

Aluminum Bottle Forming Simulation with Abaqus

Kunming Mao

DASSAULT SYSTEMES SIMULIA CORP., Central Region, West Lafayette, IN, 47906, USA

Alejandro Santamaria

The Coca-Cola Company, One Coca-Cola Plaza, Atlanta, GA, 30313, USA

Abstract: *This paper presents several modeling techniques for simulating and optimizing aluminum bottle forming using Abaqus/Explicit. Designing and tuning sheet-metal forming tools for aluminum bottles are quite complicated and time consuming tasks. These tasks must take into consideration a number of potential issues, such as the success rate of forming, bottle shape smoothness, bottle load capacity, sheet-metal over-thinning, and metal wrinkling. To shorten the design cycle and reduce the number of forming tool prototypes for the Coke Contour aluminum bottle, simulations with Abaqus served as virtual test grounds to provide valuable insight into the bottle's complex forming processes. Because of large deformation and contact interactions, Abaqus nonlinear capabilities were well suited for these tasks. This paper demonstrates Abaqus forming applications that helped resolve issues arising from realistic industrial forming design and production processes. Three bottle-forming simulations used to predict sheet-metal forming instability, metal over-thinning, and metal wrinkling are used to illustrate the effectiveness of numerical simulations.*

Keywords: *Coke bottle, sheet metal forming, forming instability, FLSD, FLD, MSFLD, surface smoothness, load capacity, die, closure system, thread forming, reverse draw forming*

1. Introduction

A few years ago, The Coca-Cola Company (Coke) decided to revolutionize the beverage industry by developing a lightweight re-sealable aluminum bottle that would use the current aluminum can-making process as its base platform. The making of this bottle involved a number of forming stages, including cupping, cylinder drawing and ironing, die forming, and top-finish forming (threading and curling). Developing the related tools was complicated and time consuming, due to the extensive list of package performance requirements, the manufacturing process robustness, cost optimization, and other factors. Numerical simulations using Abaqus provided strong virtual test grounds for investigating these issues, validating design ideas, and seeking new optimized solutions.

This paper presents several simulation techniques using Abaqus to model three aluminum-bottle-forming processes:

- The cupping forming process that turns a flat sheet of aluminum blank into a cup shape