A Remeshing Procedure for Numerical Simulation of Forming Processes in Three Dimensions

L. Giraud-Moreau, H. Borouchaki and A. Cherouat

Institut Charles Delaunay (FRE CNRS 2848)
University of Technology of Troyes
12 rue Marie Curie BP2060
10010 Troyes Cedex - France
laurence.moreau@utt.fr

Abstract. This article presents a remeshing procedure of thin sheets for numerical simulation of metal forming process in three dimensions. During simulation of metal forming processes, where large plastic deformations are possible, severe mesh distortion occur after a few incremental steps. Hence an automatic mesh generation with remeshing capabilities is essential to carry out the finite element analysis. This paper gives the necessary steps to remesh a structure in finite element simulation of forming processes. The proposed remeshing technique based on geometrical criteria includes adaptive refinement and coarsening procedures. It has been implemented with triangular and quadrilateral elements. The proposed method has been integrated in a computational environment using the ABAQUS solver. Numerical examples show the efficiency of the proposed approach.

Keywords: Adaptive remeshing, forming process, geometrical error estimator, thin sheet.

1 Introduction

The finite element method has been very successful in the numerical simulation of metal forming processes like deep-drawing, hydroforming or forging [1-2]. However, due to the imposition of large plastic strains and friction, the finite element mesh representing the workpiece undergoes severe distortion and hence it is necessary to generate a new mesh for the deformed domain [3]. It is therefore convenient to update the mesh in such a way that it conforms to the new deformed geometry and becomes dense enough in the critical region while remaining reasonably coarse in the rest of the domain. The remeshing procedure must be automatic and robust. Several remeshing methods have been proposed during the last years. Globally, three different types of adaptive remeshing strategies can be distinguished: r-adaptivity, p-adaptivity and h-adaptivity [4-5]. Strategies based on r-adaptivity consist of keeping the number of special grid points fixed, but allowing them to move into regions where a finer spatial discretization is needed. This type of adaptation is particulary powerful on problems where a large domain is needed to capture a time varying solution which has steep slopes over only a small fraction of that domain [6]. The remeshing techniques presented by Zienkiewicz et al [7], Fourment et al [8], Coupez [9], Borouchaki et al [10] are based on the computation of a size map to govern a global remeshing of the part at each iteration. Strategies based on p-adaptivity consist of changing the degree of the interpolating polynomials in appropriate regions of the mesh. This method is preferred for (linear) smooth solutions or over subregions where the solution is smooth [11]. Strategies based on h-adaptivity consist of adapting the number of grid points and changing the mesh connectivity. Grid points are added to areas where more accuracy is demanded (the interpolation will be enriched) and can be deleted where the solution is accurate enough. As part of these methods, remeshing techniques based on the computation of a size map to govern a global remeshing of the part at each iteration have been proposed [7-10]. Cho and Yang [12] have proposed a refinement algorithm based on h-adaptivity which consists in splitting each deformed element in two elements along an edge. This procedure drags to the creation of small edges and consequently degenerates elements during repetitive refinement iterations. Moreover, all similar refinement methods only based on the break of edges lead to the formation of small edges or poor shaped elements.

This paper presents a new remeshing technique for the numerical simulation of thin sheet metal forming processes. This method is based on a geometrical criterion. It is applied to computational domain after each small displacement step of forming tools. It allows, in particular to refine the current mesh of the part under the numerical simulation of the forming process in the curved area with preserving shape quality element and to coarsen this mesh in the flat area.

The mesh refinement is necessary to avoid large element distortions during the deformation. It ensures the convergence of the computation and allows an adequate representation of the geometry of the deformed domain. The mechanical fields are simply induced from the old mesh into the new mesh.

The proposed remeshing method looks like to the remeshing method presented by Meinders [13] in the case of a triangular mesh. Compared to Meinders method, the proposed remeshing technique generates a smaller number of elements, it has been implemented with triangular and quadrilateral elements and a coarsening technique is considered, in addition to the refinement technique.

This paper gives the different steps of the proposed remeshing method. Some application examples are presented in order to show the pertinence of our approach.

2 General Remeshing Scheme

The simulation of the forming process is based on an iterative process. At first, a coarse initial mesh of the part is generated with triangular or quadrilateral elements. At each iteration, a finite element computation is then realized in order to simulate numerically the forming process for a small displacement step of forming tools. This displacement step must be sufficiently small with respect to the specified minimal size of mesh elements.

Then, remeshing is applied after each deformation increment, if necessary, according to the following scheme:

- coarsening procedure applied to elements which are in flat area,
- iterative refinement to restore mesh conformity,
- refinement procedure applied to elements which are in curved area (the refinement is applied in the vicinity of nodes for which the shape of the surface is changed and only if the minimal element size is not reached),
- iterative refinement to restore mesh conformity.

This process (simulation of the forming process for a small displacement step of forming tools, remeshing of the part) is repeated until the final tool displacement is reached.

The computation convergence is principally based on the mesh refinement and coarsening procedures. The applied refinement must in particularly not introduce a mesh distortion, which could increase during iterations and stop the forming process simulation.

2.1 Geometrical Criterion

During the remeshing procedure, a geometrical criterion is used to refine the current mesh of the part in the curved area, and to coarsen this mesh in the flat area. For a given element, this geometrical criterion represents the maximal angular gap between the normal to the element and the normals at its vertices. An element is thus considered to be “curved” (resp. “flat”) if the corresponding angular gap is greater (resp. smaller) than a given threshold (for example 8 degrees). The geometrical refinement and coarsening methods based on the same geometrical criterion are thus consistent.

The normal vector $\vec{\nu}$ at node P can be defined as the weighted average of the unit normal vectors $\vec{N}_i$ (i = 1,..m) to elements sharing node P:

$$\vec{v} = \frac{\sum_{i=0}^{m} \alpha_i \vec{N}_i}{\left\| \sum_{i=0}^{m} \alpha_i \vec{N}_i \right\|} \tag{1}$$

where α_i is the angle at P of the ith element sharing P.

The computation of normal vector to the element depends on the element shape (triangle or quadrilateral). The normal vector $\vec{N}$ to a triangle $P_1P_2P_3$ is the unit normal vector to its supporting plane :

$$\vec{N} = \frac{\overrightarrow{P_1P_2} \wedge \overrightarrow{P_2P_3}}{\left\| \overrightarrow{P_1P_2} \wedge \overrightarrow{P_2P_3} \right\|} \tag{2}$$

The normal vector to a quadrilateral element is the average of the normal vectors to the four triangles defined by joining its barycentre to its edges.

The geometrical criterion applied to an element can be written as:

$$\max_i(\arccos\left\langle \overrightarrow{v_i}, \overrightarrow{N} \right\rangle) \geq \omega_g \tag{3}$$

Where $\vec{v}_i$ is the normal at vertex i of the element, $\vec{N}$ is its normal and ω_g is an angular gap threshold. In this case, the element must be refined.

2.2 Mesh Refinement and Coarsening Methods

The adaptive remeshing technique consists in improving the mesh in order to conform to the geometry of the current part surface during deformation. In the following, the mesh refinement and coarsening methods are detailed.

The refinement technique consists in subdividing mesh elements. An element is refined if it is a "curved" element (geometrical criterion). There is only one element subdivision which allows to preserve the element shape quality: the uniform subdivision into four new elements. In the case of a triangle, three new nodes are added : one in the middle of each edge. In the case of a quadrilateral, five nodes are added : one in the middle of each edge and one in the element barycentre. Figure 1 shows the triangular and quadrilateral element refinements.

Fig. 1. Triangular and quadrilateral element refinements

After each refinement procedure, an iterative refinement to restore mesh conformity is necessary. Indeed, after applying the subdivision according to the geometrical criterion, adjacent elements to subdivided elements must be modified. As the edges of the subdivided elements are divided in two, there is a node in the middle of the edges common to the subdivided element and its adjacent elements. The mesh is then not conforming. To retrieve the mesh conformity, adjacent elements to subdivided elements must be also subdivided. This last subdivision can not be a homothetic subdivision in four elements because it would result in the systematic homothetic subdivision of all mesh elements.

There are three different configurations for adjacent elements which must be subdivided in order to ensure the mesh conformity:

- no edge is saturated (i.e. containing a new added node),
- only one edge is saturated,
- at least two edges are saturated.

Fig. 2. Subdivision of elements with one saturated edge

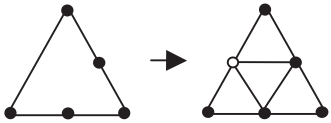


Fig. 3. Subdivision of triangle with two saturated edges

Depending on the configuration, a subdivision is applied if necessary. In the first case (no saturated edge), the element is not subdivided and is not modified. In the second case (one saturated edge), a triangular element is subdivided in two triangles and a quadrilateral element in three triangles (see figure 2). This subdivision allows to stop the propagation of the homothetic subdivision. In the third case, if all the edges are saturated the element is subdivided in four homothetic elements. Otherwise, in the case of triangular elements (having two saturated edges), all possible subdivisions lead to the formation of poor shaped elements (stretched elements). It is then necessary to add a new node in order to subdivide also this element into four homothetic elements (see figure 3). In the quadrilateral case, when only two edges are saturated and are adjacent, the quadrilateral is subdivided in four triangles (see figure 4). This subdivision allows to stop the propagation of the homothetic subdivisions. In the other cases, the element is subdividing into four homothetic quadrilateral elements (see figure 5) by adding nodes in the middle of no-saturated edges and in the barycentre of the element.

This refinement procedure is iteratively applied until no new node is added.

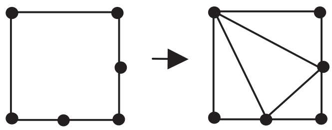


Fig. 4. Subdivision of quadrilateral element with two adjacent saturated edges

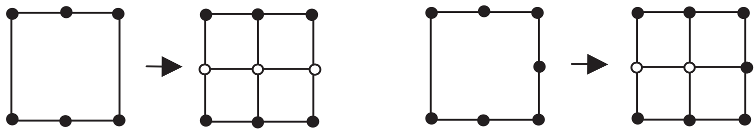


Fig. 5. Other cases of subdivision of quadrilateral element

From an algorithmic point of view, the mesh is composed of two types of element: ordinary and extraordinary. An ordinary element is a triangle or a quadrilateral without saturated edges (see figure 6). An extraordinary triangle is a triangle with one and only one saturated edge. An extraordinary quadrilateral is a quadrilateral with only one saturated edge or two adjacent saturated edges. Figure 7 shows extraordinary triangle and quadrilaterals. The remeshing algorithm must take into account these two element types. During the refinement operation, the geometrical criterion is applied to elements of both types. An ordinary or extraordinary element which is curved is then subdivided into four ordinary elements. After this operation, all ordinary elements with at least two saturated edges, except the case of two adjacent edges for quadrilateral elements, are iteratively subdivided into four ordinary elements. Then, all the elements with at least one saturated edge are transformed to extraordinary elements and the other elements remain unchanged.

At the end of the refinement operation, for the mechanical computational purpose, the extraordinary elements of the resulting mesh are transformed : an extraordinary triangle is divided in two triangles, an extraordinary quadrilateral with one middle node is divided in three triangles and an extraordinary quadrilateral with two middle nodes is divided in four triangles.

Fig. 6. Ordinary elements

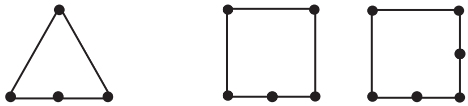


Fig. 7. Extraordinary elements

The coarsening technique is the reciprocal operation of the refinement procedure. It can only be applied to a set of four ordinary elements, called associated elements, obtained during a homothetic element refinement. Thanks to the coarsening technique, the initial element is restored when the area in which this element belongs, becomes flat (see figure 8).

Fig. 8. Triangular and quadrilateral elements coarsening

A quad tree structure can be considered to coarsening the mesh of the part. This structure allows to quickly localize associated elements. Each root of the tree is an element of the initial mesh and each edge is an intermediate element created during the refinement procedures. The leafs of the tree are elements of the current mesh and are brought together if they are associated. Figure 10 presents the quad tree structure associated to the mesh of the part on figure 9 at iteration 3. Elements whose number is underlined, are extraordinary elements.

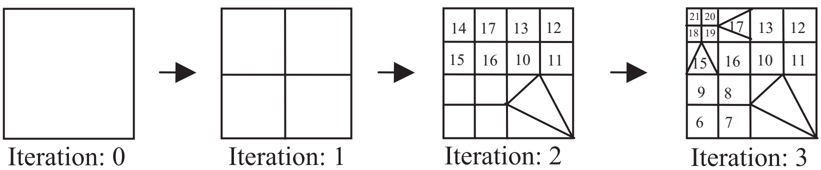


Fig. 9. Example of adaptive remeshing during three iterations

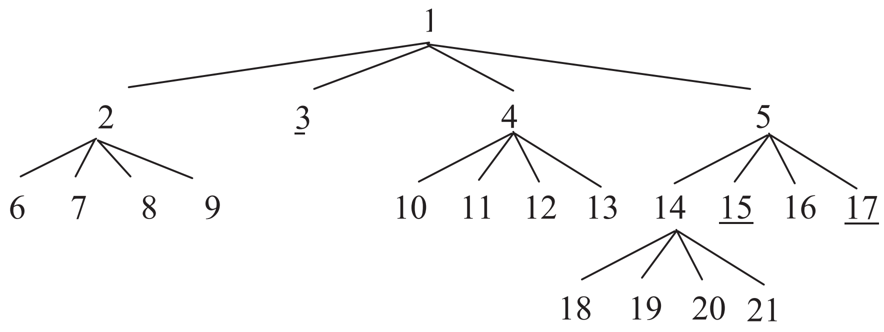


Fig. 10. Quad tree structure for the mesh of the part of figure 9

The coarsening technique is only applied to the leafs of a same level which are brought together. In the above example, associated elements which could be coarsened are: (6, 7, 8, 9) or (10, 11, 12, 13) or (18, 19, 20, 21). As in the refinement procedure, to ensure the mesh conformity, some coarsened elements could be refined if necessary. This last operation can only be applied when all the flat areas have been coarsened by the coarsening technique.

2.3 Transfer of Mechanical Fields

During the refinement procedure, the mechanical fields are simply induced from the current mesh to the new mesh. During the coarsening procedure, the mechanical fields associated to four associated elements are averaged and the result is associated to the new element. During the refinement procedure, the mechanical fields of curved elements are simply associated to the four new created elements from the subdivision.

3 Numerical Examples

3.1 Sheet Metal Stamping

An early application of adaptive mesh refinement was the simulation of 3-D sheet metal stamping example (Benchmark square box of Numisheet'93). According to Onate et al. [14], the geometric data of the square cup are: drawing depth 40 mm, sheet dimensions 150×150 mm^2, thickness h^0 = 0.78 mm, friction coefficient between the sheet and rigid tools is assumed to be $\mu = 0.144$ and the blank-holder force F = 19600 N. The material model used is an isotropic elastoplastic von-Mises model with multi-linear isotropic hardening approximating a power law yield stress curve defined as $\sigma = 567.29(0.07127 + \bar{\varepsilon}^{p})^{0.2637}$. The punch velocity is 20 mm/s and its stroke is 80 mm. The tools (punch, die and blank-holder) are supposed rigid and modeled by discrete rigid surfaces. Two examples are presented: the first example concerns the stamping of square sheet in which the angle θ between initial sheet plane frame (X,Y) and the tools orientation (x,y,z) is θ = 0° (see Figure 11a) and the second concerns the stamping of square sheet with θ = 45° (see Figure 11b). In these two cases, the solver 3D ABAQUS/EXPLICIT has been used. The element size adaptive discretization of the deformable sheet uses h_{min} = 0.75 mm, geometrical criterion = 8°.

Meshes adapted to the part curvature corresponding to different punch displacement (u = 6, 15, 24, 30, 36, 45 and 48 mm) are shown in Figures 12 to 18 for θ = 0° and θ = 45°. We can note that, the initial blank sheet is computed using an initial coarse mesh (100 quadrilateral elements), the mesh is again refined uniformly and the adaptive mesh refinement procedure is activated where elements are created automatically in regions of large curvature to even more accurately represent the complex material flow (large stretching) around the punch and die radii. The final contour of the sheet for 60 mm of punch displacement is presented in Figure 22 for θ = 0° and in Figure 23 for θ = 45°. We can note the final shape of the sheet is completely different due the initial sheet orientation.

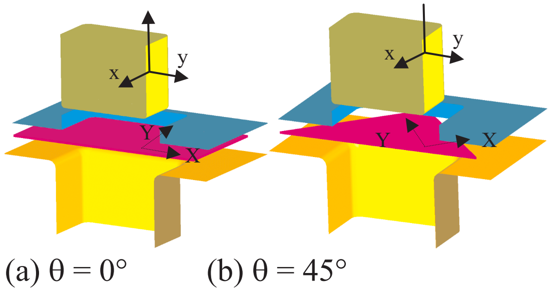


Fig. 11. Tools and initial sheet orientation

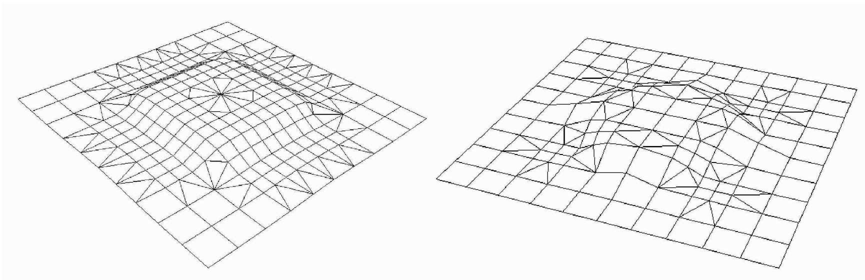


Fig. 12. Displacement u = 6 mm

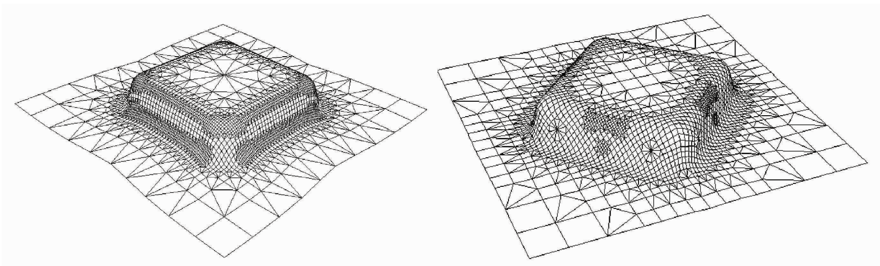


Fig. 13. Displacement u = 15 mm

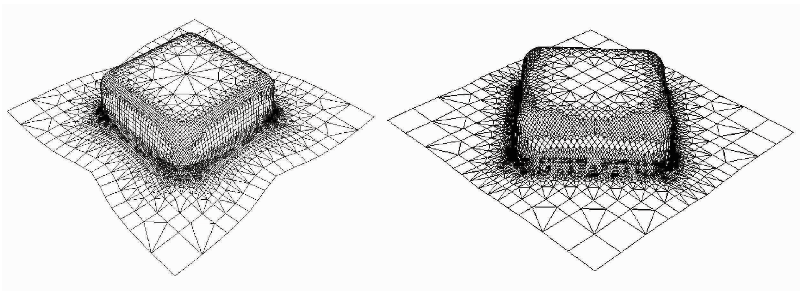


Fig. 14. Displacement u = 24 mm

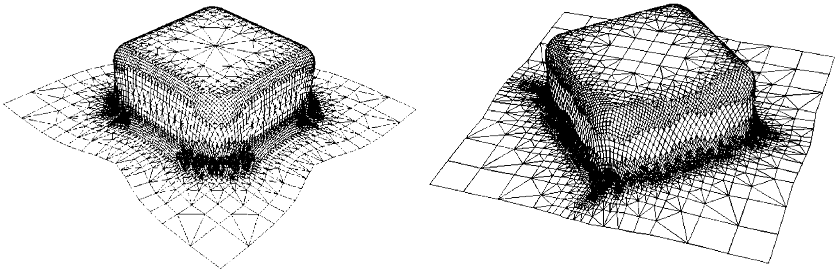


Fig. 15. Displacement u = 30 mm

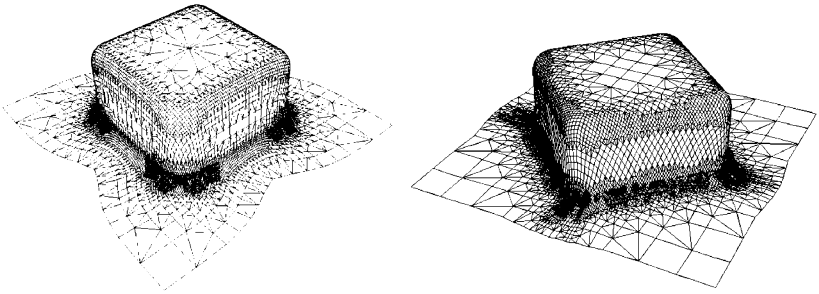


Fig. 16. Displacement u = 36 mm

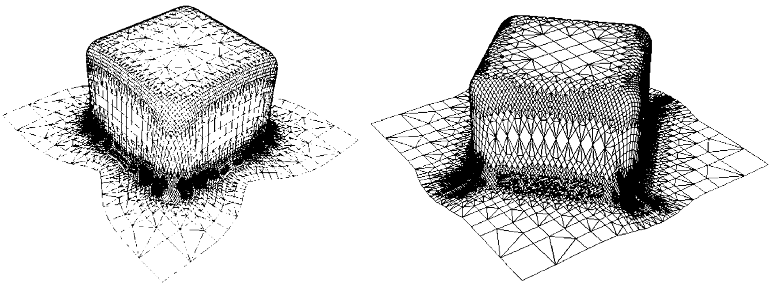


Fig. 17. Displacement u = 45 mm

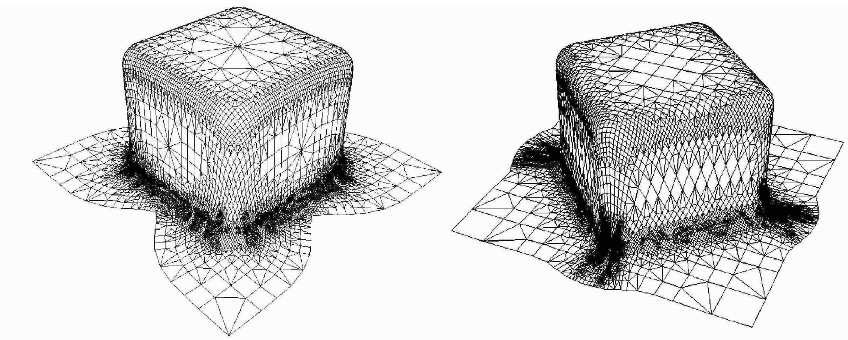


Fig. 18. Displacement u = 48 mm

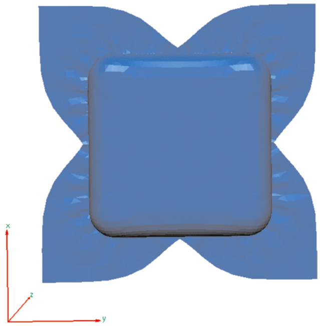


Fig. 19. Final shape for θ = 0°

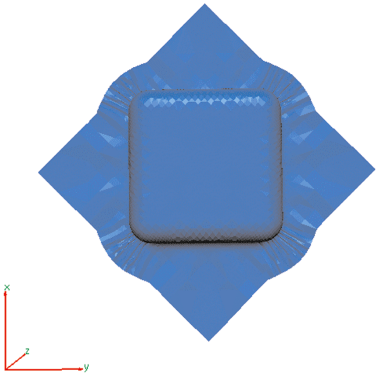
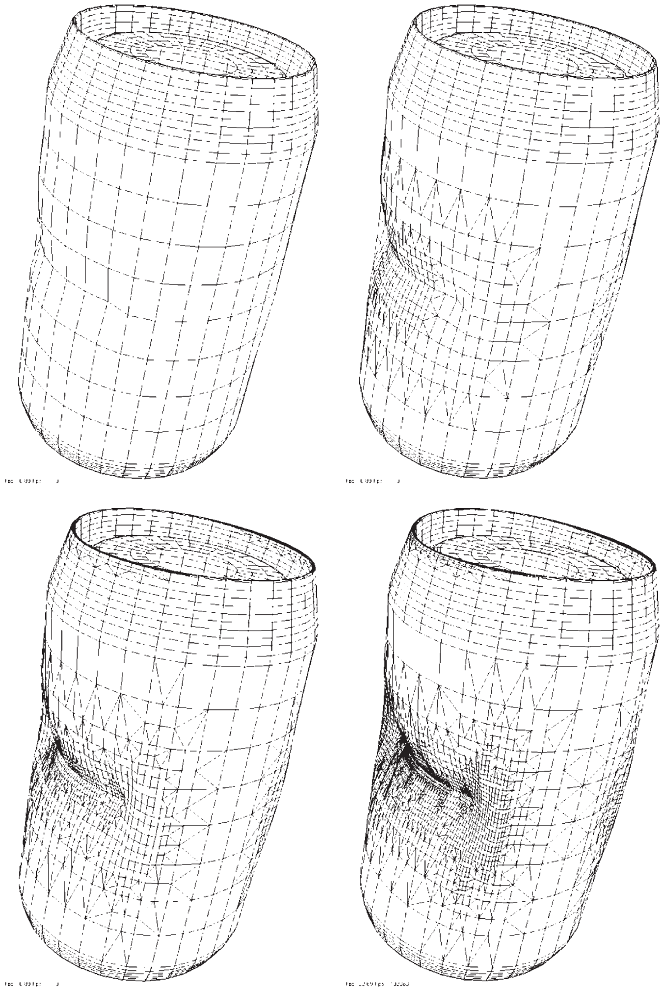


Fig. 20. Final shape for θ = 45°

3.2 Crushing of a Thin Cylinder

The second example is the crushing of a thin cylinder. The cylinder blank has initially 140 mm length, 44 mm diameter and 0.5 mm thickness. The initial mesh of the cylinder is constituted by 2048 quadrilateral sheet finite elements. Two concentrate loads diametrically opposite was prescribed using a linear ramp to simulate the crushing operation. The deformation evolution of the blank is illustrated in Figure 21. Here, the mesh refinement is localized on large deformed blank areas. The final mesh of the blank contains 30301 quadrilateral and 26714 triangular elements.



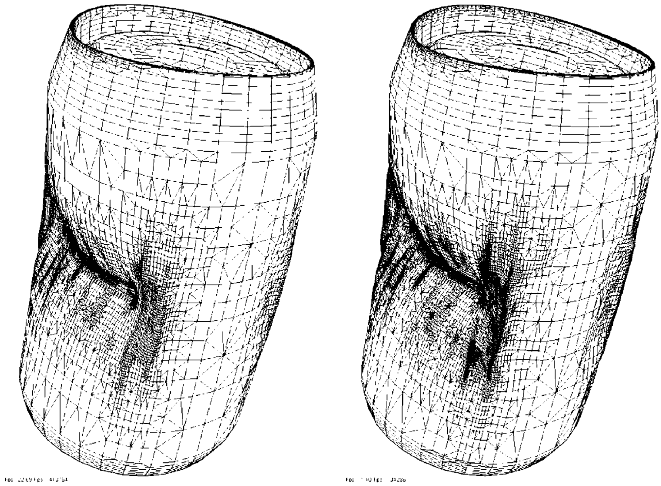


Fig. 21. Deformed cylinder for different crushing step

4 Conclusions

The different steps necessary to the remeshing of the computation domain in large elastoplastic deformations in three dimensions have been presented. The proposed adaptive remeshing technique is based on refinement and coarsening procedures using ageometrical criterion. This approach has been implemented with triangular and quadrilateral elements in the ABAQUS code. Numerical simulations of thin sheet metal forming process in three dimensions have validated the proposed approach and proved its efficiency. The extension in three dimensions for massive structure metal forming is currently under progress.

References

1. H. Borouchaki, A. Cherouat, K. Saanouni, A. Cherouat and P. Laug, “Remaillage en grandes deformation. Applications à la mise en forme de structures 2D”, Revue Européenne des éléments finis (to appear), 2002.

2. K. Saanouni, A. Cherouat and Y. Hammi, “Optimization of hydroforming processes with respect to fracture”, Esaform, Belguim, 1, 361-364, 2001.
3. P. Ladeveze and J.P. Pelle, “La maîtrise du calcul en mécanique linéaire et non linéaire, études en mécanique des matériaux et des structures”, Hermès, Paris, France, 2001.
4. D. Djokovic, “Splines for approximating solutions of partial differential equations”, Ph.D. Thesis, University of Twente, Enschede, ISBN 90-36510511, 1998.
5. A. Huerta, P. Diez, A. Rodriguez-Ferran, “Adaptivity and error estimation”, Proceedings of the 6th International Conference on Numerical Methods in Industrial Forming Processes, J. Huétink and F.P.T. Baaijens (eds), Balkema, Rotterdam, 63-74, 1998.
6. R. Drake, V.S. Manoranjan, “A method of dynamic mesh adaptation”, Int. J. Num. Meth. Eng., 39, 939-949, 1996.
7. O. C. Zienkiewicz and J. Z. Zhu, “Adaptivity and mesh generation”, in Int. J. Numer. Methods Eng. 32, 783-810, 1991.
8. L. Fourment and J.-L. Chenot, “Adaptive remeshing and error control for forming processes”, Revue européenne des éléments finis 3, 2, 247-279, 1994.
9. T. Coupez, “Génération de maillage et adaptation de maillage par optimisation locale”, Revue européenne des éléments finis 9, 4, 403-422, 2000.
10. H. Borouchaki, A. Cherouat, P. Laug and K. Saanouni, “Adaptive remeshing for ductile fracture prediction in metal forming”, C.R. Mecanique 330, 709-716, 2002.
11. Y. Li, I. Babuska, “A convergence analysis of a p-version finite element method for one-dimensional elastoplasticity problem with constitutive laws based on the gauge function method”, J. Num. Anal., 33, 2, 809-842, 1996.
12. J.-W. Cho and D.-Y. Yang, “A mesh refinement scheme for sheet metal forming analysis”, Proc. of the 5th International Conference, NUMISHEET’02, 307-312, 2002.
13. T. Meinders, “Developmsents in numerical simulations of the real life deep drawing process, Ph.D. Thesis, University of Twente, Enschede, ISBN 90-36514002, 2000.
14. E. Onate, J. Rojek, C. Garcia Garino, Numistamp: "A research project for assessment of finite element models for stamping process", Journal of Materials Processing Technology 50, 17-38, 1995.