We are IntechOpen, the world's leading publisher of Open Access books Built by scientists, for scientists

6,900
Open access books available

185,000
International authors and editors

200M
Downloads

Our authors are among the

154
Countries delivered to

TOP 1%
most cited scientists

12.2%
Contributors from top 500 universities

WEB OF SCIENCE™

Selection of our books indexed in the Book Citation Index in Web of Science™ Core Collection (BKCI)

Interested in publishing with us?
Contact book.department@intechopen.com

Numbers displayed above are based on latest data collected.
For more information visit www.intechopen.com

8

Expert System for Simulation of Metal Sheet Stamping: How Automation Can Help Improving Models and Manufacturing Techniques

Alejandro Quesada, Antonio Gauchía,
Carolina Álvarez-Caldas and José-Luis San- Román
Department of Mechanical Engineering, University Carlos III of Madrid, Spain

1. Introduction

Nowadays, competitiveness is one of the major determining factors in global markets, forcing product developers to improve their products quality and to reduce development times. Automotive industry is a clear example of this trend and sheet metal forming, as one of the most important manufacturing processes in car manufacturing industry (Samuel, 2004), is very affected by this situation.

Stamping of automotive components is a critical activity characterized by short lead times and constant technological modifications in order to improve quality and reduce manufacturing costs. The sheet metal forming process, in theory, can be viewed as relatively straightforward operation where a sheet of material is plastically deformed into a desired shape. In practice, however, variations in blank dimensions, material properties and environmental conditions make the predictability and reproducibility of a sheet metal forming process difficult (Narasimhan & Lovell, 1999). Because of this, sheet metal forming results on a process that is heavily experience based and involves trial-and error loops.

The less the experience on the part geometry and material is, the more these loops are repeated. In the innovative process design procedure, however, the trial-and error loops are reduced by means of computer simulations.

Virtual manufacturing of automotive stamped components by means of finite element computer analysis is a powerful tool that is capable of helping engineers to solve different technological tasks (Makinouchi, 1996, Silva, et al., 2004). The forming analyses of sheet metals are performed repeatedly in the design feasibility studies of production tooling and stamping dies (Taylor, et al., 1995). With these analyses, the formability of the sheet material part can be calculated, but it is also possible to estimate the deformed geometry of stamped parts.

However, FEA (Finite Element Analysis) procedure is very time-consuming and relies much on the users' experience. So, under the needs of reduction on design time, reduction on development cost, and reduction on parts weight (so called '3-reduction strategy'), there is an urgent need for more efficient and accurate method in order to improve the current design situation (Wei & Yuying, 2008).

One of the main problems in simulation of sheet metal stamping is to quantify accurately the sheet metal springback, which can be defined as the change in the shape of a sheet metal part upon the removal of stamping tooling (Gau, 1999).

The problem of springback deformations in sheet metal parts makes that most of the produced parts do not conform to the design geometry within the required dimensional tolerances right at the first time (Firat, 2007c), and this dimensional accuracy becomes a crucial factor in determining the overall quality of the part as part components get smaller and tolerances get tighter. (Ling, et al., 2005)

It is also well known that the forming limits vary from material to material. Because of these considerations, knowledge of the behaviour of sheet metal is critical for the success of the sheet forming operation (Chen, et al., 2007).

The latest trend in vehicle structure engineering is to reduce weight of vehicle body- in-white structure in order to reduce fuel consumption, forcing the automotive industry to test new materials not used before. This leads to the following problem: behaviour of new materials is not as well known as behaviour of traditional ones. Constitutive modelling for classical steels can be considered as satisfactory, whereas for new high-strength steels as well as for aluminium alloys available models are still unsatisfactory (Tekkaya, 2000). Furthermore, the use of these materials makes the springback problem more important (Morestin, et al., 1996).

Taking into account previous exposition, it is clear that a good material model is essential when trying to simulate a stamping process by FE (Finite Elements) tools. These material models usually involve a lot of parameters, and it is quite difficult for engineers to consider all of them. The selection of a proper finite element plasticity model and the efficient utilization of the material formability data are main factors controlling the accuracy of the sheet metal deformation response prediction using a computer simulation code (Song, et al., 2007)

In this work, several aspects of metal stamping FEM (Finite Elements Method) simulation are analyzed. For each aspect, the most suitable option to automate the process has been chosen. All these decisions have been included in an interface windows application that allows analyzing the process automatically. By using the application developed in this work, the user does not need to have a great knowledge about the FEM tool.

Once the stamping process is automated, a procedure to create an accurate material model is also proposed.

An initial analysis has been done to determine which material model fits better the real material behaviour. A sensitivity analysis has also been done to find the material parameters that influence more simulation results.

These parameters are optimized through a procedure that combines real test results, FEM simulations and optimization tools. This procedure allows the user to find accurate parameters for not well known materials, obtaining good simulation results for new stamping processes.

Finally, since stamping processes usually involve several steps, one of the problems found in previous studies is that a very refined mesh is needed since the first simulation to achieve good results. In fact, this mesh made of small elements is only necessary in certain areas at the lasts steps of simulation.

It seems to be a good idea to introduce adaptive meshing since the beginning, in order to reduce simulation times (Ortiz & Quigley, 1991, Quigley & Monaghan, 2002). However, this kind of mesh forces to make several changes in original procedure.

These changes are studied in this chapter and a comparison between both possibilities (adaptive and not adaptive meshing) is deeply described.

2. Simulating a stamping process by FEM

2.1 Choosing the software

Not any finite elements software is appropriate for the purposes of this work. Manufacturing processes involve intense plastic behavior of the material with deep cupping operations leading to very large deformations. Furthermore, the application of the dies is intermittent and abrupt, resulting in significant strain rates that require the consideration of the dynamic nature of the problem.

Moreover, deformation processes are carried out in several steps. Because of this, simulation must be divided into steps also and for each of them the geometry obtained after springback must be calculated, as well as the stress distribution of the material. This information is fundamental to feed the following steps.

According to previous exposition, it is necessary to take into account dynamic effects, especially those related to:

- Inertia loads produced in the material.
- Stiffening that metals present when the strain rates are important (the σ-ε curve is modified at high strain rates).

Not every software can tackle with such material models, and so the number of possibilities decreases drastically. This work adopts LS-DYNA (LSTC, 2006), specifically the integrated tool ANSYS + LS- DYNA, that allows to use the powerful LS- DYNA processor and the more friendly environment of ANSYS during pre-processor and post-processor stages. LS-DYNA is one of the softwares that passed all the NUMISHEET^{o}93 benchmark tests (Makinouchi, 1996), so it is proved to be suitable for the purposes of this work.

Even using ANSYS pre-processor, creating a finite element model of a stamping process is not a trivial task. Furthermore, in order to design an application that allows to optimize the main parameters of the materials used in the simulation it is absolutely necessary to automate the creation of the model. This implies that several assumptions must be done. These aspects are discussed in the following sections.

2.2 Explicit and implicit simulations

A general stamping process can be divided into two stages:

- Firstly, the blank is deformed by the contact of the dies.
- Secondly, the dies retire and the springback phenomenon appears.

This springback can be defined as the change in the shape of a sheet metal part upon the removal of stamping tooling (Gau, 1999). This deformation is an essential parameter that significantly complicates the design of forming dies, especially with the increasing use of high strength steels, which are not as well known as typical steels. This forces the construction of multiple prototypes (Narasimhan & Lovell, 1999) to find the dies that produces the right deformation in the black to obtain a final component with the desired shape. Because of this, to perform an accurate sheet metal forming simulation, springback effects must be taken into account.

Mathematically, the resolution of the set of equations generated to solve the finite element problem can be tackled through explicit or implicit methods. Explicit codes are usually adopted over implicit codes in industrial sheet metal applications as seen in Buranathiti and Cao (Buranathiti & Cao, 2005a, b), but implicit codes are sometimes used to simulate springback (Narasimhan & Lovell, 1999).

Explicit codes produce simulation results as accurate as the implicit FEM solvers (Belytschko, et al., 2000, Firat, 2007a) and use less computer resources, since the

computational time grows linearly with the problem size instead of the quadratic growth in the implicit codes. On the other hand, using only explicit codes forces to simulate both application and withdrawal of dies, so several iterations must be solved, resulting in much greater computational costs.

According to this, the first proposal of this work is to use explicit codes for application of dies and implicit codes for springback simulation. However, it will be seen in following sections that implict codes have several limitations that can be avoided by using explicit simulations.

2.3 Material model

One of the main points in the simulation of a stamping process by means of finite elements is the choice of the material model of the blank. For a given process and deformation geometry, the forming limits vary from material to material, so knowledge of the formability of sheet metal is critical (Chen, Gao, Zuo & Wang, 2007). The selection of a proper finite element plasticity model and the efficient utilization of the material formability data are main factors controlling the accuracy of the sheet metal deformation response prediction using a computer simulation code (Firat, 2007b).

Nowadays, the isotropic hardening plasticity models are widely accepted in the industry for sheet metal simulation, and it is assumed to be accurate enough for classical steels (Firat, 2007b). But the increasing introduction of high strength metals is showing that this model must be reevaluated. Because of this, several possible models have been taken into account in this work.

When trying to select a material model for the blank (between the more than 100 models implemented in LS-DYNA), several aspects must be considered:

- The model has to be applicable to metals.
- It has to work with shell elements (that are generally used the standard for meshing the blank (Tekkaya, 2000)).
- It must include strain- rate sensitivity.
- It has to deal with plasticity.
- It has to be able to study failure.

According to these statements, three material models have been selected for this study:

1. Kinematic / Isotropic elastic plastic.
2. Strain rate dependent isotropic plasticity.
3. Piecewise linear isotropic plasticity.

2.3.1 Selected material model

A real stamping process has been selected to compare simulation results obtained by using each of previous material models. This process (see Figure 1) is the first of the five stages needed to manufacture a part that belongs to the fix system of the spare tire of a commercial vehicle. Deformed blank obtained by this process is shown in Figure 2.

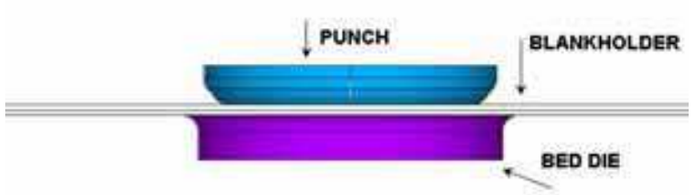


Fig. 1. Starting situation of the dies

Fig. 2. Deformed blank

The comparison between simulation results and the real deformed blank has been carried out by means of a coordinate measuring machine. The dimension used to be compared with simulation results is the stamping depth shown in Figure 3, and its real value is 15,88 mm.

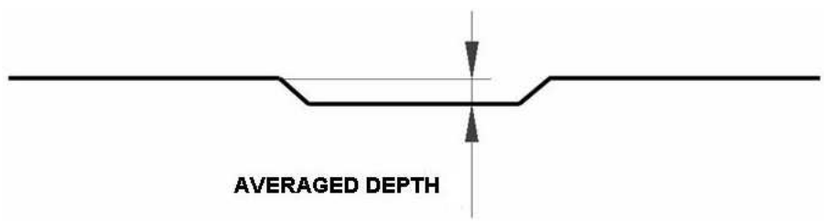


Fig. 3. Stamping depth used to compare experimental and simulation results

Table 1 shows a comparison between results obtained by using the three material models. For each model, several values of the main parameters have been tested. The maximum and minimum value obtained as well as the averaged depth are displayed.

Model	Minimum depth	Averaged depth	Maximum depth
Kinematic / Isotropic elastic plastic model	15,82 mm	15,87 mm	15,92 mm
Strain rate dependent isotropic plasticity model	15,68 mm	15,76 mm	15,85 mm
Piecewise linear isotropic plasticity model	15,85 mm	15,90 mm	15,98 mm

Table 1. Comparison between material models

According to these results, and taking into account the real obtained depth (15,88 mm) it can be concluded that any material model that has been considered in this study is accurate enough to simulate the stamping process and the behavior of the involved material.

However, the kinematic/isotropic elastic plastic model is the simplest one and the most appropriate when the material behavior is not well known. Because of this, this model has been adopted in the present work and is explained in the following section.

2.3.2 Kinematic / Isotropic elastic plastic model

This material model is described by the expresion Eq.(1) (Hallquist, 1998), based on the Cowper- Symonds model (Cowper & Symonds, 1958, Dietenberger, et al., 2005, Jones, 1983), which scales the yield stress by a strain rate dependent factor:

$$\sigma_y = \left[1 + \left(\frac{\dot{\varepsilon}}{C} \right)^{\frac{1}{p}} \right] \left(\sigma_0 + \beta E_p \varepsilon_{eff}^p \right) \tag{1}$$

Where:
σ_0 : Initial yield stress.
σ_y : Yield stress.
$\dot{\varepsilon}$: Strain rate.
β : Varying this parameter, isotropic ($\beta = 1$) or kinematic ($\beta = 0$) hardening can be obtained. In this work, isotropic hardening is supposed, so $\beta = 1$.
E_p : Plastic hardening modulus, defined by Eq.(2), where E_t is the tangent modulus and E is the elastic modulus:

$$E_p = \frac{E_t E}{E - E_t} \tag{2}$$

ε_{eff}^p : Effective plastic strain.
C and p: Strain rate parameters.
The following parameters have to be specified by the user in order to define properly this material when using LS-DYNA. Those parameters are:

- Density.
- Young's module.
- Poisson ratio.
- Initial yield stress.
- Tangent modulus.
- Hardening and strain rate parameters β, C and p.

2.4 Geometry of the dies and the blank

Finally, it is necessary to decide how to generate the geometry of the dies and the blank.
The forming tools are usually intended to impose the forming loads to the sheet metal through the forming interface. In order to reduce computation time, only the surface of the tooling has been included in the FEM model, rather than the complete geometry.
This is a common decision in sheet metal forming analysis (Firat, 2007a, c, Narasimhan & Lovell, 1999), because of the fact that the forming tools should be, theoretically, designed to be rigid and their deformation (that should be elastic with minimal shape changes) is neglected.
The fact of defining dies as rigid bodies allows applying displacement restrictions in the material definition.
The thickness defined for all the dies is 0.001mm, in order to distort the real geometry of the contact faces as less as possible.
Regarding the sheet metal blank, because of its thin geometry, it is usually meshed with shell elements (Darendeliler & Kaftanoglu, 1991, Firat, 2007a, c, Mattiasson, et al., 1995, Narasimhan & Lovell, 1999, Taylor, Cao, Karafillis & Boyce, 1995, Tekkaya, 2000).

In this work, the reduced integration Belitschko-Tsay shell element (Belytschko, Liu & Moran, 2000, Hallquist, 1998) has been used (included in the SHELL163 element implemented in LS-DYNA). Five integration points have been defined through the thickness in order to properly represent plasticity effects (Narasimhan & Lovell, 1999).
The Belitschko-Tsay shell element has proved to produce results that are similar to those obtained with more complex elements and this element is the least expensive element formulation of its kind (Firat, 2007a).
Contacts between the blank and the dies have been defined using an automatic surface-to-surface contact algorithm and a static friction coefficient and a dynamic one are considered during the simulation. With these two coefficients, the finite element simulation carries out a thorough analysis of friction.

3. Developed application

3.1 Automation procedure

Every decision discussed above is aimed at achieving an application that automatically generates the finite element model of a stamping process minimizing the user intervention.
The main steps of a FEM analysis can be resumed as follows (Álvarez- Caldas, 2009):

1. Definition of analysis parameters (materials, loads…).
2. Geometry creation.
3. Analysis.
4. Results post processing.

A different solution has been adopted to automate each one of them.

1. Definition of analysis parameters: This is the hardest step for the user, and the one that needs more automation. The designed application offers the user a window friendly environment where all the parameters needed to define the simulation can be introduced: blank thickness, material properties of the blank and the dies, loads, restrictions, displacements, contact coefficients, simulation time… This windows environment is programmed with Matlab Guide and generates a text file that can be imported to LS-DYNA.
2. Geometry creation: The user can generate the geometry entities for the blank and the dies in any CAD program, exporting them to any graphic format that can be read by LS-DYNA (as IGES).
3. Analysis: All the parameters that have been introduced through the windows environment, as well as the CAD geometries, have to be linked by the appropriate ANSYS commands. The actions that must be done can be resumed as follows:
   - Import CAD geometry of the stamping tooling and the blank.
   - Creation and assignment of material models and real constants sets.
   - Definition of frictional contact conditions.
   - Description of forming process via the prescribed displacements or forces on the tooling surfaces.
   - Meshing of the blank and the dies.
   - Resolution of the finite element model.
   - Since there are two kinds of potential users for this application (the ones that are used to employ finite elements applications, and the ones that are not), two options have been implemented:

- Blind analysis: All previous actions have been implemented in a generic subroutine that is launched by the windows environment, so that all the previous described process does not need user intervention.
- Expert analysis: The automatic process ends before the solution step, allowing the user to make any changes.

4. Results post processing: this step cannot be automated because the user must be the one to carry out the critical reviews of the results.

The proposed procedure is depicted in Figure 4, where stages that require user intervention are drawn with solid line and those that can run "blindly" are drawn with broken line.

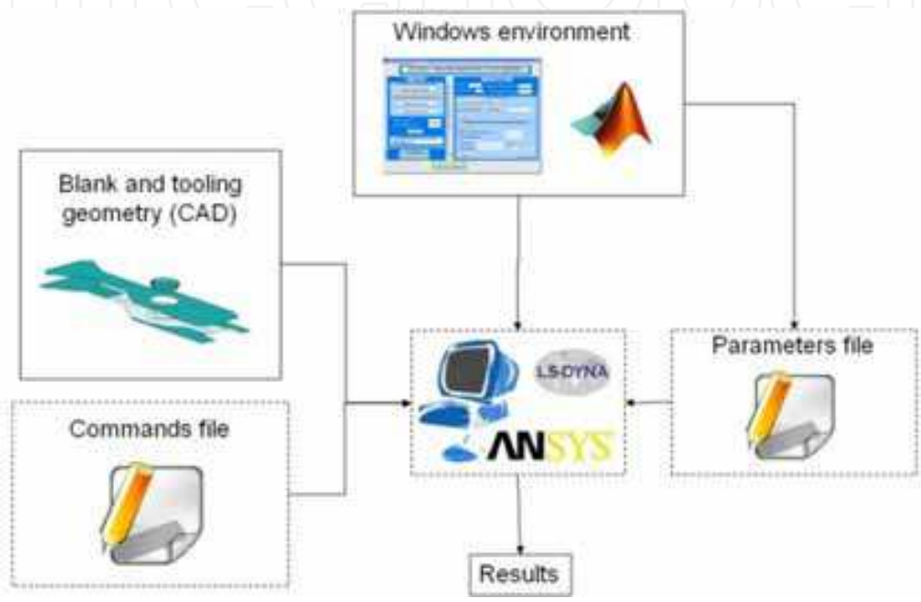


Fig. 4. Automation procedure

Once the proposed procedure is clear and taking into account that the automation may not be done by someone non specialist in ANSYS LS-DYNA it is desirable to operate within a friendly windows environment. In addition, the toolboxes available in some software such as MATLAB are of great help. Therefore, a friendly windows environment has been programmed in MATLAB by means of the GUI (Graphical User Interface) which is deeply described in the following section.

3.2 Windows environment

By means of MATLAB's GUI a friendly window environment has been designed in order to provide the user a step by step procedure that ensures the correct operations that must be done in the finite element model which simulates the stamping process. The proposed environment generates a set of files which is afterwards forwarded automatically by the software to ANSYS LS-DYNA so that it runs in batch mode, that is, under system without having to involve the user in the modelling of the stamping process. In addition, the proposed environment carries out an estimation loop so as to predict the values of the material parameters that best fit the model with experimental test results. Therefore, the software which has been developed allows the user either to simulate a stamping process or to find the material parameters that best suit the stamping process. In Figure 5 the window that allows simulating a stamping process is depicted.

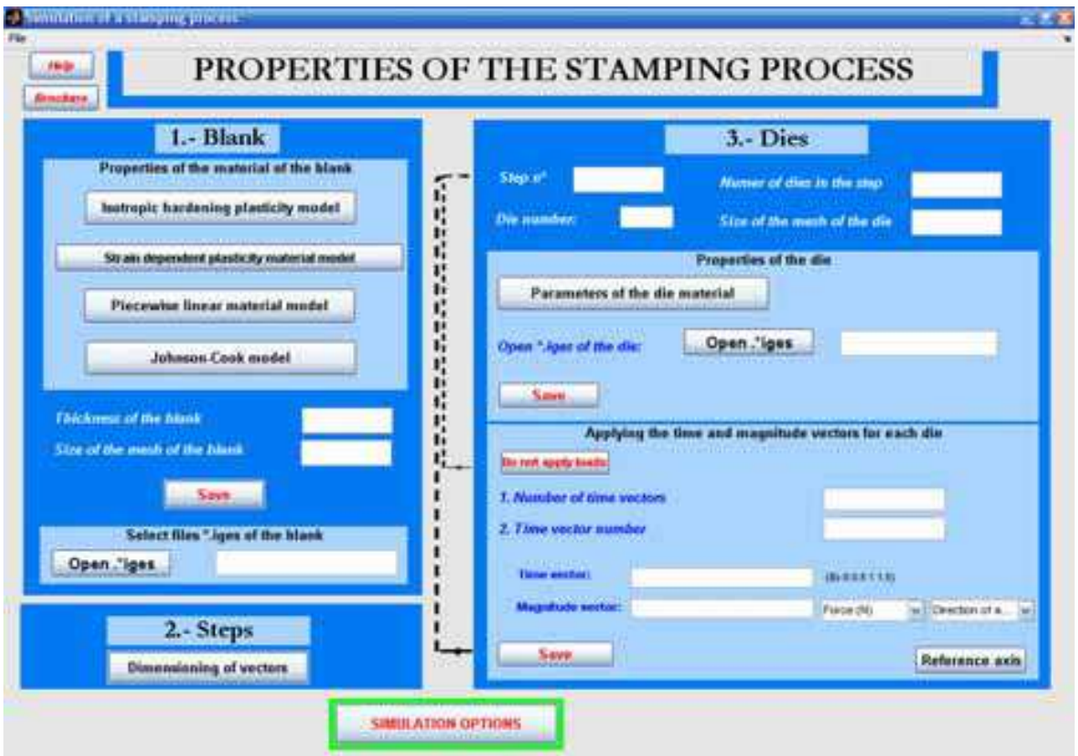


Fig. 5. Window environment of the developed software

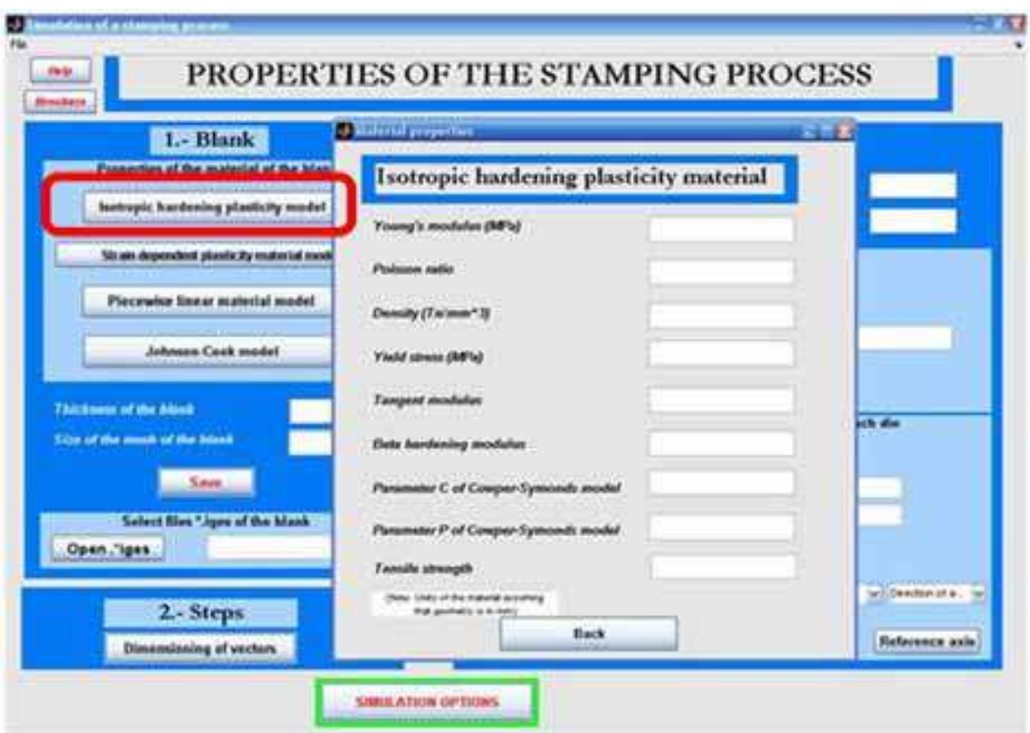


Fig. 6. Specifying the material properties of the blank

This part of the software is divided in three steps. In the first step the user must select the plasticity material model that best describes the material used as a blank. Figure 6 shows the parameters to be introduced by the user if an isotropic hardening plasticity model is selected to model the blank.

In addition, the user has to introduce the thickness of the blank, the meshing size and has to load the "*.iges" file that contains the blank geometry. Afterwards, the user must specify in the second step the number of steps in which the stamping process will be done, as well as other parameters such as the maximum number of dies which will be used during the process, etc. Finally, in the third step the properties of the dies employed during the stamping, including the die material properties (see Figure 7) and load vectors are applied. During the clicking of each of the buttons certain files are being generated automatically which will finally be the input to ANSYS LS-DYNA. In addition, the software allows distinguishing between users that have previous experience in ANSYS LS-DYNA by clicking in the simulation options button. Once clicked, the user can specify the simulation time or either open LS–DYNA in order to load the simulation and allow changes in the model before running the solution.

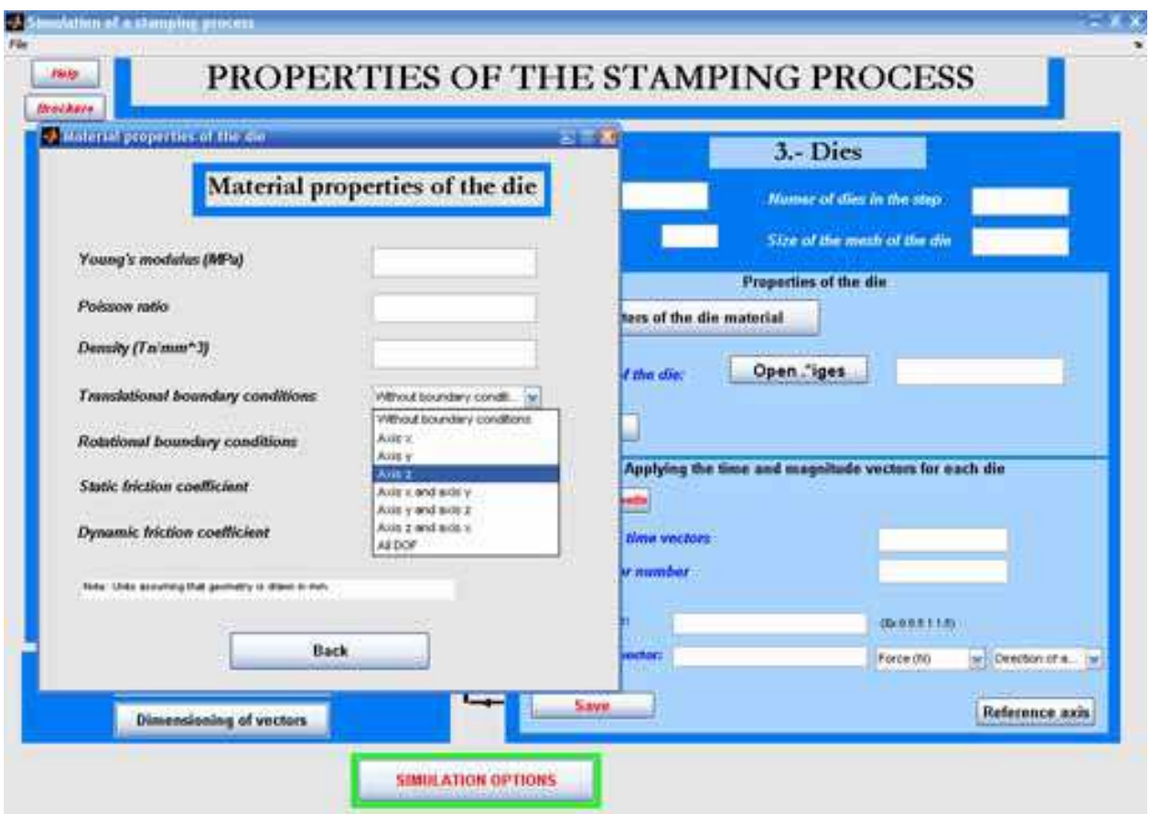


Fig. 7. Specifying the material properties of the die

One of the problems that may be encountered is that the values of material parameters are not known and therefore have to be adjusted before simulating the complete stamping process. To solve this problem the following steps are proposed:

- In the first place the user must select a certain manufacturing process to be simulated.
- Afterwards, this process will be carried out in an industry using the available dies and devices. This test will be defined as a pattern test.
- Thirdly, the pattern test will be done in the material whose parameters want to be computed. Due to the fact that the selected process is well known and defined, all the changes that take place in the final shape will be due to changes in material properties.
- Finally, once the material parameters have been clearly found other processes may be simulated once the optimum material parameters are known. This information may be used for designing new dies for new upcoming processes saving money and time as the number of experimentally tested dies has decreased a lot.

3.3 Estimation of material parameters

In order to adjust the material parameters the designed software provides a specific tool that compares the results of the finite element simulation with the results of a real experimental test (Gauchía, 2009). The user must specify at least two sets of simulations where the values of the material parameters are different. The software will create the files needed to carry out the finite element model and return a solution which will be compared with the experimental test results given by the user. From two simulations, the software provides by means of a linear interpolation an estimation of the material parameters. Because the provided values are the result of a linear interpolation the proposed material parameters may not be the most appropriate. Therefore, the user can modify the proposed values and carry out a third simulation. Once the results of this third simulation are provided the software shows different graphs that show the results obtained in the previous simulations for each of the material parameters. If for example, the depth is considered as the result to be compared with the experimental tests the prediction plots display graphs where each of the material properties is represented in the vertical axis and the depth in the horizontal axis. In addition, the user may modify the polynomial degree (linear, quadratic, etc.) for the simulated results. These graphs, represented inFigure 8, display the polynomial function and confidence bounds. Each of the results are plotted in the polynomial fit estimation and represented as a cross ("+").

The proposed software allows carrying out more than three simulations. If the user does more simulations the confidence bounds will narrow, however, the user will have to find the proper balance between computation time and exactness. It must also by noted that only some of the most sensitive material parameters can be changed by the user, as depicted in Figure 9. The material parameters the user is allowed to change are the yield stress, parameter C and parameter p of the plasticity model. The yield stress is without doubt one of the most important parameters that characterize the plasticity material model. Previous simulations (Quesada, et al., 2009) have shown that variations of approximately 14% in the depth may be encountered. However, it was found that parameters C and p do not have a great influence in the results. Previous simulations revealed that varying parameter C a 900%, produces a variation of less than 0.5% in the result and varying parameter p a 133% produces a variation of 0.4% in the final result. Therefore, the influence of other parameters can be neglected and will not be considered during the material parameter estimation.

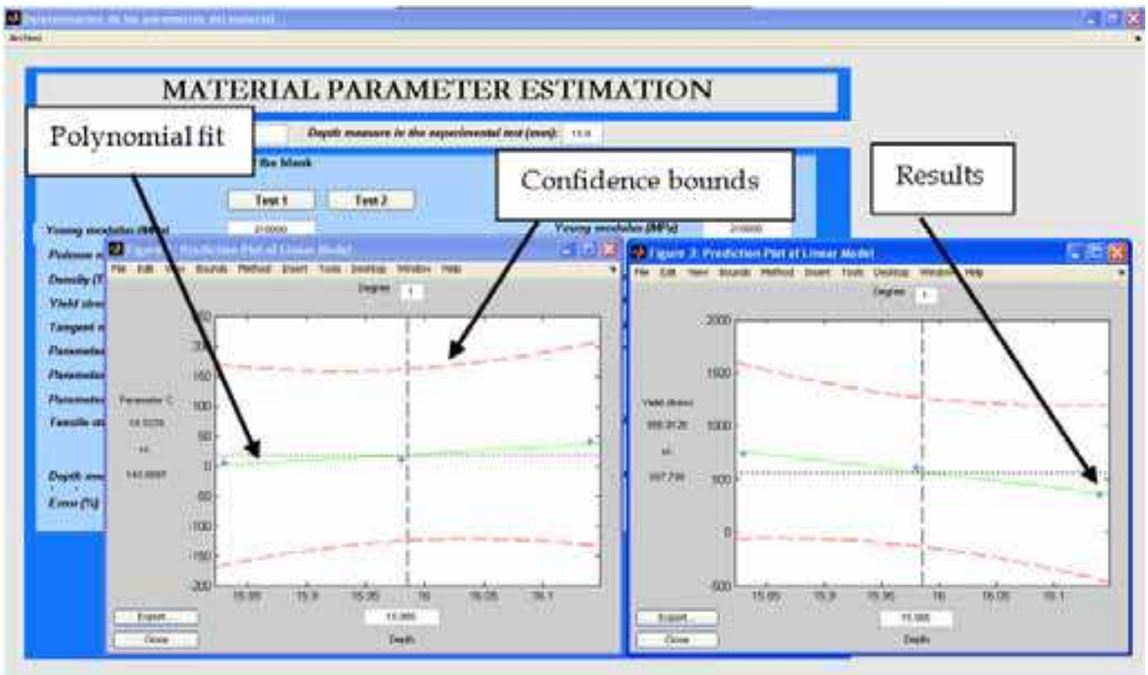


Fig. 8. Polynomial fit estimation and confidence bounds of material parameters

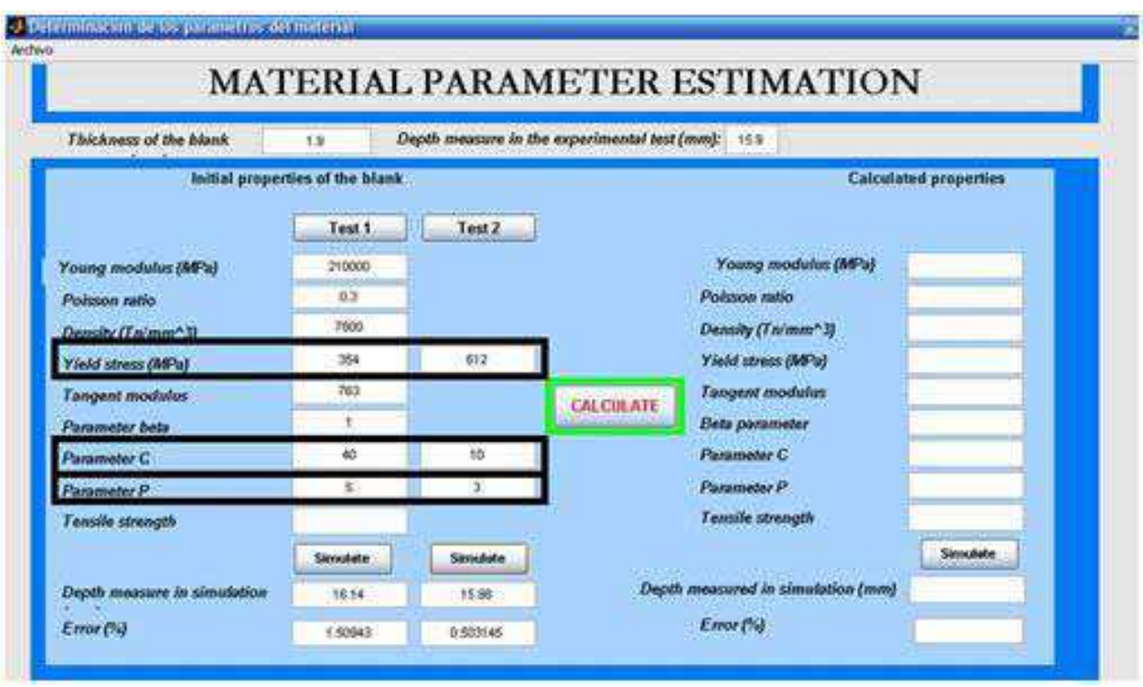


Fig. 9. Material parameters that can be modified by the user

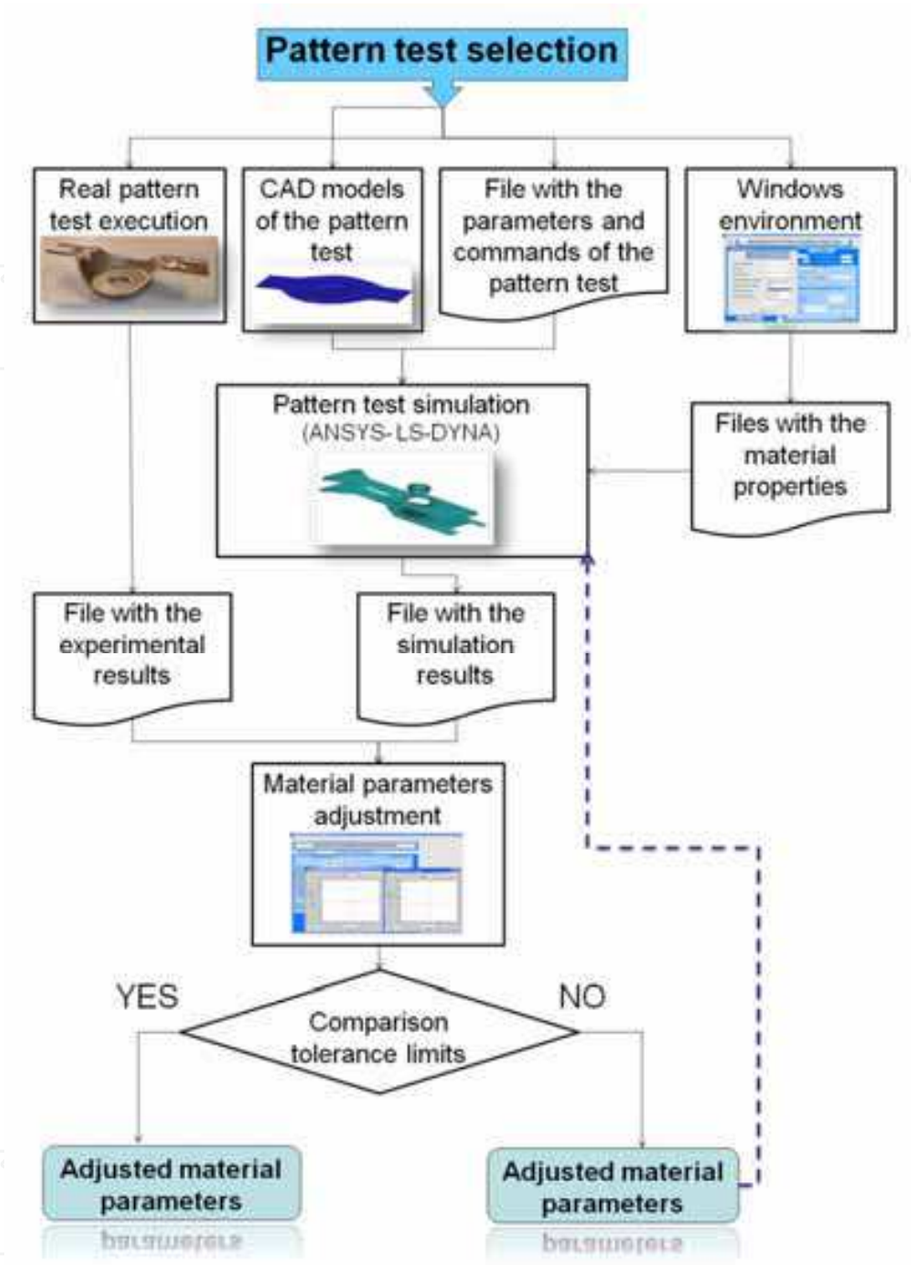


Fig. 10. Material parameters estimation procedure

4. Application example

4.1 Choosing and simulating the pattern test

The first step is choosing the pattern test. For a stamping process, the example explained in 2.3.4 has been chosen. As stated before, this test is the first of the five stages needed to manufacture a part which belongs to the fix system of the spare tire of an real vehicle. The parts involved in this step are shown in Figure 11:

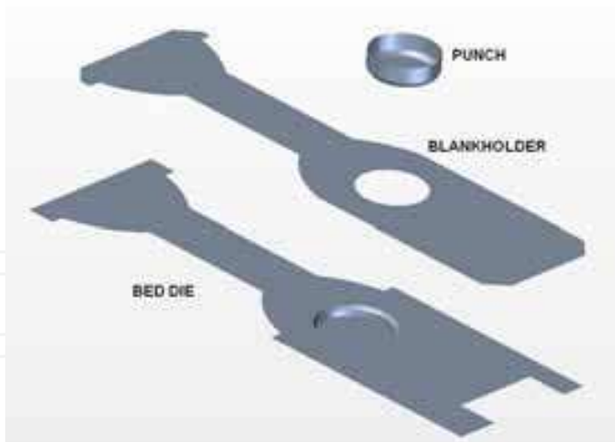


Fig. 11. First step dies

The blank is leaned on the bed die and the process starts with the movement of the blankholder, which applies a load to hold the blank once contact is established between them. After that, punch begins to go down, deforming the blank to obtain the part shown in Figure 2. Deformed blank was measured with a coordinate measuring machine, and the dimension used to be compared with simulation results is shown in Figure 3. Simulation displacements are compared with real ones because displacement measurement assures a controlled final shape of the sheet blank. Other variables such as stress or strains are not useful from a practical point of view for this purpose.

Every parameter involved in this simulation has to be adjusted according to the designer experience and taking into account the conditions of the experimental stamping process (loads, times, boundary conditions...). Boundary and loading conditions have been specified by fixing degrees of freedom of the dies or by aplying displacements and loads to them to simulate the real process (Table 2).

Time [s]	Punch displacement [mm]	Blankholder displacement [mm]	Blankholder load [N]
0	0	0	0
0,5	-38	-25	-90000
1	-78,5	-49,998	-90000
1,5	-116,498	-49,998	-90000
2	-78,5	-49,998	-90000
2,5	-38	-25	-90000
3	0	0	0

Table 2. Loads and displacements used in the pattern test simulation

Those parameters are introduced in the friendly windows environment exposed in chapter 3.2, and the stamping process is automatically simulated by ANSYS LS-DYNA according to the procedure shown in Figure 4.

As long as the patters test is well known and the real experiment can be carried out for any desired material, simulation results can always be compared with experimental values and simulation parameters can be adjusted in order to obtain a validated model.

4.2 Adjusting material parameters for a high strength steel

Once the pattern test can be simulated with great confidence, it is time to use it to adjust parameters of an unknown material in order to optimize results, predict springback and define new dies before carrying out the experimental test.

The material parameter estimation procedure needs two sets of material parameters to start. The program simulates the pattern test with these two sets and the difference between experimental and simulation results is calculated. If this difference is over the tolerance limit specified by the user, the application founds new material parameters by applying linear interpolation to previous ones and launches a new simulation with these new material parameters. The process is repeated until results fit tolerance requirements.

In the experimental test, the displacement of the punch is 16.5 mm. For this value, the final depth of the manufactured part, measured by the MMC machine, is 15.9 mm.

Initial values for the material parameters and the depths obtained for each combination can be seen in Table 3 (1st and 2nd simulations). The last column shows the parameters values obtained after optimization, considering a tolerance limit for the relative error of 0.4%.

Parameter	Number of simulation		
	1st	2nd	**Last**
Density (kg/m^3)	7800	7800	7800
Young's module (MPa)	210000	210000	210000
Poisson ratio	0.3	0.3	0.3
Yield stress (MPa)	354	425	664
Tangent modulus (MPa)	763	763	763
β	1	1	1
C (s^{-1})	40	100	10.99
p	5	5	2.5
Obtained depth (mm)	16.14	16.14	15.96
Relative error	1.5%	1.5%	0.38%

Table 3. Employed parameters

4.3 Results validation

To validate these results, obtained parameters have been used in a new deep stamping process. The selected test covers steps 1, 2 and 3 of the manufacture process of the part shown in Figure 12.

Fig. 12. Manufactured part

This process involves not only geometrical difficulty but also difficulties due to progressive stamping processes. The first step is the pattern test explained in 4.1. Dies used in steps 2 and 3 are shown in Figure 13.

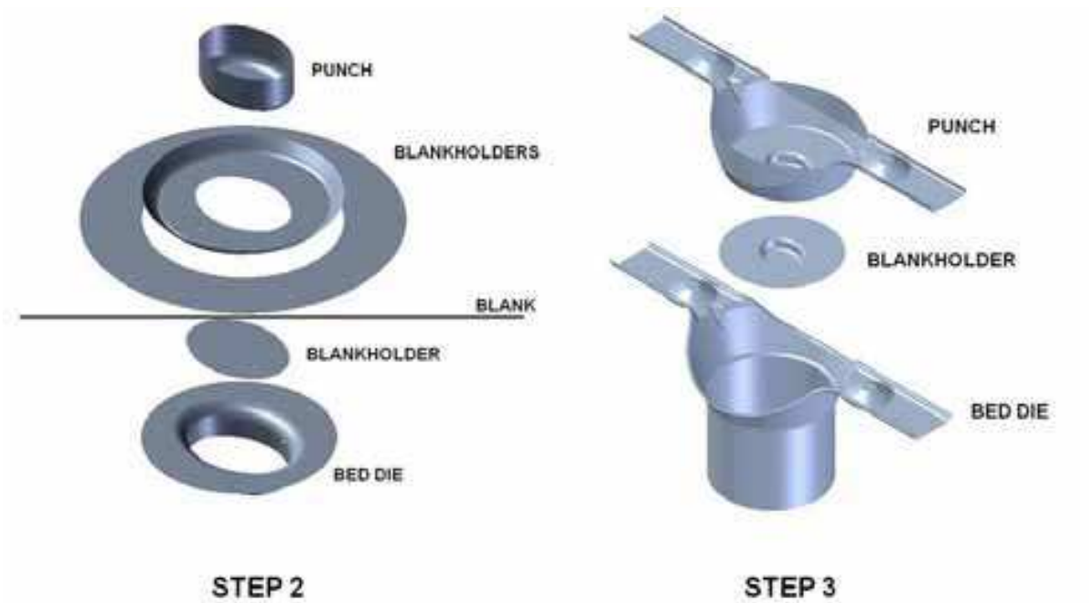


Fig. 13. Second and third steps dies

In this case, the dimension used to validate de model is the one shown in Figure 14. This dimension achieved a value of 77.21 mm in the experimental test after springback. Simulation result was 74.58 mm, representing a 3.4% error.

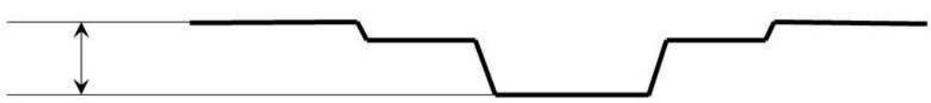


Fig. 14. Final dimension used for validation

5. Adaptive meshing

It has been mentioned before that computing time becomes an important aspect in this kind of simulations. To solve the developed models, a PC can take from several hours to a week, depending mainly on the mesh size and on the amount of plastic strain reached. Mesh size is critical not only for the results quality but for taking into account properly contact between parts. High relative speed between dies characteristic of stamping processes makes necessary to use fine mesh sizes and high contact stiffness, both of them leading to increase computational load.

In addition, to repeat many times the early steps of a multistep process is needed to adjust properly the mesh size in order to get an acceptable going of the latest steps. It multiplies at the same time programming and computing times. In this context, Numeric Calculation Adaptive Meshing (AM) technique is of paramount importance.

Using the AM tool will allow the stress analyst to save because:

- It is not needed to carry out meshing tests. An initial gross mesh can be provided, and in the first calculation it will be automatically refined in those areas in which strains

grow higher. It won't be necessary to have a prediction about the areas that are going to need remeshing neither the remeshing level. Resources are disposed at the time they are required.

- It won't be necessary to provide, for the early steps of the process, a refined mesh in the areas that are going to experiment high strain levels in the last steps. It avoids the calculation in these early steps to be unnecessarily heavy.

5.1 Adaptive meshing tool in LS-DYNA

LS-DYNA (LSTC, 1998) includes an h-adaptive method for the shell elements (Belytschko, et al., 1989). In an h-adaptive method, the elements are subdivided into smaller elements wherever an error indicator shows that subdivision of the elements will provide improved accuracy. The beginning objective of the adaptive process used in LS-DYNA is to obtain the greatest accuracy for a given set of computational resources. The user sets the initial mesh and the maximum level of adaptivity, and the program subdivides those elements in which the error indicator is the largest. Although this does not provide control on the error of the solution, it makes it possible to obtain a solution of comparable accuracy with fewer elements, and, hence, less computational resources, than with a fixed mesh.

The original mesh provided by the user is known as the parent mesh, the elements of this mesh are called the parent elements, and the nodes are called parent nodes. Any elements that are generated by the adaptive process are called descendant elements, and any nodes that are generated by the adaptive process are called descendant nodes. Elements generated by the second level of adaptivity are called first-generation elements, those generated by third level of adaptivity are called second-generation elements, etc. The coordinates of the descendant nodes are generated by using linear interpolation.

Refinement indicators are used to decide the locations of mesh refinement. One deformation based approach checks for a change in angles between contiguous elements.

5.2 Adaptive meshing programming in LS-DYNA

EDADAPT command activates AM for a part of the simulation. It should be applied to blank parts, since dies are modeled as rigid and no strains or stresses are calculated into dies. The mesh size of rigid dies can be as fine as desired because it does not imply additional calculations. For example, to activate AM for PART #1 the following command must be written:

EDADAPT, 1, ON

AM activation command is placed just before SOLVE command, and does not modify any other programming structure, which makes possible an easy incorporation to the automation scheme described in previous sections.

5.3 Adaptive meshing controls

Adaptive Meshing control parameters have to be defined by means of EDCADAPT command. These parameters are defined just below (ANSYS, 2005):

- **FREQ-** Time interval between adaptive mesh refinements.
    - **TOL-** Adaptive angle tolerance (in degrees) for which adaptive meshing will occur. If the relative angle change between elements exceeds the specified tolerance value, the elements will be refined.
    - **OPT-** Adaptivity option:

- 1. Angle change (in degrees) of elements is based on original mesh configuration.
- 2. Angle change (in degrees) of elements is incrementally based on previously refined mesh.
- **MAXLVL-** Maximum number of mesh refinement levels. This parameter controls the number of times an element can be remeshed. Values of 1, 2, 3, 4, etc. allow a maximum of 1, 4, 16, 64, etc. elements, respectively, to be created for each original element.
- **BTIME/DTIME-** Birth/Death time to begin/end adaptive meshing. It controls when AM is activated/deactivated
- **LCID-** Data curve number identifying the interval of remeshing. The abscissa of the data curve is time, and the ordinate is the varied adaptive time interval. If LCID is nonzero, the adaptive frequency (*FREQ*) is replaced by this load curve. Note that a nonzero *FREQ* value is still required to initiate the first adaptive loop.
- **ADPSIZE-** Minimum element size to be adapted based on element edge length.
- **ADPASS-** One or two pass adaptivity option:
- 0. Two pass adaptivity. Results are recalculated after remeshing.
- 1. One pass adaptivity. Results are not recalculated after remeshing.

- **IREFLG-** Uniform refinement level flag. Values of 1, 2, 3, etc. allow 4, 16, 64, etc. elements, respectively, to be created uniformly for each original element.
- **ADPENE-** Adaptive mesh flag for starting adaptivity when approaching (positive *ADPENE* value) or penetrating (negative *ADPENE* value) the tooling surface. Adaptive tool refinement is based on the tool curvature.
- **ADPTH-** Absolute shell thickness level below which adaptivity should begin. This option works only if the adaptive angle tolerance (*TOL*) is nonzero. If thickness based adaptive remeshing is desired without angle change, set *TOL* to a large angle.
- **MAXEL-** Maximum number of elements at which adaptivity will be terminated. Adaptivity is stopped if this number of elements is exceeded.

Adaptive Meshing used to simulate stamping processes has shown to work properly with the combination of control parameters revealed below:

EDCADAPT,0.1,0.5,2,3,0,1 , ,0,0,0,0,0,0,

Which means:

FREQ=0.1; TOL=0.5; OPT=2; MAXLVL=3; BTIME=0; DTIME=1.

These values can vary from one simulation to another.

5.4 Computing time saving

The 2-step stamping process analyzed in section 4 has been carried out with and without AM option, in the same computer, reaching very similar results in both cases.

In the case fine mesh is programmed from the beginning of the calculation, first step took 50 hours and second step 70 hours; 120 hours to complete the entire calculation.

In the case AM is programmed (Figure 15) over a gross initial mesh, 10 hours have been taken to complete calculation.

Additionally, these times does not take account of the efforts made by the stress analyst to find the appropriate mesh density for each blank area as a function of the final plastic strain.

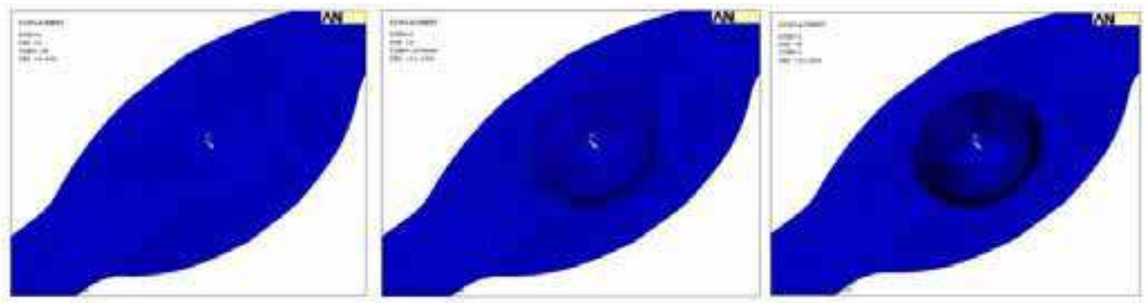


Fig. 15. Evolution of the adaptive mesh in step one simulation

5.5 Problems encountered during adaptive meshing implementation

As has been shown in section 2.2, combined "Explicit to Implicit" simulations have resulted to be the most appropriate way to simulate the complete stamping process, using Full Restart option to concatenate different stamping steps. However, ANSYS Release 10.0 Documentation says textually:

"Adaptive meshing: Adaptive meshing (EDADAPT and EDCADAPT) is not supported in a full restart. In addition, a full restart is not possible if adaptive meshing was used in the previous analysis. "(ANSYS, 2005)

So it can be concluded that using LS-DYNA AM tool to simulate a multistep stamping process forces the stress analyst to develop a unique Explicit procedure, programming different dies approximation and retiring in the same calculation.

6. Conclusions

According with previous expositions and results, it can be concluded:

- A procedure to simulate real sheet metal forming processes by means of finite elements has been established.
- To define this procedure, several options have been analyzed for each step of the process, choosing the one more suitable between the possibilities offered by finite elements software.
- Such a procedure has been automated and allows performing simulations with no user intervention, avoiding the difficulty of using a high-level program as LS-DYNA.
- By means of this automated procedure a methodology to adjust material parameters has been developed.
- Parameters involved in each material model have been identified and their influence in final results has been quantified. This is very useful to fit material properties in other simulations.
- This methodology is based in real experimental and simulation results and in a material parameter fit estimation procedure.
- Real industry experimental tests to validate the simulation results, instead of benchmark theoretical tests, have been carried out. This allows to use previous knowledge of the designer, to particularize material characterization for each kind of process and avoids building specific tooling.
- Simulation model has been validated by comparing its results with those obtained in experimental tests. An example of a real application of the industry has been presented.

- LS-DYNA adaptive meshing has been also tested. Results obtained by using it are virtually the same as those validated before and time is greatly reduced. So, it can be conclude that using adaptive meshing highly recommended.
- Using adaptive meshing forces to avoid implicit simulations in springback estimation. Therefore, a complete explicit simulation of the application and withdrawal of dies must be carried out.

7. Acknowledgment

The authors want to thank ARRAN Automoción Group for its great interest and collaboration in this work and the Government of Spain for the support given through the project 370100-103 of the PROFIT program.

8. References

Álvarez- Caldas, C., et al. (2009). Expert System for Simulation of Metal Sheet Stamping. *Engineering with computers*, Vol. 25, No. 4, pp. 405- 410. ISSN 0177-0667.

ANSYS (2005). *ANSYS LS- DYNA User's Guide. ANSYS release 10.0*. ANSYS Inc. Canonsburg, USA.

Belytschko, T., et al. (1989). Fission - Fusion Adaptivity in Finite Elements for Nonlinear Dynamics of Shells. *Computers and Structures*, Vol. 33, No. pp. 1307- 1323, ISSN

Buranathiti, T.&Cao, J. (2005a). Numisheet2005 Benchmark Analysis on Forming of an Automotive Deck Lid Inner Panel: Benchmark 1, *NUMISHEET 2005: Proceedings of the 6th International Conference and Workshop on Numerical Simulation of 3D Sheet Metal Forming Process. AIP Conference Proceedings*, Detroit, Michigan, USA, 15-19/08/2005.

Buranathiti, T.&Cao, J. (2005b). Numisheet2005 Benchmark Analysis on Forming of an Automotive Underbody Cross Member: Benchmark 2, *NUMISHEET 2005: Proceedings of the 6th International Conference and Workshop on Numerical Simulation of 3D Sheet Metal Forming Process. AIP Conference Proceedings*, Detroit, Michigan, USA, 15-19/08/2005.

Cowper, G. R.&Symonds, P. S. (1958). *Strain Hardening and Strain Rate Effects in the Impact Loading of Cantilever Beams*. Brown University. Providence, Rhode Isl, USA.

Chen, M. H., et al. (2007). Application of the forming limit stress diagram to forming limit prediction for the multi-step forming of auto panels. *Journal of Materials Processing Technology*, Vol. 187-188, No. pp. 173-177, ISSN 0924-0136

Darendeliler, H.&Kaftanoglu, B. (1991). Deformation Analysis of Deep-Drawing by a Finite Element Method. *CIRP Annals - Manufacturing Technology*, Vol. 40, No. 1, pp. 281-284, ISSN 0007-8506

Dietenberger, M., et al. (2005). Development of a High Strain-Rate Dependent Vehicle Model, *4th LS-DYNA Forum*, Bamberg, Germany 20th - 21st of October 2005.

Firat, M. (2007a). Computer aided analysis and design of sheet metal forming processes: Part I - The finite element modeling concepts. *Materials & Design*, Vol. 28, No. 4, pp. 1298-1303, ISSN 0261-3069

Firat, M. (2007b). Computer aided analysis and design of sheet metal forming processes: Part II - Deformation response modeling. *Materials & Design*, Vol. 28, No. 4, pp. 1304-1310, ISSN 0261-3069

Firat, M. (2007c). U-channel forming analysis with an emphasis on springback deformation. *Materials & Design*, Vol. 28, No. 1, pp. 147-154, ISSN 0261-3069

Gau, J.-T. (1999). *A Study of the Influence of the Bauschinger Effect on Springback in Two-Dimensional Sheet Metal Forming*. Ph.D. Degree. The Ohio State University. Ohio.

Gauchía, A. et al. (2009). Material parameters in a simulation of metal sheet stamping. *Proceedings of the Institution of Mechanical Engineers Part D-Journal of Automobile Engineering.* Vol. 223. No. 6, pp. 783- 791. ISSN 0954-4070.

Hallquist, J. O. (1998). *LS-DYNA Theoretical Manual*. LSTC. Livermore, California, USA.

Ling, Y. E., et al. (2005). Finite element analysis of springback in L-bending of sheet metal. *Journal of Materials Processing Technology*, Vol. 168, No. 2, pp. 296-302, ISSN 0924-0136

LSTC (1998). *LS-DYNA Theoretical Manual*. Livermore Software Technology Corporation. Livermore, California, USA.

LSTC (2006). *LS-DYNA: User's Manual Version 971*. Livermore Software Technology Corporation. Livermore, California, USA.

Makinouchi, A. (1996). Sheet metal forming simulation in industry. *Journal of Materials Processing Technology*, Vol. 60, No. 1-4, pp. 19-26, ISSN 0924-0136

Mattiasson, K., et al. (1995). Simulation of springback in sheet metal forming, *Proceedings of the NUMIFORM'95 Simulation of Materials Processing: Theory, Methods and Applications*, Cornell University, Ithaca, NY, USA,

Morestin, F., et al. (1996). Elasto plastic formulation using a kinematic hardening model for springback analysis in sheet metal forming. *Journal of Materials Processing Technology*, Vol. 56, No. 1-4, pp. 619-630, ISSN 0924-0136

Narasimhan, N.&Lovell, M. (1999). Predicting springback in sheet metal forming: an explicit to implicit sequential solution procedure. *Finite Elements in Analysis and Design*, Vol. 33, No. 1, pp. 29-42, ISSN 0168-874X

Ortiz, M.&Quigley, J. J. (1991). Adaptive mesh refinement in strain localization problems. *Computer Methods in Applied Mechanics and Engineering*, Vol. 90, No. 1-3, pp. 781-804, ISSN 0045-7825

Quesada, A., et al. (2009). Influence of the parameters of the material model in finite element simulation of sheet metal stamping, *7th EUROMECH Solid Mechanics Conference*, Lisbon (Portugal), September, 7th- 11th, 2009.

Quigley, E.&Monaghan, J. (2002). Enhanced finite element models of metal spinning. *Journal of Materials Processing Technology*, Vol. 121, No. 1, pp. 43-49, ISSN 0924-0136

Samuel, M. (2004). Numerical and experimental investigations of forming limit diagrams in metal sheets. *Journal of Materials Processing Technology*, Vol. 153-154, No. pp. 424-431, ISSN 0924-0136

Silva, M. B., et al. (2004). Stamping of automotive components: a numerical and experimental investigation. *Journal of Materials Processing Technology*, Vol. 155-156, No. pp. 1489-1496, ISSN 0924-0136

Song, J.-H., et al. (2007). A simulation-based design parameter study in the stamping process of an automotive member. *Journal of Materials Processing Technology*, Vol. 189, No. 1-3, pp. 450-458, ISSN 0924-0136

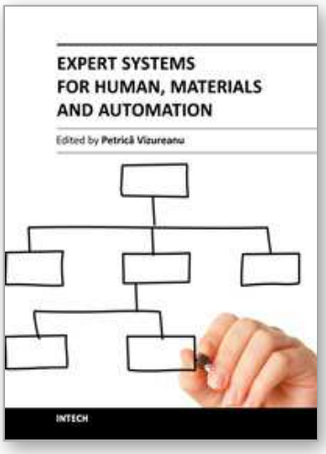
Taylor, L., et al. (1995). Numerical simulations of sheet-metal forming. *Journal of Materials Processing Technology*, Vol. 50, No. 1-4, pp. 168-179, ISSN 0924-0136

Tekkaya, A. E. (2000). State-of-the-art of simulation of sheet metal forming. *Journal of Materials Processing Technology*, Vol. 103, No. 1, pp. 14-22, ISSN 0924-0136

Wei, L.&Yuying, Y. (2008). Multi-objective optimization of sheet metal forming process using Pareto-based genetic algorithm. *Journal of Materials Processing Technology*, Vol. 208, No. 1-3, pp. 499-506, ISSN 0924-0136

IntechOpen

IntechOpen



Expert Systems for Human, Materials and Automation

Edited by Prof. PetricÄƒ Vizureanu

ISBN 978-953-307-334-7

Hard cover, 392 pages

Publisher InTech

Published online 10, October, 2011

Published in print edition October, 2011

The ability to create intelligent machines has intrigued humans since ancient times, and today with the advent of the computer and 50 years of research into AI programming techniques, the dream of smart machines is becoming a reality. The concept of human-computer interfaces has been undergoing changes over the years. In carrying out the most important tasks is the lack of formalized application methods, mathematical models and advanced computer support. The evolution of biological systems to adapt to their environment has fascinated and challenged scientists to increase their level of understanding of the functional characteristics of such systems. This book has 19 chapters and explain that the expert systems are products of the artificial intelligence, branch of computer science that seeks to develop intelligent programs for human, materials and automation.

How to reference

In order to correctly reference this scholarly work, feel free to copy and paste the following:

Alejandro Quesada, Antonio Gauchĺa, Carolina Ǻlvarez-Caldas and José-Luis San- Román (2011). Expert System for Simulation of Metal Sheet Stamping: How Automation Can Help Improving Models and Manufacturing Techniques, Expert Systems for Human, Materials and Automation, Prof. PetricÄƒ Vizureanu (Ed.), ISBN: 978-953-307-334-7, InTech, Available from: http://www.intechopen.com/books/expert-systems-for-human-materials-and-automation/expert-system-for-simulation-of-metal-sheet-stamping-how-automation-can-help-improving-models-and-ma

INTECH
open science | open minds

InTech Europe
University Campus STeP Ri
Slavka Krautzeka 83/A
51000 Rijeka, Croatia
Phone: +385 (51) 770 447
Fax: +385 (51) 686 166
www.intechopen.com

InTech China
Unit 405, Office Block, Hotel Equatorial Shanghai
No.65, Yan An Road (West), Shanghai, 200040, China
中国上海市延安西路65号上海国际贵都大饭店办公楼405单元
Phone: +86-21-62489820
Fax: +86-21-62489821

© 2011 The Author(s). Licensee IntechOpen. This is an open access article distributed under the terms of the Creative Commons Attribution 3.0 License, which permits unrestricted use, distribution, and reproduction in any medium, provided the original work is properly cited.