MSTEP INTRODUCTION

I. BASIC INTRODUNCTION

The Finite Element Inverse Approach, also referred as One-Step simulation algorithm, is applied to predict the initial blank shape, strains, thickness distribution and formability of parts using the final part shape or die surfaces. With only two degrees of freedom (DOF) on each node of a quadrilateral element and fewer input data, the Inverse Approach can quickly obtain forming result if compared to the traditional incremental finite element method. Therefore, the approach is very suitable for stamping industry to conduct fast formability analysis. Major concerns of product and process design may be determined easily and rapidly during preliminary stage of product development cycle. Moreover, the approach can be adopted to optimize technical parameters and design schemes, calculating appropriate blank outlines for complicated sheet metal parts and predicting springback tendency.

Nowadays, commercial finite element software such as AUTOFORM/One step and FASTFORM are widely utilized in the stamping industry. The backbone of these commercial software are based on traditional Inverse Approach. One of the disadvantages of traditional Inverse Approach is that the accuracy of result is low when it is utilized to simulate large sheet metal deformation. Another disadvantages is the convergence difficulties in solving complicated model.

The MSTEP module in eta/DYNAFORM is based on an improved Inverse Approach, which overcomes the fast strain localization of traditional total strain theory. The traditional Inverse Approach leads to highly nonlinear equation systems as a result of large displacements, large strain and elasto-plastic materials. Sometimes, convergence difficulties are encountered. In the MSTEP, the higher precise quadrilateral membrane element model and Discrete Krichhoff Quadrilateral (DKQ) shell element is employed. Therefore, a fast iteration convergence in solving the group of equation can be obtained. By introducing a fast sparse matrix algorithm in MSTEP to solve the group of equation, the computational speed is much faster than the traditional algorithm. Furthermore, the comprehensive influence for sheet forming by the boundary conditions, such as friction between tools and blank, blank-holder force, drawbead, etc. are accounted for in the MSTEP module.

The MSTEP is also equipped with analytical tools mode, such as the curve binder, holder, and pad. It can simulate almost all kinds of forming type in different press machine, including single action, double action, double action + pad and triple action, etc. It also can be applied to predict the size and position of the inner hole and technical lancing of the part. The formability analysis of tailor welded blank and prediction of initial shape of welded line are also made possible with the MSTEP solution.

II. BASIC THEORY AND METHOD

The conditions assumed in the simulation processes include proportional loading, minimum plastic work path and large elasto-plastic strains with full incompressibility in sheet metal forming, and plastic total deformation constitutive model with Hill's anisotropic yield criterion. The fundamental idea of the Inverse Approach is to establish the finite element (FE) equations on the final forming simulation. Then, iteratively resolving these equations. Geometric sizes and physical quantities with respect to the initial flat blank and final stamping are shown in Table 1.

Table 1 Comparison of the blank with the stamping in geometric size and physical quantities

	Initial flat blank configuration C_0	Final workpiece configuration C
Geometry shape	Unknown	known
Thickness	known	Unknown
Stress, Strain	known	Unknown
Boundary conditions	known	known

Based on the information provided in Table 1, the essential conditions and physical quantities needed in deducing the Inverse Approach are known. Only three unknown quantities will be solved using the Inverse Approach.

2.1 Kinematic Equation and Geometric Relation

Using a generalized Krichhoff assumption in sheet deformation, with the initial blank configuration C_0 and the final configuration C, the kinematic equation of an arbitrary material point p in the sheet between the two configurations is expressed as

$$\mathbf{x}_0 = \mathbf{x} - [\mathbf{u} + z(\mathbf{n} - \frac{h}{h_0}\mathbf{n}_0)] \qquad (1)$$

where $\mathbf{x_0}$ and $\mathbf{x}$ are the position vectors of point p in initial C_0 and final C, respectively. $\mathbf{n_0}$ and $\mathbf{n}$ are the unit normal vectors of point p in C_0 and C, respectively. h_0 and h are the thickness of point p in C_0 and, C, respectively. U is the displacement vector of point p_0 in the middle surface of the sheet with respect to point p. z is the normal coordinate value of the sheet.

$\mathbf{x_0}$ and $\mathbf{x}$ satisfy the following deformation relation

$$d\mathbf{x}_0 = \mathbf{F}d\mathbf{x} \qquad (2)$$

where **F** is the deformation gradient tensor from initial configuration to final configuration of point p.

Therefore, the corresponding inverse of Cauchy-Green left tensor **B** is defined as

$$\mathbf{B}^{-1} = \mathbf{F}^{-\mathrm{T}}\mathbf{F}^{-1} \qquad (3)$$

From Eq. (1) to (3), the logarithmic strains components $\varepsilon_{xx}, \varepsilon_{yy}, \varepsilon_{xy}$ of point p on the sheet mid-surface are expressed as Eq. (4)

$$\begin{Bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \varepsilon_{xy} \end{Bmatrix} = \begin{bmatrix} \ln\lambda_1 \cos^2\theta + \ln\lambda_2 \sin^2\theta \\ \ln\lambda_1 \sin^2\theta + \ln\lambda_2 \cos^2\theta \\ (\ln\lambda_1 - \ln\lambda_2)\cos\theta\sin\theta \end{bmatrix} \qquad (4)$$

where λ_1^{-2} and λ_2^{-2} are the eigenvalues of the Green's deformation tensor $\mathbf{B}^{-1}$ and θ is the direction of principal strain.

2.2 Yield Criterion and Constitutive relationship

In the plane stress state, by taking anisotropic principal axes as x and y axes, the Hill's anisotropic yield criterion is expressed as Eq. (5)

$$f = \frac{1}{2(F+G+H)}\left\{(G+H)\sigma_x^2 + (F+H)\sigma_y^2 - 2H\sigma_x\sigma_y + 2N\sigma_{xy}^2\right\} - \frac{1}{3}\bar{\sigma}^2 \qquad (5)$$

The constitutive relationship is expresses as Eq. (6) after substituting the Eq. (5) into the Hencky's deformation theory.

$$\boldsymbol{\sigma} = E_s\boldsymbol{P}^{-1}\boldsymbol{\varepsilon} = \boldsymbol{D}_s\boldsymbol{\varepsilon} \qquad (6)$$

where,

$$E_S = \frac{\bar{\sigma}}{\bar{\varepsilon}} \tag{7}$$

$$\mathbf{P} = \frac{3R_{90}(1+R_0)}{2(R_0 + R_{90} + R_0 R_{90})} \begin{bmatrix} 1 & -\frac{R_0}{1+r_0} & 0 \\ -\frac{R_0}{1+r_0} & \frac{R_0(1+R_{90})}{R_{90}(1+R_0)} & 0 \\ 0 & 0 & \frac{(1+2R_{45})(R_0+R_{90})}{R_{90}(1+R_0)} \end{bmatrix} \tag{8}$$

where **σ** is the Cauchy stress tensor, $\bar{\sigma}$ and $\bar{\varepsilon}$ are the equivalent stress and strain, respectively, R_0、R_{45}、R_{90} are the Lankford coefficients of the sheet metal.

2.3 The Element Formulation and Virtual Work Equation

The virtual work equation in the states of C is established as Eq. (9):

$$W = W_{int} - W_{ext} = \int_v \boldsymbol{\varepsilon}^{\mathrm{T}} \boldsymbol{\sigma} dv - \int_v \mathbf{u}^{\mathrm{T}} \mathbf{f} dv = 0 \tag{9}$$

where **f** is the external force vector, $\mathbf{u}^{\mathrm{T}}$ is the virtual displacement vector.

The quadrilateral iso-parametric element is introduced as:

$$N_i = \frac{1}{4}(1 + r_i r)(1 + s_i s) \qquad (i = 1, 2, 3, 4) \tag{10}$$

The stamping (the state of C) through element discretization is expressed as:

$$W = \sum_e (\boldsymbol{u}^e)^{\mathrm{T}} (\boldsymbol{F}^e_{int} - \boldsymbol{F}^e_{ext}) = -\sum_e (\boldsymbol{u}^e)^{\mathrm{T}} \boldsymbol{R}^e = 0 \tag{11}$$

For the finite element mesh of the stamping, Eq. (11) is transformed into

$$W = \boldsymbol{U}^{\mathrm{T}} (\boldsymbol{F}_{int} - \boldsymbol{F}_{ext}) = -\boldsymbol{U}^{\mathrm{T}} \boldsymbol{R} = 0 \tag{12}$$

By using the Newton-Raphson method to solve the non-linear Eq. (12), the ith iterating step is expressed as

$$\boldsymbol{R}(\boldsymbol{U}^i) = \boldsymbol{F}_{ext}(\boldsymbol{U}^i) - \boldsymbol{F}_{int}(\boldsymbol{U}^i) \neq 0 \tag{13}$$

$$\boldsymbol{K}^i_{\mathrm{T}} \boldsymbol{\Delta U} = \boldsymbol{R}(\boldsymbol{U}^i) \tag{14}$$

$$\boldsymbol{U}^{i+1} = \boldsymbol{U}^i + \boldsymbol{\Delta U} \tag{15}$$

$$\boldsymbol{K}^i_{\mathrm{T}} = \left[-\frac{\partial \boldsymbol{R}(\boldsymbol{U})}{\partial \boldsymbol{U}} \right]_{\boldsymbol{U}=\boldsymbol{U}^i} \tag{16}$$

2.4 Inverse Approach Implementation

The Inverse Approach mainly exploits the knowledge of the 3-D shape of the final workpiece. An iterative scheme is used to find the original position of each material point in the initial flat blank. Therefore, it is possible to estimate the strain and stressed in the final workpiece.

The course of Inverse Approach usually can be divided into the following two steps:

1) Guessing Initial Solution Field

Firstly, the known 3-D final workpiece is discretized by three or four nodes shell elements. Then, the most simple vertical node projection approach is used to project all the nodes of part along the vertical orientation (usually is the Z direction) to the horizontal plane or the specified curve surface. The initial solution field is achieved.

In general, there will be some bad or misshapen elements in the initial solution field using the simple vertical node projection approach when some elements belong to vertical walls of the workpiece. The efficiency and convergence rate are found quite dependent upon the initial solution. Those bad or misshapen elements may initiate the divergence problem in solving the nonlinear equilibrium equations. Such approach might be utilized in AUTOFORM/One-Step, hence it can’t resolve final parts with undercut elements.

But the projection method is improved in the commercial codes such as FASTBLANK and MSTEP. After the vertical projection initial solution filed is obtained, the equal area of element method or other method is used to adjust the inferior elements. Therefore, the convergence of the iteration solution is improved.

The initial solution field get from the projection just is the initial value for the iteration resolution. The shape and the accurate of the elements have no essential influence to the final result. In fact, the result get from the elastic method in MSTEP is just the initial solution field without considering the plastic deformation. So the accuracy of the result is lower than the accurate method.

2) Iterative Solution

Based on the initial solution field from the former step, the Inverse Approach applies all equations in Sections 2.1 to 2.3 to both the initial blank shape structure and the final stamping structure. Then, the nonlinear equilibrium equations according to the principle of virtual work are established. The total internal nodal force and the total external force (such as blank-holder force, drawbead force and pad force and so on) will get to the equilibrium after the equilibrium iterations. Then, the antidromic displacement of every node will be obtained. Consequently, the coordination of the node on initial blank shape will also be obtained after superimposing the coordination of node on final shape and the corresponding antidromic displacement. Replacing the nodal displacement filed to Eq. (1) to Eq. (3), the logarithm strain is known. Then, the stresses are computed from the constitutive relationship.

III. CHARACTERISTIC AND ADVANTAGE OF MSTEP

Some improvements on resolution arithmetic make the MSTEP more efficient and robust. The key advantages of the MSTEP module in eta/DYNAFORM are summarized as the following:

1. Adopts the modified element model. In traditional Inverse Approach, only the membrane elements are considered, hence the in-plane deformation is taken into account, while the bending effects are ignored. For a large variety of industrial applications, the membrane effects are dominant. Therefore, it is necessary to consider bending effects. For some processes with dominant bending effects or large deformation, the error is greater if compared with the experiment. The bending effects are considered in MSTEP through adaptation of the "quasi-bending" shell element in which the displacement of outer plane is artificially added.

2. Established a more accurate analytical model to describe influence of the blank-holder force, drawbead and pad, etc. In solving the nonlinear equilibrium equations, the influence of the technical parameters is implemented through transforming the extend force to the corresponding nodes, then the extend force is added to the right-hand side of the group of equation. However, experiment and experience help to obtain the appropriate proportion about those factor. There is little theoretical evidence to guide the user.

3. Adopts fast resolution algorithm in solving the nonlinear equilibrium equations with improved computational speed. For more detailed information, please refer to the paper " A generalized 2n-factor form of the inverse of a matrix, Acta Mechanica Solida Sinica, Vol.23, No.4, 2002, 446-452"

IV. CONCLUSION

MSTEP module in eta/DYNAFORM is based on the improved Inverse Approach, in which it overcomes the shortcoming of fast strain localization in traditional total strain theory. Since higher precise quadrilateral membrane element model and DKQ shell element is employed, a fast iteration convergence in solving the group of equation can be achieved. By introducing a fast sparse matrix solution algorithm into the solution of the group of equation, it is much quicker than the traditional algorithm. Furthermore, the comprehensive influence for sheet forming by the true technical conditionals, such as friction between tools and blank, blank-holder force, drawbead, etc. are considered in MSTEP module. By introducing some tools mode, such as the curve binder, holder, pad, etc. the MSTEP can simulate almost all kinds of forming type in different press machine, including single action, double action, double action + pad and triple action.

Moreover, MSTEP can be utilized to get the accurate blank contours, in addition to predict the size and position of the inner hole and technical lancing of the part. Also, the MSTEP can analyze formability of the welded blank and estimate the shape of initial welded line.