ADAPTIVE FINITE ELEMENT SIMULATION OF SHEET METAL FORMING PROCESSES USING GRADIENT BASED INDICATORS

Ramin Moshfegh
Research Associate

Xiangdong Li
Associate Professor

Larsgunnar Nilsson
Professor

Ramin Moshfegh
Division of Solid Mechanics
Linköping Institute of Technology
Linköping, SE-581 83, Sweden
Tel: +46 13 28 11 22
Email: rammo@ikp.liu.se

Keywords:
Adaptivity, Error indicators, LS-DYNA, Nonlinear finite element, Sheet metal forming.

ABSTRACT

Two mesh refinement indicators based on the gradients of effective stresses (GSIG) and effective plastic strains (GEPS), respectively, are proposed for adaptive finite element analysis of the large deformation, quasi-static or dynamic response of shell structures.

The mesh refinement indicators are based on equi-distributing the variation of stresses or plastic strains over the elements of the mesh. A program module is developed and implemented in the nonlinear explicit finite element code LS-DYNA. This module provides element-wise refinement evaluations so that selective mesh refinements are carried out in regions of the mesh where the values of local indicators exceed a user-specified tolerance.

The FE model of a conventional deep drawing process is used as numerical model, including both material and geometrical nonlinearities, in order to demonstrate the versatility of the two refinement indicators.

Four different refinement indicators, based on angle change, thickness change, GSIG and GEPS based are applied in this investigation. To verify the numerical results against experiments, the anisotropic low carbon steel, FEP04, is used as a reference material. The numerical results are compared with experimental results regarding the thickness distribution versus cup height, effective plastic strain in the deformed sheet and punch force.

It is shown that the new proposed indicators can identify finite elements, which have high gradients of stresses or strains so that the mesh is refined in the regions undergoing the most severe deformations and the numerical results are improved.

INTRODUCTION

The numerical simulation of sheet metal forming problems inevitably involves large deformation, contact/friction and nonlinear constitutive behavior. As the analysis of such highly nonlinear processes usually is computational demanding, it is desirable that the computational power is focused particularly on those parts of the finite element mesh that undergo the most severe deformations. Moreover, the accurate modeling of the final, often very complex, shapes requires the use of a fine mesh, while a coarse mesh with relatively few elements is sufficient at early stages. Therefore, it is advantageous to update the finite element mesh according to the behavior of the solution, to ensure that the mesh is sufficiently fine in regions of high gradients of stresses and/or strains, i.e., to perform an adaptive analysis.

The development of error estimates and adaptivity has reached maturity in the study of linear elliptic boundary value problems. Two types of error estimation techniques, the residual based methods introduced by (Babuska and Rheinboldt 1978) and the post-processing based methods by (Zienkiewicz and Zhu 1987, 1992), are widely used. For surveys of the existing literature and state-of-the-art reviews, see (Ainsworth and Oden 1997) and (Eriksson et al. 1995). Due to the inherent complexity of the nonlinear problems, studies on error estimation and adaptivity on these problems are far from complete. However, some advances have been recorded for certain specific problems and publications have shown a rapid growth in recent years. Among numerous contributions, the works by (Peric et al. 1994) and (Lee and Bathe 1994) for elasto-plasticity, the work by (Wiberg and Li 1994) for dynamics and the work by (Belytschko et al. 1989), (Okstad and Mathisen 1994) and (Bonet 1994) for shells may be mentioned.

Recent versions of the nonlinear explicit finite element code LS-DYNA (Hallquist et al. 1999) allow us to perform adaptive analysis of shell problems, in which two simple refinement indicators based on geometrical relative deformations, angle and thickness, are used.

Recently, two mesh refinement indicators, one based on the gradient of the effective plastic strains (GEPS) and the other one based on the gradient of the effective stresses (GSIG), were presented by the authors, see (Li et al. 1999). The refinement indicators are based on the equi-distribution of the stresses or plastic strains variation over the elements in the FE mesh. The idea behind the refinement strategy is motivated by the fact that FE solutions are generally less accurate in regions where high gradient of stresses and/or plastic strains exist. A program module to perform adaptive refinements based on these two indicators has been developed and implemented in LS-DYNA, see (Li et al. 1999) for detailed descriptions of the two gradient based refinement indicators and some numerical examples.

In this paper, the authors further investigate the performance of these two refinement indicators for sheet metal forming application. In particular, the FE simulation of a Conventional Deep Drawing (CDD) process is considered. The numerical results by using the gradient based indicators are compared with the results obtained by using two different adaptive refinement indicators in LS-DYNA, i.e., the angle and thickness based indicators, respectively. Additionally, the FE results are also compared to available experimental results.

ADAPTIVITY IN LS-DYNA

The aim of adaptive methods is to get more accurate FEM results by increasing the mesh density in areas where it is necessary and to obtain a FEM solution with a prescribed local accuracy. Traditional implementations of FEM, such as in LS-DYNA, are referred to as *h*-adaptivity options, where *h* is the characteristic size of the individual element. If such a code includes some automatic method for altering *h* in response to the characteristics of a specific problem, the method is said to be *h*-adaptive. The act of increasing the number of elements, i.e., reducing the characteristic size is called refinement.

By using the *h*-adaptivity option each adapted element, is divided into four elements, which procedure makes the mesh denser. Two new elements have one neighbor element along the common edge. This irregularity can be allowed in the displacement field if adding constraints enforces continuity. Each irregular edge contains a *new* middle node that does not contain any new degree of freedom. In order to preserve continuity with linear shape functions, the solution at this node must be equal to the average of the values at the edge's endpoints.

Two types of refinement indicators have been implemented in the LS-DYNA program. These are the angle and the thickness indicators. The angle indicator is a deformation-based indicator using the angle change, i.e., it checks angular deformations between elements in plane and out-of-plane. If the angle between two shell elements is larger than the user-specified angle the mesh is refined. A limit for the refinement has to be determined by the user. Another indicator is the thickness-based indicator. If the thickness of a shell element is reduced below a user-defined thickness, then that element will be refined. The thickness type of refinement indicator is an ideal indicator of necking, because the elements thickness will be decreased in the necking zone. Thus, the mesh density increases at the necking zone, which results in an increased accuracy as well as a visual indicator of necking, see (Moshfegh 1996).

REFINEMENT INDICATOR BASED ON GRADIENT FORMULATION

In this section, two new mesh refinement indicators are presented for nonlinear FE analysis of shells. The indicators are based on the gradient of effective stresses (GSIG) and the gradient of effective plastic strains (GEPS), respectively.

The Belytschko-Lin-Tsay (Belytschko-Lin-Tsay et al. 1984) shell element is used in our application. This is a bilinear four node quadrilateral element with single-point quadrature and a selective number of integration points through the thickness. This element is computationally efficient and also very competitive provided that the spurious hourglass modes are controlled. It is frequantly used for sheet metal forming, see (Hallquist et al. 1991) and (Hallquist and Galbraith 1993).

Indicator based on the gradient of effective stresses (GSIG)

Consider a typical shell element, denominated *K*, as shown in Figure 1. The characteristic element size can be defined as the minimum length of the sides, i.e.,

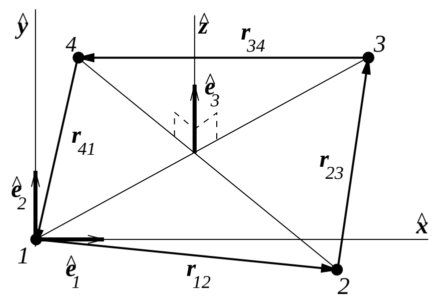

Figure 1. 4-nodes quadrilateral shell element

$$h_K = \min\{r_{12}, r_{23}, r_{34}, r_{41}\} \tag{1}$$

where r_{12} is the distance between nodes *1* and *2*, and so on. The Cauchy stresses are calculated only at the central point P_K of the element and can be written as

$$\boldsymbol{\sigma}(P_K) = (\sigma_x, \sigma_y, \sigma_z, \tau_{xy}, \tau_{yz}, \tau_{zx})^T \tag{2}$$

The corresponding effective stress is thus given by

$$\sigma_e = \frac{1}{\sqrt{2}}\left[(\sigma_x - \sigma_y)^2 + (\sigma_y - \sigma_z)^2 + (\sigma_z - \sigma_x)^2 + 6\left(\tau_{xy}^2 + \tau_{yz}^2 + \tau_{zx}^2\right)\right]^{1/2} \tag{3}$$

To introduce the refinement indicator for the $K'th$ element, a local patch is constructed which consists of *K* itself and the set N_K of neighboring elements K_i with one side in common with *K*, as illustrated in Figure 2.

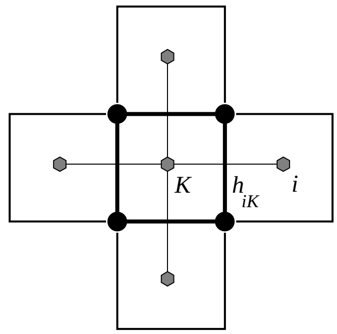

Figure 2. Nodal patch used to compute the refinement indicators

It is assumed that the gradient of the effective stresses (GSIG) at P_K can be estimated as

$$g_K^{\sigma} = \max \frac{\left|\sigma_e(P_K) - \sigma_e(P_{K_i})\right|}{\left|\mathbf{X}(P_K) - \mathbf{X}(P_{K_i})\right|} \tag{4}$$

where $\mathbf{X}(P_K)$ is the global coordinates of P_K and $|\cdot|$ stands for the Euclidean norm. It is assumed that to certain extend g_K^{σ} represents the variation of the stresses at element *K* and can be used to identify the regions of high gradients of stresses, where the errors of stresses are relatively large. Therefore, by an adaptive meshing process, the elements with large values of g_K^{σ} should be refined while the elements with very small values of g_K^{σ} could be coarsened, in order to achieve a so-called nearly optimal mesh over which the variation of stresses will be uniformly distributed.

Finally, by combining g_K^{σ} with the characteristic element size h_K, the refinement indicator for *K* based on the gradient of effective stresses is defined as

$$E_K^{\sigma} = h_K \, g_K^{\sigma} \tag{5}$$

Indicator based on the gradient of effective plastic strains (GEPS)

For some elasto-plastic problems with large deformation, strain related variables might be more suitable than stress related ones. In this subsection, another refinement indicator is presented which is based on the gradient of effective, accumulated, plastic strains. The derivation is following the same idea as above.

The local patch as shown in Figure 2 is used again and the gradient of effective plastic strains is estimated as

$$g_K^{\varepsilon} = \max \frac{\left|\varepsilon_{ps}(P_K) - \varepsilon_{ps}(P_{K_i})\right|}{\left|\mathbf{X}(P_K) - \mathbf{X}(P_{K_i})\right|} \tag{6}$$

It is assumed that to certain extend g_K^{ε} represents the variation of the plastic strains at the target element and is capable of identifying the regions with high gradient of the effective plastic strains. Thus, by an adaptive remeshing, those elements with large values of g_K^{ε} should be balanced with small element sizes, in order to achieve a mesh over which the variation of the effective plastic strains is equally distributed or the gradients of the effective plastic strains are smoothed. Therefore, the local refinement indicator is defined as

$$E_K^{\varepsilon} = h_K \, g_K^{\varepsilon} \tag{7}$$

Refinement strategy

The refinement strategy of the *h*-adaptive methods is based on seeking a so-called optimal discretization, by which discretization error is globally within a given tolerance. Global error estimation is not available in this work, but the gradient based adaptive strategy is directly aiming at a mesh over which local refinement indicators will be equally distributed.

Consider the GEPS indicator e.g., Equation 7. The average value of the refinement indicator can easily be calculated, when refinement indicators for all elements are known.

$$\overline{E}^{\varepsilon} = \frac{1}{nel}\left(\sum_{K=1}^{nel} E_K^{\varepsilon}\right) \tag{8}$$

where *nel* is the total number of elements for the part of the model where the mesh should be refined. In the most ideal situation the refinement indicator for each element would be the same as $\overline{E}^{\varepsilon}$, which seldom occurs in reality.

In principle, whether an element *K* on the current mesh should be refined or not may be conveniently guided a new local parameter

$$\xi_K = \frac{E_K^{\varepsilon}}{\overline{E}^{\varepsilon}} \tag{9}$$

and

$$\xi_K = \begin{cases} <1 & \text{enlarge element size} \\ =1 & \text{optimal} \\ >1 & \text{reduce element size} \end{cases} \tag{10}$$

From practical point of view, a refinement criteria $\beta_{tol}^{\varepsilon}$ is introduced so that the element *K* will be refined when

$$\xi_K^{\varepsilon} - 1 > \beta_{tol}^{\varepsilon} \tag{11}$$

In other words, those elements whose refinement indicators values exceed the average value with a specified tolerance level $\beta_{tol}^{\varepsilon}$ will be refined.

A program module based on the discussed refinement indicators has been implemented in LS-DYNA. Figure 3 shows a flowchart in order to explain the idea behind the adaptive solution procedure.

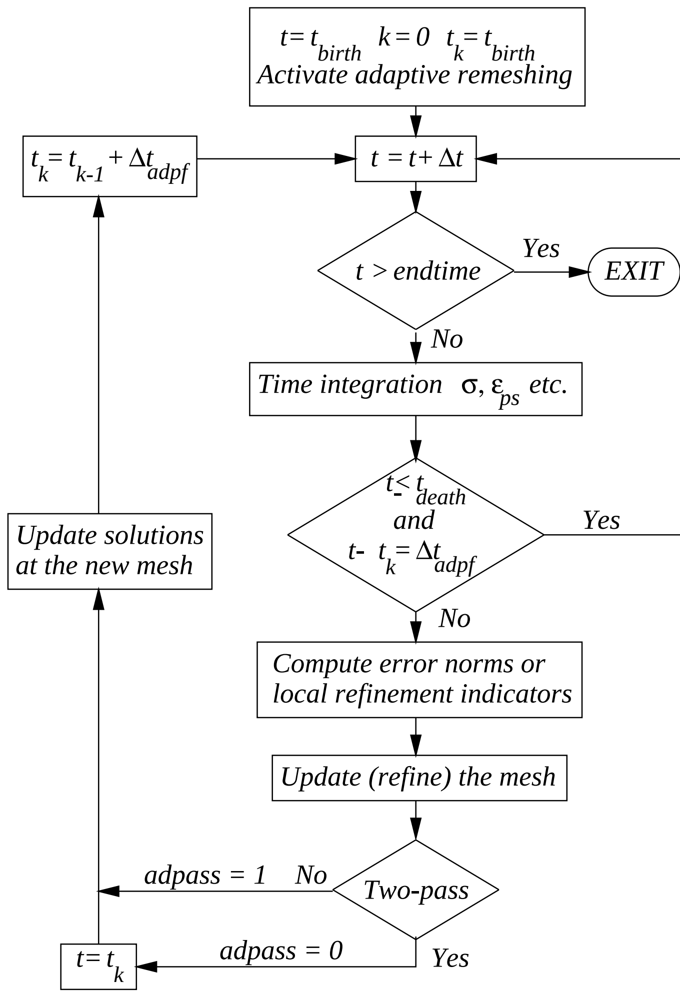

Figure 3. A flowchart illustrating the proposed adaptive procedure

APPLICATION EXAMPLE

This section presents the FE modeling and simulation of the Conventional Deep Drawing (CDD) process. Results from different adaptive methods are illustrated and compared to experimental results

Development of FE model

Finite element analysis of the CDD process has previously been conducted in several projects, e.g., see (Nielsen 1997), (Andersen 1994), and (Moshfegh et al. 1998). Thus, a large amount both of numerical and experimental data exists, which can be used for comparison with the present numerical results. The principle set up of the CDD process is shown in Figure 4, where the punch is moved downwards in order to form the cup. A blank holder is used to apply a pressure at the flange part of the blank.

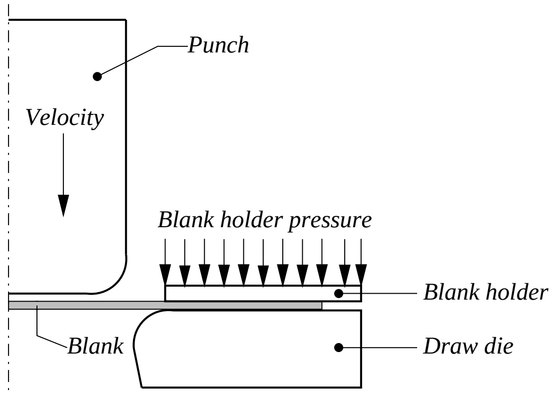

Figure 4. Illustration of the set up of the conventional deep drawing process

The complete FE model is shown in Figure 5, where the FE discretization can be seen. Due to symmetry, only one quarter of the geometry is modeled with two symmetry planes, the *xz*-plane and *yz*-plane. The set up for the conventional deep drawing process consists of the punch, the blank holder, the draw die and the blank, see figures 4 and 5.

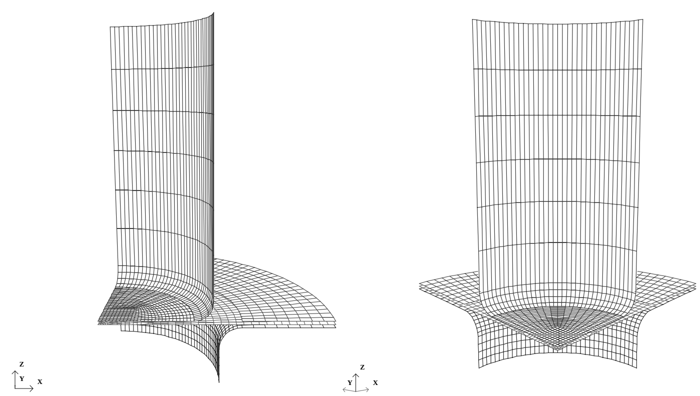

Figure 5. The complete FE model that illustrates the FE discretization

The punch, blank holder and draw die are modeled as rigid bodies. The blank material used in the experiments is the low carbon steel FEP04. The material properties are listed in Table 1. It is observed from Table 1 that the material is anisotropic. In the FE simulation, the blank is modeled as the Barlat's tri-component anisotropic plasticity material model; see (Barlat and Lian 1989).

Table 1. The properties for the applied sheet material, Steel FEP04, see (Andersen 1994)

PARAMETER	NOTATION	VALUE
Strain-hardening exponent	n	0.192
Strength coefficient	C	524

Lankford parameter in 0º	R_0	1.62
Lankford parameter in 45º	R_{45}	1.37
Lankford parameter in 90º	R_{90}	2.02
Plane anisotropy	ΔR	0.45
Average anisotropy	$\bar{R}$	1.60

The contact between the blank and tools is a standard master and slave contact interface with friction coefficient $\mu = 0.1$. The stiffness hourglass control is used in the FE simulation. No mass scaling is applied in this study.

Comparison between numerical and experimental results

In the following sections the numerical results from FE simulation of the CDD process are compared with the experimental results. Four different refinement indicators and one without mesh refinement are applied in these comparisons:

- No refinement
- Angle based refinement indicator
- Thickness based refinement indicator
- Refinement indicator based on the gradient of the effective plastic strains (GEPS indicator)
- Refinement indicator based on the gradient of the effective stresses (GSIG indicator)

The experimental results are described in (Moshfegh et al. 1998).

Forming analysis results. A comparison between the final shape of the blank from the experimental result and the FE simulation with no refinement is shown in Figure 6. As can be observed earing occurs in the expected directions for both numerical and experimental results. A parameter that can be used in connection with the earing in a drawn cup is the plane anisotropy, ΔR, see Table 1. The figure also shows close similarity between the results from the FE simulation and the experiment as to the earing tendency.

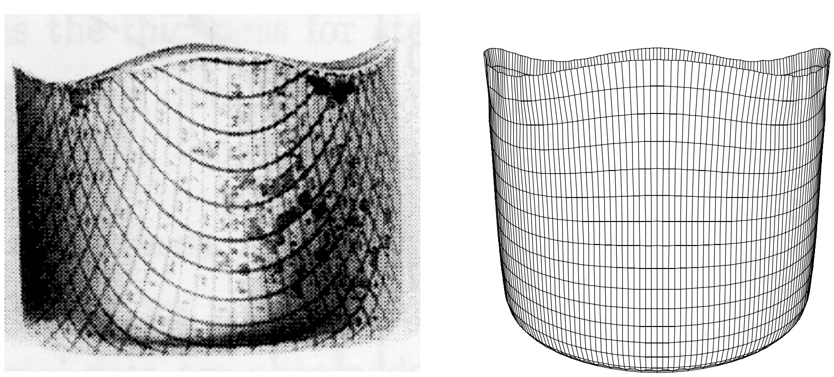

Figure 6. Earing of a drawn cup. Left: Experiment; see (Brännberg 1994). Right: Result from LS-DYNA.

The blank is modeled using 1179 Belytschko-Lin-Tsay shell elements, and no mesh refinement needs to achieve more accurate results. In order to show the efficiency in the application of the refinement indicators in the FE simulation of the CDD process, the number of elements in the initial FE model of the blank is reduced. Figure 7(a) illustrates the initial mesh density of the blank, which include 335 shell elements. Figures 7(b) to 7(d) show the deformed blank after 52.5 mm displacement of the punch. The punch is removed in these figures in order to show the contact interface between the blank and the draw die lips. As shown in these figures no penetrations between the blank and draw die lips occurs when the angle based and the GEPS indicators are applied. However, all penetrations are disappeared in the final shape of the cup.

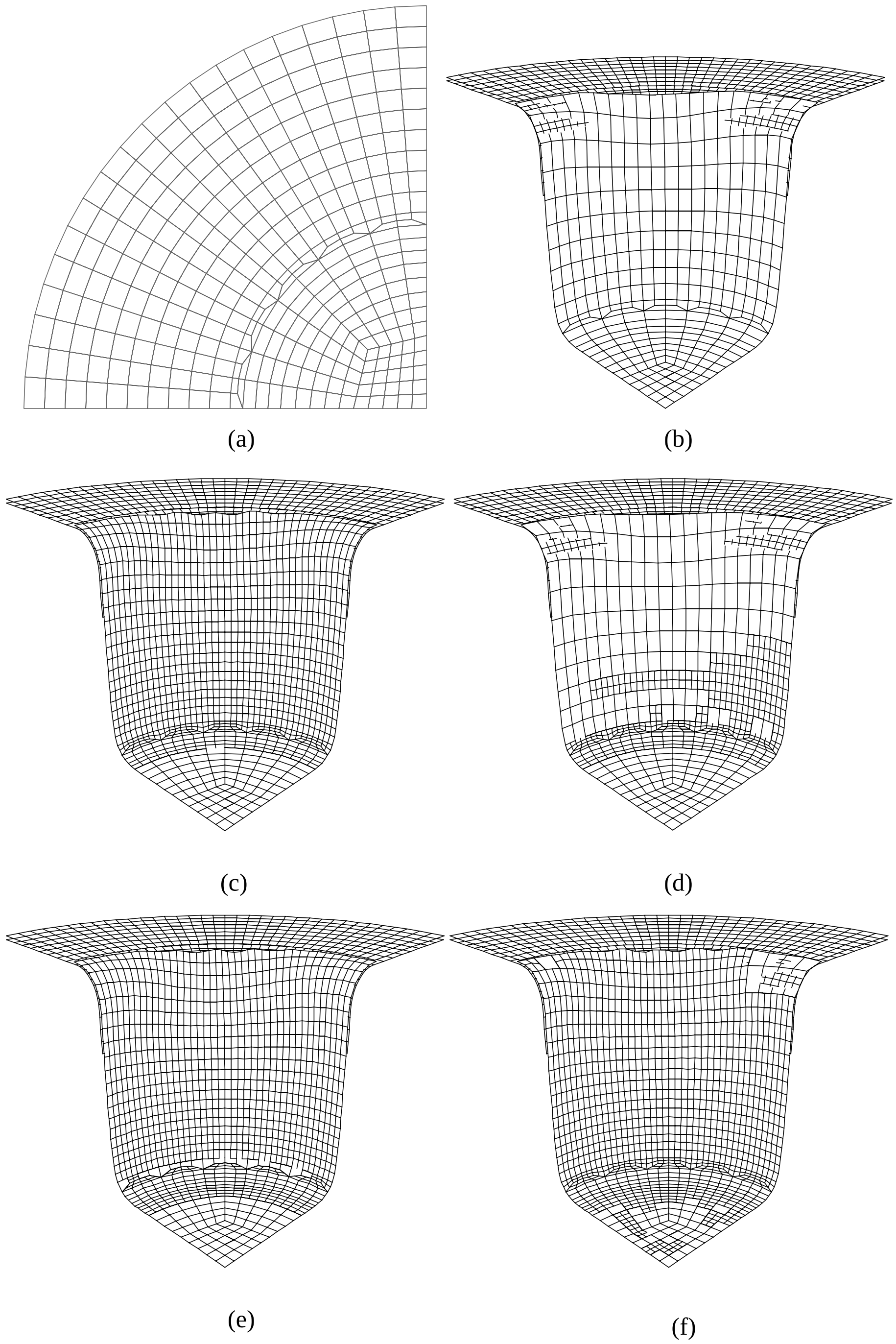

Figure 7. Deep drawing of a cup. (a): Initial geometry of the blank with 335 shell elements. (b): Deformed blank no adaptive mesh refinement. (c): Angle based refinement. (d): Thickness based refinement. (e): GEPS based indicator. (f): GSIG based indicator

Thickness distribution. The thickness distribution in the rolling direction versus cup height for the five different simulations together with the experimental results is illustrated in Figure 8.

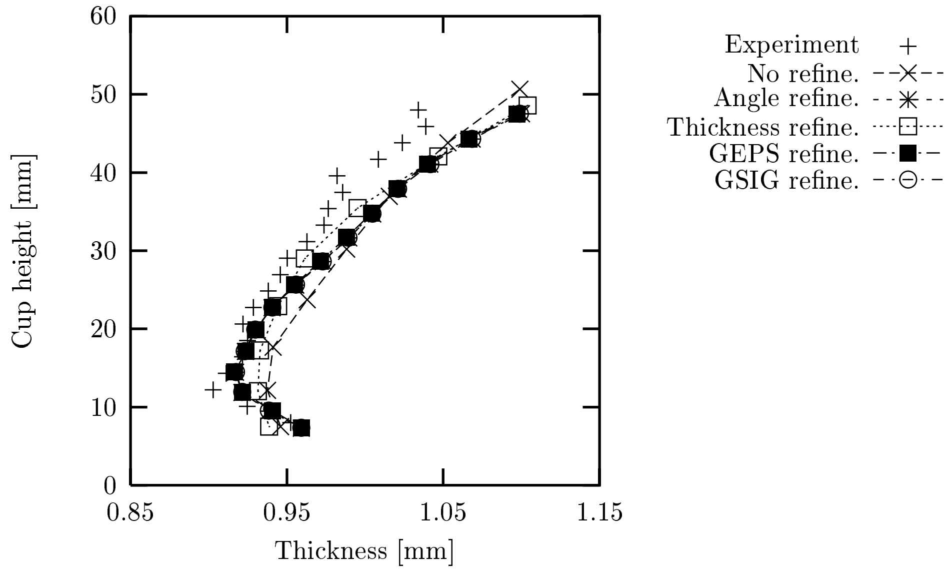

Figure 8. Thickness distribution in the rolling direction

The figure illustrates that the obtained numerical results based on angle, GEPS and GSIG refinement indicators are almost the same, specially, in the lower part of the drawn cup. The thickness distribution in 45º to the rolling direction versus cup height is illustrated in Figure 9. In this case a better agreement with the experiment is reached for angle, GEPS and GSIG refinement indicators.

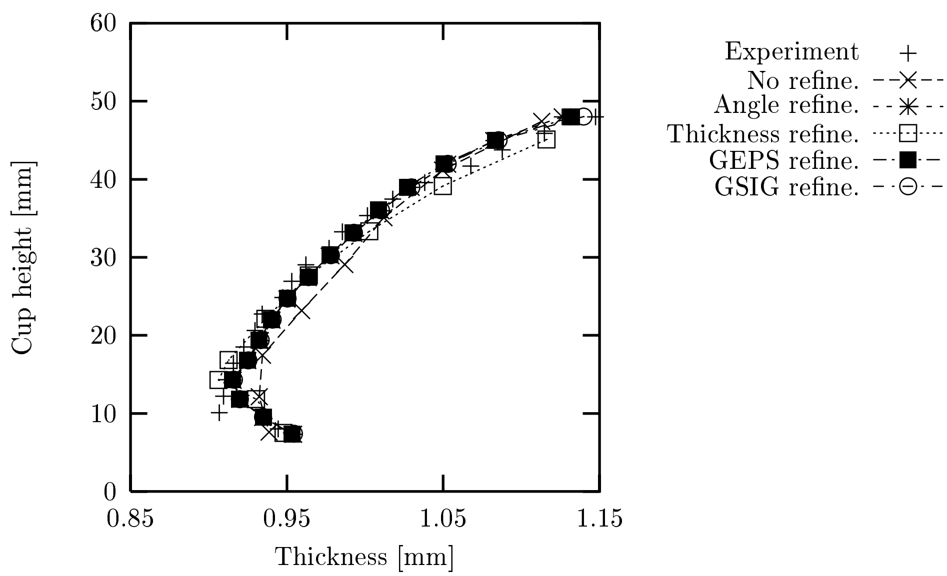

Figure 9. Thickness distribution in 45º to the rolling direction

Strain distribution. A local comparison between the numerical results, the effective plastic strain at 15 mm of the cup height versus circumferential angle is selected and shown in Figure 10. The reason for this height selection is its closeness to the punch nose and it is, in a technical sense, a more sensitive area in the CDD process. It is observed from Figure 10 that quit a good agreement between the obtained numerical results based on the angle, GEPS and GSIG refinement indicators are obtained.

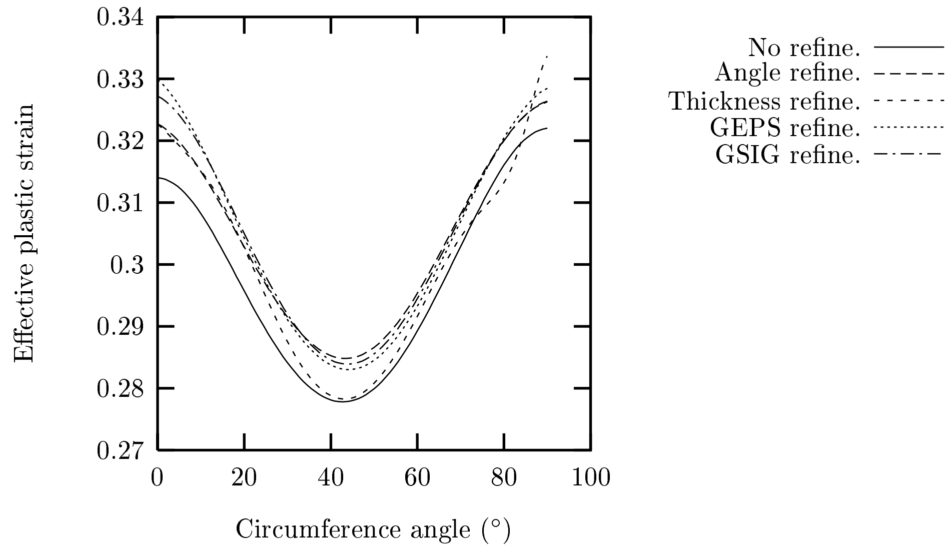

Figure 10. Effective plastic strain distribution versus circumference angle

Punch force. The punch forces from the four refinement indicators and one without refinement together with experimentally results are shown in Figure 11. The punch forces are nearly identical with a maximum value of approximately 80 kN.

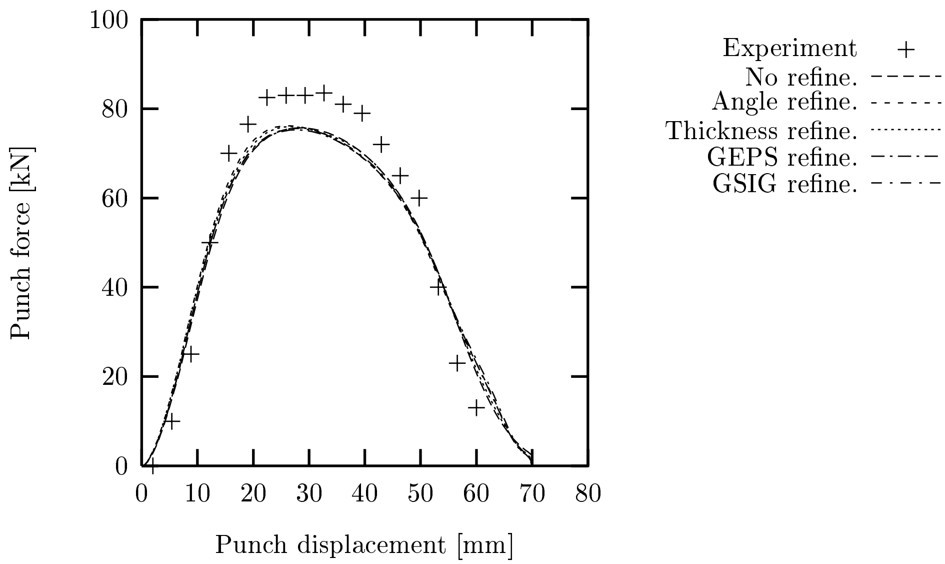

Figure 11. Numerically and experimentally obtained punch force versus punch displacement

Furthermore, the curves are seen to be relatively smooth indicating that the velocity speed up of the process is not too high. The magnitude of the maximum punch forces from different FE simulations is in quit good agreement with the experimentally obtained value as shown in Figure 11.

CONCLUSIONS

The presented work concerns the FE modeling and simulation of the Conventional Deep Drawing process using two new refinement indicators based on gradient of the effective plastic strains (GEPS) and effective plastic stresses (GSIG), respectively. A program module for implementing these refinement indicators is developed and coupled with the explicit finite element code LS-DYNA.

The idea behind the refinement strategy is motivated by the well-known fact that finite element solutions are generally less accurate in regions where high gradient of stresses or plastic strains exist.

The general applications of the proposed refinement indicators are in the adaptive finite element analysis of shells undergoing both material and geometrical nonlinearities. The mesh density of the shell elements in the blank has always some effect on the result of the numerical simulation of a sheet metal forming problem. Thus, the initial FE mesh density of the blank is less important if an adaptive FEM is applied. In general, the numerical results from the adaptive FE simulations are more accurate. However, the proposed refinement indicators can be more useful in FE simulation of sheet metal forming problems, because, these indicators can effectively identify those regions which have high gradients of stresses and strains. Thus, the mesh is refined in the regions undergoing the most severe deformations.

In this study different mesh refinement indicators, which are accessible in LS-DYNA, are compared against two new ones. To verify the numerical results against experiments, the low carbon steel FEP04 is used as a reference material. Some differences between the numerical and experimental results are observed. In the comparison between the numerical results, the angle based refinement indicator in LS-DYNA gives a result, which is in a good agreement with the results from the GEPS and the GSIG mesh refinement indicators. The compared quantities are:

- Thickness distribution in the rolling direction and 45º to the rolling direction
- Effective plastic strain near to the punch nose in the drawn cup
- Punch force

Acknowledgment- The authors would like thank Dr. John O. Hallquist for providing the possibility of using LS-DYNA in this research.

References

Ainsworth, M. and Oden, J.T. (1997). “A posteriori error estimation in finite element analysis.” Computer Methods in Applied Mechanics and Engineering, Vol. 142, pp.1-88.

Andersen, J.C.F. (1994). Hydromechanical Forming of High Strength Sheet Steels. PhD thesis, In Danish, Department of Production, University of Aalborg, Aalborg.

Babuska, I. and Rheinboldt, W.C. (1978). “A posteriori error estimates for the finite element method.” International Journal for Numerical Methods in Engineering, Vol. 11, pp. 1597-1615.

Barlat, F. and Lian, J. (1989). “Plastic behavior and stretchability of sheet metals. part 1: A yield function for orthotropic sheets under plane stress conditions.” International Journal of Plasticity, Vol. 5, pp. 51-66.

Belytschko, T., Tsay, C.S., and Lin, I. (1984). “Explicit Algorithms for the Nonlinear Dynamics of Shells.” Computer Methods in Applied Mechanics and Engineering, pp. 225-251

Belytschko, T., Wong, B.L. and Plaskacz, E.J. (1989). “Fission-fusion adaptivity in finite elements for nonlinear dynamics of shells.” Computer and structures, Vol. 33, pp. 1307-1323.

Bonet, J. (1994). “Error estimators and enrichment procedures for the finite element analysis of thin sheet large deformation processes.” International Journal for Numerical Methods in Engineering, Vol. 37, pp. 1573-1591.

Brännberg, N. (1994). Computational Aspects on Simulation of Sheet Metal Forming. Lic. Eng. Thesis, LiTH-IKP-LIC-1994:08, Department of Mechanical Engineering, Linköping Institute of Technology, Linköping.

Eriksson, K., Estep, D., Hansbo, P. and Johnson, C. (1995) “Introduction to adaptive methods for differential equations.” Acta Numerica, pp. 105-158.

Hallquist, J.O. and Galbraith, P.C. (1993). “Shell element formulations in ls-dyna3d: Their use in modelling sheet metal forming.” NUMISHEET'93, pp. 151-160.

Hallquist, J.O, Schweizerhof, K., Weimar, K. and Stillman, D.W. (1991). “Improving Standard Shell Elements, Friction Models and Contact Algorithms for the Efficient Solution of Sheet Metal Forming Problems with LS-DYNA3D.” VDI Berichte nr. 89, pp. 499-515.

Hallquist, J.O., Stillman, D.W. and Lin, T. (1999). LS-DYNA - Nonlinear Dynamic Analysis of Structures in Three Dimensions. LSTC Report. Livermore Software Technology Coporation, Livermore.

Lee, N.S. and Bathe, K.J. (1994). “Error indicators and adaptive remeshing in large deformation finite element analysis.” Finite Elements in Analysis and Design, Vol. 16, pp. 99-139.

Li, X.D., Moshfegh, R. and Nilsson, L. (1999). “Two mesh refinement indicators and adaptivity in nonlinear finite element analysis of shells.” Submitted to International Journal for Numerical Methods in Engineering.

Moshfegh, R. (1996). Finite Element Analysis of the Erichsen Cupping Test – with special reference to necking. Lic. Eng. Thesis, LiTH-IKP-LIC-1996:55, Department of Mechanical Engineering, Linköping Institute of Technology, Linköping.

Moshfegh, R. and Jensen, M.R. (1998). Finite Element Simulation of the Hydromechanical Deep Drawing Process. Technical Report LiTH-IKP-R-1032, Department of Mechanical Engineering, Linköping Institute of Technology, Linköping.

Nielsen, K.B. (1997). Sheet Metal Forming Simulation Using Explicit Finite Element Methods. PhD thesis, Department of Production, University of Aalborg, Aalborg.

Okstad, K.M. and Mathisen, K.J. (1994). “Towards automatic adaptive geometrically nonlinear shell analysis. Part i: implementation of an h-adaptive mesh refinement procedure.” International Journal for Numerical Methods in Engineering, Vol. 37, pp. 2657-2678.

Peric, D., Yu, J. and Owen, D.R.J. (1994). “On error estimates and adaptivity in elasto-plastic solids: application to the numerical simulation of strain localization in classical and cosserat continua.” International Journal for Numerical Methods in Engineering, Vol. 37, pp. 1351-1379.

Wiberg, N.E. and Li, X.D. (1994). “A postprocessed error estimate and an adaptive procedure for the semidiscrete finite element method in dynamic analysis.” International Journal for Numerical Methods in Engineering, Vol. 37, pp. 3583-3603.

Zienkiewicz, O.C. and Zhu, J.Z. (1987). “A simple error estimator and adaptive procedure for practical engineering analysis.” International Journal for Numerical Methods in Engineering, Vol. 24, pp. 333-357.

Zienkiewicz, O.C. and Zhu, J.Z. (1992). “The superconvergent patch recovery and a posteriori error estimators, Part i: The recovery technique.” International Journal for Numerical Methods in Engineering, Vol. 33, pp. 1331-1364.