# More Realistic Virtual Prototypes by means of Process Chain Optimisation

**Authors:**

Peter Gantner*, David K. Harrison**, Anjali K. M. De Silva**, Herbert Bauer*
*Aalen University, Aalen, Germany
**Glasgow Caledonian University, Glasgow, United Kingdom

**Correspondence:**

Peter Gantner
Aalen University
D-73430 Aalen
Germany

Tel: +49-(0)7361-576200
Fax: +49-(0)07361-576214
e-mail: peter.gantner@fh-aalen.de

**Keywords:**
Surface reconstruction, process chain optimisation, STEP, forming simulation, strength hardening, residual stress, residual strain

## Abstract

This paper is concerned with closing a gap in the process chain of metal forming. Tools for simulating the metal forming process like LS-DYNA® produce output geometries and stress information which cannot be easily re-imported in CAD-Systems or structure analysis programs for further processing.
A concept has been developed and implemented with a corresponding program which allows the re-importation of parts with certain topologies (tube, plane) from LS-DYNA® into any STEP-conformant CAD program. This method is mainly based on using the interconnection information which is contained in the LS-DYNA® output file. This information allows the construction of interpolating cubic B-Spline surfaces which can be represented in the STEP standard format. Thus, it is not necessary to reinvent general purpose surface reconstruction programs but rather to harvest the additional information available in the given situation.
Furthermore, a method to make the strength hardening information available in structure analysis is represented. This hardening results from the forming process and should be considered to obtain a more realistic virtual prototype and is of assistance to save weight and material costs.


## Introduction

The forming simulation by means of finite element analysis (FEA) is becoming more and more important in the field of process quality assurance and process design of mechanical and fluid media formed components.

Using the finite element simulation in the development process of hydroforming components from the first draft through to the serial production of a component provides an enormous saving of development time and costs. However, due to the constantly increasing competition in terms of costs, development time and quality, a further reduction of processing time and costs is necessary. Moreover, there is the demand for even more exact predictions and results in the area of the virtual component, in order to reduce the weight and to ensure that the component produced will have enough stability and low material costs.

Due to this demand the integration of the forming simulation into the process chain must be improved. In order to point out the optimisation potential, the sequence of the processes from the design phase to the finished component is represented in Figure 1.

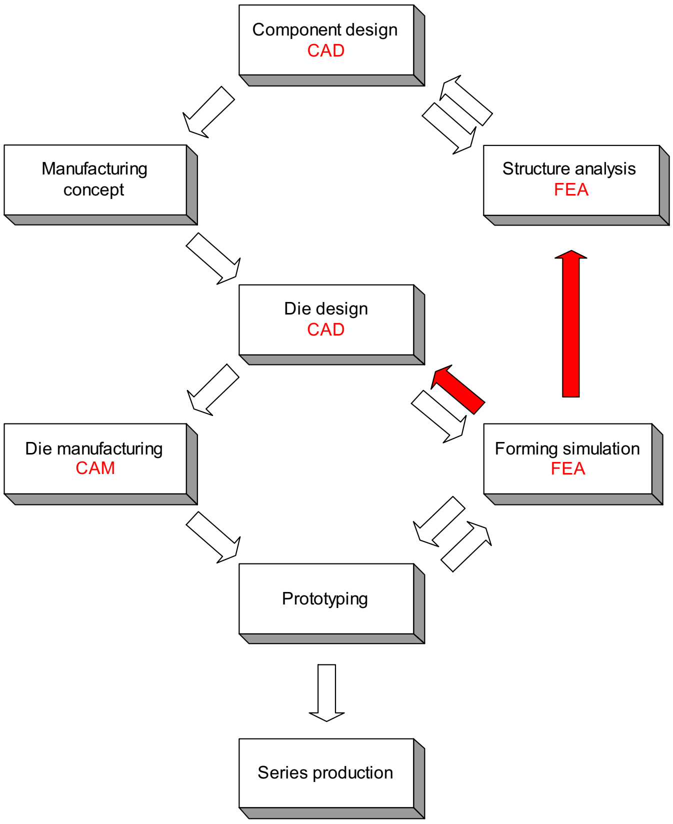



**Figure 1** A diagrammatic representation of the process chain of the development of a component

The forming simulation with the help of the FEA plays an essential role in the process chain depicted since further process steps depend on the results. For an optimisation concerning the accuracy and the processing time of a component, an improvement of the co-operation of the forming simulation with the adjacent process steps is necessary and this is shown in the Figure 1 by the dark arrows. This improvement can be achieved by interfaces, since the output of the simulation results cannot be processed directly by the adjacent processes.

The first part of this paper introduces a tool, which calculates B-Spline surfaces from the geometrical data of the forming simulation. The calculated surfaces are output in a geometry file, which can be read by all STEP-compatible programs. The reconstructed geometry can be used for e.g. optimisation of the tools or validating the real component with the simulation results.

The second part of the paper deals with the feedback of the strain hardening resulting from the forming process. At present neither the real thicknesses nor the strain hardening of the formed components are considered in the structural analyses. However, these properties can greatly affect the strength of the component. The goal of this method is, on the one hand, to be able to predict more exact results about the strength during the conception of a component. On the other hand, the goal is to reduce the weight and the costs of the component.

## 1. Surface reconstruction of results of the forming simulation

At present there are several commercial surface reconstruction programs on the market. However, the usage of these programs requires a detailed knowledge and a program-specific know-how; furthermore they cannot process the output data of the forming programs directly. Due to these facts a program was developed, which was conceived particularly for the reconstruction of simulation results. Presently, there are limits to the following basic conditions:

- Certified topologies are sheet metals and tubes
- For (adaptive) meshing square shell elements are certified

For the reconstruction of the geometric results the program uses the information contained in the output file (element -, node - and thickness specification) and calculates the boundary surfaces of the geometry as B-Spline surfaces, which are afterwards output in a STEP file.

### Forming simulation results data

The LS-DYNA® program supplies the following information about geometry as well as the stresses and strains in the formed component ("dynain file"):

- Coordinates of the element corner points on the central surface of the shell element
- Affiliation of the points to the shell elements and thickness of the shell elements
- Information about the adaptive mesh (origin during the adaptation of newly generated elements, position of newly generated nodes between corner points)
- Stress- and strain-information.

The following abstract from a LS-DYNA®-output file shows this information exemplarily:

```
*NODE
       1      -.200000000E+02       .200000000E+02 .000000000E+02 ...
...    2      -.200000000E+02       -.200000000E+02.000000000E+02 ...
...    3       .168967857E+02       .196337872E+02 .152427177E+02 ...
...
*ELEMENT_SHELL_THICKNESS
       1             ...    25            8              3              13
              .19999121E+01  .19999121E+01  .19999121E+01  .19999121E+01
       2             ...    24            9              8              25
              .19998425E+01  .19998425E+01  .19998425E+01  .19998425E+01
...
*CONSTRAINED_ADAPTIVITY
       29     24     9
       37     21     24
...
*INITIAL_STRESS_SHELL
         1         1         5
  .000E+00-7.067E-05-5.237E-04-2.818E-03 1.071E-04 1.691E-05 4.463E-04 ...
-9.062E-01-7.493E-05-1.021E-03-2.988E-03 1.070E-04 1.691E-05 4.732E-04 ...
-5.385E-01-7.320E-05-8.193E-04-2.919E-03 1.071E-04 1.691E-05 4.623E-04 ...
...
*END
```

The nodes of the shell elements with their appropriate node numbers and coordinates are indicated in the first section. The second section contains the coherence information of the nodes in the form of element definitions and the respective thickness of the elements. In the case of adaptive meshing the mesh refinement information is defined in the card *CONSTRAINED_ADAPTIVITY. The information means between which nodes a new node was generated (e.g. node 29 was generated between the nodes 24 and 9). The last two sections include the stresses and strains, which resulted from the forming. For a geometry feedback these data are not needed, however, they are important for the feedback concept of the strain hardening described in the second part of this paper.

**Calculation of the inner and outer point cloud**

The simulation of thin-walled components is always done with shell elements. Thus only the corner nodes of the elements, which are describing the middle surface, are available in the result file. In order to calculate the point-clouds of the inner and outer surface of tubes - or the upper and lower surface of sheet geometries, the thickness information and the information about the node-element relationship is used.

For this the normal vectors of each node are calculated by averaging the normal vectors of their surrounding elements. For this purpose the normal vectors of the elements are weighted according to their surface area.

The value with which the nodes are displaced along and contrary to the vector are then calculated by averaging the thickness of the adjacent elements, also weighted according to their surface. The averaged thickness is then halved and multiplied by the positive and negative external unit normal vector of the nodes and then added to the node vector, which has to be displaced. This provides two nodes, an interior and an exterior one or a lower and an upper one (Figure 2).

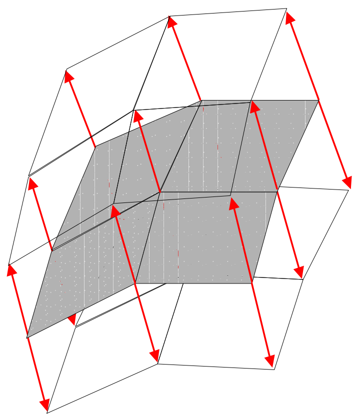
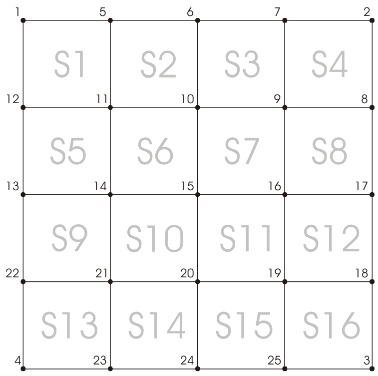


**Figure 2** A diagrammatic representation of the calculation of the inner and outer (or upper and lower) point cloud

### Calculation of the B-Spline surfaces

In order to be able to execute a B-Spline interpolation in a simple way, the information about the location of the points is needed. Therefore a sort algorithm is used to structure the point clouds and sort them into point rows in a direction by means of the node element relationship given in the element definition (Figure 3). This algorithm, however, only works if the initially defined basic conditions are kept.




```
[ 1,  5,  6,  7,  2]
[12, 11, 10,  9,  8]
[13, 14, 15, 16, 17]
[22, 21, 20, 19, 18]
[ 4, 23, 24, 25,  3]
```

**Figure 3** The determination of the point rows (where S' = element number)

For the case of an adaptive mesh with local refined elements an algorithm generates a uniform mesh with point rows containing the same number of points of same quantity (Figure 4).

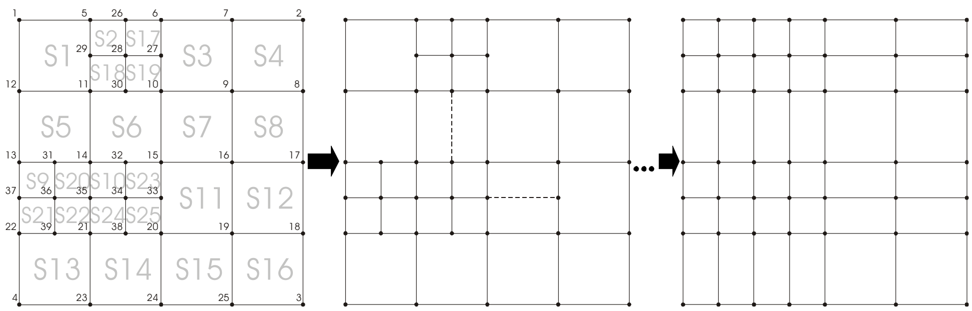



**Figure 4** The stages in the generation of a uniform mesh

The so called cubic tensor product B-Spline surfaces are used to model the free-form surfaces of the tube or sheet topologies mathematically. For the creation of such a surface the following data is needed:

- A network of control points $B_{ij}$, i=0…,n, j=0…,m
- Two node vectors ($u_0$…, $u_p$) and ($v_0$…, $v_q$) for the two parameter directions. To locate the boundary points of the control network on the B-Spline surface, the first four as well as the last four nodes are selected identically in each case in the node vectors. Such B-Splines are called "clamped" (cf. [1])

Now the cubic B-Spline surface in the parametric representation results as follows:

$$X(u,v) = \sum_{i=0}^{n}\sum_{j=0}^{m} B_{ij} N_{i3}(u) N_{j3}(v) \quad , \quad (u,v) \in \left[u_0, u_p\right] \times \left[v_0, v_q\right] \tag{1}$$

where the functions $N_{i3}$ and $N_{j3}$ are the cubic B-Spline basic functions of the two node vectors and the $B_{ij}$ are the control points which describe the B-Spline surface X in the parametric directions u and v. Essentially the method of Farin [2] is used to set up and solve the interpolation.

**Output in the STEP format**

After calculating the control points and the node vectors, the results are stored in the STEP format which is readable for many CAD and FEM systems (meshing tools).

Considering the number of ISO standards the following are important for surface reconstruction:

- ISO 10303-42 ([3]): Integrated generic resource: Geometric and topological representation. Here the substantial geometry objects are specified in the description language EXPRESS. For free form surfaces there are the B-Splines or more generally the NURBS.
- ISO 10303-214 ([4]): Application Protocol: Core DATA for Automotives Mechanical Design. This section contains (like other application protocols, cf. [5]) a collection of the essential objects for the area of "Automotive Mechanical Design". Such object quantities usually reference to object classes ("ENTITY types") from the base standards like the ISO 10303-42. Also more exact specifications or limitations can still be added (for instance with number areas of attributes). ISO 10303-214 describes different "Conformance Classes" (CC), so that there are different gradations in the standard conformity.
- ISO 10303-21 ([6]): Implementation methods: Clear text encoding of the exchange structure. This section states, how the concrete entities of the data types and object classes, described by EXPRESS, have to be formatted in an ASCII file. Besides the product data a certain header format is determined.

The ENTITIES "b_spline_surface" and the subtype "b_spline_surface_with_knots", which are indicated in the standard ISO 10303-42, are important for the surface reconstruction by means of B-Spline surfaces (cf. [3] p. 105-107). The substantial information, which is needed for the specification of concrete instances of these ENTITIES, consists of the list of the control points and of the node vectors. These data were determined in the preceding spline calculation. The other ENTITY attributes are in this case always alike (*_degree=3, *_multiplicities=[4,1,..,1,4] for "clamped" B-Splines) or have only the character of additional information (e.g. *_closed) without importance for the surface calculation. To store the instances of these ENTITIES, the specific format of the ISO 10303-21 has to be taken. For example the following data file results:

```
:
#1=CARTESIAN_POINT('',(3.263099E+000,-1.030531E+002,9.494244E+001));
#2=CARTESIAN_POINT('',(1.357688E+001,-9.846300E+001,9.370705E+001));
:
```

```
#35=CARTESIAN_POINT('',(2.184718E+001,8.223670E+001,-4.284212E+001));
#36=CARTESIAN_POINT('',(2.502546E+001,8.521254E+001,-3.244556E+001));
:
#55=B_SPLINE_SURFACE_WITH_KNOTS('',3,3,((#1,#2,#3,#4,#5,#6),(#7,#8,#9,#10
,#11,#12),(#13,#14,#15,#16,#17,#18),(#19,#20,#21,#22,#23,#24),(#25,#26,#27,#28,
#29,#30),(#31,#32,#33,#34,#35,#36)),.UNSPECIFIED.,.F.,.F.,.F., (4,1,1,4), (4,1,1,4),
(0.000000E+000, 3.058345E-001, 5.791911E-001,
1.000000E+000),(0.000000E+000,2.961919E-001,7.319924E001,1.000000E+000),
.UNSPECIFIED.);
:
```

In the first section the control points are separately indicated with a number, which is then used in the description of the B-Spline surface as reference.

### Example

The example of a tube formed by the hydroforming simulation shows the methodology. The first picture in Figure 5 shows the unformed tube in Pro/ENGINEER®. A shell model with about 14000 nodes is shown in the second picture as a result of the forming simulation with LS-DYNA®. The left picture in Figure 6 shows the formed tube as a point cloud in LS-POST®. It is also possible to import the point cloud into the CAD system Pro/ENGINEER®, as it is shown in the right picture. However, the point cloud is in each case unfit for further handling. With this technique it is now possible to produce the boundary surfaces, from which a solid model can be created immediately. This is illustrated in the left picture of Figure 7. For comparison it is shown with the picture of the real hydroformed tube.

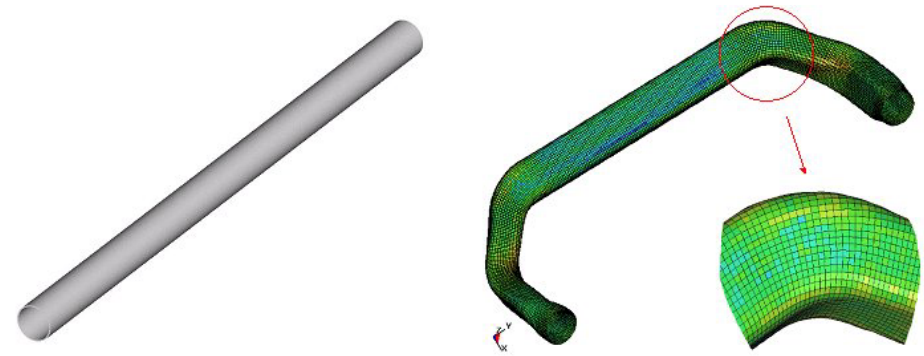


**Figure 5:** The unformed tube (left) and the result of the forming simulation (right)

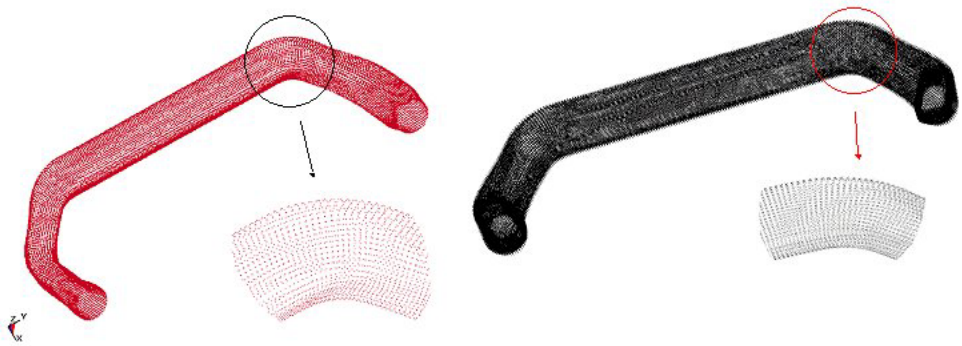


**Figure 6:** Visualization as a point cloud in LS-POST® (left) and in Pro/ENGINEER® (right)

**Figure 7:** The result of the reconstruction (B-Spline surfaces) (left) and the real component (right)

## 2. Stress/Strain-feedback from results of the forming simulation

In order to design components for lightweight constructions it is necessary to consider all results from the forming simulation in the structure-analysis. Thereby the strain hardening and wall thickness distribution resulting from the forming particularly play an important role. In the following the consideration of the strain hardening and the wall thickness distribution in a structure analysis is shown by an example of a hydroformed rear axle component of a BMW car.

### The forming simulation in LS-DYNA®

The starting point for the consideration of the strain and the wall thickness in a structural analysis is a forming simulation in LS-DYNA®. For the simulation the tools were designed as rigid body shell models and the unformed tube was designed as a deformable shell model. For the further procedure a springback calculation is necessary after the forming simulation. In Figure 8 the contours of effective plastic strain are represented after the springback simulation.

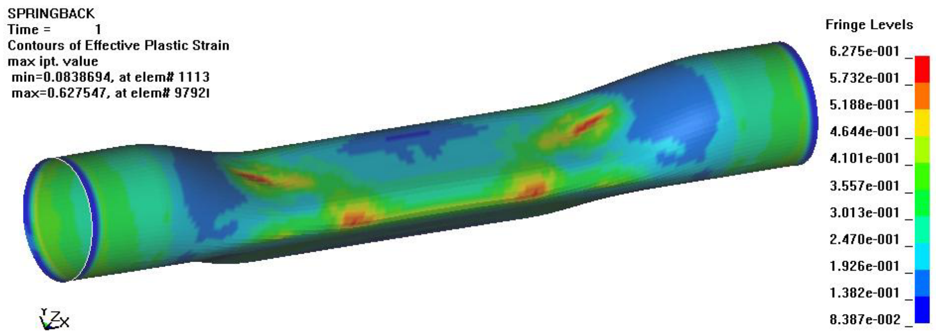



**Figure 8:** Contours of Effective Plastic Strain after the springback calculation

**Editing of the geometry and mapping of the results**

After the springback calculation the middle surface of the component is reconstructed and the trimming operations are realised in the CAD program. The close-to-reality surface model of the component is then be meshed in a pre-processor with the discretisation suitable for the structural analysis. For the transfer of the results (wall thicknesses, stresses and strains) a map process is necessary afterwards, which is done in LS-DYNA®. In Figure 9 the trimmed component with the mapped results are shown.

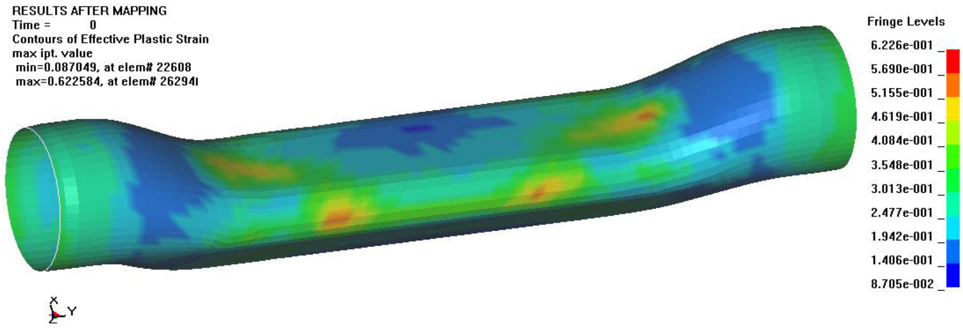



**Figure 9:** Contours of Effective Plastic Strain after the mapping

**The translation of the results from the forming simulation to the structural analysis**

To translate the results from the forming simulation into the structural analysis program a tool was developed, which calculates the new yield strength for each item from the results of the forming as follows:

- Selection or averaging of the plastic strain; there are several options for selection: strain of element on the central surface (integration point 0), the average value of the strain for each element as well as the largest and the smallest strain value.
- Afterwards a new yield strength for each element is determined by the strain value and the stress-strain curve (cf. Figure 10)

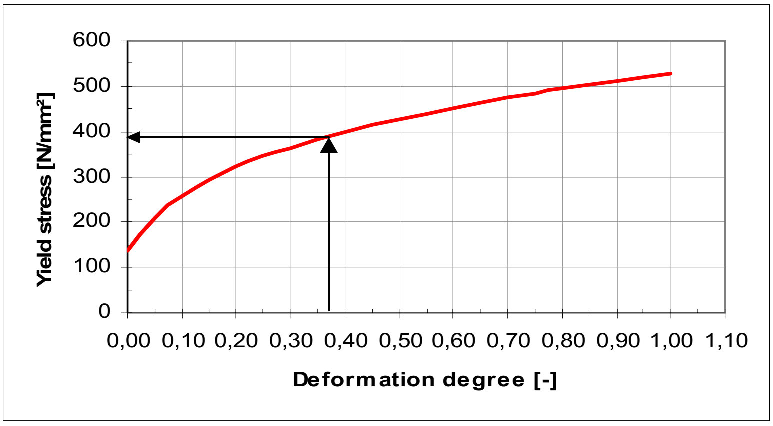



**Figure 10:** Determination of the new yield strength

For the transfer of the thickness information an averaging process is likewise necessary, since in ANSYS® the thickness is indicated for each node as opposed to LS-DYNA® for each element. To calculate the thickness of the geometry at a node the thicknesses of the surrounding elements are averaged and assigned to this node. After all values are calculated an input file for ANSYS® is produced.

**Execution of a structural analysis in ANSYS®**

To set off a structural analysis in ANSYS® the input file is read and the desired load case is calculated by defining the load conditions and the forces. In the calculation only the converted wall thicknesses of the individual elements are considered. The yield strengths from the forming are considered during the post processing by the use of an APDL-script, which calculates the safety factor for each element from the calculated equivalent stress and the yield strengths from the forming simulation. However, for the evaluation of the results in the post processing the representation of the reciprocal value (=equivalent stress/Re) of the safety factor has proved itself, as by the calculation of the safety factor a large interval width results. Thus, it would be very difficult to detect which values are critical or uncritical. By representation of the reciprocal value of safety factor this is very good to detect due to the clearer interval width between 0 to a maximum of 2.

In Figure 11 the result of an executed calculation is represented. The potential for the consideration of the strain hardening and the true wall thickness becomes evident, if one compares these results with the results of a calculation without consideration of the effects from the forming (Figure 12).

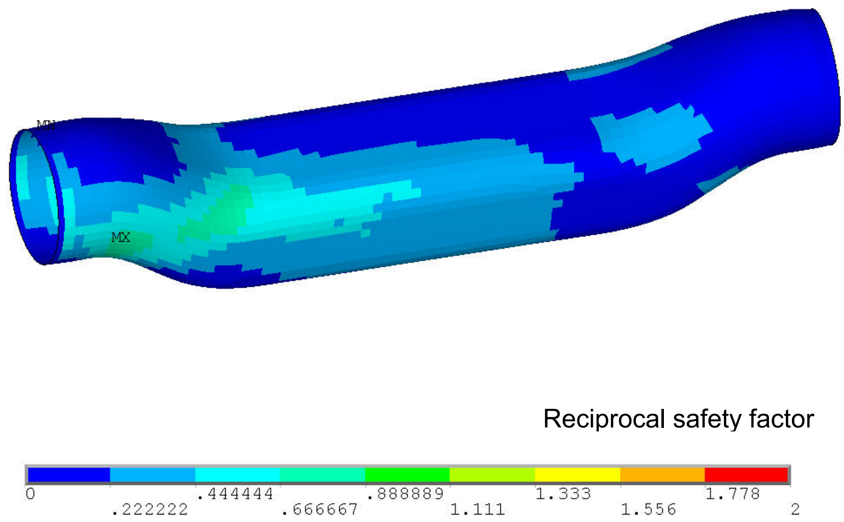



**Figure 11:** The result of the ANSYS® calculation with the related strain hardening

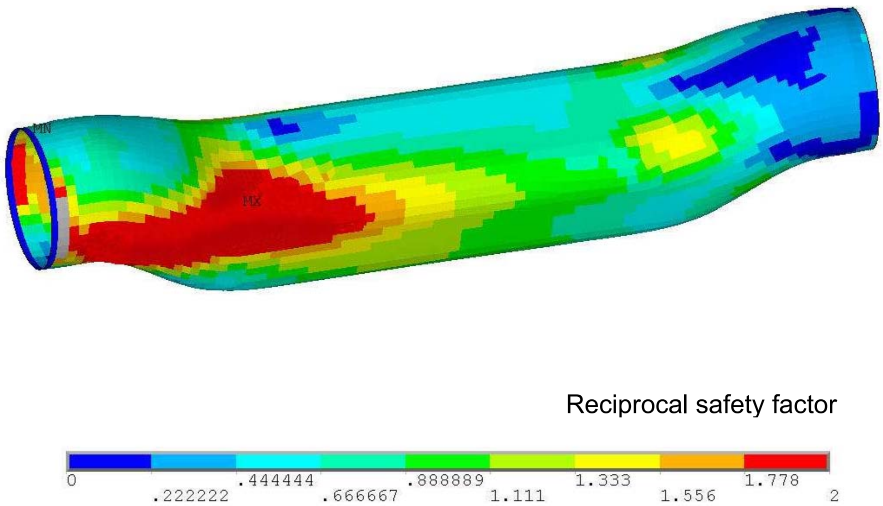



**Figure 12:** The result of the ANSYS® calculation without strain hardening

**Summary**

In the first part of this contribution a method for the reconstruction of the simulation results into the CAD world was presented. The output of a simulation tool as LS-DYNA® cannot be easily re-imported into a CAD program for further investigations due to the fact that the descriptions of geometry are completely different. With the program presented the reconstruction from LS-DYNA® is possible almost "on the push of a button" for bodies with certain topologies (tubes, sheets). The calculated boundary surfaces can be used afterwards for tool engineering with multi-stage forming processes, for more exact investigation of the virtual prototype or for the validation of the simulation result with the real component.

In the second part of this contribution a method was shown for the consideration of the strain hardening and the wall thickness distribution in strength calculations. The hardening resulting from the forming has thereby a significant influence on the strength of the component. By the consideration of these characteristics by means of the method presented it is possible to save weight and thus also material costs in the very early stage of the conception of the component.

## References

[1] Piegl, L., Tiller, W.: The NURBS Book, 2nd Edition, Berlin: Springer, 1997 p. 571. ISBN 3-54-061545-8

[2] Farin, G.: Curves and Surfaces for Computer-Aided Geometric Design, 4th Edition, San Diego: Academic Press, 1996. ISBN 0-12-249054-1

[3] ISO 10303-42: Industrial Automation Systems and Integration – Product Data Representation and Exchange - : Part 42: Integrated generic resource: Geometric and topological representation, 2nd Edition, 2000 pp. 105-107

[4] ISO 10303-214: Industrial Automation Systems and Integration – Product Data Representation and Exchange - : Part 214: Application Protocol: Core Data for Automotive Mechanical Design 1999

[5] Anderl, R., Trippner, D.: STEP – Standard for the Exchange of Product Model Data, Stuttgart: Teubner, 2000. ISBN 3-519-06377-8

[6] ISO 10303-21: Industrial Automation Systems and Integration – Product Data Representation and Exchange - : Part 21: Implementation methods: Clear text encoding of the exchange structure, 1994

[7] Hoschek, J., Lasser, D.: Grundlagen der geometrischen Datenverarbeitung, 2. Auflage, Stuttgart: Teubner, 1992. ISBN 3-519-12962-0
(Hoschek, J., Lasser, D.: Basics of the geometrical data processing, 2nd Edition, Stuttgart: Teubner, 1992. ISBN 3-519-12962-0)