Development and Validation of a Headform Impactor Finite Element Model with Application to Vehicle Hood Design for Pedestrian Protection

Costin Untaroiu, Jaeho Shin, Jeff Crandall
Center for Applied Biomechanics
University of Virginia

Scott Crino
Department of Systems & Information Engineering
University of Virginia

Abstract

Head injuries are the most common cause of pedestrian deaths in car-to-pedestrian collisions. To reduce the severity of such injuries, international safety committees have proposed subsystem tests in which headform impactors are impacted upon the car hood. In the first part of the paper, the development and validation of an adult headform impactor finite element (FE) model is presented. The skin material model was assumed as viscoelastic and its parameters were identified by FE optimization to match the quasi-static and dynamic test data reported in literature. Overall, it was shown that the geometrical and inertial characteristics of the headfom FE model developed in this study satisfy the regulations of international safety agencies. The second part of the paper presents results of a hood optimization using simulations of the headform-hood impact test. A generic hood design was assumed consisting of two plates connected by buckling structures. The reductions of head injury risk under impact and the under-hood clearance space were included in an optimization problem which considered the geometry of connecting spools and the panel thicknesses as design variables. The automatic design process was shown to converge to an optimum design after several iterations. The methodology and recommendations for future work presented in this paper may assist in the hood design of new car models to reduce pedestrian head injuries and meet new safety requirements.

Introduction

Head trauma is responsible for the most serious of pedestrian injuries and mortality in car-pedestrian accidents. Head injuries represent 80 % of severe pedestrian injuries [1] and the causes of 62% of pedestrian fatalities [2]. Therefore, countermeasures against head injuries are of the highest priority in pedestrian safety strategy.

Several test procedures [2] to asses the level of pedestrian protection for new car fronts have been proposed by international safety committees such as The International Organization for Standardization (ISO), International Harmonized Research Activities (IHRA), and the European Enhanced Vehicle Safety Committee (EEVC). These proposed regulations involve the impact of four impactors representing different parts of the human body into the front of a resting vehicle. The test which quantifies the risk of pedestrian's head injuries involves launching free flying head-forms at 32-40 km/h against the vehicle hood. The acceleration time history is recorded during the test by a sensor mounted ideally in the impactor center of gravity, and is used to asses the aggressiveness of the car front against pedestrians based on biomechanical criteria. Using physical tests to optimize the safety level of a novel hood design is costly in terms of both time and money. A fast and cheap alternative could be the use of computer simulations using the Finite Element (FE) analysis, as long as a validated FE model of headform impactor is used. Several papers [3-4] have presented the development and validation process of different FE headform models based on the dimensions of physical impactor components and the response of the impactor to several certification tests. However, these models are not public and were not available for this study; therefore, the current FE model was developed and validated based on published data.

The hood surface is the most likely region where the pedestrian's head comes into contact with the vehicle [1] during accidents; therefore, a safer hood design is the most important measure in pedestrian protection. Even though the hood is usually a relatively compliant structure and does not constitute a major risk for head injuries, the stiff underlying structures such as the engine components, can cause serious head injury when they come into contact with the deformed hood. A good design solution is to provide sufficient clearance between the hood and the underlying structures for controlled deceleration of a pedestrian's head and a headform impactor. However, considerations of aerodynamic design, styling, and cost can make it extremely difficult to adjust a vehicle's front end geometry to provide more under-hood space. A solution to this difficult problem is to develop a hood design that efficiently decelerates the impactor with minimum deformation space. Okamoto et al [5], through studies of headform impact tests and simulations, determined the energy absorbing performance of two candidate solutions for hood structure: a brittle surface material with an energy absorbing material, and an elastic surface material with a buckling structure. However, the test pieces had relatively large thickness (100mm) and were affixed to a hard floor, making the application of their results to a real hood design questionable. Recently, some other vehicle bonnet designs have been initiated, such as the multi cone design [2], but based on our knowledge some of the results concerning hood design optimization have not been published.

This study proposes the utilization of recent developments in FE software to investigate an FE optimization approach for hood design. A generic hood design including buckling aluminum spools connected to two cover plates is proposed for study. The geometric characteristics of a cylindrical spool shape, together with the thickness of upper and lower hood plates are considered as design variables. The objective function is the minimization of the total hood stroke (sum of hood thickness and bottom plate deflection) with constraints in biomechanical requirements and limited hood thickness. The optimization process was defined automatically through the connection of three software packages: TrueGrid [6] (for meshing), LS-DYNA [7] (for simulating the headform impact test), and LS-Opt [8] (for FE optimization). Even though it is ideal to use a real hood model together with its surrounding structures, it is believed that the design approach proposed in this study will help in the development of new design concepts for vehicle hood which will improve pedestrian protection.

Methods

Development and Validation of a Headform Impactor FE Model

A typical headform impactor has three main parts [9]: a steel base mounted with an accelerometer, a spherical aluminum core, and a polyvinyl chloride (PVC) skin. The FE headform model developed in this study also has these three parts, modeled with approximately 14,800 solid elements, mostly hexahedral, as illustrated in Figure 1. The geometric characteristics of the model use the published description of the physical JAMA-JARI adult headform impactor [4] as reference. Because the steel base and aluminum core component are much stiffer than the PVC skin, deformations of these components are expected to be small during the impact tests. Therefore, a linear elastic model and a rigid model were used as illustrated in Table 1 for the core and base, respectively.

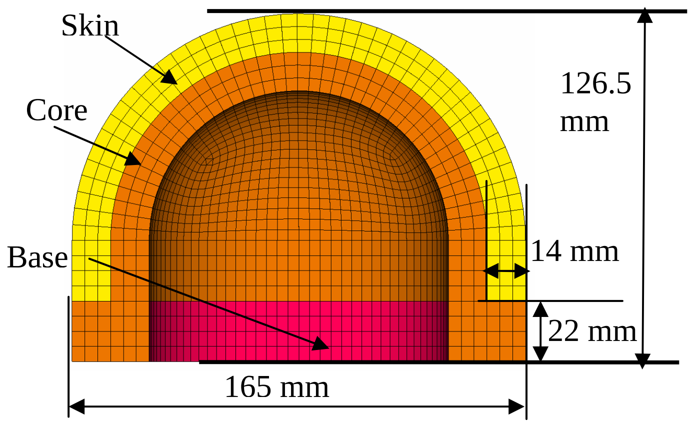


Figure 1. The FE model of adult headform impactor (Transversal cross-section)

Comp.	**Material (LS-DYNA material model)**
Base	Steel (MAT_RIGID)
Core	Aluminum (MAT_ELASTIC)
Skin	PVC (MAT_GENERAL_ VISCOELASTIC)

Table 1. Material models for FE headform components.

The first natural frequency of the headform impactor must be high enough not to influence the acceleration measurement [9]. Therefore, the first frequency of the core structure connected with the base should be higher than 5,000 Hz according to IHRA requirements. In the present study the first two natural frequencies of the core-base system were obtained using LS-DYNA and compared with the frequencies of the JARI-JAMA physical model which were measured by impact excitation tests using an impact hammer [6]. During the impact test the PVC skin is in contact with the hood and experiences high deformation due its soft structure; therefore its viscoelastic properties are critical for the fidelity of the headform impactor. Unfortunately, the material properties of skin were not found in literature, and attempts to obtain these viscoelastic properties from the skin manufacturer were unsuccessful. Therefore, a LS-DYNA viscoelastic material (MAT_76) was assigned for the skin and its coefficients were obtained using LS-Opt by material identification procedures. First, the bulk modulus (K), "static" shear modulus (G_0), and first two terms of the "dynamic" shear modulus expressed in a Prony series in Eq. 1 were determined by simulating the quasi-static test performed on JARI-JAMA physical model by Matsui et al [10].

$$G(t) = G_0 + \sum_{i=1}^{4} G_i e^{-\beta_i t} \tag{1}$$

In this compression test, the headform impactor was laid on a rigid table and half of the skin thickness (7mm) was compressed vertically by a load cell moved downward and upward at 0.1 mm/sec. The same test was simulated in LS-DYNA, as shown in Figure 2 a) using a moving rigid plate. The minimization of the root-mean square (RMS) error between simulation results and test data [10] of the rigid plate-skin contact force was defined as the optimization objective, and the material coefficients were taken as design variables. Besides the quasi-static test a dynamic drop test of the headform impactor from a height of 376 mm was simulated as shown in Figure 2 b).

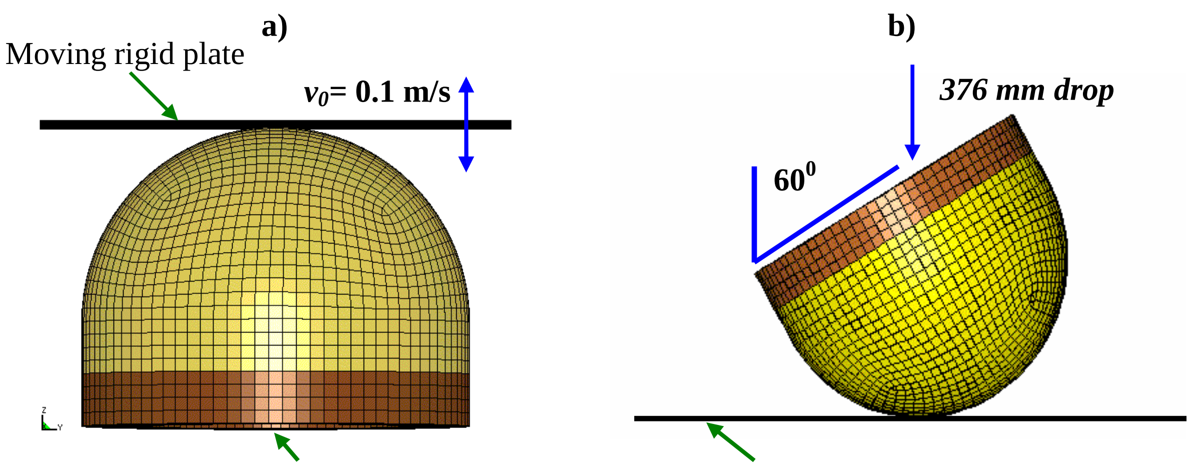


Figure 2. Test set-ups of skin validation tests; a) quasi-static test; b) dynamic test

The drop angle was 60^0 as in [10], which is close to the mean drop angle proposed by ISO (54^0) and category 1 of the IHRA (65^0). Two additional terms were added to the Prony expression of shear modulus (Eq.1) obtained by simulating the quasi-static test and their coefficients were obtained by LS-Opt. The objective function was defined as the difference between the maximum acceleration of headform impactor and 250 G – the mean value of the biofidelic corridor proposed by ISO and IHRA (225-275 G). The simulation results of impact tests (presented in the Results section) suggested that the headform FE model was in accordance with ISO/IHRA requirements. Therefore, the model was used in optimization of a generic vehicle hood using the headform-hood impact test presented in the next subsection.

FE Hood Model and Simulation of the Headform-Hood Impact Test

According to pedestrian safety regulations, the vehicle hood must not exceed the head injury criteria (HIC) limit at every point of its impact surface. This requirement has significantly changed the hood design from the traditional design which consisted of a rib structure supporting the outer panel, to a uniform structure exemplified by the multi-cone hood design [2]. Therefore, in this study, a uniform hood structure consisting of two rectangular plates (850 mm x 1450 mm) connected by a set of equidistant spools (7 rows X 13 columns) is proposed for optimization, as illustrated in Figure 3. Both outer panels and spools are made of aluminum (Al 6063), but with different yield stresses: a stiffer material for the panels (225 MPa) to meet standard load requirements [2], and a softer material for spools (100 MPa) to absorb energy by buckling deformation during headform impact tests.

The mesh of the hood model investigated in this study was performed in TrueGrid using a parametrical approach. The parameters defined in a TrueGrid batch file were: design variables, which will be explained in the next subsection, and mesh shape parameters. Mesh parameters were used to maintain a constant length of quadrilateral shell elements when the geometry of spools was modified. All hood components were modeled using Belytschko-Tsay shell elements with constant thickness and three integration points.

A fine mesh is required for better FE predictions of the spool buckling. On the other hand, small elements decrease the time step of simulation, increase the number of elements, and results in an increased computational cost for completing impact simulations and for reaching convergence to a feasible, near optimal solution. The sensitivity of the spool mesh was studied in several buckling simulations and a element size of 1 mm was found as a satisfactory model of spool buckling. In order to reduce the number of elements of the entire hood, 1 mm mesh density was considered only in the region closest to the impact point, while in the rest of hood where the amount of buckling is less, a 1.5 mm resolution was used. Overall, a typical hood model consisted of approximately 100,000 elements and the time step was about 0.1 μsec. During the impact simulation, the time step decreased significantly when the spool buckling occurred, increasing the total simulation time. Therefore, in order to maintain the total simulation time at a reasonable level the minimum time step was limited at 0.065 μsec using the mass scaling feature of LS-DYNA. However, the inertial properties of the hood were not affected by added mass which was about 0.01% of the total mass of the hood.

The hood angle was set to 10^0 corresponding to a typical sedan model and a 40 km/h headform impact test was simulated in accordance with EEVC Phase 2 [2]. The impact angle was set to 65^0 as required by ISO and EEVC protocols for a sedan type vehicle, as shown in Figure 3b). Even though the proposed hood design should be evaluated at every surface point in the same headform impact condition, only the center of hood was tested in this study to reduce total computational time. However, it must be mentioned that the chosen point of impact results in one of the highest strokes, parameter which was minimized in FE optimization.

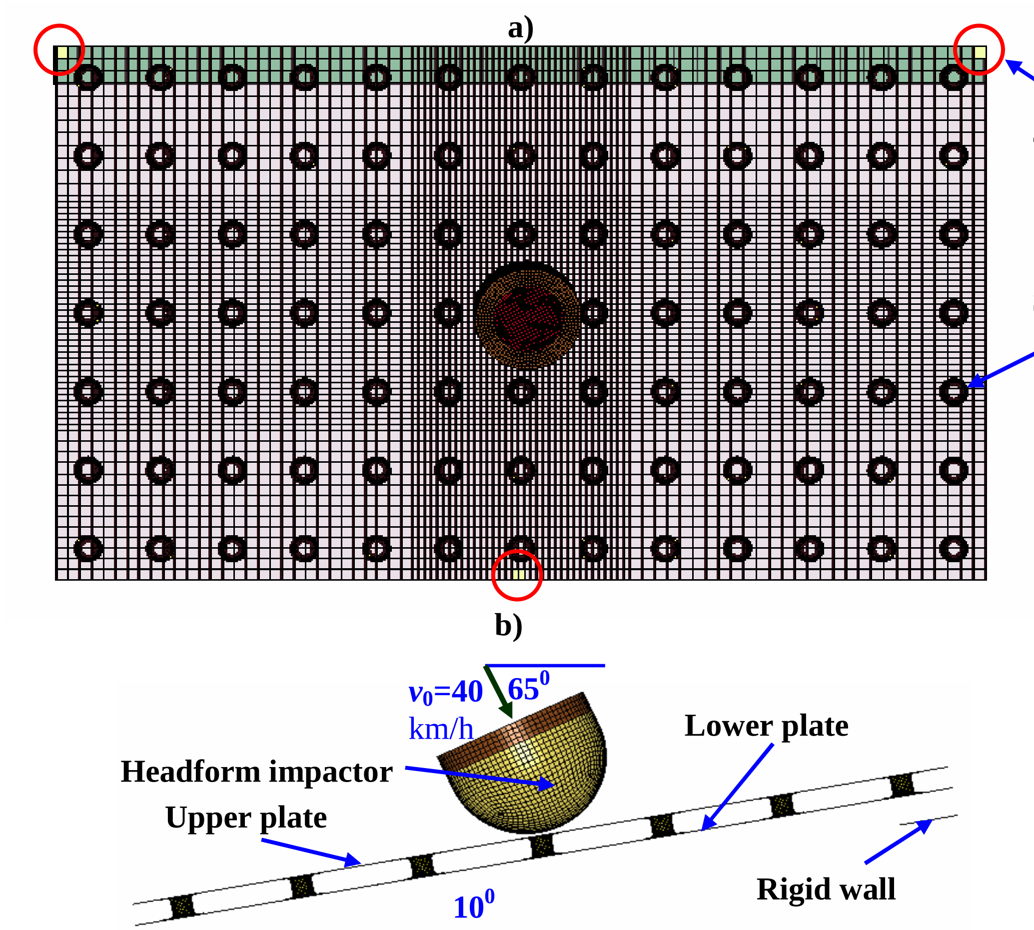


Figure 3. The FE model of a generic hood design used in FE Optimization

The design of hood edges, together with the mounting system of the hood to the vehicle structure (e.g. hood hinges, latches, and bump stops), also significantly affects the pedestrian safety, especially when the impact is closest to the hood boundaries. For simplification of the optimization problem, the mounting system of the hood was approximated with three revolute joints corresponding to two hood hinges and a latch system. The rigid region under cowl panel was also simplified with a rigid wall at 30 mm distance to the hood.

Parametric Optimization of a Generic Hood Design

The most relevant characteristics of a safer hood design are: (1) sufficient stiffness to allow a low under hood clearance; (2) an energy-absorbing structure which efficiently decelerates the headform impactor. In this study, these contradictory requirements were cast in a constrained optimization problem with the following **objective function**:

$$\textit{Minimize}\ \mathbf{TS{:}=S+H} \tag{2}$$

where the "stroke" **S** is the highest normal deformation of the lower panel, **H** is the hood thickness, and their sum **TS** is the "total stroke"
subject to the following **constraint**:

$$\mathbf{HIC<1{,}000} \tag{3}$$

HIC is the Head Injury Criteria, defined as: $HIC := \max\left[\frac{1}{t_2 - t_1}\int_{t_1}^{t_2} a(t)dt\right]^{2.5}(t_2 - t_1)$ where $a(t)$ is the time history of headform acceleration and the maximal time frame is limited to 15 milliseconds. An additional **constraint** was added to avoid some unrealistic hood designs in terms of the hood thickness:

$$\mathbf{H<40\ mm} \tag{4}$$

A cylindrical spool shape was used for the optimization analysis, as illustrated in Figure 4. Their geometrical characteristics were chosen as **design variables** with the range shown in Table 2.

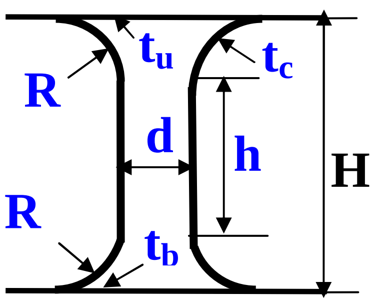


Figure 4. Geometrical parameters used in FE spool design

Table 2. Design space of spool geometric parameters

Spool Parameters		R mm	d mm	h mm	t_c mm	t_u mm	t_b mm
Variable Range	min	4	10	5	0.08	0.6	0.6
	max	10	40	30	0.2	2	2

The FE optimization was performed using the Successive Response Surface Methodology (SRSM) implemented in LS-Opt 2.2 under a Linux operating system. To reduce the number of simulations required per iteration a linear polynomial model was used. The D-Optimal design proved to give maximal confidence at low computational cost [8], and a full factorial basis with 3 points per variable was used for all FE evaluations. All design processes were integrated automatically by connecting the following software: TrueGrid (for generating the hood mesh and preparing the input files for impact test), LS-DYNA (for simulating the impact tests), and LS-Opt (for extracting and analyzing responses, and proposing new designs), as illustrated in Figure 5. This design approach may significantly reduce the time and cost required for finding the best design solution for new vehicle hoods in future studies.

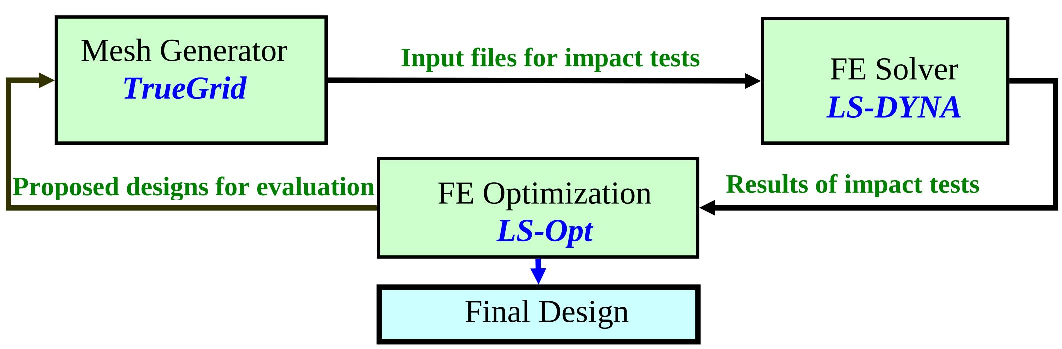


Figure 5. Schematic automatic geometry optimization of hood design

Results and Discussion

Validation of the Headform FE Model

The results of the headform FE model with optimized skin material show agreement between quasi-static compression simulation and test data in terms of the force-deflection curve, as illustrate in Figure 6 a). The value of the bulk modulus and the significant terms of dynamic shear modulus, obtained based on an ANOVA after the first iteration, are given in Table 3. Two more Prony terms of the dynamic shear modulus showed significance when matching the maximum acceleration target (250 G) in the drop certification test. The values of these terms which were added to the terms identified in the quasi-static test are given in the Table 3. The simulation results obtained in both quasi-static and dynamic tests suggest that the skin material model is relatively accurate in simulations of impact with a vehicle hood. However, in order to better characterize viscoelastic properties of skin at different strain rates, it is advisable that the Prony series of shear modulus be expanded to include more terms. This task could be accomplished by running more identification tests, with the same methodology proposed in this paper, when more test data from compression tests at different strain rates become available.

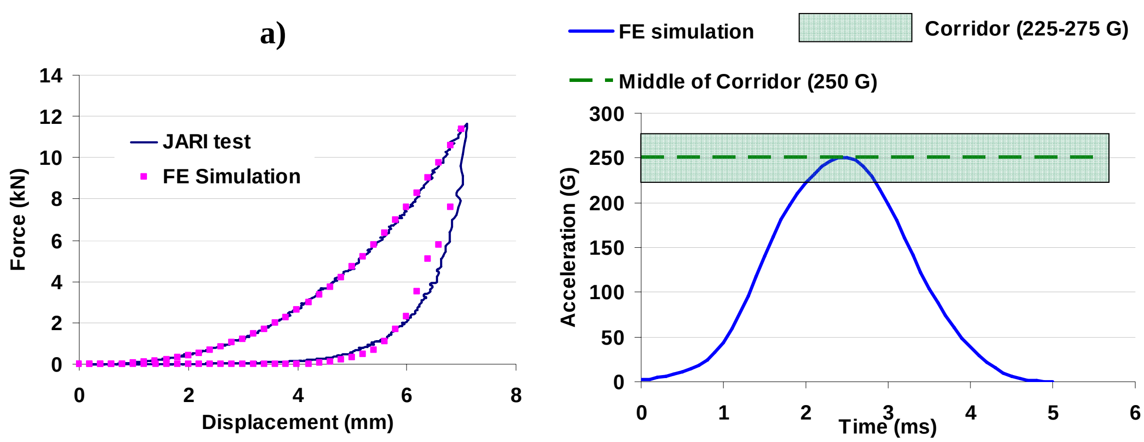


Figure 6. Results of skin validation tests; a) quasi-static test; b) dynamic test

Table 3. Identified parameters of skin viscoelastic material model (MAT_76)

K	G_0	G_1	G_2	β_1	β_2	G_3	G_4	β_3	β_4
MPa	MPa	MPa	MPa	s^{-1}	s^{-1}	MPa	MPa	s^{-1}	s^{-1}
779.6	0.483	1.345	0.01	0.1	0.01	0.389	0.005	1000	100

Identified in quasi-static test

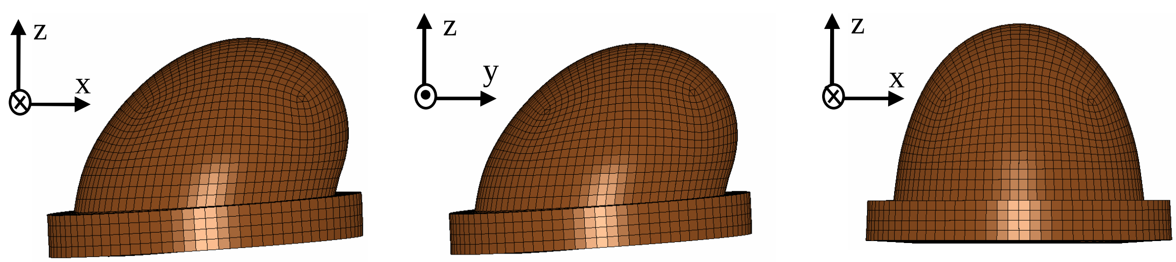
Identified in dynamic test

Various parameters (mass, outer diameter, moment of inertia, location of accelerometers, natural frequencies) of the head-form FE model, developed in this study, were compared to IHRA/ISO specifications and JARI-JAMA physical model in Table 4. The geometry of the JARI-JAMA headform base is complex and not enough information is available in [9] to accurately model this part in the FE model. The calculated natural frequencies of the FE impactor without skin were slightly lower (1-3.8%) than those measured in [4]. The first three frequency modes calculated in LS-DYNA are illustrated in Figure 7; the first mode shows a similar shape with that of the JARI-JAMA impactor [4]. Similar inertial properties of the FE model to those of the physical model shown in reference [4] suggest that the simplification of the base geometry in the FE model was reasonable. Overall, the geometrical, inertial and impact properties of the FE headform model met the IHRA/ISO specifications; therefore, the model was used in simulating headform-hood impacts, presented in the next section.

Table 4. Comparison between the specifications of physical adult headform impactor and FE model

Parameter	Units	IHRA/ISO specifications	JARI physical model	FE model
Mass	kg	4.5±0.1	4.494	4.5
Diameter	mm	165±1	164.5	165
Drop test corridor	G	225-275	255	250
Distance CG – GC	mm	± 2/± 10	0.4	0
1st natural frequency	Hz	Over 5,000	8,496	8,248
2nd/3rd natural frequencies	Hz	not specified	8,576/10,720	8,250/10,615
Moment of inertia	kg mm^2	0.0075-0.02	0.0115	0.01
Distance AL – GC	mm	± 2	0.4	0

*CG – Center of Gravity; GC – Geometry Center; AL – Accelerometer Location



a) First frequency mode b) Second frequency mode c) Third frequency mode

Figure 7. The natural frequency modes of the FE adult headform impactor.

Optimization of the Generic Hood FE Model

The design optimization process of the generic hood was performed using the SRSM method which is an iterative process involving the modifications of design parameters in an effort to improve the model responses. The SRSM method iteratively reduces the design space to a convergence region that possesses the greatest likelihood of containing the global optimum point based on successive meta-model approximations using polynomial regression.

In this study, the optimization process was stopped at the beginning of the forth iteration, which was considered sufficient for this exploratory phase. The design space after three iterations and the meta-model optimum approximation point - first point of fourth iteration are given in Table 5. The thickness and height of spool and the thickness of the upper panel have converged toward their lower limits favoring the deformation of these structures and gradually deceleration of the headform impactor. On the contrary, the thickness of the lower panel has converged toward a higher thickness increasing the stiffness of lower panel and reducing the hood stroke. The other two parameters, the interior diameter of spool and the connection radius converged to interior values of the design space and showed the smallest influence in the responses (HIC and total stroke) based on an ANOVA.

The impact responses of the hood at the design points chosen by the DOE scheme in the first three iterations are shown in Figure 8 a) in terms of a trade-off plot of the constraint: HIC_{15} versus the optimization objective: the total stroke. A high concentration of response points, which contains the first design point of the fourth iteration, is a region between 60-75 mm total stroke and 1000-1200 HIC_{15} suggesting that this region contains the global optimum design point. The time histories of headform acceleration and the deflection of lower hood panel of the optimum design point are presented in Figure 8 b). A slight violation of the HIC constraint could suggest that a small reduction of the limit (about 5 %) of this constraint could increase the number of feasible design points which verify the biomechanical constraint currently used in regulation.

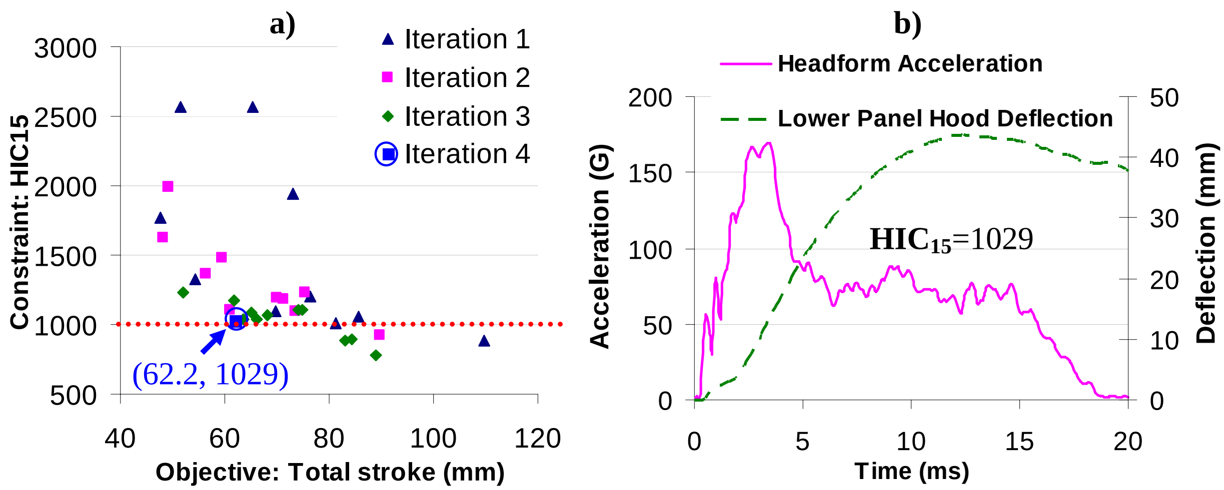


Figure 8. a) Trade-off of HIC15 versus Total stoke **b)** The time histories of headform acceleration and the deflection of Hood Lower Panel in last simulation (the fourth iteration)

Using the methodology presented in this study, the performance of other buckling structures (e.g. conical spools) should be considered as well. By varying the spool shape or spool arrangements, additional feasible solutions may result. In addition, this study showed that to accurately simulate the buckling phenomena requires fine mesh and therefore high computational power. A software solution for better simulation of buckling would be to introduce some buckling initiators in mesh structure based on the buckling modes of the structure when this feature becomes available in LS-DYNA or/and adaptive meshing . In conclusion, the convergence of the FE optimization design approach toward an optimum design suggests that this automatic method has a great potential for improving pedestrian safety, but more work is need in reducing the computational time required by FE optimization.

Table 5 Design space of spool geometric parameters at the fourth iteration

Spool Parameters		**R** mm	**d** mm	**h** mm	**t_c** mm	**t_u** mm	**t_b** mm
Variable Range	min	4.85	10	5	0.08	0.6	0.94
	max	6.77	15.7	11.8	0.11	1.22	1.55
Iteration 4		***5.81***	***12.8***	***5***	***0.08***	***0.6***	***1.25***

Conclusions

In the first part of this study, a FE model of adult headform impactor was developed in LS-DYNA based on the characteristics of the JARI-JAMA physical headform. A viscoelastic material model was assumed for the impactor skin and its coefficients were identified based on quasi-static and dynamic compression test data using an FE optimization approach in LS-Opt. Overall, the technical characteristics of the FE headform model showed to be in accordance with IHRA/ISO regulations. In the second part of this study, the headform FE model was used in the design optimization of a generic vehicle hood. The optimization problem was defined in LS-Opt according to the stiffness and energy-absorbing characteristics required by new pedestrian safety regulations. The whole process of the development of the hood FE model, evaluation of impact tests, and proposing a better design in the predefined ranges of geometric variables was integrated using an automated approach. Even though the design approach was limited to simulating only impacts at the center of the hood and only one geometric shape of the buckling structure, the successful convergence of the design suggests that the method proposed has potential for improving hood design according to new safety requirements without the high monetary and time expenses associated with physical prototype testing.

References

*1. Crandall J., Bhalla K., and Madeley N. (2002) "Designing road vehicles for pedestrian protection" British Medical Journal, **324**:1145-1148*

2. Kerkeling C., Schäfer J., and Thompson G.M. (2005) "Structural Hood and Hinge Concepts for Pedestrian Protection", Paper No. 304, Proceedings of the 19th ESV Conference

3. Frank T., Kurz A., Pitzer M., and Söllner M. (2003). "Development and Validation of Numerical Pedestrian Impactor Models" 4th European LS-DYNA Users Conference

4. Neal M.O., Kim H.S., Wang J.T., Fujimura T., and Nagai K. (2003) "Development of LS-DYNA FE Models for Simulating EEVC Pedestrian Impact", Paper No. 335, Proceedings of the 18th ESV Conference

5. Okamoto Y., Akiyama A., Nagatomi K., and Tsuruga T. (1994) "Concept of Hood Design for Possible Reduction in Pedestrian Head Injury", Paper No. 94-S7W-14, Proceedings of the 14th ESV Conference

6. TrueGrid Manual (2001) Version 2.1.0., XYZ Scientific Application, Inc.

7. John O. Hallquist (1998) LS-DYNA Theoretical Manual, LSTC.

8. Stander N., Roux W., Eggleston T., and Craig K. (2004) LS-Opt User's Manual, LSTC

*9. Matsui Y., and Tanahashi M. (2004) "Development of JAMA-JARI pedestrian headform impactor in compliance with ISO and IHRA standards", International Journal of Crashworthiness, Vol. **9**(2):129-139*

*10. Matsui Y., Takabayashi M., and Tanahashi M. (2005) "Characteristics of 3.5 kg pedestrian headform impactor prototypes developed by JAMA–JARI and ACEA–TNO", International Journal of Crashworthiness, Vol, **10**(2):197-210*