Simulation of Sheet Metal Forming Using Solid Elements using ANSYS LS-DYNA©

Mikael Schill[1], Xinhai Zhu[2]

[1] Dynamore Nordic an ANSYS Company, Linköping, Sweden
[2]ANSYS, Livermore, USA

1 Abstract

Simulation of sheet metal forming has long been a fundamental application of ANSYS LS-DYNA, predominantly relying on shell elements under a plane stress assumption. While effective in many cases, situations arise where a full 3D representation becomes imperative, particularly when modelling thicker sheets or parts with tight radii. However, transitioning from shell to solid elements poses immediate challenges related to e.g. element size and the simulation model size.

In this paper, scenarios will be highlighted when it is motivated to shift to solid elements in sheet metal forming simulations and explore how novel functionalities in ANSYS LS-DYNA, such as 3D adaptivity, address the associated challenges. By examining specific examples, the benefits of employing solid elements will be illustrated, shedding light on their practical implications for industries reliant on accurate sheet metal forming simulations.

The aim of this work is to provide insights into when and how the utilization of solid elements can enhance the fidelity and predictive capabilities of sheet metal forming simulations, ultimately advancing the understanding and application of finite element analysis in manufacturing processes.

2 Introduction

Simulations of sheet metal forming have been a cornerstone of the sheet metal stamping industry for decades. The integration of these simulations has significantly reduced lead times and costs while enhancing the quality of stamped products. This technological advancement has instilled greater confidence within the industry, facilitating the adoption of innovative forming methods and materials.

A sheet metal forming process typically consists of several steps where a blank is transferred from one step to the next. Typical forming operations are drawing, trimming to remove scrap or punching holes, flanging and hemming, see Figure 1. The most critical steps are the drawing steps where a blank is clamped between a die and a binder and stretched over a punch. The stretching causes plastic deformation in the blank while the clamping constrains the movement of the blank to allow for just enough resistance to allow for a stretching without excessive thinning in the blank that could cause failure. On the other hand, a too low resistance causes material wrinkling due to an excess of blank material. Simulations are also employed to predict other critical forming process aspects such as tool forces, surface defects, and part tolerance due to elastic springback. Springback deformation is caused by the release of residual stresses as the blank is removed from the tooling and this deformation forces the part out of tolerance.

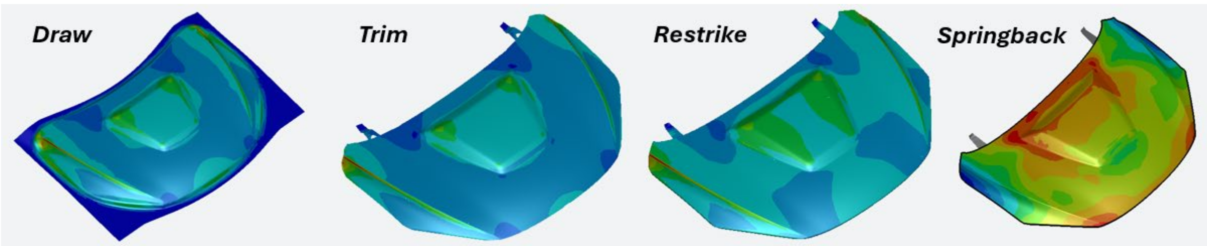


Figure 1: Sheet metal forming simulation process chain

Traditionally, the shell element has been the workhorse of these simulations due to its simplicity and computational efficiency. Shell elements are essentially derived from solid elements by assuming that the thickness is much smaller than the other dimensions. This allows the problem to be simplified by reducing the dimensionality and by incorporating appropriate kinematic assumptions. This simplification enables efficient modeling of thin structures while accurately capturing their mechanical behavior, such as bending and membrane deformations. The predominant shell theory used is the Reissner-Mindlin model which in ANSYS LS-DYNA are the underintegrated Belytschko-Tsay element and the fully integrated version type 16, see [1]. Typically, the underintegrated version is used for the forming simulations which is then switched to the fully integrated version for springback to avoid numerical convergence issues due to zero-energy modes which are present for underintegrated elements.

The Reissner-Mindlin kinematic assumption states that the cross-sectional fibres (through thickness) will remain straight during deformation but not necessarly perpendicular to the mid-plane, also known as first-order shell deformation theory, see Figure 2. This allows for the formation of transverse shear strains and it greatly simplifies the analysis of moderately thick shells. However, this assumption introduces certain limitations, particularly in situations where the shell or plate behavior deviates from this simplified model. The fact that the cross-section remains straight implies that the transverse shear strains are constant through the thickness. This might lead to less accurate results whenever the transverse shear deformation is not uniform. Research by Fleischer [2], among others, has demonstrated that this assumption can be violated under conditions involving small thickness-to-forming radius ratios or during complex bending and unbending processes. These deficiencies highlight the need for alternative approaches, such as the use of solid elements to more accurately capture the intricacies of sheet metal forming, especially in cases where three-dimensional stress states and detailed deformation patterns are critical.

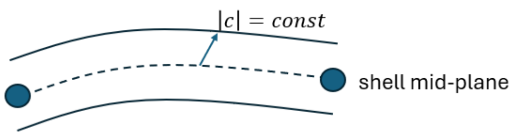


Figure 2: Reissner-Mindlin cross-sectional fibre assumption

The Reissner-Mindlin theory also states that the thickness fibre length remains constant. As a result, the thickness stretch is zero. In a forming scenario this means that the thinning and thickening of the blank is determined from the membrane straining

$$eps_{th} = -(eps_1 + eps_2)$$

Thus, any loading on the blank surface will not cause any thinning and in the event of thickening of the blank, it will not induce any through thickness stresses. This assumption could jeapordize the accuracy for forming situations with high contact pressures and thickening from excessive material draw-in.

Furthermore, shell elements operates under the assumption of plane stress conditions, where stresses in the thickness direction are considered negligible. While this assumption holds for many cases, especially when the stress state is membrane-dominant and the tool radii-to-thickness ratios are sufficiently large, it becomes less accurate in other scenarios. Allowing for a 3D stress state will increase the accuracy when a 3D-stress state is likely to form, e.g. during through thickness deformaion or bending but it will also allow for the usage of advanced material models where the anisotropy in the thickness direction is significant. Also, when the the strain localization limit is reached, a 3D stress state will form through the thickness. Here, the shell element will loose significant strength due to this deficiency while the solid element will provide a much more accurate response, see Figure 3.

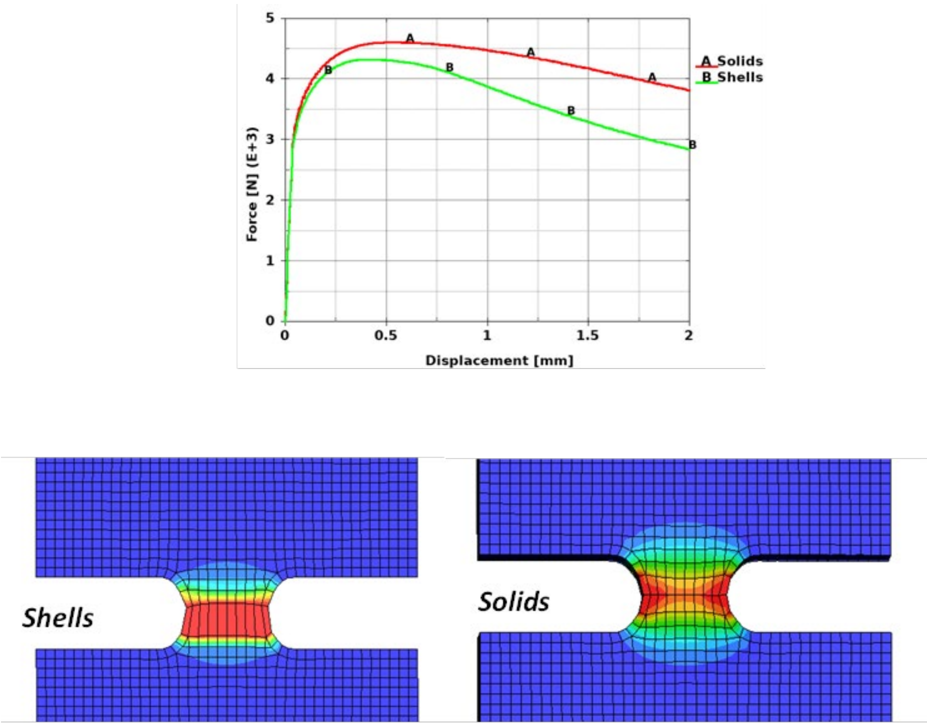


Figure 3: Shell element deficiency at strain localization limit

3 Shell element validity

In an attempt to compare shell and solid elements, two different forming scenarios are used, see Figure 4. The first forming process is a simple single action die with a blankholder force and the second one is a U-channel forming with different types of draw-bead to study the effect of bending.

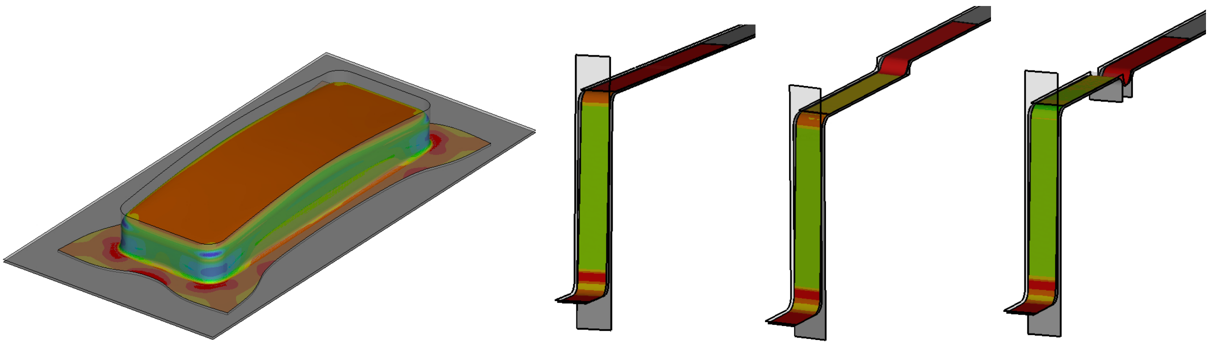


Figure 4: Single action die and U-channel model with draw-bead

The material used in the simulation is a conventional isotropic High Strength Steel, see Figure 5.

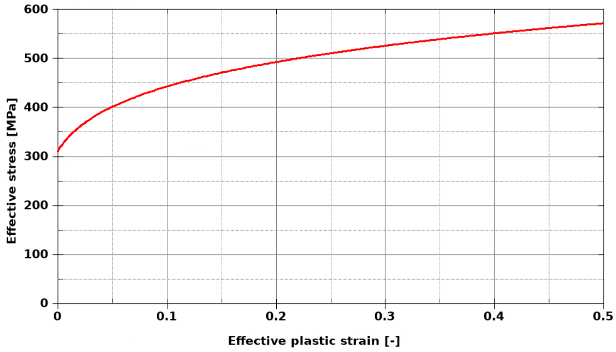


Figure 5: High strength steel material strain hardening curve

In a plane stress case, the stress components through the thickness should be zero. To quantify the magnitude of the deviation from a plane stress state, a Euclidian norm of the out of plane components is formulated according to

$$||d|| = \frac{\sqrt{\sigma_{zz}^2 + \sigma_{yz}^2 + \sigma_{xz}^2}}{\sigma_{y0}}$$

By using this measure, it is possible to visualize and quantify wherever there is a stress state that would necessitate a full 3D formulation.

3.1 Single action die

The purpose of the single action die study is to visualize the need for solid elements in a general forming scenario. To investigate the influence of the radius to thickness ratio, two different versions are used. The model data for the simulations is shown in Table 1.

	Single action die 1	**Single action die 2**
Draw die radius [mm]	5.5	2.8
Blank thickness [mm]	2	2
# Elements through the thickness	5	5
In-plane element size [mm]	2	1
Time step [s]	1.08e-7	1.08e-7
Blankholder force [tonnes]	40	18
# Blank Elements	84150	335705
Friction	0.125	0.125

Table 1: Single action die model data

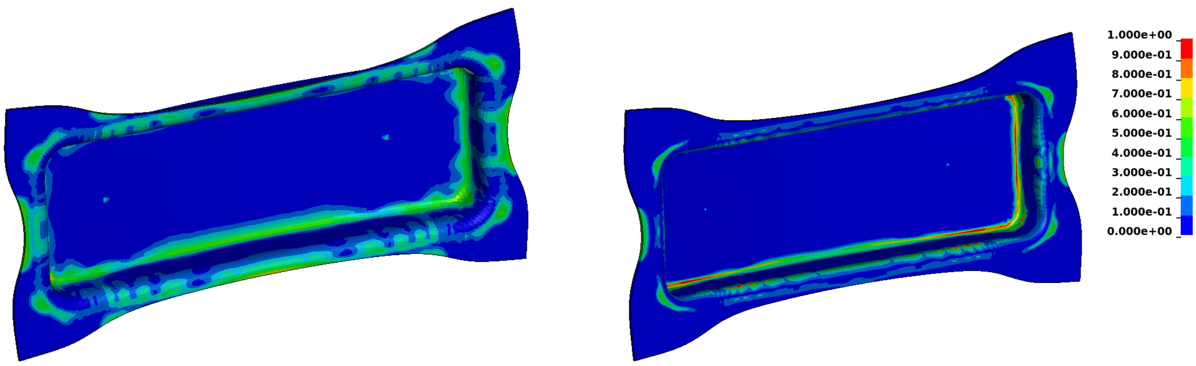


Figure 6: Plane stress deviation for single action die models

In Figure 6, the plane stress measure is presented for the final state of the two single action die setups. It shows that the deviation is quite large around the radius area which is expected since the contact pressure is likely to form a through thickness stress. Also, comparing the two models, it can be concluded that a decrease in draw die to thickness ratio will increase the plane stress deviation which follows the conclusion from previous studies. Turning the attention to the area under the blankholder, a substantial deviation is found in areas where the material is likely to thicken. The effect of the deviation is confirmed when studying the resulting thickness distribution. Comparing the findings of a shell and solid version it can be seen that the thickening under the blankholder is smoother for the solid element model while the shell model has higher maximum thickness in concentrated areas of larger plane stress deviation, see Figure 7.

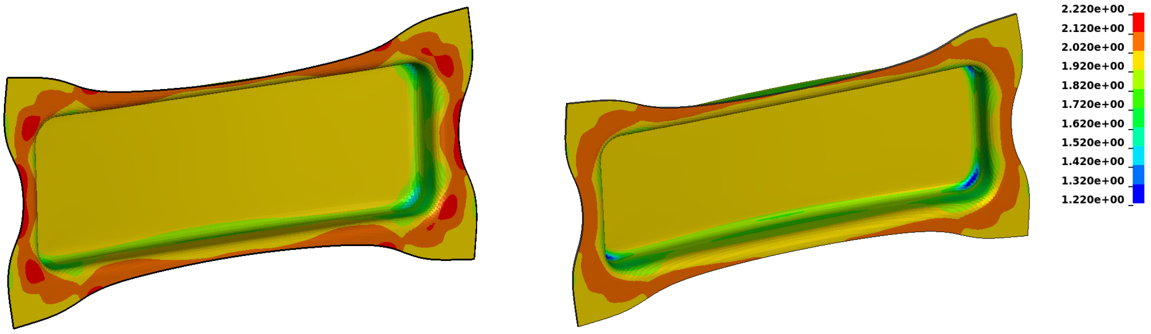


Figure 7:Thickness measurement for single action die model 1, Shell (left) and Solid (right)

The same conclusion can be drawn for the second single action die model with a decreased radius to thickness ratio. This model also shows a more distributed thickness under the blankholder. Further, both the shell and the solid element model predicts an area in the corner with high thinning, see Figure 8. This is expected due to the plane strain deformation originating from the restrained material draw-in in the corner area. Also, the solid element models in both forming scenarios predicts a decreased thickness in the punch corner due to the high contact pressure which follows the deviation findings, see Figure 9.

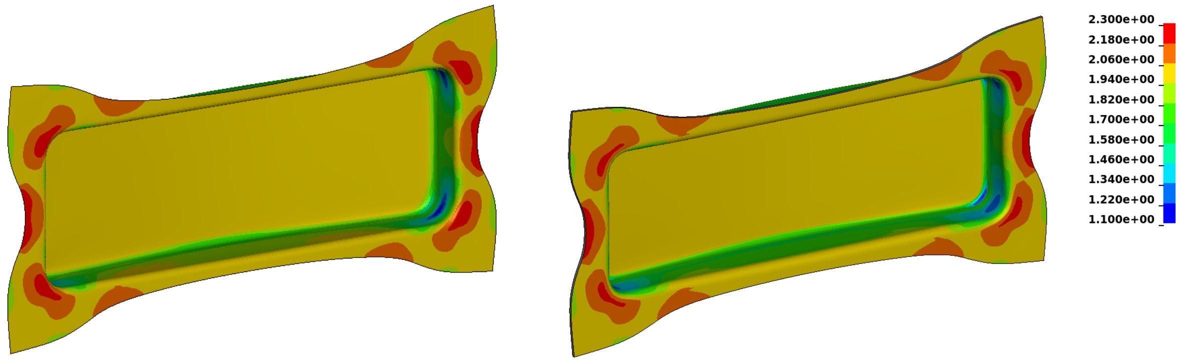


Figure 8: Thickness measurement for single action die model 2, Shell (left) and Solid (right)

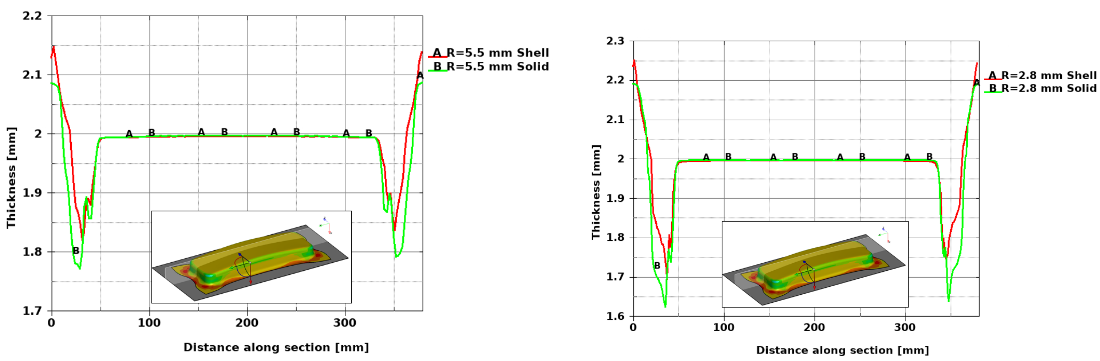


Figure 9: Shell and solid thickness plots for single action die models

3.2 U-channel model

The purpose of the U-channel models is to study the effect of bending on the shell element validity. The models are set up with different number of bending and unbending scenarios starting with a simple two-piece draw without draw bead and ending up with a full draw bead. The models are set up to have an effective plastic strain of approximately 11 % in the middle integration point in the U-bend flange. Also, for each model, the material draw-in are matched to make sure that the restraining force is the same for the shell and solid element models. The draw bead radii are 7 and 8 mm respectively and the draw-die radius is 10 mm. The blank thickness is 2 mm which yields a radius to thickness ratio of 3.5 to 5.

U-channel model without draw bead

As the blank is drawn across the radius it experiences one cycle of bending and unbending. Figure 10 shows the plane stress deviation measure along the draw die radius. Although the radius to thickness ratio is quite high in this case and the contact pressure can be considered to be low, there is a deviation from the shell assumption especially in the lower fibre along the draw die radius. In this case, the through thickness resulting stress state along the drawing direction is evaluated in the U-bend flange, see Figure 11.

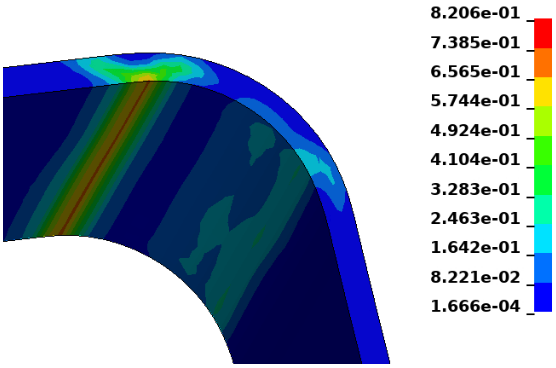


Figure 10: Shell assumption deviation for U-channel model without draw-bead

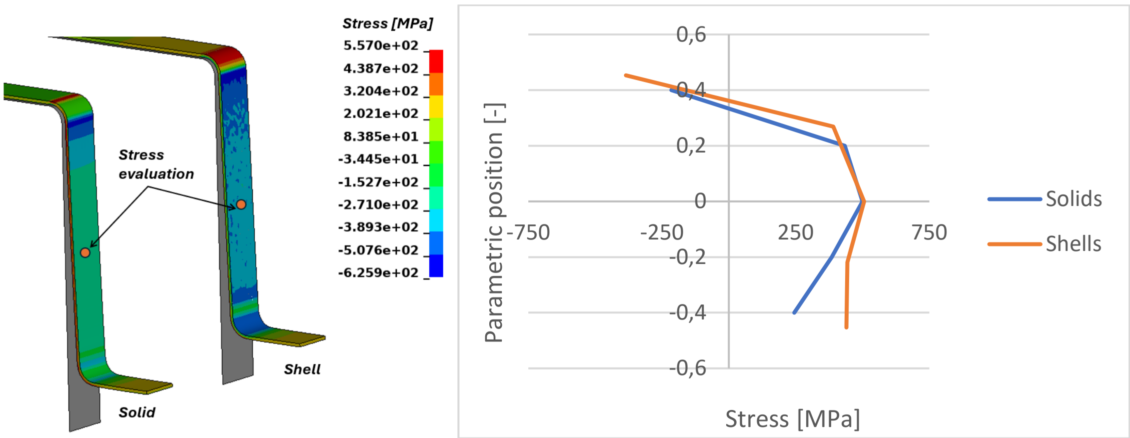


Figure 11:Drawing direction stress for U-channel model without draw-bead

It can be seen from the stress analysis that the stress state in the membrane layer is very similar between the shell and the solid model which confirms the shell validity for formability analysis. However, the stress state deviates in the lower fibre which complies well with the deviation measurement. This indicates that the bending and unbending in the draw die introduces an error and although this does not jeopardize the formability results it could affect the resulting springback deformation.

U-channel model with a blankholder radius step

In this model, a radius step is added to the die and blankholder to examine the effect from two bending cycles. In this case, the stresses are evaluated in a part of the blank that has passed through the blankholder radius step but not draw die radius. As can be seen in figures 12 and 13, the bending and unbending affects the stress state of the material passed through the radii. In this case, both the outer and inner fibres show a different stress state for the solid case compared to shells. The mean surface is however still unaffected, and the overall stress profile shows the same overall distribution through the thickness.

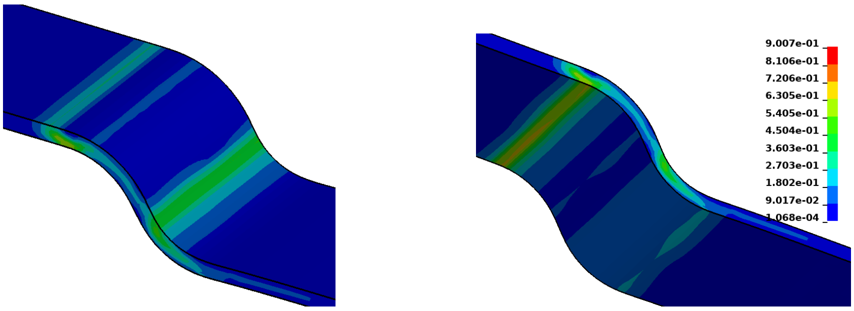


Figure 12: Shell assumption deviation for U-channel model with a blankholder radius step

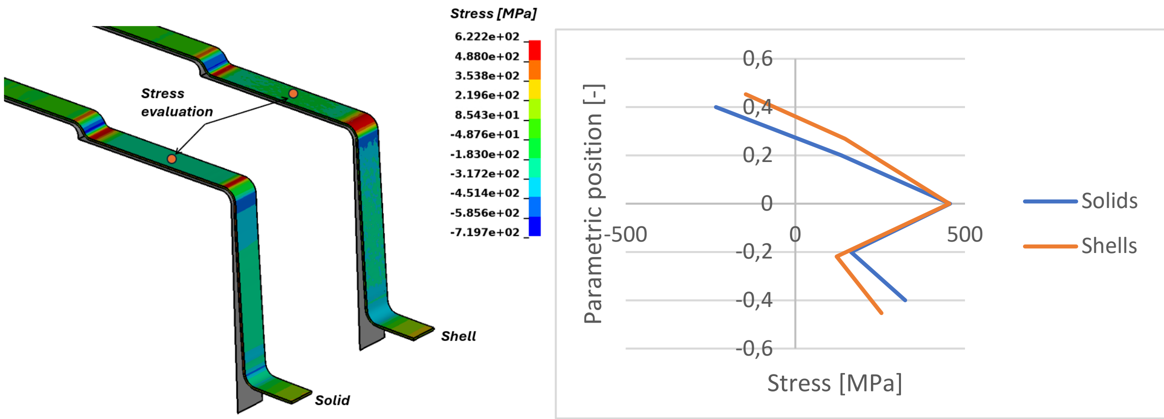


Figure 13: Drawing direction stress for U-channel model with a blankholder radius step

U-channel model with a draw bead

In this model, the blank movement is restrained by a full draw bead resulting in 3 bending and un-bending cycles. The shell element assumption deviation and the stress in the drawing direction through the thickness is shown in figures 14 and 15. The results indicate that each bending adds to the difference in stresses which is in this case quite significant, but the membrane layer shows similar results between the shell and solid representations.

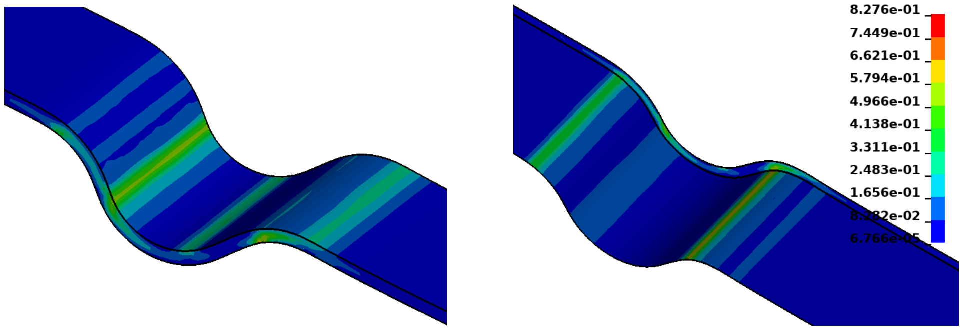


Figure 14: Shell assumption deviation for U-channel model with a draw-bead

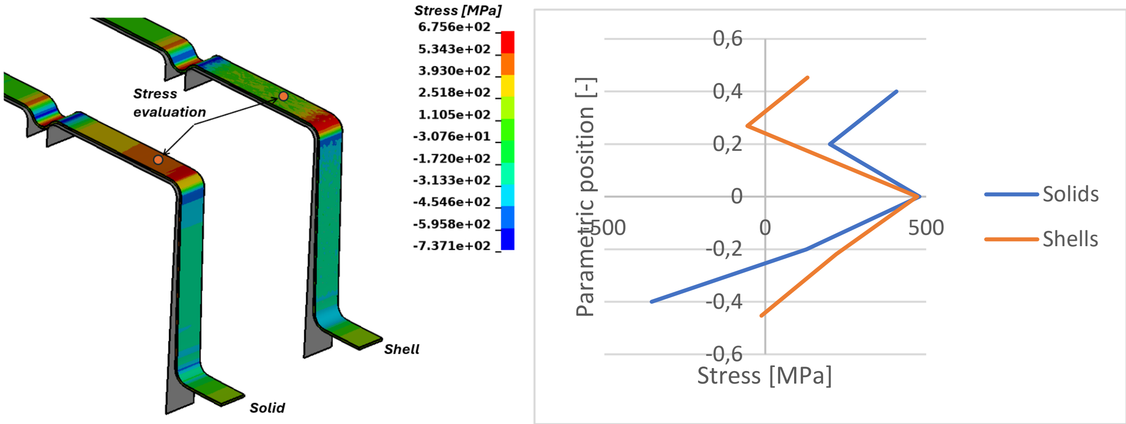


Figure 15: Drawing direction stress for U-channel model with a draw-bead

Based in the single action die and the U-channel model study it can be concluded that shell elements are very efficient and sufficiently accurate for most sheet metal forming simulations. However, there are scenarios when the assumptions of plane stress and constant out of plane shear strain can reduce the accuracy of the results. It is known that a decrease in radius to thickness ratio necessitates a need for a full 3D stress state representation. However, this is also true for general forming scenarios with large material draw in and out of plane bending as in the case of draw beads. Although the shell elements will remain as the work horse for formability simulations, having the possibility to switch to a full 3D stress state could be very useful for simulation scenarios where an extra accuracy is needed e.g. for small radius to blank thickness ratio or springback analysis.

4 Simulation of SMF using solids

4.1 Simulation timestep

One of the major challenges when switching to 3D solid elements is the critical timestep. In ANSYS LS-DYNA, the critical timestep for explicit time integration is determined based on the stability criteria of the numerical integration scheme used, specifically the Courant-Friedrichs-Lewy (CFL) condition. The critical timestep is the maximum timestep size that ensures the stability of the simulation. If the timestep exceeds this value, numerical instabilities such as non-physical oscillations or divergence may occur. The CFL condition dictates that the time step must be small enough to ensure that a disturbance (such as a stress wave) cannot travel further than the smallest characteristic length of the elements in a single timestep. Thus, the critical timestep can be expressed as

$$\Delta t \leq \frac{L_{min}}{c}$$

where L_{min} is the minimum characteristic length of an element and c is the speed of sound of the material. In the case of solid elements in sheet metal forming simulations, the minimum element length is most likely determined by the discretization through the thickness. For shells it is related to the number of elements necessary to capture the stress and strain gradients across the smallest radius of the part.

The number of integration points through the thickness of a shell element does not affect the critical timestep.

One of the ways to remedy this is to use mass scaling which is a technique used to artificially increase the critical timestep. Increasing the density of the material will decrease the speed of sound and by that increase the critical timestep. In ANSYS LS-DYNA, this is done automatically where the solver adds the necessary amount of mass to meet a specified timestep. Table 2 shows a critical timestep and added mass comparison between shells and solids for a steel material with an in-plane element length of 1 mm and 5 elements through the thickness.

While mass scaling increases the timestep, it also changes the dynamic response of the model. The additional mass can alter the inertia properties and lead to non-physical results, especially in dynamic problems where accurate representation of mass and inertia is critical. Sheet metal forming processes are generally considered to be quasi static, thus the inertia properties of the blank and tools should not influence the results. However, if the time- and mass scaling of the simulation is large enough, the inertia properties will eventually influence the blank behaviour which will lead to unphysical results.

Looking more closely at the critical timestep, it is related to the maximum eigenfrequency of the model and by adding mass of the corresponding element that eigenfrequency is decreased. The problem with conventional mass-scaling is that although it is the highest frequency that limits the timestep, increasing the mass of the element will affect all eigenfrequencies, including the rigid body motions which are the ones generating dynamic effects. Selective mass-scaling is an alternative scheme in ANSYS LS-DYNA where the mass-scaling only affects the critical (highest) eigenfrequencies. This method allows for very large mass-scaling without adding dynamic effects. However, selective mass-scaling will result in a non-diagonal mass matrix which will seriously affect the simplicity and speed of the explicit method. As a result, the mass-matrix needs to be diagonalized and the solution time for this increases with the amount of mass-scaling.

It is recommended to use conventional mass-scaling as a default due to the speed and simplicity and switch to selective mass-scaling for simulation processes that are likely to become dynamic or it the inertia effects of the conventional scheme is found to affect the simulation result.

	Shells		**Solids**	
Blank thickness [mm]	**Δt [s]**	**% mass scaling (Δt =0.5e-6 s)**	**Δt [s]**	**% mass scaling (Δt =0.5e-6 s)**
0.8	1.65E-07	15.9	2.25e-8	530.
1.5			4.2e-8	150.
2.0			5.6e-8	83.9

Table 2: Critical explicit timestep and mass scaling comparison between shells and solids.

4.2 Model size

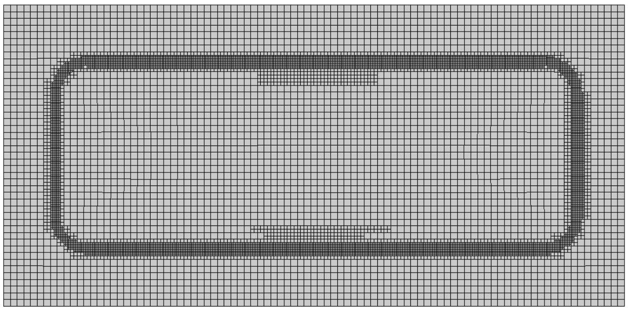
When discretizing the thickness of a blank in finite element analysis, the model size increases in proportion to the number of elements through the thickness. This can significantly raise computational demand. ANSYS LS-DYNA's explicit solver, known for its efficient memory usage, can typically handle this increased element count without causing hardware issues.

However, challenges arise when using the implicit solver, which requires assembling and inverting the full stiffness matrix, a process that is highly memory intensive. This can lead to potential memory bottlenecks, particularly during simulations involving gravity and springback stages.

To mitigate these issues, the parallel processing capabilities of MPP (Massively Parallel Processing) ANSYS LS-DYNA become crucial. This process involves splitting the model into smaller pieces, which are then distributed across several computer cores. By running large simulations in parallel and distributing the computational workload, MPP ANSYS LS-DYNA significantly reduces both the memory and computational burden on individual processors. This makes it feasible to run large-scale simulations more efficiently and effectively.

Another solution to the model size issue would be to decrease the number of elements. In sheet metal forming simulations using shells, this is commonly done using element adaptivity. Here, the blank is meshed using relatively large elements which are subsequently split into smaller elements wherever it is needed in certain areas that experience higher deformation and strain, such as sharp corners or regions with complex geometry. Since the mesh is only refined where needed, the overall number of elements and the computational load are minimized. This leads to faster simulations, allowing for quicker iterations and design optimizations.

When using adaptivity, it's crucial that adaptive refinement is performed before significant deformation occurs; otherwise, the simulation may need to step back and recalculate. This occurs because remeshing after deformation has already introduced discretization errors. Look-ahead adaptivity addresses this issue by predicting areas of high deformation and strain, allowing the mesh to be refined in advance, thus preventing the introduction of errors. In ANSYS LS-DYNA, the look-ahead adaptivity functionality, initially developed for shells, has now been extended to solid elements, see Figure 16. As the tools approach the blank, the contact algorithm predicts high deformation areas based on a user-defined distance. The elements are then split multiple times according to user-specified minimum element sizes or maximum refinement levels. This adaptivity occurs in the plane of the blank rather than through the thickness, see Figure 16. Extending this functionality to solid elements enhances both accuracy and predictive capabilities, with minimal impact on computational efficiency, pre-processing, and post-processing efforts.



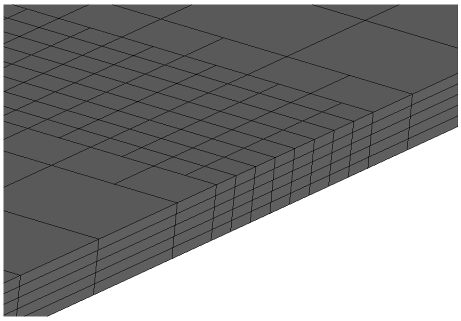


Figure 16: Look ahead adaptivity (left) and 3D-adaptivity for solid elements (right)

4.3 Material modeling

One question that arises when moving to a 3D solid formulation is how to utilize the enhanced stress description from a material modeling point of view. Using an anisotropic material model such as the Hill'48 criteria for shells reduces the formulation to a plane stress case with 4 parameters. In theory, extending to a full 3D stress state would allow the full Hill'48 criteria with 6 parameters to be used, see below

$$F(\sigma_{22}-\sigma_{33})^2+G(\sigma_{33}-\sigma_{11})^2+H(\sigma_{11}-\sigma_{22})^2+2L\sigma_{23}^2+2M\sigma_{31}^2+2N\sigma_{12}^2=\sigma_f^2$$

In practice, this would require tests through the thickness which are not easily found since most of the experiments for anisotropy and formability are based on in-plane testing such as uni-axial, biaxial tension and Nakazima tests. Consider the anisotropic material with experimental data according to Table 3. Here, σ_{ii} is the yield stress from a uniaxial tensile test with angle ii to the rolling direction. The Lankford coefficients R_{ii} is the corresponding ratio between the width and thickness strain ($^{\varepsilon_{width}}/_{\varepsilon_{thick}}$) from the angled uniaxial tensile tests. The biaxial data $\sigma_b$and $R_b$is the biaxial yield stress and in-plane strain ratio ($^{\varepsilon_x}/_{\varepsilon_y}$) from a biaxial bulge test or a layered compression test where several coins are stacked and compressed through the thickness, see Figure 17. By formulating the resulting stress deviator for the two cases it can be shown to be equivalent material tests.

σ_{00} [MPa]	σ_{45} [MPa]	σ_{90}[MPa]	R_{00}	R_{45}	R_{90}	σ_b	R_b
45	40	50	0.8	1	1.2	55	0.9

Table 3: Anisotropic material data

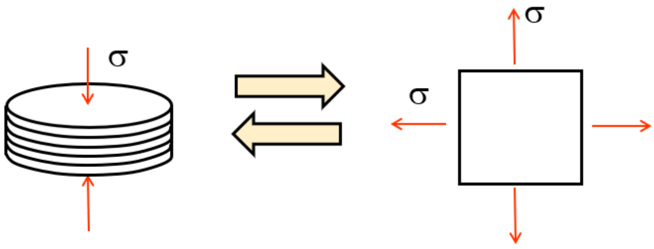


Figure 17: Stacked coin test and balanced biaxial stress state

When fitting the material data of Table 3 to the Hill'48 material model, no more than 4 tests can be fitted even though the model has been extended to a 3D stress state. In sheet metal forming simulations fitting strain data is preferred over stress. Solving for σ_{00} and R_{00}, R_{45} and R_{90}yields *F*=0.37, *G*=0.555, *H*=0.444 and *N*=1.3888. Parameters *L* and *M* are undetermined in this case and assumed to be isotropic, thus *L*=*M*=1.5. A comparison between the experimental data is presented in Table 4. It can be seen that the fitted parameters have excellent agreement as expected. This is also true for σ_{90}, but the $\sigma_{45}$and the biaxial experimental points shows poor agreement.

	σ_{00} [MPa]	σ_{45} [MPa]	σ_{90}[MPa]	R_{00}	R_{45}	R_{90}	σ_b	R_b
Experiments	45	40	50	0.8	1	1.2	55	0.9
Hill	45	46.8	50	0.8	1.0	1.2	46.8	1.5

Table 4: Experiment and Hill 48 agreement

In ANSYS LS-DYNA, the plane stress material model according to Barlat et al. [3] (*MAT_YLD2000) has been extended to 3D according to Dunand et al. [4]. The resulting yield criteria is convex and pressure independent and it reduces to the original 2d model for the plane stress case. This also implies that the same parameters used for the plane stress case can also be used to calibrate the 3D stress version and no additional tests through the thickness are required. Table 5 presents the resulting fit for the extended YLD 2000 model and Figure 18 show the $\sigma_{ii}$and R_i. It can be seen that excellent agreement is found for all tests and that commonly found in-plane material data can be used to fit the material model.

	σ_{00} [MPa]	σ_{45} [MPa]	σ_{90}[MPa]	R_{00}	R_{45}	R_{90}	σ_b	R_b
Experiments	45	40	50	0.8	1	1.2	55	0.9
YLD2000 3D	45	40	50	0.8	1.0	1.2	55	0.9

Table 5: Experiment and YLD 2000 agreement

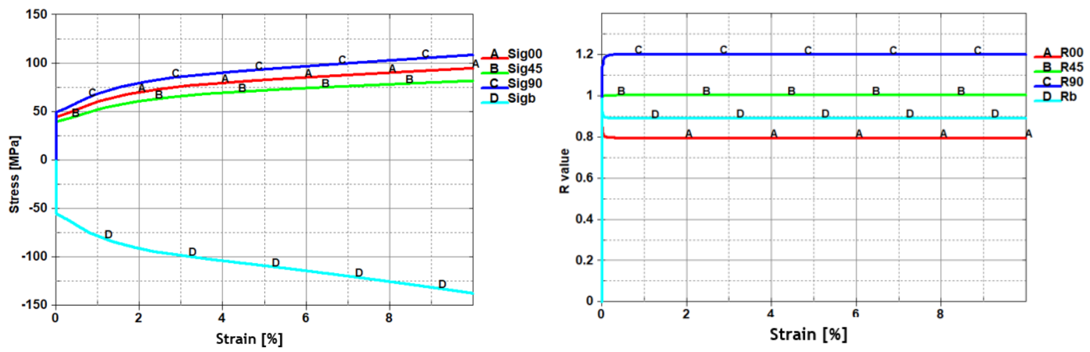


Figure 18: σ_{ii} and R_i with YLD 2000 3D extension

4.4 Element formulation

When using shell elements, it is generally recommended to select an in-plane element size that corresponds to 3-5 elements along a 90-degree arc of the blank. This guideline typically strikes a good balance between accuracy and computational efficiency. When transitioning from shell to solid elements, it is advisable to follow a similar guideline. However, depending on the thickness of the blank, this can lead to scenarios where the solid elements exhibit poor aspect ratios due to the significant disparity between the in-plane and thickness dimensions.

Poor aspect ratios magnify the issue of shear locking, a well-known problem with linear solid elements. Shear locking occurs when elements become overly stiff in bending-dominated problems, leading to inaccurate simulation results. This problem is particularly pronounced when element aspect ratios are unfavorable. In bending-dominated forming scenarios, such as those involving draw-beads, it may be necessary to use alternative element formulations to mitigate shear locking.

In ANSYS LS-DYNA, fully integrated solid elements (denoted as -1 and -2) provide a viable solution. These elements are still linear but incorporate an assumed strain approach to prevent shear locking. While both -1 and -2 elements are similar, the -1 version is implemented with a slightly less rigorous approach, resulting in lower computational demands.

Table 6 presents the results of a elements per radius and aspect ratio study. Here, the lifting force of the binder in the U-channel with a draw-bead case, see Section 3.2. The comparison is made for the underintegrated shell element type 2, the underintegrated solid element type 1 and the fully integrated solid element type -1, see [1]. It can be seen that there is a clear influence on the lifting force from the aspect ratio for the underintegrated solid element which clearly should not be used for large aspect ratios and bending dominated applications. In these cases, the fully integrated type -1 solid element should be used which shows only slight variations in the binder force even for larger aspect ratios. When reducing the number of elements per radius a deviation of the binder lifting force can be seen which is a result from a poor representation of the bending deformation across the drawbead.

#Elem/Radius	Aspect Ratio	Shell (2) [N]	Solid (1) [N]	Solid (-1) [N]
31	1	1.11e4	1.11e4	1,11
12	2.5	1.10e4	1.49e4	1.10e4
6	5	1.09e4	2.8e4	1.13e4
4	7.5	1.22e4	3.72e4	1.34e4
3	10	1.32e4	4.14e4	1.78e4

Table 6: Binder uplift force in draw bead model for different elements and aspect ratios

4.5 Discussion

When considering using solid elements instead of shell elements for sheet metal forming it is more to consider than e.g. accuracy and computational resources. As explained in the introduction, a deep drawing process stage is usually one process stage in a series of many. Firstly, the complete process needs to be set up and defined in a pre-processor. The software must be extended to allow for solid element meshing of the blank and accommodate the increased model size which could be considerate. As the simulation is finished, the results need to be extracted and visualized for the die designer in terms of e.g thinning or thickening and FLD: This is an easy task for the shell elements since it is inherent in the shell results while it will involve additional evaluation steps in the solid case.

The pre- and processing steps are usually accommodated in a tailored software where the user is guided through the setup in an environment which is recognizable for the die-designer. In the case of ANSYS LS-DYNA, the software for setting up sheet metal forming process simulations is ANSYS Forming. This tailored software also include additional functionality which is unique for the simulation of sheet metal forming processes such as blank or trim line development, springback compensation, clamping with deviation measurement and hot-forming, all of which has to accommodate for solid elements.

Focusing on the FE solver, there are a lot of functionalities necessary to perform a sheet metal forming simulation. These include e.g. scrap-trimming and hole piercing and tailored contacts which are optimized for speed. Also, for a hot-forming process this also must work for thermo-mechanical coupled simulations. Here the user can benefit from using a general FE solver such as ANSYS LS-DYNA where a lot of the functionality is available both for solid and shell elements.

5 Summary

Shell elements have been the cornerstone of sheet metal forming simulations for decades. There are several reasons for this. One is the ease of use since the bending behavior of the element is treated simply by adding integration points through the thickness. This limits the number of elements in the models which reduces the simulation time as well as the time spent on model handling such as pre- and post-processing. Also, if an explicit solver is used, the minimum timestep is determined by the smallest element length, and if the thickness of the blank is discretized this length is likely to occur through the thickness. Thus, a smaller timestep would be expected compared to a shell element model. Another important reason for using shell elements is that material characterization for shell elements is quite simple since it can be based on in-plane tests only which are relatively cheap and common to do. Extending the models to 3D often requires characterization through the thickness which are non-standard and quite complicated. On top of that, for many of the sheet metal forming applications, the shell elements provide an acceptable accuracy since the deformation is membrane dominant and the objective of the simulations is mainly focused towards predicting formability such as thinning and wrinkling.

However, shell elements have certain deficiencies that could affect the accuracy of the sheet metal forming results. Firstly, the shell element comes with a plane stress assumption which assumes the stress through the thickness is zero. While true for many applications, it can be shown that as the radius to thickness ratio decreases or as the straining of the material is localized, this assumption is violated. The plane stress assumption also limits the material modeling accuracy, where a full 3D model allows for a more accurate description. Another assumption for the shell element is that the thickness fibres remain straight. This results in a poor description of the through thickness shear deformation.

This paper has shown that there are several scenarios in general sheet metal forming simulations where a full 3D description of the stress state would be necessary. Typical situations are where the blank has high contact pressure, blank thickening, small thickness to radius forming or considerable bending deformation such as in a draw-bead or a draw-die radius. Also, it has been shown that the common obstacles like model size and time step size can be facilitated by using parallel processing, selective mass-scaling and 3D mesh adaptivity where the elements are refined in the areas where large deformations are likely to occur. Furthermore, by extending material models for shells, in-plane material testing can be used to describe the material behavior in a 3D- stress state.

Shell elements will remain a big part of sheet metal forming for a foreseeable future but the improvements in pre- and postprocessing, solver speed, material modeling and solver functionality will gradually limit the step for the sheet metal forming industry to make the switch to a full 3D description. This will increase the accuracy of the simulations and allow for an increased confidence in simulation-based design and expand the application area for sheet metal forming simulations.

6 Literature

[1] LS-DYNA KEYWORD USER'S MANUAL LS-DYNA R13, Volume 1", Livermore Software Technology (LST), An ANSYS Company, 2021

[2] Fleischer, M.: "Safeguarding the virtual process chain for subsequent operations in forming technology.", Dissertation, Chair of Statics, Technical University of Munich, Shaker Verlag, Aachen, 2009

[3] Barlat, F., Brem, J.C., Yoon, J.W., Chung, K., Dick, R.E., Lege, D.J., Pourboghrat, F., Choi, S.H., Chu, E., "Plane Stress Yield Function for Aluminum Alloy Sheets – Part 1: Theory", Int. J. Plast. 19, 2003, 1-23

[4] Dunand, M., Maertens, A.P., Luo, M., Mohr, D., "Experiments and modeling of anisotropic aluminum extrusions under multi-axial loading – Part I: Plasticity," Int. J. of Plasticity, Vol. 36, 2012, 34-49