3rd International Conference on Materials Processing and Characterisation (ICMPC 2014)

Effect of mesh parameters in finite element simulation of single point incremental sheet forming process

Kurra Suresh*, Srinivasa Prakash Regalla

Department of Mechanical Engineering, Birla Institute of Technology and Science Pilani-Hyderabad campus, Hyderabad, India

Abstract

This paper investigates the effect of element size and adaptive re-meshing technique in numerical simulation of incremental sheet forming (ISF) process. In ISF a hemispherical headed tool moves along the specified trajectory to deform the sheet into required shape. This tool path is generally very long and thus increases the computational time. Therefore, in this work adaptive re-meshing technique has been used to minimize the computational time without sacrificing the accuracy of the results. For this a varying wall angle conical frustum was simulated using shell elements with different element edge lengths and adaptive mesh. Effects of these mesh parameters on plastic strain, punch force and form accuracy of deformed geometry has been studied. The necessary simulations for this study are performed using explicit finite element code LS-DYNA.

Keywords: Incremental forming; Adaptive mesh; Numerical simulation; Plastic strain; Punch force.

1. Introduction

Single point incremental forming utilizes a hemi-spherical headed tool to deform the sheet into required shape. The tool follows the toolpath generated using computer aided manufacturing software and deforms the sheet in small increments into desired shape. Simple tooling without any dedicated dies, high formability are special features of this process.

Numerical simulation of incremental sheet forming process (ISF) helps to understand the sheet thinning, formability of blank material and forces on the forming tool. Incremental forming simulations were performed with simple geometries, due to difficulties with tool path definition in numerical simulation software and large computational time associated with the long toolpath. Kurra Suresh and Regalla 2013 proposed a methodology to convert

* Corresponding author. Tel.: +91-40-66303570; fax: +40-66303998.

E-mail address: ksuresh.iitd@gmail.com
the manufacturing codes into time position data for numerical simulation. This methodology enables to simulate any complex geometry in finite element software package. The present paper focuses on the applicability of adaptive re-meshing strategy for minimizing the computational time in ISF simulations.

Element size has significant effect on accuracy of finite element results. It affects the convergence of the problem, computational time and penetration between the master and slave surfaces. Fine mesh could improve the accuracy of results; but, it increases the computational time required to do the simulation. Coarse mesh leads to inconsistent results, penetration and convergence problems during simulation process. However, time required to complete the simulation is less due to less number of elements. One alternative to get the advantages of fine mesh with less computational time is the adaptive meshing. Adaptive meshing maintains a high-quality mesh throughout the solution by adjusting the mesh to restore aspect ratio of highly distorted elements. The improved mesh quality can prevent the divergence due to severe mesh distortion.

In adaptive meshing the initial coarse mesh is refined based on the selected refinement indicator. The mesh refinement indicators are based on equi-distribution of variation of stresses or plastic strains over the elements of mesh. Ramin Mosfegh et al. 2000 used gradient of effective plastic strain and gradient of effective stress as indicators for mesh refinement. They tested these two strategies in LS-DYNA by performing deep drawing simulations. LS-DYNA has two types of inbuilt mesh refinement indicators. These are mesh refinement based on angle change and thickness change. In case of angle based indicator the angle between the in-plane and out of plane are measured. If the angle between two shell elements is more than the user specified value the mesh is refined. In case of thickness based indicator the mesh is refined if the thickness of the blank reaches to user specified value.

Giraud-Moreau et al. 2008 proposed adaptive re-meshing scheme to solve problems, where large plastic deformations with ductile damage are possible, in metal forming processes. The proposed re-meshing method is based on refinement and coarsening of elements according to physical and geometrical criteria. The efficiency of proposed method has been tested by performing 2-D and 3-D elasto-plastic simulations in ABAQUS software. Mohd Ahmed et al. 2008 studied the influence of adaptive parameters such as field variable recovery, refinement techniques, and accuracy limit on the performance of adaptive procedure by simulating the sheet stretching operation. Lequesne et al. 2008 developed adaptive re-meshing strategy for ISF process. In ISF higher deformation occurs in the vicinity of current tool position. Therefore, the coarse elements in the neighborhood of tool are refined based on the neighborhood criteria. The proposed method has been implemented in finite element code Lagamine. The efficiency of proposed method has been tested by simulating simple line test.

2. Description of finite element model

2.1. Material characterization

Extra deep drawing (EDD) steels are the most widely used steel material today for automotive applications involving simple and complex components, which require very high formability. Exterior components such as starter end covers, petrol tanks, are made up of deep drawing grade steels. The low carbon steel sheets are also used extensively in enameling applications such as baths, sink units, kitchen ware, cooker and refrigerator panels [Singh SK et al. 2010]. The common tensile properties of these EDD steel sheets were determined by uni-axial tensile tests on a 5 ton electronically controlled UTM. The experiments were conducted at a constant crosshead speed of 2 mm/min to avoid the effect of strain rate on the mechanical properties. The load displacement curves obtained from
tensile test was used to construct the true stress – true strain curves. Strength coefficient (K) and work hardening exponent (n) were calculated from true stress, true strain data. Additionally, the Erichsen cupping test was performed to get the limiting dome height. The mechanical characteristics of EDD steel sheet from the tensile test and Erichsen cupping test was summarized in Table 1.

2.2. Governing equations

The general dynamic elastic-plastic deformation problem of sheet metal forming is governed by the following equations [Kobayashi et al. 1989]:

Equilibrium equations:

$$\frac{\partial \sigma_{ij}}{\partial x_j} = 0$$

(1)

Yield criteria:

$$f(\sigma_{ij}) = C : \bar{\sigma} = \sqrt{\frac{3}{2} \left\{ \sigma_{ij} \sigma_{ij} \right\}^{1/2}} = \bar{\sigma}(\dot{\varepsilon}, \varepsilon)$$

(2)

Constitutive equations:

$$\varepsilon_{ij} = \frac{3}{2} \frac{\dot{\varepsilon}}{\bar{\sigma}} \sigma_{ij}, \quad \varepsilon = \sqrt{\frac{2}{3} \left\{ \dot{\varepsilon}_{ij} \dot{\varepsilon}_{ij} \right\}^{1/2}}$$

With

$$\text{(3)}$$

Compatibility conditions:

$$\varepsilon_{ij} = \frac{1}{2} \left( \frac{\partial u_t}{\partial x_j} + \frac{\partial u_j}{\partial x_t} \right)$$

(4)

The unknowns for solution of quasi static plastic deformation process are six stress components and three velocity components. The above governing equations and boundary conditions prescribed in terms of velocity and traction are used to get unknowns. Since it is difficult to get the closed form solution that satisfies all of the governing equations, various approximation methods have devised. Finite element method is one among them which uses the variational principle to get the approximate solution. Variational form of elasto-plastic problem is given by

$$\int_V \bar{\sigma} \delta \varepsilon \, dV + \int_V \sigma_m \delta \varepsilon \, dV + \int_S \delta \sigma_m \varepsilon \, dV - \int S_F \delta u_t \, dS = 0$$

(5)

Once the solution for velocity field that satisfies the above basic equation is obtained, then corresponding stresses can be calculated using flow rule and known mean stress distribution. Even though the sheet metal forming problem can be treated as a quasi-static problem, numerical simulation software treat this as a dynamic problem described by the following equation.
\[ [M] \{\ddot{x}\} = \{F_{\text{external}}\} - \{F_{\text{internal}}\} \]  

(6)

Where \([M]\{\ddot{x}\}\) is the inertial force of the system and \(F_{\text{external}}, F_{\text{internal}}\) are external and internal forces of the system respectively.

The central difference formula is used to calculate accelerations of nodes \(\{\ddot{x}\}\) in terms of displacements which can be written as

\[ \{\ddot{x}\} = \frac{1}{\Delta t^2} \left[ \{x\}_{t-\Delta t} - 2\{x\}_t + \{x\}_{t+\Delta t} \right] \]  

(7)

Nodal displacements at time step \(t+\Delta t\) can be calculated by substituting the above acceleration equation in governing equation. The resultant equation is

\[ [M]\{x\}_{t+\Delta t} = [F_t] \]  

(8)

To make the solution stable time step \(\Delta t\) should be less than critical time step size. In Ls-Dyna for shell elements critical time step is given by

\[ \Delta t_c = \frac{\Delta x}{c} \quad \text{where} \quad c = \sqrt{\frac{E}{\rho(1-v^2)}} \]  

(9)

Critical time step is a function of element edge length (\(\Delta x\)) and elastic wave speed (c). Elastic wave speed is in turn a function of Young’s modulus (E), density (\(\rho\)) and poisson’s ratio (\(\nu\)) of the material.

The contact between the tool and sheet is defined using one way surface to surface contact algorithm. This algorithm is based on the penalty method. The master surface stiffness is calculated as follows.

\[ \text{Contact stiffness} = \frac{\alpha K A}{l} \]  

(10)

In the above equation \(\alpha\) is the penalty scaling factor, \(K\) is the material bulk modulus, \(A\) is the segment area and \(l\) is the element characteristic length.

2.3. Numerical model for LS-DYNA

The ISF process was simulated using explicit finite element code LS-DYNA. The finite element models of forming tool, blank and backing plate was shown in Fig. 1. The blank is square in shape with 250 mm X 250 mm X 1 mm in size. The blank is discretized with Belytschko-Tsay shell elements (Type -2) with two integration points in thickness direction. In this study, the element with edge length of 1 mm, 2 mm, 4 mm and 4 mm with adaptive re-meshing were considered to study their effect on various response variables. The nodes along all four edges of the blank are constrained to simulate the blank holder. The blank material is modeled using power law plasticity (MAT 17). The forming tool is modeled as a cylindrical rod with hemi-spherical head of 10 mm diameter. A backing plate is provided below the blank to avoid the bending and to improve the accuracy of final part. The backing plate has a hole of 110 mm diameter in the center with 5 mm corner radius. Both forming tool and backing plate are meshed with shell elements and assigned with rigid material model. The contact between the blank and backing plate, blank
and forming tool are defined using master-slave contact algorithm. The friction between different contact surfaces are defined using Coulomb’s friction law.

A varying wall angle conical frustum has been chosen as a goal shape to study the effect of mesh parameters. To get this shape the tool has to move in three dimensional space and should deform the sheet into required geometry. To define this complex tool path in numerical simulation, the part program has been generated using Pro-Manufacturing software for the selected geometry. This part program has been converted to time position data \((x,y,z,t)\) using MATLAB routine. The time position data has been given as an input to the simulation software using *BOUNDARY_PRESRIBED_MOTION_RIGID* keyword to control the motion of forming tool. To minimize the large computational time due to long tool paths in ISF simulation the punch velocity has been increased to 40 m/s and density has been increased by 10 times.

![Finite element model for numerical simulation of incremental forming.](image)

**3. Results and Discussion**

The effect of four different meshes with element edge length of 1 mm, 2 mm, 4 mm and 4 mm with mesh adaptivity have been tested (Fig. 2). In the first three meshes the number of elements is constant till the end of simulation, while in the fourth simulation (simulation with adaptive mesh) the elements are subdivided into small elements based on total angle change in degrees relative to the surrounding element for each element to be refined. In each refinement element is divided into 4 elements and maximum number of refinement levels are set to 3.

![Different mesh topologies used for numerical simulation of ISF (a) element edge length of 1 mm (b) 2 mm (c) 4 mm (d) 4 mm with adaptive re-meshing.](image)

Simulations are performed in LS-DYNA to study the effect of mesh topologies on evaluation of effective plastic strain, punch force and final shape of the part. Fig. 3 shows the distribution of maximum effective plastic strain with different meshes. The effect plastic strain with adaptive mesh is almost same as that of fine mesh with element edge length 1 mm. However, the strain distribution with coarse mesh was deviated more from the strain distribution with fine and adaptive mesh. Also, from the Fig. 3 it is clear that the element with edge length 4 mm is undergoing plastic strain after 150 sec, while the element in the model with fine mesh is undergoing plastic strain after 350 sec. Coarse mesh is undergoing more plastic strain and this strain is occurring earlier compared to fine mesh. Moreover, it was observed that the elements with edge length 4 mm (coarse mesh) are undergoing more distortion during the simulation.
The distribution of punch force in Z-direction is shown in Fig. 4. This figure illustrates that the punch force with adaptive mesh is almost same as that of fine mesh with element edge length of 1 mm. While the force with element edge length of 2 mm is slightly deviating. The force distribution with coarse mesh (element edge length of 4 mm) was deviated more compared to the results obtained with element edge length of 1 mm and 2 mm. Moreover, the estimated force with coarse mesh is lesser compared to fine mesh and is having large peaks.

The Fig. 5 shows the cross-section of deformed geometry with different meshes. The deformed cross-section with element edge length 1 mm and adaptive mesh is very close to the designed cross-section in the wall region. However, there is some deviation at the entrance and the bottom of the cup. The deviation at the entrance of the cup is due to bending of the blank over the radius provided on the backing plate. The deviation at the bottom of the cup is due to pillowing effect which is expected during forming the sheet. In case of coarse mesh, the deformed cross-section was deviated more from the goal shape. This deviation was more towards the bottom of the cup compared to the wall region. This could be due to the severe distortion of the elements towards the bottom of the cup.
The study reveals that the results obtained with adaptive mesh are as good as fine mesh. At the same time the computational time with adaptive mesh is less compared to the fine mesh. The computation time with element edge length of 1 mm was found to be 752 min, while the simulation with adaptive mesh took only 314 min. It shows that nearly 50% saving in computational time with adaptive mesh. The coarse mesh with element edge length 4 mm took only 11 min, but the results are not satisfactory. The adaptive mesh is highly efficient for numerical simulation of ISF process as the deformation is very much localized in this process.

4. Conclusions

This paper presents the effect of adaptive re-meshing technique in numerical simulation of ISF process. The results obtained with this technique are as good as the results obtained with fine mesh. Additionally, the computational time was reduced by 50% with adaptive re-meshing technique. Thus this technique can be used to overcome the problem of long computational time due to long toolpaths associated with ISF process.

References


