INFLUENCE OF THE MESH FORMAT, REFINEMENT AND WAY OF APPLICATION ON SHEET FORMING SIMULATION

Paulo Vitor Prestes Marcondes *
Sérgio Fernando Lajarin
Josef Stanislaw Falcon Magalhães
Universidade Federal do Paraná, DEMEC, Av. Cel. Francisco H. dos Santos, 210 CEP 81531-990, Curitiba, Paraná – Brazil
* marcondes@ufpr.br

Abstract. *The methods of mesh generation are the spine of the Finite Element Methods (FEM). The formats of meshes triangular, quadrilateral, hexahedral and tetrahedral are the most common types used in the dedicated softwares for simulation of great plastic deformations. In this work it was numerically evaluated the influences of the meshing type, refinement and the way of application in a metal sheet foming simulation for stretching. A validation was proposed, and with the refinement it was possible to reach similar results in cases of mesh simulation with different formats. The results are presented in terms of the true strain (ε_1, ε_2, ε_3) in the metal sheet. It was observed that elements of the Shell-type are dependent of the element format choice and the way of application on the geometry. According with the refinement is possible to compare results of meshes with different formats.*

***Keywords**: numeric simulation, meshing, sheet forming.*

1. INTRODUCTION

There are several engineering methods developed for deformation analysis on sheet metal forming. However, a more efficient analysis of the effects of the process parameters and materials has been possible with the use of finite element methods (FEM). This method consists transforming a complex problem in many others simpler problems, allowing any geometric form, loads and boundary conditions. FEM has the capacity to outline the analytical difficulties solving a mathematical problem showing complex geometries, where the equations are resolved through small elements of simple geometry (squares, triangles, beams, bars among other).

For the resolution the chosen model should be submitted to the problem parameters, especially the element geometry for meshing. Zeid (2005) presents the methods of generation of meshes as the FEM dorsal line.

In numeric simulation the meshes can be mapped or free. According to Owen (1998) the automatic generation of mesh, i.e., free is a field relatively new. Usually are used four formats of meshes: triangular and quadrilateral are suitable for the elements 2D and tetrahedral and hexahedral for the 3D elements. It is, also, important the mesh post-processing technique, that can include the mesh smoothing, cleaning and refinement.

The present work is focused in the meshing procedures seeking to qualify engineers and technicians. The objective was to study the refinements possibilities and the parameters and way of mesh application.

1.1. Meshing

Before the development of the preprocessors the finite elements meshes were generated manually. Zeid (2005) tells that the manual meshing (free) is inefficient and inclined to mistakes. For complex objects 3D the complexity of the meshing can increase and to become especially confused (hourglass effect). The actual preprocessors provide a great variety of algorithms, outlines, and methods for mesh generation. They have several automation levels for the different demands of users' input.

The most important criterion in mesh generation is to assure the validity and the perfection of the resulting mesh. It is important to observe the demands that produced the correct results in a FEM. Some are necessary while other optional ones. According to Owen (1998) the nodes must be placed inside or in the outlines of the geometric model to be worked out. It is desirable a library with great variety of elements to guarantee the users flexibility. Automatic mechanisms to regulate the meshing variations in transition areas and easy smoothing and density control. Mechanisms to convert a mesh of an element type in another type, for instance, in meshes 2D it is always possible to convert a triangular element in three quadrilateral elements (a 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 in a mesh of triangular element dividing each quadrilateral in two triangles. The mesh should agree with the geometry and topology of the object. A method of mesh generation is inherent to the geometric model to be worked out. Solid models guide the generation completely automatic. The time taken to generate a mesh and the time taken to execute FEM is crucial. To reduce both, it is important that the method of mesh generation improves the mesh and minimizes the number of nodes and elements of the mesh and still satisfying the conversion demands.

1.2. Mapped vs. Free meshing

Zienkiewics and Taylor (2000) 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 (1998) the mesh generators mapped are used usually where a rigid alignment of the elements is requested.

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

Certainly there is countless interactions among the technology of generation of mapped and free mesh, however the main characteristic that distinguishes the two fields is the interactivity that smoothing algorithms use through the generators of mapped mesh, Owen (1998).

1.3. Triangular/Tetrahedral Meshing

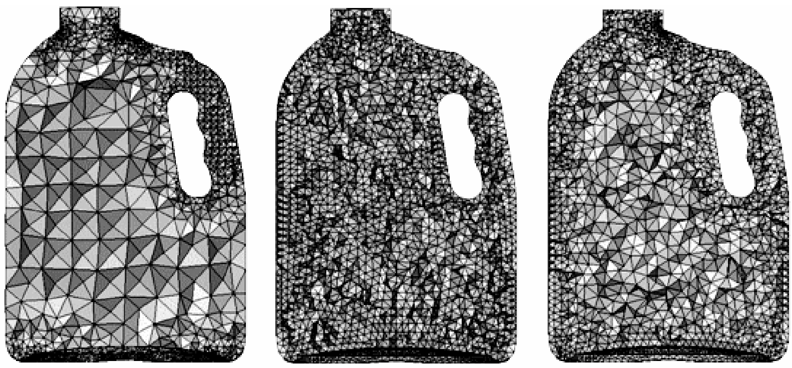
The triangular element was the first element type developed for solids 2D. The formulation is also the simplest. Liu and Quek (2003) affirm that the triangular element is less accurated compared to quadrilateral elements. Due to that, someone can usually imagine that the ideal is always to use quadrilateral elements, but the reality is that the triangular element is still a very useful element for adaptation in complex geometries.

Triangular elements are usually used to mesh 2D model of complex geometry involving deep corners. Most of the automatic meshes generators can just create triangular elements. There are automated meshing generators 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, Liu and Quek (2003).

The tetrahedron, is a tri-dimensional element, but exhibits similar properties of the triangular elements. These are without a doubt the most common form of free mesh generation. Nowadays, the most techniques can be adjusted to the one of the three main categories [2].

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 border of the initial mesh and the new nodes are inserted incrementally and locally the triangles or tetrahedrons are redefined for each new node. Finally, the third is the Advancing Front method, here; the tetrahedrons are built progressively inside of the triangular surface. 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.



(a) (b) (c)

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

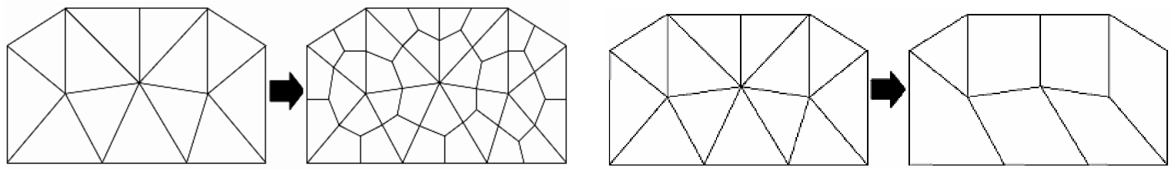
1.4. Quadrilateral/Hexahedral Meshing

Due the smallest income of the triangular meshing and with the progresses of the meshing algorithms, many models of complex geometry with sharp corners or curved extremities can be modelated using quadrilateral elements, Liu and Quek (2003).

Hutton (2004) says that quadrilateral elements are more convenient for regular geometries modeling and they could be used with triangular elements, forming the general base for the development of quadrilateral elements.

When applicable to the geometry of the object the mapped meshing Quad or Hex 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 surface mesh. 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 naming each outlined interval, Owen (1998).

The algorithms for Quad free meshing can usually be described in 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, Owen (1998).



(a) (b)

Figure 2 - (a) Quad mesh generated by the division of each triangle in three Quads and (b) Quad-dominant mesh generated by combining triangles, adapted from Owen (1998)

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 hexahedron (Fig. 3) or for composition joining tetrahedrals in order to form hexahedral elements. In the direct methods, Owen (1998) presents four strategies for the generation of hexahedral meshes. The grid-based method consists in the generation of a tri-dimensional adjustment of elements hexahedrals inside the volume. Hexahedrons are added to the outlines to fill out the openings where the regular grating of hexahedron doesn't have coherence with the surface. The medial 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 plastening consists of increasing elements beginning from the border and moving forward for 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 hexahedrals elements in each one of the directions. The objective of the algorithm is to determine where the intersections of the bending plans will happen. A hexahedron will be formed in a converging position of the three plans of bending.

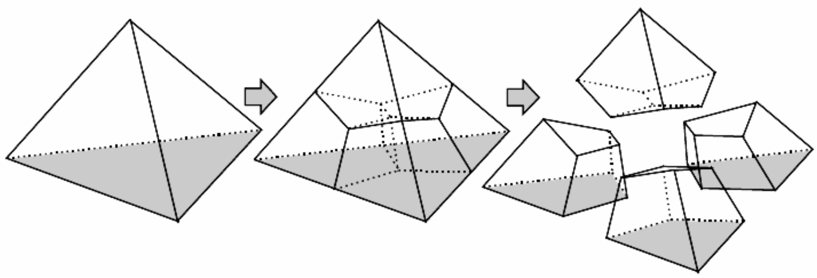


Figure 3 - Decomposition of a tetrahedral in four hexahedral, Adapted from Owen (1998).

For simulation of problems on forming metal sheets the use of hexahedral elements takes advantage of other types of elements. A disadvantage is the difficulty in the mesh generation.

Wisselink (2000) presents some suggestions to create a hexahedron mesh, like: divide the geometry in simple sub domains and generating a mesh with a mapped method or sweeping; to use a generator of tetrahedral mesh well developed for direct generation; or to use the combination of a quadrilateral mesh surface and a simple hexahedral mapped mesh inside the volume. Owen (1998) presented that elements hexahedrals should be advance as far as possible inside of the volume and the remaining empty space should be filled out with tetrahedrons.

1.5. Pos-processing meshing

It is rare some mesh generation algorithm to define a perfect mesh without any pos-processing form 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 quality of the elements. A wide variety of proposals of smoothing techniques exist. Wisselink (2000) tells that smoothing algorithms take into account as criterion for the nodes movementation the form of the element, i.e., angle, size and the position.

According to Owen (1998) cleaning methods usually apply two criteria. As a form improvement criteria for triangular meshes are frequently executed simple diagonals changes. For meshes with tetrahedrons, 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 indeed 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 is begun with a rough mesh and refinement procedures are applied untill the desired node density is reached.

The material of deformation and the element distortion become a limit to calculation. Forming process needed techniques to correct large deformations. Procedures of automatic remeshing for 2D and 3D are improved as alternative techniques to Delaunay/ Frontal methods. It is based on geometrical and topological parameters optimization and proceeds by local change. Local change can be more efficiently remeshing procedures rather than rebuilt entirely whole mesh as seen on Chenot and Massoni (2006). It is based on the combination of local improvement of the neighbourhood of nodes and edges Boussetta, et al (2006).

2. EXPERIMENTAL PROCEDURE

The objective was to evaluate the refinement influence and the way of application of different meshing types on a structural explicit solids/rigids analysis using Solid164 element-type (punch, die and blank holder) and structural explicit thin shell analysis using Shell163 element-type (sheet). The process was evaluated for great deformations of sheets, by stretching, with tools of simple geometry. The used software was the AnSys 9.0 and the Ls-Dyna. The meshing was varied in the punch and in the sheet, Table 1.

Table 1 - Components meshing variations.

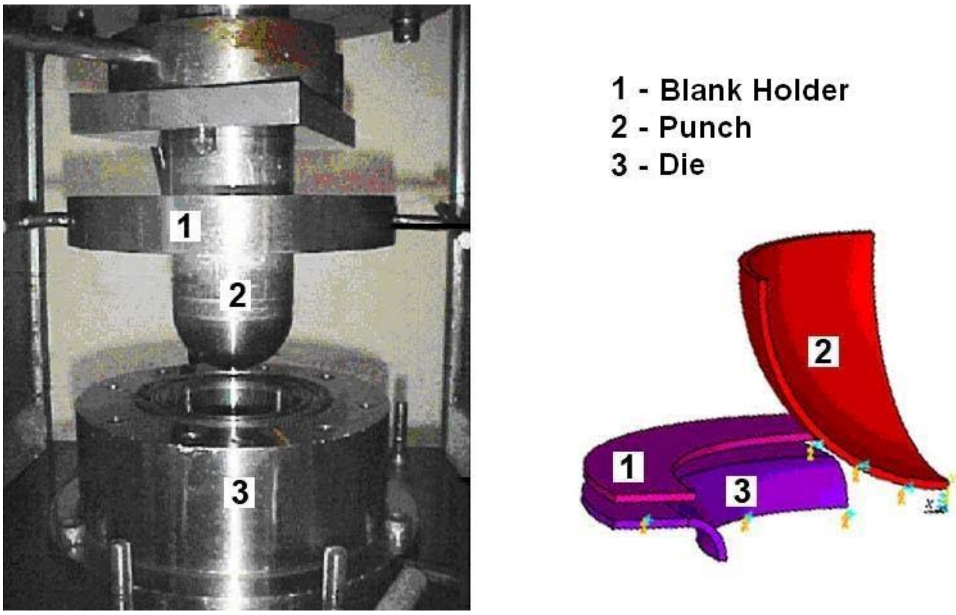
	Punch mesh	***Sheet mesh***
Case 1	Hex	Quad
Case 2	Tet	Tri (mapped)
	Tet	Tri (free)

The case 2 a variation of mapped and free meshing was made for the Tri format. The software also has disponible those options for elements in the Quad format. However, that option was not presented because it not showed differences for Quads, caused by the simplicity of the geometry used in this work.

2.1. Pre-processing

Geometry of the Problem

In a forming simulation experiment usually we have four involved bodies: punch, die, blank holder and sheet, Fig. 4a. Due the symmetry just a fourth part of the geometry were modelated, Fig. 4b.



(a) (b)

Figure 4 - Tool proposed by Nakazima, adapted of Chemin (2004) and (b) geometry model.

Mesh formulation

The Table 2 shows the number of elements used to modelate the punch and the sheet. The punch was meshed with Tet format and the sheet with mapped Tri format, i.e., the opposite of the reference case 1. The number of elements was defined so that the sheet with Tri format meshing 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 [2]. For the punch the adopted criterion was to have the same amount of divisions for face between the elements Tet and Hex. In this case, the punch is tri-dimensional and the amount of elements will larger, but the proportionality is the same.

Table 2 - Amount of used elements.

	Number of elements	
	Punch	***Sheet***
Case 1	192	1600
Case 2	1133	3200 (mapped)
	1133	3632 (free)

Material Specification

The material properties are presented in Table 3. The material was characterized as anisotropic. In the Ls-Dyna software the material model selected was the Barlat and Lian model (2006).

Table 3 - Sheet properties (Chemin 2004).

Property	***Value***	***Unit***	***Source***
Density (ρ)	7,850	g/cm^3	Literature
Elasticity Module (E)	210	GPa	Literature
Poisson (ν)	0,3	(adimensional)	Literature
Plastic Resistance constant (K)	626,8	MPa	Chemin (2004)
m	6	(adimensional)	Barlat e Lian Model (1989)
Anisotropic Coefficient 0^o (R_0)	2,0483	(adimensional)	Chemin (2004)
Anisotropic Coefficient 45^o (R_{45})	1,8659	(adimensional)	Chemin (2004)
Anisotropic Coefficient 90^o (R_0)	2,5988	(adimensional)	Chemin (2004)

2.2. Processing and pos-processing

The numeric error can be defined as the difference between the exact analytical solution of a certain variable of interest and its numeric solution, Ferziger (2001). 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 numeric model; the validation seeks to determine the proximity that the mathematical model is from the real phenomenon, through the comparison of the numeric solution with the experimental data, Oberkampf et. al (2004).

In this work, the values of the true strains (major ε_1, minor ε_2 and thickness ε_3) obtained at each simulation were compared with that ones presented by the simulation of the case 1 (Table 1), proposed by Silva (2005), assumed as a reference pattern (validation). These values were taked as referential for they have presented coherence with the experimental results done by Chemin (2004). In this work it was just considered the points of largest deformation.

It was used a Notebook with Sempron 3100 processor and RAM memory of 512MB.

3. RESULTS AND DISCUSSIONS

Case 1 - sheet Quad and punch Hex

This simulation was reproduced according to Silva (2005). In this case the location of the largest true strain was in the pole of the punch showing that the true strains were idealized concerning the tribological conditions, too. The Table 4 presents all the true strain values and the percentual deviation.

Table 4 - True strain results in function of the meshing type.

Meshing conditions		ε_1	ε_2	ε_3	***Percentual 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 Tet Sheet Tri (Mapped)	0,901	0,176	-0,944	199%	-41,5%	8,1%
	Punch Tet Sheet Tri (Free)	1,569	0,527	-1,385	347%	12,4%	58,6%

Case 2 - Sheet Tri and punch Tet

The sheet mesh types were varied of the Quad for Tri (mapped vs. free) format. The punch was variated from Hex for Tet. The Table 4 shows the ε_1, ε_2 and ε_3 of 0,901, 0,176 and –0,944, respectively. It was observed that solid objects suffer little mesh type format interference. These results were influenced also by the elements applied Tri mapped mesh format in the sheet. The applied Tri mapped mesh format promoted almost 200% and -41,5% deviations for ε_1, ε_2, respectively. In the case of the application of free Tri meshing format on the sheet it was observed that the values lifted up 347% to ε_1 and 58,6% for the thickness sheet reduction (ε_3) Table 4. This value is still more away than the previous situation. In that case it is also observed that the values of the deformations did not present uniform behavior, i.e., the true strain migrate from the pole of the punch to the borders. This could be influenced by 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 1/4 of the geometry in the simulation mainly when it will be meshed by elements with format Tri-type. In this case, the borders of ¼ of the geometry do not exist physically. They are only a symmetrical approach of the real case with time reducing computational purpose.

Time of processing

The Table 5 presents a computational time simulation comparison.

Table 5 - Simulation time.

		Simulation time
Case 1	(ref.)	17 min
Case 2	(Mapped)	8 min
	(Free)	8 min

For 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.

3th case

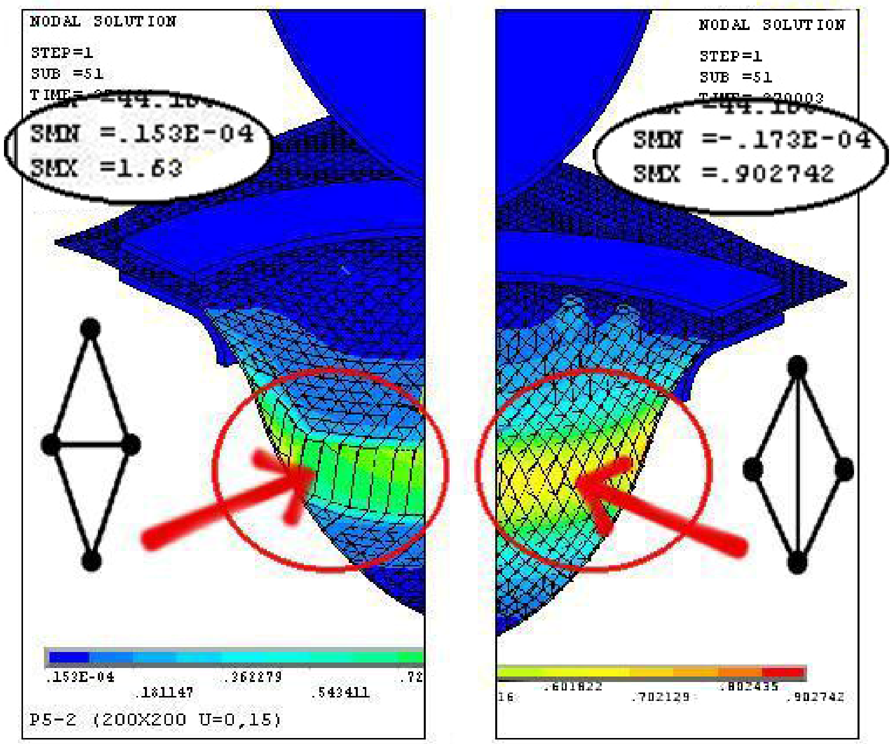
As defined in the experimental procedure the criterion used to define the sheet elements amount was according with the edge division's number. In order to equalize the results (Table 6) this experiment was a model/empiric procedure in order to find a relationship between refinement and computational time that could approximates the results of those two opposed cases. The attempts consisted in progressively to increase the number of divisions and consequently refining the sheet mesh. The approximation occurred, approximately, for the value of 54 divisions by edge, being 35% larger the refinement, providing a number of 5832 elements applied Tri in the sheet against the 1600 elements applied in case 1. The simulation time was close of the case 1 showing that this configuration needs more mesh refinement but the computational time is not increased significantly.

Table 6 - Empiric vs referential results convergence.

	ε_1	ε_2	ε_3
Case 1 (ref.)	0,451	0,424	-0,873
Case 3	0,450	0,400	-0,840
Percentual difference	***0%***	***- 4,7%***	***- 3,7%***

Considerations to the Tri meshing format

For the application of a Tri mesh format it should be observed that the software make disponible options for mesh parameters alteration. One of those choices will define the way that elements Tri will be applied in the object. The Fig. 6 illustrates the effect of that configuration.



(a) (b)

Figure 5 - Effect of the parameters choice for Tri meshing format, (a) software default configuration and (b) correct choice of parameters for improved meshing.

The Figure 5a display the result with a Tri mapped meshing without configuration of additional parameters (software default), i.e., automatic. In the Figure 5b the parameters of mesh direcioning were altered. Is worth to be note that if the mesh was not aligned in a correct way the results could be considerably changed.

4. CONCLUSIONS

This work presented some meshing considerations on finite elements algorithms with the objective of evaluating the criteria used by softwares as Ansys in the mesh application on stretch forming.

Regarding the alteration of generated mesh, it was observed that ANSYS 9.0 offers some possibilities based in the following methods: remesh with new size specifications and element format, to clean the mesh, redefine mesh control and local meshing refinement and mesh improvement (just works out with tetrahedrons format).

Besides the mesh size, Silva (2005), the mesh format also affect significantly the results convergence. Objects built with elements of the type Shell163, in other words, components that will suffer great plastic deformations are extremely sensitive to the mesh format, the refinement and the way it is applied. By other hand, objects built with elements type solid/rigid (Solid 164), punch case, do not suffer significative influence regarding the applied meshing type.

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

The efficiency of just part of the geometry in the simulation can be questioned for objects built by elements Tri under certain refinements. As presented, the areas of larger deformation in some cases were influenced by the distribution of the elements that begins for the borders in those cases.

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

New researches are being made for deep-drawing conditions and more complex geometries, closer cases of the industrial reality.

5. ACKNOWLEDGMENTS

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

6. REFERENCES

I. Zeid, 2005, "Mastering CAD/CAM", Ed. Mac Graw Hill; 1ª ed. New York, NY.

S. J. Owen, 1998, "A Survey of Unmapped Mesh Generation Technology, Proceedings", 7 th International Meshing Roundtable, pp. 239-267.

O. C. Zienkiewics; R. L. Taylor, 2000, "Finite Element Method, Solid Mechanics"; Vol. 2, 5 ed.; Butterworth Heinemann.

G. R. Liu; S. S. Quek, 2003, "The Finite Element Method: A Practical Course", Ed. Butterworth Heinemann; Burlington MA – USA.

D. V. Hutton, 2004, "Fundamentals of Finite Element Analysis", Ed. McGrawHill, pp. 193-196.

H. H. Wisselink, 2000, Analysis of Guillotining and Slitting-Finite Element Simulations; Netherlands.

J.-L. Chenot, E. Massoni, 2006, "Finite element modelling and control of new metal forming processes", *International Journal of Machine Tools & Manufacture*, Vol. 46, pp.1194–1200.

R. Boussetta, T. Coupez, L. Fourment, 2006, "Adaptive remeshing based on a posteriori error estimation for forging simulation", *Comput. Methods Appl. Mech. Engrg.* Vol. 195, pp. 6626–6645.

R. A. F. Chemin, 2004, "Avaliação das deformações de chapas finas e curvas CLC para diferentes geometrias de punções". *Dissertação de mestrado – PG-MEC*, Universidade Federal do Paraná.

H. C. Silva, 2005, "Análise Da Simulacao Numérica Do Ensaio De Nakazima Via Método Dos Elementos Finitos", *Dissertação de Mestrado – PG-MEC*, Universidade Federal do Paraná.

J. H. Ferziger; M. Peric, 2001, "Computational Methods for Fluid Dynamics", 3 rd Edition, Springer, Berlin.

W. L. Oberkampf, T. G. Trucano, C. Hirsch, 2004, "Verification, validation and predictive capability in computational engineering and physics". *Applied Mechanics Reviews*, v. 57, pg. 345-384, ASME, New York.