Turkish J. Eng. Env. Sci.
31 (2007) , 189 – 197.
© TÜBİTAK

A New End-Closure Design for DOT-4BA Propane Cylinders

Yasin KİŞİOĞLU
Department of Mechanical Education, Kocaeli University, Umuttepe, 41380, Kocaeli-TURKEY
e-mail: ykisioglu@kou.edu.tr

Received 31.03.2006

Abstract

This study presents a new end-closure design, using a numerical approach, to eliminate the buckling that occurs at the bottom end of DOT-4BA propane cylinders. When the bottom heads are subjected to test pressure they often buckle, which causes instability of the propane cylinders. That instability is the cylinder's loss of upright standing. The buckling was predicted using both experimental and numerical approaches, and a proposed solution to prevent the failure by increasing the material was previously reported. An alternative end closure design is developed in this study to eliminate the end buckling without changing the material. In order to model the new end, a computer-aided finite element analysis (FEA) design tool was employed. Two different FEA models, 2D plane and 3D shell, were created and simulated in nonlinear conditions. The results obtained from the simulations are compared to previous corresponding studies.

Key words: Buckling failure, Alternative design, Preventing failure, DOT cylinders, Propane cylinders, FEA modeling.

Introduction

DOT-4BA propane cylinders regulated and approved by the US Department of Transportation (DOT) can be filled and used for commercial, industrial, and home applications. These cylinders are nominated by different labels, such as DOT-4BA, DOT-4BW, and DOT-8AL, and are generally used for storing and transporting propane gases; therefore, they are sometimes called propane cylinders. They are portable, refillable, low pressure cylinders with a service pressure $<$ 500 psi (3.45 MPa) and are used worldwide in very large quantities, about 3×10^6 annually. They are produced in different sizes ranging from 12 lbs (5.4 kg) to 142.9 lbs (64.8 kg) with water capacities. The nominal dimensions of the cylinders can be defined by the ratio of shell thickness (t) to internal diameter (ID), which is in the range of $0.008 < t{:}ID < 0.02$, along with different cylinder lengths (Kisioglu et al., 2006). A critical DOT regulation specifies the minimum test pressure (TP) the cylinders must withstand without experiencing any measurable permanent buckling or significant failure. When the cylinder is pressure tested, the cylinder volume must not exceed 10% of the initial volume. They are designed based on the TP, which is 1.5 times the service pressure (SP). The SP is the working or operating pressure that the cylinders are filled to and used in their application areas. Both the SP and TP are specified and regulated by the DOT code.

Studies in the available literature regarding alternative head design for pressure cylinders are limited. Typical studies consider only ideal shells with perfect geometry and linear material properties. In general, they report instability pressures based on a large deflection analysis and optimum geometrical parameters of the shells. The design parameters of an end closure (head) were optimized by Kisioglu et al. (2006) to prevent the buckling of propane cylinders, using both experimental and finite element analysis (FEA) approaches. The elimination of refrigerant cylinder instabilities by designing an optimum head were also studied by Kisioglu et al. (2005), using both experimental and numerical techniques.

The weight minimization of the gas pipelines and junctions where the highest transition stresses occur was studied by Harte et al. (2003) using the computer code BOSS. The design optimization of composite submersible pressure hulls under a shell buckling strength was studied by Liang et al. (2003), who used the hybrid genetic algorithm to formulate the optimum thickness. Kisioglu et al. (2001) studied and determined the burst pressure and failure locations of refrigerant cylinders using both experimental and FEA approaches. The effects of knuckle size and length of the cylindrical flange of shallow torispheres with a sharp knuckle pressurized externally were examined by Blachut (1998), both numerically and experimentally. The buckling pressures of perfect elliptical toroids and yield stresses were studied by Combescure and Galletly (1999), who used the shell buckling computer codes BOSOR5 and INCA.

The objective of the present study was to develop an alternative head design for propane cylinders that prevents end buckling. A new end closure design was built using 2 different computer aided models, 2D plane and 3D shell. A finite element design tool was employed to simulate the models and predict the end buckling. The nonlinear material properties of the head were applied and the initial design parameters were used. The new end closure was subjected to incremental uniform external pressure, dependent on loading time. The model was successfully developed and the results were compared to previous studies (Kisioglu et al., 2006), according to buckling pressure and deflection.

Four different head designs are considered and their buckling results are compared in this study. The differences among the end models are characterized based on their design parameters. The initial head design has the same wall thickness as the cylindrical shell. The cylinder manufacturer had selected double thickness for the current end model to eliminate the buckling problem. The design parameters of the initial head were optimized by Kisioglu et al. (2006) to prevent the buckling, which can be called optimum end. Finally, the new end model was developed in this study using the different end geometry.

A Brief Description of the Buckling Problem

A DOT-4BA refillable cylinder consists basically of 2 main components, a torispherical bottom cylindrical shell and a convex end closure, as illustrated in Figure 1. They are welded together at the bottom to form the cylinder. When viewed from the cylinder bottom, the end closure is a torispherical convex shape, regarding the pressure contact surface (Figure 1). Cylinder manufacturers strive to meet the TP specified by the DOT code, while minimizing the steel content (material cost) of the end enclosures. The head was currently designed using experience and assumptions of constant sheet thickness and material properties. A case study at a cylinder manufacturer showed that a significant percentage of cylinder heads buckled under TP during the experiments. Buckling of the head means that its shape inverts, changing from convex to concave, as shown in Figure 2. The result of buckling is that the cylinder experiences a significant increase in internal volume and also loses its ability to stand flat on its foot ring. In these studies, both the cylindrical shell and head were assumed to have the same thickness, t, as seen in Figure 1. One hypothesis is that production cylinder failures are the result of variations occurring in sheet steel thickness and mechanical properties. Variations can exist in the incoming sheet steel, but can also develop during the drawing process. In a case study, the solution to the buckling problem was to double the thickness of the sheet steel used for the current head. This solution eliminated the buckling, but was expensive in terms of material cost, especially given the large number of cylinders manufactured.

Kisioglu et al. (2006) studied and predicted the buckling of the initial and current end closures, using both experimental and computer-aided analytical approaches. The design parameters of the initial head were optimized to prevent buckling with a computer-aided numerical technique. Based on their optimization results, the material (weight) of the optimum head was increased by about 30% compared to the initial model. The optimum model obtained was better than the current design, which was selected with double thickness ($2t$) by the cylinder manufacturer. However, in this study, the design parameters of the initial head are not changed; the new end closure geometry is developed by increasing the geometrical and material stiffness.

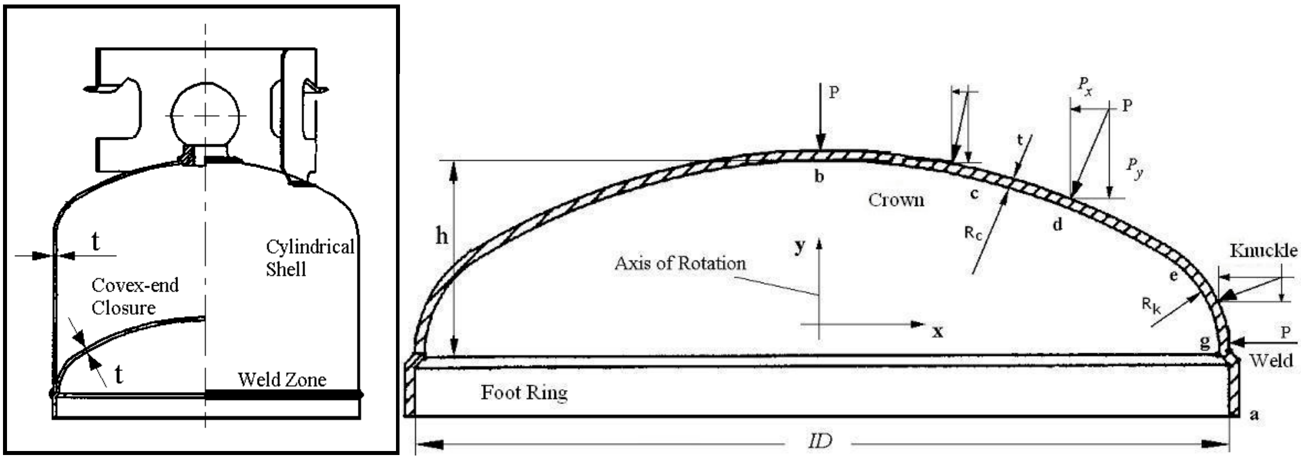

Figure 1. The DOT-4BA cylinder and its initial end closure geometry.

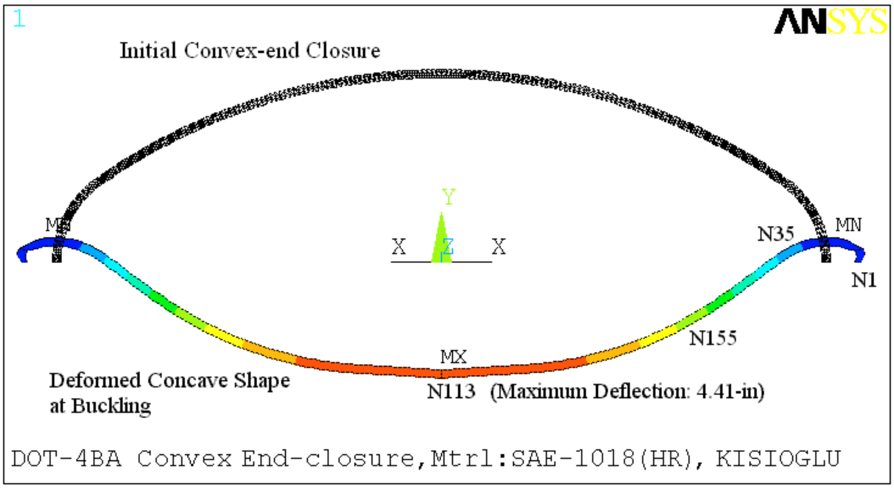

Figure 2. The buckling concave shape of the convex end closure (Kisioglu et al., 2006).

Design and Manufacturing of the New End Closure

A new end closure design is developed with 4 small dimples generated at the bottom end closure to prevent buckling (Figure 3). The 4 dimples are designed and located circumferentially at a dimple location (DL) on the crown of the head. The DL is defined as a function of θ and R components in the polar coordinate system. The dimples provide the necessary geometrical and material stiffness at the bottom to resist buckling. When the dimples are drawn, the material and the geometrical strength are enhanced. The DL is selected at a specific place where the buckling begins, which was predicted by Kisioglu et al. (2006). The dimples are designed with a dimple radius (R_d) and joined to the head with the same fillet radius of R_d since the dimples and head are not coplanar. The selection of R_d is conducted at a certain ratio of R_c:$R_d \cong 10$ to lessen the concentration of stress between the dimples and crown region.

The initial head was manufactured using the traditional blanking and deep drawing processes with a die and punch tools. The head was made of SAE-1018 hot rolled steel, which was selected by the cylinder manufacturer. To manufacture the new head with 4 dimples, similar punch and die tools can be used. The punch can be created with 4 spherical bumps (ball noses) at the end with a radius of R_d. Similarly, the counterpart can also be created with 4 spherical cavities inside the die. The construction of the new head with 4 spherical dimples can be developed during the drawing of the cylinder head, as illustrated in Figure 3.

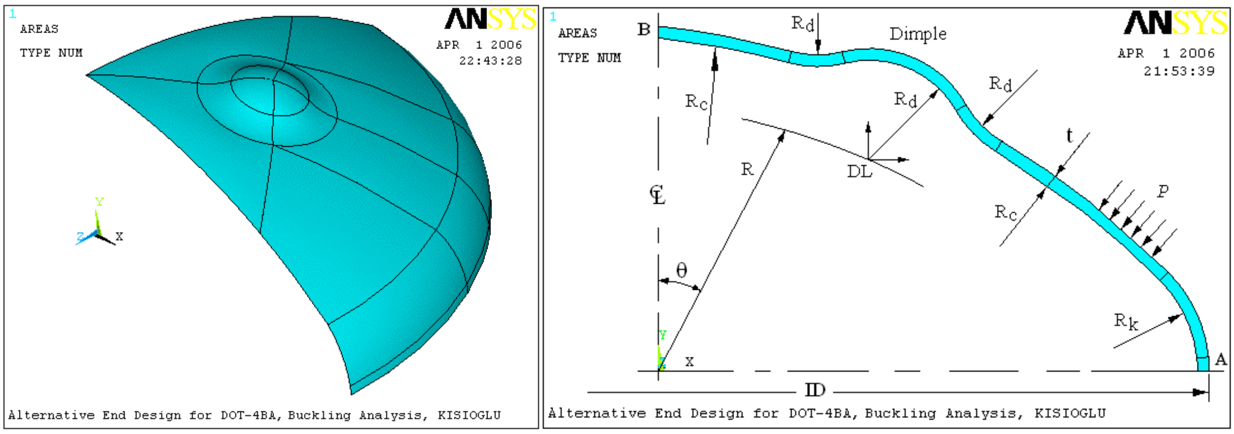

Figure 3. The new end closure model and dimple geometry.

Computer-Aided Modeling for the Buckling Analysis

The finite element-based computer code, ANSYS, was employed to model and predict the buckling at the bottom head of DOT-4BA propane cylinders. We created 2 different types of head models, 2D plane and 3D shell, which were taken into account in the computer-aided simulations. In the FEA and modeling processes, the nonlinear material and uniform geometry, except the weld zone, were taken into account using the ANSYS functions. We developed 2 suitable nonlinear FEA models in consideration of the plane-strain conditions. The initial design parameters of the end closure (see Figure 1), except the foot-ring, were considered to create both 2D and 3D nonlinear FEA models. Half-symmetry of 2D and quarter symmetry of 3D forms were generated using the initial end thickness (t), as seen in Figure 3. The geometrical models were divided into different specific areas to allow the development of an appropriate finite element mesh generation. The area divisions were also precisely specified considering the finite element shape functions. Based on the generated geometrical models, suitable boundary and loading conditions were applied, and the appropriate finite elements were selected.

Selection of finite elements, and boundary and loading conditions

It was necessary to select suitable finite structural plane and shell elements in consideration of the specifications of both 2D and 3D FEA models. Considering the element shape functions, PLANE2 and SHELL181 elements were selected for the 2D plane and 3D shell models, respectively. The PLANE2 has 2 degrees of freedom at each node, which are translational in x and y directions. The SHELL181 is specified with 6 nodes and 6 degrees of freedom at each node. Both elements are well suited for large strain, large deflection, plasticity, plane-strain, and nonlinear applications (ANSYS User Manual, 2003).

The axisymmetric boundary conditions were considered and applied to the 3D FEA model, considering the head geometry and the external uniform pressures. Both external pressure and end geometry are symmetrical with respect to the head axis of rotation (Figure 3). The symmetric boundary conditions were applied to the nodes located at point B. The nodes located at point A were restricted to the translational and rotational displacements in y and z directions, respectively (Figure 3).

The end closure was subjected to incremental external uniform pressure until the TP value of the cylinders was reached. Initially, the external pressure was applied uniformly with 10 psi (0.69-Mpa) per step during the loading (Figure 4). The figure has 2 curves, “Load incrm” and “Mx Deflect”. The Load incrm and Mx Deflect curves were plotted as a function of the load using the left-handed and right-handed y-axes, respectively. The loading increment fluctuates between 450 and 1300 loading time, shown with points “a” and “b”, respectively (Figure 4). The loading increment becomes stable between points “b” and “c”, and then the load is decreased until the buckling take places (point “d”), which is called buckling time.

Development of nonlinear FEA models

The FEA models were developed using the selected appropriate boundary conditions and the finite element as seen in Figure 5. The FEA was conducted in quasi-static analysis in plane strain conditions along with the nonlinear material and the uniform geometry. The isotropic work hardening option using the von Mises yield criteria was applied to define the nonlinear material. In order to apply the true stress-strain data into the ANSYS, the table point data command, “tbpt,defi”, was performed for each point. To apply the non-uniformity of the model to the weld zone, the wedge function (Kisioglu et al., 2001) procedure was applied.

The material properties of the end closure were specified using the well-known tensile test technique. The tensile test specimens were cut out from the end closure and weld zone (Kisioglu et al., 2006). The material properties were defined with corresponding true stress-strain data, including the weld zone properties, as illustrated in Figure 6. The figure has 2 curves, Weld and End closure, representing the true stress-strain data of the weld zone and the head, respectively. The material properties were used and input into the ANSYS program using the TB command. The multi-linear isotropic hardening option supporting large strain analysis was selected and applied. In addition, the FEA model was also created with non-uniform geometry, using the thickness variation at the weld zone circumferentially (Figure 5).

Assumptions for the modeling process

At the initial development of the FEA models, some assumptions were made to predict the buckling. First, since the end closure was constructed as very shallow (see Figure 1), the material properties of the head, including dimples and fillet zones, remain constant after the drawing process. The material properties obtained from the crown region were applied homogeneously for the entire head, including the

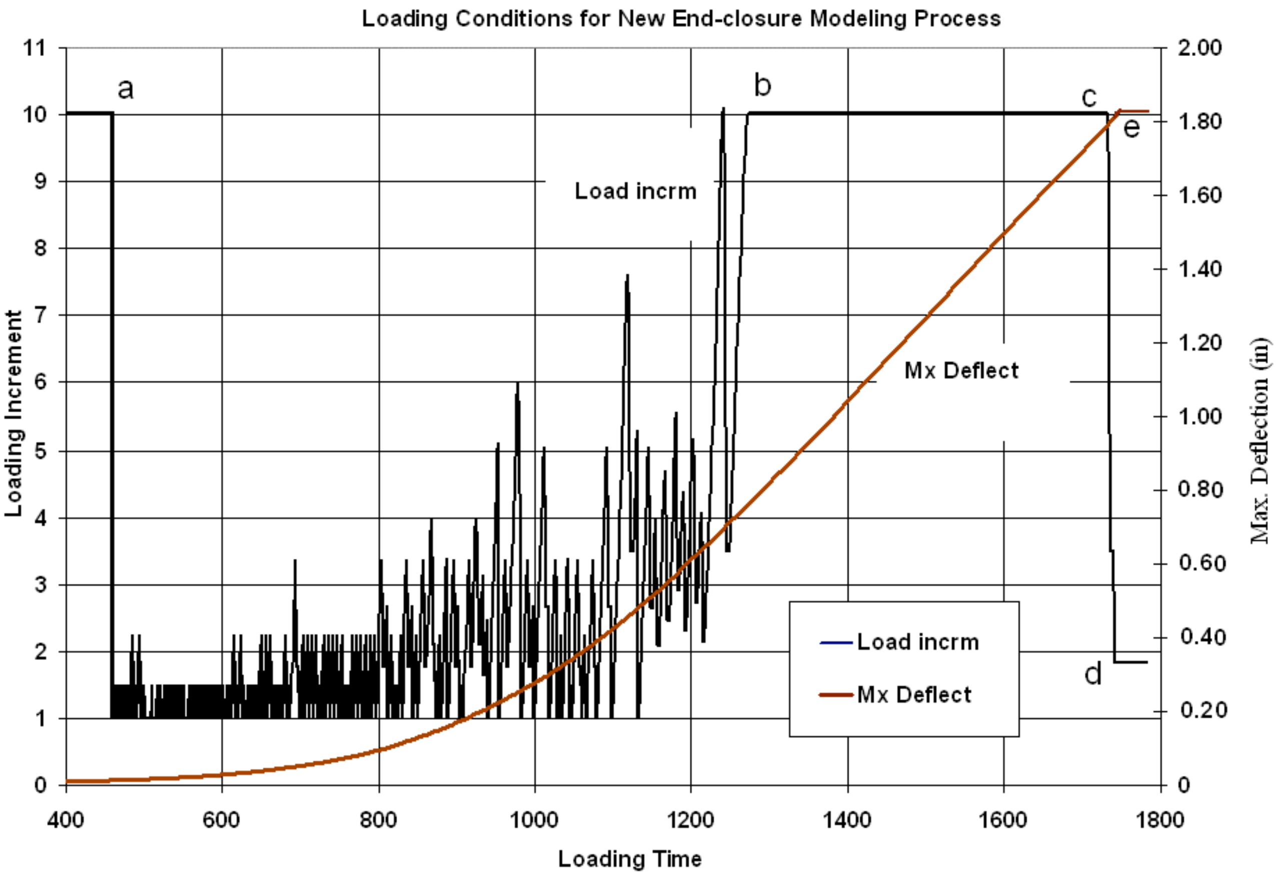

Figure 4. Loading conditions and maximum displacements of the entire new model.

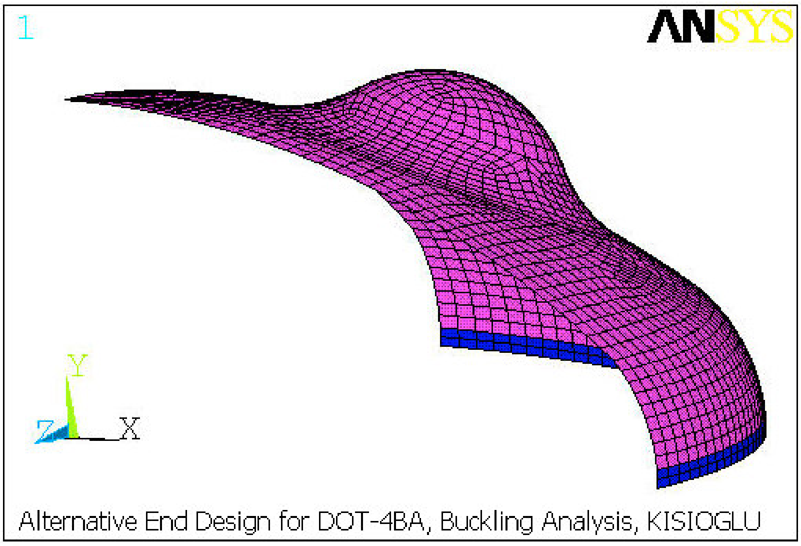

Figure 5. The 3D axisymmetric FEA shell model of the new end closure.

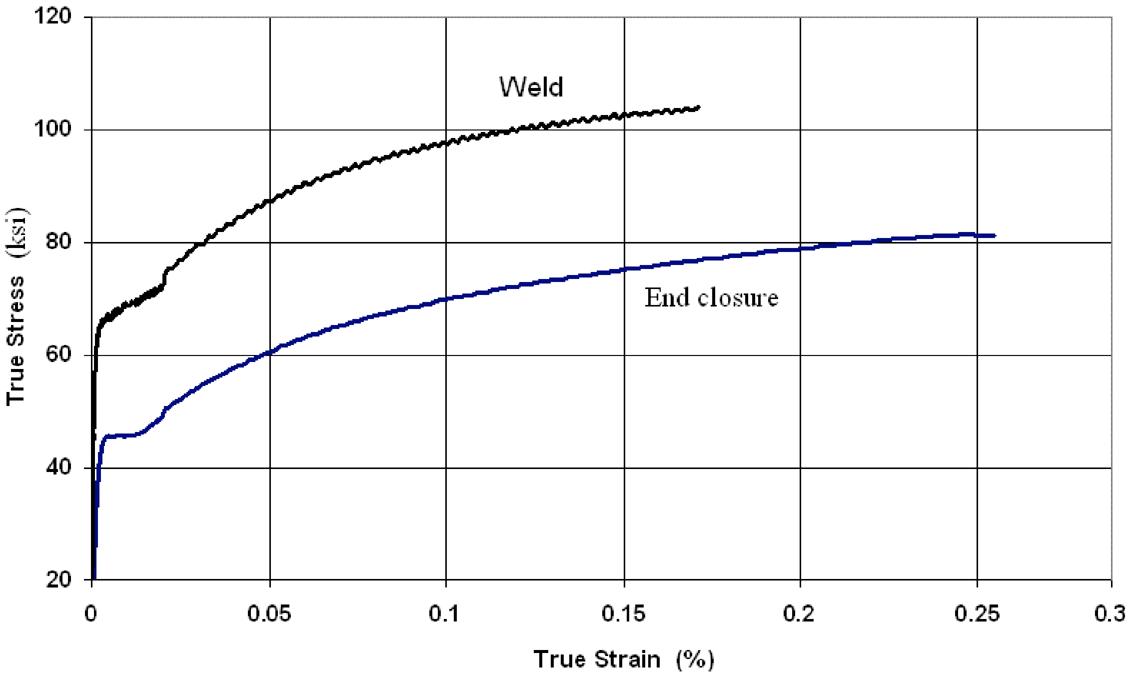

Figure 6. The true stress-strain curves of the weld zone and end closure.

knuckle region. Second, the thickness of the head also had no changes after the drawing, so that the initial thickness (t) of the blank sheet was used in the FEA (Figure 5). Finally, since the propane cylinders were being heat treated (stress relief process) after finishing all welding and assembly processes, an assumption was made that there were no residual stresses on the body of the head.

Prediction of the Buckling

To understand the buckling event predicted by the FEA simulations, the values of 3 variables, load (pressure), loading increment, and deflection, can be considered. The FEA model of the head is subjected to uniform incremental external pressure until buckling occurs, as seen in Figure 7. When the incremental pressure reaches a critical value (P_{cr}), a maximum critical deflection (U_{max}) occurs at the middle of the crown (see Figure 1). The U_{max} is the deflection at point "c" that happens just before the buckling at point "d" (see Figure 4). When the loading increment ($\boldsymbol{n}$) is increased one more step, from $\boldsymbol{n}$ to $\boldsymbol{n}$+1, the bottom head buckles (Figure 7).

The buckling can also be described in terms of the buckling deflection. When the loading increment increases from $\boldsymbol{n}$ to $\boldsymbol{n}$+1, the U_{max} changes to the buckling deflection (U_{buc}). That is, when the pressure load reaches the P_{cr} (at nth loading step), the U_{max} is about 0.26 in (6.60 mm). When the pressure increases to buckling pressure (P_{buc}) at the n+1th loading step, then the U_{buc} becomes about 1.34 in (34.04 mm), as illustrated in Figures 7 and 8. The 3D shapes of both the initial geometry and the inverted models of the head are shown in Figure 7. In addition, the nodal deflection behaviors of the selected nodes of the model (Figure 7) are plotted as a function of loading in Figure 8. The figure shows that the buckling occurs at about 1743 psi (12.02 MPa), when the displacements of the selected nodes are suddenly changed.

Comparison of the New Model with Previous Models

The buckling at the bottom head of the propane cylinders is eliminated using the developed nonlinear FEA models of the new end closure design. The buckling results of the new head design are compared to the previous models studied by Kisioglu et al. (2006), in terms of buckling pressure, in Figure 9. The figure has 4 curves, Initial Model, Current Model, Optimum Model, and New Model, which represent the different end designs, initial, current, optimum, and new models, respectively. The New Model curves and the rest are plotted as a function of the incremental external load using the right-handed and the left-handed y-axes, respectively.

The P_{buc} value of the initial end model having the same thickness (t) as the cylindrical shell (see Figure 1) was lower than the TP (Figure 9). As can be seen from the figure, the P_{buc} of the initial end model was about 745.81 psi (5.14 MPa) at point "a", which was predicted by Kisioglu et al. (2006). The P_{buc} of the optimum head was obtained at about 1080 psi (7.45 MPa) at point "b", using the optimum end design

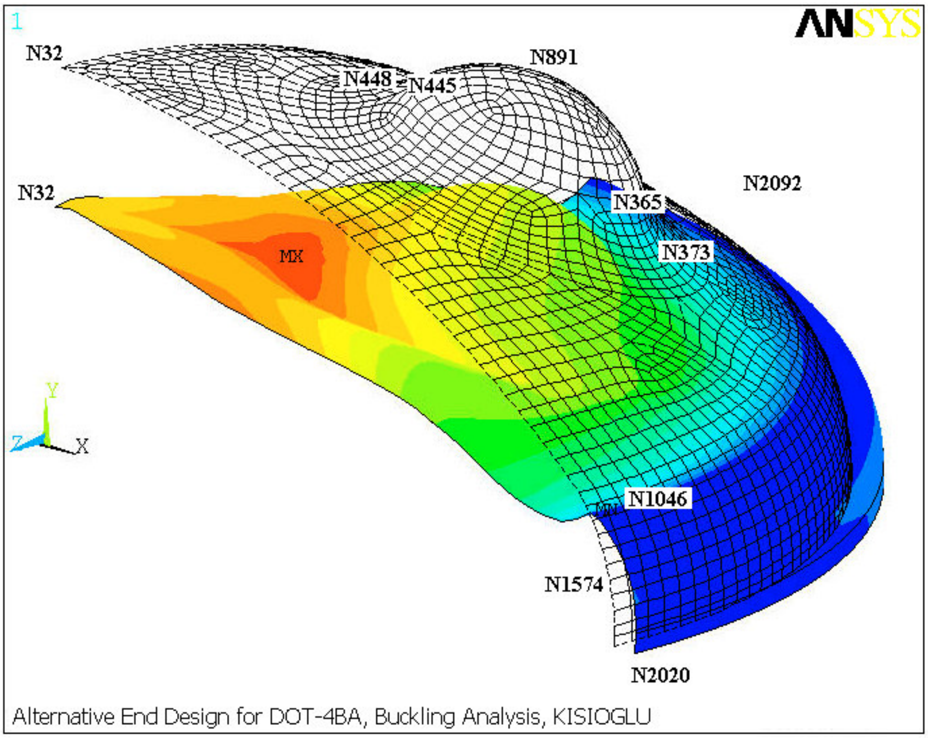

Figure 7. The deflection behaviors of the selected nodes.

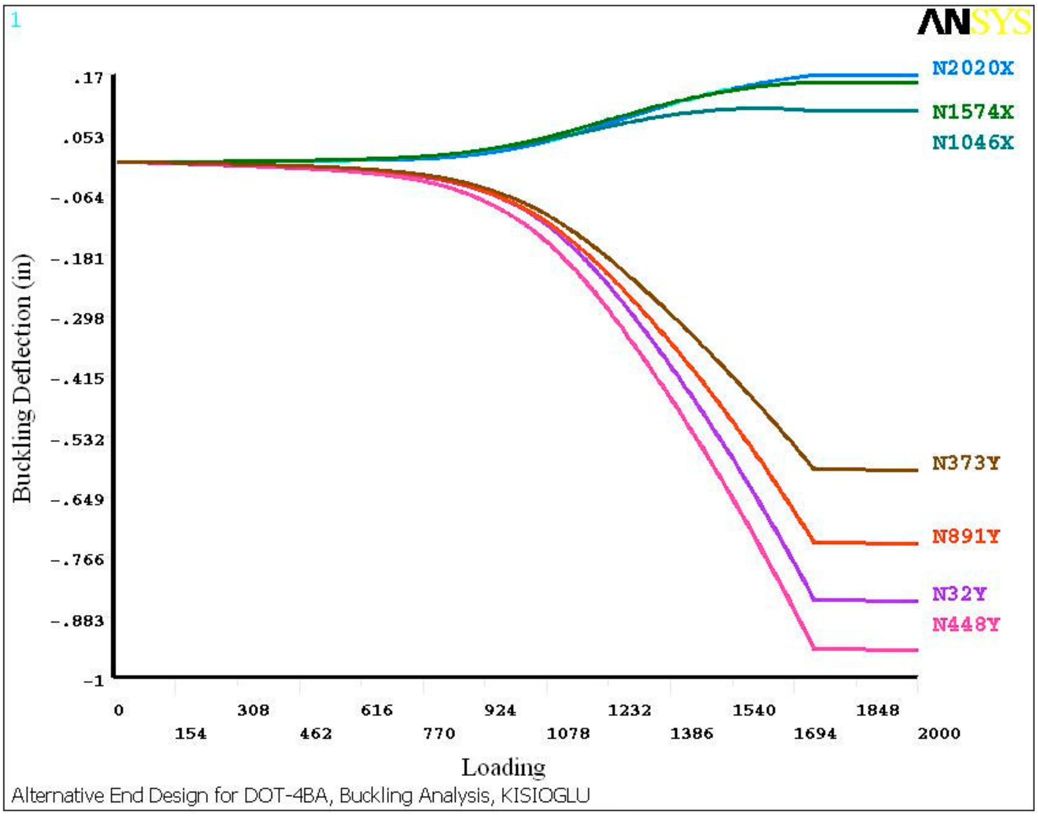

Figure 8. Deflection behavior of the selected nodes of the new end model.

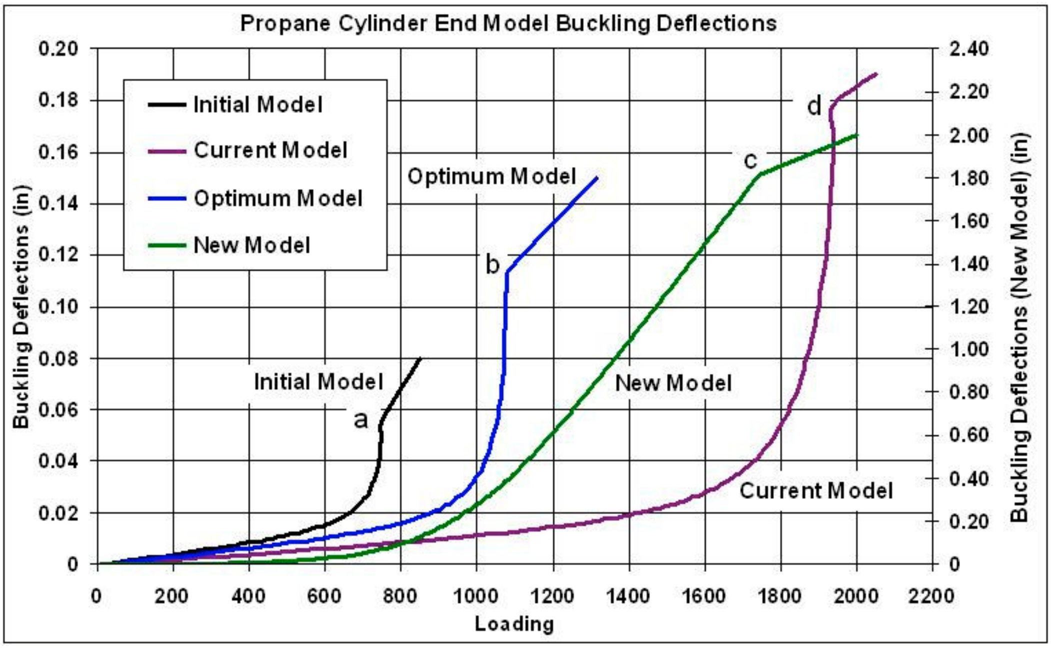

Figure 9. The buckling pressures of the different head designs.

parameters. When the end closure thickness used was doubled by Kisioglu et al. (2006) in the FEA, the P_{buc} value of the current end was also about 1938.30 psi (13.37 MPa) at point "d". The selection of the double end thickness for the current head was preferred by the cylinder manufacturer; however, using the double thickness means that the material cost doubles, which is extremely high considering the large production quantities. In this study, the P_{buc} value of the new end model was, however, much higher than the P_{buc} results of the previous corresponding studies. The P_{buc} value of the new end model was obtained at about 1743 psi (12.02 MPa) at point "c", as seen in Figure 9.

Future work

It would be good to have experimental verification of the present nonlinear FEA simulation results using the new end closure design. In order to do this, the new head model has to be manufactured with 4 dimples, using the same material, initial design parameters, and manufacturing processes. In addition, any material property changes at the dimple areas and their fillet zones can be observed. The theoretical studies for the buckling predictions have shown that the DL plays an important role rather than the R_d when the R_d is kept constant. Finally, the effects of the fillet radius, along with its drawn shell properties, should also be studied.

Conclusions

A new end closure design was presented for DOT-4BA refillable propane cylinders to prevent their end buckling, using both 2D plane and 3D shell models. The finite element formulation, which provides a solution to nonlinear problems with combined uniform geometry and nonlinear material, was derived using the standard displacement discretization procedure by introducing the incremental loading and appropriate boundary conditions. The analyses and the simulations were performed successfully using the uniform loading and the nonlinear material. The generated results for the buckling problem can be concluded as follows:

- Good agreement was found between plastic buckling predictions of the 2D and 3D models for the new end closure design subjected to external pressure. This was the case for both the buckling pressure and buckling deflection, as compared to the corresponding previous studies and the results published by Kisioglu et al. (2006).
- The new end closure design revealed that the crown region with 4 dimples was much stiffer and more resistant to buckling pressure than current models. This strength may be increased either by optimizing the DL or the R_d.
- The new end closure design maintains the current model and its material properties, and provides a much higher buckling limit, as well as reduced cost to the manufacturer.

References

ANSYS User Manual, Swanson Analysis System, R8.1, 2003.

Blachut, J., "Buckling of Sharp Knuckle Torispheres Under External Pressure", Thin-Walled Structures, 30, 55-77, 1998.

Combescure, A. and Galletly, G.D., "Plastic Buckling of Complete Toroidal Shells of Elliptical Cross-Section Subjected to Internal Pressure", Thin-Walled Structures, 34, 135-146, 1999.

Harte, A.M., McNamara, J.F. and Roddy, I.D., "Application of Optimization Methods to the Design of High Performance Composite Pipelines", Journal of Materials Processing Technology, 142, 58-64, 2003.

Kisioglu, Y., Brevick, J.R. and Kinzel, G.L., "Minimum Material Design for Propane Cylinder End Closures", Article in Press, ASME Journal of Pressure Vessels Technology, Accepted in December 2006.

Kisioglu, Y., Brevick, J.R. and Kinzel, G.L., "Bottom End-Closure Design Optimization of DOT-39 Non-refillable Refrigerant Cylinders", Journal of Pressure Vessels Technology, Transactions of the American Society of Mechanical Engineers, 127, 112-118, 2005.

Kisioglu, Y., Brevick, J.R. and Kinzel, G.L., "Determination of Burst Pressure and Location of the DOT-39 Refrigerant Cylinders", Journal of Pressure Vessels Technology, Transactions of the American Society of Mechanical Engineers, 123, 240-247, 2001.

Liang, C., Chen, H. and Jen, C., "Optimum Design of Filament-wound Multilayer Sandwich Submersible Pressure Hulls", Ocean Engineering, 30, 1941-1967, 2003.