Strength Analysis of Thermoplastic Containers Using Numerical Simulation

C. Pagliosa[1], F. C. Caires[1], D. Bataglia[2], and J. Pelaquim[2].

1 – Grupo Smarttech

2- Syngenta Brazil

***Abstract**: This study presents a numerical simulation strategy developed for strength analysis of thermoplastic containers. The method is based on the finite element analysis of both, the blow-molding process and the in-service loadings. The blow-molding analysis is performed to get the molded shape thickness distribution. The parison thickness and the coefficient of friction between the parison and the mold cavity are treated as parameters. They are varied until a good estimate of the thickness distribution is achieved. The thickness distribution obtained from the blow-molding analysis is then used in the subsequent loading analyses on the finished product. The material is defined based on the actual material properties and the loading analyses are performed to get the stress fields on the finished product subjected to drop test and vacuum load. The smoothed particle hydrodynamic analysis method is used in the drop test simulation of the fluid-filled container, whilst a uniform hydrostatic pressure is applied inside the container in the vacuum loading analysis. Simulation results are compared to the ones obtained from physical tests on an existing product and the methodology is applied to verify two new designs.*

***Keywords**: Thermoplastic Containers, Blow-Molding, Top Load, Drot test, Buckling, SPH, Collapse, Hyperelasticity, Impact, Interface Friction, Plasticity, Polymer.*

1. Introduction

The development of thermoplastic containers based on the traditional trial and error process can be expensive and time consuming. A numerical simulation methodology using Abaqus has been proposed and evaluated in order to assist engineers in the decision making process during the early phases of product development, reducing costs with physical prototypes and time to market. The method is based on the finite element analysis of both, the extrusion blow-molding process and the in-service loadings acting on the container. The blow-molding analysis is performed to get the molded shape thickness distribution which is then used in the subsequent loading analyses on the finished product. Validation of the simulation methodology is based on the results obtained from physical tests on existing product (reference model). The validated methodology is then used to compare two different design proposals of a new product.

2. Simulation methodology

2.1 Blow-molding analysis

The simulation sequence begins with the finite element analysis of the manufacturing process of the plastic container, a one litre plastic bottle, which is the extrusion blow-molding. In this manufacturing process, the melted plastic is extruded into a hollow tube (parison). The parison is then enclosed by a metal mold and air is then blown into the parison, inflating it into the shape of the container (a hollow bottle). In this study, the parison is modeled using Lagrangian finite elements (S4R), whilst the metal mold is modeled as a rigid body (R3D3 and R3D4 elements). Half symmetry is considered and special attention is given to mesh refinement in order to maintain a high-quality mesh upon large deformations throughout the analysis, **Figure 1**.

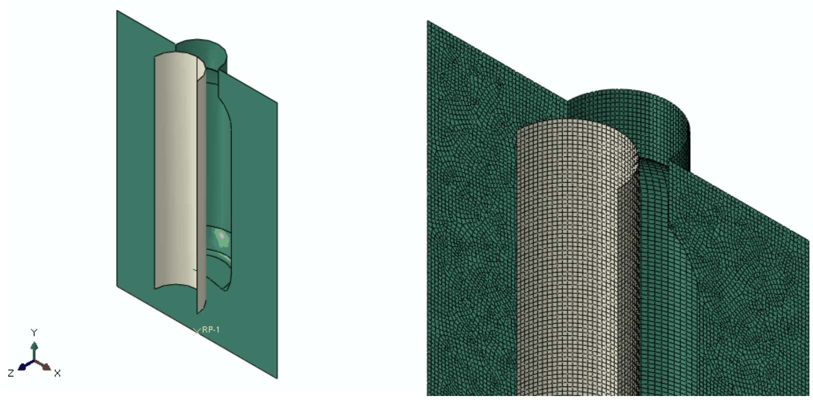

Figure 1. Model components (parison in gray and mold in green) and mesh detail.

The simulation of the manufacturing process consists of two explicit dynamic analyses. In the first analysis the parison is clamped into the metal mold, **Figure 2**. Then, the results are transferred to a second analysis and air is blown into the parison using the surface-based fluid-filled cavity, see **Figure 3**.

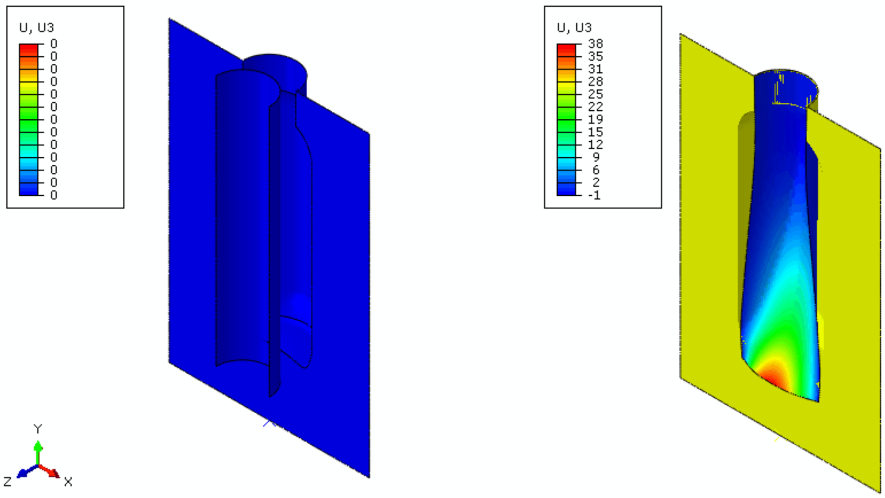

Figure 2. First analysis: parison is clamped into the metal mold.

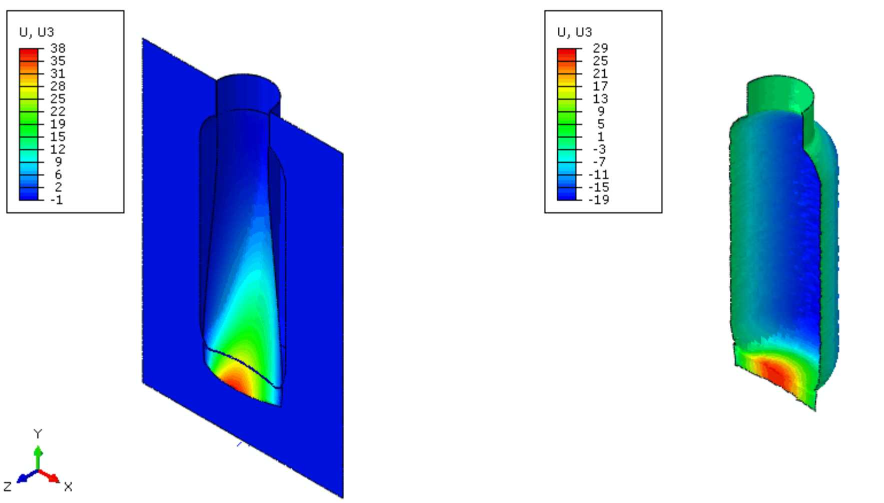

Figure 3. Second analysis: air is blown into the parison cavity to get the final thickness distribution.

The fluid cavity capability provides the coupling between the deformation of the parison and the pressure exerted by the air on the cavity boundary. The surface-based fluid cavity is defined inside the parison and uses a fluid exchange definition to model the mass flow into the cavity to blow the parison to the final shape. The mass flow can be estimated assuming an initial zero pressure inside de cavity and zero velocity outside de cavity. From Bernoulli´s equation, the velocity v of the mass flux into the cavity is defined according to Equation 1:

$$\frac{1}{2}\rho v^2 = p_A$$

where ρ is the air density (typically $1.225\ kg \cdot m^{-3}$), and p_A is the air pressure (typical blow air pressures around 0.2 to 1.0 MPa). The mass flow per unit area can then be estimated using Equation 2:

$$\dot{m}_{in} = \rho v$$

The total amount of mass blown into the cavity depends on the step time.

A single fluid behavior defines the air properties, modeled as an ideal gas, given the universal gas constant, $8.31\ J \cdot mol^{-1} \cdot K^{-1}$, and the molecular weight, $0.0289\ kg \cdot mol^{-1}$.

For rapid events such as the blow-molding, the gas expansion is so quick that no significant amount of heat can diffuse out the cavity and, for this reason, an adiabatic process is assumed. When modeling an adiabatic process for the ideal gas, the molar heat capacity at constant pressure, c_p, must be specified. In this study, it is expressed in tabular form at typical room conditions ($29.2\ J \cdot mol^{-1} \cdot K^{-1}$).

Hyperelastic material properties are assigned to the parison using the arbitrary stress-strain curve shown in the graph of Figure 4. The Mooney-Rivlin strain-energy potential is used to model the mechanical behavior of the melted thermoplastic (HDPE).

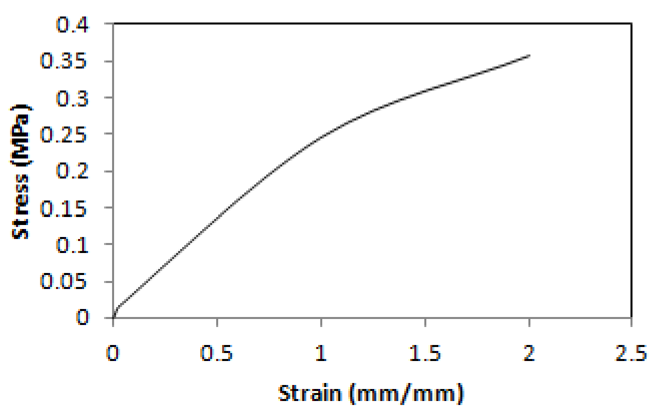

Figure 4. Stress-strain curve used in the blow-molding analysis.

The interactions between the parison and the mold are defined through contact pairs using the Coulomb friction model. During the validation phase of the blow-molding simulation method, the parison thickness and the coefficient of friction are initially treated as parameters until a reasonable estimate of the thickness distribution is achieved based on the thickness mapping of the existing product. The parison thickness and the coefficient of friction are then held constant throughout the subsequent analyses.

2.2 Top loading analysis

Validation of the loading analysis method requires correlation between results obtained from simulation and experiments. Top loading analysis of the existing product generated the simulation data for comparison with the test data available. In this test, the plastic bottle is compressed between two rigid plates. An analysis model with the designed final shape of the container is built and the final shell thickness distribution obtained from the blow-molding analysis is mapped in an average sense onto the new mesh. The container is meshed with S3R and S4R shell elements, and the rigid plates are modeled using analytical surfaces, Figure 5.

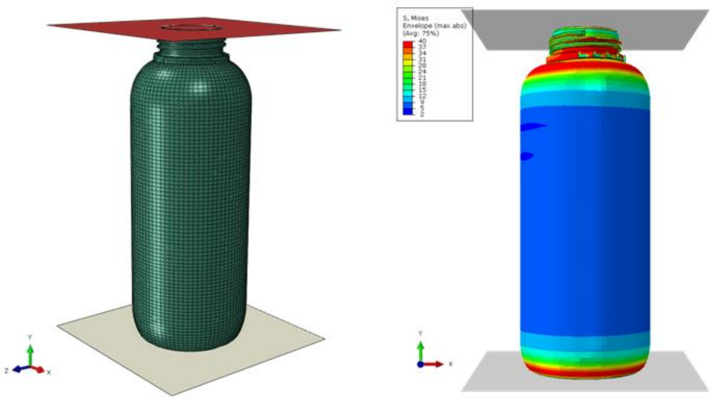

Figure 5. Top loading analysis model (left) and typical analysis results (right).

General contact is defined between the container and the rigid plates with the Coulomb friction model. A static analysis is defined considering the geometrically nonlinear effects. For the sake of simplicity, an isotropic plastic hardening model is assigned to the container with yield stress given as tabular function of plastic strain. The graph of Figure 6 shows the comparison between the experimental force-displacement curve and the simulation ones obtained using material properties from datasheet (mp datasheet) and data matching (mp calibration).

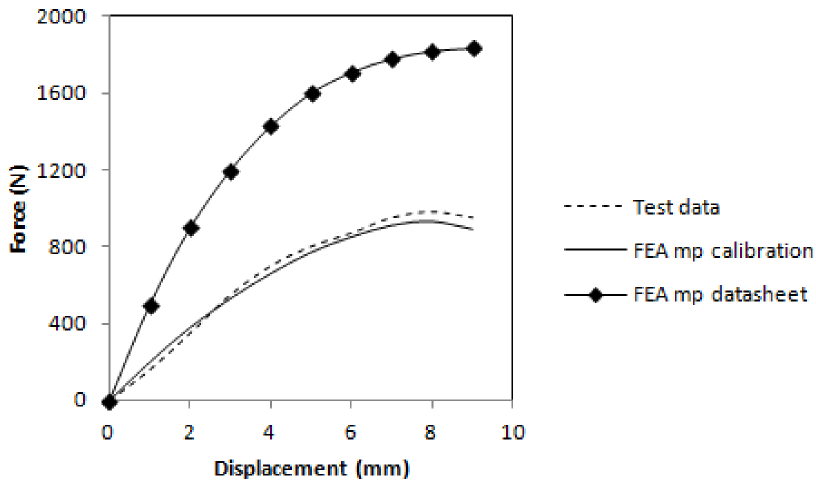

Figure 6. Force-displacement curves obtained from simulation and experimental testing.

The simulation results obtained using the mechanical properties found in the material´s datasheet are very inaccurate when compared to the experimental ones. Isight is then used to calibrate the elastic-plastic material model parameters. The Isight workflow is shown in Figure 7.

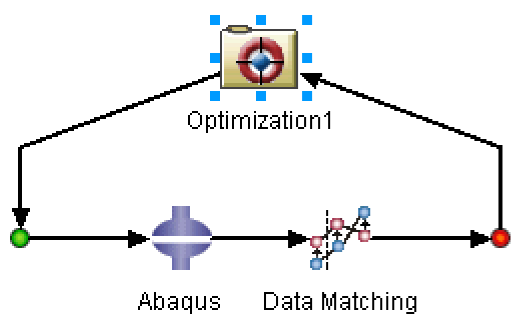

Figure 7. Isight workflow.

It uses a single Abaqus component, which runs the top loading analysis. The displacement from the test is used to drive the simulation model displacement. The Data Matching component is used to compare the simulation response to the test data. The error measure is the sum of the square difference and the optimization driver with the Hooke-Jeeves algorithm is used to perform the optimization. As can be seen from figure 6, after the calibration of the material model parameters, the simulation results are in good agreement with the test data. The subsequent analyses are performed using the material model parameters obtained from de calibration procedure. The graph in **Figure 8** shows the stress-strain curve of the material.

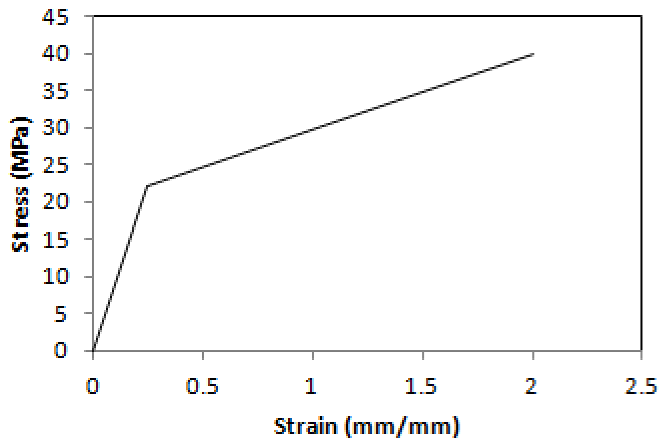

Figure 8. Stress-strain curve of HDPE.

2.3 Collapse analysis

When plastic containers are filled with hot liquid, they tend to collapse due to the vacuum pressure created by cool-down after hot-filling. The collapse analysis is performed to determine the pressure at which the container buckles. Both, material nonlinearity and geometrically nonlinear effects are considered. A nonlinear static step is defined and the step time is defined so that the time period and the applied vacuum pressure are equivalent. The mesh imperfection is introduced naturally, through the thickness distribution obtained from the blow-molding analysis, in order to get some response in buckling mode before the critical load is reached. In this class of problem, the analysis terminates due to convergence difficulties. Negative eigenvalue warnings begin to appear and are indicative of the buckling load having been exceeded. Thus, the critical pressure corresponds to the step time at which the analysis terminates. The point at which the onset of buckling and collapse occurs can also be identified through the static dissipation energy, the energy dissipated by the automatic stabilization technique.

2.4 Drop test

The drop test simulates the plastic container falling from a drop height of $1.200\ mm$ onto a flat, rigid surface, in order to investigate the bottle´s response under harsh handling conditions. The bottle is filled almost completely with water. The simulation must account for both the exterior forces on the bottle from the floor impact, as well as the interior forces of the water against the bottle. The meshless method, named smoothed particle hydrodynamics (SPH) is used to analyse the fluid-filled container drop in a purely Lagrangian environment. The water is modeled using reduced-integration continuum (C3D4) elements. A time-based criterion is used to convert the water mesh to continuum pseudo-particles (PC3D elements) at the beginning of the analysis. A single step of explicit dynamic analysis is defined with a step time of 0.04 s. The container is meshed with conventional shell (S3R and S4R) elements, and the rigid floor is modeled using an analytical surface, see Figure 9.

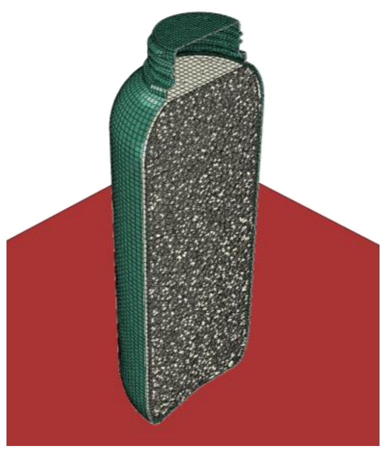

Figure 9. Cross-section view of the drop test simulation model (bottle in green, water in gray, and floor in red).

General contact is defined for the model. Contact between the fluid and the bottle is defined by including an element-based surface that includes the fluid´s interior and its initial external surface in the general contact domain.

The water is modeled using the linear $U_s - U_p$ Hugoniot form of the Mie-Gruneisen equation of state. The water has a density of $996\ kg \cdot m^{-3}$ and bulk modulus of $2.0\ GPa$. The equation parameters c_0 and s define the linear relationship between the shock velocity, U_s, and the particle velocity, U_p, according to Equation 3:

$$U_s = c_0 + sU_p$$

The values of c_0 and s are $1.45 \times 10^6\ m \cdot s^{-1}$, and 0 respectively.

Two drop conditions are evaluated: top edge experiencing initial impact and bottom edge experiencing initial impact. In both conditions, the bottle strikes the floor at a right angle, see Figure 10. The container is positioned close to the floor and an initial velocity predefined field is prescribed corresponding to the drop height.

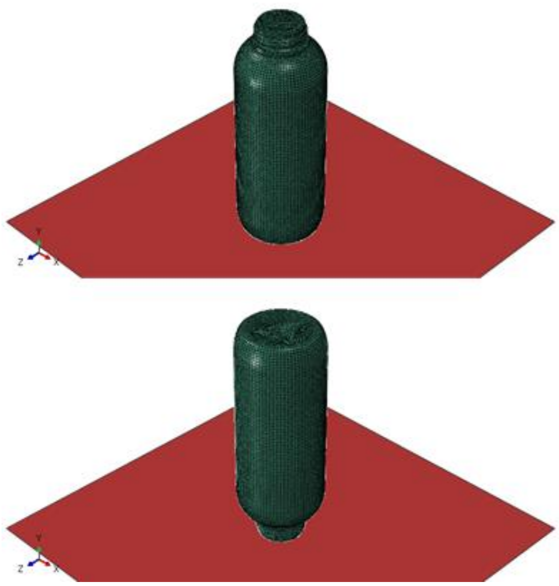

Figure 10. Drop positions.

3. Results

3.1 Blow-molding analysis

The results of the blow-molding analyses are shown in Figure 11 through Figure 13. The final thickness distribution in the reference model (Figure 11) is consistent with the thickness of the actual product. The bottle is thicker in the bottom and thinner in the transition zone which spans from the bottom to the cylindrical region. The thickness of the cylindrical region exhibits a small variation and the bottle gets thinner again in the shoulder. The neck thickness is neglected, since the actual values of this region are defined in the loading analyses.

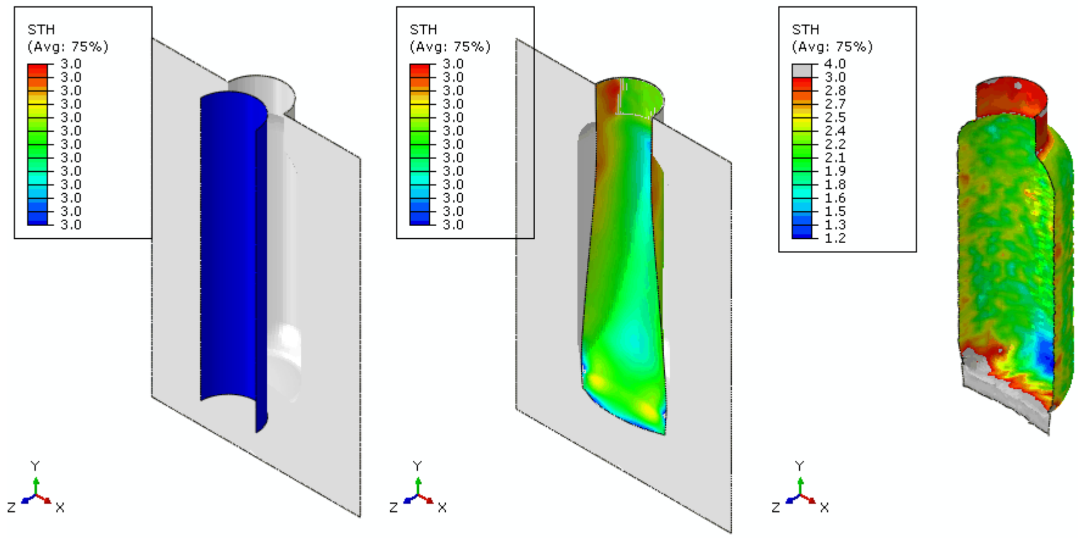

Figure 11. Thickness distribution during the blow-molding analysis sequence (reference model).

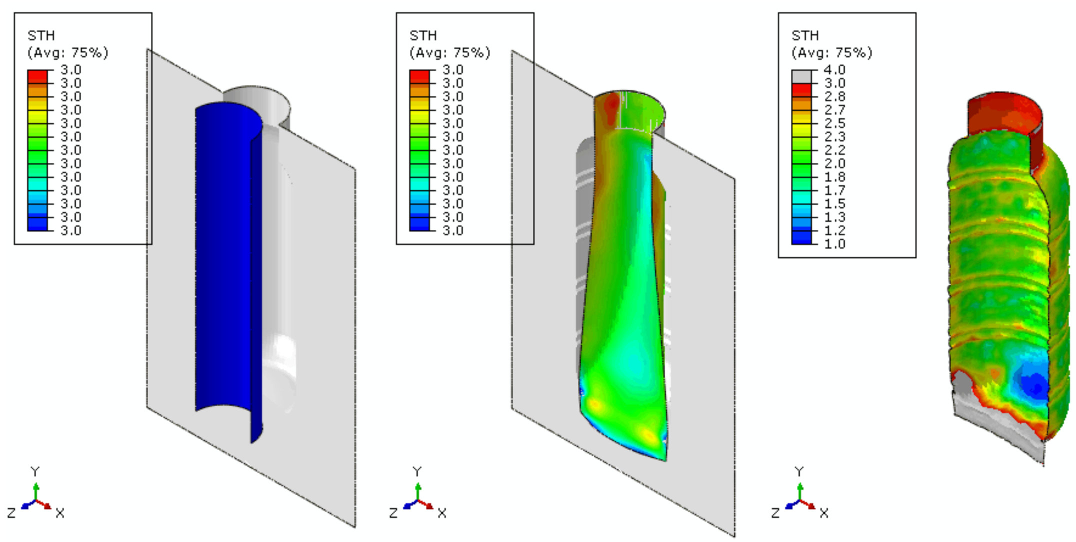

Figure 12. Thickness distribution during the blow-molding analysis sequence (design 1).

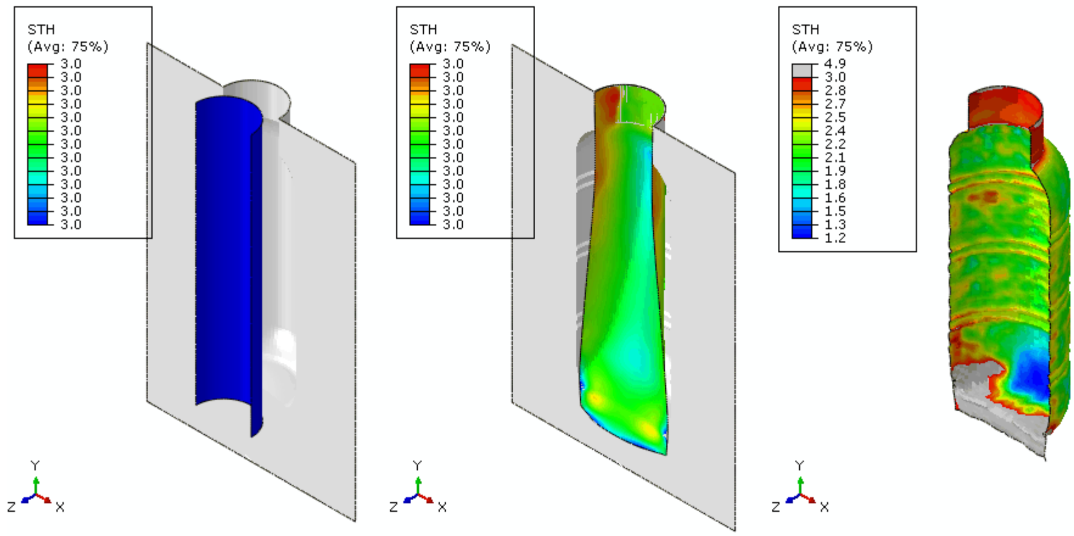

Figure 13. Thickness distribution during the blow-molding analysis sequence (design 2).

The thickness distributions in new design proposals (design 1 and 2) are comparable with the thickness distribution of the reference model.

3.2 Collapse analysis

The results of collapse analysis are presented in Figure 14. Though the different containers exhibit the similar buckling mode, all the analyses fail to converge at different step times. The design proposal 1 and the reference model exhibit the highest and the lowest critical pressure respectively. The collapse load of design 1 is 11% higher than that of the reference model.

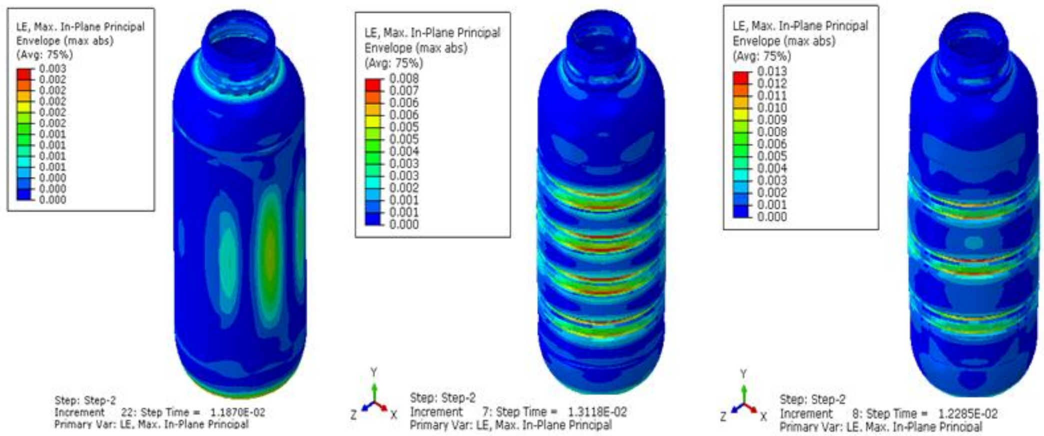

Figure 14. Collapse analyses fail to converge at different step times.

3.3 Drop test

The results of the drop test simulation are presented in Figure 15 and Figure 16. The figures show the logarithmic strains in the bottles. Peak strains occur in regions of high stress gradients and on the bottom of the bottle, along the bottle-floor interface (bottom drop), or on the shoulder of the bottle, along the neck-shoulder intersection (top drop). These strains are largely recovered after the first impact. The strain levels in the new design proposals 1 and 2 are comparable, but they are higher than the strain level in the reference model.

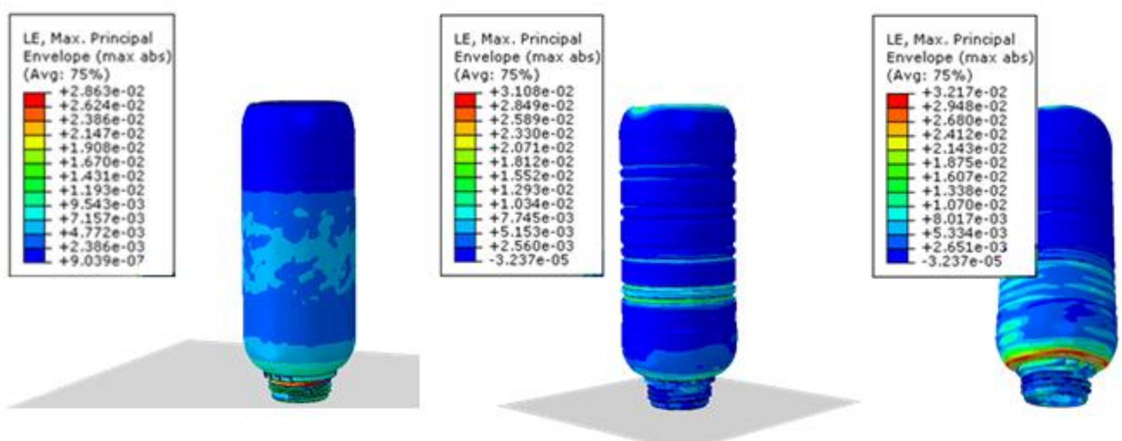

Figure 15. Logarithmic strains in the bottles (top drop).

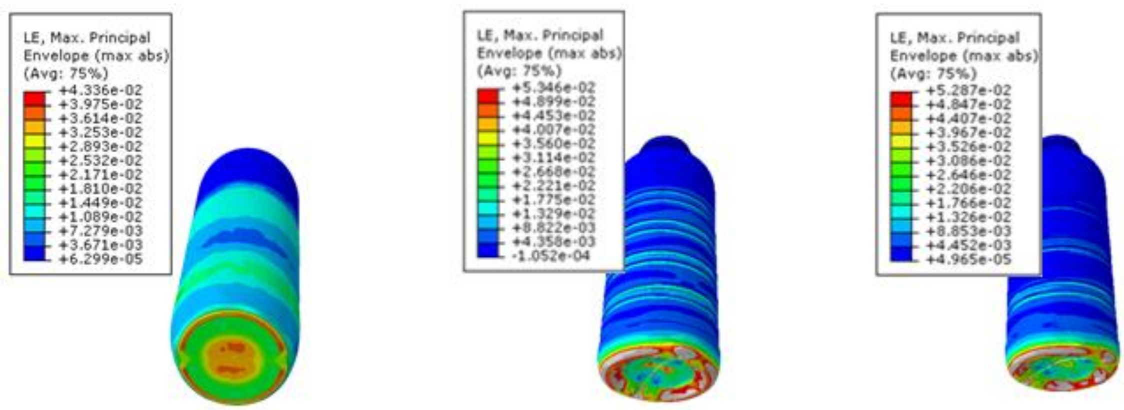

Figure 16. Logarithmic strains in the bottles (bottom drop).

4. Conclusions

A simulation methodology was proposed for the strength analysis of thermoplastic containers. Finite element analysis models of three different container models were built to get the thickness distribution resulting from the manufacturing process. These thickness values were mapped onto the final design shape of the containers, the parameters of the material model were calibrated, and loading analyses were performed to evaluate the performance of the different container models. A reasonably accurate thickness distribution was obtained from simulation with maximum 15% difference of the actual thickness of the reference container. The collapse load obtained from the analysis of the reference model is also in good agreement with test data, which is 13% higher than the simulation result. The analysis of the new design proposals showed higher values of the collapse load. The results obtained from the drop test simulations of the reference model showed predominantly elastic strains with only local plastic strains in small regions along the bottle-floor interface. Despite the higher strain levels observed in the new design proposals, the peak values occurred in small regions only, as in the reference model. Some simplifying assumptions have been made in the development of this methodology and a next step could be the inclusion of the anisotropy of the material properties. Further improvement of the simulation method could be development of an optimization process integrating the analysis of the different load cases in order to get the geometric parameters of a product that meets the performance targets with minimum mass.

5. References

1. Tanifuji, S., Kikuchi, T., Takimoto, J., Koyama, K., “Overall Numerical Simulation of Extrusion Blow Molding Process”, Polymer Engineering and Science, Vol. 40, No. 8, 2000.
2. Yang, Z. J., Harkin-Jones, E., Menary, G. H., Armstrong, C. G., “A Non-Isothermal Finite Element Model for Injection Strech-Blow Molding of PET Bottles with Parametric Studies”, Polymer Engineering and Science, Vol. 44, No. 7, 2004.
3. Yang, Z. J., Harkin-Jones, E. M. A., Armstrong, C. G., Menary, G. H., “Finite Element Modelling of Stretch Blow-Molding of PET Bottles Using Buckley Model: Plant Tests and Effects of Process Conditions and Material Parameters”, J. Process Mechanical Engineering, Vol. 218, 2004.
4. Mukherjee, S., “Virtual Simulation of Top Load Performance of Plastic Bottles”, Society of Plastic Engineers, Antec 2010.
5. Reed, P. E., Breedveld, G., Lim, B. C., “Simulation of the Drop Impact for Moulded Thermoplastic Containers”, Int. J. Imp. Eng., Vol. 24, 2000.
6. Karac, A., Ivankovic, A., “Behavior of Fluid-Filled Plastic Containers Under Drop Impact”, Proc. Int. Conf. on Computational Engineering Sciences, LA, USA, 2000.
7. Abaqus Users Manual, Version 6.10-1, Dassault Systémes Simulia Corp., Providence, RI.