The 22nd Conference of Mechanical Engineering Network of Thailand
15-17 October 2008, Thammasat University, Rangsit Campus, Pathum Thani, Thailand

A Comparison of Finite Element Models for A Thermoformed Rectangular Tray under Top Loading

Praphun Thammanontikul and Satjarthip Thusneyapan*

Department of Mechanical Engineering, Faculty of Engineering, Kasetsart University,
Bangkhen, Bangkok 10900, Thailand
Tel: 02-942-8555 ext. 1818, Fax: 02-579-4576, *E-mail: fengsjt@ku.ac.th

Abstract

Products manufactured by thermoforming process are used extensively for our daily life, such as in the food packaging. With the light weight requirement, the thickness of the thermoforming part is thin – which, frequently causes the collapse when several parts are loading on top. In order to design the parts, with greater strength, the finite-element analysis (FEA) becomes a helpful tool for design engineers. However, an improperly defined finite-element model (FEM) could yield significantly fault FEA results. This paper compared nine finite-element models, each differed by constrain and static loading conditions. The product for this study was a square tray from thermoforming process; the material was polystyrene (PS). The FEA results were compared with experimentally measured data, from the average of the vertical deformations at the four corners of the physical part. Our study found a FEM of the tray with four fixed corners at the base and distributed loading condition gave the most accurate result. The averaged deformation, in the range between 7.6 N and 38.0 N, gave the error of 3.49%; while the average of the absolute error was 5.02%.

Keywords: thermoforming, plastic tray, FEA, top load, polystyrene.

1. Introduction

Plastic packages produced by thermoforming process are in the form of box, tray and blister pack are found in our everyday life. Their applications are for food and small consumer goods. Thermoforming, also known as vacuum forming, is a process of forming a thermoplastic sheet to the three-dimensional shape of a mold. The thin sheet is heated in an oven and softened at its forming temperature. The warmed sheet is lowering and stretching onto to a mold or die, which has the shape of the required product. Then vacuuming, between the sheet and the mold, to make the sheet conform to the shape of the mold – and let it cooled.

The advantages of the products created by thermo-forming process are lightweight, easy mold production, fast production and low manufacturing cost. However, improper design of the products may result in the localized too thin of the wall thickness. This causes the products to collapse as they are stacking during storage and transportation. Therefore, a product testing is required to investigate the effects including the deformation, collapse or failure, as well as stacking hazard.

It is an interested topic to investigate the performance of packages under particular conditions of loading; mainly the top loading as for stacking. The International Organization for Standardization (or ISO) established ISO 2234:2000 Packaging – Complete, filled transport packages and unit loads – stacking tests using a static load [1]. This international standard specifies three methods for carrying out a stacking test on a complete, filled transport package, or on a unit load, using a static load. This standard is used for design conditions so that it has the strength or the protection that it offers to its contents when it is subjected to stacking.

The appropriate product thickness is too complicated to calculate - especially for complex shapes. Hence the CAD/CAE (Computer-Aided Design / Computer-Aided Engineering) is found to be a suitable application.

The finite element (FE) method, as CAE software, is an internationally accepted method for engineering design of modern products. Nevertheless, an inappropriate defined material and modeling conditions could lead to a significantly mistaken FEA result.

The objective of this research is to find the most suitable finite-element model (FEM) for analysis the top load test of square and rectangular shapes. The FEM conditions consist of material property of polystyrene (PS), constrains or boundary conditions and loading assignment.

2. Experimentally Obtained Stress-Strain Relation

The stress-strain relation of polystyrene (PS) was performed by using a tensile testing machine (Hounsfield H50KS, UK). The specimen preparation and test conditions were followed the ASTM D882-03 [2]. The testing condition for the grip separation speed was at 5 mm/min. The PS was obtained from TPI Polene Public Company Limited. Five specimens were used and their results were ensemble averaged. The averaged stress-strain data of PS are plotted in Figure 1.

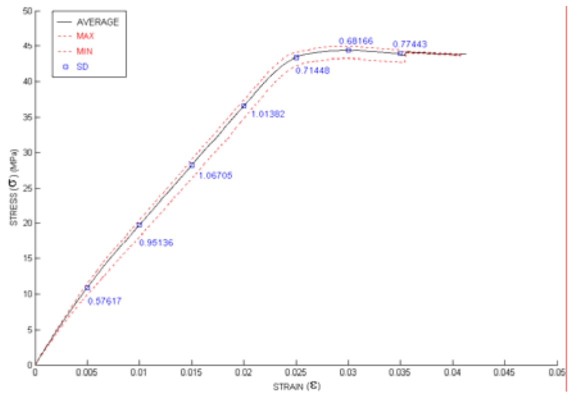

Figure 1. The experimentally obtained stress-strain relation of the polystyrene.

The numerical data of this averaged stress-strain curve was later used for the material property of the product in the finite element analysis.

3. Top Load Testing

Two shapes of tray were used in this experiment, model A and B, as shown in Figure 2. They were manufactured by the thermoforming process. The model A was a square shape with the dimension of 124.5 **x** 124.5 **x** 28.5 mm, and B was a rectangular shape with the dimension of 139 **x** 68 **x** 33 mm. Both models had the average wall thickness of 0.2 mm. The mass of A and B were 4.72 gm and 3.73 gm respectively.

Figure 2. The tray model A (left) and B (right).

The square tray (model A) was primarily used for evaluating the suitability of finite element models. While model B was later used for verifying the accuracy of the FEM – when the product shape was differed from model A.

Top load testing was used for testing the vertical deflection of the PS tray. This test is analogy to the stacking condition of the product. The test was done according to the ISO 2234:2000 Packaging – Complete, filled transport packages and unit loads – stacking tests using a static load [1]. An objective of this testing standard was to measure the vertical deflection of the product.

The setup of our top-load test is shown in Figure 3. The equipments consisted of vernier height gages, loading plates and a support for the plates. The loading plate was a square plate (130 x 130 mm) with the thickness of 3 mm. Each loading plate weighted 3.8 N ± 2% and made from steel, the exact load of each plate was engraved on the plate. The support, for supporting the load, was a plastic plate with two aligning corners.

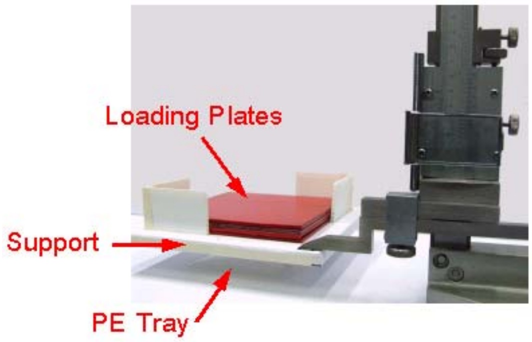

Figure 3. Setup the top load testing.

In order to measure the deflection of the tray, the product was placed on a rigid flat floor with the support positioned on the top. The base and the support were set in the horizontal position, and the heights of the four corners were measured.

To avoid the impact force, slowly place the loading plate one by one on the support; then measuring the deflection. The loading was from 7.6 N to 38.0 N or 0.77 kg to 3.87 kg, and 7.6 N increments. According to the ISO 2234:2000, the center of gravity of the total weight must be less than 50 % of the height of the product.

The total of five trays was used for the measurement. Therefore the deflection at each load was the average from the four corners of each tray of the five samples. The plot of the average vertical deflection of the model A versus the loading force is illustrated in Figure 4.

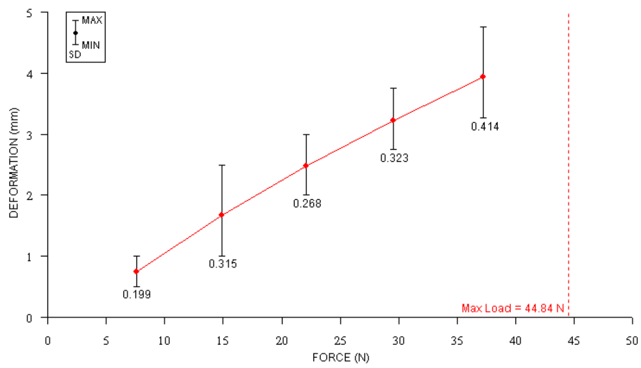

Figure 4. Plot of the experimental result of the vertical deflection against the force, of model A.

Note that the next loading increment above 38.0 N, or at 45.6 N, the deformation was unable to determine due to the collapse of the tray. Hence the maximum deformation was obtained at the load of 38.0 N.

4. Finite Element Models

To simulate the top load test, CAD Software (Pro/ENGINEER, Parametric Technology Co., USA) was used for creating the surface model of the tray. The MSC.Patran 2005 (MSC Software Co., USA) was used for pre and post processes; and MSC.Nastran 2005 (MSC Software Co., USA) was used for processing the static top loading simulation.

The CAD Model of the tray was transferred to the MSC.Patran software by the standard file exchanged format – IGES. The mesh model of the tray A was shell elements with 0.2 mm thickness. Each element was a QUAD element with isotropic property. The total of 3805 nodes and 3752 elements were created for the tray A. The node displacement was free to move for all of its six degree-of-freedom (6-DOF), except at the boundaries.

The FEM of the contact elements with contact load is shown in Figure 5. The static top load assignment for distributed load on nodes along the top edges of the tray was assigned in the vertical direction as shown in Figure 6. Another loading assignment available in MSC.Nastran 2005, the Multiple Points Constrain (MPC), was applied to the FEM as shown in Figure 7. The node displacement, where the top load was applied, was single DOF in the vertical direction or in z-axis.

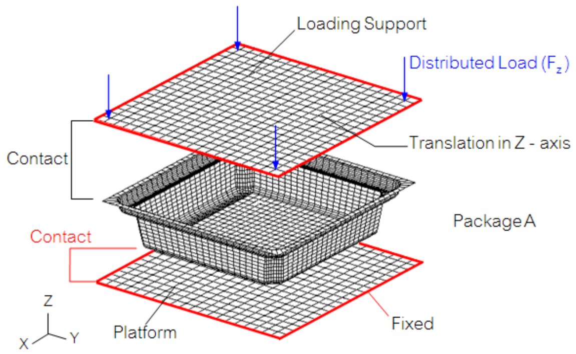

Figure 5. The FEM for contact elements-contact load.

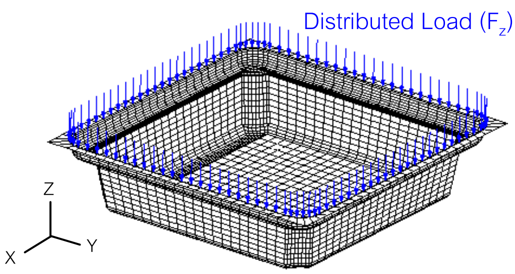

Figure 6. The distributed load on the top edges.

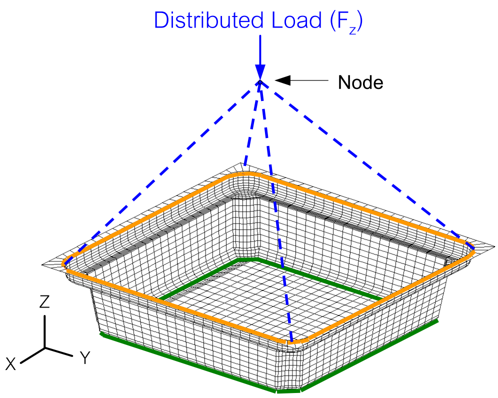

Figure 7. The top load assignment using Multiple Points Constrain or MPC.

The constrain or the boundary condition, where the nodes at the bottom edges of the tray contacting the floor, was fixed or had a zero DOF at the specified corners and/or edges – otherwise the node was a 5-DOF with no displacement in the vertical direction. Figure 8 shown four types of the boundary conditions; red color indicated fixed condition and green color for the 5-DOF nodes.

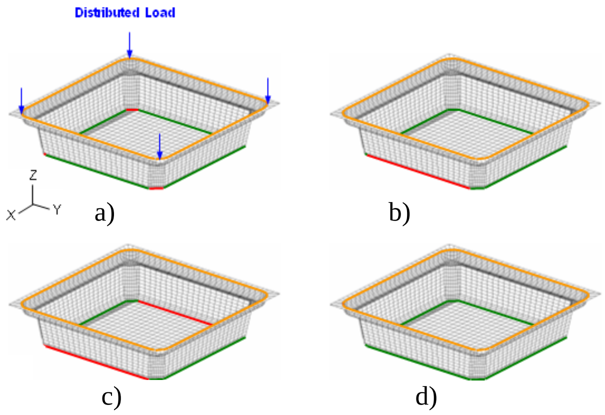

Figure 8. The constrains at the bottom of the tray: a) four fixed corners b) one fixed edge c) two fixed edges and d) free edges. Red color indicated the fixed nodes and green for the 5-DOF nodes.

Thus from the types of loading assignment and boundary condition, there were nine combinations of constrain and loading. Hence, the total of nine FEM's were examined, they were as follow:

1. contact elements – contact load (Contact)
2. four fixed corners – distributed load (Fix 4 corners)
3. two fixed edges – distributed load (Fix 2 edges)
4. one fixed edge – distributed load (Fix 1 edge)
5. free edges – distributed load (Not Fixed)
6. four fixed corners – MPC (Fix 4 corners–MPC)
7. two fixed edges – MPC (Fix 2 edges–MPC)
8. one fixed edge – MPC (Fix 1 edge–MPC)
9. free edge – MPC (Not fixed–MPC)

The static load simulation by using FEA was done according to the experiment performed as was described earlier in section 3. That was from 7.6 N to 38.0 N with 7.6 N increments. The material property of the tray applied to the FEA was that from Figure 1.

5. Results

Examples of the result FEA simulating the static top load of the tray with the load of 22.8 N for four fixed corners with distributed load and two fixed edges with distributed load are shown in Figure 9 and 10 respectively. The color contour is the deflection in z-direction.

The graph of the deformation in z-axis against the load of the nine FEM's and from the experiment is shown in Figure 11. The percentage errors of the nine FEM's at the five loading forces are summarized in Figure 12.

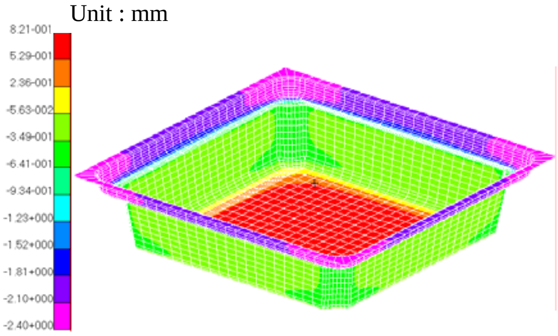

Figure 9. FEA result of the deformation in z-axis during a 22.8 N top loading by using the FEM with four fixed corners with distributed load.

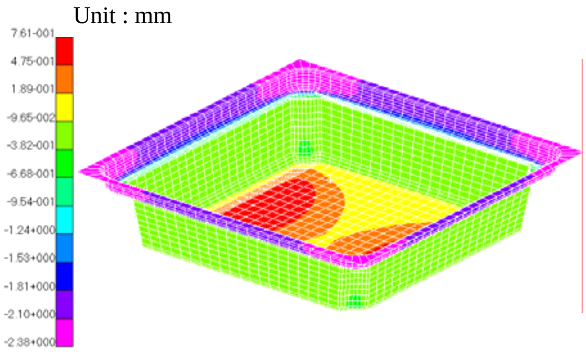

Figure 10. FEA result of the deformation in z-axis during a 22.8 N top loading by using the FEM with two fixed edges with distributed load.

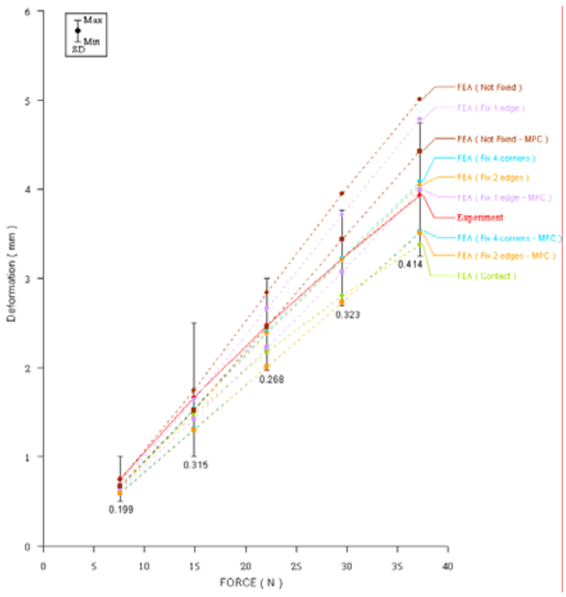

Figure 11. A plot of the deformation in z-axis against the load comparing the nine FEM's and the experiment.

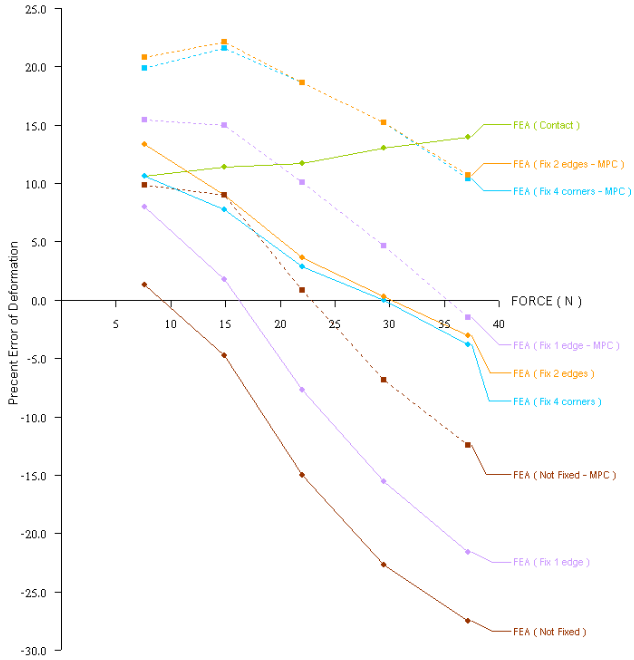

Figure 12. A plot comparing the percentage error of the deformation in z-axis against the load of the nine FEM's.

The deformation shown in Figure 9 indicated the maximum deflection was at the middle of the base (red region) and had the positive value. This positive value illustrated that the base was bending up; however we interested only the negative deflection at the top edge of the part.

The results from Figures 11 and 12 shown the curve of the FEM's with four fixed corners with distributed load (Fix 4 corners) and two fixed edges with distributed load (Fix 2 edges) were agreeable to the experimental data. The average percentage error over the load from 7.6 N to 38.0 N of the four fixed corners with distributed load was 3.49 %, and that of the two fixed edges with distributed load was 4.64%. It was interesting to note that the FEM having free edges (5-DOF for the nodes at the base) with MPC loading (Not fixed–MPC) had the average error of only -0.04%.

The percentage of the absolute value of the error of the "Fix 4 corners" was the lowest at 5.02%, followed by "Fix 2 edges" at 5.86%, while that of "Not fixed–MPC" was 7.69%.

The two worst cases were two fixed edges with MPC loading (Fix 2 edges–MPC) and four fixed corners with MPC loading (Fix 4 corners–MPC); their percentage of the absolute value of the errors were 17.60% and 17.17 % respectively.

Besides the lowest error obtained by using the fixed four corners with distributed load, it was found to have less complicated procedure for assigning the boundary and loading conditions. The processing time was also fast, 6.66 times better than the slowest (contact elements–contact load) and was slower by 1.019 times the fastest (2 fixed edges–distributed load).

The two FEM's were applied to the tray model B with the same conditions as for the model A, only the number of nodes and elements were differed (1360 nodes and 1323 elements). The plot of the deformation in z-axis against the load, comparing with the experiment, is shown in Figure 13. The experiment found the model B could withstand higher load; it was collapsed at the load greater than 45.6 N.

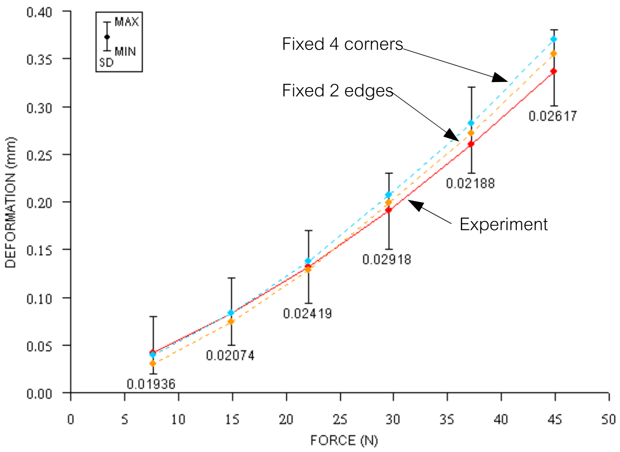

Figure 13. The deformation in z-axis against the load for the tray model B using the fixed at 4 corners and fixed at 2 edges, comparing with the experiment.

For the tray model B, the percentage of the absolute value of the error of the "Fix 4 corners" was also the lowest at 6.44%, and also followed by the "Fix 2 edges" at 9.58%.

Hence the most suitable finite-element model, suitable for simulate the static top loading of thermoformed products, was the one with four fixed corners and distributed load on the top edges.

6. Conclusion

This paper compared nine finite-element models, each differed by constrain and loading conditions. The product used for this study was a square tray, produced by thermoforming process; the material was polystyrene (PS). The FEA results were compared with the experimentally measured data, from the average of the vertical deformations from the four corners of the tray. Our study found the FEM of the tray with fixed at four corners at the base and distributed loading condition gave the lowest percentage error. The averaged deformation in the range from 7.6 N to 38.0 N was 3.49% and the average of the absolute error of 5.02%. Future analysis of a rectangular tray found the same FEM also provided the lowest percentage of the average of the absolute error of 6.44%

References

1. ISO 2234:2000 Packaging – Complete, filled transport packages and unit loads – stacking tests using a static load.
2. ASTM D 638-03 Standard Test for Tensile Properties of Plastic.