Static Analysis for Container Drop Test with a Case Study

M. Muni Prabaharan

***Abstract*—A great numbers of test are needed in designing and manufacturing of any new product to meet the safety requirements. The impact performance of new product is a major concern of a new design. The test procedure has to comply with international standards, which establishes minimum mechanical requirements, evaluates yield stress and deformation characteristics of products. The rapid developments in numerical simulation techniques, faster computing ability, and greater memory capacity, are allowing engineers to create and test new products in virtual environments. This enable design teams to consider virtually any container drop test without incurring the expense associated with manufacturing and machine time. Numerical implementation of impact test is essential to shorten the design time, enhance the mechanical performance and lower development cost. This research study deals with the simulation of impact test for a new product by using finite element analysis. Simulation was conducted to investigate the stress and displacement distributions during impact test. As a result, the use of explicit finite element method to predict the performance of new products design is replacing the use of physical test. Usually, the drop test is done by high expensive softwares where dynamic analysis can be done. But, this paper proposes the possibilities of using static analysis software.**

***Index Terms*—CAE, static analysis, new product, impact test, finite element analysis, yield stress, deformation**

NOTATIONS

K: Spring constant
F: Impact force
D: Deformation of the design object due to the impact
W: Weight of the design object
H: Height of fall

I. INTRODUCTION

The rapid developments in numerical simulation techniques, faster computing ability, and greater memory capacity, are allowing engineers to create and test new product design in virtual environments.

Through finite element analysis (FEA), these sophisticated simulations provide valuable information for designing and developing new products, as well as perfecting existing ones. Manufacturers have found this method eminently useful, as it helps them to achieve better productivity at a lower cost per unit, and develop engineering components that are easy to manufacture, and which make the most economic use of their materials.

Manuscript received December 04, 2011; revised February 18, 2012.

M. Muni Prabaharan is with New Product Development, Mahindra Satyam, Integrated Engineering Solution, Hyderabad, Andhra Pradesh (e-mail: mp.me89@gmail.com).

A. New Product Development

New Product Development industries increasingly seek to achieve higher quality level for the parts. The design of product has come to be required to be able to withstand increasingly higher stress [1]. This research study was focuses on the impact test when the product falls from a certain height. It is determined that, we can optimize the stress value during impact test in the design stage itself.

B. Drop test

An important durability test of any new product is the drop test. The test is conducted by filling the product with loads at full capacity, and dropping it from a certain height onto a hard steel or concrete floor. The product is then observed for the breakage. This conventional drop test method is costly and time consumed. With the introduction of commercially available finite element analysis (FEA) software, the drop test for the product can be digitally simulated for strength without molding and testing. FEA enables prediction the performance of the product under any realistic loading situations [2].

C. Finite Element Analysis

Finite Element Analysis is an accurate and flexible technique to predict the performance of a structure, mechanism or process under in-service or abuse loading conditions.

FEA has traditionally been associated with validating designs before committing to manufacture. However, it is now also commonly used early in a design process to try out new concepts and optimize before any physical prototypes are made and tested. Benefits include:

1) Increased innovation, as FEA encourages you to think creatively at less risk
2) Optimum rather than acceptable designs, resulting in better performance & reduced material costs, as FEA enables you to run multiple scenarios quickly and cheaply.
3) Improved understanding & control of operating envelopes, leading to higher quality & robustness, as FEA provides detailed performance information difficult to obtain from physical tests.
4) Reduced development cost & lead time, with pass/fail physical tests replaced by virtual design iterations, as FEA models are generally quicker to build than prototypes and test equipment.

With widely range of computer applications and developed, the finite element method have become numerical method in the last 40 years. It has great versatility and flexibility and can be used to solve complex boundary issues. Finite element analysis is valid modern method for the structural mechanics. It is widely used in the structure of static and dynamic characteristics, and then applied to solve the heat conduction, electromagnetic field, fluid mechanics and other continuity problems. The commonly finite element analysis software: ANSYS, PRO / MECHANICA and so on [3].

D. Advancement in Computer Technologies

The advancement in computer technologies has created a tremendous impact on several of industries, from automotive, appliance, military and aerospace industries to electrical and electronic and household industries. CAE (Computer Aided Engineering) is one of the computer technologies utilized by many OEMs (Original Equipment Manufacturer) during the design and development of products.

Today, application of CAE is commonly used to perform virtual analyses including structure, impact, drop test, thermal and computational fluid dynamic. Strength design of design objects can be accomplished by utilizing solid modeling and structural analysis software. All parts of the product can be modeled parametrically and required changes can readily be optimized via advantages that are provided by the solid modeling. Likewise, strength calculations of the designed product could be made by means of the computer aided structural analysis software.

A simplified analysis method is desirable for design engineers to perform daily quick design calculations to estimate structural sizes without the need of assistance from expensive structural simulation software [4].

II. RESEARCH PROBLEM

In order to achieve better performance and quality, the product design and manufacturing use a number of prototype tests (overload test, fatigue test, and impact test) to insure that the product meets the safety requirements. The test is very time consuming and expensive. Computer simulation of these tests can significantly reduce the time and cost required to perform a new product design.

The research objective is to optimize the stress during container drop test and to predict the failure modes. Since the cost of dynamic analysis software is expensive, we have to solve the problem by using static software itself.

III. THE RESEARCH BACKGROUND

The physics of impact necessarily involves conservation of energy and momentum. When a moving object strikes a structure the force which decelerates the mass satisfies conservation of momentum. The kinetic energy of the impacting body will be partially converting to strain energy in the target and partially dissipated through friction and local plastic deformation and strain energy 'radiated' away as stress waves. The details are very difficult to predict, but some simple estimates based on first principle can usually result is reasonable estimates for response [5], [6].

IV. ORGINALITY / VALUE

In this study, the statics finite element analysis is performed to simulate a new product during container drop test. The structural damage parameter of the product is estimated by the yield stress and the fracture criterion is based on the total Plastic work of the wheel material. Computer simulation of new product impact test can significantly reduce the time and cost required to finalize a product design [7].

V. WORKING METHODOLOGY

In this section, step by step procedure [8] is explained with a case study.

Step 1

Open the model in any static analysis software.

Step 2

Apply the impact force of 14.16 lbs at the corner where the object strikes the ground as shown in Fig. 1.

Fig. 1.

Step 3

Constrain the object at the opposite end as shown in Fig. 2.

Fig. 2.

Constraining the far end of the case is probably unrealistic but we are mainly interested in the stresses at the point of contact. This will be this highest stress area.

Step 4

See the displacement result at the load point as shown in Fig. 3.

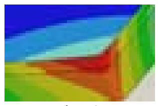


Fig. 3.

The maximum displacement is 0.1778 inch.

The results of the analysis show a force of 14.16 lbs produces a displacement of 0.1778 inches.

Step 5

To compute the spring constant for the design object

K = F / D

K = (14.16 / 0.1778) = 79.64 lb / in

Step 6

To compute the displacement at the point of impact by using the equation

D = sqrt (2WH/K)

D = 1.180 inch

Step 7

To compute the stress imposed by this displacement (from step 6)

F = KD

F = 79.64 * 1.180 = 94 lb

Step 8

Now, apply the load as 94 lb at the corner as shown in Fig. 4.

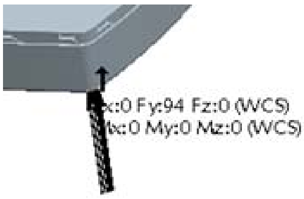


Fig. 4.

As a check, we can examine the displacement. It should be very close to 1.180 inches.

Step 9

See the results

Displacement result (shown in Fig. 5)

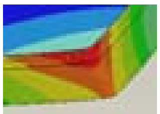


Fig. 5.

Stress result (shown in Fig. 6)

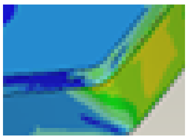


Fig. 6.

Prediction (shown in Fig. 7)

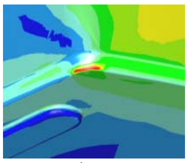


Fig. 7.

From the latest result, we can conclude that, the area (red color) where the object hits the ground, it is going to fail.

Step 10

Now, we need to change the value of radius to optimize the stress value.

VI. Optimization Summary

Iteration	Inner radius	Outer radius	Pass / Fail
Existing design	.10	.05	Fail
1	.22	.05	Fail
2	.22	.10	Pass

VII. Research Findings

The prediction of a product at impact is based on the condition that fracture will occur if the maximum yield stress of the product during the impact test exceeds the total plastic work of the product material from tensile test. The simulated results in this work show that the total plastic work can be effectively employed as a fracture criterion to predict a product fracture during impact test.

VIII. Conclusions

Design engineers can run upfront drop tests on their designs easily and quickly. Finally, we arrived the best design from the iteration 2. This methodology reduces cost to buy such expensive softwares to simulate. In the practical test, the new design is improved than previous design. Static finite element with a reasonable mesh size and time step can reliably calculate the dynamic response. The prediction of a new product is based on the condition that fracture will occur if the maximum yield stress of the design exceeds the specified material property. The simulated results in this work show that the total plastic work can be effectively employed as a fracture criterion to predict a product fracture during impact test.

Acknowledgement

I am grateful to Mr. Palanivel Subramaniyam, Product Line Leader and Mr. Karthick Srinivasan, Project Leader of Mahindra Satyam, Integrated Engineering Solution, who constantly encourages to carrying out this research.

References

[1] M. B. Ali, S. Abdullah, M. Z. Nuawi, and A. K. Ariffin, "Test Simulation using Finite Element Method" *IOP Conf. Series: Materials Science and Engineering*

[2] M. Kh. Abunawas, "A Suitable Mathematical Model of PET for FEA Drop-Test Analysis," *International Journal of Computer Science and Network Security*, vol.10 no.5, May 2010.

[3] B. Fan, G. Huang, and Maquan, "Analysis and Comparison of Application of Reducing Mill by Pro/ Mechanica and Ansys," *American Journal of Engineering and Technology Research,* vol.11, no.9, 2011

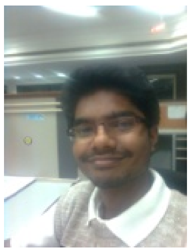
[4] A. Rios, B. Davis, and P. Gramann, "Computer Aided Engineering in Compression Molding," *The Madison Group: Polymer Processing Research Corporation, CAE*

[5] T. J. Mackin, "Calculating the impact force of a mass falling on an elastic structure"

[6] Christopher Wright P.E, "Introduction to Impact Loading"

[7] C. L. Chang, S.H. Yang, "Finite element simulation of wheel impact test," *Journal of Achievements in Material and Manufacturing Engineering,* Vol 28, Issue 2, June 2008

[8] R. Greenlee, "Chapter 6, Computing Impact Loads", *University of New Mexico*



M. Muni Prabaharan had received his Bachelor of Engineering (Mechanical) with Gold Medal from Anna University, Chennai, Tamilnadu, India in 2010. Currently, he is working in Hyderabad, India as Design Engineer for New Product Design with Mahindra Satyam, Integrated Engineering Solution. His research interests include Structural Analysis, Computer Aided Engineering, Computer Aided Designing, Design for Recycling and Project Management. He is incorporating Lean Six Sigma in Design application in his current role. He is certified Professional of Design Process and Quality Management System. He had done 65 scientific papers at International and National level as Lead author. The President of World Forum invited him as "Ambassador" for India. He honored as Outstanding Intellectual of the 21st Century, Top 100 Educators 2011, Man of the Year and International Peace Prize. He is the fellow member of IACSIT and IAENG. He had received Rajya Puraskar award from Ex- Governor of Tamilnadu, India. He had also received three awards from his institution for his Academic performance. His other roles are Scientific Committee/ Technical Committee/ Reviewer / Editorial Board Member of leading International Journals, Conferences and their Associations. He also played as "Chairman" for ISTE Chapter.