Preform Optimization Using Non-linear Finite Element Simulations

Abstract

Non-linear finite element simulations of the blow molding and thermoforming processes have been used to provide accurate predictions of the material thinning. However, this powerful simulation tool has provided limited assistance to the design and optimization of the wall thickness distribution of preforms used in injection blow molding. Trial-and-error methods are often used in design. This paper presents a technique that converts finite element analysis of the injection blow molding process into a design/optimization tool. A systematic optimization technique for preform design that uses an iterative series of non-linear finite element simulations will be described. The series of simulations converges on a preform wall thickness distribution that will result in a specified thickness distribution in the blow molded product. This design technique is especially effective for non-circular or irregular shaped products.

Preform Optimization Goal

Injection blow molding involves three stages. The first stage is the injection molding of the plastic preform that is commonly shaped like a test tube. The second stage is the conditioning of the preform, bringing its temperature to the proper level for molding. In the third stage the preform is placed inside of a mold cavity that is in the shape of the final product. Air pressure is applied to the inside of the heated preform causing it to expand like a balloon. The expansion continues until the heated plastic touches the cool mold cavity forming the product.

As the material in the preform stretches it also thins. Because of the initial geometry of the preform, the stretching and thinning will not be uniform. When the expanding material touches the inside of the mold cavity it quickly cools which stops the stretching and thinning process. The goal or optimum design is one where the initial thickness distribution of the preform is such that the final product has a uniform or specified wall thickness.

Background

Scientists and engineers in many industries, to assist in the design and optimization of products, have adopted finite element analysis to provide guidance and direction. However, the use of this analysis tool in the design of preforms for injection blow molded products has been relatively slow to develop. The high degree of difficulty in running an accurate blow molding simulation as compared to more common finite element analysis has been the primary reason for its limited use in design.

The most common finite element analysis involves linear material properties and small deflections of the structure. With the advances in technology in recent years, a linear-elastic small-deflection finite element analysis can be conducted using the average office computer and relatively inexpensive software. In many cases, the material properties needed for the analysis can be found in textbooks. Engineers have easily developed the necessary skills to use finite element analysis in their design work.

In contrast, an injection blow molding simulation involves non-linear material behavior and large deflections of the preform. Non-linearity of the blow molding simulation requires significantly more compute power and a more expensive non-linear finite element code. For some materials, the properties must be determined experimentally using specialized test equipment. In addition, the level of expertise required to run a simulation involving non-linear behavior is higher. All of these factors have combined to slow the use of finite element simulations in injection blow mold product design.

Without the aid of advanced computer analysis tools, optimization of blow molded products is as much an art as a science. Often the design process for blow molded products is the result of a significant amount of trial-and-error combined with the past experience of the product developer. The result is that most products are not optimized. Minimum thickness requirements are met in critical areas of the product while other areas have excessive thickness. This is especially true for unsymmetrical or irregular shaped products.

While the challenges of running an injection blow molding simulation are greater as compared to simulations in other industries, the potential benefits are also greater. The fact that the preform thickness distribution can be varied provides the opportunity to optimize the product.

Ideally, the polymer is at a uniform temperature prior to forming. Many polymers behave nearly identically when they are heated uniformly to the proper temperature for forming. This was first reported in a paper presented at ANTEC 89 (1) related to experimental work in thermoforming. Thermoforming is a very similar process to blow molding. The results of the study showed that three different materials, which were processed at different temperatures, yielded almost identical thickness profiles in the final product. Also, excellent correlation was found between a finite element analysis of the thermoforming process and the experimental data. Similar results have been reported with extrusion blow molding (2).

The stress-strain behavior of the polymer at forming temperatures is assumed to be non-linear and elastic or hyperelastic. A hyperelastic material is incompressible (Poisson's ratio of about 0.5). Thus the volume of the material remains constant. Inplane stretching of a thin material made of incompressible material will result in a corresponding reduction of the thickness to maintain a constant volume. The viscous behavior of the polymer will be neglected in this simulation.

Accurate finite element simulations of the injection blow molding process are possible. However, this capability alone does not allow the preform to be optimized because finite element modeling is an analysis tool and not a design tool. The product developer must still rely on trial-and-error techniques. Even though the trial-and-error techniques could be conducted on the computer, a direction toward optimization would still not necessarily be clear. What is required is a systematic technique to convert finite element analysis into a design tool. This paper presents a technique to optimize the wall thickness of preforms using non-linear finite element techniques.

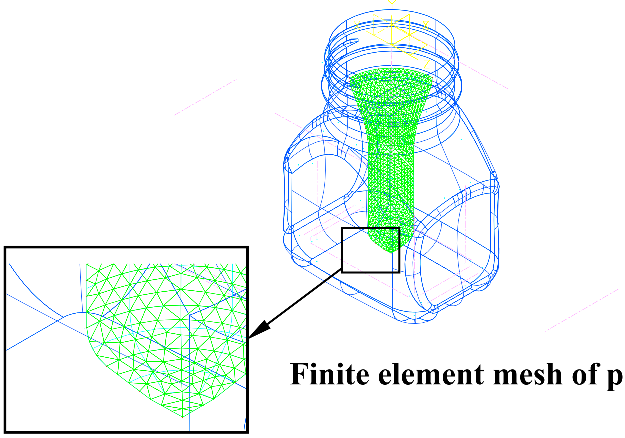
The Finite Element Model

Finite element analysis is a mathematical technique used to evaluate complex geometries by dividing the structure into small building blocks called elements. In the ideal case, the physical behavior of each element matches the actual material. The behavior of the entire structure is then determined by adding the contributions of the individual elements.

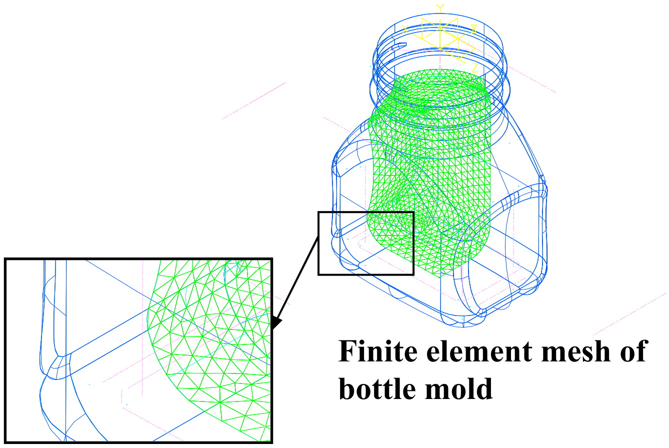
In the simulations reported in this paper, triangular shell elements were used. A shell element is used to model structures where the thickness is significantly smaller than the other dimensions and the stresses normal to the thickness direction are negligible. The triangular shell elements are joined at their corners, called nodes, to form the finite element mesh. The elements had six degrees of freedom (three in translation and three in rotation) at each node. The shell elements used in the analysis account for membrane strains and allow the shell thickness to change with element deformations.

The 3D solid computer model of the injection molded bottle and preform used for demonstration purposes in this paper is shown on Figure 1. The bottle was oblong in shape with two concave sides and two flat sides. Because of the irregular shape it was anticipated that the optimum preform thickness distribution would be non-uniform in all directions.

3D solid model of preform and bottle



Finite element mesh of preform



Finite element mesh of bottle mold

Figure 1. 3D solid model of the bottle, FEA mesh of the preform and bottle mold.

The finite element mesh of triangular shell elements is also shown in Figure 1. One mesh of deformable shell elements was developed for the preform. A second mesh was developed using rigid elements to model the bottle mold.

The simulation took advantage of symmetry by modeling only one-quarter of the bottle. Using the proper boundary conditions at the edges of the quarter symmetry model, the same solution will be obtained as modeling the entire bottle. Because the simulation can take hours to run, simplification of the model, reducing the number of elements, is very beneficial. The applied loading for the simulation was a uniform pressure on the inside of the preform.

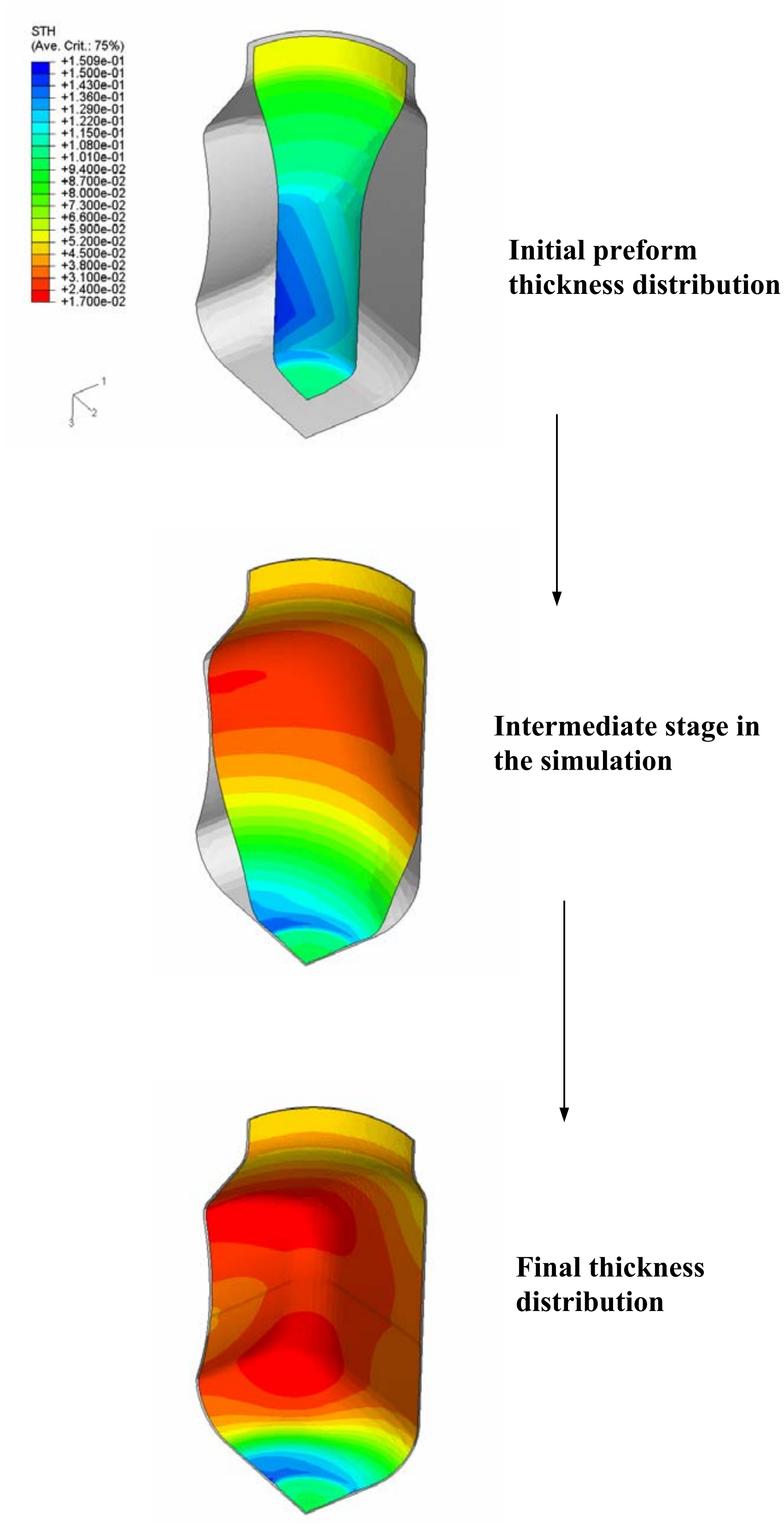
It was assumed that the material continued to stretch until it came in contact with the mold. Contact with the mold cools the material and prevents further stretching. Thus, in the simulation, the shell elements were allowed to stretch until they contact the rigid elements of the mold. By applying a high coefficient of friction between the contacting elements, further stretching of the shell was prevented.

The non-linear finite element program ABAQUS/Explicit was used to run the blow molding simulations. The thickness of the shell at each node is provided as input. The ABAQUS program reports the results of the final thickness at each node.

Correlation of Simulation with Test Results

The first task and perhaps the most difficult prior to optimization is to simulate the existing blow molding process. Usually, existing products of similar size and made under similar process conditions are available to make physical measurements. The predicted results from the finite element simulation should be checked with physical data to develop confidence that the model is accurate.

The bottle shown on Figure 1 was used to check the finite element model prior to optimization. Results of the injection blow molding simulation are shown as wall thickness contour plots on Figure 2. The predicted wall thickness from the simulation was compared with measured values at a number of locations in the bottle as shown in Figures 3,4 and 5. Figures 3, 4 and 5 plot the wall thickness (horizontal axis) versus the distance from the bottom of the bottle (vertical axis) at the concave side, corners, and flat sides of the bottle, respectively.



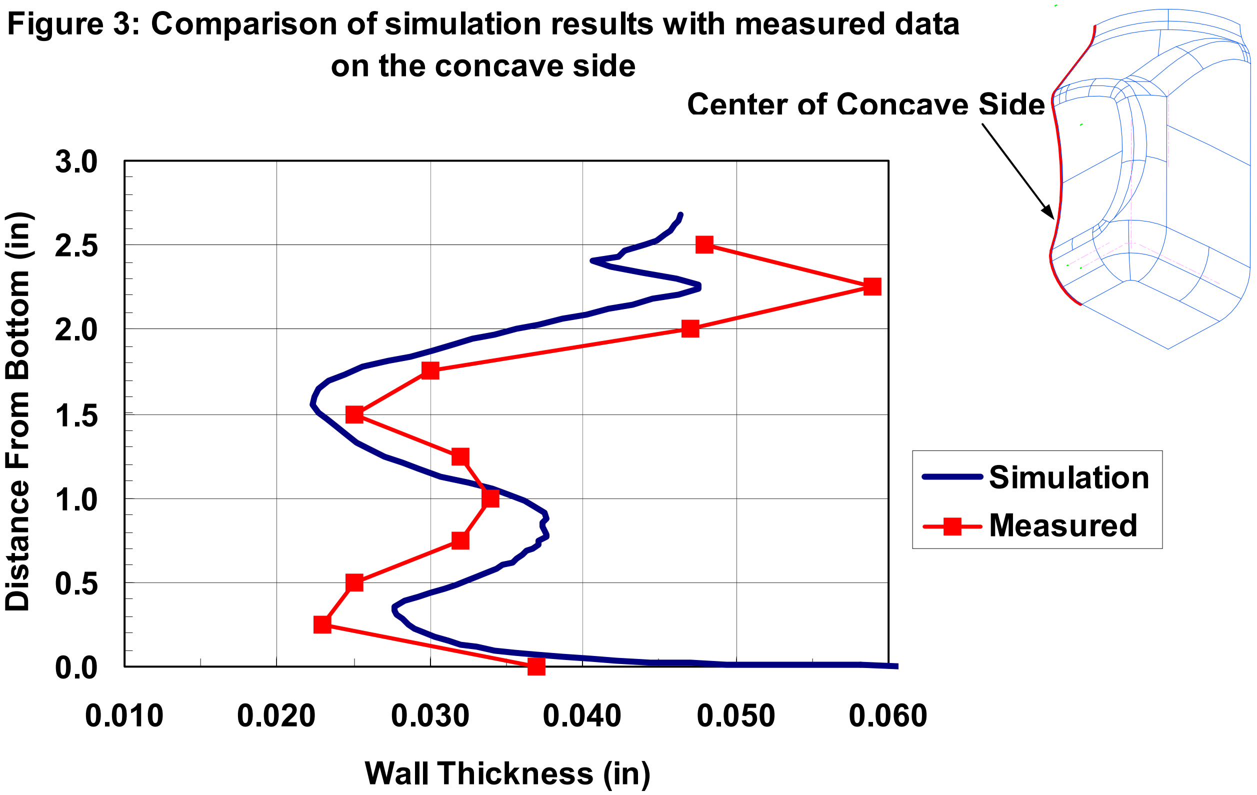
Initial preform thickness distribution

Intermediate stage in the simulation

Final thickness distribution

Figure 2. Contour plots of the wall thickness distribution during injection blow molding simulation.

Figure 3: Comparison of simulation results with measured data on the concave side



…..

Figure 4: Comparison of simulation results with measured data in the corner

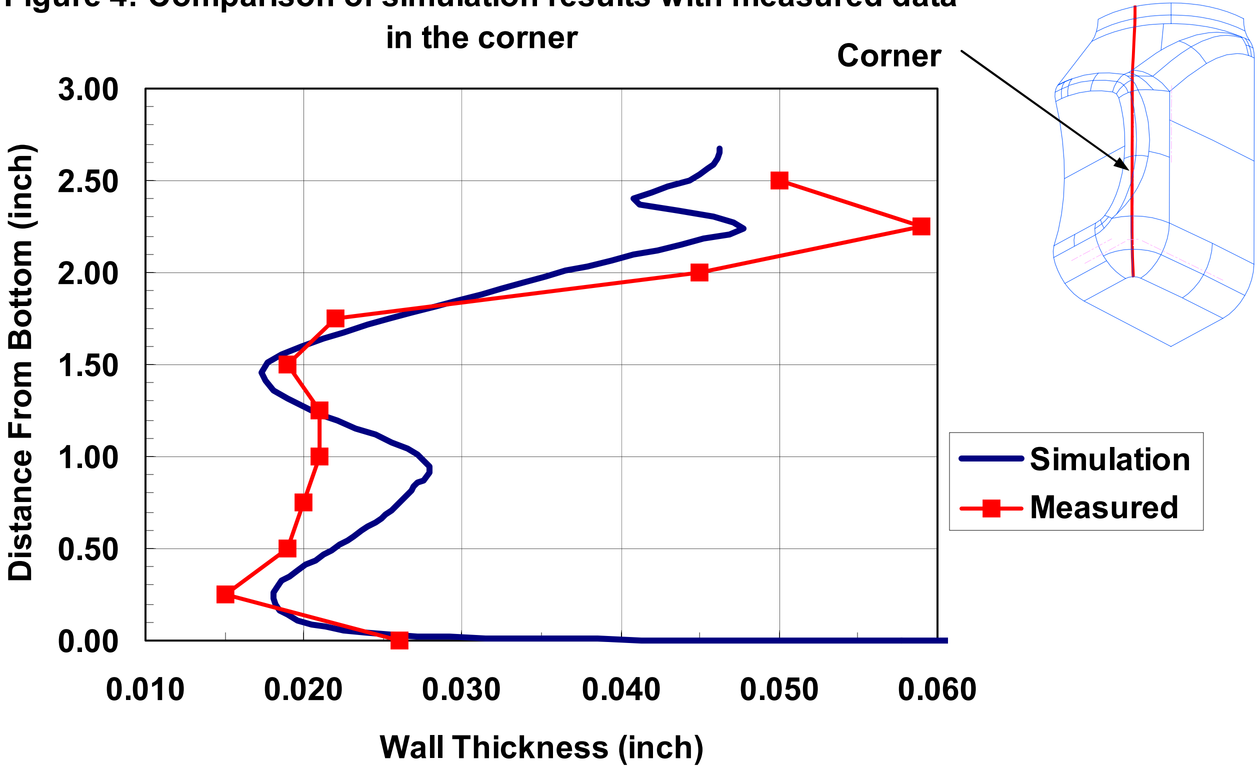
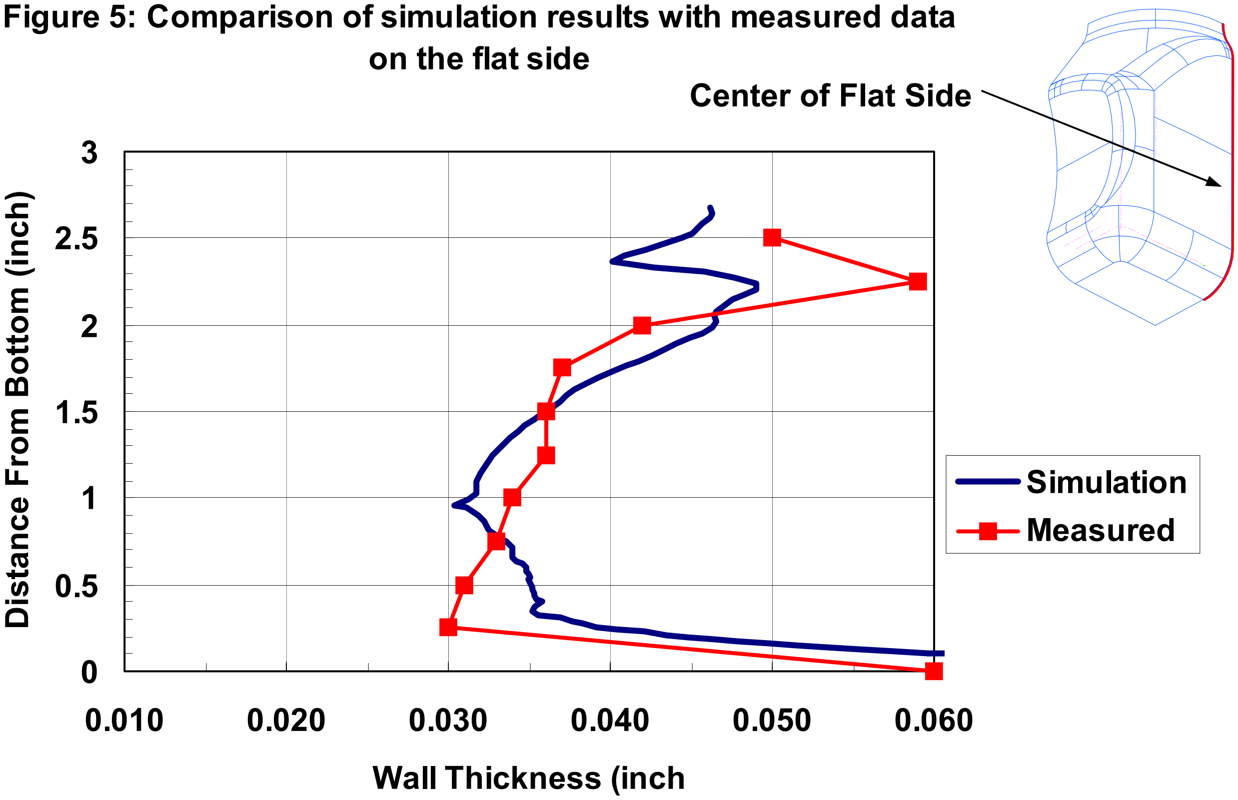


Figure 5: Comparison of simulation results with measured data on the flat side



The predicted thickness values correlate well with the measured values in all areas of the bottle, which allows a demonstration of the optimization technique.

Optimization Technique

The goal of this optimization was to produce a bottle of uniform thickness. However, the target thickness could also be a specified non-uniform distribution. The preform optimization technique is a systematic iterative process that involves a series of simulations. Normally convergence on the optimum is achieved in about ten iterations.

The first iteration starts with a preform of uniform thickness. At the end of the first simulation the thickness at each node in the model is computed. At this point we know three key thickness values at each node in the model:

1. Thickness of the preform at the start
2. Thickness in the bottle at the end of the first iteration
3. The target thickness in the bottle

An equation is written to relate the three values and compute a starting thickness value at each node in the preform (P_2) for the next iteration:

Preform Optimization Example

For demonstration purposes, the same preform and bottle used in the above simulation will be optimized. A uniform wall thickness of 0.35 inches was chosen as the target for the optimization.

Figure 6 shows wall thickness contour plots of the first iteration of the optimization process. As shown in Figure 6, the preform thickness distribution at the start of the optimization was uniform. Thickness contour plots at intermediate and final stages of the blow molding simulation are also shown in Figure 6. As expected, a significant material thinning is observed at the corners of the bottle in the first iteration.

The optimization process required ten iterations to hit the target wall thickness in all areas of the bottle. Figure 7 shows wall thickness contour plots of the tenth iteration of the optimization process. As expected, the optimized thickness distribution of the preform is non-uniform in all directions. The final wall thickness in the bottle is uniform and at the target thickness.

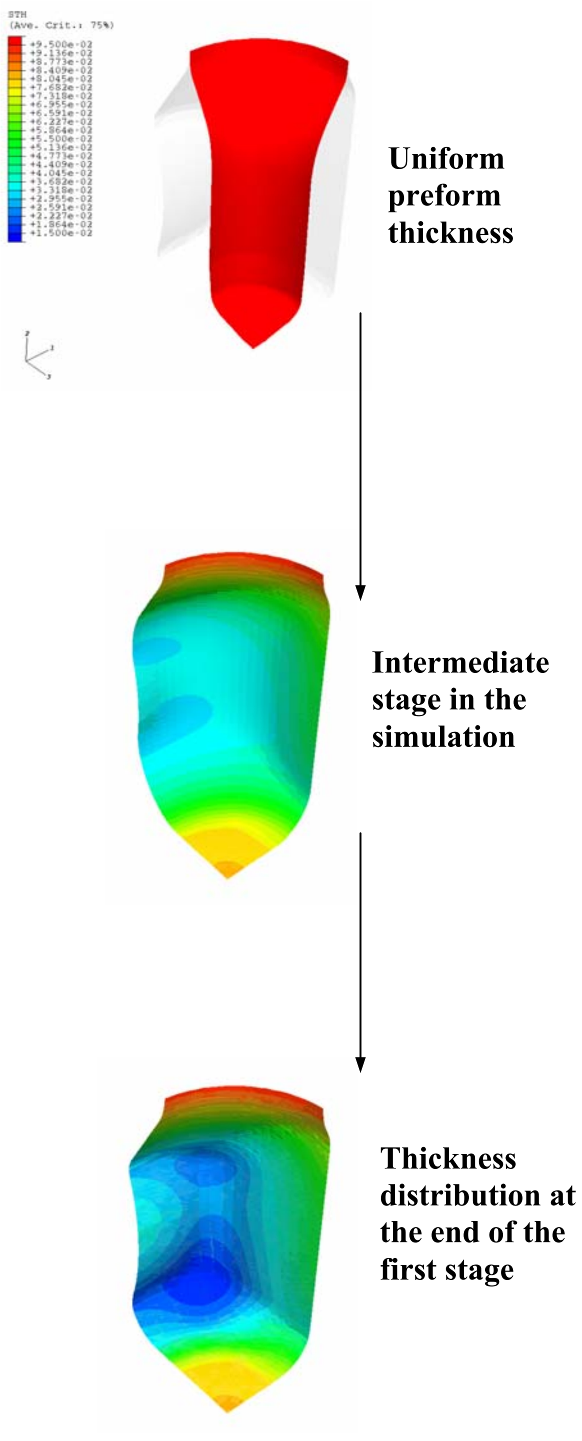


Figure 6. First iteration of the preform optimization process.

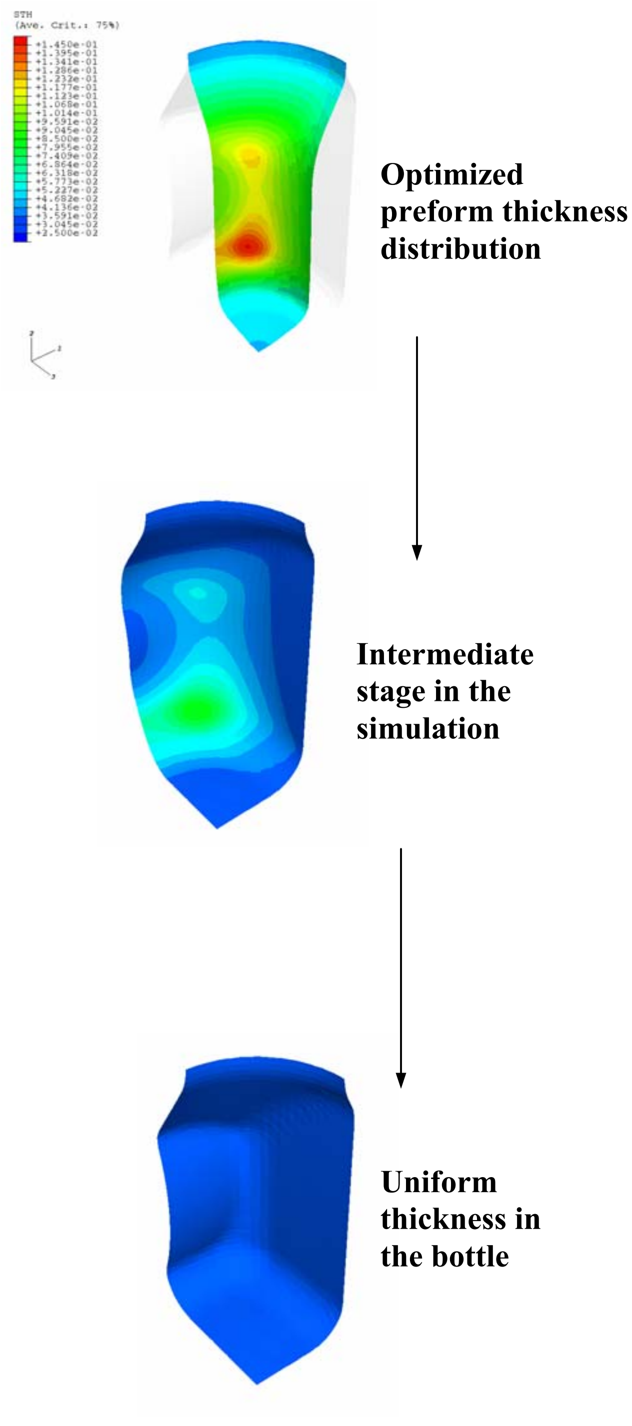


Figure 7. Last iteration of the optimization

The results of the optimization are also shown graphically in Figure 8 (at the corner of the bottle) and in Figure 9 (along the center of the concave wall). The optimized preform thickness distribution is shown at the related locations

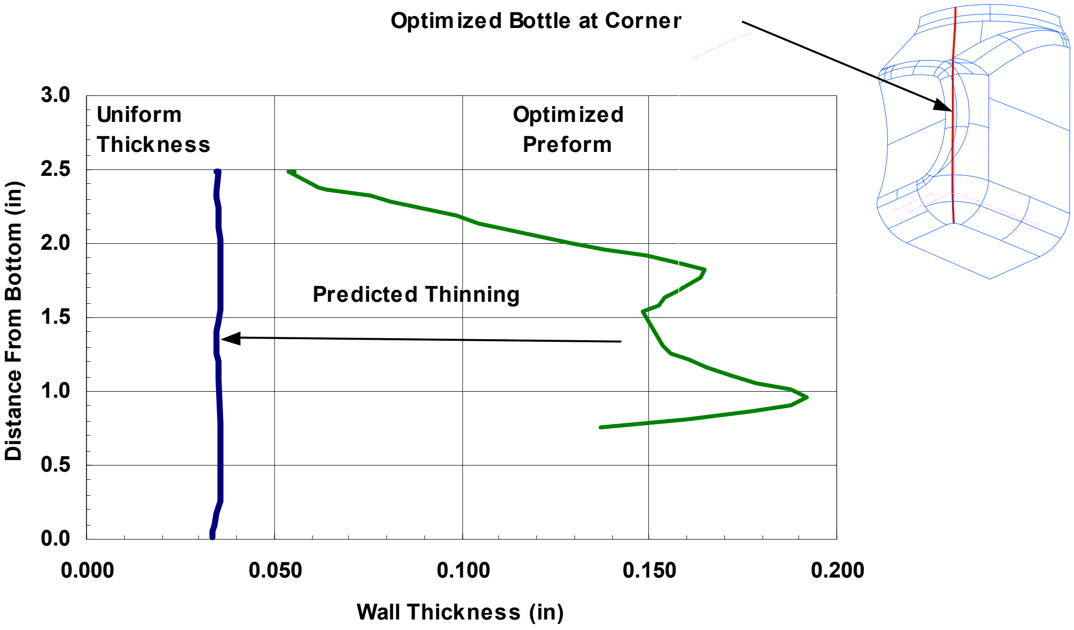


Figure 8. Results of the preform optimization at the corners of the bottle.

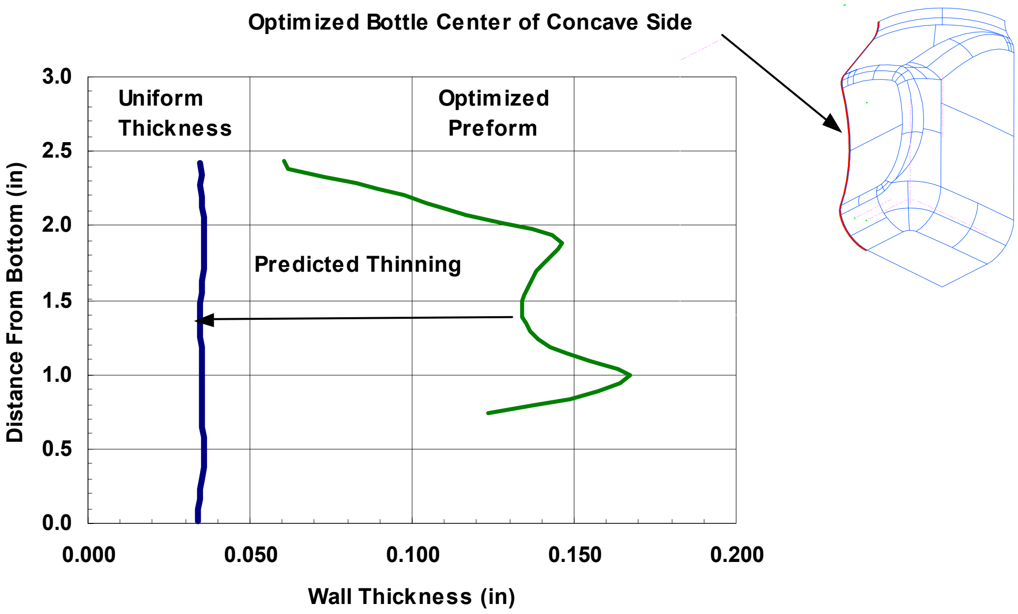


Figure 9. Results of the preform optimization at the concave sides of the bottle.

Summary

An optimization technique for injection blow molded products was demonstrated. It was shown that the technique could be used to design the preform wall thickness distribution such that the final product was at a target thickness. The accuracy of the optimization depends on knowledge of the temperature profile and related material properties of the preform at forming and on the blow molding process variations.

Acknowledgement

The author would like to thank R. J. Abramo Associates, Inc. for their contribution of the bottle and preform geometries and test samples.

References

1. Taylor, C. A.; SPE ANTEC '89 paper, Finite Element Simulation of Thermoforming – Experiments and Analysis (1989).
2. deLorenze, H. G., Taylor, C. A.; Intern. Polymer Processing VIII, The Role of Process Parameters in Blow Molding and Correlation of 3-D Finite Element Analysis with Experiment (1993).