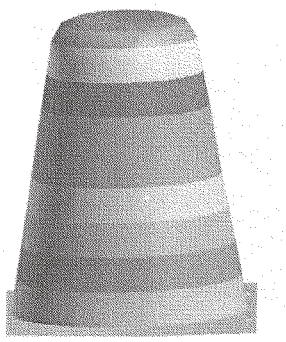

Fig. 7. Cone layers

Optimization study

Altair HyperStudy and HyperMesh software modules [12] were intensively used in conjunction with Ls-Dyna, a non-linear finite element solver [4, 13] to perform size and shape optimization in order to meet the objective. HyperStudy supports the optimization algorithms associated to the Sequential Response Surface Method and Method of Feasible Directions [12]. A user defined optimization engine can be used and linked to HyperStudy.

The target for the size optimization was to minimize the HIC(d) value for specific energy absorbing tasks, performed with truncated cone shape recesses.

A shape optimization has been performed with the purpose of promoting an annular buckling mode and a targeted strength, respectively a HIC(d) value, for the generic truncated cone.

The size optimization is more likely to be managed through injection molding while the shape optimization can be performed through thermoforming and injection molding, as well.

An optimization study was conducted to minimize the HIC(d) value for a single truncated cone subjected to an impact with a rigid moving flat plate. The rigid plate was given an initial downward velocity of 6.7m/s and a tuned mass of 0.2438kg, so it would have energy comparable to that a single cone may be expected to manage within a part. The cone mesh has been divided into nine layers parallel with the base, plus the roof or top surface area (fig. 7) to allow thickness to be assigned independently to the group of shell elements within each layer. Each layer or level was numbered such that "1" was at the cone bottom, "2" was next lowest, up to "10" at the top.

Constant wall thickness

For the first approach, the optimization of a constant thickness side wall was performed for the impact conditions described above (i.e. no independent layers were created this time) as a reference base for the next optimization with the layered cone. The initial thickness for the entire wall was 0.21 mm, and the variation of the thickness through the optimization was constrained from 0.2 mm to 0.8 mm.

The optimal or lowest HIC(d) of 1598 was obtained with a wall thickness of 0.637mm. The process was repeated with an initial thickness of 0.5 mm to yield an optimal HIC(d) of 1613 and the optimal thickness of 0.61 mm, after 13 iterations.

A layered side wall

Eight real and continuous design variables were created by using parameter command at the beginning of the Ls-Dyna deck (.dyn) file.

Their base value, min. value and max. value are 0.5 mm, 0.2 mm and 0.8 mm respectively. A study directory was created, where the input file has been placed. In order to perform the base run an input file and an execution script file (ls960-nsmp.exe) were prescribed.

The most important response which is the objective for the optimization procedure is the Head Injury Criterion or HIC(d). The expression max(hicd (v_2, 0.001*0.1019*v_1, 36, 2, 1)) was used to find the maximal HIC(d) value for a given crush simulation, where v_2 is the time vector of data to be analyzed, v_1 is the resulting acceleration vector of

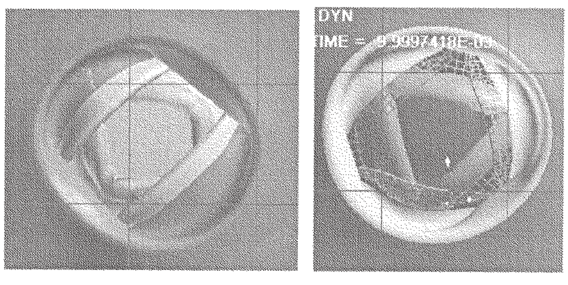

Fig.8. Two and three sided buckling modes by simulation

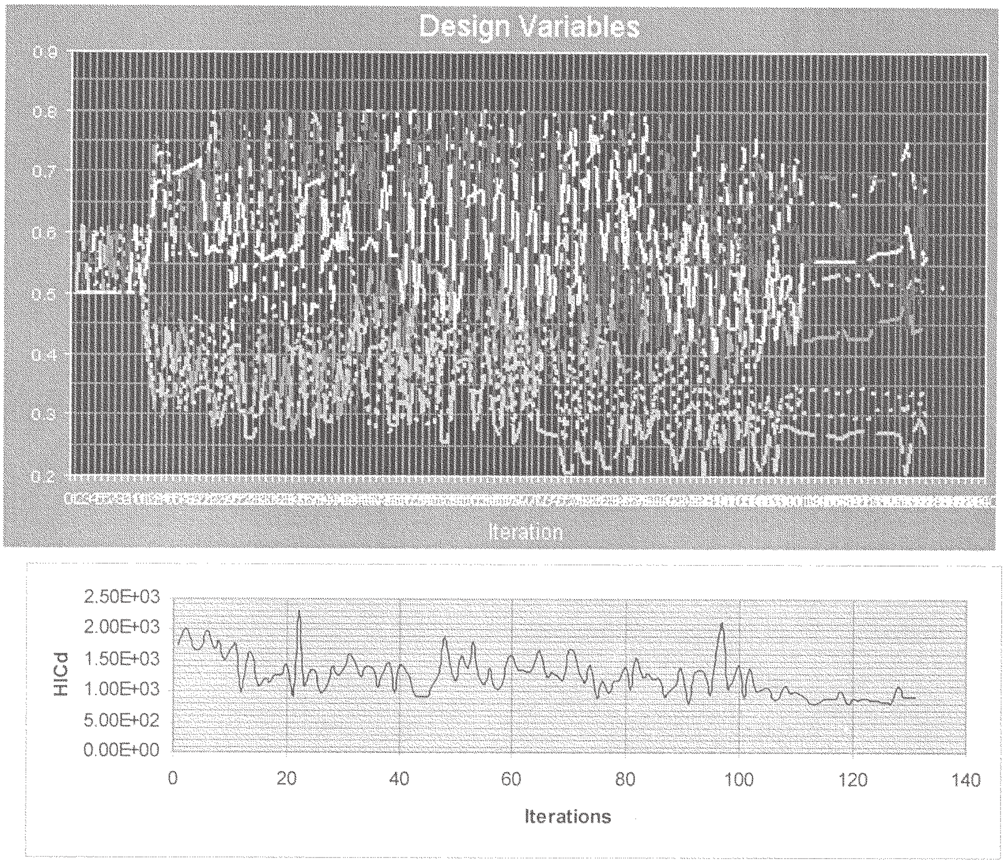

Fig. 9. Design variables and HICd variation during optimization

a representative point of the impactor, 36 is the time window (t2-t1) in milliseconds of the HIC definition (5). The representative node is located in the middle of the impacting plate. Hicd() with five parameters and max() are built-in functions. Because for a particular crash a set of HIC(d) values are computed, each one for a different pair of time instants t1 and t2 (5), max() function has to find the highest HIC(d) for that particular crash simulation.

Other responses of interest for the deforming structure, computed at each iteration are: the internal energy, the mass variation of the part under test and the ratio of the current volume of the cone and the internal energy.

For the current volume an expression that considers the side area of each ring (of the layered cone) and the design variables th_i (i=1 to 8) representing the current thickness for each layer, was used.

For each Ls-Dyna run (iteration) the output files are saved in a separate directory entitled run#, where # stands for the iteration number.

After 140 iterations of changing layer thicknesses, the HIC(d) was decreased in value by the optimization engine, from 1750 to 791 (fig. 9). The design variables at optimum thickness of the layers and associated to minimum HIC(d) value of 791, are as follows [mm]:

L1 = 2.702E-01,	L2 = 5.595E-01,
L3 = 6.504E-01,	L4 = 5.125E-01,
L5 = 4.236E-01,	L6 = 6.950E-01,
L7 = 3.031E-01,	L8 = 5.550E-01,
L9 = 5.554E-01,	L10 = 3.481E-01.

When comparing the first iteration and the optimal one, the buckling mode changes dramatically from a familiar one with two or three sides, shown in figure 6 and figure 8, to the annular one depicted in figure 10. An important aspect is the fact that the mass decreased from the initial value of 0.409e-3kg to 0.397e-3kg for the optimal shape.

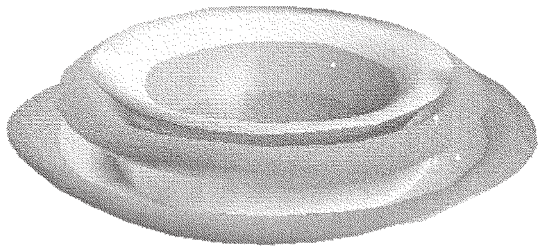

Fig. 10. Annular buckling

For verification purpose, a second optimization process was performed, the only difference being a 0.21 mm starting constant wall thickness of the cone as opposed to 0.5 mm previously used. We get the same optimal thickness distribution.

Side wall shape changes

The process of controlling the HIC(d) value for peculiar recesses of the impact energy absorber has been approached by using the shape changes in the side wall.

Using morphing capabilities (HyperMorph module) of HyperMesh, the mesh of the initial profile of the cone was modified. The cone diameter is slightly increased and decreased alternatively along the cone height paying attention to the drafting angles so that the thermoformed part can be extracted from the tool.

Shape optimization can be performed by using Ls-Dyna for crash simulation, HyperMorph for changing the wall profile (diameter of each concavity and convexity along the cone height) and HyperStudy to manage the optimization. Hence, annular buckling will be promoted targeting a HICd value for the structure.

The first stage of the study was to compare the behavior of two cones, the reference shape with straight side wall and a modified profile, depicted in figure 11. Both were subjected to crush or plastic deformation in axial direction

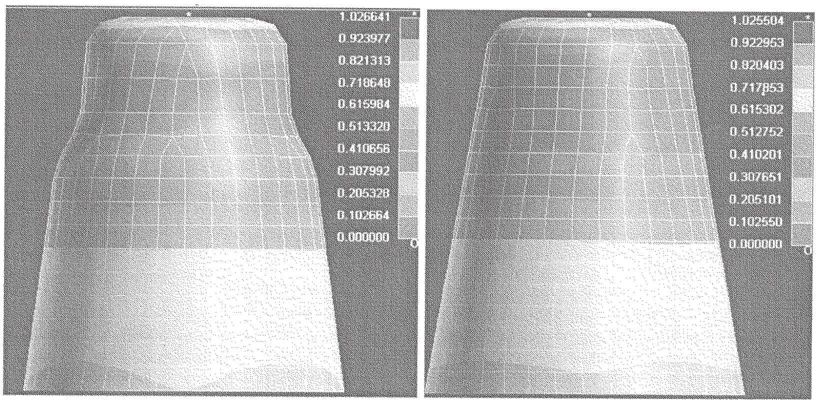

Fig. 11. Wall thickness profile after thermoforming with different tools

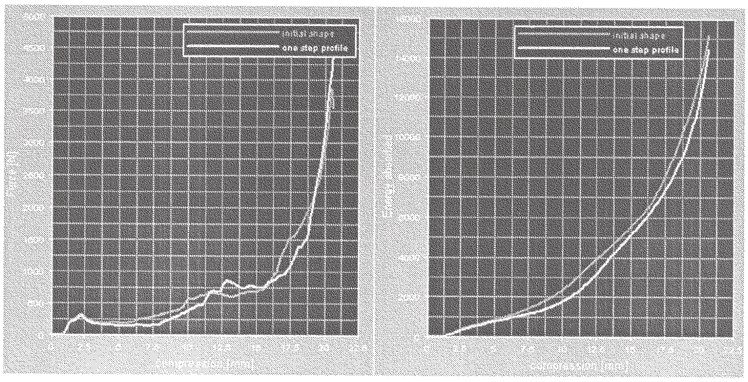

Fig. 12. Force and energy absorbed comparison for the two profiles

with an initial downward velocity of 6.7m/s and a tuned mass of 0.2438kg for the impactor.

Side wall thickness data for the shell elements was obtained by thermoforming simulation.

The design of the lower thermoforming tools was performed by using the CAD capabilities of HyperMesh. Then the tool profile is converted in stl format and imported as a lower tool in T-SIM [16]. A fine mesh of the tool's active surface is created in HyperMesh and exported in Ls-Dyna format (.dyn) which contains the card *Element_shell. The thermoforming simulation is performed with T-SIM, resulting a part with stretched net and individual thickness values for each node. The same initial geometry is coarse meshed proper for the crash simulation with Ls-Dyna. By using a special interface [7] already included in the T-SIM environment, the thickness variation is transferred from T-SIM to this coarse mesh in Ls-Dyna format assigning to each node of the mesh for the crash simulation a proper thickness value coming from the associated nodes of the thermoforming simulation mesh. In the Dyna deck file the *Element_shell card is replaced by the *Element_shell_thickness card.

The initial lower tool and the modified one are used to simulate the forming of two different parts by using the same gage of the plastic sheet and all other simulation parameters the same. The thickness variation in both thermoformed parts are shown in figure 11. The thickness variation for each part is transferred to Ls-Dyna associated models and performed the crash simulations. The resulted Force versus compression curves and Energy absorbed versus compression are depicted in figure 12.

Conclusions

A generic truncated cone has been selected for optimization by simulation. A flat plate of 0.2438 kg translating along the cone axis and having a downward velocity of 6.7m/s was used to hit the structure.

At the beginning the HIC(d) was minimized finding an optimal side wall constant thickness of 0.61 mm and a minimal HIC(d) of 1613.

In the next step the side wall was divided into concentric rings or layers with individual thickness values working as parameters in the optimization process. The splitting of the side wall in rings has been inspired by observing the side wall thickness variation of the real thermoformed truncated cones. The new model was subjected to size optimization.

The recorded drop in HIC(d) value to 791 was outstanding and has been attributed to the change in buckling mode from two or three sided mode per layer to an annular buckling mode per layer, as a consequence of the change in thickness in different rings along the cone height. In parallel, a slight diminish of the cone mass has been recorded. In all situations the cones are crushing all the way in between the flat plates.

A shape optimization has been started aiming the design of recesses at a target HIC(d) by properly changing the side wall profile and to promote annular buckling.

We conclude by observing that the use of an optimal thickness variation in annular layers can promote annular buckling with an increase of impact energy absorption capability of truncated cones, without any increase in mass.

By combining the mentioned optimizations and considering a proper forming process, injection molding or thermoforming, a targeted HIC(d) can be pursued. The method can be extended to a plurality of cones involved at a time in a specific target location during head or side impact test procedure.

Bibliography

1. CLARK, C., JOHNSON, P. , FROST, C., High Strain Rate Characterization of Thermoplastics Used For Energy Management Applications, SAE Technical Paper 940882

2. CORMIER, J., BROOKS, R., Design of Thermoformed Plastic Mechanical Energy Absorbers for FMVSS 201 Head Injury Applications, 2001 Detroit Auto Interiors Show Conference Proceedings

3. GORAL, T., Material Modeling For Engineering Thermoplastics In FEA Under High-Strain Loading, SAE Technical Paper 2000-01-1167
4. HALLQUIST, J.O., LS-DYNA Theoretical Manual, (Livermore Software Technology Corporation, Livermore), CA, 1997
5. HILL, S.I., SJOBLOM, P., Practical Considerations in Determining High Strain Rate Material Properties, SAE Technical Paper 981136
6. LUPEA, I., CORMIER, J., SHAH,S., FEM for Impact Energy Absorption with SafetyPlastic, 8-th International LS-DYNA Users Conference, Proceedings pg. 5.57-5.68, May 2-4, 2004, Hyatt Regency Dearborn, Michigan, USA.
7. LUPEA, I., CORMIER, J., Mat. Plast., **43**, nr.4, p. 340, 2006
8. MARTIN, P., CRANDALL, J.R., Constitutive Modeling of Polymers Subjected to High Strain Rates, SAE Technical Paper 2001-01-0472
9. MURTHY, T.K.S., BREBBIA, C.A., Structural Design and Crashworthiness of Automobiles, Springer Verlag, Berlin, 1987
10. NAUGHTON, P., Extension Of Material Models And Finite Element Technique To Improve The Simulation Of High Speed Impact Of Thermoplastic Material, SAE 1999-01-0300
11. WOODS, J.T., TRANTINA, G.G., Material Characterization for Predicting Impact Performance of Plastic Parts, SAE Technical paper 1999-01-3178
12.*** HyperStudy® and HyperMesh® are registered trademarks of Altair Engineering, Inc.
13.*** LS-DYNA version 970, Keyword User's Manual, Part 1 and 2
14.*** SafetyPlastic® is a registered trademark of The Oakwood Group, Dearborn, MI, USA.
15.*** The Oakwood Group patents: US Patent No. 6682128, Pat. No. 6679967, Pat. No.
6247745, Pat. No. 6199942, Pat. No. 6017084
16.*** T-SIM® is a registered trademark of Accuform, Ltd.

Manuscript received: 19.09.2007

We conclude by observing that the use of an optimal thickness variation in annular layers can promote annular buckling with an increase of impact energy absorption capability of truncated cones, without any increase in mass.

By combining the mentioned optimizations and considering a proper forming process, injection molding or thermoforming, a targeted HIC(d) can be pursued. The method can be extended to a plurality of cones involved at a time in a specific target location during head or side impact test procedure.

References

1. CLARK, C., JOHNSON, P. , FROST, C., High Strain Rate Characterization of Thermoplastics Used For Energy Management Applications, SAE Technical Paper 940882
2. CORMIER, J., BROOKS, R., Design of Thermoformed Plastic Mechanical Energy Absorbers for FMVSS 201 Head Injury Applications, 2001 Detroit Auto Interiors Show Conference Proceedings
3. GORAL, T., Material Modeling For Engineering Thermoplastics In FEA Under High-Strain Loading, SAE Technical Paper 2000-01-1167
4. HALLQUIST, J.O., LS-DYNA Theoretical Manual, (Livermore Software Technology Corporation, Livermore), CA, 1997
5. HILL, S.I., SJOBLOM, P., Practical Considerations in Determining High Strain Rate Material Properties, SAE Technical Paper 981136
6. LUPEA, I., CORMIER, J., SHAH,S., FEM for Impact Energy Absorption with SafetyPlastic, 8-th International LS-DYNA Users Conference, Proceedings pg. 5.57-5.68, May 2-4, 2004, Hyatt Regency Dearborn, Michigan, USA.
7. LUPEA, I., CORMIER, J., Finite Element Model Validation of a Polymeric Impact Energy Absorber, Revista Materiale Plastice, pg. 340, **43**, Nr.4, 2006
8. Martin, P. and Crandall, J.R., Constitutive Modeling of Polymers Subjected to High Strain Rates, SAE Technical Paper 2001-01-0472
9. Murthy, T.K.S. and Brebbia, C.A., Structural Design and Crashworthiness of Automobiles, Springer Verlag, Berlin, 1987.
10. Naughton, P., Extension Of Material Models And Finite Element Technique To Improve The Simulation Of High Speed Impact Of Thermoplastic Material, SAE 1999-01-0300
11. Woods, J.T. and Trantina, G.G., Material Characterization for Predicting Impact Performance of Plastic Parts, SAE Technical paper 1999-01-3178
12.*** HyperStudy® and HyperMesh® are registered trademarks of Altair Engineering, Inc.
13.*** LS-DYNA version 970, Keyword User's Manual, Part 1 and 2
14.*** SafetyPlastic® is a registered trademark of The Oakwood Group, Dearborn, MI, USA.
15.*** The Oakwood Group patents: US Patent No. 6682128, Pat. No. 6679967, Pat. No.
6247745, Pat. No. 6199942, Pat. No. 6017084
16.*** T-SIM® is a registered trademark of Accuform, Ltd.

Manuscript received: 19.09.2007