SHEET METAL FORMING OPTIMIZATION USING ANSA AND LS-DYNA

[1]Simone Ferrero*, [1]Caterina Tribuzi, [1]Fabio DiNatale
[1]Nova Analysis, Italy

KEYWORDS –
Metalforming, Anisotropy, Optimization, LS-Dyna

ABSTRACT

Setup of a LS-Dyna four components sheet metal forming benchmark was conducted using ANSA. Tools for meshing and process setup, like material constituting the elements, the output to be determined or the way contacts have to be treated, were exploited. With the purpose of matching the experimental data provided by the benchmark, the file implementing sheet metal forming problem was conveniently arranged to run in LS–Opt for optimizing Lankford's parameters or R–Values, which define the anisotropy of the blank in the material model employed in the simulations, and static friction coefficient in contact definition.

1. INTRODUCTION

Forming applications are processes allowing to produce an object of specific shape starting from a flat sheet. The elements characterising metal forming are numerous and one of the most important is surely the material model.

In this work, the purpose is to optimize specific parameters of a precise material model in order to match the results of a simulated metal forming process with provided experimental data. To achieve this goal, a procedure was developed, consisting of the following steps:

1. Choosing an appropriate material model. In particular, as suggested by bibliography, it was used *MAT_36 or *MAT_3-PARAMETER_BARLAT, which implements in LS–Dyna the Barlat and Lian's model. It allows the user to simulate the usual anisotropic behaviour of sheet metal, defined through Lankford's parameters or R–values;

2. To verify the effectiveness of this model, a virtual tensile test was implemented and performed. Then, a metal forming application, whose results concerning given physical quantities were available, was modelled and simulated, adopting for the parameters of interest given values;

3. An optimization problem was formulated using LS–Opt. Its objective consisted in identifying the values of Lankford's parameters which allow to simulate the same sheet metal forming application, but producing outcomes that better match the experimental ones. Also friction coefficient in contacts definition was investigated, in order to better match experimental results.

Each step of model setup both in metal forming phase and in optimization phase was developed using ANSA pre-processor. This report highlights the main steps of model setup and obtained results.

2. OVERVIEW

A metal forming process is a set of mechanical procedures by which a strip or a blank is deformed into an object of specific shape, whether plane or hollow, in one or more steps. Because of the plastic deformation the sheet is subjected to, the acquired configuration is permanently maintained. In particular, the drawing of metal or deep drawing is the process in which a flat sheet of metal is formed into a cylindrical–, conic– or box–shaped part. The final workpiece has to be achieved using minimal operations and generating minimal scrap, meeting anyway definite quality requirements.

The typical machinery involved in an usual sheet metal forming procedure is constituted by the following tools:

- **Die**: it represents the base on which the blank is initially placed. The die is characterized by a cavity whose contour reproduces the profile of the final workpiece. Its depth is related to the height the drawn piece has to achieve;
- **Punch**: it is the tool by which the blank is forced to flow into the die cavity. The punch is designed in a way that allows to obtain the expected form at each stage of the process. The face of the punch can not be flat, depending on the forming process and on the final mould. If the difference between die and punch radii is less than the blank thickness, then the foil shrinks (ironing);
- **Binder/Blankholder**: it is usually positioned over the blank with the aim of slowing down its flux into the die. The force applied by the binder is called "Holding force", thus distinguishing it from the "Drawing force", exercised by the punch. This tool is notably useful in order to avoid blemishes on the workpiece borders.
- **Drawbeads** can serve the purpose of controlling or reducing the blank flow. In the first circumstance, they are more properly called "draw rings", since they form a jutting out band along the die border. Drawbeads are designed to keep away from flaws the final workpiece. In this slant, the utilization of drawbeads allows to diminish the blankholder force acting on the sheet;

This study exploits the capability of LS-Dyna to manage metal forming problems. Case setup in LS-Dyna is made by keword file that lists every card representing a particular mathematical model of a physical quantity (contacts, materials, restraints).
Clearly, for complex problems like forming applications it is not conceivable writing manually all the keywords necessary to implement the process. It is usual to take advantage of pre–processor software, such as ANSA or LS–PrePost. These programs allow the user to set–up the process in a relatively easy way through their graphical interface and produce as output the file containing all the information in the format required by the solver.

3. VIRTUAL TENSILE TEST

The first step of this analysis was the setup of a virtual traction test in order to explore LS-Dyna material model capabilities to represent the behaviour for anisotropic materials. The tensile test is conducted according to DIN EN 10002 norm, while the geometry of the specimen is determined according to DIN50125 norm (**Fig. 1**):

• Total length: 245.28mm;

• Gauge length: L0 = 120mm.

• Gauge width: B0 = 21mm;

• Raccording radius: R = 7mm.

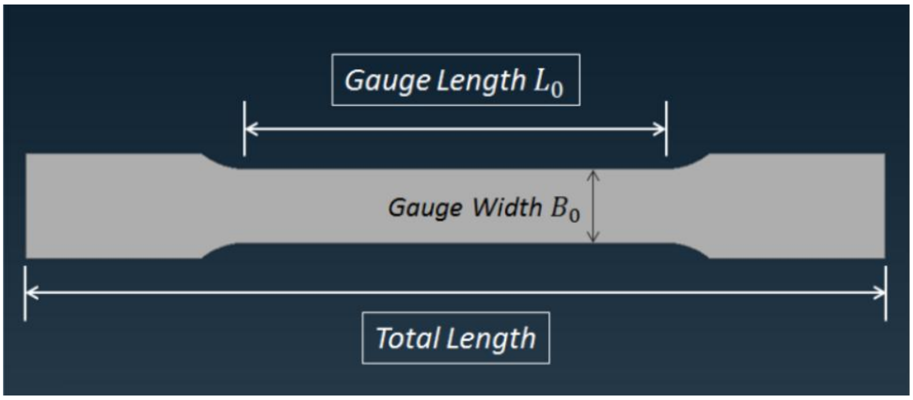


Fig. 1. Specimen dimensions

The material constituting the specimen is steel. Its main mechanical characteristics are resumed in Table 1, which comprehends also R–values.

Density r (kg/mm3)	Young's Modulus E (GPa)	Poisson's Coefficient n	R0	R45	R90
7.85e-6	210	0.3	1.7	1.3	1.9

Table 1. Summary of the mechanical properties of the steel constituting the specimen

All the features describing the tensile test are encoded in LS–Dyna language through the pre–processor ANSA, that elaborates the input data for producing the appropriate output file, in this case a ".k" or ".key" file, which will be processed by the solver.

The first step consists in creating an opportune mesh on the CAD model of the specimen. In simulation, only Mid surface of the specimen was used. Its shape, sufficiently regular, allows to use the MAP algorithm, thus creating a mesh constituted only by QUAD elements (**Fig. 2**).

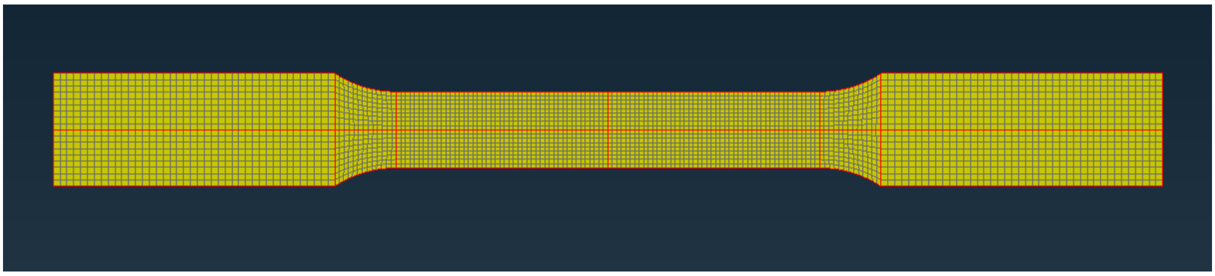


Fig. 2. Specimen mesh

The gripping of real tensile testing machine on the specimen was modelled using the *BOUNDARY_SPC_SET keyword, while the tension exerted by the machinery itself on the specimen was modelled using the *BOUNDARY_PRESCRIBED_MOTION_SET keyword. Both keywords are available inside ANSA in the LS-Dyna deck inside the BOUNDARY section.

The linear translation imposed to the specimen is defined through a curve using the *DEFINE_CURVE keyword, that can be directly recalled from the *BOUNDARY_PRESCRIBED_MOTION_SET window.

In order to reproduce material behaviour, *MAT_36 (Barlat and Lian model) keyword is employed, with the parameters listed in Table 1 and the Stress vs. Strain curves specified for each rolling direction, using curves provided in (4). The ANSA control panel of the keyword with all the fields filled with the appropriate values is displayed in **Fig. 3**

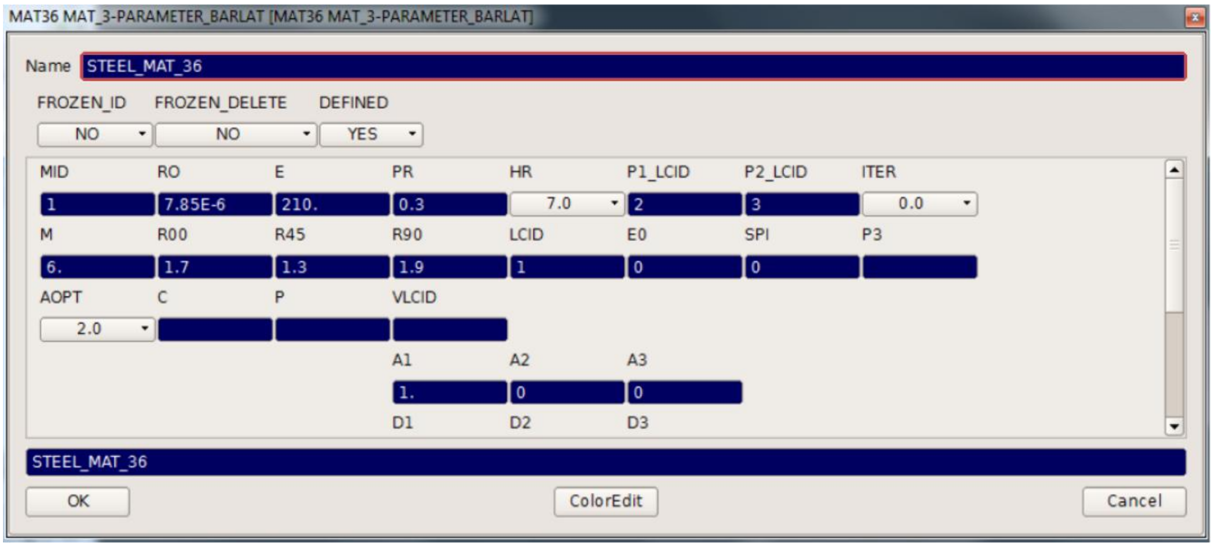


Fig. 3. ANSA Material definition panel

Correctly defining the AOPT parameter in the material card, that permits to change the referential rolling direction of the material, the simulated hardening laws in 0, 45 and 90 degrees directions can be easily determined with three tests.

Finally, some controls on the simulation have to be defined, such as *CONTROL_HOURGLASS, which prevents that in the simulation verifies “hourglassing“, i.e. anomalous deformations of the elements constituting the mesh. Another important keyword is *CONTROL_SHELL, since the mesh is composed by “shell“ elements. It allows defining the theory describing shell behaviour or how thickness changes have to be simulated. Also *CONTROL_TERMINATION plays an important role, since it establishes when the solver has to stop calculations. In particular, the field ENDTIM is set equal to 200ms.

In order to get physical quantities for the post processing phase, also the following keyword were set up in ANSA:

*DATABASE_BINARY_D3PLOT: contains the information about thickness or stresses on each shell element;

*DATABASE_CROSS_SECTION_SET: this keyword was used to register the forces acting on nodes lying in the middle of the specimen;

*DATABASE_HISTORY_NODE_SET: two nodes at the ends of the gauge section are observed, thus capturing the relative displacement to be used in the determination of true strains;

Results

The diagrams of the output and input curves of Stress-Strain along the three rolling directions, respectively displayed with continuous lines and points, are shown in **Fig. 4**.

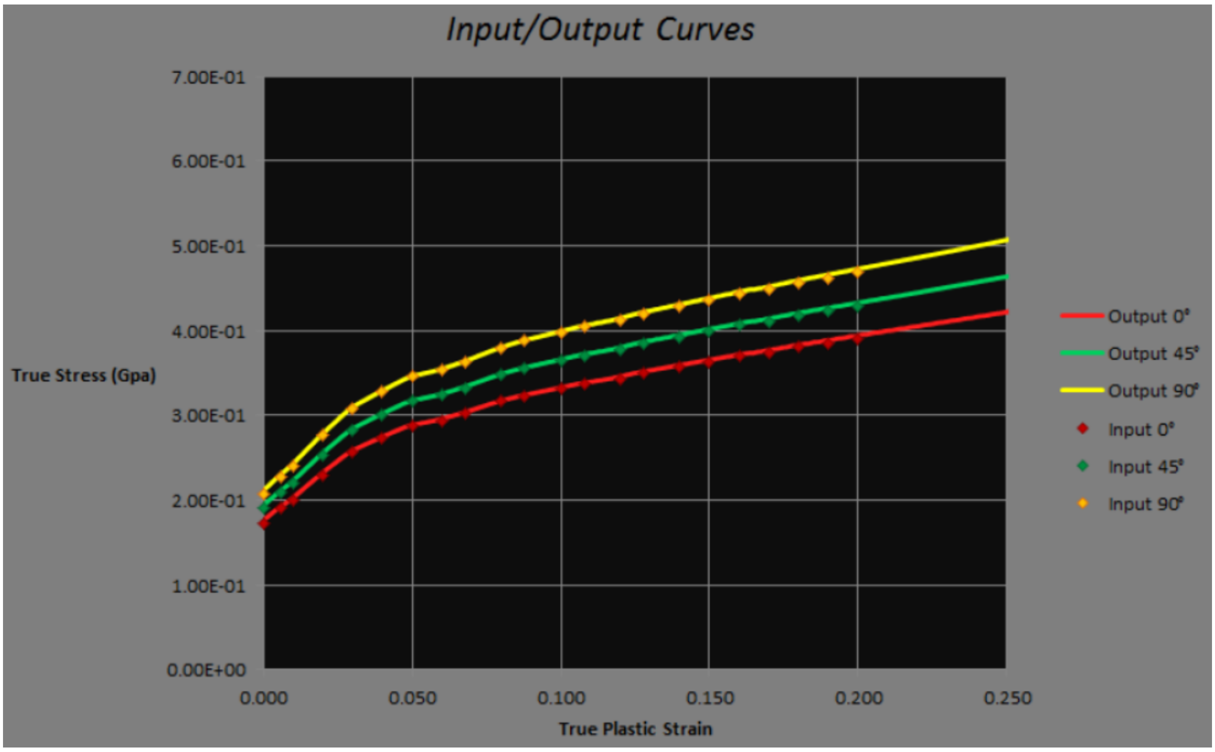


Fig. 4. True Stress vs. True Plastic Strain Input/Output Curves

This result confirms that the employed mathematical model is able to reproduce real behaviour of the material.

The development of the process is reported by means of **Fig. 5**.

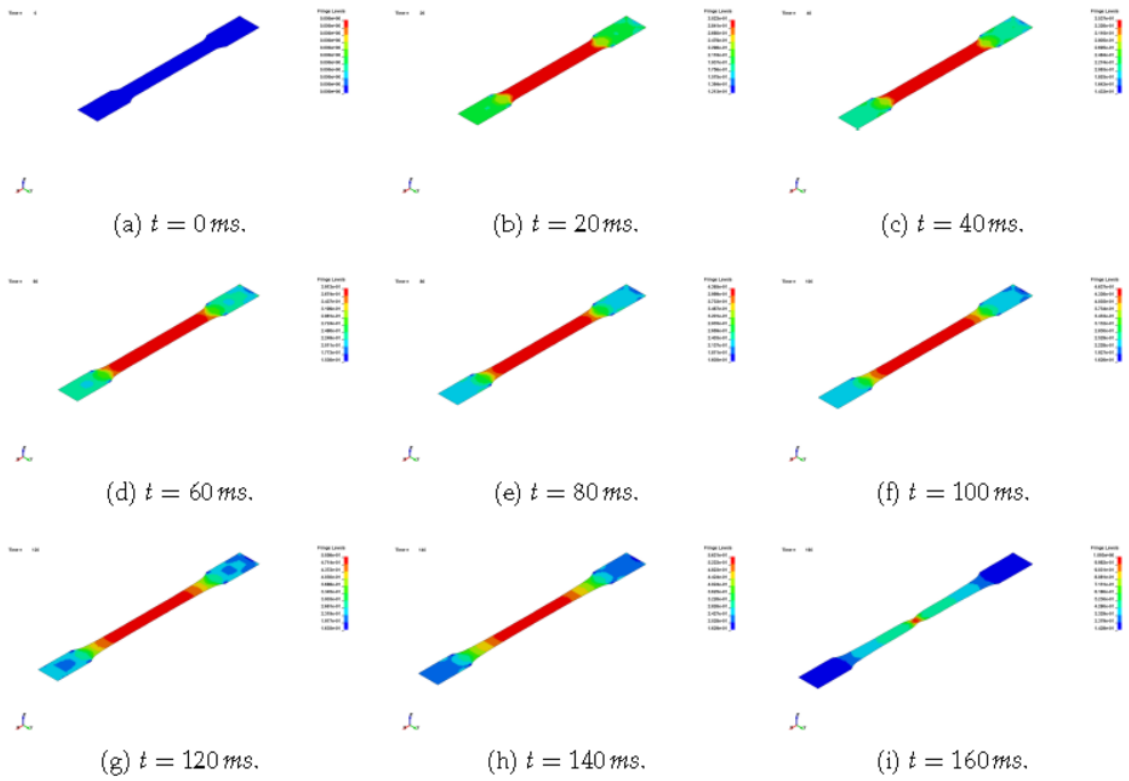
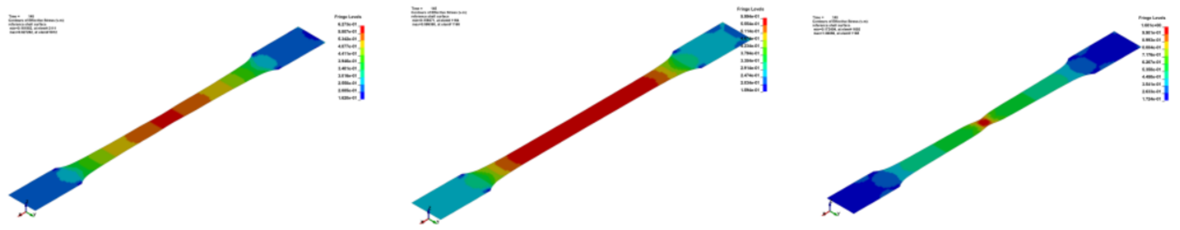


Fig. 5. Von Mises Stress on the specimen during tensile test

Particularly, the captured instants refer to the virtual tensile test in which a = (1, 0, 0), that is the rolling direction coincides with x–axis. The physical quantity represented on the specimen is the Von Mises stress, which assumes its maximum values in the central part of gauge section, since it's the zone where the greatest forces develop.



(a) Von Mises Stress for $t = 145\,ms$ when $\mathbf{a} = (1, 0, 0)$.

(b) Von Mises Stress for $t = 145\,ms$ when $\mathbf{a} = (1, 1, 0)$.

(c) Von Mises Stress for $t = 145\,ms$ when $\mathbf{a} = (0, 0, 1)$.

Fig. 6. Different specimen deformations due to different rolling directions

Fig. 6 illustrates for the same time–step (t = 145ms) the trend of Von Mises stress on the specimen. As expected, changing the rolling direction of the material entails drastic variations on its behaviour when an uniaxial force acts on it. Particularly, when the rolling direction is perpendicular to the direction of the force, i.e. when the rolling direction is orthogonal to x–axis, figure (c), the specimen collapses earlier, as can be seen from its necking, which is almost finished when in the other two tests are in their initial phases.

4. DECKLID INNER PANEL DRAWING SIMULATION

<u>Metal forming process</u>

The considered problem is the Benchmark 1 proposed at the 6th Numisheet Conference in 2005, in which the forming of a decklid inner panel has to be carried out.

The utilised forming scheme considers a sequence of machines in which the die is placed at the pinnacle, the punch is the lowest apparatus, while the binder, supported by hydraulic cylinders, lies in the middle. The punch is stationary, and participates in deforming the sheet metal only in the last moments of the drawing procedure. The tools moving and originating major stresses and deformations on the blank are the die and the blankholder.

The displacement takes place exclusively along z–axis and has a total duration of approximately 150ms. It can be splitted in two parts: in the first one, only die begins to move, thus commencing deformation process, in particular on blank's borders. In the second part, definitely briefer, the blank is clutched between die and blankholder and, through their coordinated motion, continues to be deformed.

The holding force exercised by the blankholder is equal to 1334 kN. This force begins to act only in the second step of the drawing process, when the blank is tight between die and binder. To avoid contingent wrinkling, the force is applied through drawbeads.

The sheet metal used in this process has a trapezoidal form. With the objective of simplifying the procedure, the blank is not flat, but it is pre–bent, thus allowing a better positioning between the tools. The geometry, i.e. the CAD model of the blank, is provided.

The material considered for simulation is BH180 Steel. For this material, besides the main mechanical properties such as density ρ or Young's Modulus E, also the Stress vs. Strain curve is given. In particular, for describing the usual sheet metal anisotropy, the benchmark provides three curves for each of the rolling directions, i.e. 0, 45 and 90 degrees (labelled respectively with L, D and T).

Setup

The procedure for simulating this deep drawing application starts when the CAD data are read, i.e. the virtual models of blank and machinery are opened in ANSA. When the working space comprises all the elements essential for deep drawing it is necessary to correctly arrange the components: the imported geometries overlap, but, taking advantage of the command “Translate“ in “TRANSF.“ panel, the elements can be settled in their proper locations.

The blank needs to be rotated to be correctly oriented with respect to other tools; to achieve this result TRANSF. > MOVE > Transform functionality was used.

The CAD (Computer–Aided Design) model of these appliances is shown in **Fig. 7**; the blank is located between die and blankholder.

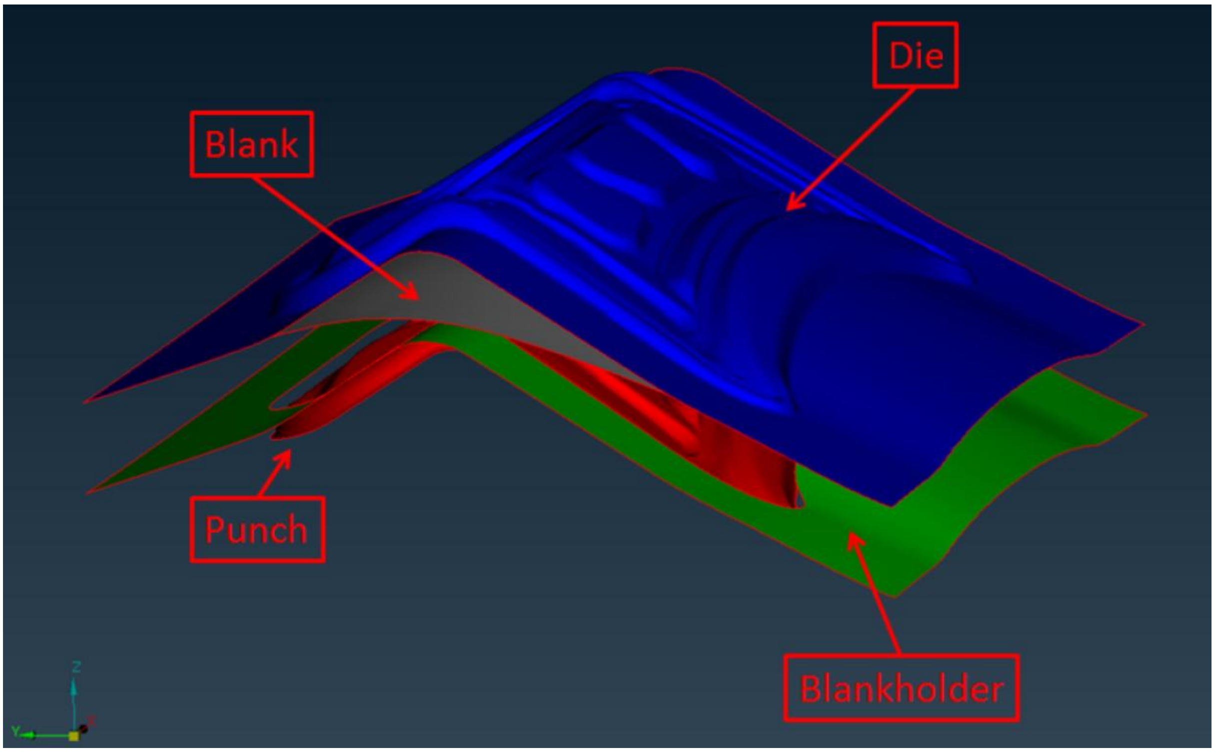


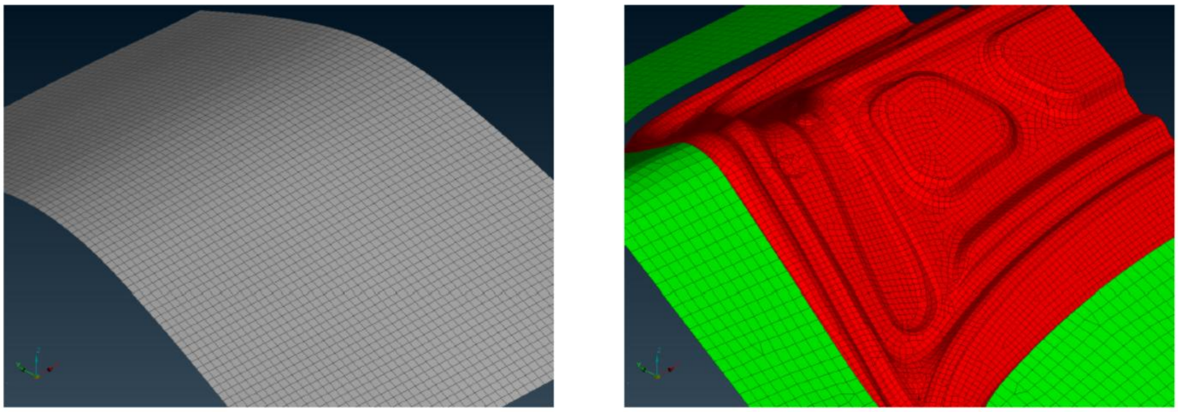
Fig. 7. Tools arrangement with the sheet metal

Even if it’s not particularly difficult to fulfil, this preliminary step takes on a crucial part, since erroneous relative positions of the tools can provoke failure of simulation runs or highly inaccurate results.

The mesh of the blank, which has a trapezoidal form, is generated using MAP algorithm, which creates exclusively quandrangular (QUAD) elements.

For tools, BEST algorithm was used, since their complex shapes require the use of triangular (TRIAS) elements.

The generated mesh has 27.684 QUAD elements and 1.754 TRIAS elements (29.438 total elements). **Fig. 8** (a) and (b) show the result obtained.



(a) Blank mesh generated with MAP algorithm.

(b) Portions of the mesh of punch and binder, obtained with BEST algorithm.

Fig. 8. Generated mesh for the forming process

Tools movement was imposed using the keyword *BOUNDARY_PRESCRIBED_MOTION_RIGID on die and blankholder, while punch remains stationary for all the process.

Motion is defined using load curves.

In the first stage of the process, only the die is in move. In few seconds, it reaches the assigned velocity, which is maintained until the last moments of the phase, when it resets. The values characterising this motion are reported in **Table 2. Motion curves in step 1 and 2**. This move stops acting at 140.5307 ms since TDEATH = 140.5307 ms is imposed in *BOUNDARY_PRESCRIBED_MOTION_RIGID.

	Time (ms)	**Velocity** (mm/ms)
Step 1	0	0
	1	1
	139.5307	1
	140.5307	0
Step 2	0	0
	1	5
	9.3138	5
	10.3138	0

Table 2. Motion curves in step 1 and 2

For making the blankholder stationary in this step on this tool is active a zero–velocity curve. In the second stage, die and blankholder come into contact. The behaviour is the same as in the first step, but times are scaled (using in the opportune *BOUNDARY_PRESCRIBED_MOTION_RIGID keyword TBIRTH = 140.5307ms) and velocity increases. Data are reported in **Table 2**. This motion dominates the zero–velocity curve imposed on the binder, thus even this tool starts moving.

Since the virtual model was halved, thus taking advantage of its symmetry, it's necessary to assign fitting boundary conditions in the cut areas for preventing erroneous behaviours of the objects, which may corrupt the simulated data.

The keyword *BOUNDARY_SPC_SET is used for achieving this purpose. It acts on a previously defined set of nodes on the blank: clipped border has the possibility of translating only along y and z axes and of rotating around x axis. In this manner, natural displacements, i.e. the ones that would be made if the entire model were considered, are perfectly reproduced.

The material constituting the blank is BH180 Steel, whose main properties are defined in **Table 3**:

Parameter (Symbol)	Value (Unit of Measure)
Density (ρ)	7.85 10^-6 kg/mm3
Young's Modulus (E)	210 GPa
Poisson's coefficient (ν)	0.3

Table 3. BH180 Steel parameters

Remaining significant features, e.g. Lankford's parameters ("R–values"), can be gained from benchmark documentation. Experimental "Stress vs. Strain" curves in three rolling directions (0, 45, 90 degrees) originate from different tensile tests are also available. These values are resumed in **Table 4**.

Lankford's Coefficient	Value
R0	1.604
R45	1.388
R90	1.991

Table 4. Lankford's coefficient of BH180 Steel

All these characteristics can be implemented using *MAT_36 or *MAT_3-PARAMETER_BARLAT keyword. The model developed in this keyword consents to simulate the anisotropy of the blank and also permits to describe hardening by means of three different curves in three rolling directions.

The material model used for tools is the one implemented in *MAT_20 or *MAT_RIGID keyword.

When simulating processes like sheet metal forming, contacts handling represents decidedly one of the most important feature.

LS–Dyna makes available a specific class of *CONTACT keywords expressly developed for metal forming applications. They are based on the "AUTOMATIC" type contact, which allows improving performances with respect to the traditional "TWO SURFACE" type contact. In this case, the *CONTACT_FORMING_ONE_WAY_SURFACE_TO_SURFACE keyword is employed. The contact is handled with the Penalty Based Method, by which an adequate force is assigned to the nodes of the blank moving towards tools mesh. This keyword only requires the correct orientation of the objects and it is recommended since better solutions may be obtained when also adaptive remeshing is applied.

It is necessary to define three distinct keywords for implementing the expected three contacts, that are “Die–Blank“, “Binder–Blank“ and “Punch–Blank“.

In the considered deep drawing application, it’s needed the adoption of Drawbeads, positioned, as is usual, on the blankholder surface. Their contribution is implemented through *CONTACT_DRAWBEAD keyword (this feature was not available inside ANSA).

Every control requested by LS-Dyna solver is imposed inside ANSA using the LS-Dyna Deck. In particular, the most relevant keywords used were:

*CONTROL_ADAPTIVE: at regular time intervals it refines the mesh where computed quantities show wide changes in their values, for the whole model or for specific parts (e.g. the blank in this deep drawing process).

*CONTROL_TERMINATION: allows to define the exact moment the simulation stops;

*CONTROL_TIMESTEP: allows to define “time–step“ setup using mass–scaling technique

*CONTROL_CONTACT: it permits to manage general contacts parameters, such as shell thickness or initial penetrations

*CONTROL_HOURGLASS, *CONTROL_BULK_VISCOSITY

*CONTROL_SHELL: shell elements properties

Other controls are employed (*CONTROL_PARALLEL, *CONTROL_OUTPUT, etc.) with default values, since their influence on the process is minimal.

Even post processing requests were defined through LS-Dyna cards available inside LS-Dyna deck in ANSA:

*DATABASE_BINARY_D3PLOT

*DATABASE_OPTION

*DATABASE_RCFORC

*DATABASE_BNDOUT

*DATABASE_HISTORY_OPTION

*DATABASE_NODAL_FORCE_GROUP

Results

Simulations run on 4 processors on an Intel I7 4core machine. Time requested to run the model was about 60 minutes.

Some instants of decklid deep-drawing process are shown in **Fig. 9**.

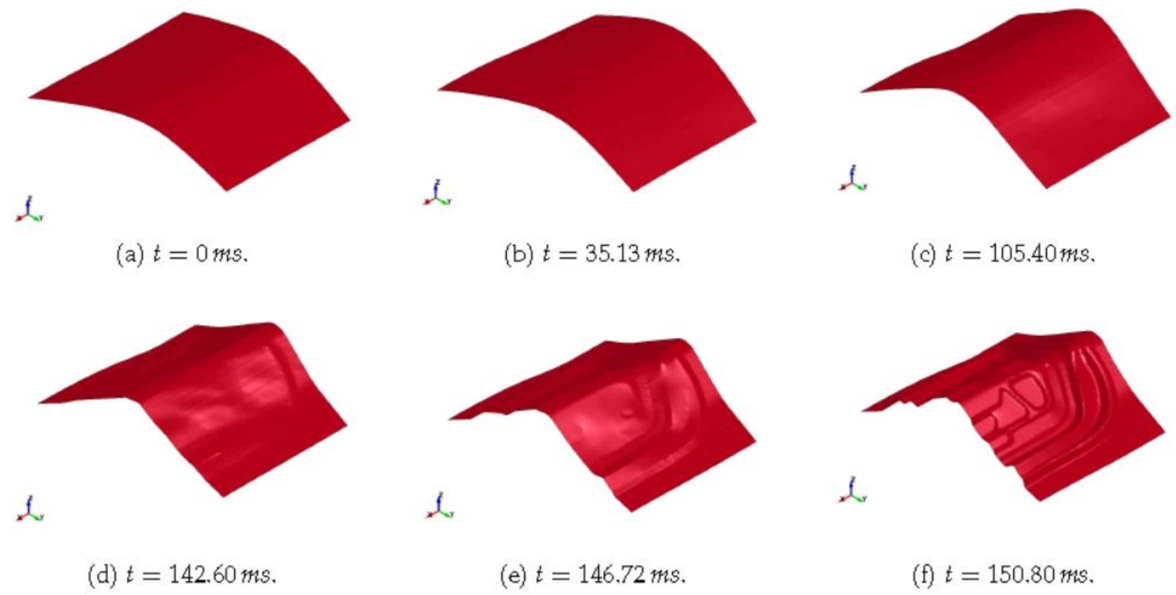


Fig. 9. Progressive decklid panel deep drawning

Adaptivity refinement on the blank is shown in **Fig. 10**.

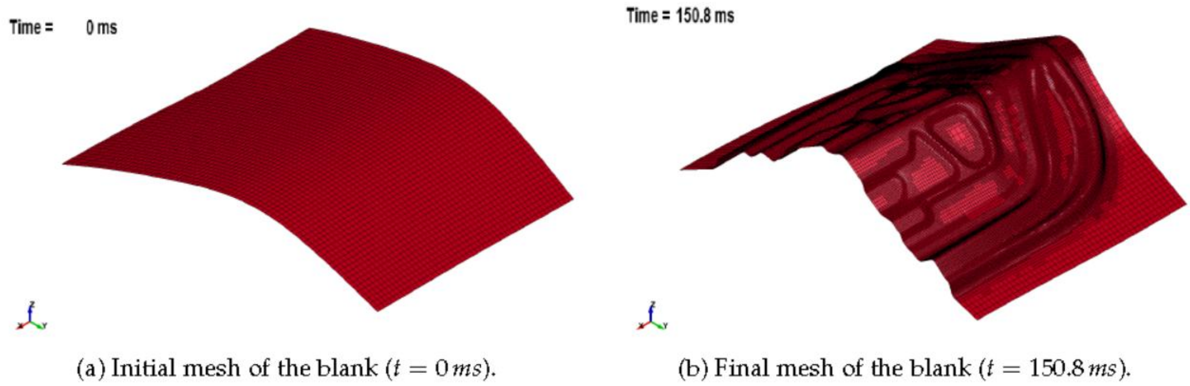


Fig. 10. Adaptivity on the blank mesh: while the border is unchanged, the internal part is extremely thicker

The patterns of thickness and of Von Mises Stress on the blank are selected and displayed in **Fig. 11** and **Fig. 12** respectively.

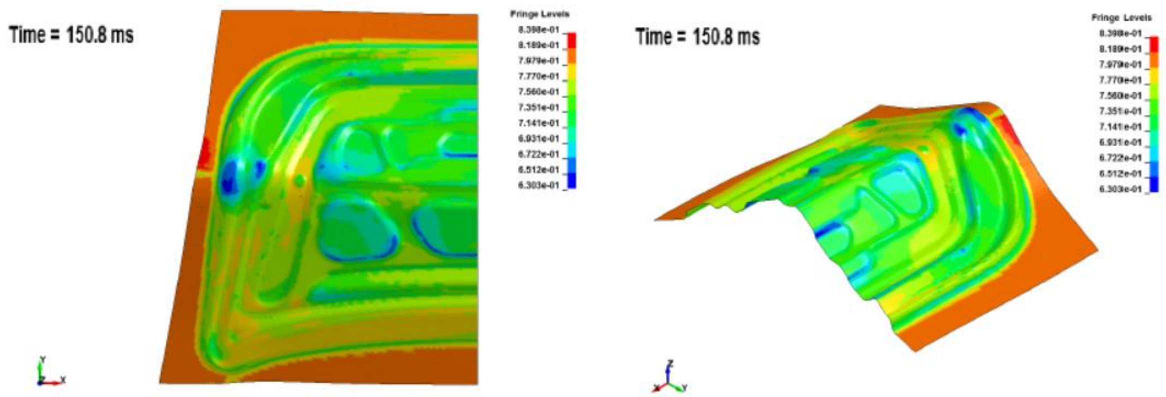
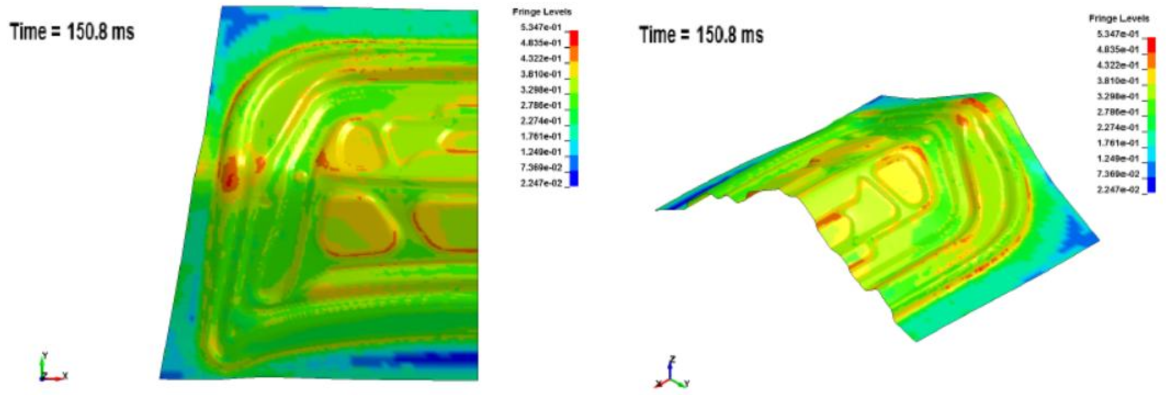


Fig. 11. Decklid thickness after forming



(c) Top view of the blank with Von Mises contours.

(d) Isometric view of the blank with Von Mises contours.

Fig. 12. Decklid Von Mises stress after forming

As expected, the thickness decreases till reaching minimum values of 0.65mm in the complex curved shapes of the blank, since here the sheet metal experiences the greatest deformations. This trend is confirmed by Von Mises Stress, which in these same zones assumes its highest values, approximately equal to 0.5 GPa, that is a reasonable and bearable value in this type of processes.

Benchmark documentation provides data about thickness values along specific sections. Particularly, for specific points of each section, their x and y coordinates in the global coordinate system are given together with the registered value of thickness at the end of the deep drawing.

The given sections are highlighted in red in **Fig. 13**.

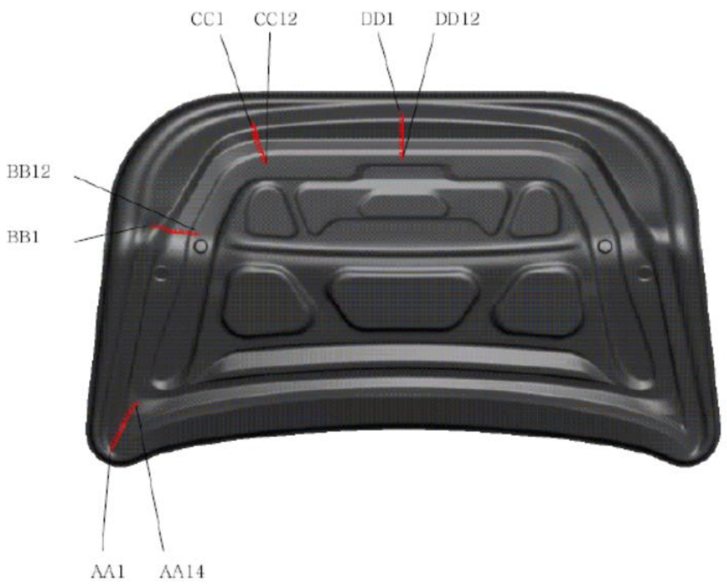


Fig. 13. Sections for which benchmark documentation provides thickness data

The results obtained in the simulation are compared to experimental data and plotted along sections B and C in **Fig. 14**.

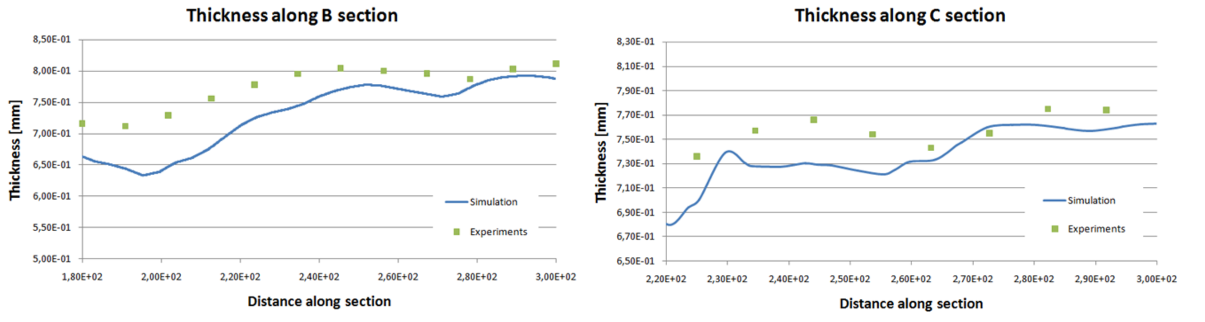


Fig. 14. Comparison of thickness values with experimental data

The curves show that the model is able to reproduce the real process, since the simulated results (blue lines) exhibit the same trend of experimental data (green points), particularly along section C. However, they do not perfectly match, that is a not negligible difference remains. In order to eliminate this discrepancy, an optimization procedure can be settled, thus finding values for specific parameters that improve the results.

Particularly, in this work the influence on the deep drawing process of R–values in material model and static friction coefficient in contacts definition is investigated using the software LS–Opt.

5. OPTIMIZATION

The success of a forming application, i.e. producing a final workpiece showing the required expectations, can be influenced by many variables, especially the ones characterising material model and physical parameters as friction.

A parameters identification procedure using LS–Opt is thus arranged in order to find, for the considered application, the optimal estimates of R–values and static friction coefficient, undergoing specific objectives and constraints. The choice falls on examining these parameters as experimental measurements (hardly obtainable by means of tensile tests) are usually affected by errors or are inaccurate, so it’s opportune being aware of the effects their variations may have on the process and on its products.

Material and process parameter identification is a non–linear optimization procedure used for calibrating material or system properties.

It is an iterative procedure consisting of the following steps:

1. Simulation of the examined non–linear problem. The model parameters to be optimized are provided as input;

2. Extraction of the required responses from the simulation;

3. Comparison of the numerical results with measured data: the deviation of the computed results from the measured data is calculated and a choice is make on how to proceed in the optimal search.

The software LS-Opt was exploited.

In order to vary in LS–Opt the values assigned to the optimization parameters, thus creating different designs at each generation, the keyword *PARAMETER has to be defined in the

“.key“ file. Using this keyword, the R–values and FS values are inserted in the respective slots without a numerical value, which is assigned only in the initialization stage. The values to be assigned are defined in the *PARAMETER keyword and are different for each design.

The *PARAMETER keyword is modified by LS–Opt for each design, since diverse values, randomly generated respecting defined bounds, are inserted in the VALUE field and then are automatically inserted in the pertaining slots of material keyword.

A Direct Optimization method is chosen. This choice entails higher computational costs and long time computation, but, on the other hand, does not use an approximated model, thus providing more accurate results. The number of individuals constituting a population is set equal to 10, a value lower than the default one, mainly because of the complexity of the considered process.

The required values of thickness to be extracted from simulations are determined using an LS–PrePost script and saved into a “.txt“ file. Finally, “.txt“ files containing experimental thickness values, provided in benchmark documentation are loaded as File Histories.

The maximum number of generations is chosen to be equal to 5.

It’s decided to initially employ just one set of data relative to a certain section for running the optimization procedure. In particular, the thickness values along sections B and C are considered. These two sets of data are then combined in only one simulation.

Thickness values provided in the benchmark documentation are compared with the corresponding values obtained simulating the deep drawing process using material parameters furnished in benchmark files and with the corresponding ones determined with the optimization procedures (one section at a time and two sections simultaneously). Also the values of the objective functions are compared.

Optimization run are carried out using two objective functions, i.e. minimizing the distance between simulated and experimental data along the considered sections simultaneously.

The optimal R–values are quite different from the ones employed in the original simulation, see **Error! Reference source not found.**.

Section	R0	R45	R90	FS
Initial guess	1.604	1.388	1.991	0.1
B,C	1.925	1.68	2.12	0.15

Table 5. R–values and FS value resulting from multi–objective optimization

FIG. 15 reports resulting thickness of the multi-objective simulation in comparison with exprimental data.

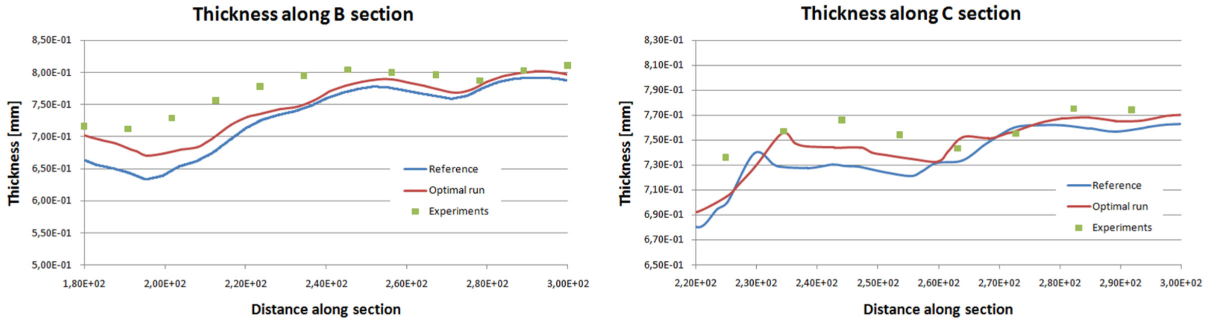


Fig. 15. Graphical comparison of thickness values along section B and C in multi–objective run.

Fig. 16 reports a comparison between the optimal run (set of parameters derived from optimization) and results obtained by participants of the Numisheet Benchmark whose data were available from the Numisheet web site. All the participants reported used LS-Dyna in their analyses. It can be seen that the in both sections optimized curves follow the trend of other simulations.

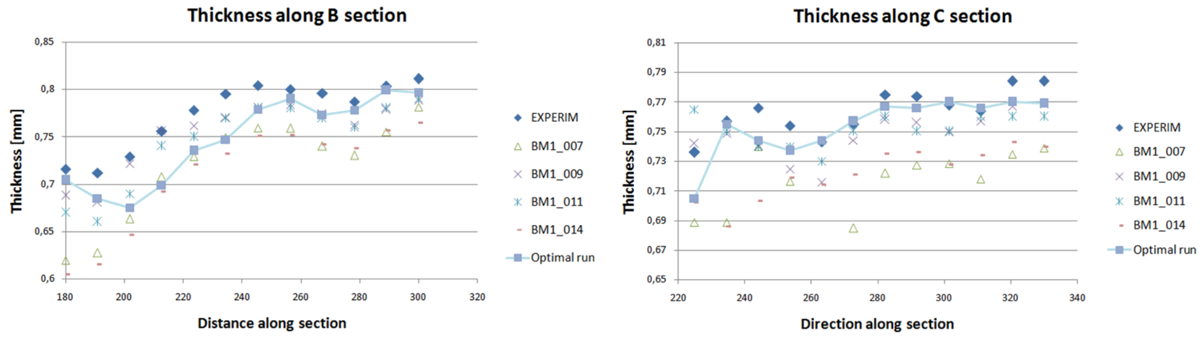


Fig. 16 Graphical comparison of thickness values along section B and C with benchmark results

6. CONCLUSIONS

A four step metal forming process was set up using ANSA, analyzed with LS-Dyna and optimized with LsOpt.

System parameters identification procedures demonstrate that R–values and static friction coefficient actually influence the analyzed physical quantities, i.e. thickness values along different sections, and that using optimal R–values and friction coefficient instead of the provided data permits to obtain results that better match experimental data.

Good agreement with experimental data and also with other benchmark participants results was achieved.

Further developments on this work are planned. In particular, the implemented optimization procedure will be improved modifying the physical quantities to be compared, as stresses or strains, for which experimental data are available.

REFERENCES

(1) ANSA version 15.0.0 User's Guide, BETA CAE Systems S.A., 2014

(2) 2005 Numisheet Benchmark 1 – Springback Prediction of Decklid Inner Panel, in "6th Numisheet Conference", 2005;

(3) LS–Dyna Keyword User's Manual, Volume I, Livermore, CA, Livermore Software Technology Corporation, 2007;

(4) Bletzinger K. , Borrwall T. , Fleisher M. , Experience from using recently implemented enhancements for Material 36 in LS–Dyna 971 performing a virtual tensile test, in "6th European LS–Dyna Users Conference", 2007;