Simulation of Peen Forming Process of Aluminum Aeronautic Panel

M. Laspalas[1], J. Gómez[1], F. Martín de la Escalera[2], R. Sánchez[3], M.A. Jiménez[1]

[1]Área de Investigación, Desarrollo y Servicios Tecnológicos. Instituto Tecnológico de Aragón
(I.T.A.). C/ María de Luna 8, 50018 Zaragoza -SPAIN

mlaspalas@ita.es

[2]Aernnova Engineering Solutions Ibérica

Avda. Manoteras 20 4-5 Planta San Chinarro 28050 Madrid-SPAIN

[3]NMF EUROPA (AEROMETALLIC)

C/ Andalucía s/n. Polígono Industrial de Tarazona. 50500 Tarazona, Zaragoza-SPAIN

Abstract: The shot peen forming is a cold work deformation process consisting of treating the surface of a work piece with a stream of round shots (usually made of steel) with high enough energy to deform plastically the surface. This plastic deformation causes compressive residual stresses under the surface that are able to create a curvature on the component. This process is increasingly being used to create smooth curvatures in large pieces, for example, in the manufacturing of aerodynamic panels in the aeronautic sector.

In collaboration with AERNOVA group, a methodology to simulate this process by the FE method has been developed. First, the methodology includes material characterization, the transient dynamic simulation by means of Abaqus/Explicit of the impact of an isolated shot against an aluminum plate, including validation of the generated dimple with experimentally measured data. Next, the simulation of multiple impacts is performed, were residual compressive profiles are obtained as a function of the percentage of surface impacted or coverage. The determined profiles are imposed to simulate with Abaqus/Standard the deformation process of initially flat plates in different processing conditions, correlating the results against measured deformation data from validation tests. Finally the model is applied to the simulation of the peen forming process of a complex panel, including the geometrical features typically found in aerodynamic panels, thickness variations, stiffening ribs, etc.

Keywords: Aircraft, Forming, Impact, Plasticity, Residual Stress

1. Introduction

The use of a stream of steel shots impacting the surface of a metallic part (see Figure 1) is a common surface treatment method used to improve its fatigue strength or the corrosion under tension properties. Each shot impacting acts as a small hammer and deforms plastically the material producing a small dimple on the surface. Under the surface layer, the material tries to restore this deformation to its original state, this creates a compression residual stress that makes more difficult the cracks initiation. This treatment process is known as shot peening.

With the same basis but using shots of higher size, the residual stress profile generated is high enough to deform the part, constituting a cold drawn forming process named peen forming.

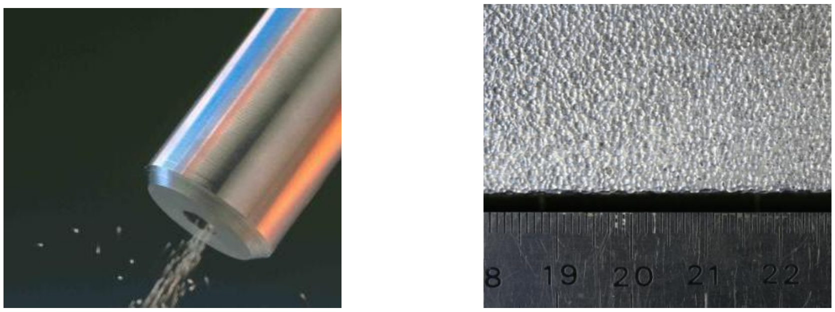

Figure 1. Peen forming process.

This method has a great potential applicability in the aeronautic sector, where the manufacturing of aerodynamic profiles requires creating smooth and usually double curvatures on big parts. The common practice to produce this sort of components is to machine it from a whole material block envelope, wasting many tons of material in form of shaving (till 90%). The use of peen forming process allows starting with a flat pattern, machined from a material block substantially smaller and applying afterwards the desired curvature (see Figure 2).

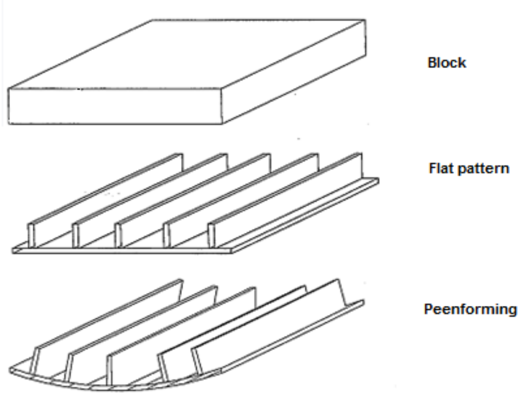

Figure 2. Peen forming process.

In collaboration with the AERNOVA group, ITA is developing a simulation methodology to predict the final shape of parts manufactured by this method, which would allow the selection of the process parameters such as the intensity of the shot stream, the coverage or the pre-stress given to the component.

The proposed methodology starts with the development of a single impact model based on a transient dynamic simulation with Abaqus/Explicit, where a number or sensitivity analysis is performed in order to establish the proper modelization decisions such as mesh size, model dimensions, boundary conditions, element type, etc. Afterwards a multiple impact simulation model is set up in order to obtain residual stress profiles at different coverage values.

The residual stress values are transferred to a model of the component to simulate its deformation process as a spring back simulation in Abaqus/Standard. This procedure is validated in a first step simulating single plates treated in different conditions and comparing simulation and experimental results. Finally the simulation methodology is applied to a complex panel, representative of those found in the aeronautical sector.

A bibliographic research reveals a considerable number of papers dedicated to the simulation of single impacts by means of FEM (Al-Hassani, 1999, Hong, 2008, Meguid, 1999). Fewer references are found dedicated to simulate multiple impacts (Meguid, 2007, Miao, 2009). Finally few references address the final deformation process at component level (Levers, 1998, Gardinier, 1999, Wang, 2006).

This project has been partially supported by the Centre for the Development of Industrial Technology (CDTI) at the 2008 call of the Applied Research Program (Aerospace Subprogram) of the National Plan for Scientific Research, Development and Technologic Innovation 2008-2011.

2. Single impact model

2.1 Finite element model

The FE model for the simulation of a single impact consist of a steel ball, considered as a rigid analytical surface, that impact with a specified velocity against an aluminum plate of 8 mm thickness. The geometric model of the plate extends only the area close to the impact point, which is the area of interest where the residual stresses are developed. This area of interest has been fixed performing a sensitivity analysis, establishing that a distance of 3 times the ball diameter is enough to catch the deformation process.

Since only perpendicular impacts are going to be modeled, an axisymmetric model would be possible, however as multiple impacts simulation is the next step, a 3D model has also been preferred in this phase (although for this simple impact model one fourth symmetry has been used).

The shot has a diameter of 1/8 inch (3.175 mm). The Figure 3 shows the finite element mesh of the target used. In the impact zone the size of the elements in 0.02x0.02x0.02 times the shot diameter. The element type used is a linear hexahedron with reduced integration (C3D8R) available in the element library of the calculation program (Abaqus 6.9.1). The mesh size is increased progressively out of the impact area to reduce the total size of the model.

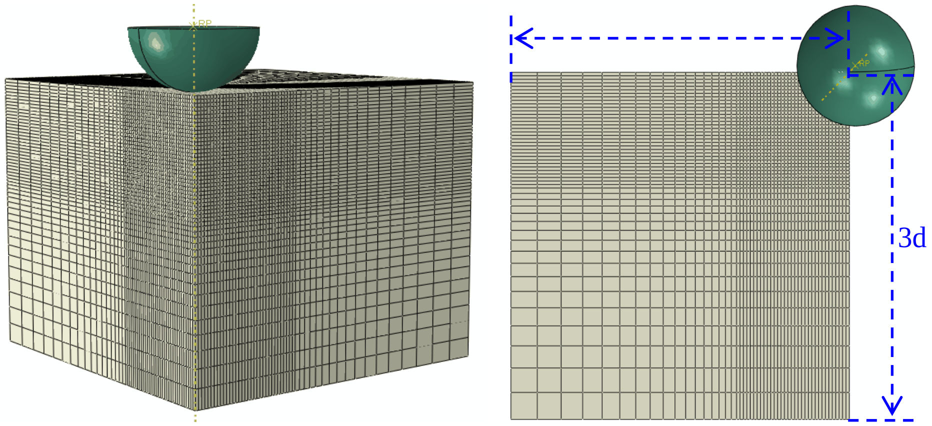

Figure 3. Single impact model.

2.2 Material model

The material model used for the aluminum is a J2 elastoplastic model with isotropic hardening and strain-rate independent. The stress-strain curve necessary to feed this model has been obtained experimentally by means of uniaxial tension tests performed on machined specimens. This curve is represented in Figure 4.

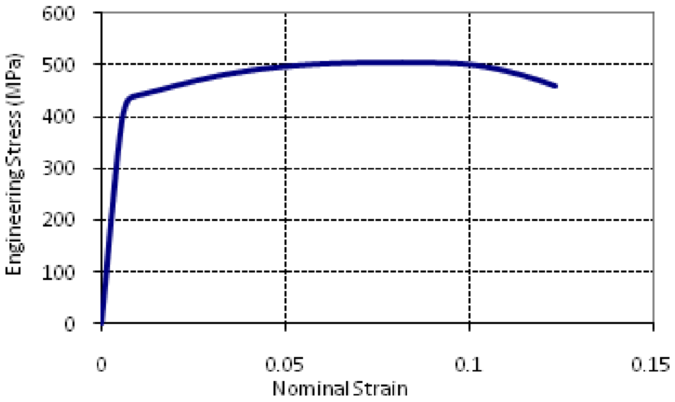

Figure 4. Stress-strain curve.

In order to eliminate the elastic oscillations produced by the impact, material damping is included in the simulation through the Rayleigh model. This model incorporates mass and stiffness proportional damping according to (1). In this case only mass proportional damping is used ($\beta = 0$), taking the α value the corresponding to a critical damping ξ of 0.5 for the lowest natural frequency of the system ω_o , determined by (2).

(1) $$\xi = \frac{\alpha}{2 \cdot \omega_o} + \cdot \frac{\beta \cdot \omega_o}{2}$$

(2) $$\omega_o = \frac{1}{H}\sqrt{\frac{2 \cdot E}{\rho}}$$

where:

E: Young modulus
ρ: Density
H: Height

2.3 Boundary conditions

The boundary conditions used are indicated next:

- Restriction in the displacement component perpendicular to the symmetry planes.
- Restriction in the displacement component perpendicular to the other two lateral limits of the model.
- Vertical displacement restriction in the inferior side.
- The impact velocity of the shot, perpendicular to the plate surface, has been set to different values ranging between 5 and 100 m/s to assess the sensitivity o this parameter.

2.4 Interactions

The interaction has been modeled with a tangential behavior according to the Coulomb law, with a friction coefficient of 0.47.

2.5 Calculation type

The type of the calculation is transient dynamic solved with Abaqus/Explicit. The simulated time is 50 ms, high enough to reach a stable solution after the impact.

2.6 Results

The fundamental result obtained is the residual stress distribution on the plate around the dimple after the impact. Figure 5 shows this distribution for one of the plane components and corresponding to an impact velocity of 35 m/s.

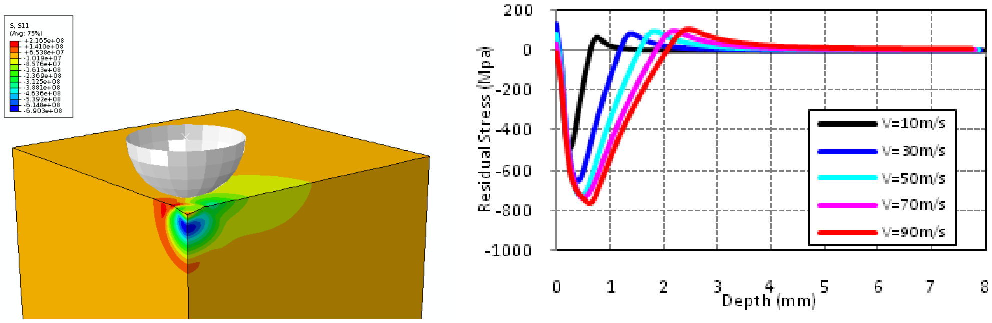

Figure 5. Stress distribution and profile (S11).

The graph in Figure 5 shows the evolution in the thickness of this stress component at the centre of the impact, where it can be appreciated the compression profile in the thickness generated at several velocities. Notice that the higher the velocity the higher and wider the maximum peak in compression.

Figure 6 represents the geometric profile of the dimples as a function of the impact velocity.

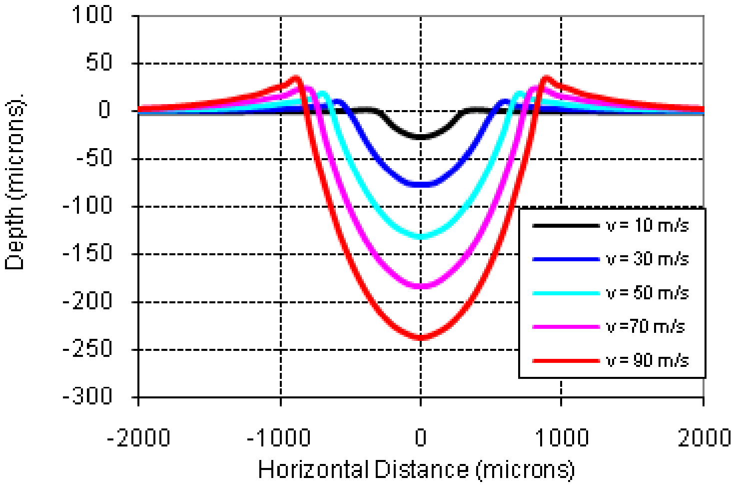

Figure 6. Dimple profile.

2.7 Experimental measurements

Since the resolution of the measurement equipments for the residual stresses determination (hole drilling, X-Rays, Neutron diffraction, …) is not high enough to analyze the distribution in such a small area with big gradients, in this work we have been opted to validate the simulation results at this step through other variables more easily quantifiable, as for example, the dimple diameter, the depth of the dimple and the depth of material plasticized.

Using a peen forming industrial equipment, the pressure conditions have been varied in order to obtain different impact velocities. These velocities have been measured using a high speed camera FASTCAM-X1280 PCI 1k. Figure 7 represents three consecutive frames, separated 1 ms each, where the shot is observed in its trajectory till the plate surface.

Using a profilometer, the profile of each generated dimple is obtained in the perpendicular directions. From these profiles, the depth and diameter of each dimple is obtained.

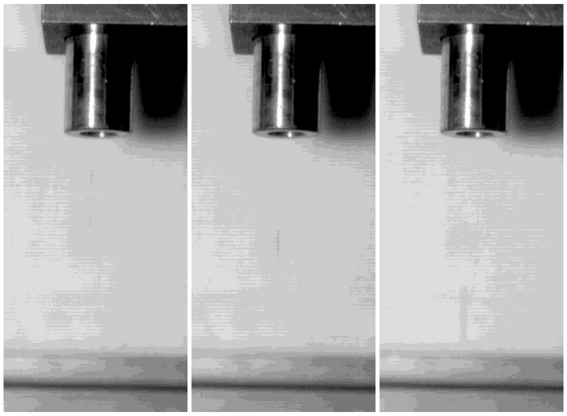

Figure 7. Impact velocity measurement.

Next some dimples have been analyzed using metallographic techniques. Taking profit of the grain texture coming from the lamination process, the depth of material plasticized can be estimated just observing the deformation of the grain boundaries, as it is represented in Figure 8.

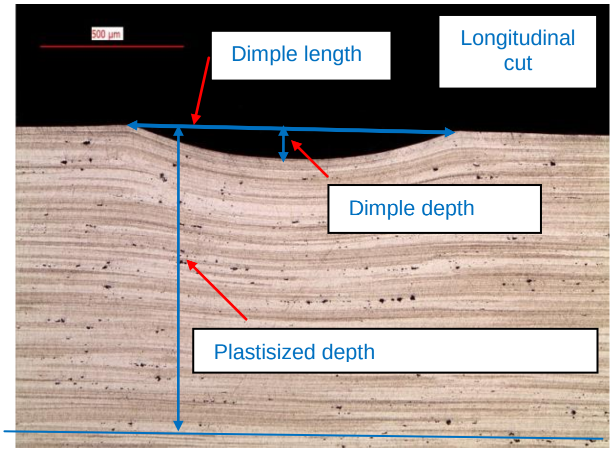

Figure 8. Metallographic analysis.

2.8 Numerical-Experimental correlation

The experimentally obtained dimple diameters are represented against the impact velocity and compared with the simulation results. Figure 9 represents this comparison where the differences found are lower than 5%.

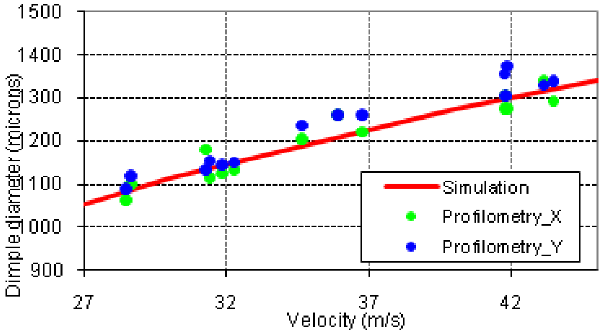

Figure 9. Dimple diameter comparison

The dimple profile is compared for one of the dimples in Figure 10, where the simulation predicts a depth of the dimple slightly higher than the experimental one (difference lower than 10%).

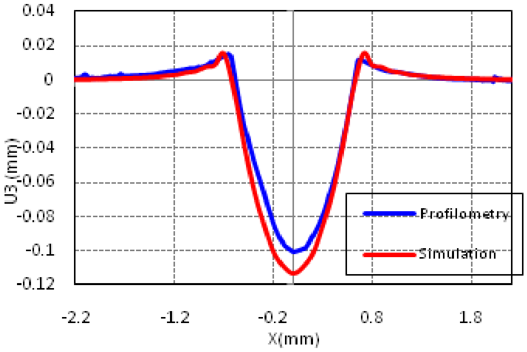

Figure 10. Dimple profile comparison

The same tendency is observed in the depth of material plasticized (see Figure 11), slightly higher in the simulation with regard to the experimental measurements.

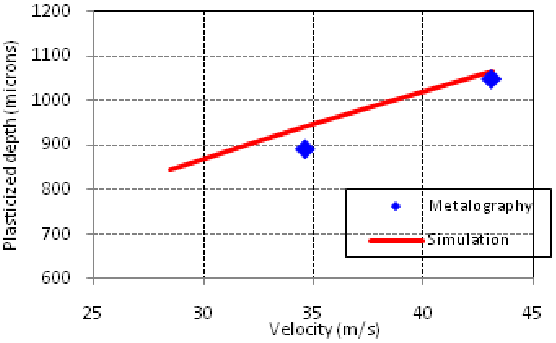

Figure 11. Plasticized depth comparison

This fact can be due to not having considered the strain rate dependence of the material; however, the differences are again lower than 5%.

3. Multiple impact model

3.1 Finite element model

The following step on the development of the methodology is to create a multiple impact model that allows analyzing real coverage levels. Some approaches are found in the literature where several shots are simulated impacting at regular positions, however this approach does not take into account the random nature of the process.

In this work an analytical analysis has been performed by means a code in Matlab© in order to analyze the dependence of the theoretical coverage as a function of the number of impacts and the area of the target. The result of this analysis allowed us to fix the dimensions of the target for the finite element simulations.

In this case we have selected a 26x26 mm^2 area where only a central area of 8x8 mm^2 is impacted. Several thickness values of the target have been simulated in order to obtain the residual stress values as a function of the thickness. Around 300 shots have been introduced in the model as analytical rigid surfaces. The positions of the shots have been produced by a random algorithm in Matlab© checking that there is not overlapping between them. The separation in vertical direction of the shots is constant and it is the minimum required to allow the target to stabilize in the time interval from an impact to the following one. Figure 12 represents this model.

The rest of characteristics of the model (mesh size, element type, boundary conditions, material …) are the same used in the single impact model.

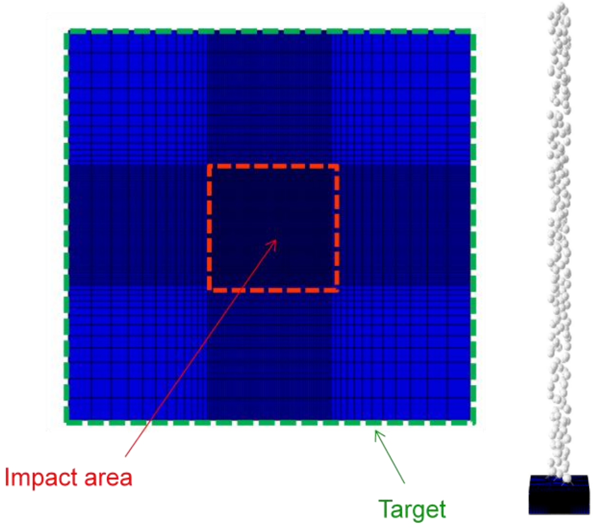

Figure 12. Multiple impact model

The model size details and computation cost are summarized in Table 1.

Table 1. Model size details and computation cost.

Thickness	N. Elements	N. Nodes	N. CPUs used	Computation time	Hardware characteristics
Model 2mm	229312	245276	1	175 h	Machine with 4 CPUs (Intel(R) Xeon(TM) MP CPU 3.66GHz) and 8 GB RAM
Model 5 mm	259864	274124	2	48 h	Machine with 8 CPUs (Intel(R) Xeon(TM) X5355 CPU 2.66GHz) and 32 GB RAM
Model 5 mm	428132	446174	2	87 h	Machine with 8 CPUs (Intel(R) Xeon(TM) X5355 CPU 2.66GHz) and 32 GB RAM

3.2 Results

Results have been recorded in 20 frames in order to have data representative of different coverage levels. Figure 13 represents the vertical displacement at different frames, where the progression of the coverage is observed.

Figure 14 represents the Von Mises equivalent stress in the treated part of the target, where the texture created on the surface can be appreciated. As it can be observed, the stress distribution is not completely uniform in the impacted area, in consequence, in order to obtain average residual stress values, a program has been prepared that averages the stresses of all the elements belonging to the same depth. The result of this process is shown in Figure 15 for two different velocities of the shots.

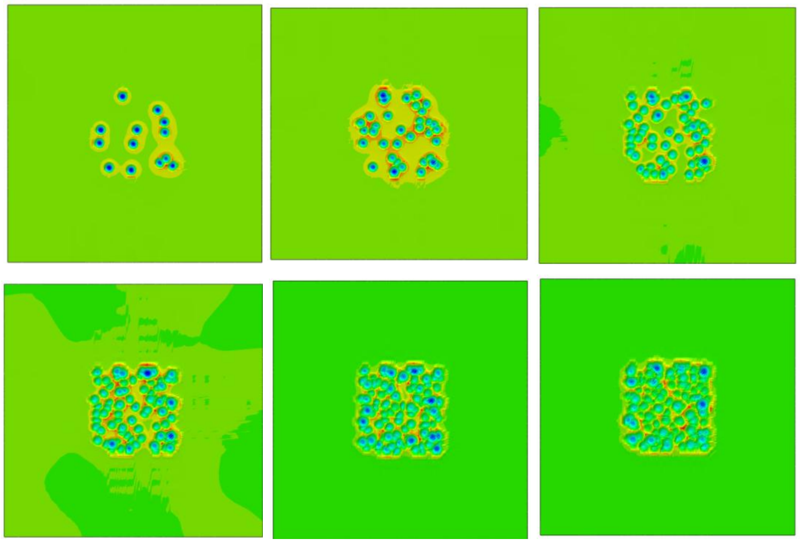

Figure 13. Dimples on the surface

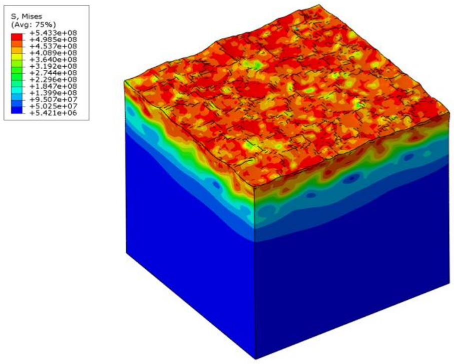

Figure 14. Von Mises equivalent stress

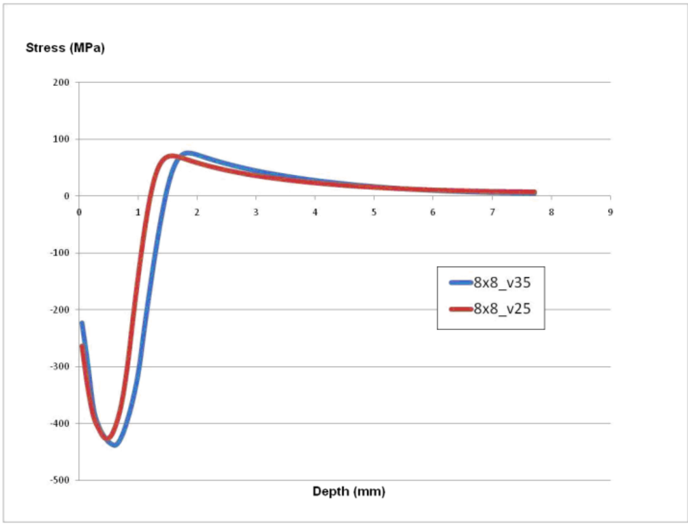

Figure 15. Residual stress values

This method has been repeated in order to obtain a database of different residual stress profiles corresponding to different target thickness, impact velocities and coverage levels.

4. Simulation of peen forming process of flat plates

The residual stress profile corresponding to a set of processing conditions is applied to the component level in order to simulate de whole deformation process. In this case a 300x300 plate has been manufactured with different uniform thickness. These plates have been treated by shot peening with different processing conditions settings and measured in a 3D coordinate measuring machine (Figure 16).

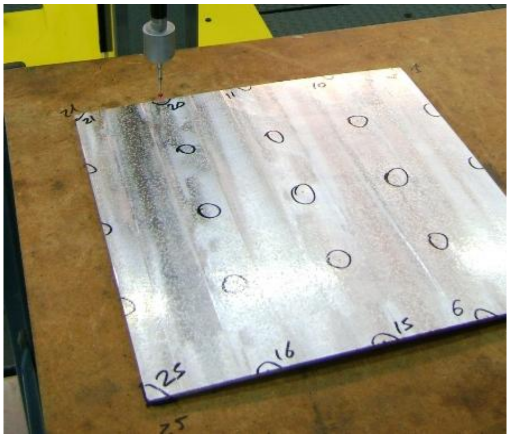

Figure 16. Dimensional control on treated plate

The residual stress profile is introduced in the simulation as an initial condition by an equivalent thermal method. A shell layered model is used in Abaqus/Standard to model the plate. Using a proper thermal expansion coefficient the residual stress profile can be introduced via a temperature profile across the layers of the shell elements.

4.1 Finite element model

The simulation has two steps. In the first one the plate is completely restricted and the temperature is made uniform. In this step the temperature profile imposed as initial condition is transformed in the corresponding stress profile. In a second step, the restrictions of the plate are released, keeping only the symmetry conditions and the vertical displacement at a point to avoid solid rigid motion (Figure 17).

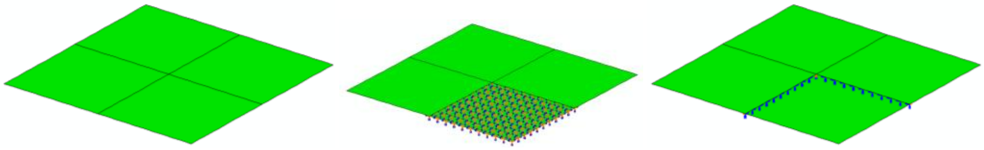

Figure 17. Boundary conditions

In this second step, the residual stress profile finds a new equilibrium condition deforming the plate.

4.2 Results

In Figure 18, the vertical deformation of the plate is represented; the undeformed shape is also superimposed for comparison.

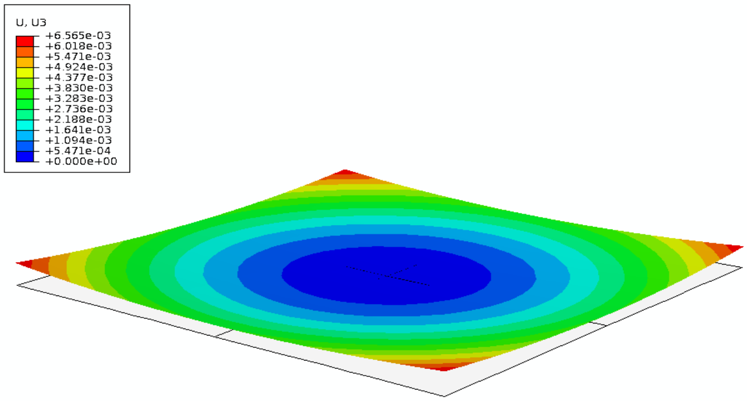

Figure 18. Plate deformation

When comparing with the experimental results, the numerical results show good agreement for different thicknesses and processing conditions. Figure 19 shows a magnified deformation 3D graph for one of the simulations where it is represented in pink the experimental result and in green the numerical one.

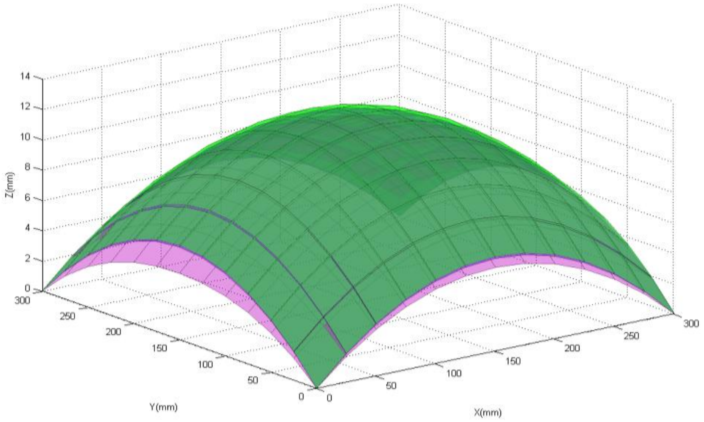

Figure 19. Numerical-Experimental correlation

5. Simulation of the peen forming process of an aerodynamic panel

5.1 Finite element model

This work finalizes with the simulation of a complex panel, representative of those found in the aeronautic industry. This 1600x1100 mm panel has areas with different thickness (see Figure 20) and, in addition, incorporates 5 mm thickness stiffening ribs in longitudinal direction.

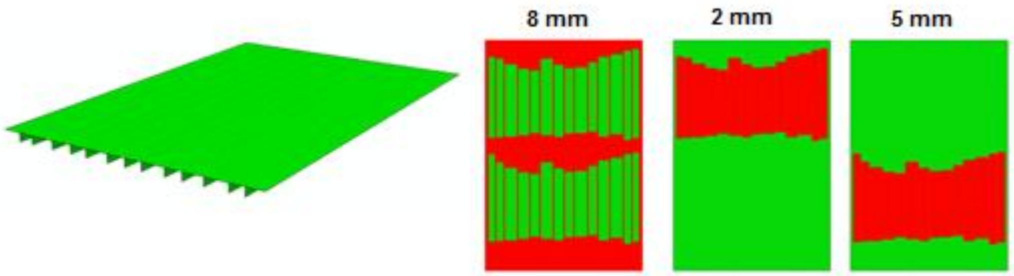

Figure 20. Panel geometry

5.2 Results

Using the same procedure described in the former point, the deformation process of the panel has been simulated. Figure 21 shows the obtained deformation represented as the vertical displacement (the perpendicular to the plane of the panel).

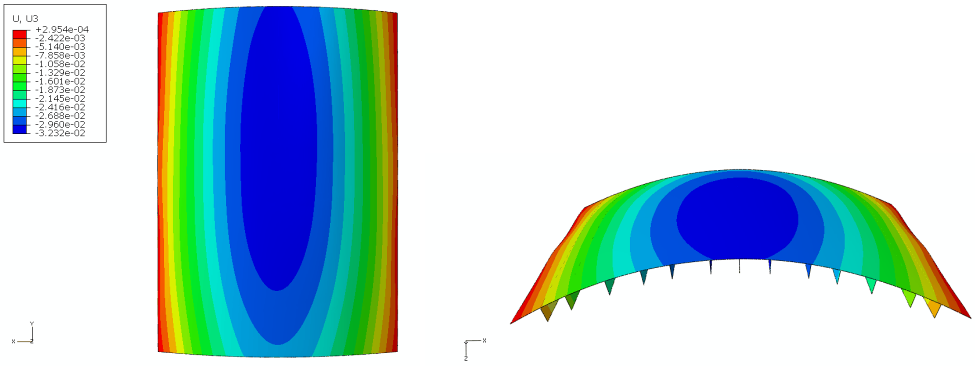

Figure 21. Panel deformation

6. Conclusions

The calculation model reproduces with a satisfactory approach the lower level of the peen forming process, which is the impact of a single shot. Some simplifications on the material behavior, considered in this work as rate independent, would explain the small differences found between the simulations and the tests.

The multiple impact model developed accounts for the random nature of the peen forming process, balancing the computational cost of the simulation with the expected statistical variation.

In order to simulate the peen forming process of real components, the obtained residual stress distribution in the multiple impact simulation must be transferred to the component. This has been done through a thermal profile imposed as an initial condition. This method is easily implemented and effective, allowing the simulation of the deformation process to be inexpensive computationally.

Further developments of the methodology would include the possibility of using the tool for optimizing process parameters in order to obtain a final geometry specification.

7. References

1. Al-Hassani, S.T.S., K. Kormi, K and D.C. Webb, “Numerical simulation of multiple shot impact”, The 7th international conference on shot peening ICSP7, Warsaw, 1999.
2. Gardinier, D.S. and M.J. Platts. “Towards peen forming process optimisation”, The 7th international conference on shot peening ICSP7 Conference, Warsaw, 1999.
3. Hong, T., J.Y. Ooi, and B.A. Shaw, “A numerical study of the residual stress pattern from single shot impacting on a metallic component”, Advances in Engineering software, vol. 39, pp. 743-756, 2008.
4. Levers, A., and A. Prior, “Finite element analysis of shot peening”. Journal of Materials Processing Technology, vol.80-81, pp. 304-308, 1998.
5. Meguid, S.A., G. Shagal, J.C. Stranart, and J.Daly, “Three-dimensional dynamic finite element analysis of shot-peening induced residual stresses”, Finite element in analysis and Design vol. 31, pp. 179-191, 1999.
6. Meguid, S.A., G. Shagal, and J.C. Stranart, “Development and Validation of Novel FE Models for 3D Analysis of Peening of Strain-Rate Sensitive Materials”, Journal of Engineering Materials and Technology, vol. 129, pp. 271-283, 2007.
7. Miao, H.YT., S. Larose, C. Perron, and M. Lévesque, “ On the potential applications of a 3D random finite element model for the simulation of shot peening”, Advances in Engineering Software, vol. 40, pp. 1023-1038, 2009.
8. Wang, T., M.J. Platts, and A. Levers, “A process for shot peen forming”, Journal of Materials Processing Technology, vol. 172, pp. 159-162, 2006.