A benchmark study for different numerical parameters and their impact on the calculated strain levels for a model part door outer

E. Berger [1], T. Brenne [2], A. Heath [3], B. Hochholdinger [4], K. Kassem-Manthey [5], L. Keßler [6], N. Koch [7], G. Kortmann [8], A. Kröff [9], T. Otto [10], G. Steinbeck [11], E.Till [1], H. Verhoeven [10], T.-C. Vu [12], K. Wiegand [13]

[1] voestalpine Stahl GmbH, Linz, Austria
[2] AutoForm Engineering Deutschland GmbH, Dortmund, Germany
[3] Engineering System International GmbH, Eschborn; Germany
[4] DYNAmore GmbH, Stuttgart, Germany
[5] Gesellschaft für Numerische Simulation GNS, Braunschweig, Germany
[6] ThyssenKrupp Stahl AG, Dortmund, Germany
[7] Adam Opel AG, Rüsselsheim, Germany
[8] Wilhelm Karmann GmbH, Osnabrück, Germany
[9] Salzgitter Mannesmann Forschung GmbH, Salzgitter, Germany
[10] Volkswagen AG, Wolfsburg, Germany
[11] Stahlinstitut VDEh, Düsseldorf, Germany
[12] BMW AG, München, Germany
[13] DaimlerChrysler AG, Sindelfingen, Germany

Abstract. To increase the accuracy of finite element simulations in daily practice the local German and Austrian Deep Drawing Research Groups of IDDRG founded a special Working Group in year 2000. The main objective of this group was the continuously ongoing study and discussion of numerical / material effects in simulation jobs and to work out possible solutions. As a first theme of this group the intensive study of small die radii and the possibility of detecting material failure in these critical forming positions was selected. The part itself is a fictional body panel outside in which the original door handle of the VW Golf A4 has been constructed, a typical position of possible material necking or rupture in the press shop. All conditions to do a successful simulation have been taken care of in advance, material data, boundary conditions, friction, FLC and others where determined for the two materials in investigation – a mild steel and a dual phase steel HXT500X. The results of the experiments have been used to design the descriptions of two different benchmark runs for the simulation. The simulations with different programs as well as with different parameters showed on one hand negligible and on the other hand parameters with strong impact on the result – thereby having a different impact on a possible material failure prediction.

INTRODUCTION

Issues concerning the feasibility of forming small radii with mild steel and high-strength materials are routinely encountered in practice. To address this problem, a working group was set up at Stahlinstitut VDEh in year 2000 as part of the German Group of the International Deep Drawing Group (IDDRG) to investigate potential deviations between simulated and practical trial results in a systematic manner. The addressees included automotive manufacturers, component suppliers and producers of semi-finished products as well as academic institutes and research departments.

To ensure that the working group would operate effectively, the objective was defined as follows:

CP778 Volume A, *Numisheet 2005,* edited by L. M. Smith, F. Pourboghrat, J.-W. Yoon, and T. B. Stoughton

"The aim of the working group is not to capture a momentary image of the performance capability of FE systems, but to understand the deviations between simulated and practical sheet forming results. In addition, the causes of such deviations are to be examined through a longer-term discussion and experience sharing effort."

This discussion should address both the core project tasks and the choice of components. It deserves to be noted at this point that there exists a close correlation between software capabilities and the skills and experience of the users, a fact which should be taken into account in evaluating results.

Based on the first two discussion sessions, it was determined that the initial steps of this joint project should be dedicated to the accurate simulation of cracking phenomena in the door handle recess area. The expectations placed on the trials and the definability of boundary conditions by working group members were likewise established. Implementation of the requirements specified by the working group was handled by voestalpine Stahl GmbH on the experimental side and by ThyssenKrupp Stahl AG at the level of integration into the simulation environments. For this purpose, voestalpine Stahl AG agreed, firstly, to build a new punch into which the door handle recess of the current Golf A4 model, made available by VW as a CAD data record, was then integrated with a minimum tool radius of 2.2 mm, and secondly, to conduct the actual experimental investigations. The simulation was described after, and in awareness of the experiments conducted. However, the result of the experiments was not known to the participating working group members at the time of its first meetings.

EXPERIMENTS

The experiments were carried out with a double-acting 6300 kN hydraulic press using the modular trial toolset, which was of a pan-shaped design embodying the door handle recess (refer to Fig. 1). To prevent material draw in out of the blank holder area, the blank had to be spot-welded to a 2.5 mm gauge support sheet since the maximum blank holder force of the press (2500 kN) was insufficient to securely lock the material. In order not to introduce another unknown quantity (friction) into the benchmarking process, no lubricant was used between the punch and the sheet metal (each blank was wiped dry). For the simulation, a friction coefficient of μ = 0.14 was stipulated. The selected drawing speed was 94 mm/s.

For the deep drawing trials, grade DC04 ZE75/75 and HXT500X+ZE (DP500) sheets (both electrolytically galvanized) with a nominal thickness of 0.8 mm were selected and made available by voestalpine Stahl GmbH and ThyssenKrupp Stahl AG, respectively. The sheets measured 1000 by 1500 mm, their lengthwise dimension (coordinate *x* in Fig. 1) coinciding with the direction of rolling.

In pressing the blanks, the punch was initially lowered full stroke several times until tearing of the metal occurred (reproduction of the same result). The punch stroke was then progressively reduced until the part could be reliably produced without visible necking. Lateral drawing-in of the blank was neither visible nor measurable. The component was then measured on its top side (refer to Fig. 1) using the ARGUS automatic strain analysing system (Gesellschaft für Optische Messtechnik GmbH). The strain distribution was also analysed in marked sections and validated through sheet thickness measurements at defined points. The basic geometry of what was identified as the most critical location in the experiments is illustrated once again in Fig. 2 along a part of section B2*_S5. From the slightly curved line of the punch (left), the surface descends slightly in the door handle mounting area before continuing into the handle recess with a 2.2 mm die radius (right side).

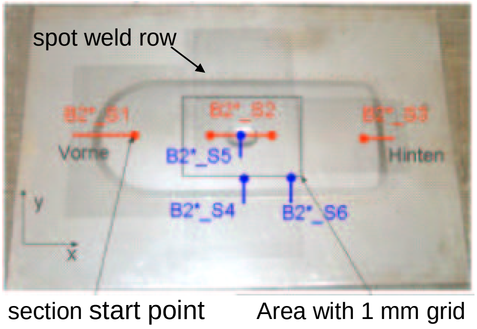

FIGURE 1. Model part and measured sections for simulation run B2*, shown on the component..

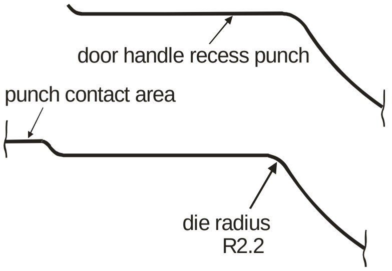

FIGURE 2. Part of the geometry for the door handle recess along section B2*_S2

SIMULATION BENCHMARK RUNS

Definitions for runs B1 and B2

It is assumed that, in their day-to-day work with FEM software for sheet metal forming simulations, users model components on the basis of given parameter specifications which are often derived from default parameters defined only once. However, since the result of a metal forming simulation will always depend on user/software interaction and on the user's experience with specific component categories, an attempt was made here to permit a separation of individual effects via the definition of the simulation runs.

Based on the experimental results and the requirements placed on the working group's objectives, the following boundary conditions were established for the definition of the simulation runs:

- On the one hand, the FEM programs employed should analyse the general feasibility of the components for a stated punch stroke (simulation runs B1*). This was carried out mainly with "normal", i.e., software-defined standard (default) parameters, and only partially with accurately specified parameters (e.g., friction, material).
- On the other hand, users were to adopt what in their experience were the best settings for the specific simulation software with the given component (simulation runs B2*).

The objective of the B1* simulation runs was to test the automatic parameter-finding capability of the examined programs with the selected geometry. Specifying a drawing depth at which necking or tearing had occurred in the trials, this simulation was to be used exclusively for inter-comparison of all results obtained. The main information provided by this simulation run was an evaluation of the simulation result regarding component feasibility.

The B2* simulation run, on the other hand, allowed the user to adapt specific software settings to the sheet metal forming problem on hand. The intention here was to enable each user to obtain the best result he could possibly achieve. The drawing depth defined for this simulation was one which, in the trials, had yielded acceptable components and permitted an appropriate strain analysis.

It was a feature common to all phase B1* und B2* simulation runs (cf. Table 1.) that the participating members were not aware of the trial results. To standardize the model definitions adopted by the working group members, the stress-strain curves of the materials used (B*A = DC04 and B*B = HXT500X) were supplied to them in the form of an extrapolated yield curve section or as a software-specific "material data" library file with the appropriate forming limit curve. To avoid deviations attributable to tool meshing, square or triangular tool grids were generated on the basis of standard parameters by Engineering Systems International GmbH (ESI) and made available in a Nastran format.

TABLE 1. Software version used in the simulations.

Software	Version	Simulation run
AutoForm Incremental	3.2	B1x, B2x
LS-DYNA	970	B1x, B2x
Pam-Stamp	2000/ 2G	B1x, B2x
Indeed	7.3	B1x, B2x
Optris	6.1 b	B2x

Results of the first simulation block

Simulation run B1A and B1B

Results obtained along the specified, defined sections with the AutoForm, Pam-Stamp, LS-Dyna and Indeed software were submitted for the first simulation run (B1*)

Overall, this data led to the following conclusions:

- Locations found to be critical in the trials were not identified as such by all participants with their software. In some cases, the critical nature of these points was greatly underestimated.
- The differences between the maximum strains calculated for critical points lay in the region of approx. two-tenths (0.2) of the major strain. This result varied greatly, even among users of the same program.
- The degree of major strain determined by the simulation tended to be greater with increasing number of mesh elements.

- The greatest difficulties were encountered in detecting and modelling the entrance radius of the door handle recess (generation of elements necessary to simulate this radius).

- The differences between modelling and experimental results arose with both materials at the specified drawing depth.

To draw a conclusion from the purely numerical simulation run, it may be observed that the programs, when employed with their standard parameter settings, will provide highly different assessments of the component's feasibility and manufacturing reliability. In many cases the tooling would have been cleared for production on the basis of the simulations conducted although its process capability was, in reality, inadequate.

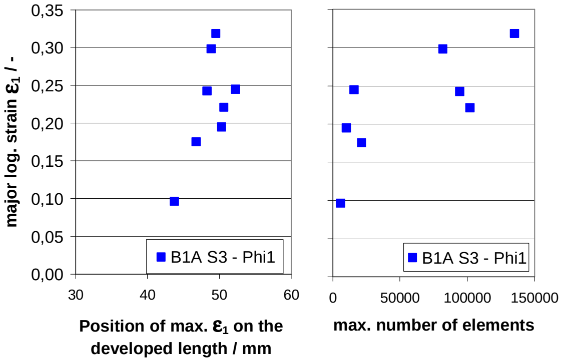

FIGURE 3. Results of run B1A calculated by all programs with material mild steel, evaluation of door handle recess radius in section B1*_S3, corresponding to section B2*_S5

Simulation runs B2A and B2B

Results obtained with the AutoForm, Pam-Stamp, LS-Dyna, Optris and Indeed software were submitted for simulation run B2*. This time, contrary to the B1* run, program users were free to adapt additional simulation parameters to the application at hand in order to improve the simulation result. However, to avoid interpretation problems, the tool mesh as well as a description of material characteristics (stress-strain curve, forming limit curve, friction) were once again specified. A further difference from simulation run B1* was that the drawing depth defined for the punch was based on the experimentally determined values. At a punch stroke of 40 mm, components drawn from the DC04 mild steel material exhibited barely onset of necking in the door handle recess; for the material dual phase 500, a 34 mm punch stroke was determined as the limit where necking was reliably prevented.

The simulation results were compared initially to one another and then with the experimentally determined strains along the six defined component sections.

The comparison of the numerical simulation results can be summarized as follows:

It emerged from the data submitted that the critical point was not identified by all software/user combinations. However, a certain improvement in results was detectable with regard to the critical door handle recess areas. This improvement often correlated with the use of a much increased number of elements (compare Fig. 4). Areas with smaller strains were modelled more accurately, across all software/user combinations, than areas with high local strain peaks.

Overall, the result quality was improved through a more precise prediction of actual (measured) deformations at the drawing depths used.

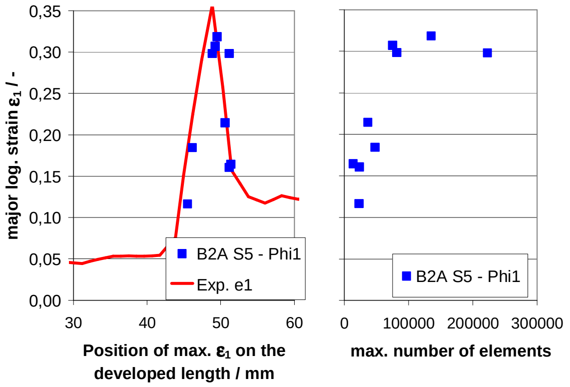

FIGURE 4. Results of run B2A (example): calculated maximum major strain at the door handle recess radius for all programs.

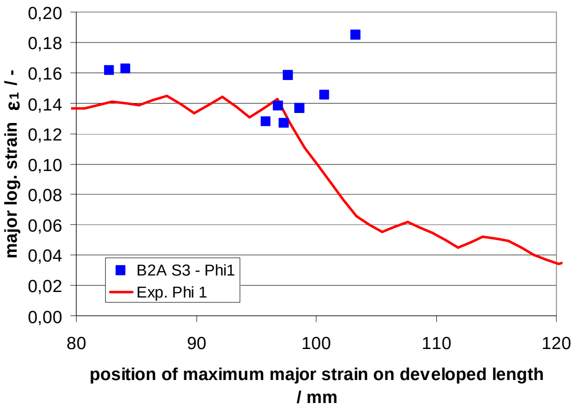

FIGURE 5. Result of run B2A (example) for a low-strain area (punch edge radius – section B2A_S3).

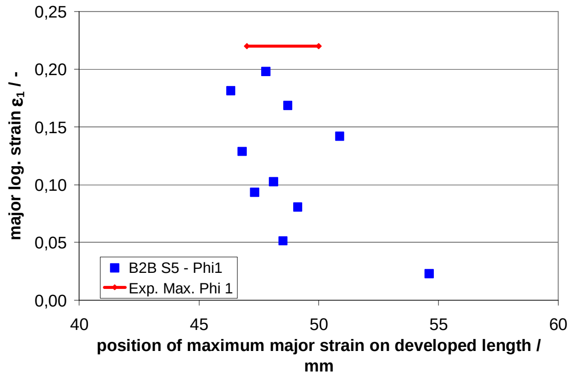

FIGURE 6. Result of simulation run B2B for section _S5 (example). Representing a reduced drawing depth compared to mild steel.

Overall, it emerged from the evaluation of the two simulation runs and their comparison with the trial data that the most critical point of the component – a section through the radius of the door handle recess in its short dimension – was underestimated by many users. Only in a few cases was the part rated as potentially critical. As had been the case in run B1*, the quality of the results obtained varied greatly, even where these had been obtained with the same software.

In the light of this observation, the software manufacturers were requested to examine the key factors influencing the simulation results and the differences between the models.

The following factors were identified as key parameters for these investigations:

- Influence of the strain rate on critical deformations (stability of the stress calculation process)
- Initial blank mesh (number and orientation of elements)
- Number of refining steps and selection of the refinement strategy/criterion (permitted steps, etc.).
- Evaluation of the influence of the friction coefficient (impact on result)
- Differences in simulation models used by the individual participants

RESULTS OF THE GENERAL PARAMETER OPTIMIZATION PROCESS

Based on the analysis of the simulations, the potentially influencing factors and parameters expected to yield an improved simulation result were selectively examined. In the present paper we shall limit our review to the two items outlined below.

Influence of strain rate sensitivity

The rationale for examining the influence of the strain rate was that, for both materials tested, the critical deformations determined were of a magnitude where yield behaviour may also be influenced by the material's strain rate sensitivity due to reduced hardening. This influence was evaluated on the basis of an identical simulation where the strain rate sensitivity for DC04 was assumed to be m = 0.025. The results of the two calculations are shown in Table 2. It emerges that the simulation in which the strain rate is <u>not</u> taken into account yields an overestimated maximum strain for all sections (values above 100 %), whereas the simulation in which the strain rate was considered tends to underestimate these values at the critical points (S2 and S5).

TABLE 2. Effect of considering strain rate on maximum strain in the sections (example mild steel).

Run B2A	Measurement	Simulation / measurement ratio (in %)	
Section point	**max. ε_1**	**Without strain rate**	**With strain rate effects**
S1	0,12	135	117
S2	0,21	106	92
S3	0,15	138	110
S4	0,07	121	101
S5	0,36	104	92
S6	0,07	114	97

Influence of initial mesh and refinement strategies/criteria on the simulation result

To verify the effects of mesh element size on the calculated deformations, simulation runs were carried out with and without adaptive mesh adjustment for the blank. It was found that an element edge length ≤ 1.5 mm was required to model the local deformation gradients at critical points. However, an adaptive refinement – with adjustment of parameters to the simulation problem – yielded only marginal differences from the simulations carried out with a constant mesh adjusted for the problem at hand.

It emerged from the investigations that the detection of major strains on small radii requires the choice of a fine initial mesh and a sufficient number of refinement steps. This approach will lead to better results if the refinement criteria themselves (e.g., angle criterion) are adapted as well. Only then will the simulation user be able to model local loads and hence resolve the local deformation distribution properly.

To estimate the effects of the tool mesh on the simulation result, the tool was re-meshed by DaimlerChrysler AG. This was followed by a series of simulation runs. The results of these simulations shall not be discussed here.

Based on all investigation done in this working group each individual software company produced comments and recommendations concerning the settings and procedures to be adopted with the respective programs. These settings were published in [1].

ACKNOWLEDGMENTS

We thank all partners in the German Deep Drawing FEM Working Group for fruitful discussions and the generated input. Especially the involved software companies for their support and help.

REFERENCES

1. Keßler, L. et al., „Verbesserte Versagensbewertung enger Radien durch angepasste Simulationsparameter am Beispiel einer Türgriffmulde," in 3rd Virtual Materials Processing Proceedings, Neue Materialien Bayreuth, 2004, pp. 107-126.

CONCLUSION

The investigations pursued to date by the working group with a view to improving the simulation of small radii have shown that in some sheet metal forming applications, closer attention to software settings on the part of simulation users is required. The wisdom of merely relying on the program's default values should be routinely questioned with regard to the specific geometry under simulation. As could be demonstrated, a selective adaptation of simulation parameters in the various software systems produced a significant increase in the accuracy with which failure probabilities were simulated.

The originally intended longer-term discussion and open experience-sharing process between all participants proved very helpful in achieving this goal. Now that the results have been published for the benefit of a wider circle of users, further topics in the field of metal forming simulation will be addressed. These will include issues for which no satisfactory modelling or interpretation approach has been developed to date.