Numerical analysis of multistep ironing of thin-wall aluminium drawpiece

Artur Rękas[1], Tomasz Latos[1], Łukasz Brodawka[2], Maciej Kociołek[2], Michał Siedlik[2], Robert Budzyn[2], Andrzej Furman[2]

[1]AGH University of Science and Technology
[2]Can-Pack S.A.

1 Abstract

This work presents results obtained from numerical analysis of ironing with use of finite element mesh. The base drawpiece, that was a starting point in the analysis, includes a full history of deformation that resulted from simulations of preceeding operations, that is drawing and redrawing. Such a complex approach allows for a complete analysis of each successively conducted process. In the case of materials of thicknesses smaller than 0.250 mm, work hardening is a significant aspect which determines further plastic working of the element. Local work hardening allows for carrying stresses that are necessary to form material in following operations. Numerical simulations of ironing cylindrical elements formed from stripes are highly problematic due to difficulty in defining base material. Such simulations require a reological model of the material including a forming limit curve and a hardening curve. The numerical analysis was carried out for ironing a side wall of a drawpiece from a thin aluminium stripe. The analysis was conducted in Dynaform 5.9.1 with LS-Dyna solver. The material used in the analysis was 3104 aluminium alloy in H19 temper of original thickness 0.250 mm. The final thickness of a wall after ironing is smaller than 0.100 mm. The process of increasing height of a side wall at the cost of reduction in its thickness was conducted in 3 stages. It was a consequence of a degree of reduction in thickness, which was calculated. The data resulting from each individual operation were imported as input data to the analysis of a following operation. The study was carried out in 2 stages. In the first one, the material model was optimised taking into account the problematic character of ironing. The second stage was the analysis of results from the simulation. The state of deformation and stress in the numerical model was analysed. Then, the results were compared to the physical process which is currently used in the industry. The compared results embraced: reduction in thickness of a wall, localisation of the transition to a thin wall of a drawpiece, geometry of a drawpiece.

Keywords: Finite element analysis, ironing process, aluminium alloy, thin-wall drawpiece, LS-Dyna

2 Introduction

Aluminum beverage cans are widely available packages that are often used around the world. Each year new improvements are made in the geometry of the can. These changes do not affect the functionality but have a crucial impact on the strength parameters, which must be checked in continuous and rigorous quality tests. In addition, companies constantly try to minimize the amount of material required for a can body production. Companies are trying to reduce material use by production of cans with thinner side walls. At this point a wall thickness less than 0.1 mm is commonly used. There are many papers devoted to the topic of optimization, or attempt to delve into various stages of can or thin drawpiece stamping processes, such as necking process [1] dome forming [2], redrawing process [3]. In paper [4] simulating of ironing process is taken into account. However, in contrast to the method described in this article, the material has not been previously deformed and only one stage of ironing was simulated.
This paper focuses on the results of numerical simulation of gradually ironed side wall of a beverage can body. The analysis was made taking into account deformation that occurred in two previous stages: cup forming and cup redrawing. The ironing operation was divided into 3 separate stages which corresponds to the actual production process. In each stage, input material is a product obtained from the previous step. In order to verify obtained numerical results, simulated can body thicknesses were compared with the thickness of an actual product.

3 Ironing Process

The process of can side wall ironing is preceded by a number of processes needed to obtain the appropriate geometry of the semi-product:

- Cutting out a proper blank shape from metal sheet,
- Drawing of a cup on a vertical press (Cupper Press),
- Further cup redrawing on a horizontal press (Bodymaker) which provides a final input geometry to an ironing process,

The mechanism of a horizontal press is equipped with a punch (Punch Sleeve) and a special die (Punch Nose) which are mounted on the end of a ram. The Ram forces a cup to go through a redrawing ring followed by a set of rings which gradually decrease thickness of the wall and extend the height of the component. The scheme of a horizontal press which demonstrate arrangement of tools is shown in Fig.1.

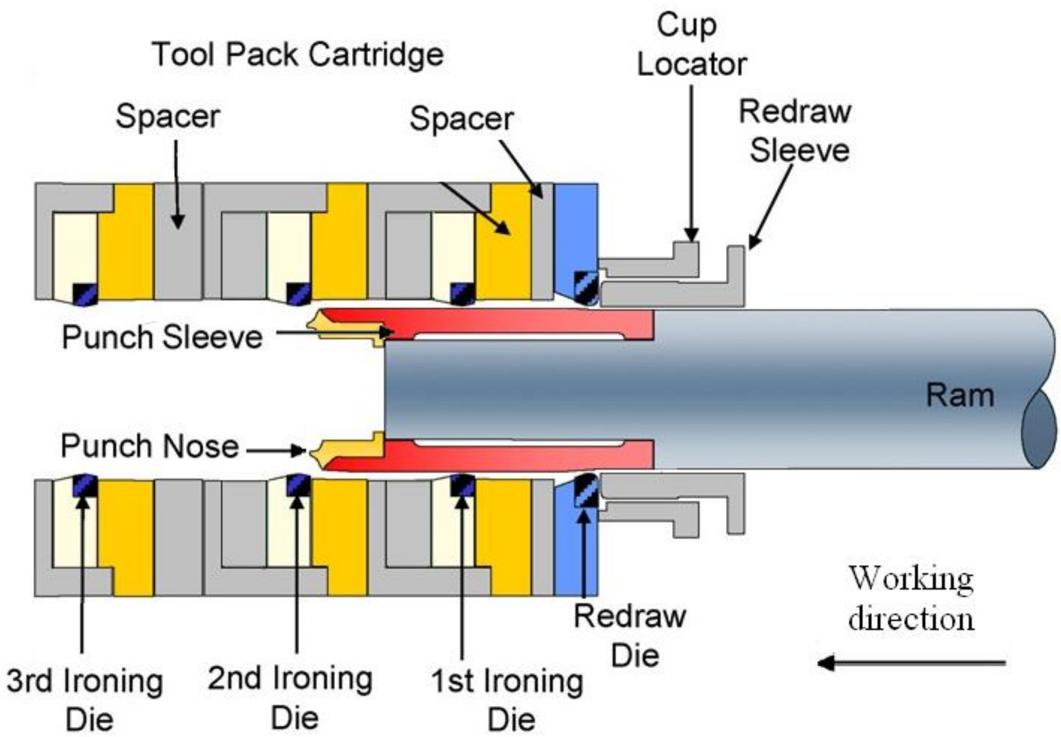

Fig.1: Scheme of a horizontal press - Bodymaker [5]

The toolkit which is required for a proper formation of a final product consists of:

- A punch made mostly of steel or carbide whose main function is to form the thick and thin wall of the can Fig.2.
- A die mounted to the front part of the punch which gives shape to the bottom part of a can,
- Cartridge in which all rings: one redrawing die, 3 ironing dies and spacers are arranged in appropriate distances from each other (distance between the rings should be selected in such a way that the material is formed only in one ring at a time).

Cooling and lubricating emulsion is needed when reducing thickness of a wall. Its main function is to reduce the friction during the process, remove the heat and flush out metal particles which are created during forming.

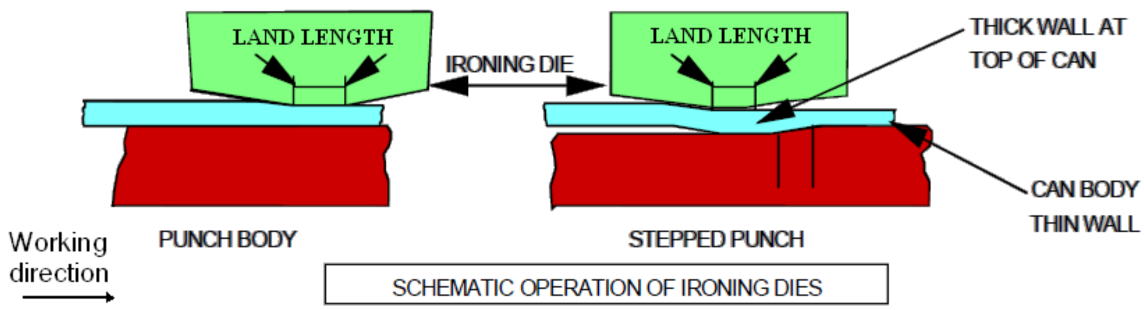

Fig.2: Schematic of wall ironing, taking into account the main forming tools [6].

4 Finite element simulation

4.1 Elements and matrial

The input drawpiece for the numerical simulation is geometry received in the previous two forming stages with full history of stress and strain - Fig.3.

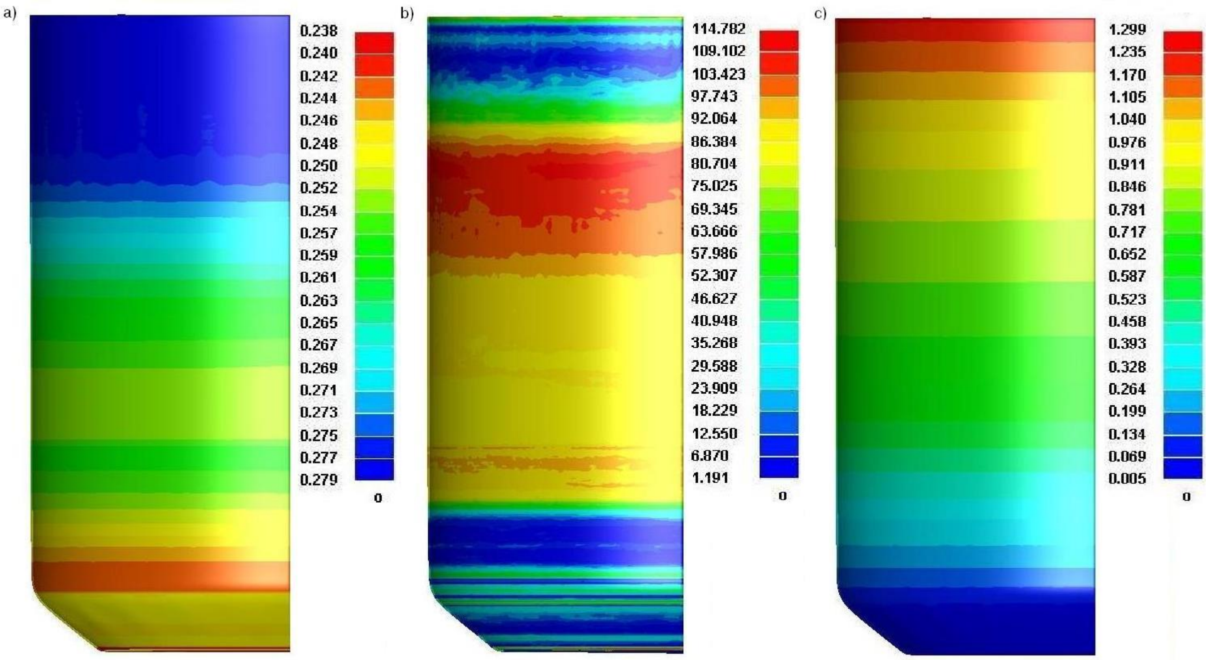

Fig.3: The input drawpiece determined to the ironing process: the average height 80.75mm, a) thickness distribution, b) max Von Mises stress, c) total strain

Among the many attempts on various shell-type elements, it was necessary to use more time consuming solid-type elements. Shell type elements could not catch material deformation in normal to the surface direction. Selected hexahedron 8 node solid elements, Element Formulation = 2 (ELFORM) Fully Integrated S/R, reproduced the change in material thickness during ironing. Element Formulation = 1 Constant Stress, caused too many "dynamic effects" such as material vibration or undesirable deformation (most of it in the bottom of the can). Due to the use of extremely thin thickness of the base sheet, the problem of element locking occurred especially on the die radius in the earlier forming stamping stages of cup forming and redrawing. Locking problem could be bypassed by use of only one layer through the thickness of the material. This method is not desirable but further reduction in element size will result in unacceptably long computation time. Thanks to the axisymmetric nature of the process, two planes of symmetry were used which meant that only a quarter geometry of tools and blank was necessary. Input material for blank - **`MAT_24 Piecewise_Linear_Plasticity`** - isotropic material suitable for solid type elements. Strain rate and failure criterion were not taken into account. More detailed information about conducted tests determining properties of the blank material (Tab.1.) can be found in article [7]. All forming tools were set as shell elements with the default material **`MAT_20 RIGID`**.

Table 1: Material used In simulation

Blank Material		
Material	Aluminium 3104 H19	
Material Model	Piecewise Linear Plasticity	
Density	2.72e-009	
Young's Modulus	58200	
Poisson's ratio	0.33	
Failure PL Strain	0	
Step Size for EL DEL	0	
Rate effect Form.	0	
Strain Param (C)	0	
Strain Param (P)	0	
Hardening curve $\sigma=K\varepsilon^n$	K	n
	356	0.0425

4.2 Contact

`Contact Forming_one_way_surface_to_surface` was applied to all the tools. All kind of trials with penalty method contacts that had been tested gave unacceptable material penetration through tool mesh. The solution to this problem, just as in the article [4], was a change in SOFT method from 0 (penalty method) to SOFT = 4 (constrained). This treatment significantly improved behavior of the material during ironing resulting in almost none material penetration. Because of the complexity of the actual forming process, many factors affect the friction coefficient value which is hard to designate. Estimated constant value of 0,08µ was assumed for all forming tools.

4.3 Mesh

The part that was the subject of the research was divided into three separate stages in which 4 tools were used: one punch and three ironing dies. In each step material was ironed through one die. Two forming surfaces of the punch sleeve and punch nose were merged together in order to maintain the continuity of the surface between tools Fig.4.

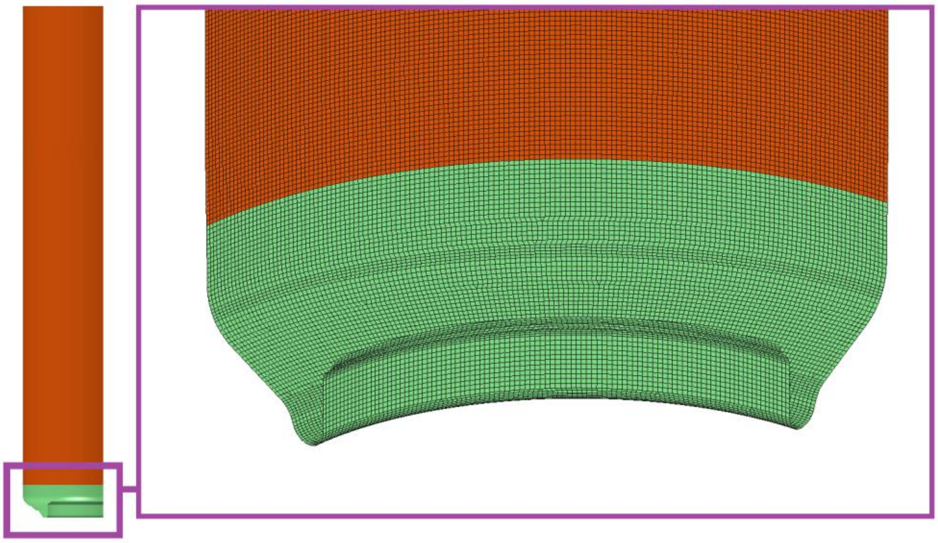

Fig.4: Punch surface of the stamp and its bottom mesh. Brown color indicates the working part of the punch, green working part of the "punch nose".

All tools were imported from external CAD files and meshed by internal mesher included in Dynaform 5.9.1 software using the "tool mesh" method.

Fig.5: One of ironing dies surface and mesh views

Small size of the blank and tools finite element mesh was applied to maintain high accuracy of the thickness distribution results. Maximum length of elements for all forming tools - 0.35 mm, minimum length 0.2 mm. The small size of the elements enabled creation of two elements in the main working part of Ironing Dies. Larger size of the mesh resulted in formation of longitudinal strips of non-uniform thickness on the can body wall. Initial blank geometry was meshed as a quarter circle with radius of 79,5mm. Blank mesh type – "disc mesh" gave better distribution of stresses and strains than default "blank mesh". Minimum element length in the radial direction 0,115mm (greater element length caused gradual loss of uniform thickness in a thin wall after third ironing), the thickness of the batch material was 0.250mm. More details about used elements, mesh and tools shape:

- Ironing die 1 - diameter 66.408mm, 3929 quad elements, 5 triangle elements
- Ironing die 2 - diameter 66.310mm, 3907 quad elements, 3 triangle elements
- Ironing die 3 - diameter 66.174mm, 3903 quad elements, 5 triangle elements
- Punch - thin wall diameter 65.989, thick wall diameter 65.872, 94028 quad elements, 54 triangle elements
- Blank – 78 943 hexaheadrons, 27 wedges

4.4 Proces

All boundary conditions and process parameters were determined using the Autosetup module in Dynaform 5.9.1 Software. Each simulation stage began and ended at the 0 mm/ms speed of a punch. Ironing dies were made static. Whole forming process was made at a constant speed of 2000 mm/ms. Increase and decrease in the speed of the punch between the extreme values took 0,001ms.
`KEYWORD DAMPING_GLOBAL` was used due to observed motions of the material in unsupported by any tool areas (the bottom part). It prevented bottom of the can from unnatural waving and too much thinning of the material.
The SLSFAC value suggested by the software was reduced from 0.08 to 0.01. With a value of 0.08, despite the use of an isotropic material, non-uniform deformation occurred making an edge "earing" at an angle of 45° to the rolling direction. Reduction of this value minimized uneven edge height but did not exclude it completely. Values below 0.01 caused too many errors and instability of the simulation.
The arrangement of tools on each forming step was similar to the arrangements shown in Fig.6. The distance between Punch Nose and the input geometry in vertical direction was set at about 0.1mm.

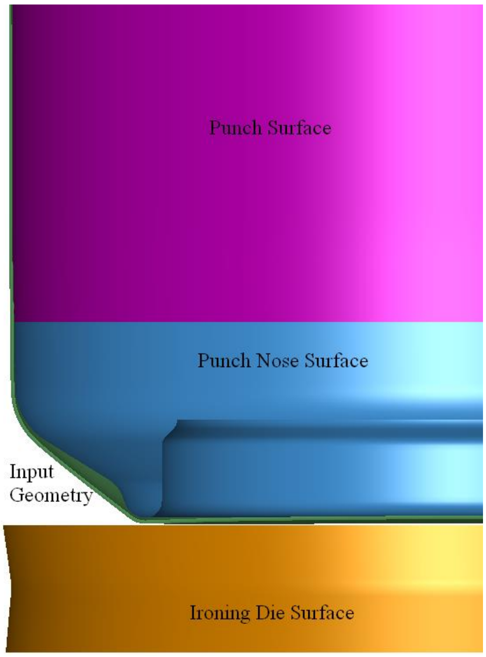

Fig.6: Initial position of the tools in the beginning of each ironing process

5 Results

Images in Fig.7 – 9 shows the results obtained after each of the ironing steps. The main concern is focused on the final can body after the 3rd ironing. Fig.10 shows thickness distribution in a can body taken from numerical results and samples taken from the actual production line. Fig.11. is a visualization of numerical geometry before and after all ironing processes.

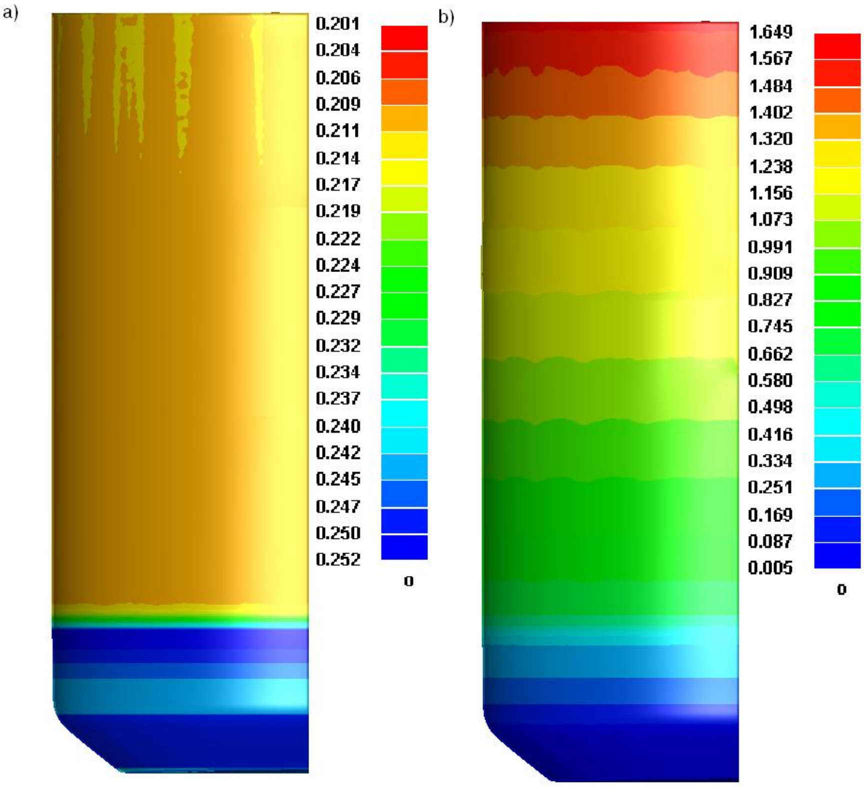

Fig.7: Drawpiece after first ironing: average height – 97.24mm, a) thickness distribution, b) total deformation

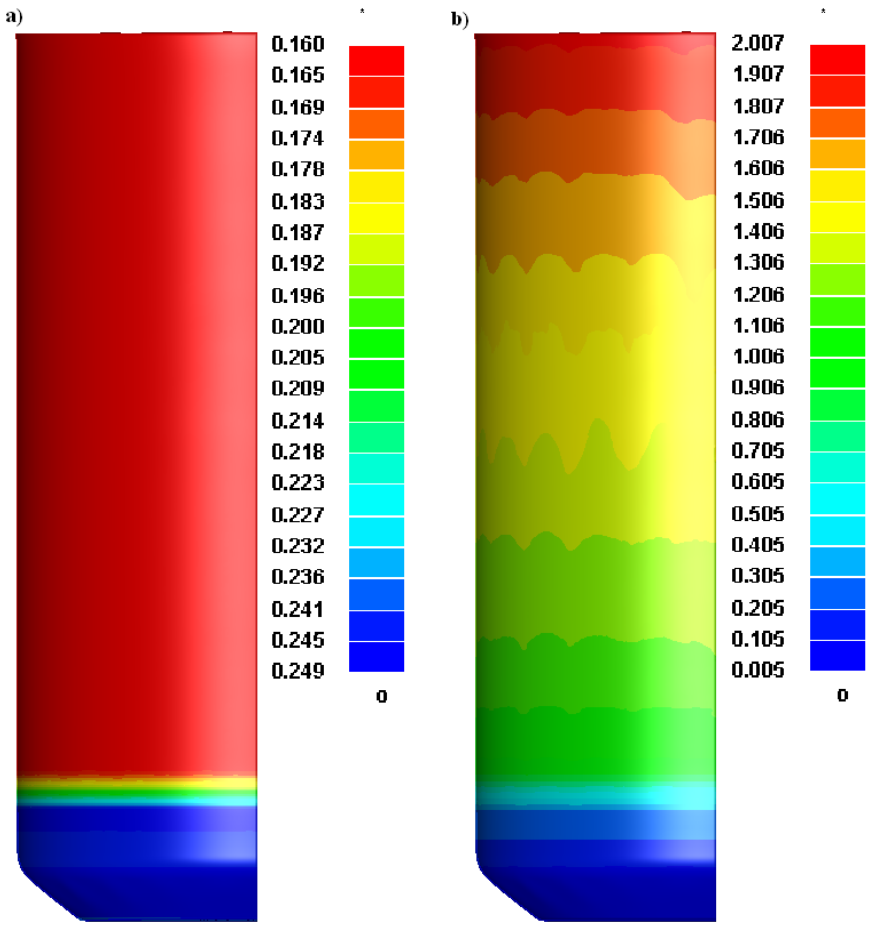

Fig.8: Drawpiece after second ironing: average height 122.85mm, a) thickness distribution, b) total deformation

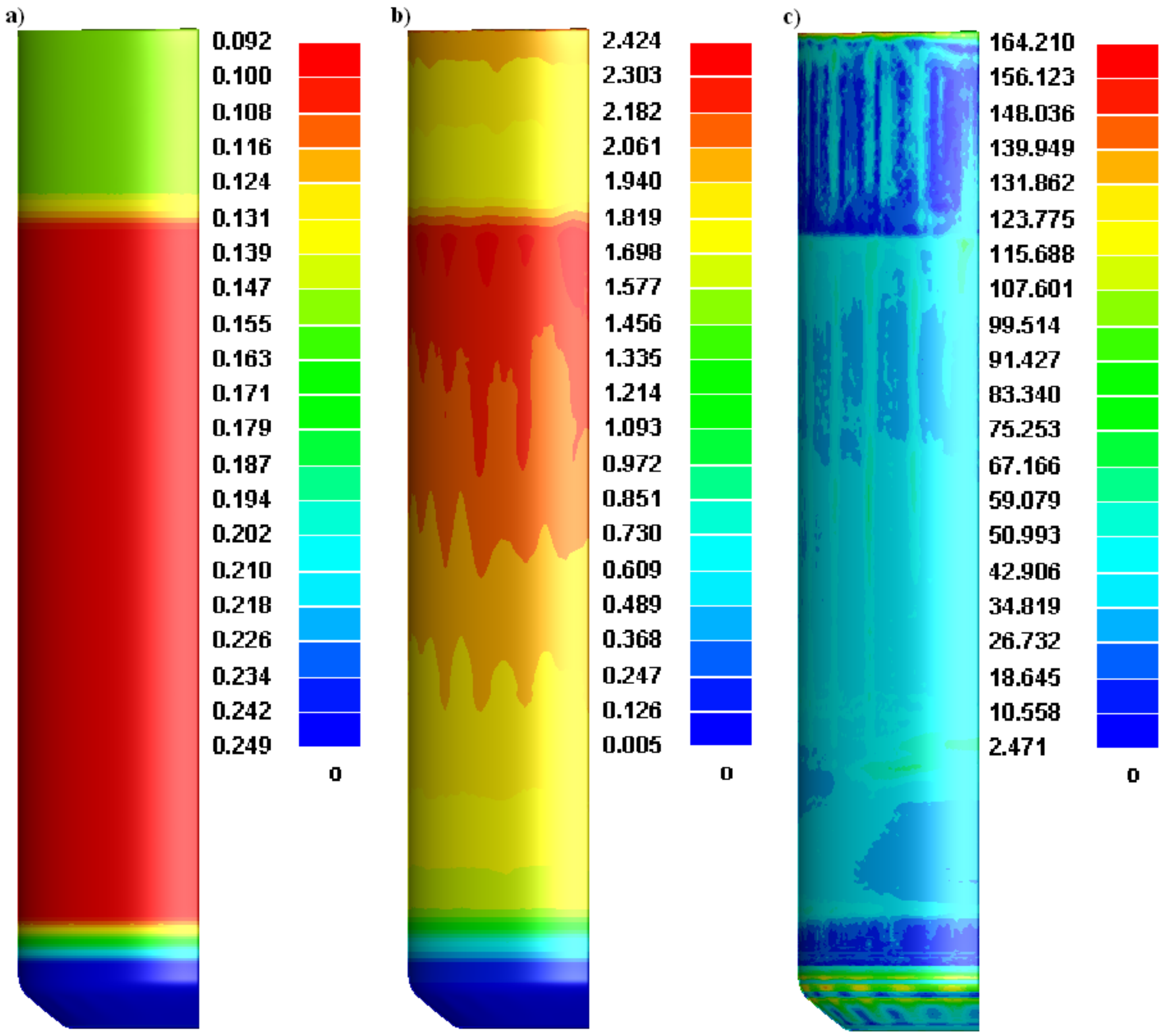

Fig.9: Drawpiece after the third ironing: average height 182.30mm, a) thickness distribution, b) total deformation, c) max Von Mises stress

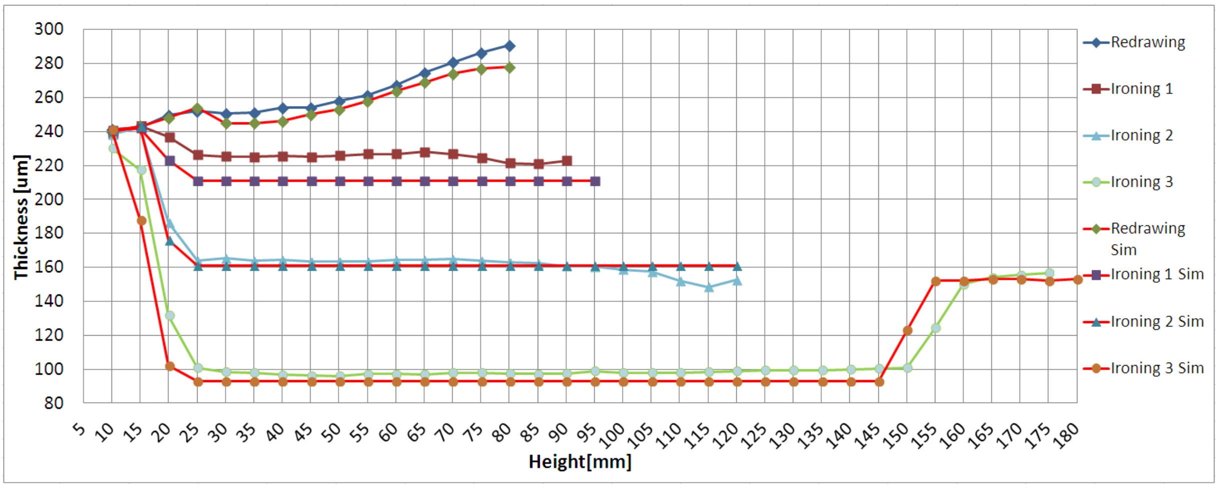

Fig.10: Thickness distribution as a function of can body height, after redrawing process and after each sidewall ironing compared with the results of numerical analysis (Sim)

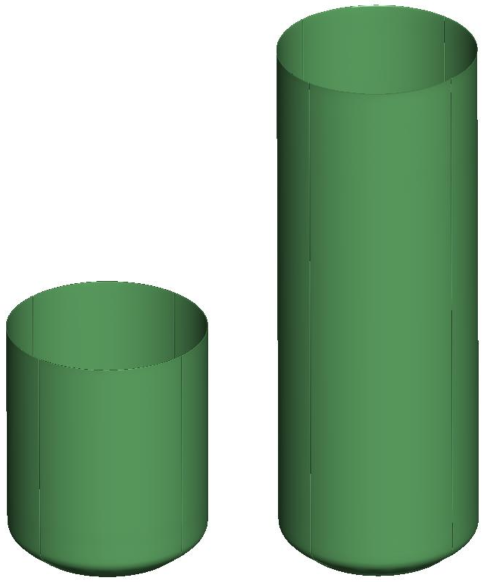

Fig.11: Drawpiece after redrawing process and final can body after three ironing processes Visible vertical edges caused by mirror image since only a quarter of the actual material was formed.

6 Conslucions

Thickness distribution analysis of numerical simulation give promising results in confrontation with measurements made on actual samples taken from production line. The discrepancy observed especially when compared to the thickness as a function of height for first ironing shown in Fig. 10 is not an error but a result of difference between tool reduction used in the real production and in numerical simulation. It was difficult to get a set of samples from a working production line, which would ideally suit numerical simulation assumptions. The different transition height and minimum wall thickness in the simulation after third ironing resulted from the use of different design of a punch in horizontal press.

The simulation results show a perfectly homogeneous course of a thin wall which is a result of perfectly aligned tools, their ideal working surface and the use of isotropic material instead of anisotropic one (which is actually used in production).

Gradual increase in heterogeneity of total deformation for subsequent stages of ironing can be observed in Fis.7-9 b). One possibly occurrence of this effect could be caused by not good enough selection of element size in initial blank. Elements undergo large deformation from its original shape in several forming processes which can lead to numerical instabilities (initial elements with radial length of 0,115mm were stretched up to 0.6mm while edge elements were compressed from width 0.873mm to 0.365mm). Second cause might occur because of too large tools mesh which could lead to similar phenomenon. Mesh refinement may be a solution, but as a drawback, increase in computing time can rise greatly. It is a difficult task to find a compromise between reasonable computation time and high accuracy of obtained results.

Heterogeneity in Von Mises stress which appears clearly in bottom of a can Fig.9 c) is a result from too low binder force at an earlier stage of cup redrawing. It is not certain if small wrinkles in the bottom part of the body influenced ironing process itself. Visible effects on stress and strain graphs did not affect in any way material thinning. A longitudinal heterogeneity in von Mises stresses on the side of thin and thick wall correlate in position with heterogeneity of total deformation.

The final geometry of the numerical beverage can body has comparable shape with the actually produced one. Numerical thickness distribution along the can height is very comparable with produced can which indicates that simulation was carried out properly. In order to define properly the ironing simulation, use of solid elements is preferable. Contact type **`forming_one_way_surface to_surface`** gives promising results but only with constrained method (SOFT=4). However, should be taken into account that constrained method is not suitable for tools which acts with force into material (binders, dumpers).

Use of a tiny solid elements with thin products, in order to capture the effect of accurate thickness reduction causes large computation times and thus are time-consuming in optimization. Properly set numerical simulation is able to provide input data (geometry, stress and strain history) for subsequent can body forming processes, i.e. trimming, dome forming or thick wall necking. Caution should be taken into account when setting all contacts, mesh sizes or material type because even the smallest parameter change can lead to unsatisfactory results or unexpected simulation termination. Determining appropriate parameters and finite mesh geometry may require multiple attempts especially in case when material mesh does not refine itself as in the case of solid 8 node blank elements presented in this paper.

7 Literature

[1] Jordan-Cordera, A., Miranda-Valenzuela, J.C.: “Simulation and Analysis of the Beverage Can Necking Process Using LS-DYNA”, 8th International LS-DYNA Users Conference (2004), Session 9 - Metal Forming Technology.

[2] Dick R.E., Yoon J.W.: ”Effect of Material Characteristics on Wrinkling During Dome Forming of a Beverage Can using LS-DYNA”, 10th International LS-DYNA Users Conference (2008), Session 17 – Metal Forming.

[3] Dick R.E.: “Improvements to the beverage can redraw process using LSDYNA”, 7th International LS-DYNA Users Conference (2002), Session 15 – Metal Forming Technology.

[4] Danckert j.:”LS-DYNA used to analyze the manufacturing of thin walled cans”, 6th European LS-DYNA Users’ Conference (2007),

[5] Kang K.:”Impact of die wear and punch surface textures on aluminium can Wall”, An International Journal on the Science and Technology of Friction, Lubrication and Wear, Wear 266 (2009) 1044–1049.

[6] Wootton E.,” Case Study on Can Making” Training in Aluminium Application Technologies, European Aluminium Association, Information on http://www.alueurope.eu/talat/lectures/3710.pdf

[7] Rękas A.,Latos T.,Budzyn R., Furman A., Siedlik M.:”The Analysis of Mechanical Properties of 3XXX series Aluminium Sheets Used for Beverage Can Production”, Key Engineering Materials, Vol 641, Apr. 2015, pp. 246-256,