FINITE ELEMENT ANALYSIS OF PLASTIC CUPS

[1]JESSUPA SARAKHUN, [2]SOMBAT TAMNA, [3]EK-U THAMMAKORNBUNJUT

[1]Master Program in Engineering Technology, Faculty of Engineering, Thai-Nichi Institute of Technology, Bangkok 10250, Thailand
[2,3]Applied Mathematics and Mechanics Research Laboratory, Faculty of Engineering, Thai-Nichi Institute of Technology, Bangkok 10250, Thailand
Email: [1]jessupa.s@gmail.com, [3]ek-u@tni.ac.th

Abstract: In the past, many research about forming of plastic sheets, especially thermoforming, have been carried out to predict wall thickness of containers and effects of forming parameters to the thickness distribution. In contrast, this paper demonstrates the use of finite elementsto improve material usage in manufacturing process. By varying thickness and applied loads in the simulation, stiffness of a cupcan then be compared and used to choose theproper value of the wall thickness that meet the actual requirements. Results also display that theincreasing rate of the stiffness will reduce as a thickness increase.

Index Terms: Plastic Cup, Polypropylene, Stiffness, Finite Element Method, ANSYS.

I. INTRODUCTION

Nowadays many food packaging are usually made of plastics because of its lighter in weight, flexibility, recyclability, and low cost. The manufacturer's demand causes rapid development of various plastic forming processes, for example, injection molding, extrusion molding, blow molding, and thermoforming. For a thin and not deep container such as food packaging, pharmaceutical blister pack, and small electronic housing, the thermoforming process is the most popular method to form these products. This method use high temperature to heat plastic sheet to a pliable forming temperature and then stretched onto a lower temperature mold by pressure with or without assisted plug. For interested reader, more details can be found in [1]. In the past, some research intend to predict the wall thickness of the container by using finite element method [2-4] while another investigate effects of its parameters to thickness distribution [5-7].

These research result in the improvement of thermoforming process, machine, and efficiently use of energy. However in the packaging design process, especially in the mechanical design point of view, some container are designed much more than their actual requirements.For example, a cup used for a cat's food container as shown in the Figure 1 has a limit of its minimum wall thickness at 0.3 mm and a mean value of the thickness is 0.35 mm. Consequently, the cup is too stiff and very strong. If we can reduce wall thickness of this kind of package we can save materials and energy consumption in its manufacturingprocesses.

This research aims to demonstrate the relation between thickness and stiffness of a cup by finite element method. Simulations are carried out by a finite element software, ANSYS.



Figure 1: Example of plastic cups

II. COMPUTER SIMULATION

A sample plastic cup with some dimensions as shown in Figure 2 is axial symmetry. It made of polypropylene which has a tensile modulus of 1.3 GPa, a yield stress of 24 MPa, and Poisson's ratio of 0.42. In ANSYS, a uniform mesh with 200058 elements and 400572 nodes, which already passed a mesh independent check, are formulated on the model.
The thicknesses of the cup are 0.2, 0.25, 0.3, and 0.35 mm. And the applied loads are 1, 3, 5, 7, and 9 N which is enough for observed the effect of thickness of the cup. The applied load acts normally at the annular area at the bottom of the cup and the top area of the cup is set as a fixed support as shown in Figure 3. Vertical deflections along the measurement line from point 1 to point 2 as shown in Figure 4will bedepicted for analysis.

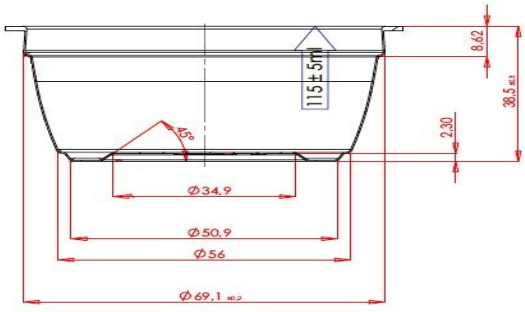


Figure 2: A sample cup dimensions

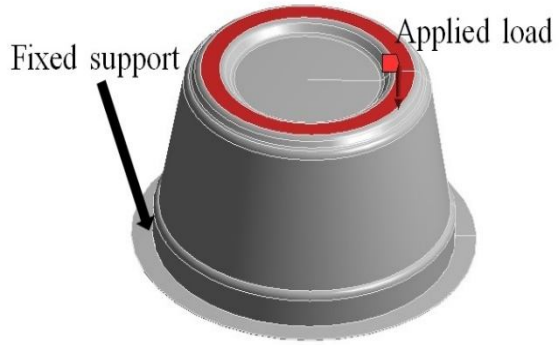


Figure 3: Applied load and support conditions

III. RESULTS AND DISCUSSION

From numerical simulations, the maximum von Mises stress, which is 13.5 MPa, has occurred at a cup of 0.2 mm thickness subject to a vertical force of 9 N. It is quite far from the yield stress so that it will not be seriously considered in this analysis. As an example, Figure 5 shows the von Mises stress of a 0.25 mm thickness cup which has a maximum value of 8.52 MPa.

Figure 6 – 9 are vertical deflection along the measurement line of cups which has wall thickness of 0.2, 0.25, 0.3, 0.35 mm, respectively. As seen from the figures, we treat this problem as a large deformation because the deflections are higher than one-tenth of the thickness. A cup with small thickness has a lower stiffness than a thick one as expect.In addition, we can observe a little nonlinear effect at each step of load increasing. This may be come from shape of the cup which act as a cylindrical shell.

From these Figures, it should be recommend to manufacturer to use a tool like finite element analysis to design their products. They may reduce the average value of wall thickness to minimum as long as the strength and stiffness are passed the actual requirements. As an example in this study which give the average thickness of the cup to be 0.35 mm. By comparing Figure 7, Figure 8, and Figure 9, one may choose a thickness of 0.3 mmif maximum deflection is still in the acceptable criterion.

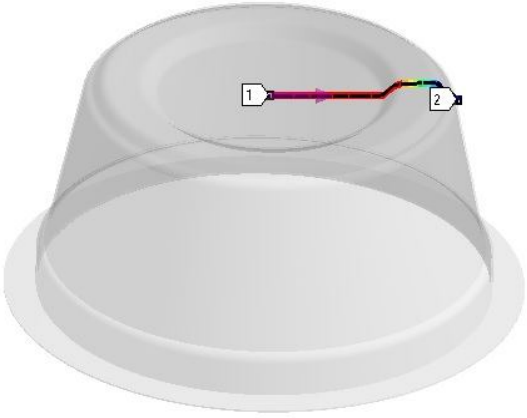


Figure 4: Measurement line for depict the vertical deflection

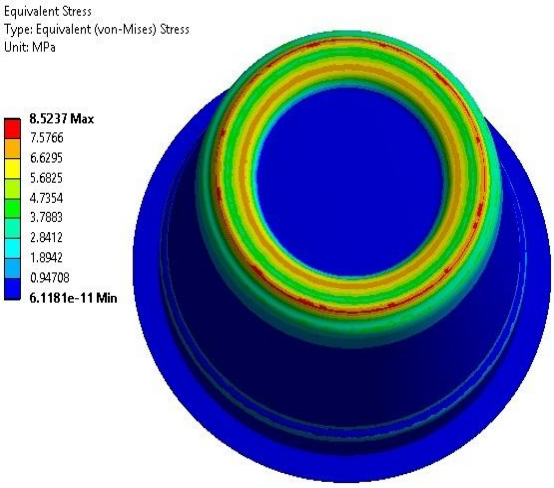


Figure 5: von Mises stress of 0.25 mm thickness cup

As shown in Figure 10 and 11, effect of the thickness at constant load is observed. The results of 3 and 9 N loads are selected and shown. Although the applied loads are different but both results show the same trend. That is an increase of the thickness from 0.2 mm to 0.25 mm the stiffness of a cup has been improved largely. However the thickness that is increased from 0.25 mm to 0.3 mm and from 0.3 mm to 0.35 mm does not make a rapidly improvement of the stiffness of the cup as it was.

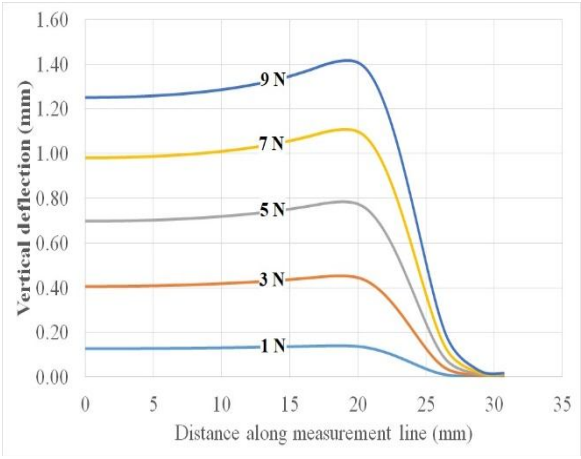


Figure 6: Deflection of a 0.2 mm thickness cup

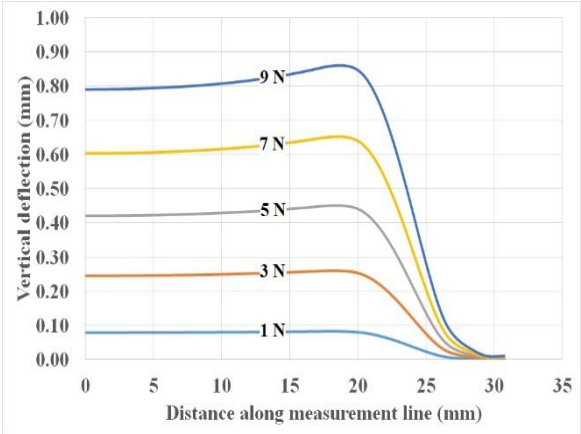


Figure 7: Deflection of a 0.25 mm thickness cup

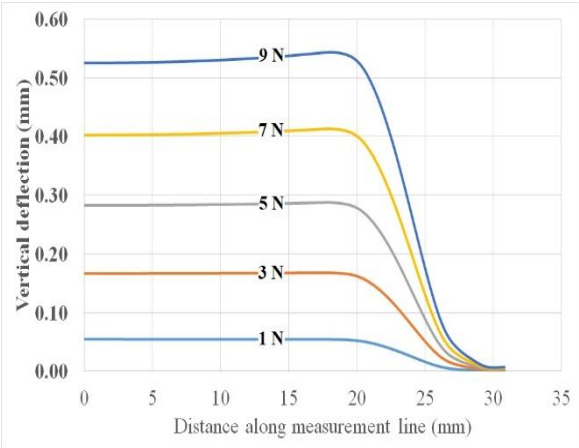


Figure 8: Deflection of a 0.3 mm thickness cup

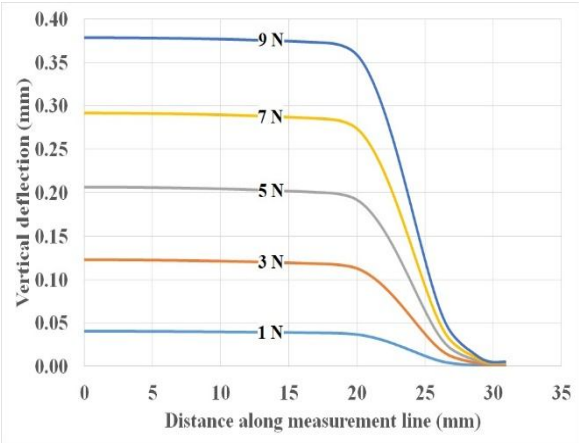


Figure 9: Deflection of a 0.35 mm thickness cup

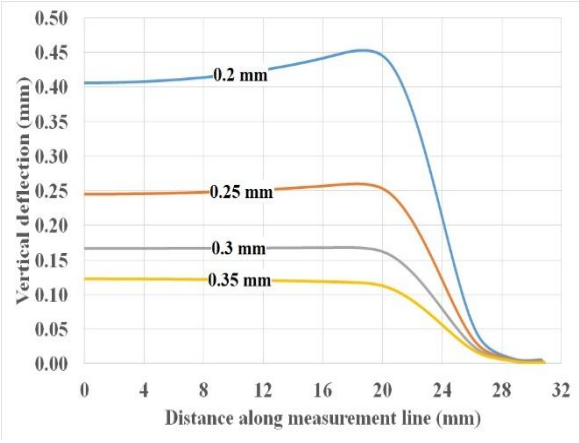


Figure 10: Deflection of a cup subject to 3N load

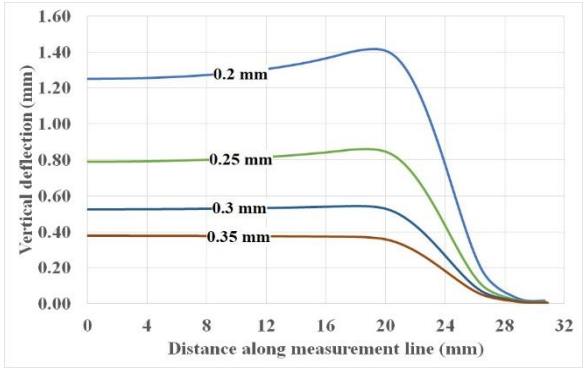


Figure 11: Deflection of a cup subject to 9N load

CONCLUSION

The present study demonstrates the way to improve the product design before its manufacturing process by simulate strength and stiffness of a plastic cup with different wall thickness subject to vertical loads.In this case, a stiffness of a cup or simply its deflections should be a main parameter to be considered instead of maximum or von Mises stresses. Additionally, increasing rate of the stiffness will reduce as a thickness increase.

REFERENCES

[1] J. L. Throne, *Understanding thermoforming* 2nd edition, Carl HanserVerlag, Munich, 2008.

[2] R. J. Crawford and S. K. L. Lui, "Prediction of wall thickness distribution in thermoformed mouldings," *Eur. Polym. J.*, vol. 18, no. 8, pp. 699–705, 1982.

[3] C. P. J. O'Connor, G. Menary, P. J. Martin, and E. McConville, "Finite element analysis of the thermoforming of polypropylene," *11th ESAFORM Conference on Material Forming*, Lyon, France, April 23-25, 2008.

[4] E. S. Erdogan and O. Eksi, "Prediction of wall thickness distribution in simple thermoforming moulds," *Strojniškivestnik - Journal of Mechanical Engineering*, vol. 60, no. 3, pp. 195–202, 2014.

[5] R. McCool, P. J. Martin, and E. Harkin-Jones, "Process modelling for control of product wall thickness in thermoforming," *Plastics, Rubber and Composites*, vol. 35, no. 8, pp. 340–347, 2006.

[6] K. Pepliński and A. Mozer, "ANSYS Polyflow software use to optimize the sheet thickness distribution in thermoforming process," *J. Polish CIMAC*, vol. 6, no. 3, pp. 215–220, 2011.

[7] C. P. J. O'Connor, P. J. Martin, J. Sweeney, G. Menary, P. Caton-Rose, and P. E. Spencer, "Simulation of the plug-assisted thermoforming of polypropylene using a large strain thermally coupled constitutive model," *Journal of Materials Processing Technology*, vol. 213, no. 9, pp. 1588–1600, 2013.

[8] E. Ventsel and T. Krauthammer, *Thin Plates and Shells: Theory, Analysis, and Applications* Marcel Dekker, 2001.

★★★