Mesh refinement study and experimental validation for stretch bending of sheet metals.

M Raupach¹, S Kreissl², L Vuaille², T Möller³, H Friebe³, W Volk¹

¹Lehrstuhl für Umformtechnik und Gießereiwesen, Technische Universität München, Walther-Meißner-Straße 4, 85748 Garching
²BMW Group, Knorrstraße 147, 80788 München
³GOM GmbH, Schmitzstrasse 2, 38122 Braunschweig

marco.raupach@partner.bmwgroup.com

Abstract: For sheet metal parts with small radii and large bending angles, the sheet metal forming simulation reaches their application limits. Alternatives are complex shell formulations and volume elements. For volume elements, the necessary number of elements over the thickness is important. Valid values are not available depending on discrete radii. Therefore in this work, a convergence study is performed using the example of an angular stretch bend test with a radius to thickness ratio of 1. For various states of mesh refinement, simulations are performed, various results are presented, analysed and discussed with regard to convergence behaviour to the necessary number of elements in thickness direction. Recommendations for suitable validation variables are derived.

Based on the refinement study, a simulation model for an experimental validation is developed. The experiments are carried out in a sheet metal forming machine. Experimental angular stretch bend test with a punch radius of 1 mm are performed until failure and the strain distribution on the top side of the sheet is measured. Finally, simulation and experiments are compared based on the surface strain.

1. Introduction
Decades ago finite element (FE) formulations have been developed, that allow for an efficient simulation of structures in which one dimension – the thickness – is significantly smaller than the other two. These shell-elements have become state of the art for various applications in the automotive industry where sheet-like structures are widely used. Common applications are crash-simulations as well as forming simulations of body-in-white (BIW) components. With increasing complexity of the part geometries the assumption of one dimension being small compared to the other two may no longer hold. An example where the above assumption is violated – at least locally – are zones in BIW components where the ratio of bending radius to sheet thickness is small. Volume elements do not rely on the model assumptions of shell elements but require much more elements to approximate physical bending behavior, leading to high computational costs. Using a high-fidelity model based on volume elements the current study analyzes the requirements regarding mesh refinement in a simple angular stretch bend test (ASBT).
2. Element formulations
Shell-elements commonly used in forming simulations, e.g. [1], are often based on the Reissner-Mindlin-Theory of plates that assume that stresses normal to the mid-plane \( \sigma_z \) are zero [2]. However, for small bending radii it can be shown, that the stress \( \sigma_z \) is not equal to zero [3] and thus violating the Reissner-Mindlin assumption. Another assumption of common shell elements is that the cross sections remain straight [1]. Fleischer showed that this is no longer the case for small bending radii [6]. To compensate for the shortcomings of common shell elements many studies focused on more advanced shell-formulations, e.g. [4]. Various thick shell formulations have been made available in commercial codes, e.g. [5], but are rarely used for industrial applications. While these thick shells allow for non-zero \( \sigma_z \), common 8-node (tri-linear) brick-like thick shells, such as [7], still cannot capture nonlinear cross sections. In order to capture a nonlinear deformation of the cross-section multiple of these thick-shells are necessary.

3. Angular Stretch Bend Test
The current study is based on the Angular Stretch Bend Test (ASBT) to analyze the requirements regarding mesh-refinement for small bending radii. The layout of the test is depicted in Figure 1. It consists of a blank, a binder, a punch and a die. The geometry of the punch is defined by the punch opening angle \( \alpha_p \) and the punch radius \( r_p \). The parameter \( r_D \) defines the entry radius of the die. In this study, the ratio between punch radius and sheet thickness is 1, leading to severe bending deformations. Many experimental studies employed the ASBT for understanding failure and identifying the influential parameters in stretch-bending, e.g. [8] and [9]. Hora et al. investigated the necking phenomenon in the ASBT [10]. Kitting et al. compared solid-element based simulations with experimental results of an ASBT and obtained good agreement regarding major strains [11]. Neuhauser et al. studied critical strains through the thickness of sheet metal using the ASBT [12]. 2D and 3D numerical studies regarding failure modes for different ratios of punch radius to sheet thickness have been conducted by Gonzalez et al. [13]. Haufe et al. investigated the applicability of a shell element formulation for the ASBT [14].

4. Mesh refinement study
For the convergence study the number of elements over the blank thickness, \( t_B \) is varied. To save computational cost only a narrow strip of width \( w_B = t_B \) is modelled, see Figure 2. The boundary conditions are clamped ends of the blank under the binder, and a plane-strain condition at both sides of the strip normal to the y-direction.

The commercial FE software LS-Dyna is used to solve the problem. For the numerical study the blank is discretized with solid elements of type -1 [15] combined with a three-dimensional isotropic elasto-plastic material model, MAT 24 [16]. All Elements are cubes with an aspect ratio of 1. The tools are modelled as rigid bodies. A mortar contact is used between punch and blank [17]. The remaining contacts are modelled with a one-way automatic contact. Friction is captured by a validated user-defined friction model. The problem is solved with an explicit time integration scheme without mass scaling. The geometry parameters corresponding to the tool set-up are listed in chapter 5.1.
4.1. Convergence of force measure

Often times reaction forces are used to either measure the convergence of a numerical model or show that a numerical model yields results in agreement with an experiment. Figure 3 illustrates that the reaction forces are very insensitive to the level of mesh-refinement. This observation is confirmed by an error measure for the force $e_F$, defined as:

$$
e_F^N = \frac{1}{n_{end} - n_{start} + 1} \sum_{n=n_{start}}^{n_{end}} \frac{F_P^N(n) - F_P^{20}(n)}{F_P^{20}(n)} \cdot 100$$

(1)

$F_P^N(n)$ and $F_P^{20}(n)$ are the punch force at state n for mesh with $N$ and $N=20$ elements in thickness direction. To avoid division by zero in Eq. (8) due to the fact that the forces at the beginning of the simulation are approximately zero, see Figure 3, the error measure only considers states from $n_{start}=30$ to $n_{end}=100$. As Figure 4 illustrates, the force error is already less than 0.005% for $N = 2$ elements.

![Figure 3: Punch force](image1)

![Figure 4: Punch force error over number of elements N](image2)

4.2. Convergence of normal stress in thickness direction

Common shell elements approximate a plane-stress condition with the stress normal to the mid-surface being zero, $\sigma_\perp = 0$. The solid-element based model used in this study allows for these normal stresses: Figure 5 shows the stress distribution over the blank thickness at the pole of specimen for various levels of mesh refinement $N$ at the final state.

![Figure 5: Distribution of the normal stress component](image3)

Given the free surface at the top ($t_B = 1$) the stresses are close to zero for all levels of mesh refinement. At the bottom ($t_B = 0$) compressive stresses $\sigma_\perp < 0$ occur due to contact with the punch. The graph of
the reference solution with \( N=20 \) elements shows a highly nonlinear normal stress distribution. For a quantitative measure, a stress integral at state \( n \) is calculated as follows:

\[
f_N^I(n) = \sum_{k=1}^{N} a_k^I(n) h_k(n)
\]

(2)

\( h_k(n) \) is the height of the \( k \)-th element at simulation state \( n \); \( N \) is the number of elements in thickness direction. Based on Eq. (2) a measure for capturing the error of the normal stress at state \( n \) is be calculated:

\[
e_N^I(n) = \frac{f_N^I(n) - f_{20}^I(n)}{f_{20}^I(n)} \cdot 100
\]

(3)

\( f_N^I(n) \) and \( f_{20}^I(n) \) are the integrated normal stresses of a mesh with \( N \) and \( N=20 \) elements in thickness direction at state \( n \). Figure 6 shows the errors \( e_N^I(n) \) over all states. To mitigate the effects of dynamic oscillations at the beginning of the simulations, only states from \( n_{\text{start}}=30 \) are considered for the summed error measure:

\[
e_N^I = \frac{1}{n_{\text{end}} - n_{\text{start}} + 1} \sum_{n=n_{\text{start}}}^{n_{\text{end}}} e_N^I(n)
\]

(4)

Figure 7 shows the error \( e_N^I \) for different levels of mesh refinement. Even with \( N=10 \) elements the error is larger than 5%, indicating that the solution has not yet converged.

4.3. Convergence of internal energy measure

The previous section analyzed the convergence of a single stress component. The stress state however is defined by stress tensor containing six separate stress components. In order to obtain a more holistic measure of converge for both stress- and strain-tensor the internal energy is used. The definition for the internal energy at state \( n \) is as follows:

\[
E_{\text{int}}^i(n) = \frac{1}{2} E^i(n) : S^i(n) V^i(n),
\]

(5)

\( E_{\text{int}}^i(n) \) is the internal energy for the \( i \)-th element \( E^i(n) \), \( S^i(n) \) and \( V^i(n) \) describe the Green-Lagrange strain tensor, the second Piola-Kirchhoff stress tensor and the volume of the respective element, at state \( n \). The arms of the blank - connecting the clamped section under the binder with the area around the pole of the punch are subjected to uniaxial stress. Given that distribution of stress/strain in the arms is mostly uniform in thickness-direction, these areas are rather insensitive regarding mesh-refinement. Therefore, measuring the convergence of the internal energy of the entire blank strip might yield misleading results.
Hence only elements located within a square box above the pole of the punch, with an edge length $l_{box}$, are considered for measuring the convergence, see Figure 8.

![Figure 8: Square box selecting elements for convergence criteria (at time $t=0$).](image)

The subset of blank element indices $i$ located within the box at the initial state will be denoted the index $j$ in the following. The energy density of this subset of elements at the state $n$ is defined by:

$$\rho_{int}(n) = \frac{1}{\Sigma j \cdot Vj(n)} \sum_j (\frac{1}{2} E^j(n) \cdot S^j(n) \cdot Vj(n))$$

(6)

The graph in Figure 9 shows the energy density values for different mesh-refinement levels. Given the low levels of internal energy at the beginning of the simulation only states from $n_{start}=30$ are considered in the following. To capture the error of different mesh-refinement-levels $N$, the error $e_{int}^N$ is defined:

$$e_{int}^N = \frac{1}{n_{end} - n_{start} + 1} \sum_{n=n_{start}}^{n_{end}} \frac{\rho_{int}(n) - \rho_{int}^{20}(n)}{\rho_{int}^{20}(n)} \cdot 100.$$  

(7)

$\rho_{int}(n)$ and $\rho_{int}^{20}(n)$ is the internal energy density of a mesh with $N$ and $N=20$ elements in thickness direction at state $n$. The graph in Figure 10 shows the evolution of the averaged error converges to zero with increasing mesh refinement.

![Figure 9: Internal energy density](image)

![Figure 10: Energy density error over number of elements $N$](image)

4.4. Convergence of spring back measure

A geometrical measure to characterize the convergence of the numerical model is the level of spring back. The springback is simulated based on the last state for various levels of mesh refinement. Figure 11 shows the boundary conditions for the springback simulation. The spring back of a mesh with $N$ elements in thickness direction is measured as the distance of a point before spring back, $p_i^N$ and the same point after spring back, $p_{SB}^N$: $\Delta_{SB}^N = \|p_i^N - p_{SB}^N\|$, see Figure 11. The spring back error of a mesh with $N$ elements in thickness direction is computed as:

$$e_{SB}^N = \frac{\Delta_{SB}^N - \Delta_{SB}^{20}}{\Delta_{SB}^{20}} \cdot 100$$

(8)
The graph in Figure 12 shows that the spring back error decreases significantly with the number of elements $N$ in thickness direction.

![Figure 11: Measure of spring back](image)

![Figure 12: Springback error over number of elements $N$](image)

4.5. Summary of mesh convergence study

The difference convergence measures introduced in Section 4.1 to 4.4 indicate that the solution converges with an increasing level of mesh refinement. However, the different measures show different rates of convergence and significantly different levels of error. While for $N=2$ the punch force error produces a negligible error of $e_F^{N=2} < 0.005\%$, the error of the internal energy and the spring back error indicate that $N=2$ elements in thickness direction is not sufficient for ensuring a converged solution: $e_{\text{int}}^{N=2} > 30\%$ and $e_{SB}^{N=2} > 70\%$. The reason why considering the convergence of the reaction force may yield misleading conclusions for the ASBT lies in the fact that the computed reaction forces just have to satisfy a mechanical equilibrium. Even coarse meshes are sufficient to achieve such an equilibrium. Both the internal-energy- and the spring-back-measure capture a more complex characteristics: levels of strains and stresses. Since the quality of the formed part is, amongst others, determined based on formation of cracks, wrinkles, spring back, etc., it is advisable to ensure that stress and strain levels are captured accurately, in order to be able to predict the above mentioned quality criteria. For the numerical model in the following sections it was therefore decided to use $N=8$ elements in thickness direction as a compromise between accuracy and computational cost.

5. Validation: comparison of numerical and experimental results

5.1. Tool Setup of the ASBT and experimental procedure

Given that the reaction forces are insensitive regarding mesh refinement, it is assumed that reaction forces are also not suitable for validation purposes. Instead, the strain around the pole is used for comparison between simulation and experiment. The tool setup is shown Figure 13 and the tool geometry parameters listed in table 1. The dimensions of the specimen are 220 mm x 60 mm x 1 mm.

| Parameter             | Value  |
|-----------------------|--------|
| Punch radius          | 1 mm   |
| Punch opening angle   | 70 deg |
| Die entry radius      | 12 mm  |
| Die opening           | 62 mm  |
| Distance between drawbeads | 125 mm |
| Drawbead radius       | 2 mm   |

All specimen are machined at their edges to avoid negative influences from shear cutting. The punch is made of GGG70L corresponding to large deep-drawing tools. The material of the specimen is AlSi0.6Mg0.5-Fo. Figure 14 shows the formed parts of three experiments, each with the stochastic pattern for the strain measurement.
In addition to the strain on the top side of the part, the binder force, punch force and punch translation is measured for each test. The functionality and test execution is similar to a standard Nakajima test. Through the opening in the die, the sample can be observed over the entire test period. For the measurement of the deformation and the strain values of the test specimen the Digital Image Correlation (DIC) measuring system ARAMIS 5M is used [18]. A stochastic pattern is applied to the specimen's surface and synchronized stereo images of the specimen are recorded at different loading states during the tests. Figure 15 shows the undeformed and the deformed configuration for a specimen. In this case the image of the undeformed configuration of the left camera is divided into a large amount of subsets (facets). The center of each facet is shown as a cross, five of the facets are shown as quadrangle. For each facet the corresponding area is calculated for the right camera also for all loading states. Based on the well-known geometry of the optical setup for each center of a facet (each cross in Figure 15) a 3D point is calculated for all loading states. For each point following result values are derived: 3D coordinates, displacements, velocities, the surface strain tensor (strain x, -y, xy, major, minor, thinning) and the strain rates.

5.2. Simulation Setup of the ASBT and procedure
The FE-model is equal to the model of the convergence study above except for few differences: drawbeads are included and the dimensions of the blank as well as the rigid tool surfaces are adjusted according to the sample size. Boundary conditions of the blank are adapted for a half-model with symmetry at the xy-plane and the blank is discretized by solid elements (type -1) with an initial aspect ratio of 1 and N=8 elements in thickness direction. In order to compare the surface strain between experiment and simulation a layer of shell elements (thickness of 1 µm) was added on the top of the sheet.
5.3. Averaging and full field comparison between numerical simulation and ARAMIS measurement

The ARAMIS software provides a module to import results from numerical simulations. This tool can be used to compare results from simulation and measurement as well as compare or average several independent measurements conducted with the same testing setup [19]. The ARAMIS measurement results are derived from the pattern on the surface of the measured specimen and the strain results are calculated with help of a surface strain tensor. Before the comparison of simulated and measured strains a representative sample must be selected. For this purpose, one measurement serves as reference and the strain distributions of the remaining two measurements are mapped on the reference mesh. Figure 16 shows the strain distribution of the three samples and their mean value at a state close before fracture at section A-A. The entire evaluation and the comparison are carried out using three sections. The defined sections on the reference sample are shown in Figure 17. The strain profiles of the three samples agrees well for the shown state, but each measurement consists of 330 individual states.

![Figure 16: Major strain distribution of three samples and their average over the section length](image)

![Figure 17: Section positions at the reference sample](image)

Therefore the selection of the appropriate measurement is made by the relative strain distribution error over all states for every section. The error for one section is calculated according to:

\[
e_{\varphi_{\text{I}}}^C = \frac{1}{n_{\text{end}} \sum_{n=1}^{n_{\text{end}}}} \int_{s_0}^{s_{n}} \varphi_{\text{I}}^C(n) ds - \int_{s_0}^{s_{n}} \bar{\varphi}_{\text{I}}^C(n) ds \cdot 100
\]

\( \varphi_{\text{I}}^C \) is the strain distribution for one measurement and \( \bar{\varphi}_{\text{I}}^C \) the corresponding mean value at state \( n \) for section \( C \in \{A-A, B-B, C-C\} \). Figure 18 shows the relative errors of the three samples and three sections. The largest deviation of 2.5% presents sample \( V_2 \) at section B-B. In total sample \( V_3 \) presents the best agreement and is therefore used for the FEM validation.

![Figure 18: Strain distribution error for various sample sections](image)
The mesh data from the numerical simulation and the data from sample V₃ are imported in the ARAMIS software and surface strains are computed similarly to achieve comparable results. Generally, the coordinate systems of the simulation and the experiment differ, what makes an alignment of both 3D datasets indispensable. Alignment algorithms are included in the ARAMIS software module. Since the positions of the ARAMIS measurement points and the nodes of the simulation differ, the results of these 3D points cannot be directly compared. Therefore a re-calculation (mapping) between simulated and measured points is performed, including 3D coordinate and results value interpolations. After these steps - import, strain calculation, 3D alignment and mapping - full field deviations between numerical simulation and measurement results are calculated. Analogous to Figure 17 the comparison is carried out for three sections: Figure 19 shows the major strain $\phi_I$ for both simulation and experiment at the three sections at the state just before failure occurs in the experiment.

\[ \text{Figure 19: Comparison of the experimental major strain distribution and the major strain distribution from the FE-simulation from three sections} \]

The combination of the small surface area of the outer radius and the relatively high local deformation is a challenge for the measuring system itself. Despite the small ratio of punch radius to sheet thickness simulation and experiment show good agreement, cp. Figure 19. Due to the fact that the simulated major strains are slightly higher than the experimental ones, which is visible to the left and right next to the strain peak at section C-C. In total all strain distributions are qualitatively and quantitatively comparable. This indicates that N=8 elements over the thickness are sufficient to capture the strain distribution.

6. Summary and outlook

Employing the common ASBT with a punch radius to sheet thickness ratio of 1, different convergence measures were analyzed in a mesh refinement study. It was be shown that a simple measure like the reaction force acting on the punch may yield misleading results. Measures that capture stress and strain distributions seem to be more feasible for the ASBT. Based on the different convergence studies a simulation setup with N = 8 elements in thickness direction is chosen as a compromise between accuracy and computational cost for a comparison between numerical simulation and experiment. The comparison between the latter two is carried out based on strain distributions on the surface of the specimen, where the strain of the experimental specimen is captured with a Digital Image Correlation measuring system. Given the agreement of numerical and experimental results, other element formulations such as shells, thick shells can be rated against the solid element solution. Based on such a study the limits of applicability of shell elements for small bending radii was determined.
References

[1] Belytschko, T.; Lin, J.I.; Tsay, C.S.: Explicit Algorithms for Nonlinear Dynamics of Shells. In: Computational Methods in Applied Mechanical Engineering 42 (1984), pp 225-251.

[2] Fung, Y.C.; Tong, P.: Classical and Computational Solid Mechanics. In: Advanced Series in Engineering Science – Volume 1 (2001), ISBN 981-02-3912-2.

[3] Marciniak, Z.; Duncan, J.L.; Hu, S.J.: Mechanics of Sheet Metal forming. Butterworth-Heinemann (2002), ISBN: 9780750653008.

[4] Bischoff, M.: Theorie und Numerik einer dreidimensionalen Schalenformulierung. PhD thesis at the Institute for Structural Mechanics, University of Stuttgart (1999).

[5] Chatiri, M.; Güll, T.; Matzenmiller, A.: An assessment of the new LS-Dyna layered Solid element: basics, patch simulations and its potential for thick composite structure analysis. In: 7th European LS-Dyna Conference (2009).

[6] Fleischer, M.: Absicherung der virtuellen Prozesskette für Folgeoperationen in der Umformtechnik, PhD thesis, Chair of Structural Analysis, Technical University of Munich (2009).

[7] Belytschko, T.; Liu, W.K.; Moran, B.; Elkhodary, I.; Nonlinear Finite Elements for Continua and Structures. Wiley (2008), ISBN: 978-1-118-63270-3.

[8] Demeri, M.Y.: The Stretch-Bend Forming of Sheet Metal. In: Journal of Applied Metalworking - Volume 2 (1981), pp 3-10.

[9] Charpentier, P.L.: Influence of Punch Curvature on the Stretching Limits of Sheet Steel. In: Metallurgical and Materials Transactions A - 6: 1665, Springer-Verlag (1975), ISSN: 1073-5623.

[10] Hora, P.; Berisha, B.; Gorji, M.; Manopulo, N.: A Generalized Approach for the Prediction of Necking and Rupture Phenomena in the Sheet Metal Forming. In: Proceedings IDDRG (2012), pp 79-93.

[11] Kitting, D.; Koplenig, M.; Ofenheimer, A.; Pauli, H.; Till, E.T.: Application of a “Concave-Side Rule” Approach for Assessing Formability of Stretch-Bent Steel Sheets. In: International Journal of Material Forming - Volume 2 Suppl. 1 (2009), pp 427-430.

[12] Neuhauser, F.M.; Terrazas, O.R.; Manopulo, N.; Hora, H.; Van Tyne, C.J.: Stretch bending – the plane within the sheet where strains reach the forming limit curve. In: Materials Science and Engineering – Volume 192 (2017).

[13] Gonzalez Zuniga, A.; Rodríguez González C.; Hendrichs Troeglen, N.; Elias Zuniga, A.: Finite Element Analysis of Stretch-Bending for Advanced High Strength Steels. MEMORIAS DEL XVI CONGRESO INTERNACIONAL ANUAL DE LA SOMIM, ISBN: 978-607-95309-3-8.

[14] Haufe, A.; Andrade, F.; Feucht, M.: Zu den Grenzen der Versagensprognose mit Schalenmodellen in der Umformsimulation. In: Proceedings 22. Umformtechnisches Kolloquium Hannover (2017), pp 144-156.

[15] LS-DYNA THEORY MANUAL (2006).

[16] LS-DYNA KEYWORD USER’S MANUAL I (2016).

[17] Borrvall, T.: Mortar Contact for Implicit Analysis In: 11th European LS-Dyna Conference (2012).

[18] Winter, D.: Optische Verschiebungsmessung nach dem Objektrasterprinzip mit Hilfe eines flächenorientierten Ansatzes, PhD Thesis, Technical University of Braunschweig (1993).

[19] Friebe, H.; Klein, M.; Galanulis, K.: Numerical comparison and verification of FEA in sheet metal forming by optical measurements of large and complex parts, Proceedings of IDDRG (2010), pp 935-944.