Journal
of Achievements in Materials and Manufacturing Engineering
VOLUME 49
ISSUE 2
December
2011

Numerical and experimental true strain assessment on sheet forming using mapped versus free meshing

S.F. Lajarin, J.S.F. Magalhães, P.V.P. Marcondes*
Mechanical Department, Universidade Federal do Paraná,
Av. Cel. Francisco H. dos Santos 210, CEP 81531-990, Curitiba - Paraná - Brazil
* Corresponding author: E-mail address: marcondes@ufpr.br

Received 17.10.2011; published in revised form 01.12.2011

Analysis and modelling

ABSTRACT

Purpose: The main aim of the present study was to analyze the influence of the algorithmic theories of generation and control of triangular, quadrilateral, hexahedral and tetrahedral meshing, i.e. which are the most common types of meshes used in the software of finite elements for large plastic deformation. The importance of these methods is due to the fact that they are the spine of Finite Element Methods (FEM).

Design/methodology/approach: It was numerically evaluated the parameters influencing mapped (structured) and free meshing on sheet forming simulation (stretching). For the tests a stretching tool with geometry proposed by Nakazima was used. The study presents the results in terms of the major true strains (ε_1, ε_2, ε_3) and a comparison with experimental data was carried out (validation).

Findings: The analysis showed that Shell-type elements are dependent of the element format choice and the way of application in the geometry. Objects built with Shell type elements, i.e. components that will suffer large plastic deformation are extremely sensitive to the mesh format, refinement and way that it was applied. A relationship was also shown among equivalent meshes for elements in the format Tri and Quad.

Research limitations/implications: To describe the complete influence of the type of meshing are beyond the scope of this study as it was used only one commercial software and one method of forming.

Practical implications: The correct choice of the meshing parameters can provide more accurate results during the simulations of sheet stretching process.

Originality/value: The paper shows the differences and implications of the correct choice of meshing during finite element analysis.

Keywords: Numerical modelling; Meshing; Sheet forming; True strain; Stretch forming

Reference to this paper should be given in the following way:

S.F.Lajarin, J.S.F. Magalhães, P.V.P. Marcondes, Numerical and experimental true strain assessment on sheet forming using mapped versus free meshing, Journal of Achievements in Materials and Manufacturing Engineering 49/2 (2011) 323-330.

1. Introduction

Manufacturing processes such as forming, machining and welding are widely used in industry and have always been very dependent of trial-and-error procedures. However, in recent decades, the employment and the development of mathematical modelling, numerical and computational methods are appearing as techniques for a significant reduction of cost and time [1-4].

There are several engineering methods developed for the analysis of sheet metal forming. However, a more efficient analysis of the effects of the process parameters and materials has been possible by using Finite Element Methods (FEM). The FEM method consists of transforming a complex problem into a number of simpler problems. For the correct solution, the chosen model should be appropriate to the problem parameters, i.e. especially the element geometry for meshing. Zeid [5] defined the methods of mesh generation as the FEM dorsal line.

In numerical simulation, the meshes can be mapped (structured) or free. Usually four mesh formats are used: triangular and quadrilateral suitable for 2D elements and tetrahedral and hexahedral for 3D elements. Also important is the mesh post-processing technique, which can include mesh smoothing, cleaning and refinement.

The present work was focused on the meshing procedures. Due to the breadth of the free mesh generation field, this research was limited on the study of the current meshing generation capabilities (Ansys software).

1.1. Meshing

Before the development of the preprocessors, the finite element meshes were generated manually. Zeid [5] reported that the manual meshing (free) was inefficient and susceptible to mistakes. For complex 3D objects, the meshing procedure can become complex (hourglass effect). The actual preprocessors provide a great variety of algorithms, outlines, and methods for meshing generation. They show several automation levels for the different user inputs.

The most important criterion in meshing generation is the mesh perfection. According to Owen [6] the nodes must be placed inside or on the outlines of the geometric model to be worked out. It is also desirable to have a library with a variety of elements to allow flexibility for users. Automatic mechanisms to regulate the meshing variations in transition areas and easy smoothing and density control are also needed. Some mechanisms exist to convert a mesh of an element type into another type, for instance: with 2D meshes it is always possible to convert a triangular element into three quadrilateral elements (one tetrahedral can be subdivided in four hexahedral) or to combine two triangular elements to produce a quadrilateral element. A mesh of quadrilateral elements can be converted into a mesh of triangular elements dividing each quadrilateral into two triangles. The mesh should agree with the geometry and topology of the object. In resume, a method of meshing generation is inherent to the geometric model to be worked out. For solid models we can conduct the meshing generation completely automatic. The time taken to generate a mesh and the time taken to execute FEM is crucial.

1.2. Mapped vs. free meshing

Zienkiewics and Taylor [7] have shown that basic elements, uni-, bi- or tri-dimensional can be mapped in simple or complex geometries. A mapped mesh is easily identified for having all their interior nodes with a similar number of adjacent elements. A mapped mesh generator is typically defined in the quadrilateral (Quad) or hexahedral (Hex) format. According to Owen [6] the mapped mesh generators are usually used where a rigid alignment of the elements is requested.

For free meshing, usually triangular (Tri) and tetrahedral (Tet) meshes are chosen, although quadrilateral (Quad) and hexahedral (Hex) can also be free.

Certainly there are countless interactions among the technology of generation of mapped and free meshing; however the main characteristic that distinguishes the two fields is the interactivity that smoothing algorithms use through the generators of mapped meshing [6].

1.3. Triangular/tetrahedral meshing

The triangular element was the first element type developed for 2D solids and its formulation is the simplest. Liu and Quek [8] reported that the use of triangular element can give less accurate results when compared to quadrilateral elements. Due to that, someone can imagine that the ideal is always to use quadrilateral elements, but the reality is that the triangular element is still a very useful element for adaptation in complex geometries [9,10].

Usually, triangular elements are used to mesh 2D complex geometries involving deep corners. There are automated meshing programs that can generate a quadrilateral mesh, but they still use triangular elements as some kind of patches for difficult situations, and finishes with a mesh of combined elements [8].

The tetrahedron is a tri-dimensional element, but shows similar properties of the triangular elements. These are without a doubt the most common form of free meshing generation. Nowadays, the techniques can be classified into three main categories [6].

The first one is the Octree method where the cubes contained in the geometric model are recursively subdivided until the required resolution. The second is the Delaunay method that uses a typical approach of elements Tri for the initial mesh border. The new nodes are inserted incrementally and the triangles or tetrahedrons are locally redefined for each new node. Finally, the third is the Advancing Front method. In this method the tetrahedrons are built inside the triangular surface progressively. A bi-dimensional example is an area outlined by Tri elements and later irregularly filled out by others Tri. In three-dimensions, for each triangular surface the computer defines an ideal place for a forth new node. The Fig. 1 illustrates the three mesh generation criteria.

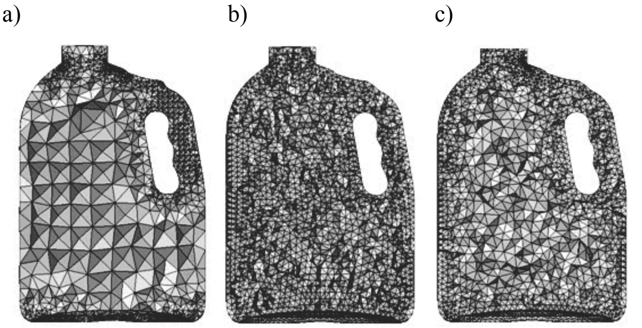

Fig. 1. Meshing flexibility: Octree = Robust, (b) Delaunay = Fast and (c) Advancing Front = Smoothed (adapted from Ansys Homepage)

1.4. Quadrilateral/hexahedral meshing

Due the smallest efficiency of the triangular meshing and with the meshing algorithms progresses many models of complex geometry, with sharp corners or curved extremities, can be simulated using quadrilateral elements [8].

Hutton [11] reported that quadrilateral elements are more convenient for regular geometries and they could be used with triangular elements. When applicable Quad or Hex mapped meshing will usually produce better results. However, for the mapped meshing to be applicable, the opposite extremity of the meshed area needs to have a similar number of divisions. In 3D models, each cube contrary face needs to have the same meshing on the surface. This can frequently be impossible for an arbitrary geometric configuration or it can involve the user's considerable interactions to decompose the geometry in areas of mapped meshing [6].

The algorithms for Quad free meshing can usually be described as direct and indirect approaches. With an indirect approach, the domain is meshed first with triangles and then several algorithms are used to convert the initial triangles in quadrilateral elements, Fig. 2. In the direct approach the quadrilateral elements are generated directly [6].

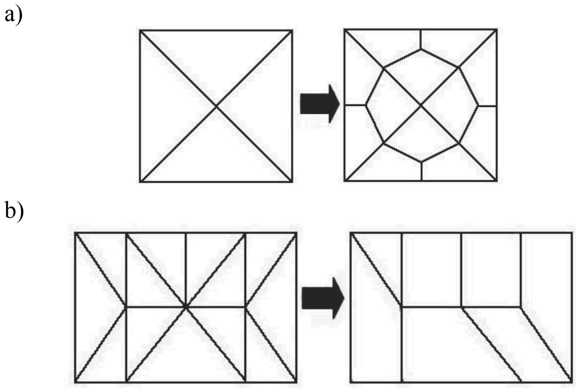

Fig. 2. Meshing approaches: (a) Quad mesh generated by the division of each triangle in three Quads and (b) Quad-dominant mesh generated by combined triangles

Similar to the quadrilateral meshing, there are direct and indirect methods for free hexahedral meshing. In the indirect methods each tetrahedron, in a solid, can be subdivided in four hexahedrons (Fig. 3) or can be used a composition of tetrahedral in order to form hexahedral elements. In the direct methods, Owen [6] showed four strategies for the generation of hexahedral meshes. The grid-based method consists in a tri-dimensional adjustment of hexahedral elements inside the volume. Hexahedrons are added to the outlines to fill out the openings where the regular grating of hexahedron does not have coherence with the surface. The midlles surface method involves an initial decomposition of the volume similar to the method of quadrilateral meshing. However, it is limited for most of the geometries. The method consists to increase elements beginning from the border and moving forward to the center of the volume. Individual quadrilateral elements are projected for the interior of the volume in order to form hexahedrons. The whisker weaving method is an arrangement of interlaced surfaces that shows bifurcating hexahedral elements in each one of the directions. The algorithm objective is to determine where the bending plans intersections will happen. A hexahedron will be formed on a converging position of the three plans of bending.

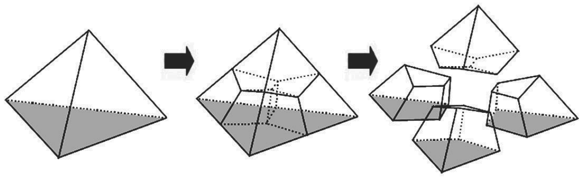

Fig. 3. Decomposition of one tetrahedral into four hexahedral elements

For metal forming simulation the use of hexahedral elements can offer advantage compared with other types of elements. A disadvantage is the difficulty of meshing generation.

Wisselink [12] showed some suggestions to create a hexahedron mesh, like: divide the geometry in simple sub domains generating a mesh with a mapped method or sweeping; to use a generator of tetrahedral mesh developed for direct generation or to use the combination of a quadrilateral mesh surface and a simple hexahedral mapped mesh inside the volume. Owen [6] presented that hexahedral elements should advance as far as possible inside of the volume and the remaining empty space should be filled out with tetrahedron elements.

1.5. Post-processing meshing

Normally, the mesh generation needs a post-processing procedure to improve the global quality of the elements. The main categories of mesh improvement include smoothing, cleaning and refinement. Most of the smoothing procedures involve some form of interactive process that adds individual nodes to improve the local elements quality. A wide variety of proposals of smoothing techniques exist. Wisselink [12] reported that smoothing algorithms take into account as criterion for the nodes improvement the form of the element, i.e. angle, size and position.

According to Owen [6] cleaning methods usually apply two criteria. As improvement criteria for triangular meshes are frequently executed simple diagonals changes. For meshes with tetrahedron, some local transformations are projected to improve the quality of the element. These transformations can include the changing of two adjacent interior tetrahedrons that share the same face for three tetrahedrons, or equally, three tetrahedrons can be substituted by two tetrahedrons. The topology improvement criterion is a method to try to improve meshes by decreasing the number of extremities that share the same node.

Refinement is defined as any operation executed that reduces the size of the local element. The size reduction can be demanded in order to capture a local physical phenomenon, or simply to improve the local quality of the element. Usually the process begun with a rough mesh and refinement procedures are applied until the desired node density is reached. The material deformation and the element distortion become a limit to calculation. Forming process usually requires techniques to correct large deformation. Procedures of automatic 2D and 3D remeshing are alternative techniques to the traditional Delaunay/Frontal methods. This method is based on geometrical and topological parameters optimization and proceeds by local change. Sometime, local change can be more efficiently remeshing procedure rather than rebuilt the whole meshing [13]. This local change is based on the combination of local improvement of the neighborhood of nodes and edges [14].

In this work the main objective was to evaluate the influence of different meshing types on a structural explicit solid/rigid analysis using Solid element-type (punch, die and blank holder), and structural explicit thin shell analysis using Shell element-type (sheet). The process was evaluated for large deformation of sheet metal (stretching) with tools of simple geometry.

2. Experiments

The ANSYS 9.0/LS-DYNA software was used. The following options for the Shell163 elements were used: S/R Hughes-Liu type formulation, number of integration points in the thickness direction equal to five and Gauss quadrature as integration rule. The meshing was varied in the sheet and punch, Table 1.

Table 1.
Meshing condition

	Punch mesh		Sheet mesh	
	format	Nº of elements	format	Nº of elements
Case 1	Hex	192	Quad	1600
Case 2	Hex	192	Tri (mapped)	3200
	Hex	192	Tri (free)	3631
Case 3	Tet	1133	Quad	1600

As the meshing can be mapped or free four different combinations for simulations was carried out, Table 1. The sheet meshing was varied from mapped and free condition when it was composed by Tri-type elements. In Tri-type condition it was observed considerable differences. Mapped meshing was always used with Quad-type elements and when Hex and Tet were applied on the punch. In these cases they did not show any differences between the conditions mapped and free.

2.1. Pre-processing

Geometry

In the numerical simulation of forming process usually we have four involved bodies: punch, die, blank holder and sheet, Fig. 4a. Due the symmetry a quarter of the geometry was considered, Fig. 4c.

Mesh formulation

Table 1 shows the number of elements used for the punch and the sheet. For the sheet, the number of elements was defined so that the sheet with Tri meshing format had approximately the double of elements of that meshed with Quad format. It was considered that two Tri forms a Quad, based on the indirect method of generation of Quad free mesh [6]. For the punch the adopted criterion was to have the same amount of face divisions between the elements Tet and Hex. In this case, the punch was tri-dimensional and the amount of elements was larger, but the proportionality was the same.

Material Properties

A series of experiments were carried out by Chemin and Marcondes (2008) to ascertain the Forming Limit Curve of 0.7 mm thick DC 06 steel (DIN 10152) using the test proposed by Nakazima. The anisotropic material properties are presented in Table 2. In the LS-DYNA software the material model selected was the Barlat and Lian.

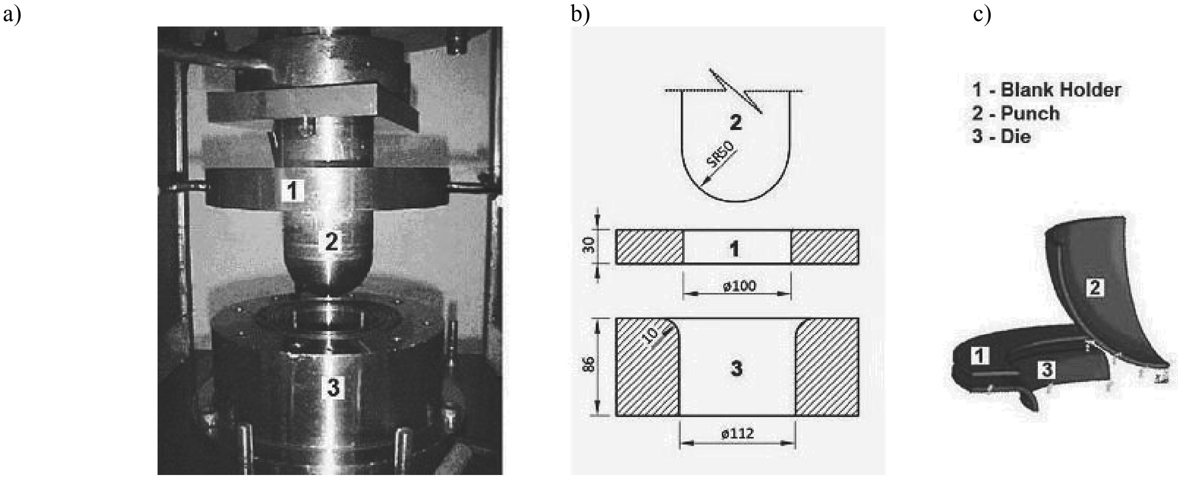

Fig. 4. Experimental tool proposed by Nakazima: (a) adapted from Chemin and Marcondes (2008), (b) tool dimensions in mm and (c) FEM model

2.2. Processing and pos-processing

Numeric error can be defined as the difference between the exact analytical solution of a certain variable of interest and its numeric solution. The main processes to estimate and to evaluate the error in simulation programs are called verification and validation. While the verification is the evaluation of the computational solution accuracy in relation to the numerical model; the validation seeks to determine the proximity of the mathematical model of the real phenomenon, i.e. through the comparison of the numerical solution with the experimental data [17].

In this work, the values of the true strains (major ε_1, minor ε_2 and thickness ε_3) obtained by simulation were compared with that ones presented by the simulation of the case 1 (Table 1), i.e. assumed as a reference pattern for validation, Silva [15]. These values were taken as a reference because their coherence with the experimental results done by Chemin [16]. In this work it was just considered the points of largest strain. It was used a Laptop with Sempron 3100 processor and memory RAM of 512GB.

3. Results and discussions

Case 1 - Sheet Quad and punch Hex

The numerical simulation was reproduced according to Silva [15]. Fig. 5a shows the major true strain located on the punch pole showing that the true strain was idealized concerning the tribological conditions. The Fig. 5d, punch with Tet meshing format and sheet with Quad meshing format showed the best results, i.e. very similar to the Case 1 (pattern). Table 3 presents all the true strain values and the percentage deviation.

Case 2 - Sheet Tri and punch Hex

In this condition the sheet meshing format was varied of Quad for Tri (mapped). It was observed that the major deformations migrate away from the punch pole, Fig. 5b. The true strain $\varepsilon1$ was of 0.902 and Table 3 shows 0.179 for ε_2 and -0.945 for ε_3; the true strain values are considerable away from the reference values. This deviation was directly influenced by the elements Tri applied on the sheet showing that they have larger instability to analyze true strains when compared to Quad type mesh format.

Figure 5c illustrates the case of the application of Tri meshing format (free) on the sheet. It was observed that the values lifted up 340% to ε_1, showing the material failure (Table 3). This value is still more discrepant than the previous situation. In that case it is also observed that the strain values did not present uniform behavior, i.e. the true strain migrated from the punch pole to the borders. Probabily, due to the distribution of the elements on the sheet as the software chooses the 'best way' of distributing the elements on the object. Here could be questioned the use of only a quarter of the geometry in the simulation mainly when it was meshed by elements with format Tri-type. In this case, the borders of quarter geometry do not exist physically. They are only a symmetrical approach of the real case for reduction of the computational time.

Case 3 - Sheet Quad and punch Tet

The punch meshing type was varied of Hex for Tet format. It was observed that the larger strain was located again in the punch top (punch pole), Fig. 5d. Table 3 shows ε_1, ε_2 and ε_3 of 0.485, 0.457 and -0.943, respectively. These values are very close of the referential ones. This results showed, like expected, that solid elements (Solid164/rigid) suffer little interference of the meshing format.

Table 2.
Material properties [15]

Property	Value	Unit	Source
Density (ρ)	7.850	g/cm3	Literature
Elasticity Module (E)	210	GPa	Literature
Poisson (ν)	0.3	(dimensionless)	Literature
Plastic Resistance constant (K)	626.8	MPa	Chemin and Marcondes [16]
m	6	(dimensionless)	Barlat e Lian Model
Anisotropic Coefficient 0° (R0)	2.048	(dimensionless)	Chemin and Marcondes [16]
Anisotropic Coefficient 45° (R45)	1.866	(dimensionless)	Chemin and Marcondes [16]
Anisotropic Coefficient 90° (R90)	2.599	(dimensionless)	Chemin and Marcondes [16]

Table 3.
True strain as a function of the meshing type

	Meshing conditions	ε_1	ε_2	ε_3	Percentage difference in relation to the case 1 (reference)		
					ε_1	ε_2	ε_3
Case 1	Punch Hex Sheet Quad (Mapped) Ref.	0.451	0.424	-0.873	-	-	-
Case 2	Punch Hex Sheet Tri (Mapped)	0.902	0.179	-0.945	100%	-42%	8.2%
	Punch Hex Sheet Tri (Free)	1.537	0.528	-1.352	340%	12.4%	15.5%
Case 3	Punch Tet Sheet Quad (Mapped)	0.485	0.457	-0.943	7.5%	7.8%	8%

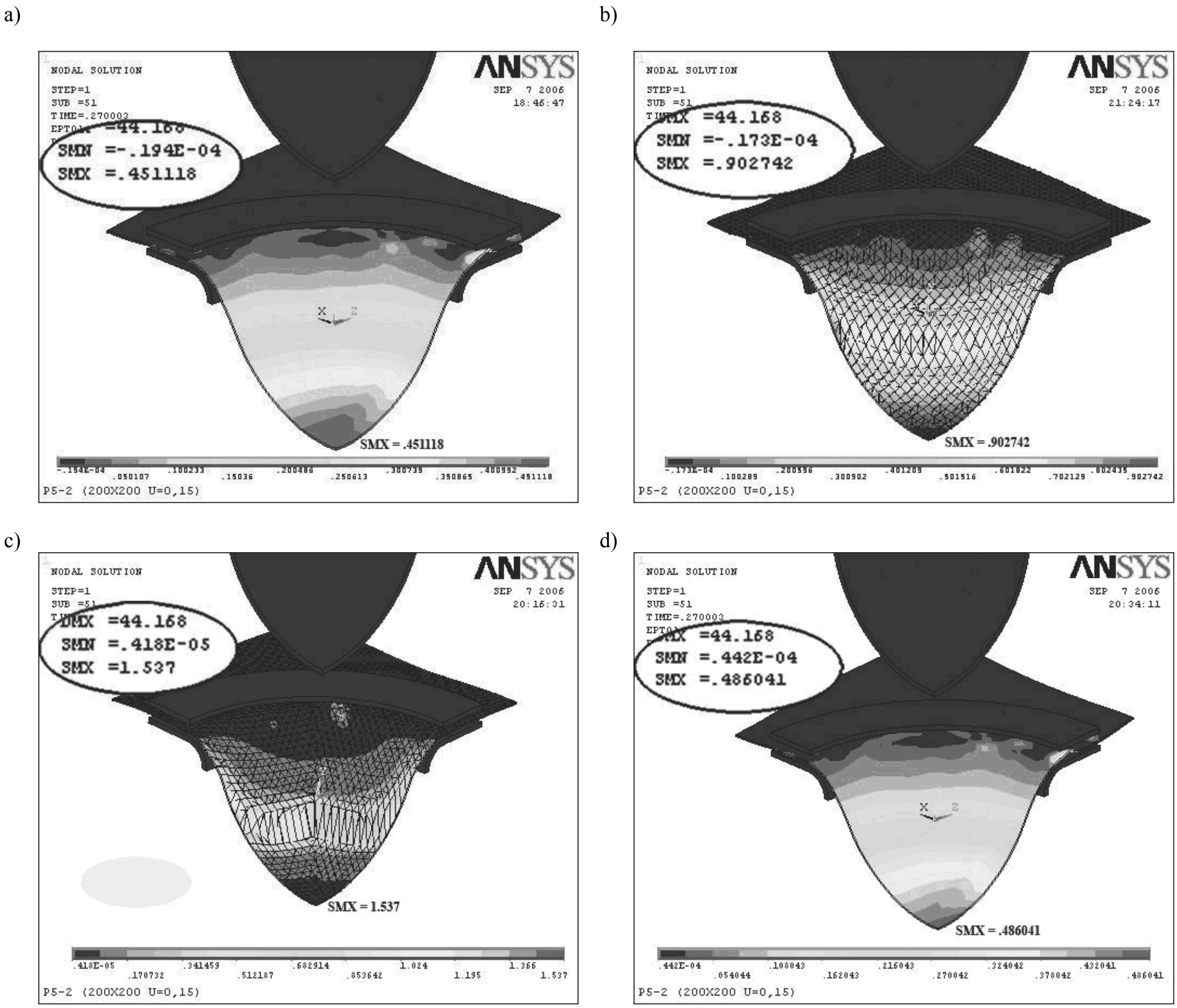

Fig. 5. Meshing variations: (a) Case 1 (reference pattern) sheet Quad and punch Hex - Silva [15], (b) Sheet with Tri mapped meshing and punch Hex, (c) Sheet with Tri meshing free and punch Hex and (d) Sheet with Quad meshing and punch Tet

Time of processing

The simulations times were of 17 min in case 1, 10 min in case 2 and 15min in case 3 with computational time closer of the case 1 (reference). It was confirmed that for solid elements the mesh format type variation does not interfere considerably. The sheet elements processed with format Tri (cases 2) presented a reduction of almost 50% in the time of simulation but showed high deviation for ε_1, ε_2, ε_3 from de referential (case 1).

Case 4 - Sheet mapped Tri and punch Tet

As defined in the experimental procedure the criterion used to define the sheet elements amount was according with the number of edge division. In order to equalize the processing time, the punch was meshed with Tet format and the sheet with mapped Tri format, i.e. the opposite of the reference (case 1).

This experiment was an empirical procedure in order to find a relationship between refinement (as mentioned in the section 2.4) and computational time that could approximates the results of those two opposed cases. The attempt consisted of a progressively increase of the number of divisions and consequently refining of sheet meshing. The approximation occurred for the value of 54 divisions by edge, being 35% larger the refinement, providing a number of 5832 elements (Tri) applied on the sheet against the 1600 elements applied on case 1 (Table 4). The simulation time was close of the case 1 showing that this configuration needs more mesh refinement but the computational time was not significantly increased.

Table 4.
Empirical vs referential result convergence

	ε_1	ε_2	ε_3
Case 1	0.451	0.424	-0.873
Case 4	0.450	0.400	-0.840
Percentual difference	0%	- 4.7%	- 3.7%

Design Implications - Considerations to the Tri meshing format

For a Tri meshing format it should be observed that the ANSYS/LS-DYNA software has available options for the alteration of the meshing parameters. One of those choices will define the way that elements Tri will be applied on the object. The Fig. 6 illustrates the configuration.

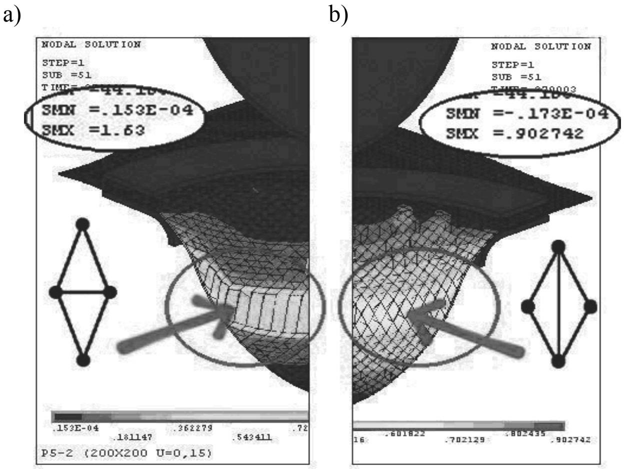

Fig. 6. Effect of the parameters choice for Tri meshing format: (a) software default configuration and (b) correct choice of parameters for improved meshing

Figure 6a shows the result with a Tri mapped meshing without configuration of additional parameters (software default), i.e. automatic. In the Fig. 6b the parameters of meshing were altered. It is worth noted that if the mesh was not aligned in a correct way the results could be considerably changed, even indicating a premature material failure.

4. Conclusions

This work presented some meshing considerations with the objective of analyze the criteria used by ANSYS software on the meshing application on stretch forming. Regarding the alteration of generated meshing, it was observed that ANSYS 9.0 offers some change possibilities based in the following methods: re-meshing with new size specifications and element format, cleaning the mesh, redefine mesh control and local meshing refinement and mesh improvement (work out with tetrahedrons format).

Besides the meshing size the meshing format also affects significantly the results convergence, Silva [15]. Objects built with elements of the type Shell163, i.e. components that will suffer large plastic strain are extremely sensitive to meshing format, refinement and way it was applied.

The considerable reduction of simulation time reached by the application of Tri format meshing in the sheet was unfeasible by the discrepancy of the true strain results achieved (ε_1, ε_2, ε_3).

The efficiency of quarter geometry in the simulation could be questioned for objects built by elements Tri under certain refinements. As presented, the areas with larger plastic strain were influenced by the distribution of the elements that begins from the borders.

In order to compare true strain results and simulation time, for the conditions of the present work, a meshed sheet with Tri format should be 35% more refined than with Quad format.

Acknowledgements

To the ESSS Company and Mr. Nicolau Botelho for the temporary supply of the Ansys 9.0 software.

References

[1] N. Cappetti, L. Garofalo, A. Naddeo, M. Nastasia, A. Pellegrino, A method for setting variables in Super Plastic Forming process, Journal of Achievements in Materials and Manufacturing Engineering 38/2 (2010) 187-194.

[2] A. Śliwa, L.A. Dobrzański, W. Kwaśny, M. Staszuk, Simulation of the microhardness and internal stresses measurement of PVD coatings by use of FEM, Journal of Achievements in Materials and Manufacturing Engineering 43/2 (2010) 684-691.

[3] M.J. Jackson, Numerical analysis of small recessed silicon carbide grinding wheels, Journal of Achievements in Materials and Manufacturing Engineering 43/1 (2010) 27-37.

[4] W. Kapturkiewicz, A.A. Burbelko, E. Fras, M. Górny, D. Gurgul, Computer modelling of ductile iron solidification using FDM and CA methods, Journal of Achievements in Materials and Manufacturing Engineering 43/1 (2010) 310-323.

[5] I. Zeid, Mastering CAD/CAM, Mac Graw Hill, New York, 2005.

[6] S.J. Owen, A survey of unmapped mesh generation technology, Proceedings of the 7th International Meshing Roundtable (1998) 239-267.

[7] O.C. Zienkiewics, R.L. Taylor, Finite Element Method, Solid Mechanics 2, Butterworth Heinemann, 2000.

[8] G.R. Liu, S.S. Quek, The Finite Element Method, A Practical Course, Butterworth Heinemann, Burlington, 2003.

[9] J.C. Cuillihre, An adaptive method for the automatic triangulation of 3D parametric surfaces, Computer-Aided Design 30 (1999) 2 139-149.

[10] W. Min, Z. Tangt, Z. Zhang, et al. Automatic mesh generation for multiply connected planar regions based on mesh grading propagation, Computer-Aided Design 28 (1996) 671-681.

[11] D.V. Hutton, Fundamentals of Finite Element Analysis, McGrawHill, 2004, 193-196.

[12] H.H. Wisselink, Analysis of guillotining and slitting-finite element simulations, Netherlands, 2000.

[13] J.-L. Chenot, E. Massoni, Finite element modelling and control of new metal forming processes, International Journal of Machine Tools & Manufacture 46 (2006) 1194-1200.

[14] R. Boussetta, T. Coupez, L. Fourment, Adaptive remeshing based on a posteriori error estimation for forging simulation, Computer Methods in Applied Mechanics and Engineering 195 (2006) 6626-6645.

[15] H.C. Silva, Numerical analysis of the test of Nakazima via Finite Element Method, Master thesis - PG-MEC, Federal University of Paraná, 2005 (in Portuguese).

[16] R.A.F. Chemin, Evaluation of sheet deformations and FLC curves for different geometries of punches, Master thesis - PG-MEC, Federal University of Paraná, 2004 (in Portuguese).

[17] W.L. Oberkampf, T.G. Trucano, C. Hirsch, Verification, validation and predictive capability in computational engineering and physics, Applied Mechanics Reviews 57 (2004) 345-384.