Explicit and Implicit FE Simulations of Material Tests for Subsequent Durability Analyses

Philipp Thumann[1], Marcus Wagner[1], Bertram Suck[2], Josef Meinhardt[2], Steffen Marburg[3]

[1]Ostbayerische Technische Hochschule Regensburg, Laboratory for Finite Element Analysis and Structural Dynamics, Galgenberstraße 30, 93053 Regensburg, Germany

[2]BMW Group, Production System, Technical Planning, Tool Shop, Plant Construction - Standards, Innovations, Knorrstr. 147, 80788 München, Germany

[3]Technical University of Munich, Department Mechanical Engineering, Chair of Vibroacoustics of Vehicles and Machines

1 Abstract

Because of increased stroke rates the loads on forming tools increase too. To ensure a save design of components, durability analyses are intended. For this, simulation results from FE analyses are necessary. Therefore, it is desirable to use elements with quadratic function, because of a good stress approximation.
The goal of the described investigation is to show if calculation results created by LS-DYNA [1] can be used for durability analyses. Especially the use of quadratic elements is investigated. For the evaluation, on the one hand explicit FE analyses of a special durability test are carried out. These analyses are validated by available test data. To create results for later durability analyses further FE analyses with implicit time discretisation are carried out. In this paper results of the investigations are presented and evaluated critically.

2 Introduction

To improve the efficiency in sheet metal manufacturing, more and more press systems with increased stroke rates are in use. Due to the increased stroke rates, the structural-dynamic loads increase too. Thereby, not only the press systems are highly loaded, but also the tools for manufacturing the blanks. This has to be accounted for in the early design phase of the tools in the virtual process chain.
Because of safety requirements, high safety factors are used during the design of the tools, leading to high manufacturing costs. Furthermore, stroke rates are limited due to the movement of high masses of the tools, reducing the productivity of the press system.

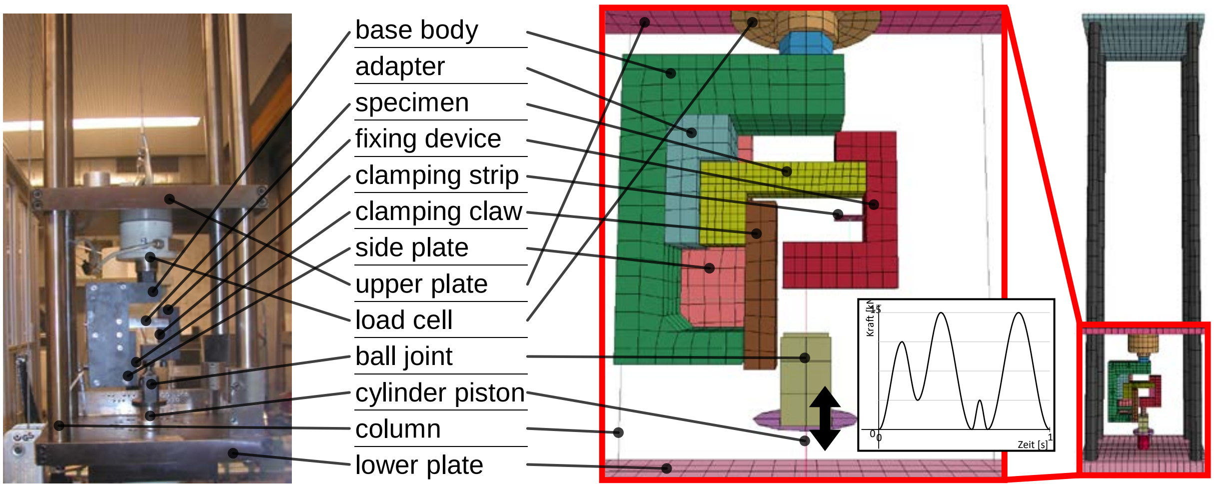

Fig.1: Test facility in reality (left) and discretized for the FE simulation (left).

To reach smaller allowable safety factors, resulting eventually in fewer costs, and to improve the stroke-limitation due to the moved masses, a simulation methodology for the evaluation of the durability of selected tool components is developed. Therefore, special material tests were designed and carried out. The tests have to be investigated with FE simulations for subsequent durability analyses. Therefore, the test facility was modelled in ANSA [2] for the LS-DYNA [2] solver with explicit time discretization, see section 3. The explicit FE simulation of the whole test facility delivers boundary conditions for a subsequent substructure model with implicit time discretisation, described in section 4. In section 5 additional investigations according to the available solid elements in LS-DYNA with the corresponding results are mentioned. In section 6 conclusions and an outlook are presented.

3 Simulation of test facility

First a calculation of the test facility with explicit time discretisation is carried out. The explicit solver is used because stress values are not of interest. For the simulation of the test facility only the displacements are important, because for the subsequent simulation of the substructure model only the node displacements are as boundary conditions. Therefore, only the most important components of the test equipment are modelled in the pre-processor ANSA. Fig. 1 shows the real test facility on the left side. On the right side, the modelled test facility is illustrated. For a better overview, the side plate is invisible in the FE discretized test device. The most important component is, of course, the test specimen, which is fastened by bolts and the clamping claw in the base body. The base body is connected to the load cell, which connects to the hydropulse system. Also the ball joint belongs to the hydropulser and is connected to the movable cylinder piston. The cylinder piston is loaded by a force over the time function, as shown in the diagram in Fig. 1 on the right side.

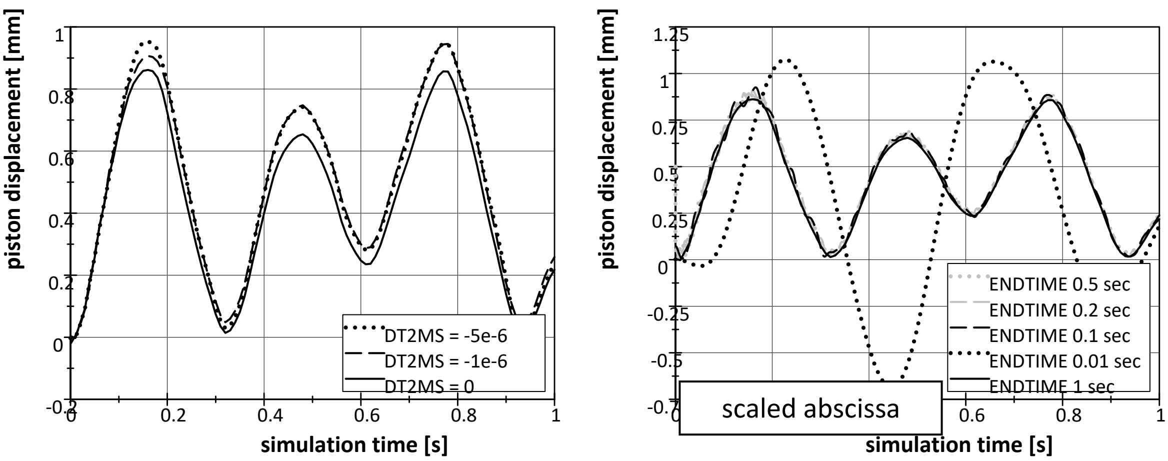

Fig.2: FE results from carried out investigations.

To save CPU time the following investigations of several influence factors were carried out.

- Mass scaling
- Simulation time
- System damping

In Fig. 2 the results of the investigations are shown in diagrams. In the diagram on the left side the influence of mass scaling is shown. The parameter DT2MS in **`*CONTROL_TIMESTEP`** was set from zero to -5e-6. With this setup the displacement progression of the cylinder piston over the time is hardly influenced. Smaller setups for DT2MS, e.g. -1e-6 also lead to an influence of the piston displacement. Hence, the mass scaling is set to zero.

However, to save CPU time the simulation time was reduced according to the legend in Fig. 2 in the right diagram. Fig. 2 shows these simulation results on the right side (values for abscissa are scaled for a better comparison). As can be seen a vibration builds up if the simulation time is set to 0.01 sec. By raising the time to 0.1 or higher there occurs no building up of vibrations. But now there are small oscillations. It seems that a resonance frequency of the system is hit. To avoid the oscillations a synthetic damping is added to the system by using **`*DAMPING_GLOBAL`**. A curve with a constant value according to the hit resonance frequency of 50 Hz is defined, see the diagram in Fig. 3 on the left side. This operation leads to improved results by shorter CPU time, see the diagram in Fig.3 on the right side.

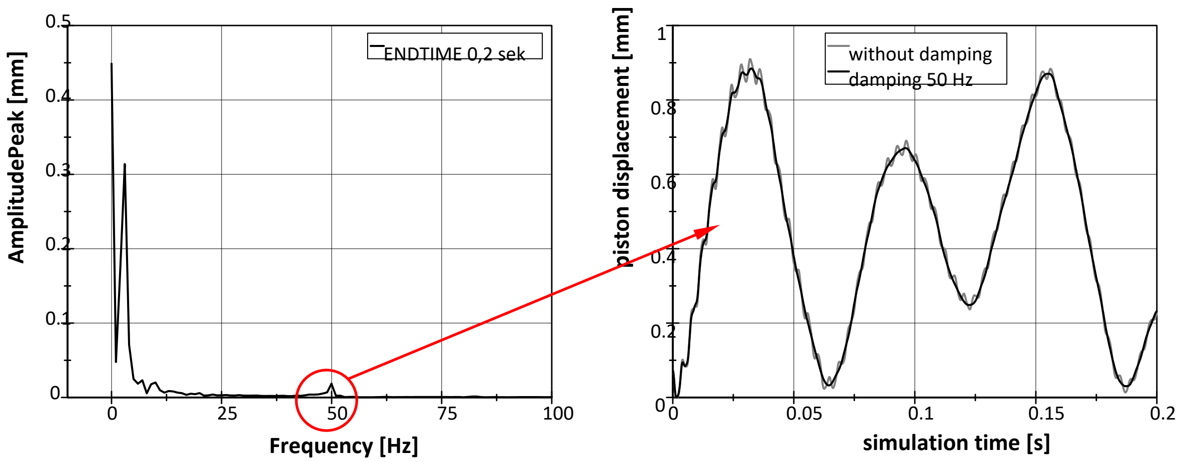

Fig.3: FE results from carried out investigation.

To save further CPU time some volumetric parts are discretized with beam and shell elements, e.g. the columns of the hydropulser or the different bolts for locking the specimen in the fastening device, see the enlargement in the middle of Fig. 1.
To be sure that the simulation of the test facility delivers reasonable calculation results a verification is done. Therefore, the displacement of the cylinder piston from the test measurements are compared to the one from the calculation. Fig. 4 shows this comparison in the diagram on the left side. As can be seen there is nearly no deviation between the simulation and the reality.
For the subsequent simulation of the substructure model, the interfaces for the node displacements, used as boundary conditions in the substructure model simulation, have to be defined. The output is defined by using **`*INTERFACE_COMPONENT_SEGMENT`** where only a **`*SET_SEGMENT`** has to be defined in the area of the cross section. To write out a file with the node displacements the keyword **`*INTERFACE_COMPONENT_FILE`** is used. The keywords have to be put in two times for the two cutting planes for the boundary conditions, see Fig. 4 on the left side. To check if the modelling with the single parts of the specimen, connected by **`*CONTACT_TIED_SURFACE_TO_SURFACE`**, a further verification with the measured and calculated piston displacement is done, see the diagram in Fig. 4 on the right side. There is no influence.

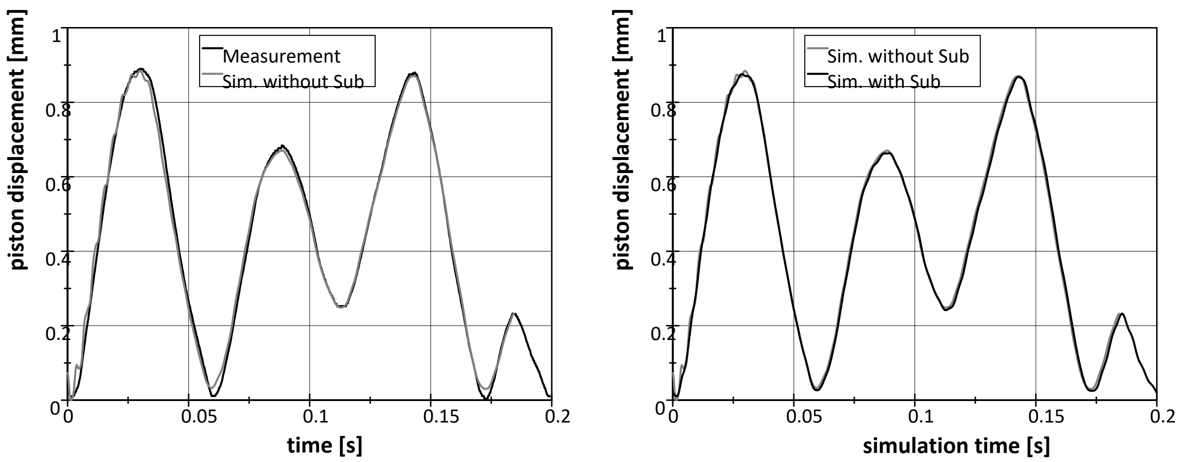

Fig.4: Comparison between measurements during tests and FE results of the piston displacement for verification (left) and comparison between FE results from simulations without and with inclusions for substructure modelling (right).

4 Simulation of the substructure model

As mentioned before the outputs from the explicit simulation of the test facility are used as boundary conditions in an implicit simulation of the substructure model. The implicit solver has the advantage that quadratic elements can be used. Generally the quadratic elements deliver better stress values,

which are necessary for subsequent durability analyses. Furthermore, to activate the quadratic element formulation the parameter ELFORM has to be set to 23 for hexahedrons or 17 for tetrahedrons in ***SECTION_SOLID**.
The displacements of nodes in the cross section areas will be applied to the cutting planes of the substructure, see Fig. 5. The cross section areas are defined with ***INTERFACE_LINKING_SEGMENT**. Once again the ***SET_SEGMENT** has to be indicated. For the import the keyword ***INTERFACE_LINKING_FILE** can be applied.

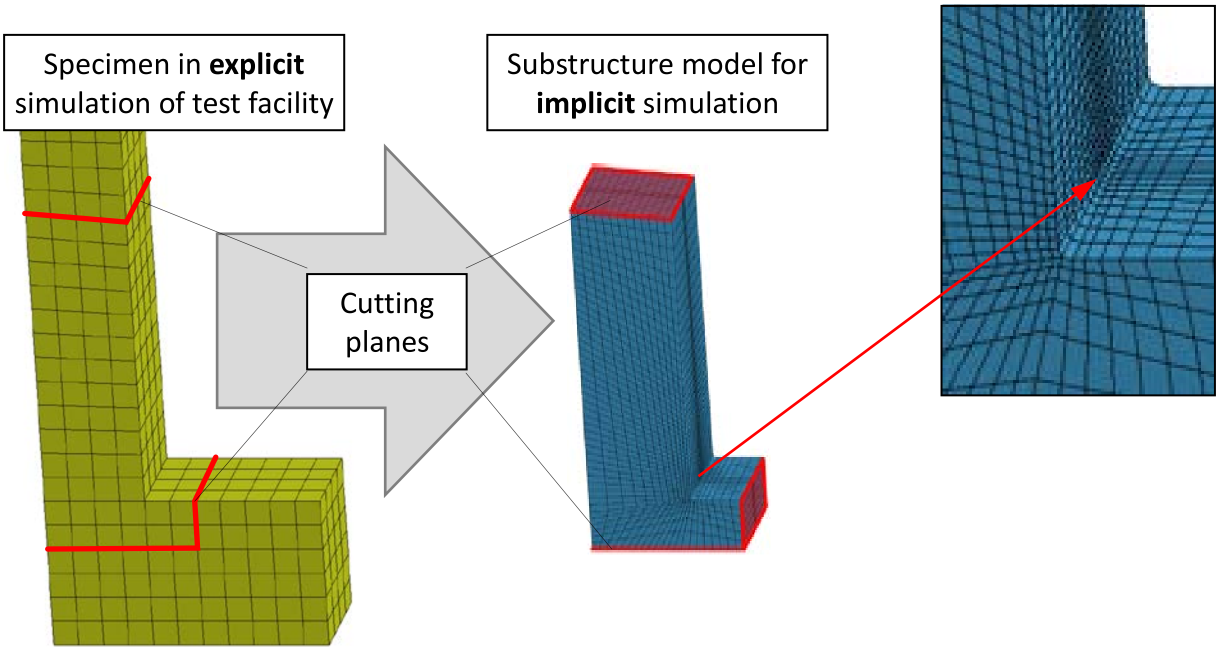

Fig.5: FE model of the specimen for simulation of the test facility (left) and the substructure (right).

The big advantage of using substructure modelling is the opportunity to create different meshes in the substructure without calculation of the whole test facility every time. That saves a lot of CPU time. The variation of the mesh is described in chapter 6. An example of mesh refinement is shown in Fig. 5. The mesh size is much smaller compared to the simulation of the whole test facility explained in chapter 3.
A further advantage is the increase of the discretisation accuracy. As can be seen in the enlargement in Fig. 5 the radius at the notch is included. The simulation of the whole test facility does not contain this radius, because there is no influence to the displacement at the cutting planes.

5 Element investigations

To get the best possible mesh for later durability analyses element investigations were carried out. At first there is an investigation according to the output of elements from ANSA to LS-DYNA. A simple geometry was created, for the purpose of investigation. The discretised geometry is shown in Fig. 6 on the left side. Also the boundary conditions are illustrated.
For durability analyses, volumetric elements with middle nodes (quadratic elements) are most suitable. So it is the goal to use quadratic elements. Hence, this type of elements should be read out from ANSA. In the case of tetrahedrons it is possible, but not in the case of hexahedrons. ANSA enables the output of amongst others LS-DYNA input-deck files. If this opportunity is chosen a k-file will be created. But the middle nodes will not be written out. That means that the quadratic elements are backspaced to linear elements. And the advantage for the later durability analyses is no longer given. If the output type is changed from LS-DYNA to NASTRAN the middle nodes can be written out. But in this case the order of middle nodes is not the right one for the element definition. Hence, the single middle nodes have to be allocated to the quadratic elements manually. For the simple geometry, which is used in this study, it is possible, but not in the case of the specimen (or other complex forms). That’s why a MATLAB [3] code was written to bring the nodes in the right order in the k-file. If elements with middle nodes are used the element formulation ELFORM has to be changed in ***SECTION_SOLID** to 23 for hexahedrons or 17 for tetrahedrons.

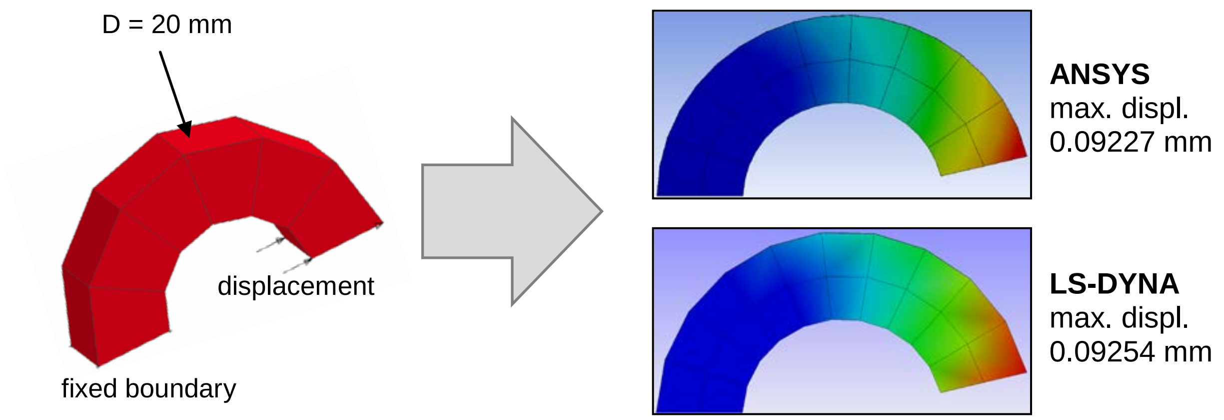

Fig.6: Modelling of simple discretized geometry for element investigations (left) and FE results by using quadratic hexahedrons.

On the right side Fig. 6 shows the simulation results from LS-DYNA compared to a reference simulation carried out with ANSYS [4]. As shown in the two plots on the right side the node displacements are nearly the same (extreme high scale factor for enhanced presentation). However, the stress values from the LS-DYNA simulation converge very slowly compared to the ANSYS results by the use of hexahedron elements, see Tab. 1 (decreasing El. Size). Hence, the stress outputs from calculations with quadratic element formulations by using LS-DYNA are not recommended for a subsequent durability analyses because of high CPU time. In Tab.1 the highlighted cells shows the converged calculations. In this investigation the definition of convergence was defined as following: If the maximum stress is the same or lies below the value from the calculation before, then convergence is reached.
Results from calculations with linear elements (without midnode, see Tab. 1) in generally are not preferable for subsequent durability analyses because of linear approximation of the geometry. That leads to notches at every node on the surface due to the discontinuous intersection from element to element.

Table 1: Investigation with respect to the mesh size and the calculated stresses by variation the element formulation ELFORM with highlighted cells for converged calculations.

Implicit, displacement x=0.05mm									
ANSYS		**LS-DYNA**							
Hexahedron						Tetrahedron			
with midnode		without midnode		with midnode		without midnode		with midnode	
quadr. ELFORM		ELFORM 1		ELFORM 23		ELFORM 13		ELFORM 17	
El.size	Stress v.M.	El.size	Stress v.M.	El.size	Stress v.M.	El.size	Stress v.M.	El.size	Stress v.M.
[mm]	[Mpa]	[mm]	[Mpa]	[mm]	[Mpa]	[mm]	[Mpa]	[mm]	[Mpa]
8	305							5	318
4	393			2,5	315			2,5	375
2	387	1,25	259	1,25	358	1,25	401	1,25	392
1	398	0,625	324	0,625	382	0,625	410	0,625	399
0,5	404	0,417	349	0,417	390	0,417	413	0,417	403
0,25	406	0,278	367	0,278	396	0,185	408	0,278	404
0,125	407	0,125	388	0,185	400			0,185	405
0,1	407	0,083	395						

Fig. 7 shows the comparison between LS-DYNA and ANSYS results in diagrams with the maximum stress over the number of nodes. As can be seen by using the FE solver from ANSY with quadratic elements a steady state subjected to the maximum stress is reached at a fraction of number of nodes compared to LS-DYNA. Especially in the case of hexahedron elements, which should be preferred. That makes the use of LS-DYNA for durability analyses doubtful with respect to the CPU time.

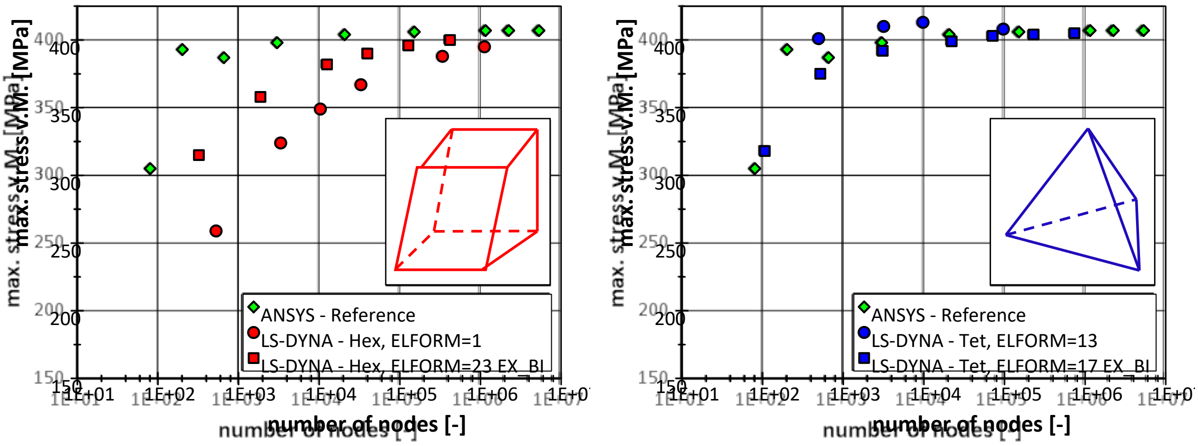

Fig.7: FE results from simulations of simple discretised geometry with hexahedrons (left) and tetrahedrons (right) by varying the mesh size.

6 Conclusions and Outlook

The results from the simulation of the test facility by using explicit time discretization are very successful. A verification was presented by a comparison of the piston displacement from measurements and from FE simulations.

The interaction between ANSA for preprocessing and LS-DYNA for FE solving is not advised in case of using quadratic hexahedron elements. If a k-file is created with ANSA the middle nodes will not be written out. The middle nodes can be written out by using a NASTRAN output. NASTRAN uses another node order for 20 node hexahedron elements. So the middle nodes have to be brought into the right order manually or with a self-made routine, e.g. with MATLAB.

Because of no possibility to create a proper hexahedron mesh at current the only publicly available and usable quadratic elements are the tetrahedrons. But there is still a disadvantage of the mesh convergence with respect to the maximum stresses in the notch compared to other FE tools. E.g. ANSYS needs only a fraction of quadratic elements to reach convergence.

7 Literature

[1] LSTC Inc. LS-DYNA *(a general-purpose finite element program)*.
[2] BETA CAE Systems. ANSA *(CAE pre-processor software)*.
[3] The MathWorks, Inc. MATLAB *(multi-paradigm numerical computing environment)*.
[4] CADFEM GmbH. ANSYS Workbench *(a general-purpose finite element program)*.