ROLE OF A CUSTOMIZED MSC/PATRAN INTERFACE IN LOWERING COSTS AND "TIME TO MARKET"

Anthony J. Scott and Donald A. Traub
Owens-Brockway, a unit of Owens-Illinois

In today's competitive market, part cost and timely delivery are the main driving forces. Finite element analysis (FEA) software coupled with custom PCL can enable companies to rapidly verify how geometry changes in a plastic container design will affect its mechanical properties. Owens-Brockway (OB) is in the plastic container market. "Time to market" is very important to its customers. The faster OB can get a new functional product into production, the better chance it has of getting ahead of its competition. One of the biggest concerns in the design of a new container is top-load capability. OB, through the use of a custom FEA tool, can determine the best design of a plastic bottle to maximize the top-load capacity. This will be accomplished by using finite element analysis software that has been tailored to reduce the set-up time. FEA has the reputation for being an analyst's tool, as a consequence many manufacturing companies prefer to construct prototype parts, perform destructive test, and modify the design based on the results. Major cost reductions can be realized by eliminating the need for such a prototyping approach.

Introduction

Blow molding is gaining popularity in the container industry. Compared to other plastic product manufacturing processes it is fairly new. As a result, the blow molding process is still very much an 'art'. This is particularly apparent in the capability of strength prediction of final products. A majority of strength and failure analyses are done after a prototype is made. Changing from more traditional bottle materials such as glass or aluminum to more economical materials (plastics) can cause concerns about product structural stability. Finite element analysis (FEA) provides a means for testing a design under different loading conditions before expensive prototype molds are cut and the container is molded.

This paper describes the successful implementation, into a mixed designer-engineer environment, of a complex non-linear analytical FEA tool. This is achieved through the use of a customized MSC/PATRAN interface specific to the analytical problem. The problem is the prediction of top-load (axial compressive load) capacity of a plastic bottle of any size, material, material thickness, and geometrical complexity.

Background

CAE

Design economy is a powerful driving force in today's bottle manufacturing marketplace, and computer aided engineering (CAE) is now a necessary part of design. CAE encompasses a large field, including computer-aided design (CAD), finite element analysis (FEA) and flow analysis (typically CFD). All of these tools have at least two common purposes — a reduction of product development time and enhanced product quality.

Traditionally engineers have referenced a bookshelf of information for the equations and empirical rules necessary for product design and analysis. Much of this analysis was simplistic and difficult to apply to real world products. In many instances it was, and still can be, cheaper to manufacture and test a prototype part.

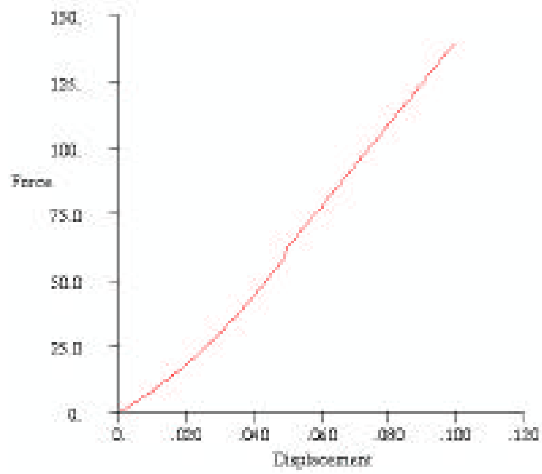
In the bottle industry the use of computer simulation is growing rapidly. FEA is replacing much of the manual computation and providing more accurate predictions of product behavior. As a result the roles of designer and engineer are becoming less distinct as design tools move towards greater and greater analytical accessibility and capability.

At Owens-Brockway CAD modeling software has become the foundation of design and engineering. Once a product design is completed, it can be easily transferred to MSC/PATRAN for the conduct of any one of a number of FEA analyses, one of which is top-load analysis. The relative ease and speed of transference of designs and development of top-load model allows engineers and designers to assess multiple iterations in a relatively small amount of time. In many cases a product with acceptable top-load performance can be devised without the need of a prototype.

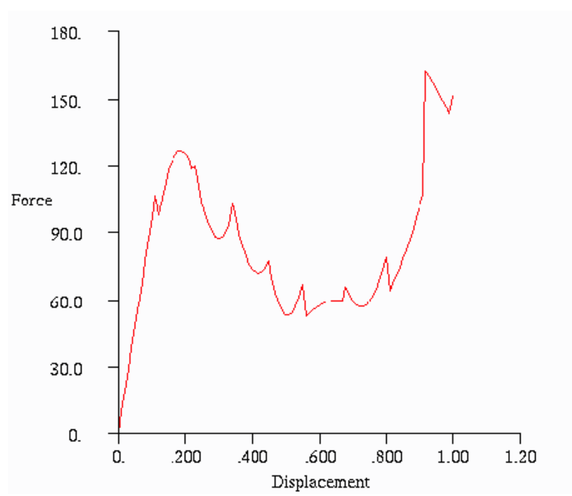
Use of an Explicit Solver

Top-load simulation has proven to be difficult to address because of the high degree of material and geometrical non-linearity inherent in bottle response. Under deflection controlled compressive load most bottles exhibit a number of distinct buckling modes, all of which are of interest to the designer. After numerous trials it was found that in this application the stability of an explicit solver permitted a load-deflection curve to be generated which encompassed the entire load-deflection history. This was typically not possible with implicit solvers, which struggled to meet convergence criteria during rapid stiffness changes (Figure 1). Indeed it was often found that a solution could not be reached with an implicit solver. Thus, though explicit solvers are designed for transient dynamic simulations, there is a proven place for them in highly non-linear static applications.

MSC/DYTRAN is the explicit solver used at OB. It is a three-dimensional analysis code for analyzing the dynamic, quasi-static, and non-linear behavior of solid components, structures and fluids. It uses explicit time integration (Figure 1a) explicit solver P-δ curve with a central difference integration scheme. Though equipped with both Lagrangian and Eulerian solvers, and the ability to couple the two for fluid-structure



interaction problems, only the Lagrangian solver is required for vented top-load simulation (though unvented simulations may



employ coupled F-S analysis).

Figure 1 Implicit Solver

Figure 1a. Explicit Solver P-δ Curve

The smallest element and the speed of sound within it define the time step for an explicit solver. The solver uses a time step small enough to maintain numerically stability in the solution. Explicit codes are perfect for problems that are short in duration, have material, geometrical and contact non-linearities, and have large deformations. This makes MSC/DYTRAN an excellent choice for top-load analyses. The loads are applied for a relatively short period of time and the part undergoes large deformations. The variety of material models makes the solver ideal for plastics. An additional characteristic inherent to the solver is the ability to handle self-contact. This allows the element mesh to contact itself without the individual elements passing through each other - a distinct advantage when random buckling events may take place.

MSC/PATRAN Command Language (PCL)

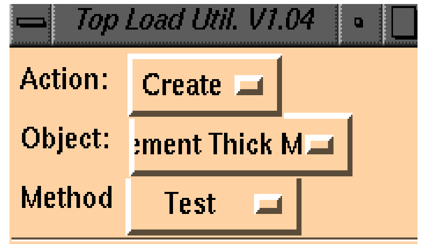
MSC/PATRAN is a modeling system that can be used with many CAD sources and many FE solvers. It is an open modeling environment with an open architecture based upon PCL. Any function or action executed within MSC/PATRAN can be accessed by a similar externally issued command. Once a user learns the language, programs and interfaces can be devised to achieve certain tasks. These tasks may be simple or very complex. The use of PCL allows time consuming and error prone modeling processes to be automated.

Top-Load Simulation

Overview of the Top-Load Interface

A top-load test is characterized by:

1. A plastic bottle whose wall thickness varies over its entire surface.



2. An elastic-plastic material.
3. Two parallel steel test machine platens - one of which is fixed (beneath the bottle), the other (above the bottle) which translates parallel to the bottle axis at a fixed velocity.
4. Bottle displacements which include a series of buckles and may include self-contacting deformations.
5. A load-deflection curve characterizing the bottle response.

In the case of top-load simulation the time consuming and error prone tasks were:

- The assignment of specific material thicknesses to all elements of the bottle model.
- The generation of top and bottom machine platen rigid bodies and their materials, properties, and boundary conditions.
- The generation of contacts between platens and bottle, and bottle-bottle.
- The development of load-deflection curves from load-time and deflection-time results.

Using non-automated methods a moderately experienced user could generate a top-load simulation, with a roughly defined material thickness distribution, in about 3 days.

By generating a customized interface for the generation of top-load simulations the set-up time is dramatically decreased. Also, it makes top-load simulation available to users that might not have the appropriate background to economically conduct the analysis. In addition, manual additions/modifications at deck level have been eliminated completely.

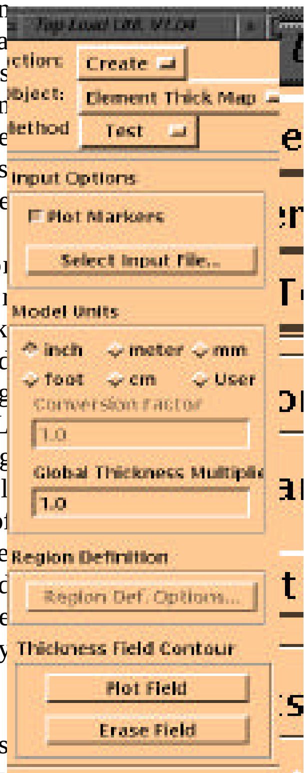
In preparation for use of the simulator the analyst need only define a continuous FE mesh on the bottle geometry (comprising quadrilateral or triangular elements), define a bottle material and a bottle element property set. The bottle is oriented with its axis parallel to global Y.

Figure 2. Main Menu

The simulator comprises a simple drop down menu with MSC/PATRAN style Action/Object/Method structure (Figure 2). The simulator is set-up as an electronic 'checklist' that makes model completion nearly 'fool-proof'.

Element Thicknesses

Normally, a field might be created within MSC/PATRAN in order to define a wall thickness distribution in space. This feature is not available as a standard in the MSC/PATRAN preference for MSC/DYTRAN, nor would such a task be easily achieved manually, requiring either a PCL function describing the mathematical variation of thickness in space or an element field describing the thickness for every element.



The simulator uses a new methodology based upon a linear thermal analogue. The user first provides a text file containing point thickness descriptions for various points in space located on (or near) the bottle surface (Figure 3.). The file has the format of X,Y,Z,t quads (Figure 4.). One spatial point with associated thickness per row. The simulator reads the thickness file as a cloud of points and executes a 3-dimensional interpolation

```
0,.5,.840,.032
1.45,.5,0,.013
.867,.5,.646,.019
-1.45,1,0,.016
-.867,1,.646,.025
0,1,.840,.031
1.45,1,0,.017
.867,1,.646,.022
-1.45,1.625,0,.018
```

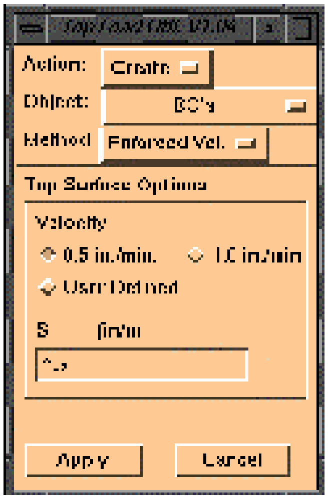
routine (thermal analogue) to determine the thickness at every element centroid. Thicknesses are based upon the four nearest defined thickness values. This results in a spatial FEM field.

The spatial FEM field is compatible with other MSC/PATRAN preferences and is therefore transportable into other analysis classes.

The thickness file can be created manually in a spreadsheet (using empirical wall thickness prediction tools) or imported as output from some inflation simulation software.

Figure 3.
Thickness Mapping Menu

Figure 4.
Section Of Wall Thickness File



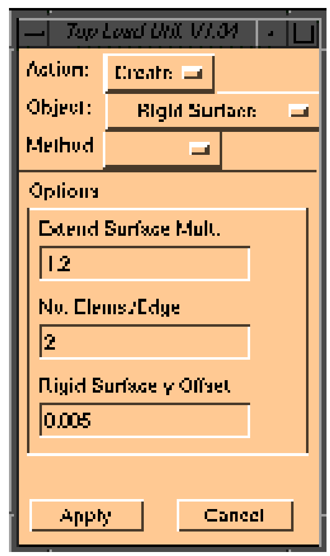
Rigid Surfaces

The simulator automatically creates two rigid surfaces at the top and base of the bottle by determining the limits of the bottle mesh and sizing the surfaces accordingly (Figure5). The lower rigid surface is automatically fixed against translation and rotation. The upper surface is fixed against rotation, and is given a user defined enforced downward velocity (Figure 5a). In this manner the test machine platens are modeled.

Figure 5.
Rigid Surface
Menu

Figure 5a
Enforced Velocity

Contacts



Two contact boundary conditions are generated between the bottle top and bottom n% of its length (n being user defined) and the platens, with user defined friction coefficients (Figure 6).

An optional self-contacting region can also be defined by user selection of a group of elements. This is so those self-contacting regions during buckling can be correctly simulated.

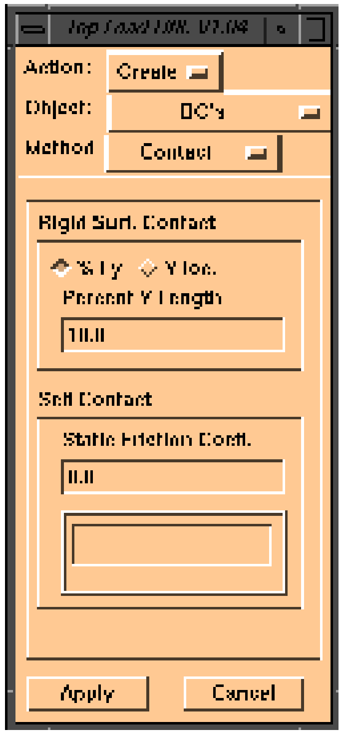


Figure 6.
Contacts Menu

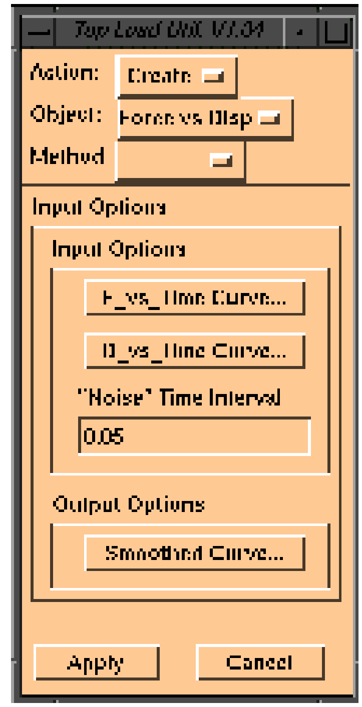


Figure 7.
Load vs.
Deflection Menu

Results Post-Processing

In addition to the use of standard MSC/PATRAN post-processing capabilities, the PCL is even tailored to generate smoothed force versus displacement graphs from the results, which are inherently, time dependent (Figure 7).

Illustration by Example

The following describes the top-load evaluation of a specific bottle using the simulator and MSC/DYTRAN.

CAD Model

The CAD system Pro/Engineer is used for the bottle design. Features that are determined to be insignificant to the analysis are removed from the model. For example, threads and alphanumeric embossed areas are always suppressed. The geometry is checked for quality prior to export. The geometry is exported as a Pro/E part file.

Conventional FEA Model

The Pro/E part file is imported into MSC/PATRAN (Figure 8). If the bottle is axi-symmetric then the geometry may be reduced to a single quadrant. This provides economy of solution time but requires that symmetry plane boundary conditions be applied. The geometry is then meshed with 2-D shell elements of either QUAD4 or TRIA3 topology. Care is taken to ensure that relevant features are adequately meshed, providing for enough elements to resolve high stress gradients without using unnecessarily small elements (Figure 9).

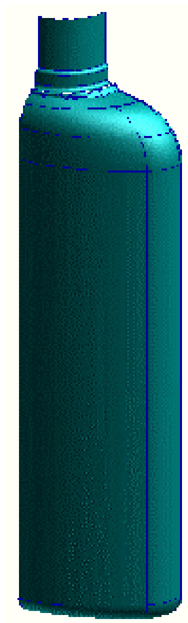


Figure 8.
Imported Geometry

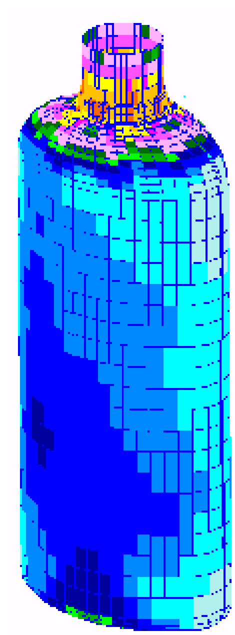


Figure 9.
Meshed Bottle

The current analyses assume that the bottle material is isotropic. The minimum desired properties are those of mass density, Poisson's ratio, and elastic modulus and yield point. In situations where the data permits, it has been found advantageous to use piecewise-linear stress-strain curves to describe the yield response. It is possible to describe the material as 2-D orthotropic, though in most cases the test data is not available to accurately achieve this. The material is described in MSC/PATRAN in the conventional manner.

It is assumed that regions of the bottle will suffer plastic deformation and that significant warping of elements may occur. For this reason the KeyHoff formulation employing 5 through thickness integration points is used.

Use of the Simulator

Once the bottle is meshed, and the material and element properties assigned, the simulator is used to complete the model. The element thickness map is applied (Figure 10). The rigid surfaces (and their materials and properties) are created at the top and bottom of the bottle (Figure 11).

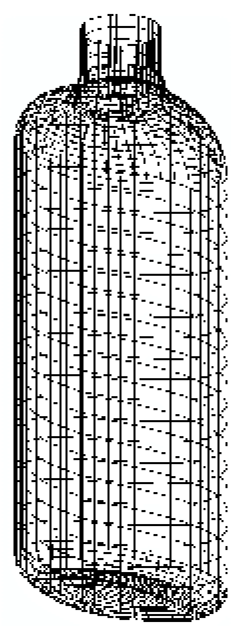


Figure 10.
Mapped Element
Thicknesses

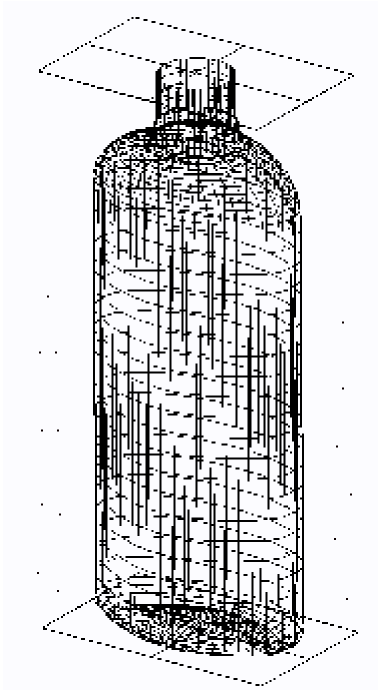


Figure 11.
Rigid Surfaces

All load and boundary conditions are applied, including rigid surface constraints, velocities, and contacts (self and surface to surface) (Figure 12).

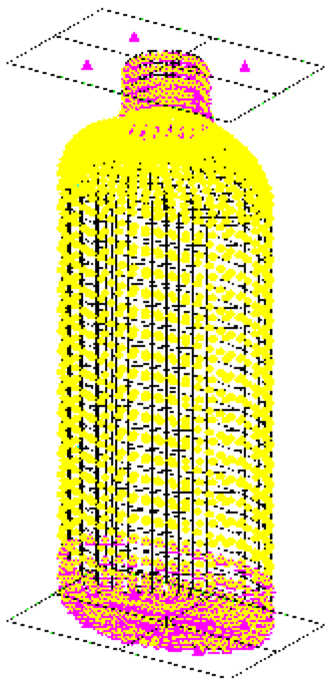


Figure 12.
Contacts

The simulation was devised to achieve a 0.125in top platen displacement at a velocity of 3.33 in/sec. Once complete the model is sent to MSC/DYTRAN for solution. When analysis is complete the simulator is used one last time to automatically create a force v displacement graph of results (Figure 13).

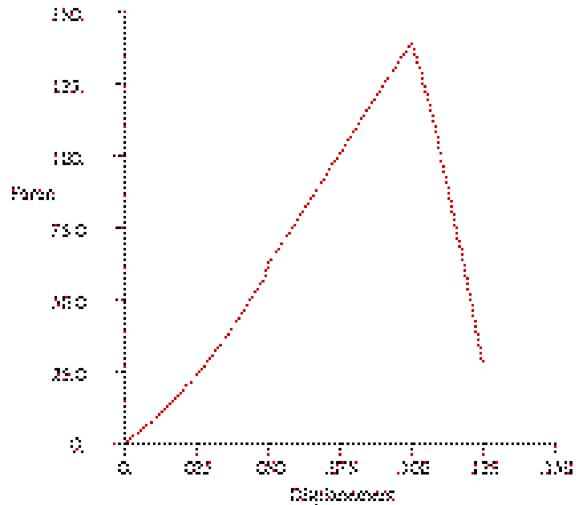
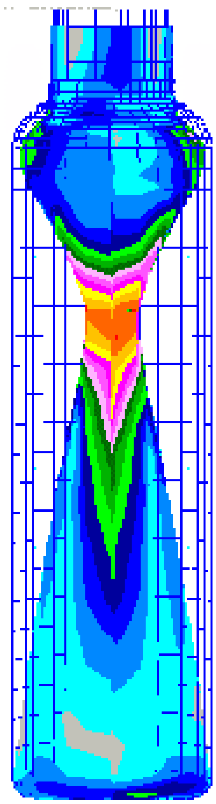


Figure 13
Load vs. Deflection

Results

The analysis indicated that the bottle would buckle just below the shoulder region (Figure 14) with a total displacement of 0.10 in. and a force of 139.1 lbs. (Figure 13). The actual bottle test indicated bottle failure in exactly the same region as the predicted computer model at a load of 117.4 lbs. And displacement of 0.12 in. (Figure 15). The head speed of the top load tester was 1 in/min. The difference between actual and predicted peak loads is approximately -15%.

Figure 14.
Deformed Shape



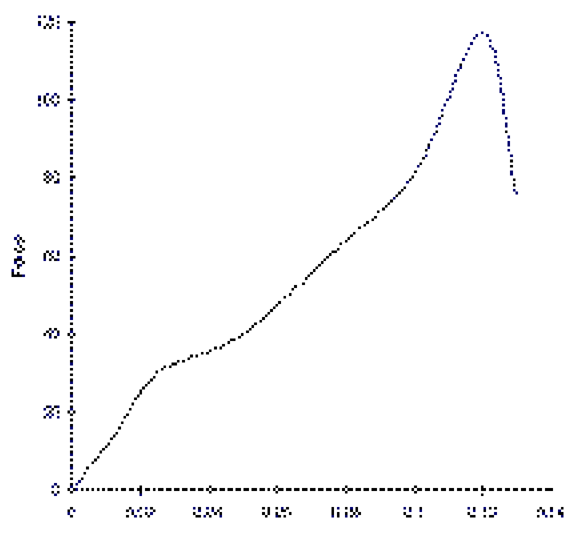


Figure 15
Actual Bottle

The FEA analysis was done before the bottle molds were constructed. So, there were not any bottles available on which to take measurements. The three most important parts to an analysis are: Accurate wall thickness distribution; Definition of material properties; And geometry definition. Without an actual specimen the former two of these factors can be difficult to predict.

The bottle wall thickness is determined by scaling methods applied to bottles of similar gram weight and geometry. This process does not predict possible molding problems such as uneven distribution of material.

The second and less common is the material property definition. If the resin used is a new grade the information used in the analysis must be provide by the supplier. In many cases the properties provided are obtained from samples that are injection molded. The values from these tests tend to be higher than those taken from blow molded parts. The molecules in injection molded parts are aligned in the direction of testing. The molecules in a blow-molded part are stretched bi-axially. This can cause such properties as the yield stress to be much lower.

Conclusions

- The top load simulator is a great asset in many areas. It is helping us reduce the gram weight in many bottles prior to first production. Additionally, bottles in production at higher gram weight because of un-quantified top load concerns can now be optimized for top load to save material costs. The simulator allowed us to quickly identify the areas of a bottle that could be thinned, thus reducing material costs.
- The simulator has reduced the amount of time required for analysis. It used to take 2 days to setup a model. It now takes less than an hour. Additionally, with the implicitly solved model there was no confidence of reaching convergence beyond initial (and lesser important) buckle. The explicit solver will always solve to completion and capture the entire load-deflection response.
- The simulator opens the door to many design engineers that may not have the

experience to run FEA software or the time to be subjected to problem specific training. The check list style makes it easy to define a correct. A designer requires only a few days of background training to be able to complete a simulation, therefore company does not need to invest time and resources into one dedicated FEA analyst.

- Reduced simulation time, solve time, and increased confidence in results make it possible to run many more iterations. This allows designers and customer to be more creative. They can try ideas that were originally discarded because of concerns about top load capacity and the inability to simulate this response.
  The amount of prototyping time is also reduced. Many designs can be eliminated by the analysis. The most direct impact is on unnecessary prototype molds.
- The use of the simulator is a powerful marketing tool. Being demonstrably a user of new technology results in enhanced customer confidence. FEA simulation has associated with it an aura of believability and a reputation for accuracy.

Acknowledgments

- A special thanks to Stuart J. Wright and MSC for their technical support.