Product Development Process Improvement by Using Finite Element Simulation

Thawisanee Pattamapradit, Chalearmchai Chaithongrat and Somsak Siwadamrongpong*

School of Mechanical Engineering, Suranaree University of Technology, Nakhon Ratchasima 3000

*Corresponding Author: somsaksi@sut.ac.th, Tel. 044-224-234, Fax. 044-224-613

Abstract

The designing, manufacturing and testing steps are the important steps in product design and development processes. It begins with a design that searches for required dimensions and shape that conform to its applications. Then, parts are manufactured and tested for performance evaluation and making of corrective design. These steps generally consume long time and high expense. It should be more advantage, if these steps could be carried out by computer simulation. Therefore, this research aims to use a finite element method to simulate and analyze failure of the polycarbonate product. Fixtures are designed and constructed for tensile testing of the current product. The result comparison between simulation and testing shows approximately 90% of accuracy. It implies that simulation model and software are verified. The results from this research were used to improve the current design. It is expected that using simulation techniques could significantly reduce development time and expense.

Keywords: failure analysis, computer simulation, polycarbonate, product development

1. Introduction

Nowadays, many technologies have growth rapidly. In each of industries has high competition even in manufacturing. In order to ensure quality and performance, the step of product development begins with designing. It requires dimensions and shape that conform to applications and consumers. Then it is manufactured and tested for performance evaluation. Test is an important procedure of product development process. It uses to investigate failure problems and able to reduce amount of scrap. There are many available testing modes such as tensile test, drop test, vibration test, impact test etc. In generally, the product development process has followed as these steps: designing, manufacture and test as shown in figure 1. Nevertheless these steps consume long time and high expense especially testing procedure. In addition, some product must be sent to test in foreign country. It should be more advantage, if these steps could be carried out by computer simulation. Therefore, this research aims to use finite element method to analyze product failure. The result of simulation indicates the weak point. It is able to find the way to improve the model before damaged happened.

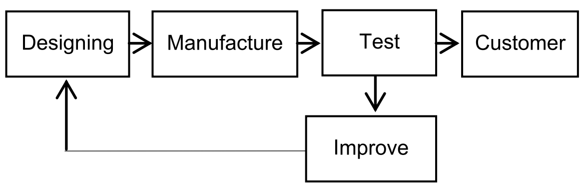

Fig. 1 product development process

2. Experimental Procedure

Finite element analysis (FEA) is one method that used as failure analysis tool. Many industrials use it to improve and develop product. It is more convenient, faster and less expense than testing. This research has studies product failure by simulation together with making of fixture for testing. The project activities follow in figure 2.

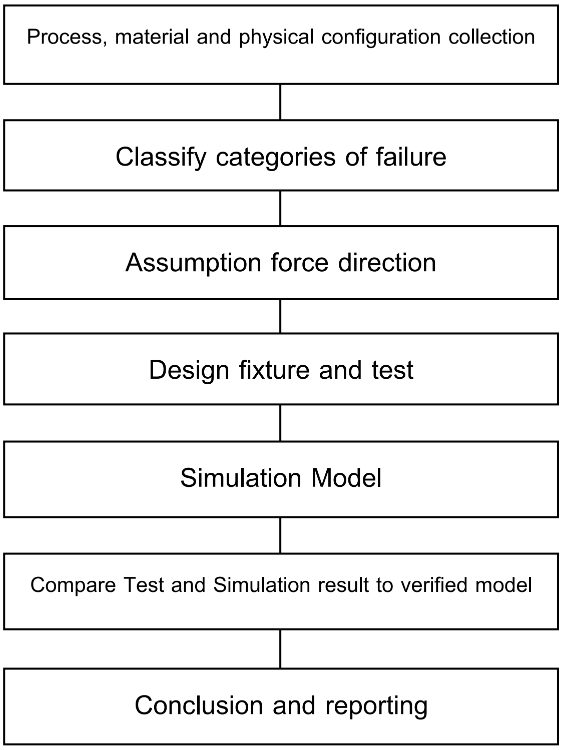

Fig. 2 Project activities

The project activities start with studying manufacturing process, material of product and physical configuration. Then collecting data by classify product amount 200 pieces in order to identify critical failure area. Afterwards assume the load directions that may cause part fracture. The fixtures are designed and applied to use with Universal Testing Machine (UTM 5560). The sample product will be tested by applying load on critical area in assumed directions. To analyze product failure by using Finite Element will use ANSYS Workbench 12.0. It uses for simulate mechanical static failure and determine the result of stress distribution which occurred on the model. The simulation is indicated by specify appropriate element type and size. The result of simulation will be compared with fracture testing sample to verified directions and force magnitude which cause damaged. Eventually, it implies that simulation model and software are verified and reliable with approximately 90% of accuracy.

3. General information about product

Polycarbonate was selected as the material for the study. In this session describes about material properties, feature of product and failure appearance as follows.

3.1 Material

Polycarbonate (PC) is engineering material that widely used in many industries. It is a group of thermoplastic that has organic chemistry structure as shown in figure 3. It was well known about its high strength, good hardness, flexible, impact durability, high temperature and chemical stability compared to other polymers consequently, it is used as part and tooling and fabricate in many components. Polycarbonate properties as shown in table 1.

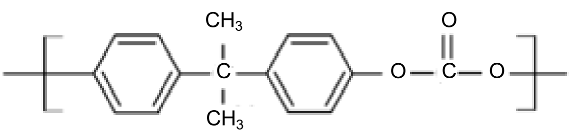

Fig. 3 PC organic chemistry structure

Table 1 Polycarbonate Properties

Properties	
Specific gravity	1.4
Tensile Strength (Mpa)	64
Tensile modulus (Mpa)	2400

Impact Strength (J/m)	70
Thermal Conductivity (W/m.k)	0.25
Melt temperature (°C)	280-310
Electrical Resistivity (ohm.cm)	<1E4

3.2 Feature of product and failure appearance

According to the feature as shown in figure 5, the part was designed and injected mould into irregular shape that suitable for assembly with other components. There are 4 parts (A, B, C, and D) which protrude from this product. The damaged mostly occurs on these parts. From the result of 200 PC part failure collected data as shown in table 2 found that part A and part B are mainly fracture cover 66% of failure. The reasons that make part failure cause from external load occur during the process and design of the model. The critical area could damage easier than others also the damaged cause scrap increasing in the process. Therefore part A and part B are mainly considered in this study.

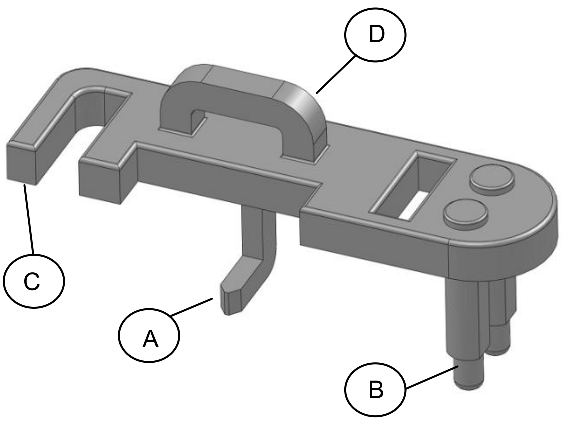

Fig. 4 PC Feature of product

Table 2 Percent of PC part failure

Part	**Percent**
A	42
B	26
C	10
D	8
Others	14
Total	**100**

3.3 Force assumption for failure analysis of part A and part B

The failure of part A can be classified into 2 manners. First, the part has broken out of product. Some case found rip at the corner of protrusion area before it broken. Second, the part is bended in left or right direction of the product. From these actual failures, the force assumption can be supposed in 2 directions (A_1 and A_2) at the tip of the part as shown in figure 5.

As for The failure of part B can be classified in only one manner. The part has broken out of product without bending. Due to the face that the feature of part B was designed in tubular as symmetry shape so the failure can occur in many directions. Hence this part failure would be supposed in 4 directions (B_1 B_2 B_3 and B_4) as shown in figure 6.

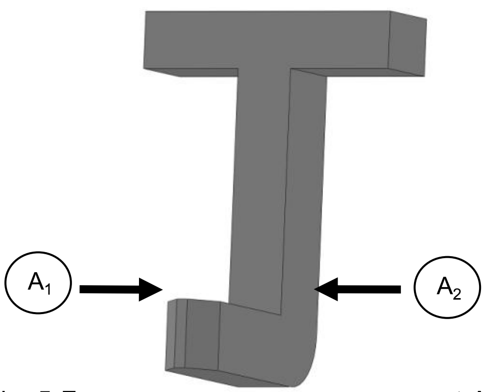

Fig. 5 Force direction assumption of part A

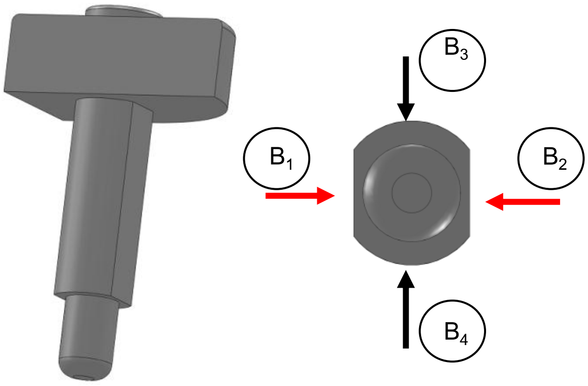

Fig. 6 Force direction assumption of Part B

3.4 Design fixture and test

In this section will design fixture for test with UTM testing machine as shown in figure 7. The fixture composes of 2 parts as shown in figure 8. First is fix fixture, it use for attach part and assembled with UTM testing at lower platform. Second fixture is assembled with UTM testing at upper platform. It uses for test by put on the test position. Then it moves down to compresses part with speed 1 mm/min. This test proposes to find maximum force magnitude that makes part A and B damaged in assumption directions. The failure test result would be compared with simulation result later.

Fig. 7 Universal Testing Machine

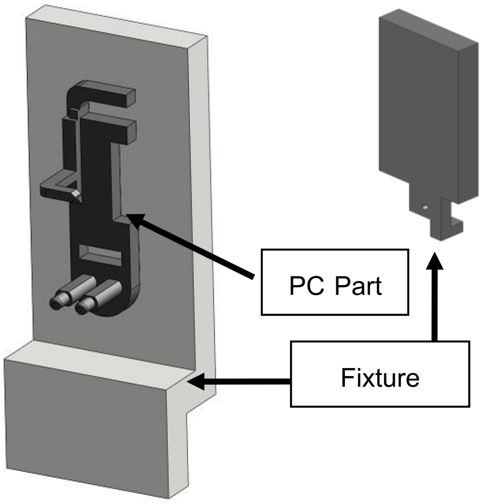

Fig. 8 Fixture and part

3.5 Finite element failure analysis

Finite element model of a PC part was constructed to simulate with static analysis. The model is studied by performing finite element analysis (FEA) using commercial program ANSYS 12.0. It presents a 3D finite element calculate of stress distribution that occur on model during apply loading act on. In order to yield the exactly result, the simulation must identify the boundary conditions that are similarly to the actual as possible.

3.5.1 Finite Element Modeling

To analyze product failure would be set the initial conditions for simulation as follows

- Material behavior identify in linear isotropic elasticity
- Using von Misses Yield criteria to indicate failure
- Specify Mechanical properties of Polycarbonate as shown in table 3

3.5.2. Element Type and elements size

The element type that defines for this feature is second order quadrilateral as shown in figure 9. It yields high accuracy to compare with other types. This study specified element size between 0.2-0.5 mm. to consider the effect of element size.

Table 3 Mechanical properties of Polycarbonate

Mechanical properties	
Poison ratio	0.38
Young Modulus (Mpa)	2,400
Yield Strength (Mpa)	58
Tensile Strength (Mpa)	64
Tensile Elongation (%)	5.0-10.0

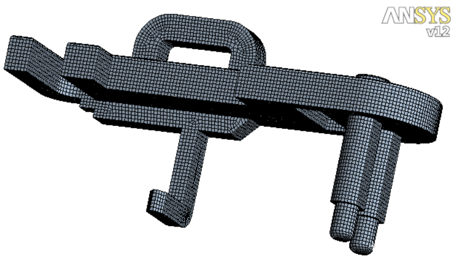

Fig. 9 Meshing feature

3.5.3 Boundary Conditions

A) Fix Support

In the FEA model, the fixing areas as shown in fig. 10 are referred from fix fixture assembly.

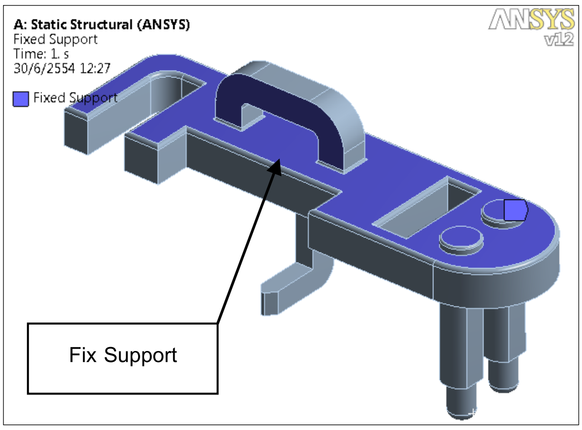

Fig. 10 Fix Support area

B) Load applying

This simulation will apply load act on part A and part B with assumption directions that refer in section 3.3.

- Part A : entering load in 2 directions (A_1 and A_2)
- Part B : entering load in 4 directions (B_1 B_2 B_3 and B_4)

The simulation analysis will consider deformation and stress distribution that occurs on model. After apply load until cause ultimate tensile stress, the maximum load simulation result will be compared with test result.

4. Result of analysis

4.1 effect of element size

This research considers feature failure by study effect of element size. The result of stress after applies load on this variety element size model as shown in table 4. It found that if the element size decreases, the trend of maximum stress that occurs on weak point would be increase then approach stable. The stress error at element size 0.2 mm. is within ±5% acceptable interval. The relation between element size, maximum stress distribution and stress error of part A and part B as shown in figure 11 and 12 have represented the response of element size change. Therefore, the element size 0.2 mm is employed to investigate part A and part B failure simulation.

Table 4 show maximum stress

Part	Direction	Element Size (mm.)			
		0.2	0.3	0.4	0.5
A	A_1	15.3	14.43	13.19	12.64
B	B_1	25.62	24.58	22.71	21.29

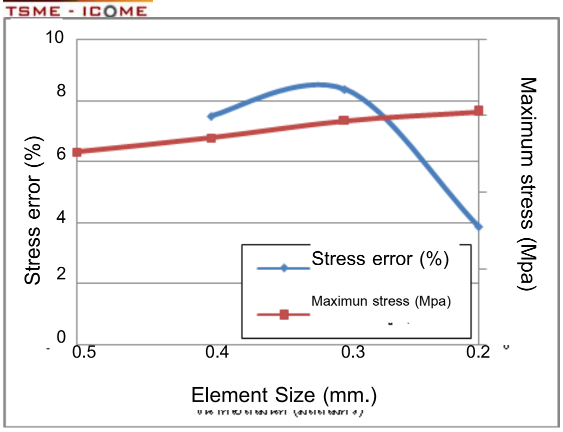

Fig. 11 Relation between element size, maximum stress distribution and stress error of part A

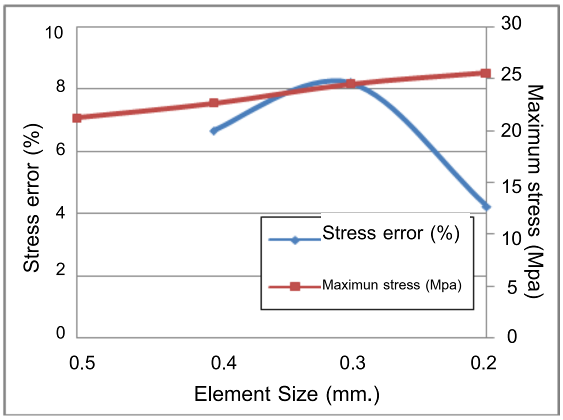

Fig. 12 Relation between element size, maximum stress distribution and stress error of part B

4.2 Testing failure result

The failure testing provides maximum force magnitude that causes part fracture with assumption directions as shown in table 5.

Table 5 Testing result

Part	Direction	Maximum Load (N)
A	A_1	5.2201
	A_1	5.1666
B	B_1	10.4721
	B_2	10.2331
	B_3	8.1669
	B_4	8.4168

4.3 Finite element simulation result

This section shows the result of FEA failure analysis. It represents deformation and stress distribution occur on model after apply load on interested area. The criterion of failure evaluation refers to ultimate tensile stress.

4.3.1 Failure simulation result of part A

The Investigation of part A failure is considered apply load in 2 direction. The simulation result as shown in figure 13 and 14 is represented stress occur by apply load on A_1 and A_2 direction respectively. It found that the maximum load is 5.2624 N. would be cause ultimate tensile stress and the failure is occurred at the corner of the part in both sides.

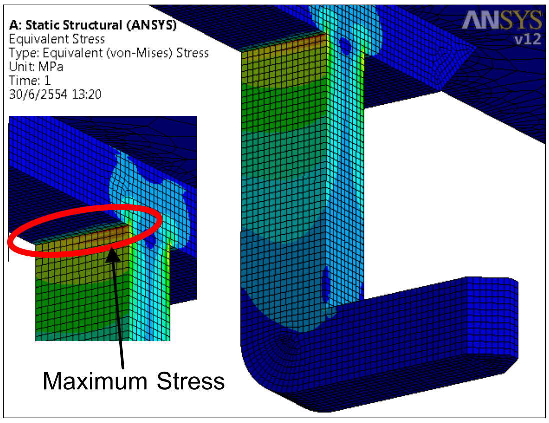

Fig 13 Stress distribution of Part A during apply load in A_1 direction

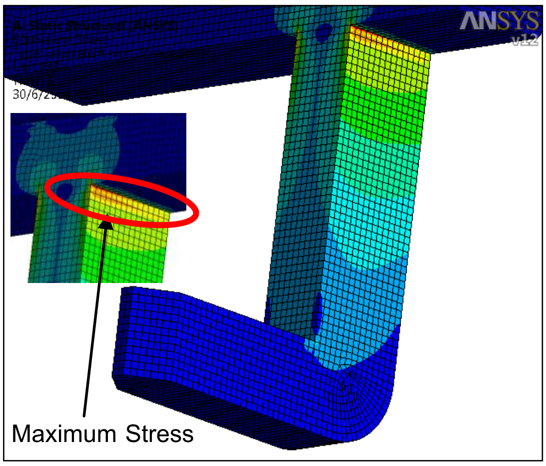

Fig 14 Stress distribution of Part A during apply load in A_2 direction

4.3.2 Failure analysis simulation result of part B

The Investigation of part B failure is considered loading perform in 4 directions. The simulation result as shown in figure 15 found that the failure is happened on the base of part B in the same side as load applying. The maximum stress is occurred as shown in table 6. It found that apply load in B_1 and B_2 direction causes maximum stress further than B_3 and B_4 direction.

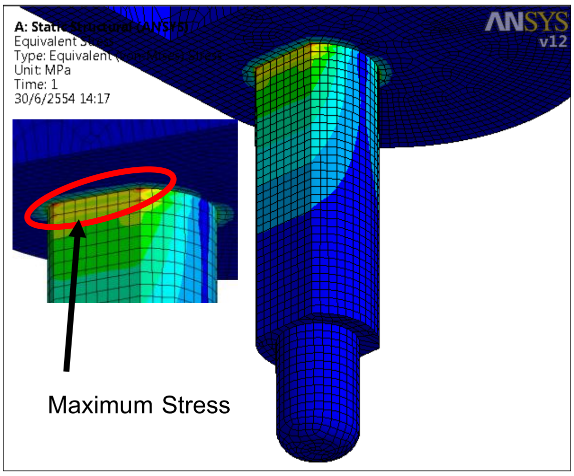

Fig 15 Stress distribution of Part B during apply load in B_1 direction

Table 6 Maximum stress result of part A and part B simulation

Part	Direction	Maximum Load (N)
A	A_1	5.2624
	A_1	5.2624
B	B_1	10.764
	B_2	10.764
	B_3	8.6030
	B_4	8.6030

4.4 Comparison of failure testing result and finite element simulation result

The FEA simulation result is compared with testing result as shown in table 7. It found that the experiments error is acceptable with error of about 5%. In addition, the maximum stress distribution simulation occurred at the damage area that similar to failure testing as shown in figure 16.

Table 7 Comparison of testing result and simulation result of part A and part B

Part	Direction	Maximum Load (N)		Error
		Testing	Simulation	(%)
A	A1	5.2201	5.2624	0.81
	A2	5.1666	5.2624	1.85
B	B1	10.4721	10.764	2.79
	B2	10.2331	10.764	5.19
	B3	8.1669	8.6030	5.34
	B4	8.4168	8.6030	2.21

Part A Part B

Fig 16 configuration of part A and part B failure

5. Conclusion

This research aims to reduce development time and expense in product development process by using FEA simulation to analysis. Therefore the research has studied a product failure that produces from polycarbonate. Part A and part B is feature of this product that mainly fracture cover 66% of total failure.

They are considered by using finite element failure analysis together with building fixture for test. The test is investigate by assumes force direction that may cause part failure. As the result, FEA simulation achieved less than ±5.5% error compared with testing result. In addition, the maximum stress distribution simulation that occurred on the model is similarly to failure testing result. Thus it indicates that simulation model and software are verified. The result from this study can be applied to investigate the other product failure by FEA simulation with accuracy and reliability. Eventually, the failure problem can be solved and improved rapidly.

6. Acknowledgement

The authors would like to acknowledge Seagate Technology CO., LTD. for supporting this research.

7. References

[1] J.P.F. Inberg, Fracture of polycarbonate/ABS blends. (2001). Thesis, University of Twente, Enschede, The Netherlands pp. 1-139

[2] Monika, G., Albert, J., Shih, E., and Ronald, O. (1999). Finite-Element Analysis of Stress Concentration in ASTM D 638 Tension Specimens. Journal of Testing and Evaluation, Vol. 31, No. 1.

[3] Nasser, F. and Chenhua, L. (2008). Optimal simple step stress accelerated life test design for reliability prediction. Journal of Statistical Planning and Inference, pp. 1-10

[4] Yusaku, F. and Dong-Wei, S. (2008). Impact force measurement of an actuator arm of a hard disk drive. International Journal of Impact Engineering 35 (2008), pp. 98–108

[5] Roy A. B. Engelen, Ron H. J. Peerlings, and Marc G. D. Geers. (2003). Ductile Failure Modes in Plasticity,Eindhoven University of Technology, Netherlands Institute for Metals Research

[6] Neville W. Sachs, P.E., (2005). Fracture Features, Journal of failure analysis and prevention, Vol.5,

[7] Sai Sarva, Adam D. Mulliken, Mary C. Boyce. (2007). Mechanics of Taylor impact testing of polycarbonate. International Journal of Solids and Structures 44 pp. 2381–2400

[8] Jongpradist, P., Rotbunsongsri, R., Sukkana, C., Sungtong, W. (2008). Parametric Study of Baseplate Geometry Using Finite Element Analysis, Thesis, KMUTT, Thailandx

[9] Gu, B., Shu, D.W., Luo, J. and Shi, B. J. (2006). FEA Simulation of Linear and Rotary Drop Test for Small Form Factor HDD. International conference on Mechatronics and automation June 25-28, 2006, pp. 1264-1268.