Numerical Procedures for the Improvement of the Structural Response of Thermoplastic Manufactured Parts

STEFAN TABACU[1], ANTON HADAR[2]*, DANUT MARINESCU[1], MARIANA IVĂNESCU[1], VICTOR BALASOIU[3]
[1] University of Piteşti, Faculty of Mechanics and Technology, 1 Targu din Vale str., 110040, Piteşti, Romania
[2] University Politehnica of Bucharest, IMST, 313 Splaiul Independenţei, 060042, Bucharest, Romania
[3] University Politehnica of Timisoara, 1 Mihai Viteazul Blvd., 30022, Timisoara, Romania

For the modeling and usual analysis of mechanical structures it is considered that the whole structure and its components accomplish one of the fundamental conditions of strength of materials – the condition of continuity and homogeneity. This paper presents some modeling techniques useful for the optimization of the numerical models in order to improve the structural response for impact applications. As some parts have to be damaged a material model capable of material failure must be used. The differences between the results obtained using a model without failure and a model with failure compared to the experiments were discussed. The second procedure is regarding the random distribution of the thickness properties of the elements for a selected area within the user specified tolerance. Procedures implemented in custom written code and the algorithms presented for the random distribution were presented.

Keyword: material homogeneity, material model, random thickness, numerical simulation

One of the most demanding market in terms of quality, reliability, on board equipment, safety and price is related to the automotive industry. Therefore efforts are paid in the direction of minimizing cost and improving product performance. Costs are reduced using high performance design and engineering products including tools for numerical simulation. One of the main advantages of the numerical simulation is that once the environment is set and the numerical model defined, various scenarios can be tested.

But there is still room for physical testing, because numerical models must be validated using experiments. But even so, the expenses on testing are limited because the number of experiments can be reduced to minimum. Starting from initial design or from a previous model the current model is validated and updated according to specifications. But there may be one thing that is still missing and this is regarding the removal of some uncertainties from the numerical model. Most of the simulation models are built considering a homogenous distribution of properties. Unfortunately in some cases the manufacturing process alters the initial performances of the designed product [1]. Thus the numerical model must comply with these changes in order to improve the results.

A vehicle interior assembly, the cockpit module, was tested and analyzed using experiments and numerical simulation. The assembly must comply with the ECE 21 (in Europe) or FMVSS 201 (U.S.A.) safety measures regarding the passengers' protection in case of impact. The testing device, experimental results, development and validation of the numerical model were presented in a previously published paper [2].

The cockpit module consists in the instrument panel, steering column, air conditioning, glove box, cross car beam, different storage area, navigation and audio system and different decorative elements. The main components are manufactured by polypropylene (PP) like ExxonMobil Chemical EXXTRAL® BMT 222 for the dashboard and ExxonMobil Chemical EXXTRAL® HMU 202 for the windshield demister.

The material model and materials stress – strain curves were analyzed. A special test machine manufactured by Zwick/Roell was used. The traction test velocity was of 50[mm/min] according to the materials' specification datasheet. Engineering stress – strain data was obtained and used for the definition of real stress – strain data used for the numerical simulation [2- 9].

The numerical models were solved using LS-Dyna v.970 code, an explicit, general purpose, and widely appreciated solver for transient, slow and high velocity dynamics simulations [10-12].

Structure response, impact parameters, failure model and damage pattern were analyzed and results from the numerical simulation were compared with the experiment.

Experimental part
Numerical procedures
Virtual testing world allows different models to be developed and tested. But even if this virtual world is quite permissive the engineer must design and test in agreement with the real world information and data.

Experimental methods are very costly by this point of view, because a detailed investigation of the event requires a very sophisticated instrumentation using strain gauges, force and displacement transducers. Even in this case the results may reflect localized phenomena that can lead to an under or over evaluation of the generalized response.

By this point of view numerical methods allow a much in depth analysis of the structures. The structured numerical model contains finite elements that are organized in parts according to their mathematical and material model and, in case of shell elements, thickness. Finite element method consists in the decomposition of complex structures, and the application generally known rules and equations in order to obtain the results.

The element stiffness matrices can be derived from the principle of minimum potential energy I.

$$I = strain\ energy - work\ done\ by\ external\ forces \qquad (1)$$

* email: antonhadar@yahoo.com; Tel.: 40 021 4029204

The total energy of a mechanical system must be conserved. In case of a mechanical system with initial kinetic energy the total energy balance must be reflected in the internal energy accumulated due to deformation, friction, heat and/or sound wave propagation and the change of the kinetic energy.

The total energy of the system is, in this case, the initial kinetic energy of the false head used both for the experiment and for the numerical simulation.

As mentioned, the virtual model can work in ideal condition. It is obvious that the impact test will produce damages and ruptures in the tested parts and all of these because in some areas the stress will be above the ultimate stress value.

Firstly, the model is designed in order to allow high strains so that parts will not collapse. This step is performed in order to check the energy balance in order to ensure for the good performances of the numerical model.

Stress – strain data are implemented in the computational model using specialized cards. When these data are available, the most used material model for LS-Dyna simulations is *MAT_24 or *MAT_PIECEWISE_LINEAR_PLASTICITY. The material model is using stress – strain curves defined by points and also the strain at failure may be defined.

Although the numerical model is constructed using shell elements LS-Dyna solver uses the shell thickness in order to compute the stress strain profile. The number of integration points through the shell's thickness is specified using *CONTROL_SHELL card. As a consequence the *MAT_24 material model was updated in order to handle the shell thickness and therefore *MAT_123 or *MAT_MODIFIED_PIECEWISE_LINEAR_PLASTICITY can be used [2, 4, 5, 7].

This adaptation is required by the model that may deform by bending. Setting a number of integration points that must fail before the elements fails will ensure a better response of the structure for both in-plane loads (tension, compression) and out-of-plane load (bending, twisting).

Figure 1 presents the energy balance in terms of kinetic energy, internal (deformation) energy and total energy. There are no changes or important deviations from the nominal value in the total energy. If any, they are due to the failed elements. The procedure for the model with damage, implemented via the material model, is to remove from the calculation the elements with high stress or strain.

From the components used to define the geometry of the numerical model the parts that contribute to more than

10% of the system internal energy were selected. Although the model consists in a quite high number of parts (52) the previously mentioned condition is fulfilled only by two components – the impact section of the dashboard and the stiffening structure located under (as the windshield air deflector).
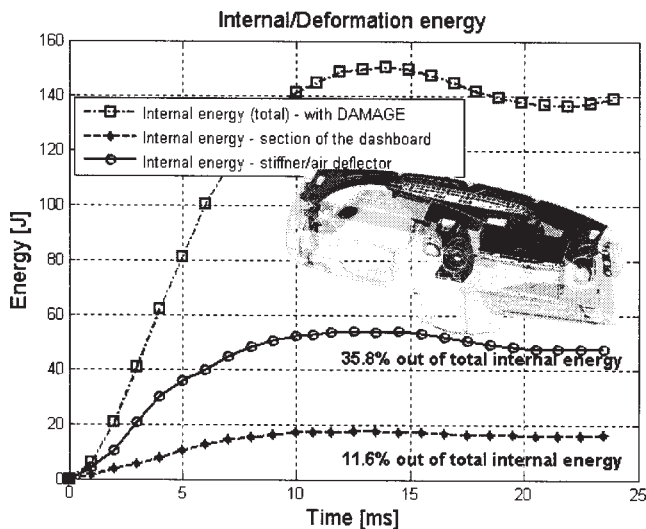

Fig. 2. Internal/deformation energy

Figure 2 presents the parts that play a significant role in the internal energy history.

Thus the potential area of parts for optimization can be defined because changes in the structural performances of these parts will improve the response of the entire assembly.

Also different methods for assigning part properties in terms of thickness and material formulation can be applied to a few numbers of parts and the control of these changes or modeling improvements is more accurate [1, 2, 8].

After the experiment the stiffening structure was damaged. The small fragments of the part were analyzed and the thickness was measured. The nominal value of the part is of 2.5 mm. For some fragments the thickness was below this value. It may be a result of the manufacturing process and therefore the numerical model must be updated in order to consider these aspects. One procedure consists in very detailed measurements of the part and an update to the numerical model. But it may be a very expensive procedure in terms of time and equipment.

The free edges of the main part and failed fragments can be easily measured (fig. 3). Using these values the deviation from the nominal thickness is computed. As the variation of the thickness occurred for more than one instances and for more than one sections of the analyzed part a new requirement for assembly modeling has to be stated.

The section selected for the random distribution is the one that was damaged during the impact. In order to accomplish this task a collaborative environment between the finite elements solver and new software is required.

A custom code written in Matlab was created and used for the random distribution. The input is the initial file with the numerical model and the outputs are a file with the sections for which the random procedure was applied and the second one is the initial file with the lines containing the elements selected for random distribution remove. There are two files generated by the code.

Figure 4 presents the organization of the custom written code.

The code can update the thickness, material properties or both of them for one element. The material properties
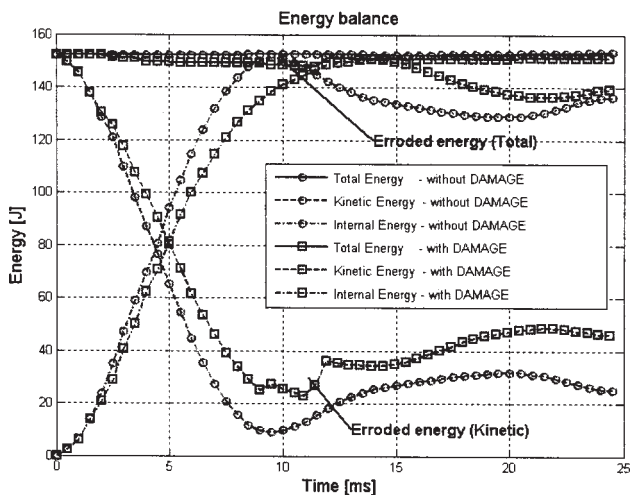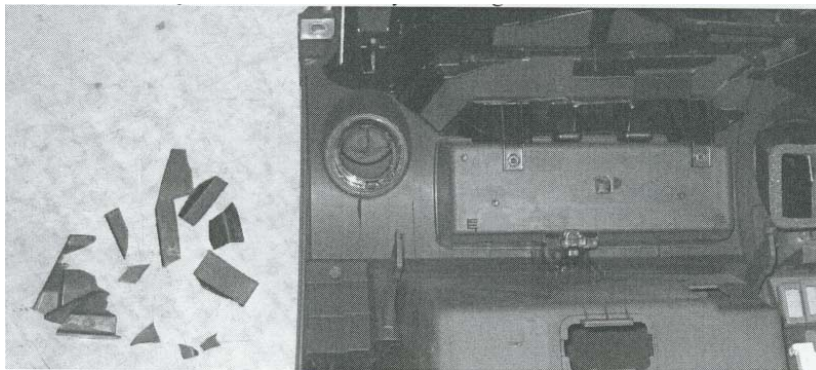

Fig. 1. Energy balance

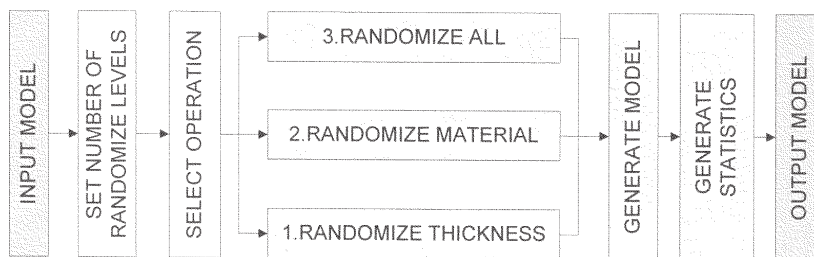Fig. 3. Dashboard assembly after the experiment
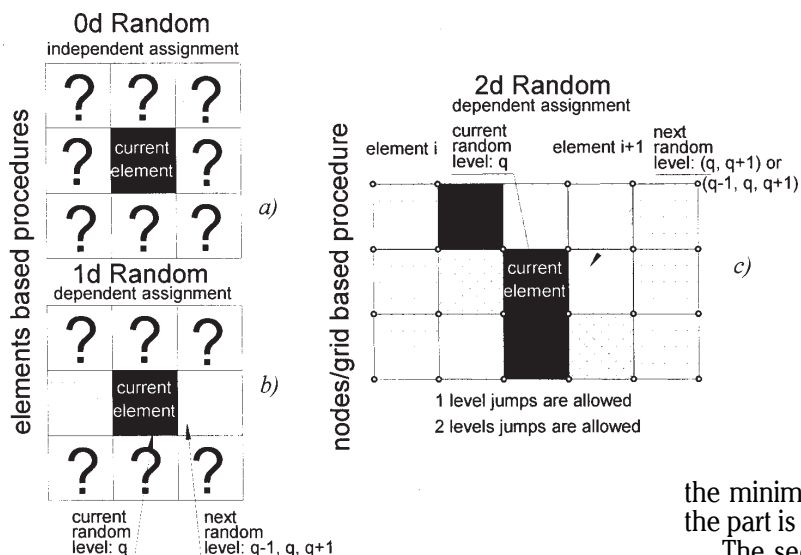

Fig. 4. Organization of the custom code

Fig. 5. Random procedure
a)elements based – independent assignment
b)element based – dependent assignment
c)node based – dependent assignment

may be added because in some cases, due to the manufacturing process, the model is not homogenous.

A procedure, to consider these changes from the nominal value of the thickness, is to set a random distribution within the elements consisted by the selected part.

The paper will further concern only the application of the first option, regarding the changes in the part thickness.

The parameters of the random procedure are the deviation from standard expressed in percents and number of levels for the thickness distribution for the elements.

Three random procedures may be implemented. The difference between these procedures is regarding the number of dependencies created between neighbor elements.

The first one is an element based procedure because it is element oriented. In this case the neighbor elements or the currently selected element are not taken into account (fig. 5a). The property assigned to the current element may be any of that defined by setting the random level. So, it is possible for two elements in row to have one with the nominal/maximum thickness and the next one to have

the minimum thickness. The distribution of thickness in the part is not accurate.

The second one is also element based (fig. 5b). The current elements and its neighbors are identified and even if it is about a random procedure there are some rules added. Considering the number of order, two consecutive elements can not have one the maximum (nominal thickness) and the next one the minimum thickness. The random distribution can be done only to a level of -1, 0 or +1 starting from the current element. The extreme values are also considered and for the minimum thickness the distribution is 0, +1, while for the maximum thickness the distribution is -1, 0.

The third random procedure is a 2 dimensional procedure (fig. 5c). This procedure is related to the nodes that are used to define an element. The custom built code searches through the numerical model definition to identify any existing elements that may have one or more nodes in common with the currently selected element. Once these elements are identified the random procedure is applied in to order to assign the different properties. The random level is stored for each element.

The numerical procedure for this assignment is shortly described. The code starts with -1 entry for the element random level and the identification of neighbor elements is applied. If in the main list, containing the elements, the
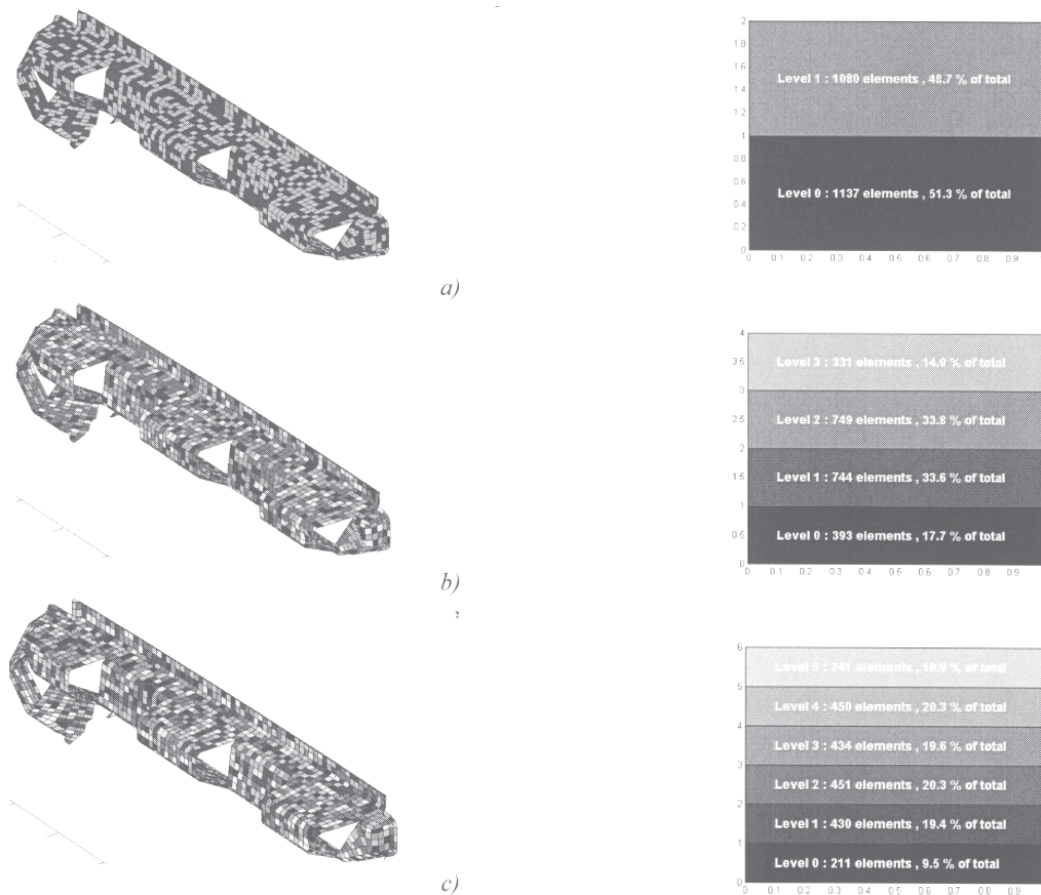
Fig. 6. Random assignment of the properties
a) number of levels: 2; b) number of levels: 4; c) number of levels: 6

random level for the current elements is -1 then a property is assigned. For the neighbor a property of -1, 0, +1 from the current level is assigned. In this case for any two elements the differences between the random level is limited to 2.

If an element it was already selected as a neighbor of a previously defined element, than, its random level property is used as a start in order to define the random level for its own neighbors and the procedure is repeated.

Using the internal energy history the parts with greater participation were selected. The model can be even more sub structured in order to decrease the area for design update. As a consequence from the stiffener / air deflector part (fig. 2) only the section that was damaged (fig. 3) was selected for the random procedure.

The code is applied for different numbers of random level. The finite elements structure is colored according to the property assigned to each element (fig. 6). Also the corresponding legends with some statistical results are presented. A random level of 2, 4 and 6 is applied. The deviation from the nominal thickness if of -3% (minimum thickness is 2.42 mm).

The results, obtained using the custom code, are presented in figure 6.

The distribution of the elements is in a good agreement with the Gaussian or standard normal distribution. This fact and the results of the random distribution are used to validated the custom written code.

Results and discussion

A number of numerical simulations were performed and the results compared. The first simulation is with a material model without failure. The second model has a material formulation with failure and it was previously validated using experiment [2]. At this stage the main criteria for the evaluation of the numerical model's performance was the value of HIC (Head Injury Criteria) that is used in order to validate the components from the vehicle interior in case of impact. The event duration is less than 36 milliseconds thus HIC was calculated during a 15 milliseconds time interval. For the numerical simulation $HIC_{15ms} = 259.9$ and for experiment $HIC_{15ms} = 248.8$ with an error between measured and calculated data of 4.4%. The parameters are within the limits of ECE 21 specifications.

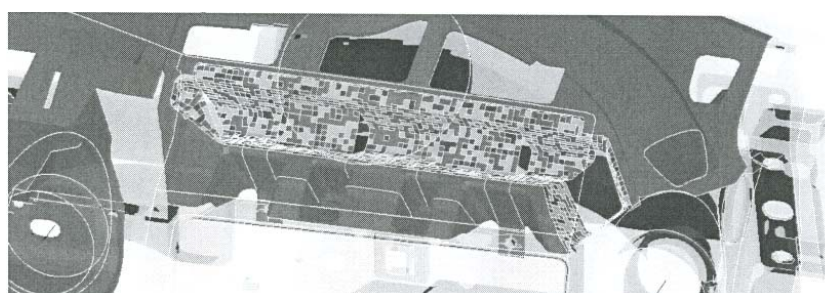The third model uses the random distribution of the

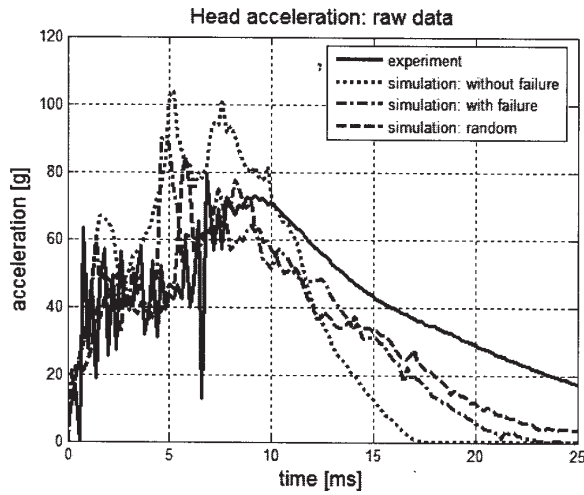Fig. 7. Application of the random distribution of the properties
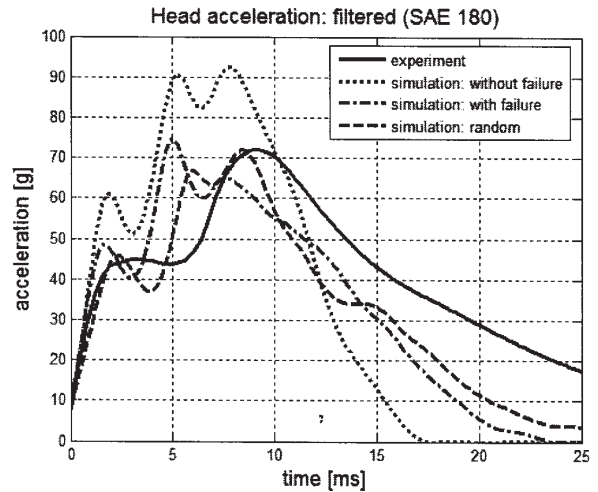
Fig. 8. Raw acceleration data

Fig. 9. Filtered data (SAE 180 Hz)

thickness as a method for improving the accuracy of the results. Figure 7 presents the section used for the random distribution as it is integrated into the computational model.

Two sets of data are available. There are the raw data as obtained from the experiment and simulation and there are the filtered data. Filtering is required in order to remove the noise or some other reads that may alter the results, thus making the evaluation of the post processed data more effective in terms of decisions efficiency. Figure 8 presents the raw data used as inputs for data filtering. The measurements were performed at the gravity centre of the headform.

Regarding the safety criteria (e.g. HIC) raw data can be

Table 1
HIC SAFETY INDEX

	HIC [-]	Error
Experiment	249	-
Model without failure	508	104%
Model with failure (constant thickness)	260	4.4%
Model with failure (random thickness)	255	2.8%

used as they are obtained. Table 1 presents the value of the safety parameter as obtained from the experiment and simulations.

Past work [2] concerned the validation of a numerical model. As mentioned the differences between the experiment and the previously analyzed model are in a good agreement, within a generally accepted error (+4.4%). It is important to underline the differences

between the experiment and the simulation model using a material formulation without failure. There is an important difference between the results (+104 %). As a consequence part damage and material failure must be taken into account.

Regarding the simulation with a random distribution of the thickness, there is an improvement of the results, the error for HIC index as obtained from the simulation and experiment is of only +2.8%.

But, as mentioned, in order to outline the differences between the computational models a specialized filter was used (SAE 180 Hz). The filter is used in automotive industry in order to smooth data and make results more readable [13]. Figure 9 presents the filtered data.

There may be noticed some similarities between the results. All curves show two peaks located at different times. The location in time of the first peak is for all the curves within a very tight interval. By that time all of the components of the cockpit module are in contact, including the metallic section (cross beam) used for fixing the assembly. The acceleration values are different due to the material modeling (without failure, with failure) and to the thickness assignment to the parts (constant thickness, random thickness).

The time location of the second peak is different. By now some of the section should have failed. For the model without failure the parts integrity is maintained. Thus from all of the cases, this one is the stiffer and the recorded acceleration does prove this statement. For the models with failure the maximum recorded accelerations are

Table 2
PARAMETERS FROM IMPACT ANALYSIS

Model	experiment	Model without failure		Model with failure (constant thickness)		Model with failure (random thickness)	
Parameter	measured	measured	error	measured	error	measured	error
a_{max} [g]	72,02	92,46	28,38%	74,76	3,80%	72,03	0,01%
$a_{average}$ [g]	29,53	29,93	1,35%	28,4	-3,83%	28,49	-3,52%
a_{peak_I} [g]	44,96	61,13	-	48,48	-	45,71	-
a_{peak_II} [g]	72,02	92,46	-	74,73	-	72,03	-
t_{peak_I} [ms]	3,5	1,9	-	1,7	-	2,3	-
t_{peak_II} [ms]	9,2	7,9	-	5,1	-	8,5	-
Δt_{peaks} [ms]	5,7	6	5,26%	3,4	40,35%	6,2	8,77%
Error (sum)		35,00%		47,98%		12,31%	

comparable. There is only an issue of time location of the peaks.

Table 2 presents the results obtained form the simulations and experiment.

As presented in table a number of parameters were analyzed: a_{max} [g] - maximum recorded acceleration, $a_{average}$ [g] the average acceleration, a_{peak_I} [g] - acceleration recorded at the first peak, a_{peak_II} [g]- acceleration recorded at the second peak, t_{peak_I} [ms] - time location of the first peak, t_{peak_II} [ms] - time location of the second peak, Δt_{peak} [ms] - time interval between peaks.

The reference model is considered the experiment. The parameters that were compared, and have an error computed, are the maximum recorded acceleration, average acceleration and time interval between the two peaks.

The simulation model without failure offers good results in terms of time between peaks and average acceleration, but the maximum acceleration and the HIC index are outside the acceptable tolerance interval (±5% from nominal for accelerations, ±10% for time because the time scale is more discrete).

The simulation model with failure offers good results in terms of maximum acceleration, average acceleration and HIC index. In terms of structural performances this numerical model can be, and it was, validated for virtual testing [2].

The simulation model with failure and random distribution of the thickness with an -3% deviation from the nominal offers very good results in terms of maximum acceleration, average acceleration, HIC index and time between peaks. Concluding with these results it may be said that particular modeling techniques are applicable for improving the performances of numerical models.

Conclusions

For the modeling and usual analysis of mechanical structures it is considered that the whole structure and its components accomplish one of the fundamental conditions of strength of materials – the condition of continuity. The great variety of problems in engineering practice yields sometimes situations where discontinuities appear in some components [1].

This paper presents some modeling techniques useful for the optimization of the numerical models in terms of improving the structural response for impact applications.

Using experiments performed with equipment constructed according to ECE 21 / FMVSS 201 regulations [2], the results obtained using three numerical models are compared and solutions for numerical modeling are presented.

The first step was concerning the material model used. As some parts have to be damaged a material model capable of material failure must be used. The differences between the results obtained using a model without failure

and a model with failure compared to the experiments were discussed.

The second procedure is regarding the random distribution of the thickness properties of the elements for a selected area with a user specified tolerance. Procedures implemented in custom written code and the algorithms presented for the random distribution were presented.

This application was required because the parts measurements presented some deviations from the nominal value, probably due to the manufacturing process.

The performances of the numerical model were evaluated and compared with the experiment.

Even the second model used (model with failure and constant thickness) offered good results, the third model (model with failure and random thickness) can bring the results much closer to the ones from the experiment.

Although simulation codes are very powerful in terms of computational performance, accuracy of results a collaborative environment between these codes and custom written computer programs can improve the structures' response by considering the real life distribution of properties within the analyzed parts.

References

1. HADĂR, A., CONSTANTINESCU, I.N., JIGA, G., IONESCU, D.S., Mat. Plast., **44**, nr. 4, 2007, p.354
2. TABACU, S., HADĂR, A., MARINESCU, D.G., MARIN, D., DINU, G., IONESCU, D.S., Mat. Plast., **45**, nr. 1, 2008, p.113.
3. CURTU, I., MOTOC, D.L., Mat. Plast., **45**, nr. 4, 2008, p.366
4. LUPEA, I., CORMIER, J., Mat. Plast., **44** ,nr. 4, 2007, p.339
5. DuBOIS, P.A., KOLLING, S., KOESTERS, M., FRANK, T., Int. J. Impact Eng., **32**, nr. 5, 2006, p.725
6. ARRIAGA, A., LAZKANO, J.M., PAGALDAI, P., ZALDUA, A.M., HERNANDEZ, R., ATXURRA, R., CHRYSOSTOMOU, A., Polym. Test., 26 nr. 3, 2007, p.284
7. HUBERT, F., HIERMAIER, S., NEUMANN, M, Material Models for Polymers under Crash Load. Existing LS-DYNA Models and Perspective, 4th German LS-DYNA Forum, Bamberg, 2005
8. FRITZSCHE, P., WEDER, M., WYSS, I., HÖRMANN, M., MÜLLER, Compos. Struct., **85**, nr. 4, 2008, p.337
9. KRIVACHY, R, RIEDEL, W., WEYER, S., THOMA, K, Int. J. Crashworthiness, **13**, nr. 5, 2008, p.559
10. Livermore Software Technology Corporation, 'LS-DYNA Keyword Manual v970', Livermore, CA, Livermore Software Technology Corporation, April 2003
11. PAUN, V.P., Mat. Plast., **40**, nr. 3, 2003, p. 127
12. PAUN, V.P., Mat. Plast., **41**, nr. 4, 2004, p. 262
13. TABACU, S., PANDREA, N., Int. J. Crashworthiness, **13**, nr. 4, 2008, p.387