

Concepts to Take Elastic Tool Deformations in Sheet Metal Forming into Account

A. Haufe^{*}, D. Lorenz^{*}, K. Roll^{**}, P. Bogon^{**}

^{*}DYNAMore GmbH, Industriestrasse 2, 70565 Stuttgart, Germany
{andre.haufe; david.lorenz}@dynamore.de

^{**}Daimler AG, PK-PWT, 71059 Sindelfingen, Germany
{karl.roll, peter.bogon}@daimler.com

Abstract

In recent years the development of more and more niche products, i.e. cars with different external appearance, has become a remarkable trend in the automotive industry. This trend, however, generates higher costs for individual tooling geometries that are traditionally made as stiff as possible. A second trend is the increasing use of high and ultrahigh strength steel grades for bodies in white. Here too the design philosophy for the tools in sheet metal forming is based on a rather rigid and stiff tool approach. It is clear though, that a tremendous amount of money could be saved by designing the tools such, that their elastic deformation during the forming process is taken into account. This would lead to lighter and hence more inexpensive tools.

The traditional approach to design the tool geometry by finite element simulations with rigid tools. Clearly, if elastic deformations are to be accounted for in such models, the assumption of a rigid tooling geometry needs to be abandoned. Here the straight forward approach would be to discretize the tool by a sufficiently accurate full 3D finite element model. Additionally the machine stiffness may be added to the model for completeness. Obviously this will lead to prohibitively increased computing time especially for large parts. A simple yet effective way to take the elastic deformations nonetheless into account is to condensate the discretized machine and tool geometry once and reuse it in subsequent simulations runs. The paper will discuss recent features in LS-DYNA[®] that allow the static condensation of elastic tool and machine geometries. Furthermore the application of the “deformable rigid bodies”-approach is shortly discussed.

1. Introduction

In recent years the development of more and more niche products, i.e. cars with different external appearance, has become a remarkable trend in the automotive industry. As another trend the increasing use of high and ultrahigh strength steel grades for bodies in white should be mentioned. Both trends generate higher costs for individual tools that are traditionally made as stiff as possible. It is clear though, that a tremendous amount of money could be saved by designing the tools such, that their elastic deformation during the forming process is taken into account. Subsequently the tool geometry may then be designed to compensate the elastic deformations. This would lead to lighter and hence more inexpensive tools.

In the present paper some of the possible approaches to take elastic response of tools in finite element sheet metal forming simulations into account are discussed, namely so called deformable rigid bodies, elastic super-elements and full 3D discretization of respective geometries. Furthermore the application of a coarser underlying tool mesh might be a suitable way to save labor time during simulation setup and also computing time during the solution phase. However, special attention must be put on the technique to couple the coarse (solid) tool mesh with the finer discretized (surface) tool mesh.

2. Elastic Response of Forming Machine

Reducing the stiffness of the forming tools adds also new challenges to the modeled forming system: Now also elastic deformations of the press equipment, i.e. the stiffness of the frame and table, need to be taken into account. One possible approach is of course to model the whole press equipment by finite elements to study the elastic response (see Figure 1, left to middle). It must be taken into account however, that the modeling effort may be very high and usual engineering assumptions, might already influence the results very much. Hence, it might be much easier to do a limited number of real world stiffness measurements of the machine that can in turn be used as boundary conditions, i.e. via elastic translational and rotational springs (see Figure 1, right). Hereby worn out connections and unwanted clearance in the press equipment is automatically included in a subsequently defined finite element model.

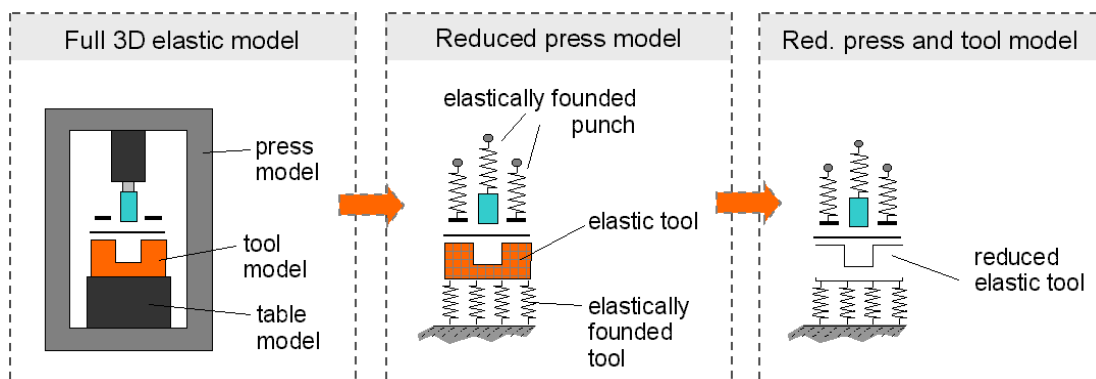


Figure 1: Reduction of model size

The next step to reduce the model size further is to reduce the degrees of freedom (*dof*) of the 3D modeled tools, i.e. die, punch and binder (see Figure 1, middle to right). Details how this could be done will be discussed in the next section. It should be emphasized though, that a modular setup for such elastic forming simulations can be easily achieved. Standard elastic stiffness parameters for each press combined with standard discretization techniques for the tools help to simplify the rather complex setup.

3. Methods to take Elastic Tool Deformations into Account

As mention in the preceding paragraph some of the methods readily available in LS-DYNA[®] to take elastic tool deformations into account shall be discussed in the following.

3.1 Deformable Rigid Bodies

Modal methods are classical techniques in linear dynamic analysis to compute the response of elastic structures. In general a problem dependent reduced number of eigenmodes needs to be found. This basis of eigenmodes does not necessarily include all eigenmodes. Hence all possible (or available) modes may be separated in rather useful and important ones on the one side and rather unimportant ones on the other side. The subsequent linear superposition of the basic modes leads to the desired final deformation. Clearly, linear superposition is limited to small strain applications. LS-DYNA offers the possibility to combine shape functions (i.e. eigenmodes) from a previously computed model basis with deformable parts that are modeled

with finite elements. This approach allows to consider at least the major elastic deformations of a body with fairly little effort. Of course, if the basis needs to be larger, i.e. the expected deformations are more complex, then more effort has to be taken to identify, compute and store the respective modes. Obviously there is a break even point when the effort does not pay off as fast as expected.

The implementation of the method into LS-DYNA has been described in [2]. In addition, an interesting and comprehensive discussion about the application of rigid deformable bodies to head impact simulations was demonstrated in [1]. The method is therefore not covered in detail in the present paper.

3.2 Condensation of Tool Matrices

Another rather classical method to reduce the number of degrees of freedom in finite element simulations is to reduce internal *dofs* by static condensation. This approach is also known by the name of super-elements and has been investigated by Roll & Hoffmann [4,5]. Clearly, this method is again only applicable to linear and small strain problems and hence may be applied to tools in sheet metal forming simulations. The procedure to reduce unnecessary internal *dofs* from a stiffness matrix is well known to the reader from early university courses about statics and mechanics of structures but the approach will be presented here for convenience nevertheless:

In classical sheet metal forming simulations one is only interested in the surface geometry of the tools. Therefore it is a common approach to discretize the effective surface only. In order to compute the elastic response of the tool it must be modeled in a fully 3D manner where for convenience tetrahedron shaped finite elements are usually applied.

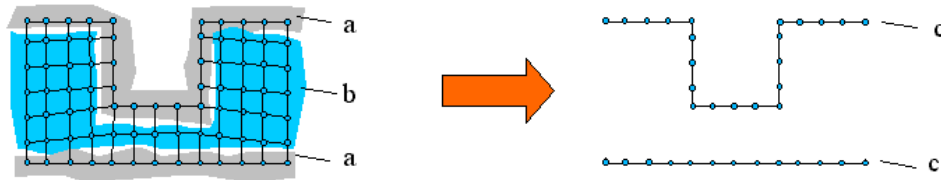


Figure 2: Condensation of tool surface; *b*-region

Since only the *dofs* at the upper and lower surface (compare Figure 2, region denoted with *a*) are of interest and consequently all internal *dofs* (compare Figure 2, region denoted with *b*) may be eliminated. The remaining nodal stiffnesses are afterwards bound to the nodes denoted with *c* in Figure 2. In classical static condensation the stiffness matrix is split into four different regions; denoted in the following with K_{aa} , K_{ab} , K_{ba} and K_{bb} . Here the subscripts refer to the nomenclature introduced in Figure 3.

$$\begin{bmatrix} K_{aa} & K_{ab} \\ K_{ba} & K_{bb} \end{bmatrix} \begin{bmatrix} u_a \\ u_b \end{bmatrix} = \begin{bmatrix} F_a \\ F_b \end{bmatrix} \quad (1)$$

Here u_a and u_b denote the displacements associated with the respective degrees of freedom and F_a and F_b represent the corresponding external forces. After some maths the following expression may be gained:

$$\underbrace{\left(K_{aa} - K_{ab} K_{bb}^{-1} K_{ba} \right)}_{K_{cc}} u_a = F_a - K_{ab} K_{bb}^{-1} F_b \quad (2)$$

Here K_{cc} represents the newly gained condensed stiffness matrix. Also, it should be mentioned that in the present application no external forces are expected to act on internal nodes, i.e. on the node set denoted with b . Hence $F_b = \{0\}$ and equation (2) reduces to $K_{cc} u_a = F_a$.

A similar approach may be taken for the diagonally shaped mass matrix and also for the damping matrix. In sheet metal forming applications a useful procedure seems to be to reduce the *dofs* of the tools to surfaces that are in contact with other parts.

In LS-DYNA 971 R4-beta [3] the procedure to condensate unnecessary *dofs* from a part may be launched in future by the new keyword *CONTROL_IMPLICIT_STATIC_CONDENSATION. Here the target part (or part set), a set of nodes with appropriate *dof*-attributes that define the remaining nodes and *dofs* needs to be given as well as the name of the stiffness and mass matrices. Furthermore, since the condensated matrices are written to the hard disk a filename must be assigned. The file may be written in standard NASTRAN-ASCII format or alternatively in binary format.

In the subsequent forming simulation the file containing the condensated matrices is imported by the keyword *ELEMENT_DIRECT_MATRIX_INPUT. This feature is available in LS-DYNA for several years now and has been used extensively also for other applications. One of the drawbacks of the super-element approach is the file size and therefore the uneasy handling of condensated matrices within the file system. Firstly it should be noted that a condensated matrix does not have any zero (i.e. blank) entries. Despite being symmetric one still has to save $\frac{1}{2} * n * (n - 1)$ real values to the hard disk. Here n is the number of remaining *dofs* (i.e. *dofs* of the a region). Even for small test problems this generates an enormous amount of data that easily sums up to a couple of Gbytes.

3.3 Full 3D Discretization

Finally one should also mention that a simple and brute force approach to model elastic tools is by just adding an enormous amount of finite elements. If the underlying solver is capable of handling a large amount of elements effectively this approach is definitely worth investigating. In the case of explicit forming simulations with LS-DYNA where an effective multi-processor strategy (MPP) on powerful computers may also be applied this idea seems to be very promising, too.

4. Coupling of Different Tool Mesh Sizes

When investigating different methods to model the elastic response of forming tools one soon finds out that a fine resolution of the tool deformations is probably not necessary. On the other hand however, the fine surface details of commonly used rigid tool surfaces demand a rather fine mesh close to the surface for the 3D modeled tool. This discrepancy could be solved if some means of interpolation between the fine surface mesh and the rather coarse discretization delivered by a possible 3D tool mesh can be applied (see Figure 3). Here the necessity of interpolation between the fine surface mesh and the coarse solid tool mesh is pictured.

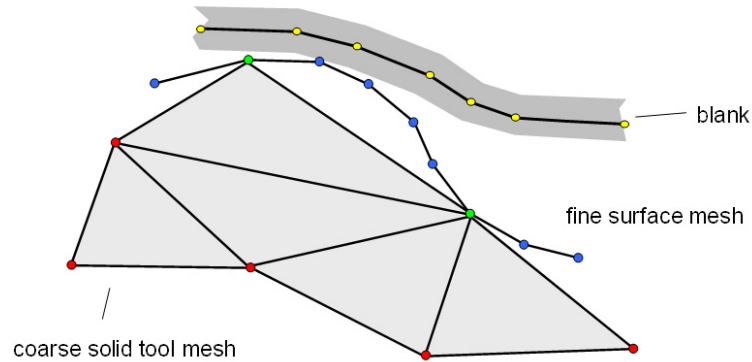


Figure 3: Coupling of different tool meshes

In principle one may think of rather complex methods to interpolate between these two discretizations. Besides some kind of loose displacement coupling (i.e. not in every explicit cycle) and complicated mortar formulations, a very simple, effective and probably practical method could be to apply the tied-offset-contact definition already available in LS-DYNA. This approach will fix nodes from the fine surface mesh to corresponding segments (i.e. elements) of the coarse 3D tool mesh. Clearly, this will lead to node patches that will act as rigid facets on the deformable tool surface. Further investigation of such non-matching meshes will show if this approach is admissible in terms of the sought improved accuracy of deformable tool discretization.

5. Examples

In the next paragraph three examples where the aforementioned approaches were applied will be presented. Clearly, the present approach is still under development. Some features are usable within a beta-version of LS-DYNA but might still need some refinement. Also, the usability may need some improvement.

5.1 U-Shaped Rail

This first example uses the geometry of the NUMISHEET Benchmark of 1993 but the die has been discretized with hexahedron finite elements. These solid elements were condensed to their respective effective surfaces on the top and on the bottom. In order to be able to visualize the tool and also to be able to define contacts with the super-element, a layer of dummy shells has been put atop of the outermost nodes of the effective surface (see Figure 4a).

Comparisons were done of the solid and the super-element simulation (see Figure 4b). And it was found that the results in terms of displacements of the elastic die were very much the same. Also the nodal displacements scaled by a factor of 100 are depicted in Figure 5. Here the red coloured elements belong to the full 3D simulation while the blue coloured elements represent the dummy shell atop of the super-element.

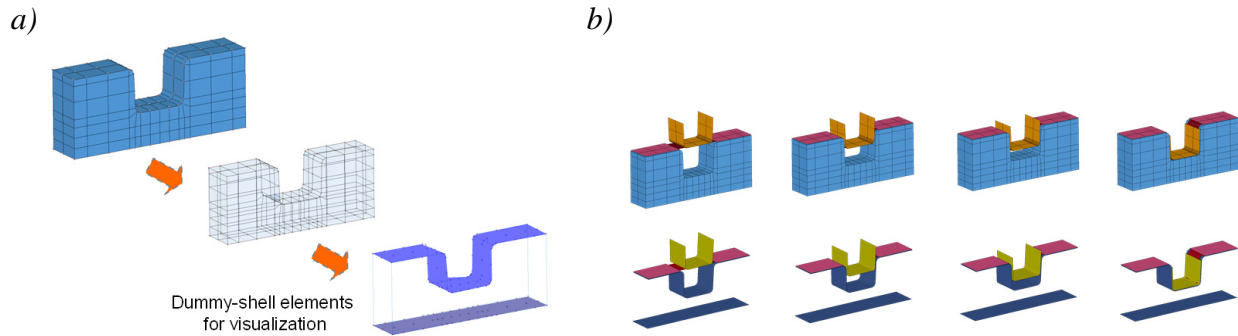


Figure 4: a) Dummy shells to visualize the effective surface of the super element
 b) Comparison of 3D and condensed simulation

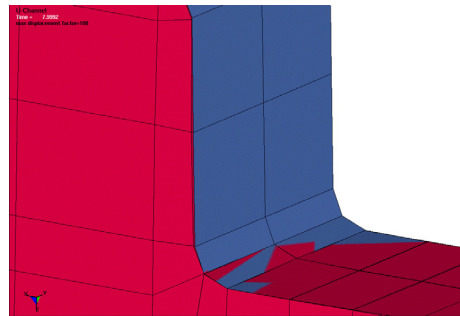


Figure 5: Displacements of 3D elements and condensed super-element scaled by a factor of 100

It can be seen that the deviations along the vertical axis are very small, therefore the “simplification” done through the application of the super-element seems to be admissible in this case. Clearly, small deviations were expected since the standard stabilisation methods of explicit finite element simulations, i.e. hourglass-control, are not applicable to the super-element. In this example the condensed stiffness and mass matrices used up 8,7 Mbytes of disk space in standard NASTRAN format. But the matrices had only 342 *dofs* in this case.

5.2 S-Rail

A very similar numerical experiment was done with a modified s-rail geometry (see Figure 6a). The binder was supported by a set of barrels (pins) that were given different reaction forces as depicted in Figure 6b. Here the binder was discretized with hexahedron solid elements while the die was kept rigid.

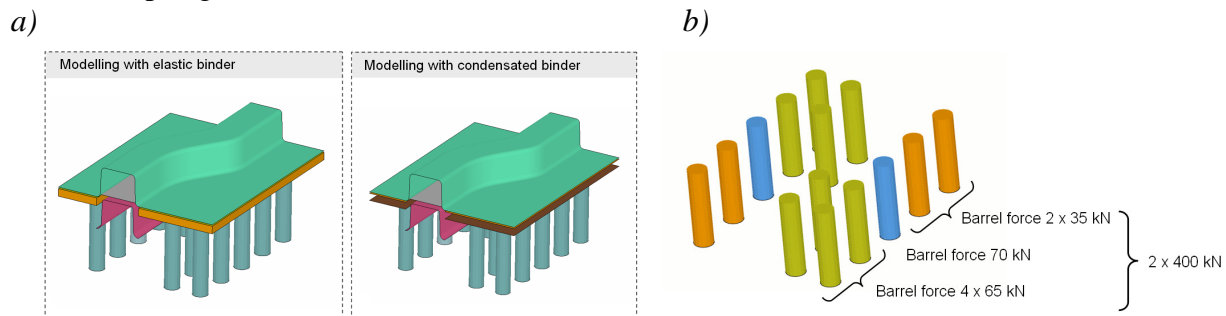


Figure 6: a) 3D binder vs. condensed binder
 b) Forces applied by pins

In Figure 7a the effect of a rigid binder vs. the 3D modeled elastic binder is depicted. It can be seen that the effect of the elastically supported binder as well as binder deformations itself lead - as expected - to a different drawing-in contour. Of course all other data to set up the simulation was kept constant.

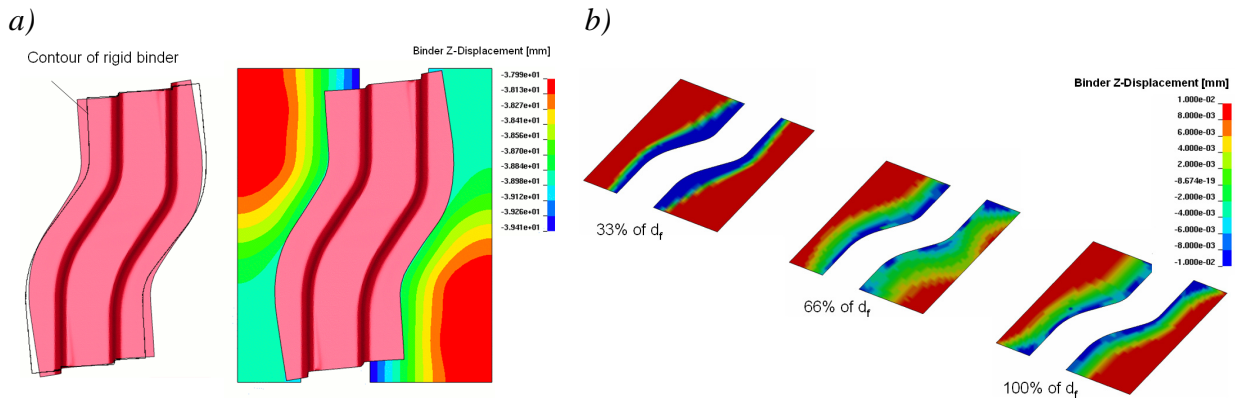


Figure 7: a) Drawing-in compared between rigid and elastic 3D modeled binder
b) : z-displacement at different stages of condensed binder simulation

In the next step the 3D binder was condensed as described before. Also dummy shells for visualization reasons were put atop of the binder surface. In Figure 7b the z-displacement of the condensed binder surface is depicted in three different stages of final drawing distance d_f . In this example the binder consists of a symmetrical left and a right part. For one part 4050 hexahedron elements consisting of 5280 nodes were used. The final effective binder surface had 880 nodes at the top and 880 nodes at the bottom. Hence the NASTRAN ASCII-format of the super-element file used up 291 Mbytes on the hard disk while the proprietary binary LS-DYNA format only needed 71 MBytes. It should be also mentioned, that only *dofs* in z-direction were allowed. Therefore the size of the matrices could be minimized compared to a full set of *dofs* at the nodes, too.

5.3 U-Shape Rail with Non-Matching Meshes

As discussed in section 4 one of the main problems of large model sizes may be avoided if some means of coupling between coarse solid and fine surface shell mesh would be available. One way to solve this problem seemed to be a special contact option available in LS-DYNA. Therefore the aforementioned u-shaped rail was used again with a very coarse 3D discretization atop of which a fine shell discretization was used to model the effective surface (see Figure 8).

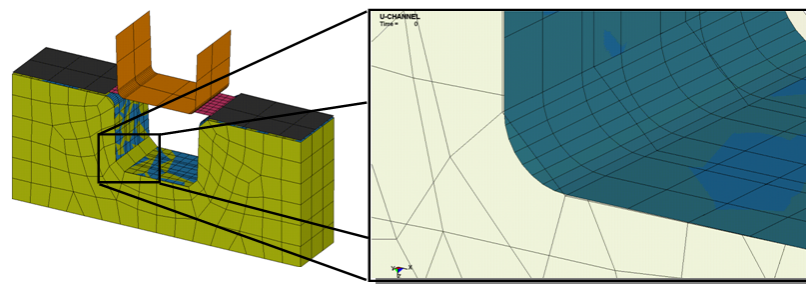


Figure 8: Non matching meshes for solid die and effective surface shell

The applied tied-offset-contact option worked as expected and similar results comparable to the first examples were generated. But on the downside one must note, that the model setup needs to be done with great care. Especially the accuracy of the generated meshes is of importance due to the fact that the contact algorithm depends on a properly defined location to correctly tie the nodes.

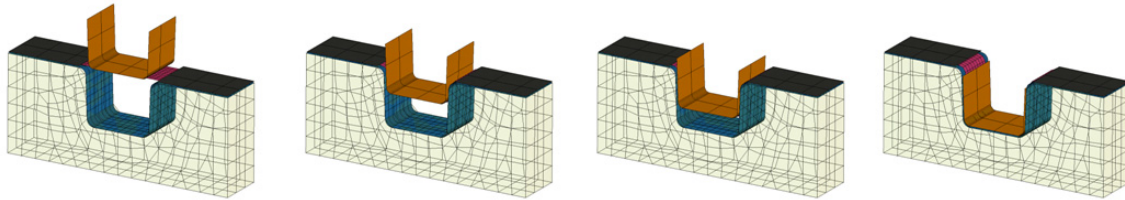


Figure 9: Drawing of the u-shaped rail with non matching meshes of the elastic die

6. Conclusions

A short overview on existing methods to model elastic tools for sheet metal forming was given. Among these methods the method of static condensation (i.e. super-elements) was chosen and implemented into LS-DYNA. The exchange of stiffness and mass matrices may now be done in ASCII or binary format. First tests lead to the expected conforming results, however, the shear size of the files may limit its application in future.

As a possible solution coarser meshes for elastic tools coupled by a tied-contact-algorithm to the effective tool surface was investigated. Here the results were very promising. This method may have a large potential – combined with regular 3D meshes, condensed super-elements or even deformable rigid bodies. Future work will focus on larger models and hence sound recommendations as to which method may be superior can be given.

Acknowledgement

The present work is partially supported by the German Federal Ministry of Education and Research within the cooperative project "Innovative Methoden zur Auslegung von Umformwerkzeugen im Fahrzeugbau (IMAUF)" by grant #02PU2006. Furthermore the authors express their gratitude to Roger Grimes of Livermore Software Technology Corporation, CA, for his valuable support.

References

- [1] Bitzenbauer J., Franz U., Schweizerhof K.: *Deformable Rigid Bodies in LS-DYNA with Applications – Merits and Limits*. Proceedings 5th European LS-DYNA User Conference, Birmingham, May 2005.
- [2] Maker B., Benson D.: *Modal Methods for Transient Dynamics Analysis in LS-DYNA*, Livermore Software Technology Corporation.
- [3] *LS-DYNA Keyword User's Manual*, Version 971 Volume I+II, May 2007, Livermore Software Technology Corporation, Livermore, CA, ISBN 0-9778540-2-7.
- [4] Hoffmann J.: *Berücksichtigung der elastischen Werkzeugeigenschaften in der Blechumformsimulation*, Diploma-Thesis, FH Lausitz in Cooperation with DaimlerChrysler AG, 2005.
- [5] Roll K., Hoffmann, J. *Eine Möglichkeit zur Berücksichtigung der elastischen Werkzeugeigenschaften bei der Blechumformung*, Proceedings 4th LS-DYNA Anwenderforum, Bamberg, Germany, October 2005.