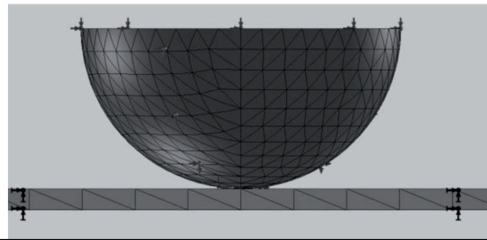
Simulation and validation of incremental forming to ASTM B-265 titanium alloy sheet



SIMULACIÓN Y VALIDACIÓN DE CONFORMADO INCREMENTAL EN CHAPA DE TITANIO ASTM B-265

DOI: http://dx.doi.org/10.6036/7296 | Recibido: 15/09/2014 • Aceptado: 26/12/2014

Luis Miguel Gómez-López[1,2], Valentín Miguel-Eguía[1], Alberto Martínez-Martínez[1], Juana Coello-Sobrino[1], María Carmen Manjabacas-Tendero[1]

[1] UCLM. Universidad Castilla La Mancha. Avda. España, s/n - 02006 Albacete.
[2] IDI. Industrial Development Institute. Albacete Science and Technogical Park. Paseo de la Innovación, 1 - 02006 Albacete.

RESUMEN

- La embutición es uno de los métodos tradicionales de estampado empleados para el conformado de chapa en lotes grandes. Cuando se quiere producir un prototipo o un lote pequeño de piezas, el conformado incremental de chapa debe ser considerado. El proceso de conformado incremental de chapa mediante un punto es una de las opciones más empleadas debido que las instalaciones especiales son poco complejas. Existen trabajos publicados en relación a este tema, especialmente para aleaciones de aluminio y acero. En este trabajo se ha experimentado con la aleación de titanio ASTM B-265 grado 2. Se ha efectuado un trabajo de simulación mediante elementos finitos mediante Solidworks®, software que no es empleado habitualmente para la simulación de este tipo de procesos. Tres experimentos han sido llevados a cabo para establecer la potencialidad del software en la resolución de problemas no lineales, lo que constituye el objetivo principal del trabajo. Se ha construido un prensachapas experimental para medir las fuerzas que aparecen en el contacto herramienta-pieza. Los resultados experimentales y la simulación concuerdan, lo que demuestra que las no linealidades existentes en el proceso son bien simuladas mediante este software. También se establece que el alcance del software está limitado a trayectorias cortas de la herramienta lo que no permite simular piezas muy grandes. El método experimental presentado aquí permite la elaboración de piezas completas.
- **Palabras clave:** Conformado Incremental de Chapa, titanio, simulación, FEM.

ABSTRACT

Deep drawing is one of the traditional stamping processes used for producing large-batch sheet metal parts. When a prototype or a small batch of parts have to be produced, the Incremental Sheet Forming (ISF) technique should be considered. Single Point Incremental Forming (SPIF) process is one of the mostly-used options as they do not require complex facilities. Many studies of SPIF have been shown in the literature, especially for aluminum alloys and low carbon steels. In this work, a SPIF process of ASTM B-265 grade 2 Titanium sheet has been experimented. An introspection to the simulation has been carried out by the finite element method (FEM) in a Solidworks® environment that is not usually used in this arena. Three different simplified experiments have been performed in order to establish the potential of Solidworks FEM SPIF simulation. An instrumented blankholder has been built to measure the forces that act on the sheet and the tool. Experimental and simulated results were compared and a high grade of agreement was found what demonstrates that typical non-linearities involved in the process are well simulated with this kind of software. It has been found that Solidworks® FEM is limited to a short path processes. The experimental method herein presented has been well established by forming complete shapes.

Keywords: Incremental Sheet Forming; titanium, simulation, MEF.

1. INTRODUCTION

The research carried out in the field of Single Point Incremental forming process [1-3], has shown that this process can be done in a numerical control milling machine aided with a design and manufacturing software (CAD/CAM) to develop the shape of the sheet before the forming process and to control the forming tool path, respectively. This process is dieless unlike the traditional deep drawing processes that need expensive dies. SPIF is a process in which there is a single forming tool whose motion is in the horizontal sheet plane labeled as the X-axis and the Y-axis, and the vertical Z-axis being the direction in which deformation takes place [4].

With *Solidworks®* standard module it is possible to obtain the expected design of the sheet after the forming process [5]. On the other hand, *CAMworks®* is a Solidworks® module to program the forming tool path. Finally, the *Solidworks Simulation®* module is used to obtain the stresses, displacements and strains involved in the forming operation by the finite elements method. In this way, the part design, the forming tool path and the process simulation are developed by the same software package and consequently, the process compatibility and integrity is guaranteed. Gómez et al [5] simulated a squared base pyramid of DC-05 steel with Solidworks® package and obtained as a main result that permanent strains were very small because the tool only was able to reach 8 mm in Z axis; that is, the maximum depth that they could simulate was 8 mm. Thus, the involved stresses were very small and hardly overcame the material yield point. In addition, the X, Y, Z steps of the forming tool path were large compared to the real SPIF processes values, around 0.5 mm.

Previously to the indicated recent work [5], many other works about SPIF simulation and experimentation have been developed. An incremental sheet forming process was carried out with two robots on DC-01 steel sheet and this procedure was called *Roboforming* by the authors [6]. The design and the forming tool path were carried out with Solidworks® and CAMworks® respectively, while simulation was realized with *LS-Dyna®* software using an explicit dynamic solver. This kind of solvers is more robust for that processes in which many non-linear situations take place. The problem of Roboforming is the low stiffness of the robots sometimes used in the process and therefore the CNC milling machine may be a better option to drive the forming tool.

Numerical simulation in *LS-Dyna®* software using explicit dynamic solver and experimental validation of micro squared base pyramid was developed on copper steel sheet [7]. The height and the side length of the pyramid were 4 mm and 5 mm respectively. Another similar work [8] used the DYNA3D® software to simulate the process of SPIF of a small squared base pyramid with a height and a base side length of 10 mm for both of them.

Pyramids of larger dimensions have been simulated with the LAGAMINE non-linear FEM code [9]. This one consists of a lagrangian based method developed by the ArGEnCo department of the University of Liège. The code can simulate large displacements and deformations having available a large library of finite elements and constitutive laws.

The general trend in these works is to use the LS-Dyna® software [10] with dynamic explicit solver to do the simulation of small parts because the non-linearities like the behavior material, the contact between tool and sheet, the force variation and large strains and displacements turn the simulation into an extremely slow process with very large computation times. In some cases a specific algorithm or application is developed to do the simulation.

Although SPIF process has been simulated with Solidworks, the results have not been sufficiently compared to experimental results due to the limited dimensions that can be formulated. However, the authors of the present work think that the simulation with Solidworks should permit to evaluate simple paths of the tool or isolated effects. The aim of this work is to do a local simulation consisting of the tool pushing on the sheet as a very short part of the SPIF process and to validate experimentally the simulation results in order to obtain the capacity of this software for simulating SPIF processes. At the final stage of the work a complete experimental SPIF process of two pyramids (squared and rounded) will be showed in order to demonstrate the capability of the process. Titanium alloys have a great interest for forming parts in biomedical applications. The experimental material selected in this work is a ASTM B-265 titanium alloy that it is thought to be a good formability.

2. MATERIALS AND METHODS

2.1. SIMULATION PROCEDURE

Along the Incremental Sheet Forming history a great effort has been done to improve the simulation of this process for a better understanding of the involved mechanics fundamentals and to be able to predict the material behavior by some mechanics analysis software, being the finite elements (FEM) the most employed methodology for those analysis. This method is applied in mechanic analysis software like *Abaqus®*, *LS-Dyna®* or *Solidworks Simulation®*, like in this case. As it is well-known, FEM consists of the division of a mechanical system into many subsystems forming a mesh. Differential equations are formulated and solved in the nodes of the mesh. The equilibrium equation proposed at each node is written in Eq. (1).

$$F = M \cdot \ddot{x} + C \cdot \dot{x} + K \cdot x \qquad (1)$$

In Eq, (1) M is the mass matrix, C is the viscosity matrix, K is the stiffness matrix, x is the displacement vector and F is the force vector. One or two points on the x variable represent the velocity and the acceleration, respectively. There are two methods for solving these differential equations, the implicit one and the explicit one.

On one hand, the implicit solver starts calculating the displacement vector (x) for which it is necessary to invert the stiffness matrix. Computers need large power and time to invert this matrix; that is, a great processor and RAM memory are needed. In addition, implicit method is an iterative process that solves the state in the time t + Δt based on the available information just at that moment and thus, it increases the computation time and provides convergence problems in high nonlinear contact or large strain cases [11]. The form of numeric method most commonly used for implicit solver is the Newton Raphson´s one and it leads to a long solving time in nonlinear or in contact problems and even has convergence difficulties for achieving the solution of the problem. There is a Newton Raphson´s modified method that improves the performance for nonlinear problems.

On the other hand, the explicit solver starts calculating the acceleration vector ($\ddot{x}$) for which it is necessary invert the mass matrix, that is a diagonal matrix. This is more quickly than doing the inversion of the stiffness matrix and thus, computation times and power can be reduced. The explicit solver is not an iterative method and obtains the state t + Δt based on the available information at the previous time t. For those reasons explicit solver works better than implicit one regarding high nonlinear, contact and large deformations problems. The disadvantage of explicit solver is that it is necessary increments of time, Δt, very short to get an accurate and stable solution and sometimes it is not possible to work with sufficiently short time increments of time and solutions are not accurate enough Thus, implicit solvers lead to more accurate solutions in those cases.

In sum up, both implicit and explicit solvers get good results but implicit ones have difficulties to achieve the solutions of high nonlinear contact and large deformations problems. Therefore it is recommended to use implicit schemes with adaptive remeshing [12]. Robust explicit codes as *ANSYS WORKBENCH*® can also be tested..

Solidworks Simulation® uses two implicit solvers to solve nonlinear problems: Direct Sparse and FFEPlus. Therefore achieving the simulation of a complete SPIF pyramid process is very difficult. For this reason, a simulation of simple paths, like a straight line at X, Y or Z axes, has been carried out.

There are different study cases in Solidworks Simulation module like, static, dynamic, nonlinear, frequency response, fatigue… The most suitable for SPIF simulation is dynamic nonlinear study, because it considers all possible nonlinearities like material plasticity, contact between tool and sheet, large deformations and the variation of the tool position along the time.

Previously to the creation of the simulation study, a design of the tool tip and the sheet has been created in Solidworks®. There are several steps to develop the FEM simulation environment in preprocessing. For this work, six steps were carried out and are indicated below.

2.1.1. Material sheet definition

The sheet material was ASTM grade 2 titanium with a thickness of 0.8 mm whose chemical composition is indicated in Table 1. A Von-Misses plasticity behavior was considered for the material. The material was created in the Solidworks Material Library for which it was necessary to introduce the stress-strain curve experimentally obtained by tensile tests according to EN-ISO 6892-1. Three different directions respect to rolling were considered, 0º, 45º and 90º, respectively. The tensile curves were turned into true stress-strain models, σ-e and correlated by a Hollomon´s function according to Eq. (2).

$$\sigma = k\varepsilon^n \qquad (2)$$

An average function was considered according to the plasticity theory [13] for calculating the values of the constants in Hollomon´s equation, k and n, Eq. (3) and (4). The stress-strain curve approximation was considered from the yield point as Figure 1 depicts, defining an ideal rigid-hardening plastic model [14]. Thus, the plastic behavior of Titanium alloy sheet involves a first non-linearity in this work.

$$K = \frac{k_o + 2k_{45} + k_{90}}{4} \qquad (3)$$

$$n = \frac{n_o + 2n_{45} + n_{90}}{4} \qquad (4)$$

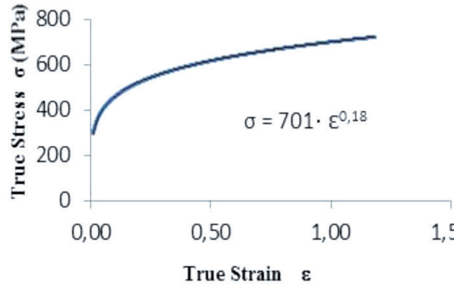


Fig. 1: True Stress (MPa) – Strain curve for ASTM B-265 grade 2 titanium sheet

The material used to forming the SPIF tool was AISI 420 stainless steel with linear elastic isotropic behavior. This material exists in the Solidworks® library and therefore it was selected from it directly. Table 1 contains the chemical composition of the tool material. Table 2 collects the properties of sheet and tool material and that were considered in SPIF simulation.

Parameter	AISI 420 Steel	Grade 2 Titanium
Elasticity Modulus	2.10 e5 (MPa)	1.18 e5 (MPa)
Poisson Coefficient	0.28	0.33
Mass density	7,740 (kg/m^3)	4,510 (kg)m^3)
Ultimate tensile strength	1,500 (MPa)	419.50 (MPa)
Yield	1,200 (MPa)	294.25 (MPa)
True strain at yield point	-	0.0085
Hardening factor	-	0.182
Reference Rule	AISI	ASTM
Material Model	Linear elastic isotropic	Von Misses-plasticity

Table 2: Material properties of AISI 420 and ASTM B-265 titanium alloy

The titanium sheet and the forming tool have been modeled like a deformable body and a rigid body respectively. In the case of the titanium sheet only the non-constrained region has been modeled. Respecting the forming tool only the hemispheric tool tip zone has been modelled in order to simplify the simulation.

2.1.2. SPIF tool – titanium sheet contact definition

The selected contact was the option of node (tool) – surface (sheet) without penetration and friction. The friction was not considered because friction can be neglected if a tool rota-

	%	C	S	Cr	Mn	Ni	P	Si	H	Fe	O	Ti
ASTM 2 Titanium		< 0.08	< 0.02	0.08	0.12	< 0.03	---	---	< 0.015	< 0.30	< 0.25	99.5
AISI 420 Steel		0.37	0.03	13.56	0.75	---	0.04	0.36	---	---	---	---

Table 1: Chemical composition of ASTM B-265 grade 2 titanium commercial sheet and AISI 420 steel

tion speed is applied according to the tool-sheet rolling condition [4] and if the contact is suitably lubricated. Different testing simulations carried out with low friction coefficient showed that the influence of friction on the simulation results was about 1 per cent and it could be neglected in early studies. In addition the computation time is larger if friction is considered. This contact type supposes a second non linearity.

2.1.3. Constraints definition

There are two kinds of constraints in the simulation that correspond to the sheet clamping and to the forming tool displacements, respectively. Sheet clamping constraints of fix geometry were created at sheet sides. In this way, displacements and rotations were blocked in these zones, simulating the blankholder effect. The Forming tool displacements were simulated like movements constraints in X, Y, Z axes and rotation constraint respect the Z axis. Tool rotation increases the computation time and hardly had influence, less than 1%, on the simulation results, once the friction phenomena is neglected as above indicated. For that reason, the tool rotation was not further considered. The displacement of the tool along the time involves a third non-linearity.

2.1.4. External load application

In the simulation no load were directly applied because the contact between the forming tool and the sheet provides the forces that are one of the aims of this study. The knowledge of the forces is very important because on one hand, helps understanding the process better from a mechanical viewpoint. On the other hand it allows evaluate the forces that the milling machine has to support in order to avoid its damage.

2.1.5. Mesh definition

The mesh created was a solid mesh for both parts, tool and sheet. The sheet could have been meshed with 2D shell elements but the computation time of a mixture mesh is at least as high as a solid mesh or even higher. The selected elements, in which the system equilibrium equations were proposed, were tetrahedrons with sides of 3 mm length and 4 nodes. Figure 2 depicts the mechanical system meshed and constrained.

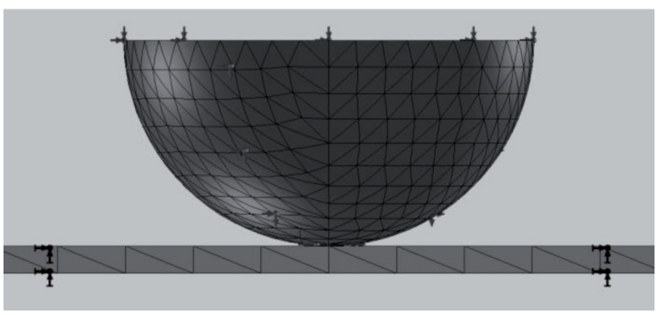


Fig. 2: Mechanical system meshed and constrained

2.1.6. Configuration of the simulation parameters

Several parameters had to be defined, as the solver, the integration method, the simulation time, the step time, the convergence condition and its tolerance, the displacement and the deformation options. As it was previously mentioned, Solidworks® has two implicit solvers, Direct Sparse and FFEPlus. Direct Sparse is a direct method that inverts the stiffness matrix and this is a process for which the computer memory consumption is very high. FFEPlus is an iterative method that looks for the solution proposing deformation values of the system. When the system has many degrees of freedom, i.e. more than 300 000, the FFEPlus works better than Direct Sparse. Otherwise, for multi contact problems the Direct Sparse solver is the best option and thus, this solver was selected. Moreover, large displacements and deformations option was chosen.

2.2. Case Study. Experimental details

Three cases corresponding to different simple paths were considered in the simulation. Forming surface of the samples was 120x120 mm^2 squared sheet of 1mm thick. In case 1 the tool follows a vertical path along Z axis down to 3 mm in the centre of the sheet. Case 2 corresponds to a movement of the tool from the middle of a sheet side to the opposite one through the centre. The depth selected for the path was 1 mm. In Case 3 the tool describes the first squared of a pyramid near to the border of the sheet with 0.5 mm of depth.

Experimental processing of the study cases described before were carried out in a Machining Center Dekel Maho model DMC 835V. The study cases corresponding to simple paths, that is 1 and 2, were carried out by writing down directly the ISO language orders in the CNC module. For the cases in which a completed pyramid was formed, a program had to be created by using a CAM software like CAMworks® at this case.

An instrumented blankholder was built in order to clamp the sheet and to measure the forces acting on the sheet. The device was instrumented with five load cells that were monitored by a data acquisition system, Fig 3. One load cell was located at the base of the blankholder for measuring the, compressive force of the tool on the sheet. Four cells were disposed in the sidewalls of the instrumented blankholder that supporting them in order to probe if some useful information

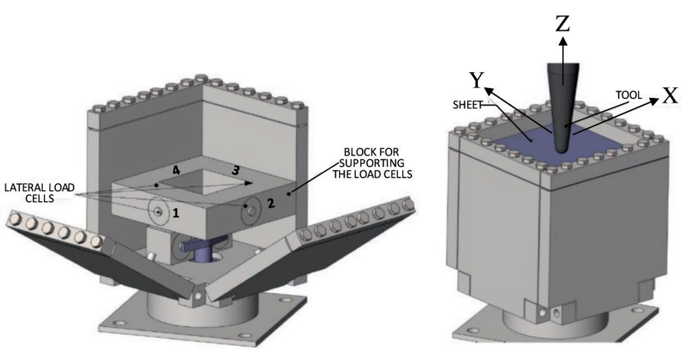


Fig. 3: An instrumented blankholder for monitoring SPIF process

can be obtained through them related to the tensile forces acting on the sides of the sheet.

A hemispherical tool of a diameter of 12 mm was machined in a CNC lathe and was quenching and tempered up to 50 HRc hardness. Finally the tool tip was polished to improve the contact with the sheet.

The sheet was cutted, drilled and placed in the blankholder to develop the different tests. The blankholder was fixed to the table of the machined center.

2.3. Establishment of a methodology for experimental research in SPIF of titanium alloys

In order to validate the possibility of using the experimental methodology proposed herein, squared and round base pyramid was formed and the simplified paths for experimenting cases 1 to 3 were selected according to that. In all cases the SPIF parameters were selected taking into account the existing data in literature for other materials. Figure 4 shows the different parameters that act in the experimental performance of a squared pyramid and in the experimental tests that correspond to the case 3 described before. L_s is the side length of the sheet, L is the side length of the first square, L_c is the constrained length of the sheet, P_{xy} is the incremental step in X and Y axes, P_z is the same as P_{xy} but in Z axis, t is the steel sheet thickness, R_t is the tool radius, F_t is the tool feed rate and S_t is the tool spindle speed. In addition the time parameters must be defined too, t_s is the step time and t_f is the full simulation time. In Table 3 the value of these parameters can be seen.

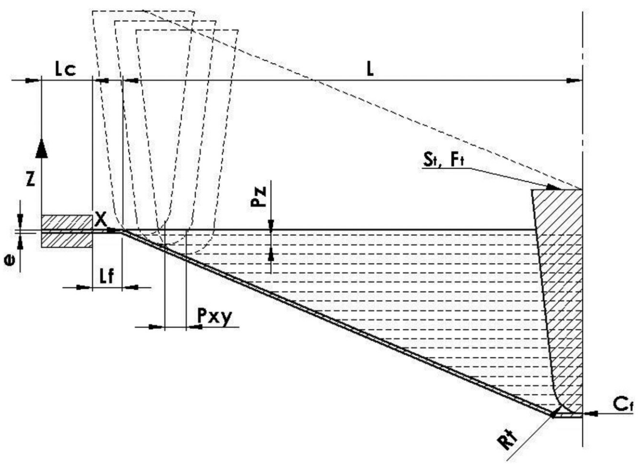


Fig. 4: Squared pyramid experimental parameters

3. RESULTS AND DISCUSSION

Figure 5 depicts the experimental results for the study case 1. As unexpected, it can be observed that the measuring tensile forces in the sheet are not exactly the same for one side and the opposite side and in both directions (X, Y). This might be caused by the unavoidable building non-symmetry of the blankholder and by the friction phenomenon existing in the axes in which the blankholder walls are placed. The normal force in Z axis was also registered. Forces in cells 1 to 4 must be divided to a half in order to obtain the real forces in the sides of the sheet according to the position of the cells in the blankholder. In Figure 6, the total forces in X, Y and Z directions have been represented. Logically, the X and Y forces are not totally zero due to the asymmetry observed. In any case, a good approximation between experimental and simulation results is established.

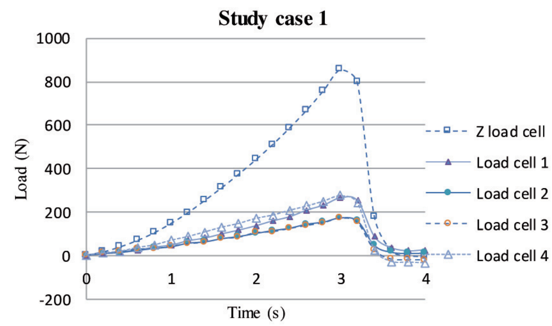


Fig. 5: Experimental measured forces by the blank-holder load cells for the study case 1

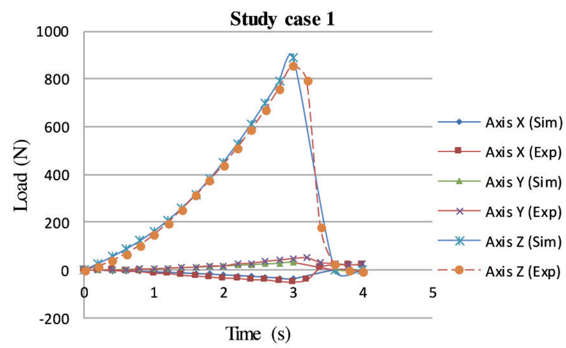


Fig. 6: Resultant forces in axis X, Y and Z. Experimental and FEM results for the study case 1

The simulation and experimental results corresponding to the case 2 are represented in Figure 6. Experimental and simulation results for case 2 present some grade of agreement although some mistmacht between the curves, if the Z axis results are observed. This discrepancy can be considered as an off-set value that can be corrected. The measured values in axes X and Y are very low and no conclusions can be considered. In fact, the force values in X and Y axes are in the same range than the resolution of the cell loads. These forces will not considered for the rest of the cases and the above result suggests that no relevant information is given about the process. Thus, the simulation process can be validated according to the comparison results with Z axis. Case 3 can be considered as a composed form of the case 2 according to the experimental results presented in Figure 8.

According to the results presented for cases 1 to 3, it can be stated that Solidworks® is a suitable environment for simulating processes defined with different non linearities, as SPIF. Experiments have been planned according to realistic SPIF parameters in contrast to other authors whose simulation planning cannot be experimentally carried out [15]. The simulation results present a good agreement with the experimental ones according to other authors, although due to the different selected materials and SPIF parameters avoid a direct comparison [16]. But the reach of the simulation process is limited.

L_S (mm)	L (mm)	L_C (mm)	P_{XY} (mm)	P_Z (mm)	t (mm)	R_t (mm)	$F_{t,XY}$ (mm/s)	$F_{t,Z}$ (mm/s)	S_t (rpm)	t_s (s)	t_f (s)
147	100	14	0.5	0.5	0.8	6	10	1	40	0.2	-

Table 3: Parameters to simulation and experimental tests (unity: mm)

As authors demonstrated in a previous work [5] short paths can be simulated with this software environment. In any case, Solidworks® points out some potential for simulating small pieces for medical implants in which the forming operations were smooth [17].

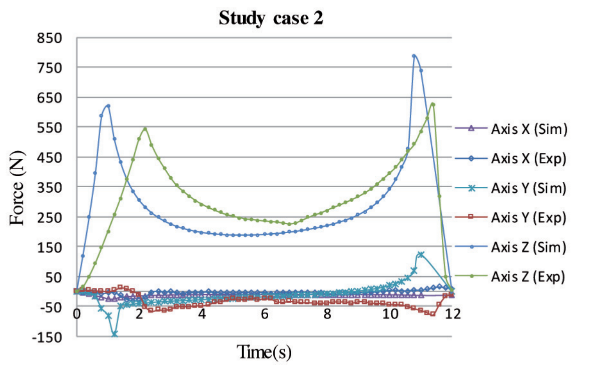


Fig. 7: Experimental and simulation forces for the study case 2

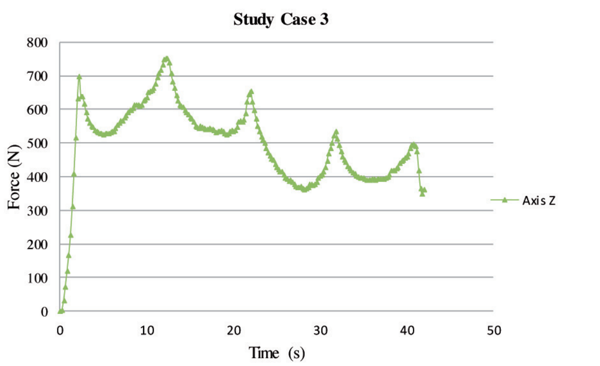


Fig. 8: Experimental force obtained for the study case 3

Otherwise the simulation process, the tool spindle speed must be taken into account in the experimental procedures for

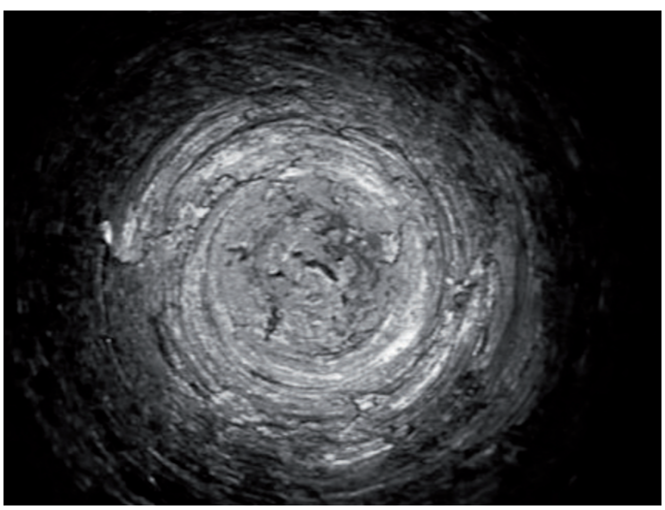


Fig. 9: Consequences of high spindle speed of the tool on the sheet

obtaining square and round base pyramids because, although some authors establish that high spindle speeds improve the conformability [4], the experiments demonstrated that adhesions was produced and that the sheet material was pulled out producing a bad surface finish. Figure 9 depicts the results of a high spindle speed of the tool during experimental SPIF.

In order to avoid this problem the tool spindle speed was calculated according to the rolling condition [4]. The tool spindle speed depends on the tool feed speed and the tool pass deep in the Z axis. On the other hand, a mineral oil was used to reduce the friction phenomenon.

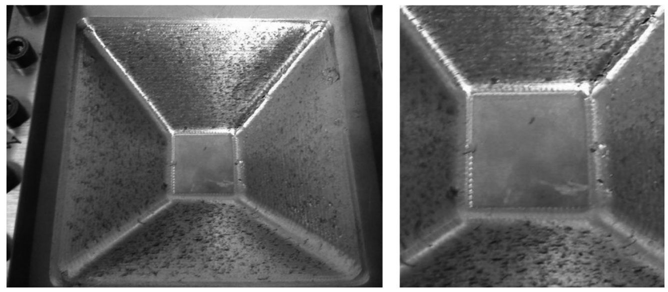


Fig. 10: Pyramid formed with constant Z path strategy

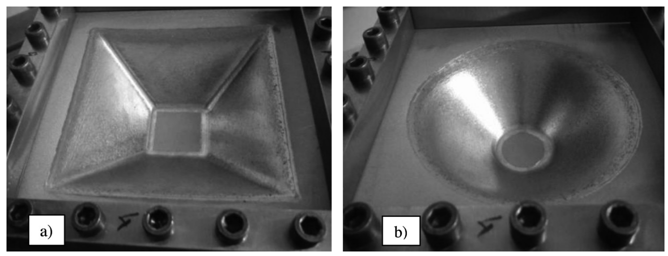


Fig. 11: a) Squared base pyramid formed by helical path strategy; b) Rounded base pyramid formed by helical path strategy

Two different strategies to program the forming tool path were considered. The first one consisted of applying different pass values in the sheet plane for a constant Z value. The second strategy was to establish a helical path for the tool. As it can be seen in Figure 10, the first strategy causes cracks at the line that joins the starting points of each pass. Figures 11a and 11b show the formed pyramids using the helical path strategy.

Exigent forming processes, as indicated pyramids, are out of the Solidworks®´s reach and other software media must be considered. Experimental methodology herein presented permits to carry out experimental research in SPIF processes ap-

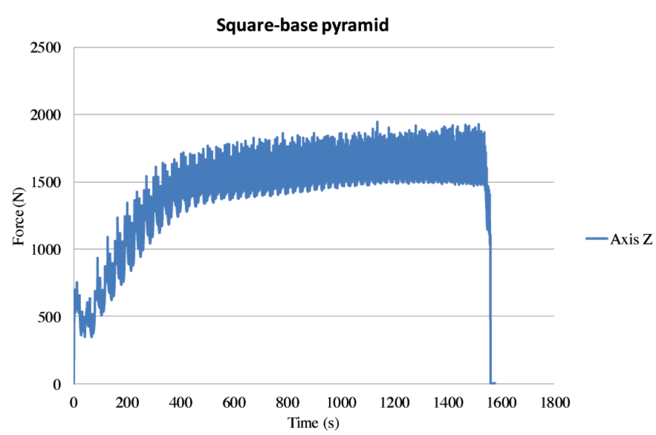


Fig. 12: Force in the Z axis for a complete Square-base pyramid of ASTM B-265 Titanium Alloy

plied to titanium alloys that can be cold formed. Normal force can be measured along the process as it is indicated in Figure 12 what permits to optimize simulation processes in order to research the conformability of titanium alloys that can be cold formed and to optimize the process parameters taking a better understanding of the SPIF process.

4. CONCLUSIONS

A procedure of incremental forming processes simulation has been developed in a Solidworks environment. The parameters of the simulations were realistic as they were previously experimentally proven. For that, elemental cases have been considered.

An experimental device has been developed in order to measure the forces that act on the sheet in SPIF. This device permits to measure the tensile forces on the sides of the sheet and the normal contact force on the tool. A simplified experiment was planned for validating both procedures, the experimental and the simulation ones, and the obtained results demonstrated a great grade of agreement.

Athough Solidworks®´s environment is limited for simulating processes that requires large paths, non-linearities can be well simulated and thus, this FEM software could be used for short formability paths with the advantage of increasing the compatibility of simulation and design processes when this software package is used as in integrated way.

ACKNOWLEDGEMENTS

This work has been conducted thanks to financial support from the National Research Plan, promoted by the Spanish Ministry of Science and Innovation (Project MAT2009-13877) and FEDER funds. Authors are very grateful to The Institute for Industrial Development belonging to the Technological Park of Albacete for providing its installations where some part of this work was carried out.

BIBLIOGRAPHY

[1] Jeswiet J, Hagan E, "Rapid Proto-typing of a Headlight with Sheet Metal", *Proceedings of Shemet*, April 2001, p. 165-170.

[2] Leach D, Green A. J, Bramley A. "A new incremental sheet forming process for small batch and prototype parts". *9th International Conference on Sheet Metal*, Leuven, 2001, p. 211-218.

[3] Filice L, Fantini L, Micari F. "Analysis of Material Formability in Incremental Forming", *Annals of the CIRP*, vol. 51/1/2002, p. 199-202.

[4] Jeswiet J, Micari F, Hirt G et al. "Asymmetric single point incremental forming of sheet metal". *Ann. CIRP* Vol. 54/2, 2005, p. 623–649. (doi: http://dx.doi.org/10.1016/S0007-8506(07)60021-3)

[5] Gómez L M, Miguel V, Martínez A, et al. "Simulation and Modeling of Single Point Incremental Forming Processes within a Solidworks Environment", *Procedia Engineering*, Vol.63, 2012, p.632-641. (doi: http://dx.doi.org/10.1016/j.proeng.2013.08.253)

[6] Meier H, Zhu J, Buff B, et al., "CAx Process Chain for Two Robots Based Incremental Sheet Metal Forming". *Procedia CIRP* Vol.3, 2012 p. 37 – 42. (doi: http://dx.doi.org/10.1016/j.procir.2012.07.008)

[7] Thibaud S, Ben Hmida R, Richard F, et al. "A fully parametric toolbox for the simulation of single point incremental sheet forming process: Numerical feasibility and experimental validation". *Simulation Modelling Practice and Theory* Vol. 29, 2012, p.32–43. (doi: http://dx.doi.org/10.1016/j.simpat.2012.07.004)

[8] Yamashita M, Manabu G, Shin-Ya A. "Numerical simulation of incremental forming of sheet metal", *Journal of Materials Processing Technology*, 2008, Vol. 199 p. 163–172. (doi: http://dx.doi.org/10.1016/j.jmatprotec.2007.07.037)

[9] Guzmán C. F, Gu J, Duflou J, et al. "Study of the geometrical inaccuracy on a SPIF two-slope pyramid by finite element simulations". *International Journal of Solids and Structures.* 2012, Vol. 49 p. 3594–3604. (doi: http://dx.doi.org/10.1016/j.ijsolstr.2012.07.016)

[10] Taleb B, Araghi G.L, Manco M, et al. "Investigation into a new hybrid forming process: Incremental sheet forming combined with stretch forming". *CIRP Annals - Manufacturing Technology.* 2009, Vol. 58 pp. 225–228.

[11] Harewood F J, McHugh P.E, "Comparison of the implicit and explicit finite element methods using crystal plasticity", *Computational Material Science.* 2007. Vol. 39 p. 481- 494. (doi: http://dx.doi.org/10.1016/j.commatsci.2006.08.002)

[12] de Sena, J. I. V, Guzman, C. F., Duchene, L., et al. "Numerical simulation of a conical shape made by single point incremental", *AIP Conference Proceedings* 1567, 852 (2013) (doi: http://dx.doi.org/10.1063/1.4850104).

[13] Hill R, "The Mathematical Theory of Plasticity", Oxford, Clarendon Press (Oxford Classic Texts in the Physical Sciences), (1998).

[14] Rees D, Basic Engineering Plasticity: An Introduction with Engineering and Manufacturing Applications, Ed. Butterworth-Heinemann, 2006. p. 309-313.

[15] León, J,, Salcedo, D., Ciáurriz, C. "An et al. "Analysis of the influence of geometrical parameters on the mechanical properties of incremental sheet forming parts", *Procedia Engineering* 2013 Vol 63 pp. 445-453 (doi: http://dx.doi.org/10.1016/j.proeng.2013.08.206)

[16] Bouffioux, C., Eyckens, P., Henrard, C. et al. "Identification of material parameters to predict Single Point Incremental Forming forces", *International Journal of Material Forming* 2008 Vol.1, 1 suplement pp.1147-1150 (doi: http://dx.doi.org/10.1007/s12289-008-0183-0).

[17] Eksteen P.D, Van der Merwe A.F, "Incremental sheet forming (ISF) in the manufacturing of titanium based plate Implants in the bio-medical sector, *CIE42 Proceedings*, 16-18 July 2012, Cape Town, South Africa.