Wave propagation in automotive structures induced by impact events

Steffen Mattern, Karl Schweizerhof
Universität Karlsruhe, Institut für Mechanik

Institut für Mechanik
Kaiserstr. 12, Geb. 20.30
76128 Karlsruhe
Tel.: +49 (0) 721/ 608-2071
Fax: +49 (0) 721/ 608-7990
E-Mail: ifm@uni-karlsruhe.de
www.ifm.uni-karlsruhe.de

Wave propagation in automotive structures induced by impact events

Steffen Mattern   Karl Schweizerhof

1 Introduction

The prediction and numerical simulation of the propagation of high frequency vibrations caused by impact is very important for practical purposes and proper modeling is an open issue. In present automobiles more and more technical devices, which are getting their informations from sensors, are used to improve the safety concerning passengers. For example the activation of airbags and safety belts is controlled by sensors. For this reason it is important to assure that sensors perform reliably at a specific frequency range.

For the development of sensors, it is very important to know the kind of signal, which has to interpreted. When an impact happens at a specific location of the structure, a wave starts to propagate immediately. At the time, when the oscillations reach the point where the senor is located, they have already been influenced by many structural and material parameters, e.g. edges, spotwelds, foam material. It is necessary to calibrate the sensor in a way that if and only if a sufficiently hard impact has occurred, e.g. the airbag is activated.

In order to reduce costs for complex and expensive experiments, it is useful to perform numerical simulations of suitable structural parts to obtain general informations about wave propagation mechanisms in complex structures and to find out about the proper modeling with FE-programs. In this project, the powerful, highly parallelized commercial finite element code LS-Dyna [7, 6] is used to simulate different models and evaluate the influence of several modeling modifications and of other simulation parameters. In this contribution, a partial structure is described in detail and some of the performed modifications are explained. Different contact formulations are used as well as different material models for special parts. Damping is applied in parts of the structure and spotwelds are modelled with different methods, which is also a very important issue of simulation in automotive industry.

Because of the large number of numerical investigations, performed during the project, only a few results are shown in the following sections. The results of the simulations are compared to experimental data with a special focus on amplitudes and frequencies at the locations of the sensors.

The models contain about $25000 - 50000$ solid and shell elements and the simulated time of the impact event was in all cases 0.1 s, which lead to over $2 \cdot 10^6$ time steps per computation, using the explicit time integration available in LS-Dyna. With LS-Dyna, such kind of problems are particularly suited for computing on a high performance parallel computer, such as the *HP-XC6000 Cluster* at the University of Karlsruhe. Only with parallelization, it is possible to perform parametrical studies, which are necessary to obtain general knowledge about wave propagation and their proper modeling in complex structures in an appropriate time. Within the project, also investigations on academic examples with rather few elements were processed in order to gain information about wave propagation mechanisms as e.g. discussed in [5, 1] and their simulation with finite elements. However, several important effects can only be recognized at sufficiently complex models, which motivates the costly studies, presented in the following.

The results of the experiments were made available by other sources, thus they will not be explained further in the following. The main issue will be the simulation of the numerical models and the influence of different parameters.

2 T-shaped, spotwelded structure impacted by a rigid ball

2.1 Description

The structure, discussed in this contribution is a steel construction of top-hat profiles and sheets, connected with spotwelds, which is impacted by a metal ball at the top. This is a typical part of an automobile containing all relevant structural issues. In the experiments, the displacements were measured at one (S_1) and the accelerations at four sensors (locations $A_1 - A_4$), as shown in Figure 1. The finite element model (Figure 2) consists of about 33500 shell elements [3, 4] and 145 spotwelds, each modeled with one 8-node solid element. The steel plates are modeled with an elastic-plastic material. For the spotwelds, the specific material `*MAT_SPOTWELD` in LS-Dyna, which consists of an isotropic hardening plasticity model coupled to four failure models [7] is used. The metal ball, the impactor, is assumed to be a rigid body. Contact between impactor and plate is realized by an automatic penalty based segment-to-segment contact formulation. In order to connect

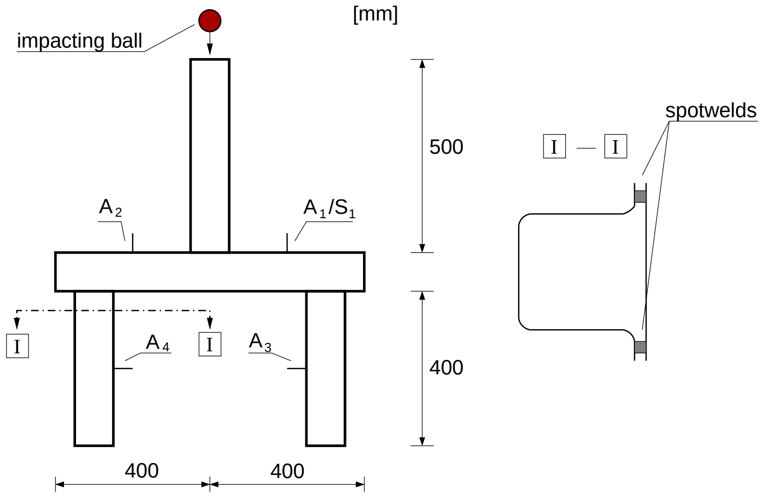


Figure 1: Geometry of the T-shape structure, sensor location denoted by S_1 and $A_1 - A_4$

two metal plates with spotwelds, the *CONTACT_SPOTWELD is applied, which ties the nodes of a solid element to the two neighboring deformable shell surfaces with constrains [7].

The results from the simulation of this basic finite element model, depicted in Figure 1, compared to experimental results provided to the authors, are shown in Figure 3. This is then taken as a basis for several parametrical studies, computed on the *HP-XC6000 Cluster*.

2.2 Parametrical studies

2.2.1 Stabilization of spotwelds

As already mentioned, the spotwelds are modeled with a standard eight-node hexahedral finite element using one-point under-integration combined with a stabilization against unphysical hourglass modes, as shown exemplarily in Figure 4. The chosen hourglass stabilization – Belytschko-Bindemann assumed strain co-rotational stiffness form – is described in detail in [2, 4] and works properly for all applications performed in the project. By default, i.e. if no hourglass stabilization is chosen, a standard viscous hourglass control is used [7].

In Figure 5, the results of the simulations with standard viscous and with Belytschko-Bindemann stiffness stabilization of the spotwelds are given. As expected the structure reacts slightly stiffer with stiffness controlled stabilization of the spotwelds, which leads to higher frequencies in the displacements.

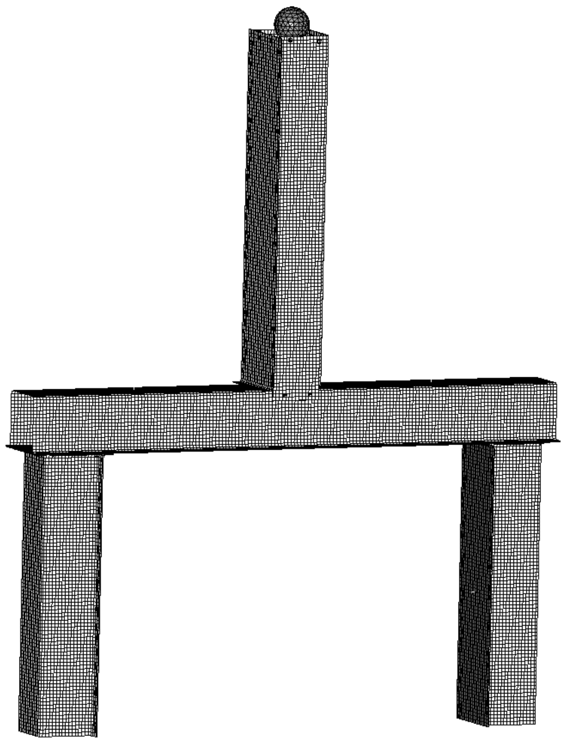
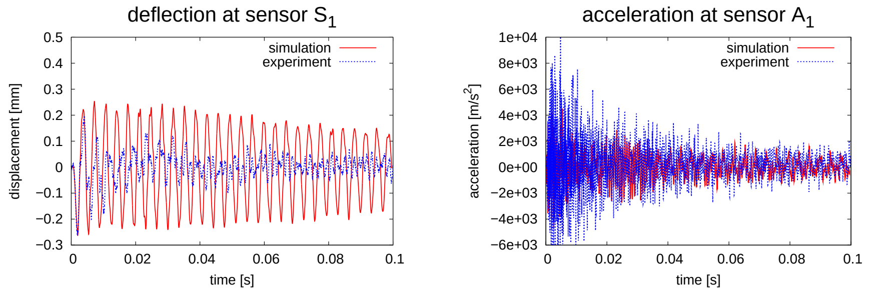
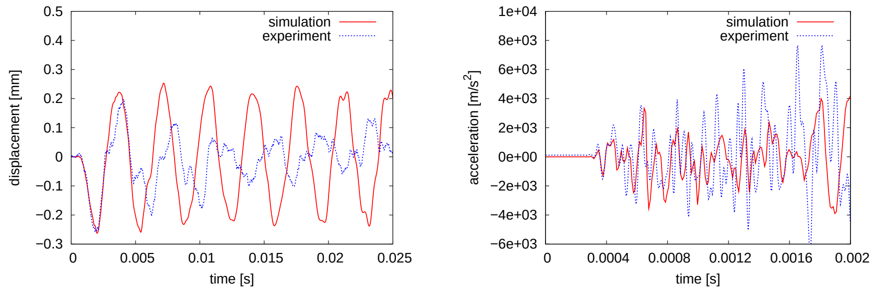


Figure 2: FE-Model of the T-shaped structure



(a) full simulated time



(b) cutout – beginning of the simulation

Figure 3: Experimental and simulated displacements and accelerations at sensor 1

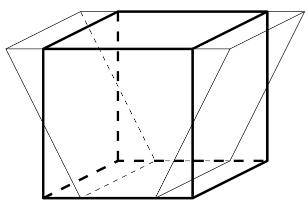
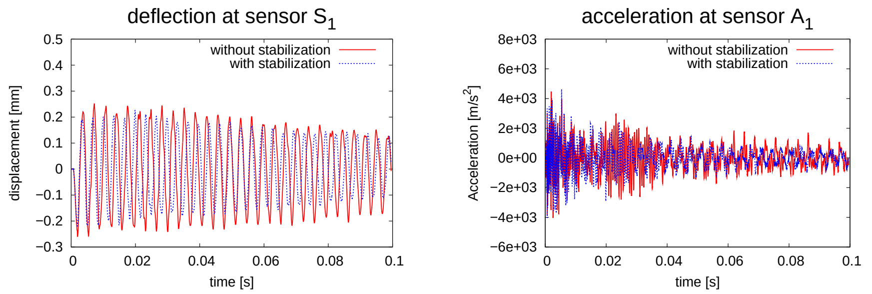
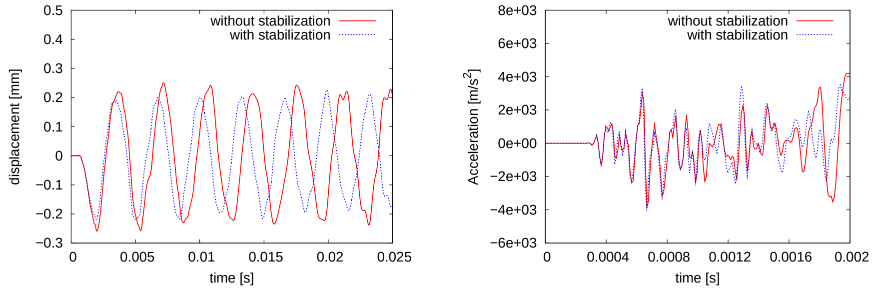


Figure 4: Example for an hourglass mode



(a) full simulated time



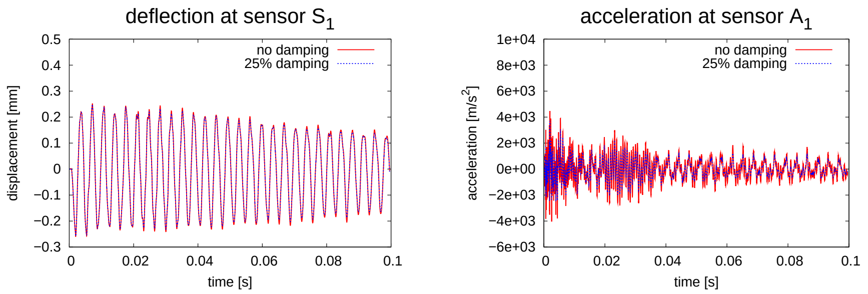
(b) cutout – beginning of the simulation

Figure 5: Simulated displacements and accelerations with standard viscous- and with stiffness-controlled hourglass stabilization of the spotwelds

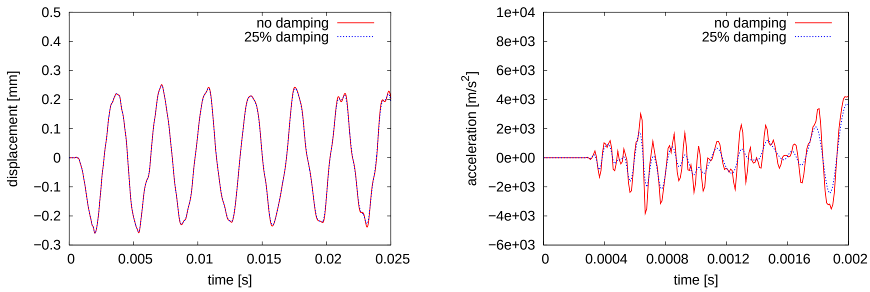
Also a reduction of the amplitudes can be noticed. The right side of figure 5 shows also only small changes in the accelerations. The properties concerning propagation of waves through the spotwelds have slightly changed as a consequence of stiffness controlled stabilization, which shows that there is a small influence of unphysical hourglass modes on the results of the original simulation. For problems with larger displacements and higher amplitudes, the effect would be more obvious, which means that for such kind of problems (e.g. crash simulations), hourglass stabilization of under-integrated finite elements is mandatory.

2.2.2 Applied damping

As can be seen in Figure 3, the displacements decay very quickly in the experiment, but in the simulation, the amplitudes stay almost constant. This leads to far overrated displacements at the end of the simulation. The reason for the energy loss in the experiment is system damping, which is due to material damping as well as due to dissipation at boundaries and joints. Mechanically, this effect can be described with Rayleigh damping, where the



(a) full simulated time



(b) cutout – beginning of the simulation

Figure 6: Studying the influence of stiffness proportional damping

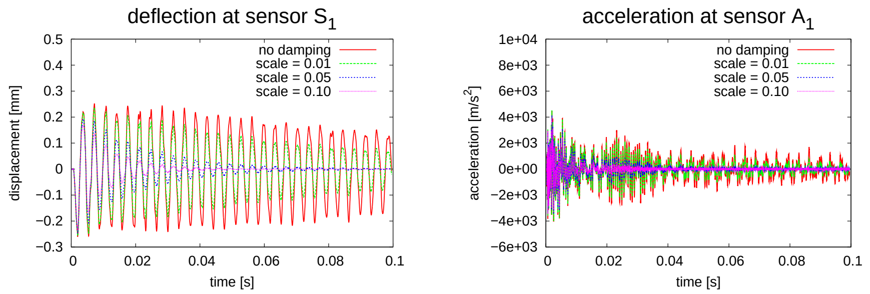
damping matrix can be defined as

$$\mathbf{C} = \alpha\mathbf{M} + \beta\mathbf{K}. \tag{1}$$

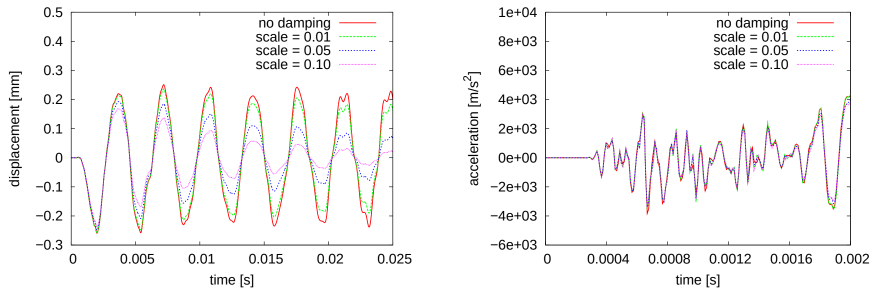
Mass proportional and stiffness proportional damping is possible, which can be defined for the whole structure, or only for specific parts. Fairly problematic is the validation of the damping parameters α and β, thus parametrical studies are necessary.

First, only stiffness proportional damping is applied, which leads to damping of high frequencies. As coefficients, values from 0.10 to 0.25 were chosen, which roughly corresponds to 10% to 25% of damping in the high frequency domain [7]. For better visualization, in Figure 6 only the results with maximum damping (25%) are plotted. It can be seen in figure 6, that damping of high frequencies has almost no influence on the displacements, and leads to a rather small reduction of amplitudes of the accelerations.

Then pure mass proportional damping was applied. Mass weighted damping is used to damp all motions including rigid body motions, this means damping in the lower frequency range. It is also possible to damp only special parts, or chose different damping coefficients for different parts. The results in figure 7 show – as expected – strong influence of the mass weighted damping on the amplitudes of displacements and acceleration.



(a) full simulated time



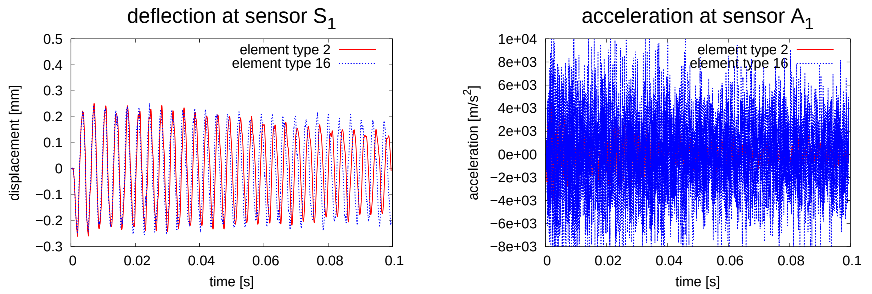
(b) cutout – beginning of the simulation

Figure 7: Studying the influence of mass proportional damping

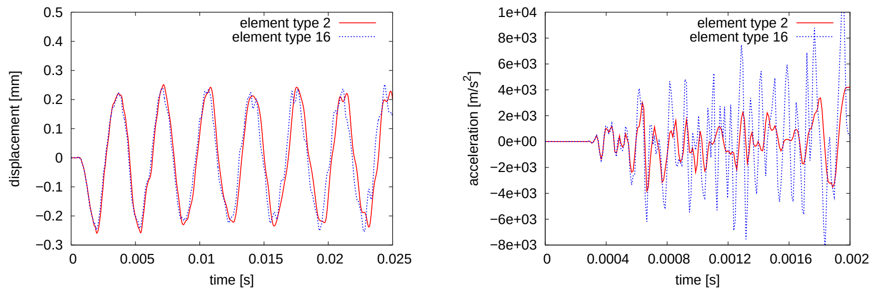
Comparing the results of both described formulations, mass proportional damping leads to results, which are closer to the experimental data. Obviously, the main part of energy loss happens in the lower frequency domain. Another reason for dissipation of energy is the impact event. In the simulation, the ball is modeled as a rigid body, hence here no loss of energy is implied. To obtain results, closer to reality, also a contact damping could be applied, which means a dissipation of energy at the impact event. Further investigations concerning the combination of different ways of modeling damping are necessary.

2.2.3 Influence of shell element formulation

In order to investigate the influence of the element formulation on the simulation result, besides the standard shell elements with hourglass-stabilization also fully integrated shell elements with an assumed strain formulation for the shear terms [7] are used to model all metal sheets. Spotwelds, impactor and contact formulation, as well as all boundary conditions are kept unmodified, so differences in the results can be associated exclusively to the element formulation. As expected, the fully integrated shell elements react much stiffer, which is depicted in Figure 8. Although the displacements show



(a) full simulated time



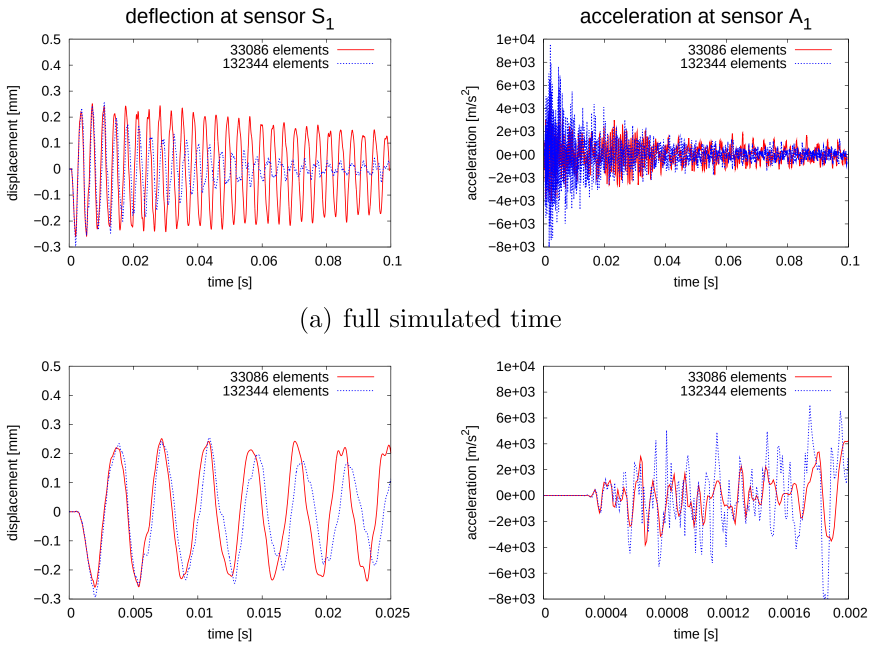
(b) cutout – beginning of the simulation

Figure 8: Comparison of standard (type 2) shell element with viscous hourglass control and fully integrated (type 16) shell element

only slightly higher frequencies, the differences in the accelerations can be clearly recognized. Frequencies as well as amplitudes are considerably higher than in the simulations with hourglass stabilized elements, which indicates a stiffer structure. In addition we have to note, that the choice of the element formulation strongly affects the efficiency. The computation with fully integrated shell elements takes about three times as long as the computation with under-integrated elements. Concerning the use of different shell element formulations, we have to conclude that, fully integrated elements are better suited for predicting the higher frequency domain. However the higher stiffness of this formulation leads to overrated amplitudes especially regarding the accelerations.

2.2.4 Refinement of the mesh

As described in section 2.1, the original discretization, depicted in Figure 1 contains 33086 shell elements. The average element length in this model is 5 mm. In order to investigate the influence of mesh refinement on the results, the element length in both directions was reduced to about 2.5 mm, which lead to 132344 shell elements. All computations of these models were per-



(a) full simulated time

(b) cutout – beginning of the simulation

Figure 9: Two levels of discretization of the steel structure

formend on 8 processors of a so called fat-node on the *HP-XC6000 Cluster*. The CPU-time per processor of this problem was about $3 \cdot 10^4$ s, which leads to a simulation time of approx. 8.5 h for the complete analysis. The results in Figure 9 show higher frequencies and amplitudes in the accelerations and quickly decreasing amplitudes in the displacements. Compared to the experimental data, given in Figure 3, frequencies and amplitudes correlate much better for the refined mesh. However, in the first few cycles, both simulations show very similar results, which is clearly visible for the displacements.

Obviously the lower frequency response is well captured already by the coarse mesh. However, the capability of the finer mesh to model more high frequency content – important in wave propagation – appears to be extremely important. A closer view on this partial issue by a Fourier decomposition is depicted in Figure 10. It shows that lower frequencies can be simulated correctly even with the coarse mesh, but higher frequencies require finer meshes, even beyond the currently used refined mesh. The experimental results show also that there is a fairly high energy content between 500 and 1000 Hz which must be represented by the simulation model. This is a fairly surprising result, as studies with a ball impacting a plate had shown very reasonable correspondence with experimental observations.

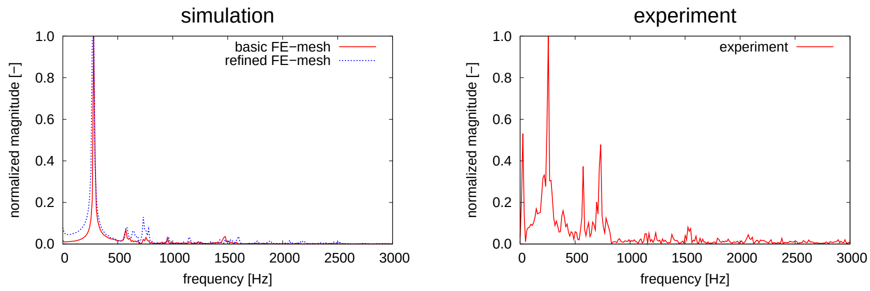


Figure 10: Frequency amplitude spectra of displacements - simulation and experiment

3 Conclusions

The simulation results presented in section 2 show only a part of the project work on the *HP-XC6000 Cluster*. The goal of the project was to generate knowledge about the proper FE-simulation of complex structures by modifying several relevant model parameters. In another part of the project, numerical examples with rather small numbers of elements are computed, to gain general information about wave propagation simulation with finite elements. Some of the performed modifications do not advance the results at all, compared to the experimental results, but general rules concerning the usage of different tools and procedures of the Finite Element Method can be obtained from these simulations. However the mesh refinement has shown that the computation of very large models on parallel computers is absolutely necessary. Though in many investigations it is not required, to aim at an absolute realistic simulation of the experimental results, because the numerical effort is much higher than the obtained advantage, it has proven that a constantly refined mesh is of vital importance. As described in section 2.2.4, it may be often possible to simulate e.g. lower frequencies with rather coarse meshes. In order to realize the presented parametrical studies in an appropriate time, they were all carried out first with the rather coarse mesh. As the goal of the investigations was, to find out the general influence of the discussed parameters, for which the used discretization was sufficient.

The project is continued and computations on other complex structures with rather fine meshing will follow, which presumes the availability of high performance parallel computers, such as the *HP-XC6000 Cluster*. In further investigations, also the application of damping on the refined mesh from Section 2.2.4 with the experience from Section 2.2.2 will be analyzed. Other interesting issues are a) the influence of the boundary conditions on the experimental results and how accurate these can be captured in the simulation, and b) how the simulation reacts on changes in the joining achieved with a contact formulation used for the spotwelds. It is important to investigate all these parameters separately, which requires again many parametrical studies with high numerical effort. The final achievement of the study will be to develop modeling rules for wave propagation simulation in complex structures such as automotives under impact as found in crashworthiness events.

References

[1] J. D. Achenbach. *Wave propagation in elastic solids.* North-Holland, 1973.

[2] T. Belytschko and L. P. Bindemann. Assumed strain stabilization of the eight node hexahedral element. *Computer Methods in Applied Mechanical Engineering*, 105:225–260, 1993.

[3] T. Belytschko and C. S. Tsay. A stabilization procedure for the quardilateral plate element with one-point quadrature. *International Journal for Numerical Methods in Engineering*, 19:405–419, 1983.

[4] Ted Belytschko, Wing Kam Liu, and Brian Moran. *Nonlinear finite elements for continua and structures.* Wiley, 2000.

[5] K. F. Graff. *Wave motion in elastic solids.* Ohio State Univ. Press, 1975.

[6] J.O. Hallquist. *LS-DYNA Theoretical Manual.* Livermore Software Technology Corporation, 1991-1998.

[7] J.O. Hallquist. *LS-DYNA Keyword User's Manual.* Livermore Software Technology Corporation, 1992-2005.