Finite element calculations of hole expansion in a thin steel sheet with polynomial yield functions of four and six degrees

Seung-Yong Yang1, Wei Tong2∗ and Mohammed Alharbi3
1 School of Mechanical Engineering, Korea University of Technology and Education, Chonan, Republic of Korea
2 3Department of Mechanical Engineering, Southern Methodist University, Dallas, Texas, USA
E-mail: 1 ysy@koreatech.ac.kr
E-mail: 2∗ wtong@lyle.smu.edu, (corresponding author)
E-mail: 3 malharbi@mail.smu.edu

Abstract. A three-dimensional finite element simulation was used to study the anisotropic plasticity behavior of sheet metal forming. Both Gotoh’s (fourth order) yield function and the more flexible sixth order polynomial yield functions with an associated flow rule were implemented as user material subroutines in the FE code ABAQUS. Parameter values in the yield functions were decided by fitting the yield stresses and plastic strain ratios along various directions of uniaxial and biaxial tension. To verify the FE implementation and to evaluate the modeling capabilities of the developed yield functions that were certified to be convex, the hole expansion experiment by Kuwabara et al.[1] was considered as the target example. The simulation results using the sixth order yield function showed a better agreement with the experimental results than those of lower order yield functions such as Hill’s second order or Gotoh’s fourth order yield functions.

1. Introduction
A computational analysis for sheet metal forming is an essential part of the manufacturing process nowadays. It can be extremely helpful in reducing the cost and time in the development of the metal forming process. Extensive progresses have been made to improve the finite element method for the plastic forming analysis in the various areas such as material modeling, contact treatment, element development, and so on. Phenomenological continuum approach is favored in the material modeling of plasticity behavior over the microscopic crystal plasticity approach, because the micromechanical approach requires significant computational cost, although the latter approach may provide some information about the fundamental mechanism of sheet metal plasticity. The present study aims at presenting the finite element implementation procedure of two non-quadratic polynomial yield functions to model the orthotropic plastic deformation behavior of a sheet metal. Fourth and sixth order homogeneous polynomial yield functions have been implemented in the user material subroutines UMAT and VUMAT for the commercial finite element software ABAQUS. A hole expansion experiment on a thin steel sheet conducted by Kuwabara et al.[1] are simulated using the developed user material subroutines to verify and validate the FE implementation.
2. Theory

Hill [2] suggested a polynomial stress expression for the yield function of the orthotropic sheet metal under a state of plane stress in the following form

\[ P_n (\sigma_x, \sigma_y, \tau_{xy}) = \sum_{i+j+k \leq n} A_{ijk} \sigma_x^i \sigma_y^j \tau_{xy}^k, \]

where \( x \) and \( y \) axes align with the in-plane orthotropic symmetric axes and \( z \) axis is normal to the sheet plane. The yielding condition can be defined by the following non-dimensional form or in terms of an equivalent yield stress \( \tilde{\sigma} \) and the current flow strength \( \phi_f \) of the material as

\[ P_n^* (\sigma_x, \sigma_y, \tau_{xy}) = 1, \quad \text{or} \quad f = \tilde{\sigma} (\sigma) - \sigma_f (\tilde{\sigma}) = 0 \]  \hspace{1cm} (1)

where the bold face Greek letter \( \sigma \) represents the Cauchy stress tensor and \( \tilde{\sigma} \) is an equivalent plastic strain for isotropic hardening. The scalar equivalent stress of a given state of stress \( \tilde{\sigma} (\sigma) \) is defined as being numerically equal to the uniaxial tensile stress along the rolling direction which is on the same instantaneous yield surface as the given state of stress \( [3] \).

Only homogeneous polynomials were considered in this work, and for the case \( i+j+k = n = 2 \) and \( k = 0 \) or 2, the specific polynomial is the well-known quadratic yield function by Hill

\[ \Phi_2 (\sigma_x, \sigma_y, \tau_{xy}) = A_1 \sigma_x^2 + A_2 \sigma_x \sigma_y + A_3 \sigma_y^2 + A_4 \tau_{xy}^2. \]

In a similar way, for \( i+j+k = n = 4 \) and \( k = 0, 2, \) or 4 (the so-called Gotoh’s yield function [4]),

\[ \Phi_4 (\sigma_x, \sigma_y, \tau_{xy}) = A_1 \sigma_x^4 + A_2 \sigma_x^3 \sigma_y + A_3 \sigma_x^2 \sigma_y^2 + A_4 \sigma_x \sigma_y^3 + A_5 \sigma_y^4 \]
\[ + A_6 \sigma_x^2 \tau_{xy}^2 + A_7 \sigma_x \sigma_y \tau_{xy}^2 + A_8 \sigma_y^2 \tau_{xy}^2 + A_9 \tau_{xy}^4, \]

and for \( i+j+k = n = 6 \) and \( k = 0, 2, 4 \) or 6,

\[ \Phi_6 (\sigma_x, \sigma_y, \tau_{xy}) = A_1 \sigma_x^6 + A_2 \sigma_x^5 \sigma_y + A_3 \sigma_x^4 \sigma_y^2 + A_4 \sigma_x^3 \sigma_y^3 + A_5 \sigma_x^2 \sigma_y^4 + A_6 \sigma_x \sigma_y^5 \]
\[ + A_7 \sigma_y^6 + A_8 \sigma_x^4 \tau_{xy}^2 + A_9 \sigma_x^3 \sigma_y \tau_{xy}^2 + A_{10} \sigma_x^2 \sigma_y^2 \tau_{xy}^2 + A_{11} \sigma_x \sigma_y^3 \tau_{xy}^2 \]
\[ + A_{12} \sigma_y^4 \tau_{xy}^2 + A_{13} \sigma_x^2 \tau_{xy}^4 + A_{14} \sigma_x \sigma_y \tau_{xy}^4 + A_{15} \sigma_y^2 \tau_{xy}^4 + A_{16} \tau_{xy}^6. \]

For practical analyses using a finite element program, it can be more useful to consider the three-dimensional case. The two-dimensional fourth-order and sixth-order polynomials above can be extended for the 3D per the suggestions given in [5, 6]. For the case of the sixth order yield function, two possible examples are

\[ \Phi_{6a}^{3D} (\sigma) = \Phi_6 (\sigma_x - \sigma_z, \sigma_y - \sigma_z, \sqrt{\tau_{xy}^2 + \tau_{yz}^2 + \tau_{zx}^2}) + (A_{17} - A_{16}) \tau_{yz}^6 + (A_{18} - A_{16}) \tau_{zx}^6, \]
\[ \Phi_{6b}^{3D} (\sigma) = \Phi_6 (\sigma_x - \sigma_z, \sigma_y - \sigma_z, \tau_{xy}) + A_{17} \tau_{yz}^6 + A_{18} \tau_{zx}^6. \]

If two additional out-of-plane shear yield stresses are not available, \( A_{17} = A_{18} = A_{16} \) is assumed. The 16 material parameter values were derived by fitting uniaxial yielding stress ratios and \( r \)-values along several loading directions. The material parameters are given in Table 1 and the experimental \( r \)-value and fitting curve are displayed in Figure 1.
3. Numerical algorithm

An implicit finite element method is composed of local constitutive stress integration and structural iterative parts. In our work, Fortran user material subroutines UMAT and VUMAT were developed for the finite element software ABAQUS. The implicit procedure implemented in UMAT updates the local stress and state variables and returns the Jacobian matrix to the main program, while the explicit VUMAT routine just integrates the stress and state variables. Our numerical integration algorithm will follow the general three-dimensional procedure as given by Belytschko et al. [7].

3.1. Local stress update by predictor and corrector

It is necessary to integrate solutions like plastic strain and stress with time history in a path dependent problem like plastic deformation. In ABAQUS, a total strain increment is given to the user subroutine. The new solution variables are updated for the next time step \( k+1 \) in the user material subroutine. This section explains an iterative procedure to update the solutions.

In implicit finite element procedure, the variables are updated by an iterative process. The updated total strain and the plastic strain at the next time step are represented by

\[
\epsilon_{k+1} = \epsilon_k + \Delta \epsilon, \quad \epsilon^p_{k+1} = \epsilon^p_k + \Delta \epsilon^p.
\]

If the plastic yielding condition Eq.(1) is satisfied, the increment of plastic strain is obtained from the consistency condition. The flow rule \( \Delta \epsilon^p = \Delta \lambda p_{k+1} \) can be used, where \( p \) is the flow direction tensor which is given by the derivative of the flow potential \( p = \frac{\partial f}{\partial \sigma} \). The flow potential is the same as the yielding function in the associated flow rule. Then the stress increment can be written by

\[
\sigma_{k+1} = \sigma_k + \Delta \sigma = \sigma_k + C : (\Delta \epsilon - \Delta \epsilon^p) = \sigma_{\text{trial}} - C : \Delta \epsilon^p,
\]

where \( C : \Delta \epsilon^p \) is called the plastic corrector added to the elastic prediction of the stress tensor. The increment of the flow strength is given by

\[
q_{k+1} = q_k + \Delta \lambda h_{k+1},
\]

where \( q \) was used instead of \( \sigma_f \) to model the current flow strength of the material in terms of a set of internal variables, \( h \) is the hardening modulus and \( \Delta \lambda \) is the increment of the loading parameter. In the case of the associated flow rule, \( d\lambda = d\bar{\epsilon}^p \) using the property of Euler’s homogeneous function. The quantities at the next time step \( k+1 \) should additionally satisfy the yield condition as well if plastic deformation occurs, namely \( f(\sigma_{k+1}, q_{k+1}, \lambda_{k+1}) = 0 \).

This formulation is an implicit procedure, because the flow direction \( p \) and the hardening stiffness \( h \) are the values at the new equilibrium state at the next time step. The unknowns, plastic strain \( \Delta \epsilon^p \), internal variables \( \Delta q \), and loading parameter \( \Delta \lambda \) are obtained by solving the following nonlinear equations,

\[
a = -\epsilon^p_{k+1} + \epsilon^p_k + \Delta \lambda p_{k+1} = 0, \quad b = -q_{k+1} + q_k + \Delta \lambda h_{k+1} = 0, \quad f(\sigma_{k+1}, q_{k+1}, \lambda_{k+1}) = 0.
\]

If the equation is not satisfied at the \( j \)-th iteration, a modification can be chosen to satisfy the equation in the next iteration

\[
a^{j+1} = a^j + \delta a^j = 0.
\]
3.2. Structural iteration
The standard Newton procedure (path dependent strategy) is not suitable in the plasticity problem because it can cause spurious loading-unloading [8]. Instead, a secant Newton method (path independent strategy) should be used for the structural iteration part. It is necessary to supply the Jacobian matrix or algorithmic stiffness to the ABAQUS main program in UMAT (the explicit procedure does not however need this process), namely

$$C_{alg} = \frac{d\Delta \sigma}{d\Delta \epsilon}$$

where $\Delta$ means the increment between the time steps $k$ and $k+1$, and the algorithmic stiffness means the ratio between the variations of the total increments of stress and strain during the time increment. The expression of the consistent tangent modulus was derived by following the procedure described in [7]. This algorithmic stiffness can be an unsymmetric matrix if a non-associate flow rule is used.

3.3. VUMAT
The user material subroutine for ABAQUS/Explicit is called VUMAT and the arguments have different names and meaning from implicit UMAT. In our work, UMAT was developed first and then a small subroutine was added to UMAT to convert the program to be adequate for the purpose of VUMAT using relSpinInc variable. The components of the stress tensor are referred to the corotational coordinate axes, or neutralized, so a routine for the stress transformation is necessary here.

4. Finite element calculation
4.1. Verification examples
To verify the developed user material subroutines, simple test problems were first considered. If the numerical finite element results are identical to the analytic results of the simple problem, then it can be said that the coding is implemented properly.

In the first example problem, the anisotropy coefficient or the plastic strain ratio R-value was examined using an uniaxial tensile loading along various loading angles $\theta$ to the rolling direction. That is, the longitudinal axis of the specimen is at a different angle with respect to the $x$ axis. Using the sixth order yield function $\Phi_6$, the analytic expression for the R-value can be found in [6] and the ABAQUS results for five loading angles $\theta = 0, \frac{\pi}{8}, \frac{\pi}{4}, \frac{3\pi}{8}, \frac{\pi}{2}$ were compared with the analytic result. The material model uses the calibrated sixth order polynomial yield function denoted by POLY6a in Table 1 and one three-dimensional continuum element was used in the finite element analysis.

In the second example, yield condition defined by the same yield function was tested. Figure 2 displays the yielding locus for different amount of the shear stress $\tau_{xy}$ at $\bar{\epsilon}^p = 0.03$. The yielding condition Eq.(1) was used with the flow strength of a steel sheet,

$$\sigma_f(\bar{\epsilon}^p) = 1138(\bar{\epsilon}^p + 0.0002)^{0.099} \text{ MPa}. \quad (2)$$

The finite element model is POLY6a in Table 1 and biaxial and shear stresses were imposed on the one element model. The finite element results show the almost identical results as displayed by bullets in Figure 2. From these observations, the user material subroutines are verified to be accurately programmed.
4.2. Hole expansion simulation
Deformation behavior of 780MPa grade dual-phase steel sheet was investigated via hole expansion experiments by Kuwabara et al.[1]. In our work, fourth and sixth order polynomial yield functions were used to simulate the experiment. The experimental material testing data have been reported by Kuwabara et al.[1]. The parameters listed in Table 1 were fitted by a least square method for the fourth-order and sixth-order yield functions based on in part the direct calibration approach [5, 6]. All four cases of the non-quadratic yield functions listed in Table 1 were certified to be strictly convex using a numerical minimization algorithm as described in [5, 6]. The finite element model is shown in Figure 3: 851 8-node solid elements with 8 integration points (C3D8) were used to model the sheet metal and the punch and holder were modeled by rigid shell elements. The dimensions were taken as the same value to the experimental condition. Effect of the small friction coefficient was negligible, so it was set zero in the analysis.

Figure 4 shows the thickness strain at 1mm away from the bore edge. POLY6b gave the closest result among the tested models. Figures 5 and 6 display the thickness strain along the rolling and transverse directions. The finite element calculation was performed using a PC with Intel Core i7-3770 single CPU, 3.7GHz clock speed, 12GB RAM. The total elapsed wall-clock time for the computation was about 85 minutes. The same problem was solved by VUMAT in a Dell workstation PC with Intel Xeon double CPU E5-2667, 3.2 GHz clock speed. 30 logical processors were used. ABAQUS Explicit program is known to be vectorized and the elapsed wall-clock time was reduced to 13 minutes.
Table 1. Coefficient values for the polynomial yield functions.

| Model name | Hill1948 | POLY4a | POLY4b | POLY6a | POLY6b |
|------------|----------|--------|--------|--------|--------|
| $A_1$      | 1        | 1      | 1      | 1      | 1      |
| $A_2$      | -0.9637  | -1.94  | -1.94  | -2.89  | -2.9   |
| $A_3$      | 1.055    | 2.99   | 3.00   | 5.99   | 6.5    |
| $A_4$      | 2.7507   | -2.02  | -2.08  | -7.46  | -8.2   |
| $A_5$      | -        | 0.93   | 0.96   | 6.49   | 8.9    |
| $A_6$      | -        | 4.34   | 5.86   | -3.26  | -3.1   |
| $A_7$      | -        | -5.65  | -5.87  | 1      | 0.95   |
| $A_8$      | -        | 4.92   | 5.27   | 3.28   | 5.87   |
| $A_9$      | -        | 6.54   | 8.07   | -7.80  | -14.8  |
| $A_{10}$   | -        | -      | -      | 19.44  | 31.2   |
| $A_{11}$   | -        | -      | -      | -8.17  | -21.6  |
| $A_{12}$   | -        | -      | -      | 3.46   | 7.36   |
| $A_{13}$   | -        | -      | -      | 21.87  | 15.77  |
| $A_{14}$   | -        | -      | -      | -36.67 | -34.27 |
| $A_{15}$   | -        | -      | -      | 24.98  | 21.33  |
| $A_{16}$   | -        | -      | -      | 21.92  | 24.78  |

5. Conclusion

A finite element analysis of the hole expansion experiment on a dual-phase steel sheet by Kuwabara et al. [1] was carried out using ABAQUS user material subroutines UMAT and VUMAT. Convex fourth-order and sixth-order homogeneous polynomial yield functions were obtained by fitting uniaxial and biaxial yield strengths and plastic strain ratios. Sixth order polynomial yield function has more material parameters and degrees of freedom to fit the experimental data. The numerical results gave a similar tendency to the experimentally observed thickness strain on the bore edge and along the rolling and transverse directions of the sheet metal. The oscillatory behavior of the thickness strain along the bore edge depended on the yield stress ratio and the local minimum and maximum values of the thickness strain have changed by the r-value behavior on the bore edge. The use of an explicit finite element code could reduce the computation time by using multiple CPUs. In our computation, 30 CPUs have been used in the explicit computation.

References

[1] Kuwabara T, Hashimoto K, Iizuka E and Yoon J W. 2011 Effect of anisotropic yield functions on the accuracy of hole expansion simulations Journal of Materials Processing Technology 211 475
[2] Hill R 1950 Mathematical Theory of Plasticity Theory (Oxford: Clarendon Press)
[3] Calladine C R 2009 Plasticity for Engineers Theory and Applications (Chichester: Horwood publishing limited)
[4] Gotoh M. 1977 A theory of plastic anisotropy based on a yield function of fourth order (plane stress state) I Int. J. Mech. Sci. 19 505
[5] Tong W and Alharbi M. 2017 Comparative evaluation of non-associated quadratic and associated quartic plasticity models for orthotropic sheet metals Int. J. Solids & Struct. 128 133
[6] Tong W 2018 Calibration of a complete homogeneous polynomial yield function of six degrees for modeling orthotropic steel sheets Acta Mech. https://doi.org/10.1007/s00707-018-2113-7
[7] Belytschko T, Liu W K, Moran B Nonlinear Finite Elements for Continua and Structures (Chichester: John Wiley Sons)
[8] Dodds R 1987 Numerical techniques for plasticity computations in finite element analysis Comp. Struct. 26 767