ISSN XXXX XXXX © 2018 IJESC

Research Article Volume 8 Issue No.6

A Review of Finite Element Analysis of Drop Test for Home Appliances

Amol Kamale[1], Dr. Sanjay R. Bahulikar[2]
M.Tech Student[1], Professor[2]
Department of Mechanical Engineering
Vishwakarma Institute of Technology, Pune, India

Abstract:
This paper discusses the procedure to be follow for the drop simulation of electronic home appliances. Every home appliance has to pass the drop test to ensure product will safely reach to customer. It identifies weak design points during the impact behavior of home appliance. In industry physical testing method widely used to investigate the behavior of product under drop that may occur during handling and transportation. But physical testing is very costly, time consuming and very few parameters can investigate at a time. Finite element analysis provides numerical models without spent so much money on physical testing methods.

Keywords: Finite element analysis, LS-Dyna, Home appliance, Drop test, Hypermesh.

I. INTRODUCTION

Now a day's home appliances are necessary in every household. Seeing to this many new and latest home appliances has available in the market. Manufactures are working hard to meet the demands and requirements of the consumers and the competitions in the market for these products are really high. A product will be more profitable if it delivers customer benefits and requirement, faster to market, less costs to produce. Customer will be satisfied if the product will function well throughout its life without any major issues. Therefore to maintain the integrity of structure and performance of electronics/electrical packages through the life of the product is very important. The accidental drop of these appliances during handling and transportation can harm the integrity of structure and performance of electronic/electrical components [1][2]. Several forces and accelerations acts on product when it fall, forces depends on the drop height, material, weight, shape orientation at impact surface and several other factor. Therefore it is important for designer to look at those aspects at the design stage only. At the design stage, FE method plays a more important role. It is very difficult for the designer to do the physical test of the newly designed components at the preliminary stage of the design. And experimental testing provide provides little feedback to improve the design. The FE simulation will help designers understand how components interact in the assembly and how the failure mode and mechanism is developed under the impact loading.

II. FINITE ELEMENT ANALYSYS

Finite element analysis (FEA) is the modeling of products in a virtual environment, to find out and solve the potential (or existing) structural or performance issues. FEA is the practical application of the finite element method (FEM), which is used to mathematically model and numerically solve complex structural, fluid and multi physics problems. There are two types of analysis that are used in industry 2-D modeling and 3-D modeling. While 2-D modeling conserves simplicity and allows the analysis to be run on a relatively normal computer, it tends to yield less accurate results. 3-D modeling, however, produces more accurate results while sacrificing the ability to run on all but the fastest computers effectively. Within each of these modeling schemes, the programmer can insert numerous algorithms (functions) which may make the system behave linearly or non-linearly. Linear systems are far less complex and generally do not take into account plastic deformation. Non-linear systems do account for plastic deformation, and many also are capable of testing a material all the way to fracture. FEM allows entire designs to be constructed, refined and optimized before the design is manufactured. This powerful design tool has significantly improved both the standard of engineering designs and the methodology of the design process in many industrial applications. The use of FEM has significantly decreased the time to take products from concept to the production line. One must take the advantage of the advent of faster generation of personal computers for the analysis and design of engineering product with precision level of accuracy.

III. FLAT DROP PROCEDURE IN LS-DYNA

A drop test involves orienting an object with respect to an assumed gravitational field and allowing it to drop from some specified height under the influence of gravity onto a flat, rigid surface. In a typical drop test, the object is dropped from rest and the target lies in a plane perpendicular to the direction of the acceleration due to gravity (g). The drop analysis predicts deformations and stresses in the object over a user-specified time interval to characterize the implications of the impact of the object with the target. General Drop Test Analysis procedure that many times followed in industry is as follows, (Note: This procedure may vary from industry to industry).

Step 1: Import the CAD model
Designer must import the model of the object to be dropped in meshing tool like Hypermesh. It is possible to import the CAD geometry in the formats like PRT, ASM, STEP, IGES, UG, Parasolid etc. in Hypermesh.

Step 2: Generate meshing and connections
Mainly three types of meshing are used i.e. 1D, 2D/Shell Meshing, 3D/Solid Meshing. Select the meshing type Based on

the part dimensions, complexity and material. For each type of meshing there are different types of element shape is available. Mesh quality criteria like Accept ratio, warpage, skew, minimum length, Jacobian, Minimum and Maximum quad angle should be satisfied.

Step 3: Define the material properties

There are number of material models available in LS-Dyna according to material behaviors (elastic, plastic, brittle, rigid, foam, ceramics etc.). Material property is defined with the help of Mat_keyword is used. Example, MAT20: MAT_RIGID.

STEP 4: Define the magnitude of initial velocity and Gravity The Load collectors are used to define initial velocity, acceleration due to gravity (g), body loads, temperatures etc.

STEP 5: Create rigid base for drop

To drop the product a rigid base is required. Therefore model the rigid base below the component at distance equal to the actual distance to be simulated.

STEP 6: Specify contacts between assembly components

In drop test, contacts between the assembly components need to be defined. But in LS-dyna defining automatic contact for those components of assembly is possible.

STEP 7: Specify solution controls

In this step we have to specify when the analysis starts (near impact time or at drop time), as well as the run duration time after impact. Under the Number of Results Output heading, specify the results to be analysed from this drop test.

STEP 8: Solve

Before solving, make sure that you are satisfied with the information in the Drop Test Set-up dialog box. Initiate the LS-DYNA solution.

STEP 9: Post-processing

LS-Dyna solver generates the D3plot, D3plot1, D3plot2 ….so on. Check all energy curves, drop direction its behavior by animation and the stress values and Plastic strains in assembly.

IV. LITERATURE REVIEW

Oguzhan Mulkoglu, Mehmet A. Guler, & Hasan Demirbag, examined the nonlinear explicit finite element code LS-DYNA® is used for the drop impact simulations. They build a finite element analysis (FEA) model of the drop test of the free-standing dishwasher structure in order to determine the critical regions in the assembly. The dishwasher is dropped from a predefined height with an initial velocity in order to decrease analysis time. For all parts finite element mesh is generated using HYPERMESH. The mesh size is chosen to be 8 mm for the parts where there is not much deformation and a minimum mesh size of 2 is used for the critical parts. Fully integrated shell element formulation is used for shell elements since it decreases analysis time compared to other formulations. Fully integrated solid elements (S/R solid elements) are chosen to prevent excessive distortions that lead to negative jacobian errors. MAT 20 is defined for rigid wall. Type 24 which is an elasto-plastic material type, is used for steel parts of the dishwasher. Type 124 used for plastic components of dishwasher. Type 63 material model is used for packaging module which is made from Expanded Polystyrene Foam (EPS). They have validated the foam material model with a simple foam compression test and compared with the simulation results in LS-DYNA. *AUTOMATIC_SINGLE_SURFACE contact card is used to define the self-contact of each part. Contact between the rigid wall and the dishwasher is defined with *AUTOMATIC_SURFACE_TO_SURFACE contact card. Contact between corner post foams and the bottom foam is defined with *CONTACT_TIED_NODES_TO_SURFACE card. Simulation time is reduced by decreasing the drop height of the dishwasher and the analysis is started at that point with an *INITIAL_VELOCITY_GENERATION card. Performed finite element simulations for inclined to side and vertical drop tests. In vertical drop test simulation observed the critical regions on the components that come to contact with bottom foam. In inclined drop test simulation the parts at the inclined side of the dishwasher is the most affected parts during impact to the ground.

D. Hailoua Blanco, A. Ortalda, F.Clementi have described the methodology to perform the drop test of the Eletrolux refrigerator with packaging in LS-DYNA to verify the performance and suitability of the packaging and its interaction with the structure in case of damage occurring during transportation or delivery. In this study they have divided the simulation process in 6 tasks. In material characterization task they have used *MAT_24 (Piecewise linear plasticity) for steel panels and thermoplastic materials, MAT_083 (Fu Chang Foam) for packaging and cabinet foams and MAT_181 (Simplified Rubber Foam) for compressor support pads. In refrigerator modeling task they simplified the geometry of refrigerator to reduce the number of elements. For solid elements of foam they used 4 mm of mesh size. Shell elements provided to boundary components of refrigerator. Then they meshed the packaging with 1 point integration tetrahedron elements. After meshing the provided contact by (*CONTACT_TIED_NODES_TO_ SURFACE) and (*CONTACT_ AUTOMATIC_ GENERAL) card. They simulated free fall edge impact at 10 degrees with respect to the floor. The falling height is set up to 250 mm, which corresponds to an impact speed of 2.21 m/s. In Experimental testing, they have identified two kinds of problems, the compressor crossbeam excessively bent during the impact, side wrapper bend at bottom and simulation exactly replicated the observed experimental damage. Lim & low (2002) examined the drop impact response of portable electronic products at different impact orientations and drop heights. Measurement of level of the shock experienced by electronic components on the printed board circuit (PCB) during impact is done. During drop impact examined the impact force, strain and the level of shock induced at the PCB. Experimental drop test is done with the help of Impact drop tester which was capable to drop portable device at any orientation and height. Accelerometer is used to register shock experienced in PCB, and strain gauge is used to register the dynamic strain induced in the PCB. Load cell is used to measure the impact force experienced by the cellular phone and Oscilloscope is used to capture the drop impact response from accelerometer, strain gauge and load cell. Five tests performed with different height and orientation. Observed increase in impact force with increase in drop height. PCB was bending in compression and tension during and after impact. Acceleration level arising from the same drop height was different because of different orientation. T.T. Mattila, L. Vajavaara, J. Hokka, E. Hussa, M. Mäkelä, V. Halkola (2014) performed the controlled product level drop tests with the help of the customized board-level drop tester (Salon Teknopaja Oy). Eight smart phone devices of different brand were dropped only in the horizontal (display side downwards) orientation on the basis of the prior knowledge that impacts perpendicular to the display or the back of the device are the most severe. The mechanical response of the devices to the shock impact was characterized with the help of a miniature accelerometer and strain gauges with varying sizes and numbers of axes. Wei Liu, Hongyi Li discussed the drop test simulation of a new cell phone design whose steel band is split into three segments. LS-DYNA, the non-linear explicit FEA code was selected for the analysis due to its robust capability of handling impact phenomenon. The cell phone was dropped on the garnet floor from the height of 1m with different orientations including face drop, edge drop, corner drop, tilt face drop. Due to introduction of split band did increase the stresses for the cover glass and other glass layers. E. Tempelman, M.M.S. Dwaikat, C. Spitás investigated experimental and analytical study on the dynamic behaviour during impact of portable products with internal shock mounting. The analysis was carried out using elastic impact theory, rigid body kinematics, and dynamics equations. Hertzian non-adhesive elastic contact theory is used to compute the impact force, contact time, and compressed depth of contact. Impulse dynamics is used to compute the response of the box system and the internally-mounted plate. Special drop tower with guiding frame used for controlled-angle free-fall drop impact, representative products are dropped at different angles and the acceleration is recorded both on the outer case and on an internally-mounted plate using accelerometer. Karen E. Jackson and Edwin L. Fasanella, developed the LS-DYNA simulation of a vertical drop test of an ATR42-300 twin-turboprop high-wing commuter-class airplane. They performed the drop test of aircraft on the concrete impact surface using Dynamic Drop Test Facility at the FAA Technical Center. The purpose of the test was to evaluate the dynamic structural response of the aircraft when subjected to a severe, but survivable, impact. The total weight of the aircraft was 41900 lb. The finite element model of the aircraft was developed from geometric data gathered from direct measurements of the aircraft. The geometry model was discretized into a finite element mesh, element and material properties were assigned, contact and initial velocity conditions were defined, and the model was executed to generate analytical predictions of structural deformation and time history responses. The model was simulated in LS-DYNA, a commercial code for performing explicit transient dynamic simulations. The analytical predictions correctly validated the major damage mode seen during the test which was collapse and failure of the fuselage structure beneath the wing. The predicted acceleration responses showed a high level of correlation with the test data.

V. CONCLUSION

Finite element analysis (FEA) is one of the most important tools in designer’s arsenal of digital tools for design and analysis of products and processes. FEA has become a vey good solution for identifying the failures due to impact of the product by showing problem area in a components and assemblies, and thus allowing designer to see all of the theoretical stresses within the product failure criteria and help to select the parameters of the components. FEA analysis provides the information about the number of failure that can occur during physical testing and reduces the manufacturing costs by reducing the number of physical prototype testing.

VI. REFERENCES

[1]. Oguzhan Mulkoglu, Mehmet A. Guler, & Hasan Demirbag, "Drop Test Simulation and Verification of a Dishwasher Mechanical Structure.", 10th European LS-DYNA Conference 2015, Würzburg, Germany.

[2]. Gurjinder Singh, Jaswinder Singh, "A Review on Finite Element Analysis of Free Fall Drop Test on Mobile Phone", ISSN (Online): 2319-7064.

[3] Lim C.T. & Low Y.J. (2002), “Investigating the drop impact of portable electronic products,” Electronic Components and Technology Conference, pp. 1270 –1274.

[4]. Mattila T.T., Vajavaara L.,Hokka J.,Hussa E., & Halkola V., (2014), “Evaluation of the drop response of handheld electronic products.” Microelectronics Reliability, pp. 601 – 609.

[5]. Liu W., & Li H., (2011) "IMPACT ANALYSIS OF A CELLULAR PHONE,” 4th ANSA & μETA International Conference, June 1-3 2011.

[6]. Tempelman E., Dwaikat M.M.S., & Spitas C., (2012), “Experimental and Analytical Study of Free-Fall Drop Impact Testing of Portable Products.” Experimental Mechanics, pp. 1385 – 1395.

[7]. Meng-Kao Yeh, Tzu-Heng Huang, “Drop Test and Finite Element Analysis of Test Board”, Procedia Engineering 79 (2014) 238 – 243.

[8]. Karen E. Jackson and Edwin L. Fasanella,” Development of an LS-DYNA Model of an ATR42-300 Aircraft for Crash Simulation” NASA Langley Research Center Hampton, VA 23681-2199.

[9]. Module 1: Introduction to Finite Element Analysis, NPTEL, IIT Kanpur.

[10]. Chapter 8: New Product Development* by John R. Hauser, MIT and Ely Dahan.

[11]. ANSYS LS-DYNA User's Guide.