Blucher Mechanical Engineering Proceedings
May 2014, vol. 1 , num. 1
www.proceedings.blucher.com.br/evento/10wccm

DD3LT - A UNIFIED PLATFORM FOR LEARNING AND TEACHING THE VIRTUAL TRY-OUT OF SHEET METAL FORMING PROCESS

R. M. Heleno[1] , M. C. Oliveira[1], J. L. Alves[3], L. F. Menezes[1]

[1]CEMUC, Mechanical Engineering Center of the University of Coimbra, Department of Mechanical Engineering, University of Coimbra (luis.menezes@dem.uc.pt)

[2]CT2M, Center for Mechanical and Materials Technologies, Department of Mechanical Engineering, University of Minho

Abstract. *This work describes the DD3LT platform developed to support the learning and teaching of virtual try-out of sheet metal forming processes, based on the awareness of the problems associated with the careless use of finite element analysis codes. This platform is supported on the DD3IMP code, which is an in-house FE solver that has been continuously developed and optimized to simulate sheet metal forming processes. The DD3LT platform integrates an interactive application, in order to help the model pre and post processing, as well as an extensive database of sheet metal forming benchmark problems.*

Keywords: *FEM, Sheet metal forming, DD3IMP, Virtual try-out, Teaching*

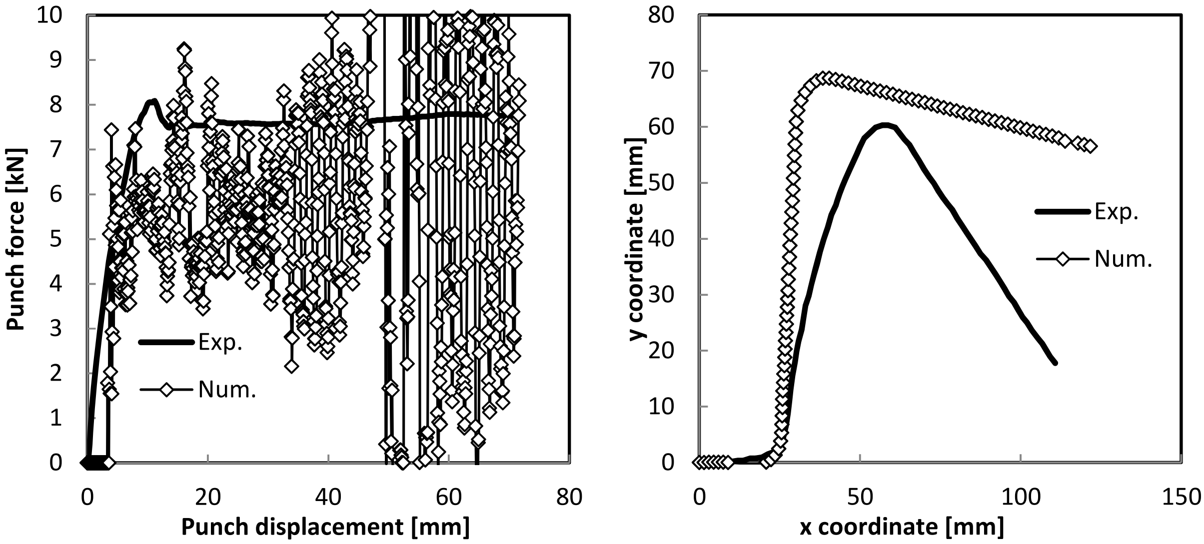
1. INTRODUCTION

Nowadays, powerful numerical tools are available to support mechanical engineers decision process. In fact, computational virtual try-out enables detailed studies and development of a myriad of problems in different engineering areas, changing the way to approach the design of more complex industrial parts. In this context, FEA of engineering problems has become an indispensable tool, since it allows the optimization of the parameters that dictate the success of the technological processes. Sheet metal forming is one of the technological processes for which FEA virtual try-out has assumed an undeniable importance in the last years, leading also to increasing technological developments. The determination of the process design window involves the analysis of many parameters such as, tool geometry, material parameters, etc. The FEA virtual try-out reduces the design and manufacturing time as well as costs. Therefore, sheet metal forming industry relies on FEA for the design and manufacturing process of stamping tools, particularly, for more complex multi-stage processes.

Sheet metal forming is a technological process in which the original flat geometry of a thin metal sheet is modified to the desired shape, by applying external forces that induce plastic deformation of the material. This process allows high volume production of sheet metal parts of different complexities. Thus, one of the most important industries exploring the advantages of this kind of process is the automotive, due to the high production rates. In the last years, this industry has been continuously driven by new environmental and security rules, energy conservation laws and strong demands on sustainable development, which lead to steadily increasing requirements for stronger and lighter materials. In fact, there is a high demand for decreasing the vehicles weight so that fuel efficiency is improved as well as their security by improving crash performance. In this context, the application of deep drawing processes to new materials is being highly influenced mainly by this industry. Thus, new materials like high strength steels and aluminum alloys have found an increasing use in the automotive industry since they can lead to lightweight components. In fact, advanced high strength steels are being used for more than 60% of the body parts of modern cars [19]. The growing complexity of deep drawing components has been leading to a greater dependence of virtual production concepts, in particular the numerical simulation of metal forming processes resorting to the finite element method and the extension of its use throughout all the production chain [28]. The numerical simulation allows the virtual validation of forming tools and process parameters, leading to a time and costs decrease related when compared to its experimental testing. In fact, it allows predicting the material flow, analyzing stress, strain and temperature distribution, determining forming forces, forecasting potential sources of defects and failures, improving part quality and complexity and reducing manufacturing costs. Nowadays, in an integrated manufacturing environment modeling and simulation are often integrated parts of product and process design [32]. In addition, numerical simulation can help optimizing the entire production chain, from the raw material to the assembled product. This can translate in an enormous profit in both economic, time and technical terms, crucial in the current highly competitive market.

All these factors make the use of Finite Element Analysis (FE) of the utmost importance in order to virtually design and optimize sheet metal forming processes. Such virtual try-out approach is consensually accepted as the main factor for the huge decrease in the time-to-market life cycle of new formed parts as well as for the notable savings in terms of money, time and effort in their design, production and process set-up. Nowadays, there are many commercial codes specifically developed to the numerical simulation of forming processes, which are widely used in the industry. The increasing accuracy of the numerical simulation results and of the computational power also contributed to the high industrial interest for tools virtual try-out, since nowadays it is possible to analyze components and processes of increasing complexity.

Despite the clear potential of numerical simulation in technological processes analysis, its uncontrolled use is extremely dangerous. The CAE engineer should be aware that all the numerical simulations are model dependent and, consequently, imperfect and somehow wrong. Therefore, the correct interpretation of results requires specialized personnel with a detailed knowledge of the technological process and also of the code, such as, numerical methods, numerical parameters and algorithms, modeling of the mechanical behavior of the materials, etc. In fact, there is also a continuous effort in making the use of FEA more simple and interactive. The industry expects that the usage of the code should be so simple, that there is no need for an extra finite element expert. Furthermore, the simulation tool should be available there where it is needed, that is, it must be usable in the design office and not only in the computational department of the company. Therefore, user-friendliness is a critical issue for the application of the simulation tool in industry [30]. However, the risks associated to the incorrect usage of this powerful tool are high and are potentiated by its increasing versatility, as highlighted by the benchmark results for the NUMISHEET conferences. For example, in the *BM 4 - Pre-strain Effect on Spring-back of 2-D Draw Bending*, proposed by the benchmark committee of NUMISHEET'11 conference, one of the participants reported the results presented in Figure 1. This figure compares the experimentally evaluated punch force evolution with the punch displacement and the profile, after springback, with the numerical result predicted by one of the participants [7]. The incorrect control of the numerical parameter of a well-known dynamic explicit code leads to the inaccurate prediction of both results.



(a) (b)

Figure 1. Comparison between experimental and numerical results for BM4, NUMISHEET'11: (a) Punch force evolution with punch displacement; (b) profile after springback [7].

The awareness of the problems related with the careless use of FEA codes for sheet metal forming problems, this work presents the DD3LT platform, developed to support the learning and teaching of virtual try-out of sheet metal forming processes. This platform is supported on the DD3IMP code, which is an in-house FE solver that has been continuously developed and optimized to simulate sheet metal forming processes [21,24,25]. It also integrates an extensive database of sheet metal forming benchmark problems and an interactive application for pre and post processing the models. In the following section, the main features of DD3LT platform are described.

2. DD3LT PLATFORM

2.1. DD3IMP in-house code

The finite element code DD3IMP (which stands for Deep-Drawing 3D IMPlicit code) has been specifically developed to simulate sheet metal forming processes. The mechanical model takes into account large elastoplastic strains and rotations, and assumes that the elastic strains are negligibly small with respect to unity. Elastic behavior is assumed to be isotropic. The plastic behavior is described through phenomenogical constitutive models based on the definition of: (i) an associated flow rule; (ii) a yield criterion and (iii) a work-hardening law. There are several yield criterion implemented in DD3IMP, considering isotropic (von Mises, 1993 [33]; Drucker, 1949 [10]; and Hosford, 1972 [12]) and orthotropic (Hill, 1948 [11]; Bartlat et al., 1991 [3]; Karafilis and Boyce, 1993 [15]; Cazacu and Barlat, 2001 [5]; Drucker +L [3,5,15] and Cazacu et al. 2006 [6]) behavior. The isotropic work hardening behavior can be described either using the Swift, 1947 or the Voce, 1948 laws, which can be combined with the Prager, 1955 [26] and the Lemaître and Chaboche, 1985 [17] kinematic work hardening laws. The work hardening behavior can also be described using the complete or the simplified Teodosiu and Hu, 1998 models [31].

The updated lagrangian formulation implemented is based on the principle of virtual velocities proposed by McMeeking and Rice (1975) [20]. An explicit approach is used to calculate an approximate first solution for the nodal displacements, the stress states and frictional contact forces. A $r_{\min}$ strategy is implemented to impose several restrictions on the size of the time increment in order to improve the convergence [35]. The first trial solution is iteratively corrected, using a Newton–Raphson algorithm, finishing when a satisfactory equilibrium state in the deformable body is achieved. It is then possible to update the blank sheet configuration, as well as all the state variables, passing on to the calculation of the next increment. This is repeated until the end of the process [21]. Table 1 presents a resume of DD3IMP main algorithm.

In sheet metal forming processes the boundary conditions are dictated by the contact established between the blank sheet and the tools. Such boundary conditions continuously change during the forming process, increasing the importance of correctly evaluating the actual contact surface and the kind of contact that is established at each point of the deformable body. A master–slave algorithm is adopted, with the tools behaving as rigid bodies. Coulomb's classical law models the friction contact problem between the tools and the blank sheet (deformable body). The contact with friction problem is treated by an augmented lagrangian approach [21,24,25]. Then the above mentioned fully implicit Newton–Raphson scheme is used to solve, in a single iterative loop, all the problem non-linearities associated with either the contact with friction problem or the elastoplastic behavior of the deformable body.

The forming tools are modeled using parametric surfaces, Bézier or Nagata type [23]. The blank sheet is discretized with 3D solid finite elements. Although penalized in this type of applications by computational cost and effectiveness, solid elements have many advantages. Among others, they allow the accurate evaluation of the contact forces through an accurate description of contact evolution and thickness change; the simultaneous contact on both sides of the sheet is naturally solved without any particular strategy or tricky algorithms. Also, solid elements are required for accuracy in FE springback simulation when the ratio between the tool radius and blank thickness is lower than 5–6 [18]. These facts have motivated many studies on the improvement of solid elements for sheet metal forming simulations [2,14,27,34]. In DD3IMP the traditional tri-linear eight-node hexahedral finite element can be applied using full integration, reduced integration or associated with a selective reduced integration scheme (SRI) [13]. Although the SRI scheme in torsion-dominant problems can exhibit spurious zero-energy modes, this kind of finite elements allows efficient computation of the thickness evolution as well as the through-thickness stress gradients [1,22], depending on the type of applications and on the number of elements thought thickness and in sheet plane. There are other types of solid elements available in the finite element library, including 20-node serendipity elements and the tri-quadratic 27-node hexahedral finite elements.

Table 1. DD3IMP main algorithm.

START
➢ Read and verify the input data
➢ Initialize the increment number $N = 1$
➢ Repeat
▪ ***Prediction***
✓ Impose the trial increment
✓ Impose contact with friction conditions
✓ Calculate the tangent stiffness matrix and nodal force vector
✓ Solve the system of equations for the imposed trial increment
✓ Calculate strains and stresses
✓ Calculate the $r_{\min}$ value to define the actual increment size
✓ Update the sheet and tools position
✓ Update the contact variables
▪ ***Correction***
✓ Update the contact with friction conditions
✓ Calculate the strain and rotation increments
✓ Integrate the material’s behavior law
✓ Solve the system of equations
▪ Validate an eventual change of phase in the process
▪ Actualize the increment number $N = N + 1$
Until the end of the process
END

DD3IMP allows the use of three different strategies to simulate the unloading phase. The first one can be understood as a simple continuation of the forming process, as the tools’ motion is reversed and the computation is carried out until the end of the process (loss of contact between the tools and the formed part). This unloading strategy is in very close agreement with the physics of the real process itself, since it allows the changes in the contact areas between the blank sheet and tools during the unloading phase to be tracked. However, this procedure leads to a significant increase in CPU time due to the reversing tools’ displacement and can lead to convergence problems due to the discrete character of the contact. The second possible strategy consists of removing the tools, one by one, using only one time increment per tool (punch, die,…), forcing the equilibrium at each step by an implicit equilibrium iterative loop. The third strategy performs springback in only one step, removing all the tools simultaneously and forcing the blank sheet to attain equilibrium. In this last strategy, named ‘‘One Step Springback’’, all the constraints imposed by the tools vanish at the beginning of the unloading phase. There is no need to perform a trial solution since the initial solution for the implicit scheme corresponds to the configuration at the end of the forming phase.

The model is defined using ANSI ASCII input files with a predefined format, which is a commonly adopted approach in many FEA solvers. The standard input files necessary to define the model are presented in Figure 2. Globally, these files contain the following information:

- DD3_bcon file is used to impose the problem boundary conditions. These can be planes with restrained displacement (e.g. the symmetry planes) or specific points with fixed displacements (e.g. points used to control the springback phase).
- DD3_contact file is used to define the contact sets, i.e. to associate specific regions to specific tools, in order to minimize the contact search problem dimension. The Coulomb friction coefficient between the blank and the tool is also defined in this file. It is possible to define a global friction coefficient, different friction coefficients for each contact set (e.g. two different friction coefficients between the upper blank surface and the tool and the lower blank surface and the tool) and a different friction coefficient for each patch, used to define the tools geometry.
- DD3_input file is used to define all numerical parameters (e.g. convergence criteria, maximum number of iterations, tolerances and residues) as well as the output data (e.g. output files for results visualization, variables stored in the output post-processing files).
- DD3_mater file is used to define the material parameters, according to the previously selected yield criteria and hardening law. This file is also used to define the rolling direction according to the global axis.
- DD3_mesh file is used to define the blank finite element discretization: coordinates of each node and each finite element connectivity (i.e. the nodes belonging to each element). There are several formats available for this file, based on the preprocessor used to define blank discretization.
- DD3_phase file is used to define the forming process conditions, i.e. the total number of tools and phases and the role of each tool in each phase. The initial displacement of the tools is also defined in this file. Finally, in this file each tool is related with one of the contact sets, previously defined in DD3_contact.dat.
- DD3_tool file is used to define the tools geometry. The parametric surface description can be defined using either Bézier or Nagata patches. When using Nagata patches it is necessary to define a finite element discretization. In order to recover the surface normal with the Nagata patches, it is necessary to know the normal in each node of the surface discretization. When the information about tool geometry is available in IGES format, the normal in each node can be evaluated using this CAD file. The tool discretization can be obtained using GID preprocessor. There-

fore, when using Nagata patches, it is necessary to define for each N tool: DD3_toolN.igs; and a DD3_toolN.msh file.

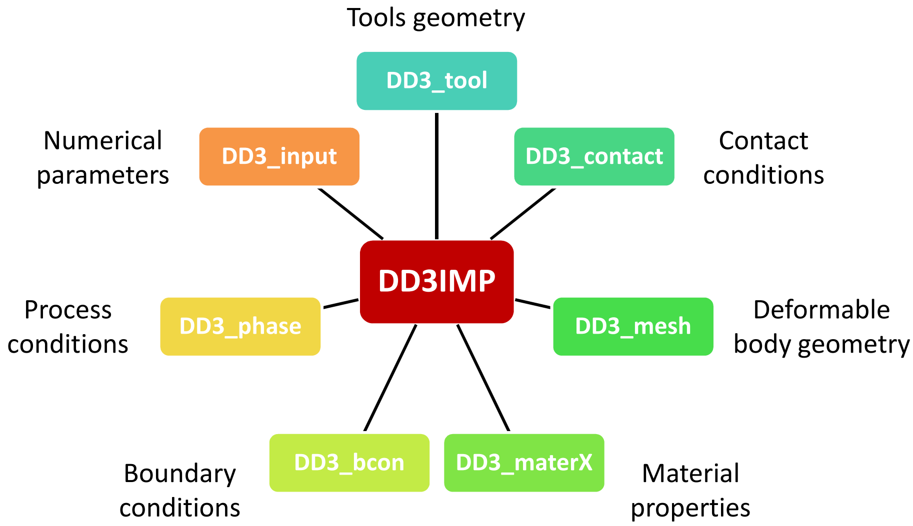


Figure 2. Standard input files necessary to define the FEA model in DD3IMP.

Figure 3 presents an example of the DD3_input standard ANSI ASCII input file, necessary to control the numerical parameters of DD3IMP solver. This figure highlights the characteristics common to the majority of the input files: the information is structured in a columnar form; each parameter as a fixed number of characters associated to its format and; it is important to know the parameters description in order to input valid values. It is important to mention that this type of layout favors the error occurrence, which is difficult to detect due to the large number of parameters. In order to circumvent this disadvantage, an interactive platform was developed, which is described in the following section.

```
===================+============================================================
Simulation and     |     NSTART      NEND      NOUT      iGID       INC       DEV
Output Data        |          1     20000        05         5        51         0
-------------------+------------------------------------------------------------
Tolerances and     |      TOLEQ    TEQOUT     RAPEQ     TOLST      CUNL    DUdamp
residues           |   1.0E-02   1.0E-01   1.0E+09   1.0E-08     0.999      1.00
-------------------+------------------------------------------------------------
Maximum number     |      IRMAX    IEQMAX    NMAXST
of iterations      |          1        50        25
-------------------+------------------------------------------------------------
Max. Increments    |      DEMAX     DWMAX    DSNMAX    DSTMAX
for each NST       |     0.0500    1.5000       6.0       3.0
-------------------+------------------------------------------------------------
Rmin               |       RINF      RSUP    DFNMAX   DFT1MAX   DFT2MAX
Strategy           |     0.0010     5.000       0.0       0.0       0.0
-------------------+------------------------------------------------------------
Solver             |      LEVEL    TOLCGV
Parameters         |          4   1.0E-13
-------------------+------------------------------------------------------------
Input data,        |     MEPOPT    iphOSS
Cep                |          1         3
===================+============================================================
```

Figure 3. DD3_input default input file.

2.2. Interactive application

The main goal of the interactive application was to reduce the complexity in the use of DD3IMP solver. The user access to a large amount of different type of parameters is important in order to be able to explore them, but contributes for the increasing complexity in a first approach. Therefore, the goal was to diminish the time and effort necessary to achieve the results analysis, which is the more important phase for learning sheet metal forming technologies. The interactive application was built based on the underlining principle that the user may not be familiar with all the parameters available in the input files. Thus, although the parameters are visible and the user will receive information about them, the application will control its range of validity.

The interactive application works based on the selection of a previously defined model. The idea is that the user will modify the parameters associated to that initial model. The user will be able to change all type of parameters, including:

- Numerical parameters, e.g. increment size for each phase, number of iterations in each increment, penalty parameter for the augmented lagrangian method;
- Process conditions, e.g. friction coefficient between the tools and the blank, blank-holder force value, tools' displacement, springback strategy;
- Material properties, e.g. material work hardening law, yield criteria or a different material;
- Blank characteristics, e.g. finite element type, finite element integration strategy, dimensions, finite element discretization;
- Tools' geometry, e.g. change the die radius, remove the blank folder.

The interactive application also helps to organize the information concerning the models, associating each set of input files to a different directory. The changed input files are saved in the working directory by the interactive application, which also launches the selected/ altered simulation. The interactive application allows following the running simulation in real time, by visualizing the updated information concerning the increment and the tools displacement. It also generates an EXCEL© file report with the tools' force evolution and allows visualizing same fields (e.g. strain, stress, contact forces).

The interactive application was programmed in C++, on MS Visual Studio 2010 Professional SP1, using the MFC10.0 classes available on .NETFramework 4.0. Special care was taken in order to guarantee compatibility between different platforms (x86 and x64) as well as different operating systems (Windows XP, Windows Vista, Windows Seven). For visualization purposes the 3D (vtk) rendering library was selected. The model for the application was developed based on object oriented programming, based on the Model-View-Controller architecture.

Figure 4 presents an example of the interactive platform showing the DD3_input standard ANSI ASCII input file, as well as the pane for controlling the numerical parameters. As show also in Figure 4, there is a description associated to each parameter. In order to avoid incorrect input values, validation tests were implemented for the numerical parameters. When a parameter is changed it is indicated in bold format and highlighted in the input file, emphasizing the changes introduced by the user. However, the input files can also be changed without the visualization of the input ANSI ASCII format. The possibility to visualize the default input files was introduced mainly to highlight the amount of different parameters that can be controlled by the user.

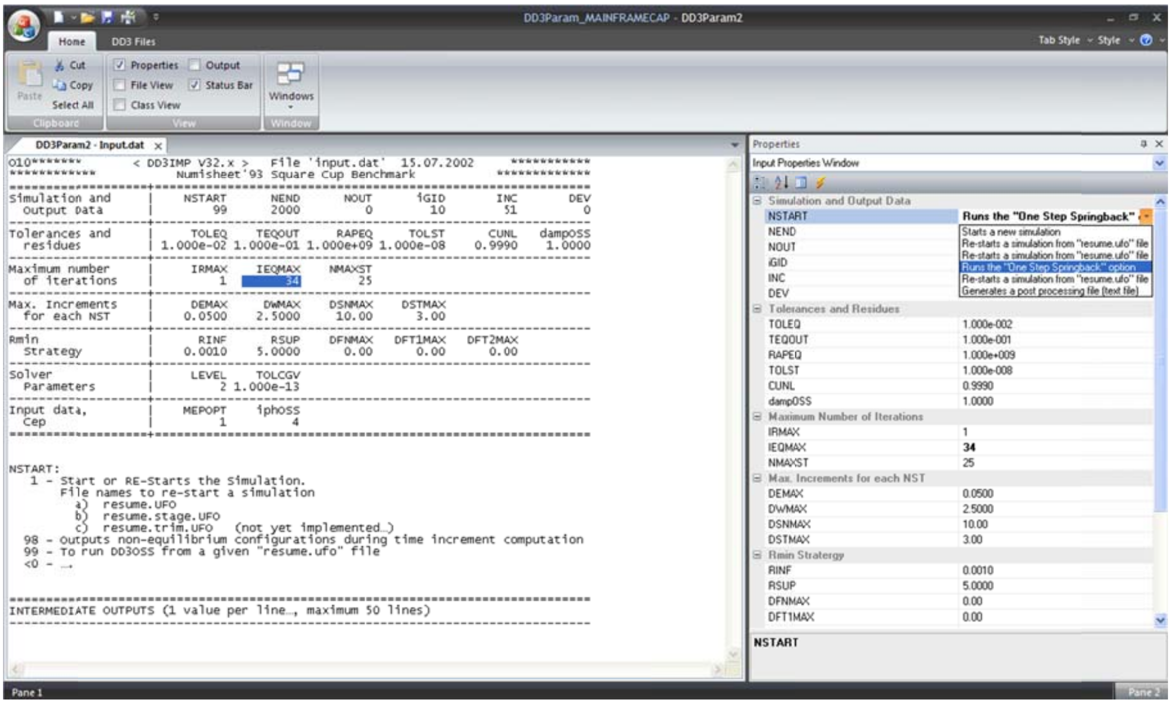


Figure 4. Example of interactive control of the DD3_input file through the DD3LT application.

2.3. Benchmark Database

As previously mentioned, the interactive application works based on the selection of a previously defined model. These models are stored in a database that was built using Microsoft Access©. The interactive application accesses the database through the Microsoft Access Driver - Aceodbc.dll, present in Microsoft Access Database Engine 2010 Redistributable.

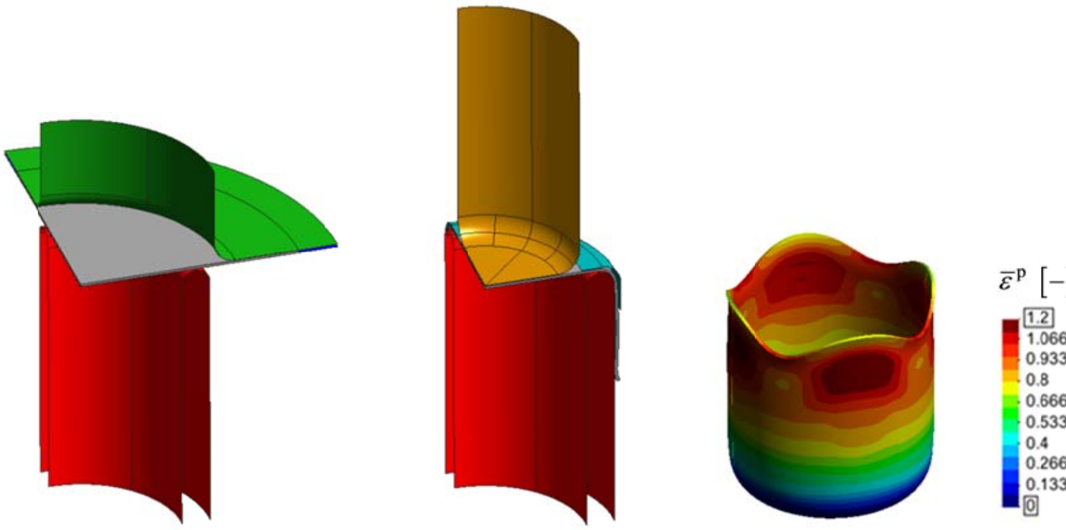
The database was built in order to store the data from previous models in an organized structure. Therefore, it enables the access to the input files of previous models, minimizing the preprocessing work and helping to keep track of the parameters used in previous simulations. The structure selected for the developed database is relational, with the following order:

- Tools geometry: this is the first parameter to select, because it is related with the process conditions (i.e. number and type of phases) and contact conditions (i.e. contact sets). Typically, this selection also dictates the boundary conditions, although the complete tool geometry can be used to model only half or a quarter of the formed component;
- Material: this selection includes not only the material but also the constitutive model (i.e. work hardening law and yield criteria) adopted;
- Blank geometry: this selection is based on the main geometrical characteristics of the blank, i.e. square, rectangular and circular. Based on this initial selection different types of in-plane and through thickness discretizations can be selected. In

fact, the interactive platform enables the scaling of the blank discretizations stored in the database;

The examples available in the database can be directly executed with the aid of the interactive application. The database was fed with a large set of default examples, which include many benchmarks proposed by the NUMISHEET conferences series. Some of the benchmarks included are the:

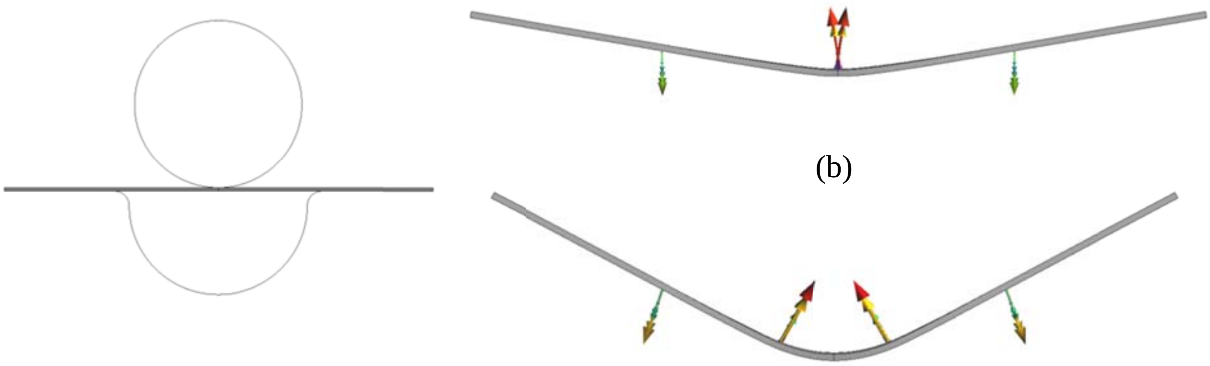
- NUMISHEET'99 Reverse Deep Drawing: the aim of this test was to evaluate the solvers capability to predict thickness evolution along the cup wall in processes involving strain-path changes [8]. This example involves two stages, as shown in Figure 5, being the punch of the first stage used as die for the second stage. In both stages the blankholder is controlled by imposed displacement. In fact, one of the parameters to be determined was the gap between the die and the blankholder, which is sufficiently small to allow the material flow but large enough to avoid wrinkles. This example, as any cylindrical geometry is also interesting to analyze the influence of the material orthotropic behavior, as shown in Figure 5 (c). This figure also highlights the influence of the strain-path changes induced by the reverse deep drawing in the equivalent plastic strain distribution.



(a) (b) (c)

Figure 5. NUMISHEET'99 Reverse deep drawing: (a) tools geometry and initial blank for the first stage; (b) tools geometry and initial shape for the second stage and (c) equivalent plastic strain distribution on the predicted final shape.

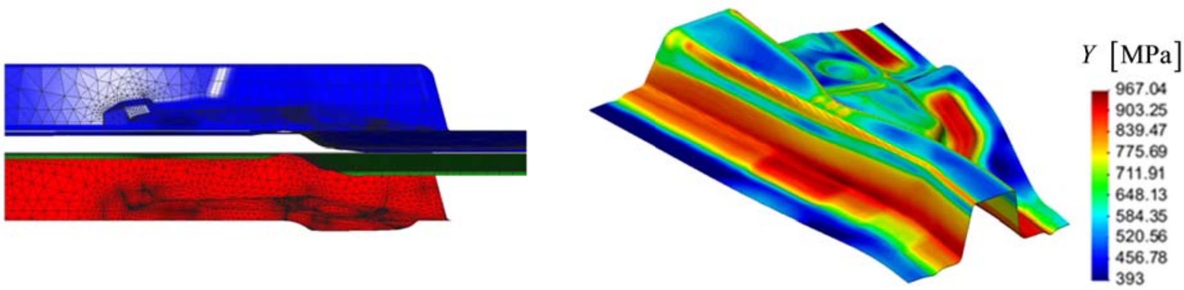
- NUMISHEET'02 Unconstrained Cylindrical Bending: the aim of this test was to evaluate the solvers capability to predict contact conditions evolution and springback [16]. The test consists in bending a rectangular blank with a cylindrical punch and die, as shown in Figure 6. This example is quite sensitive to the blank discretization adopted as well as the type of finite element and integration. All these parameters are known to influence the springback prediction. In this example, the in-plane discretization of the rectangular blank will also influence the prediction of the punch displacement for which the contact changes from one point to two points, as shown in Figure 6 (b) and (c), respectively. The displacement stroke corresponding to the contact bifurcation is influenced by the blank discretization.



(a) (c)

Figure 6. NUMISHEET’02 Unconstrained cylindrical bending: (a) tools geometry and initial blank; (b) continuous contact conditions (c) two point contact conditions.

- NUMISHEET’05 Automotive Underbody Cross Member: the aim of this test was to evaluate the solvers capability to accurately predict springback in forming processes where bending effects play a dominant role [4]. This example highlights some important features present in real industrial components. Figure 7 (a) presents the tools geometry and the blank positioning, highlighting the problems associated to contact detection for non-flat tool surfaces. In this case, the initial contact depends on the blank discretization adopted. Thus, finite element numerical details that can be easy analyzed in simple problems, such as the unconstrained cylindrical bending, can be extended to real industrial problems. Also, this example highlights the influence of the drawbead geometry in controlling the material flow and the material hardening, as shown in Figure 7 (b).

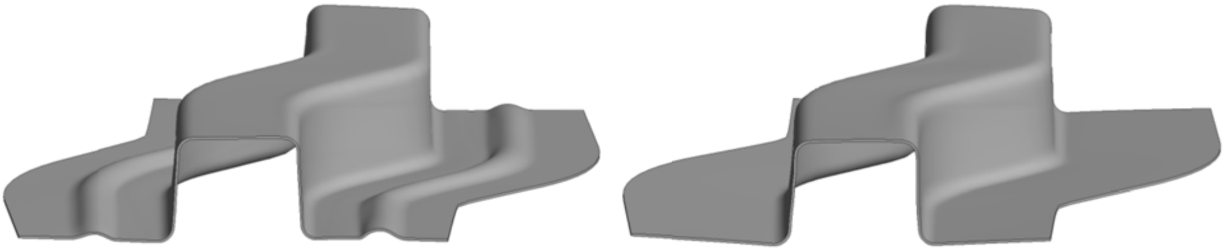


(a) (b)

Figure 7. NUMISHEET’05 Automotive Underbody Cross Member: (a) tools geometry and initial blank and (b) flow stress distribution on the predicted final shape.

- NUMISHEET’08 Influence of Drawbeads on the Springback Behaviour («S-Rail-08»): the aim of this test was to evaluate the solvers capability to accurately predict drawing and springback for models with different drawbead scenarios. An S-rail tool was developed in order to be able to adapt smooth and locking drawbeads

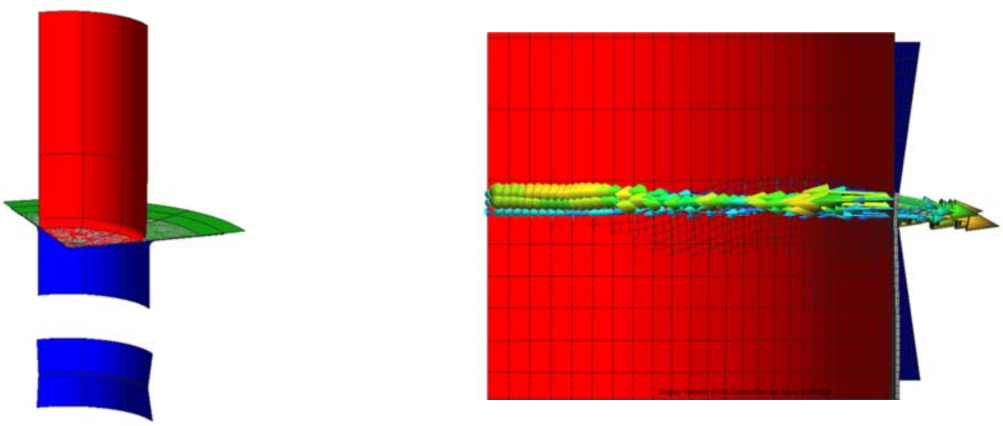
[29]. The same component can be modeled with or without drawbead, using the same blank holder, leading to different springback predictions, as shown in Figure 8. This component is a good example to analyze the influence of the contact conditions on springback, by studying different blankholder forces and friction coefficient distributions. It is also a good example to analyze the presence of surface defects, as it is also shown in the example presented in Figure 8.



(a) (b)

Figure 8. NUMISHEET'08 Influence of Drawbeads on the Springback Behaviour («S-Rail-08»), blankholder force of 400 kN: (a) final shape with smooth drawbead and (b) without drawbead.

- NUMISHEET'11 Earing Evolution During Drawing and Ironing Processes: the aim of this test was to evaluate the solvers capability to accurately predict earing evolution during drawing and ironing for advanced material modeling. A special die which involves only drawing and ironing within one punch stroke was designed to simplify real processes [9], as shown in Figure 9 (a). The ironing operation, which consists in wall thinning, is known by contributing to the earing phenomenon reduction, allowing a more uniform wall thickness of the component as well as increased cup height results. The blank sheet discretization using 3D solid elements allows an accurate contact description on both sides of the blank, as shown in Figure 9 (b). This is a good example to analyze the influence of the yield criterion selected in the earing profile as well as lubricant conditions, particularly in the ironing process.



(a) (b)

Figure 9. NUMISHEET'11 Earing Evolution During Drawing and Ironing Processes: (a) tools geometry and initial blank and (b) detail of the contact force vectors in the ironing process.

3. CONCLUSIONS

Generally the accuracy requirements have increased with the usage of the simulation tool, promoting the continuous development of FEA solvers for sheet metal forming processes. However, the quality of the results depends not only on the FEA solver but also on the appropriate training of the CAE engineers. The DD3LT platform aims to help the training of sheet metal forming CAE engineers, providing in an interactive form, a large range of different problems, which can be used to explore the influence of many different process parameters.

Acknowledgements

This work was co-financed by the Portuguese Foundation for Science and Technology (FCT) via project PTDC/EME-TME/103350/2008 and by FEDER via the "Programa Operacional Factores de Competitividade" of QREN with COMPETE reference: FCOMP-01-0124-FEDER-010301.

3. REFERENCES

[1] Alves J.L., Menezes L.F., "Application of tri-linear and tri-quadratic 3-D solid finite elements in sheet metal forming simulations". *Proceedings of the Numiform'01 on Simulation of materials processing: Theory, Methods and Applications*, Mori K.-I. (eds.), Balkema, Rotterdam, 639–644, 2001.

[2] Areias P.M.A., César de Sá, J.M.A., Conceição António C.A., Fernandes A.A., "Analysis of 3D problems using a new enhanced strain hexahedral element". *Int. J. Numer. Meth. Eng.* 58 (11), 1382–1637, 2003.

[3] Bartlat F., Lege D.J., Brem, J.C., "A six-component yield function for anisotropic materials". *Int. J. Plasticity*. 7, 693-712, 1991.

[4] Buranathini T., Cao J., "NUMISHEET'05 Automotive Underbody Cross Member". *NUMISHEET'2005, 6th International Conference and Workshop on Numerical Simulation of 3D Sheet Forming Processes - On the Cutting Edge of Technology*, Cao J., Shi M.F., Stoughton T.B., Wang C.T., Zhang L. (Eds.), American Institute of Physics Conference Proceedings, Part B, 1121-1133, 2005.

[5] Cazacu O., Barlat F., "Generalization of Drucker's yield criterion to orthotropy". *Math. Mech. Solids*. 6, 613-630, 2001.

[6] Cazacu O., Plunkett B., Barlat F., "Orthotropic yield criterion for hexagonal closed packet metals". *Int. J. Plasticity*. 22, 1171-1194, 2006.

[7] Chung K., Kuwabara T., Verma R.K., Park T., Huh H., Bae G., "BM 4 - Pre-strain Effect on Spring-back of 2-D Draw Bending". *NUMISHEET 2011, The NUMISHEET 2011 Benchmark Study of the 8th International Conference and Workshop on Numerical Simulation of 3D Sheet Metal Forming Processes*, Huh H., Chung K., Han S.S., Chung W.J. (eds.), 171-209, 2011.

[8] Danckert J., Nielsen K.B., Højberg, "NUMISHEET'99 Benchmark C: Reverse Deep Drawing of a Cylindrical Cup". *NUMISHEET'99, 4th International Conference and Work-*

shop on Numerical Simulation of 3D Sheet Forming Processes, Gelin J.C., Picart P. (Eds.), University of France Comté and E.N.S.M.M., BURS (Eds.), Besançon, 871-928, 1999

[9] Dick R.E., Yoon J.W., Huh H., Bae G, "BM1 - Earing Evolution during Drawing and Ironing Processes". NUMISHEET'11, The NUMISHEET 2011 Benchmark Study of the 8th International Conference and Workshop on Numerical Simulation of 3D Sheet Metal Forming Processes, Huh H., Chung K., Han S.S., Chung W.J. (eds.), 11-44, 2011.

[10] Drucker D.C., "Relation of experiments to mathematical theories of plasticity". *J. App. Mech.* 16, 349-357, 1949.

[11] Hill R., "A theory of yielding and plastic flow of anisotropic materials". *Proceedings of Mathematical, Physics and Engineering Science,* Royal Society of London, A193, 1948.

[12] Hosford W.F., "A generalized isotropic yield criteria". *J. Appl. Mech.* 39, 607-609, 1972.

[13] Hughes T.J.R., "Generalization of selective integration procedures to anisotropic and nonlinear". *Int. J. Numer. Meth. Eng.* 15, 1413–1418, 1980.

[14] Jiao Z.-P., Li C., "A new formulation of eight-node hexagonal solid element". *Comput. Meth. Appl. Mech. Eng.*, 187 (1–2), 213–217, 2000.

[15] Karafilis A.P., Boyce M.C., "A general anisotropic yield criterion using bounds and a transformation weighting tensor". *J. Mech. Phys. Solids*. 41, 1859-1886, 1993.

[16] Kim H.J., "NUMISHEET'02 Benchmark Test B - Unconstrained Cylindrical Bending". *NUMISHEET'2002, 5th International Conference and Workshop on Numerical Simulation of 3D Sheet Forming Processes - Verification of Simulation with Experiment*, Yang D.Y., Oh S.Y, Huh H. Kim Y.H. (eds.), 2, 757-811, 2002

[17] Lemaître J., Chaboche J-L., "Mechanics of solid materials", *Cambridge University Press*, Cambridge, Reino Unido, 1985.

[18] Li K.P., Carden W.P., Wagoner R.H., "Simulation of springback". *Int. J. Mech. Sci.* 44 (1), 103–122, 2002.

[19] Lingbeek R.A., "Aspects of a Design tool for Springback Compensation". *Msc. Thesis*, University of Twente, Netherlands, 2003.

[20] Mc-Meeking R.M., Rice J.R., "Finite element formulations for problems of large elastic-plastic deformation", *Int. J. Solids Struct.* 11, 601-616, 1975.

[21] Menezes L.F., Teodosiu C., "Three-dimensional numerical simulation of the deep-drawing process using solid finite element". *J. Mater. Process. Tech.* 97, 100-106, 2000.

[22] Menezes L.F., Teodosiu C., Makinouchi A., "3-D solid elasto-plastic elements for simulating sheet metal forming processes by the finite element method". *Proceedings FE-simulation of 3-D Sheet Metal Forming Processes in Automotive Industry,* VDI Berichte (eds.), 864, 381–403, 1991.

[23] Neto D.M., Oliveira M.C., Alves J.L., Menezes L.F., "FEA of frictional contact problems using Nagata patches for surfaces description", WCCM 2012, 10th World Congress on Computational Mechanics, 8-13 July, S. Paulo, Brazil, (accepted to be presented), 2012.

[24] Oliveira M.C., Alves J.L., Menezes L.F., "Algorithms and Strategies for Treatment of Large Deformation Frictional Contact in the Numerical Simulation of Deep Drawing Process". *Arch. Comput. Meth. Eng.* 15, 113-162, 2008.

[25] Oliveira M.C., Alves J.L., Menezes L.F., "Improvement of a frictional contact algorithm for strongly curved contact problems". *Int. J. Numer. Meth. Eng.* 58, 2083-2101,

2003.
[26] Prager W., “The theory of Plasticity: a survey of recent achievements”. *Proc. Inst. Mech. Eng.*, 169, 41-57, 1955.
[27] Reese S., “On a physically stabilized one point finite element formulation for three-dimensional finite elastoplasticity”. *Comput. Meth. Appl. Mech. Eng.* 194 (45–47), 4685–4715, 2005.
[28] Roll K., Rohleder M., “Complex testing tool for the investigation of springback deviations”. *NUMISHEET 2002*, *The 5th International Conference and Workshop on Numerical Simulation of 3D Sheet Metal Forming Processes*, Yang D.Y., Oh S.I., Huh H., Kim Y.H. (eds.), 131-136, (2002).
[29] Roll K., Wiegand K., Hora P., “Benchmark Study Report - Benchmark 2 - Influence of drawbeads on the springback behaviour (“S-RAIL-08”)”. *NUMISHEET’08, 7th International Conference and Workshop on Numerical Simulation of 3D Sheet Metal Forming Processes - Part B*, Hora P., Volk W., Roll K., Griesbach B., Kessler L., Hotz W. (eds.), 58-112, 2008.
[30] Tekkaya A.E., “State-of-the-art of simulation of sheet metal forming”. *J. Mater. Process. Tech.* 103, 14–22, 2000.
[31] Teodosiu C., Hu Z., “Microstructure in the continuum modelling of plastic anitropy”, *19th Riso International Symposium on Materials Science: Modelling of Structure and Mechanics of Materials from Microscale to Products*, Riso National Laboratory, 149-168, 1998.
[32] Tisza M., “Numerical modelling and simulation in sheet metal forming”. *J. Mater. Process. Tech.* 151 (1–3), 58–62, 2004.
[33] von Mises R., “Mechanik der festen Körper im plastisch deformablen Zustand”. *Göttin. Nachr. Math. Phys.* 1, 582–592, 1913.
[34] Wang J., Wagoner R.H., “A practical large-strain solid finite element for sheet forming”. *Int. J. Numer. Meth. Eng.* 63 (4), 473–501, 2005.
[35] Yamada Y., Yoshimur N., Sakurai T., “Plastic stress–strain matrix and its application for the solution of elastic–plastic problems by the finite element method”. *Int. J. Mech. Sci.* 10 (5), 348–354, 1968.