Simulations on Structure Performance of 3C Thin-Wall Injection-Molded Parts

Shia-Chung Chen, Hsing-Ling Wang, Juan-Po Chen, Hsin-Shu Peng

Mechanical Engineering Department, Chung Yuan Christian University, Chung-Li, Taiwan 32023, Republic of China

Received 8 May 2001; accepted 25 March 2002

ABSTRACT: Manufacturing of 3C (Computer, Communication, and Consumer Electronics) products toward weight reduction, thin-wall, and minified-size is an inescapable trend for the future 3C industries. However, the induced damage information from drop impact, including exterior housing fracture, liquid crystal display (LCD) cracking, solder-joint breaking, or interior component failure, is still derived experimentally and involves very complicated parametric analyses, such as a dynamic impact process, drop orientation, contact behavior, and large deformation during the impact instance. In the present study, numerical simulations for the drop test and bending strength were applied to a thin-wall computer dictionary (Model CD-66) housing to understand the key factors that affect the part drop test performance. The appropriate modeling that would affect simulation accuracy as well as the associated nodal degree of freedom and computer time were also investigated. A housing of CD-66 was redesigned to be 1 mm thick and structurally verified with two different plastics: polycarbonate (PC) and acrylonitrile butadiene styrene (ABS). The simplification of the PC board and LCD backlight circuit in finite element modeling (FEM) only causes about a 10% difference, while saving many modeling costs. The numerical simulations also indicate that both its bending strength and drop-impact strength were decreased only about 5%, whereas the product quality still met its strength requirement if only the top housing plate thickness was reduced while the remaining sidewall thickness was kept unchanged. © 2002 Wiley Periodicals, Inc. J Appl Polym Sci 86: 3064–3071, 2002

Key words: 3C plastics product; structure performance; numerical drop test; drop/impact force; bending test simulation

INTRODUCTION

Portable telecommunication devices, such as cellular phones, personal data assistants (PDAs), portable notebooks, and computer dictionaries, are widely utilized in everyday activities. Weight reduction, for multitelecommunication functions within a compact design, consequently, becomes the most appealing and competitive characteristic in the consumer electronics market. However, the drop-impact-induced failure for such a thin-wall structure is of great concern due to product reliability and performance in usage. Thus, the current study investigated some crucial design criteria on the part performance during a drop test for 3C products.

The induced damage from drop impact may include exterior housing fracture, liquid crystal display (LCD) cracking, solder-joint breaking, or interior component failure. These failure models involve very complicated parametric analyses such as the dynamic impact process, drop orientation, contact behavior, large deformation, and path of the elastic propagating wave front during the impact instance. Owing to a highly competitive environment, devices must be manufactured and delivered to the market in mass quantity and good quality within a very short period; at the same time, design failure has to be eliminated as much as possible before the product is merchandised. Conventionally, a product reliability test to prevent impact-induced damage is carried out empirically by a design–prototype–test–redesign procedure. Physical prototype drop test procedures have been performed and verified[1–9]; however, high cost, time consumption, and lack of analytic information are major deficiencies of such a test. A repetitive design/post tooling/redesign engineering study not only delays the production schedule, but also increases the product manufacturing cost. Moreover, the design methodology can rarely be derived from such drop tests because it is quite difficult to mount sensors at the desired locations in a small, compact product to obtain the required test information. Computer simulation, on the other hand, can provide more comprehensive mechanical information at any location of the analyzed objects as compared with those segmental messages acquired from sensors during a physical test. Simulation-based analysis can be performed in every design stage in an efficient and timely manner. By providing detailed information for the drop test event, computer simulation can reduce the number of physical drop tests

---

Correspondence to: S.-C. Chen (shiachun@cycu.edu.tw).

Contract grant sponsor: National Science Council; contract grant number: 89-2622-E194-001.

Contract grant sponsor: Chung Yuan Christian University.

Journal of Applied Polymer Science, Vol. 86, 3064–3071 (2002)

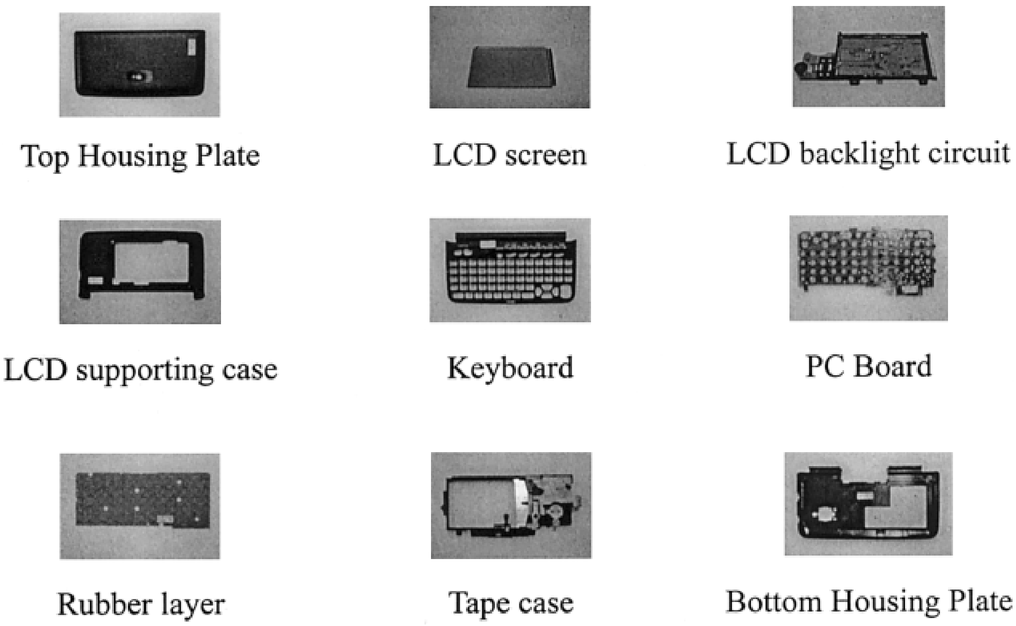
Figure 1 Computer dictionary (CD-66).

required for product certification. This, in turn, reduces the time and costs associated with the product development cycle. To obtain accurate predictions from computer simulations, the right tool, the right methodology, and engineering experiences are essential.

In the present study, a Belta computer dictionary (Model CD-66) with a 1.6-mm original thickness in the top housing was utilized as an application case for the structural performance of a bending-strength analysis and drop-impact simulation. The drop-inpact simulations were carried out under different finite element modelings (FEMs), particularly in the interior components of the CD-66. Simulation accuracy and efficiency regarding model simplification were studied and the results are discussed. The top housing of the CD-66 was redesigned to be 1 mm thick and structurally verified with two different plastics: polycarbonate (PC) and acrylonitrile butadiene styrene (ABS). Bending and drop-impact performance were then simulated and the results are discussed.

METHODOLOGY OF NUMERICAL SIMULATION: CD66 MODELING

Numerical simulations based on FEM have been widely applied in industrial applications. The computer dictionary CD-66 and its disassembled components are shown in Figures 1 and 2, respectively. In this study, commercial FEM software (MSC® Dytran[10]) was utilized for the numerical analyses. Several key factors including simulation accuracy and efficiency due to FEM simplification, unbalanced weight distribution modeling, and structural redesign criteria are presented in this section. Attention is focused on the stress level and distribution on the top housing of the computer dictionary CD-66. The full FEM consists of all major parts: top housing plate, LCD screen, LCD backlight circuit, LCD supporting case, keyboard, PCB, rubber layer, tape case with metal insert, and bottom housing plate, as shown in Figure 3. Several major integrated circuit (IC) components were attached on a uniform-weight plate to model the PCB and LCD backlight circuit. The FEM contains totally 9432 shell elements, 348 solid elements, and 156 discrete elements (beams and rigid link pairs). The time step used in this study is of seconds. It is noted that both the PCB and LCD backlight circuit are modeled



Top Housing Plate
LCD screen
LCD backlight circuit
LCD supporting case
Keyboard
PC Board
Rubber layer
Tape case
Bottom Housing Plate

Figure 2 Major nine components of CD-66.

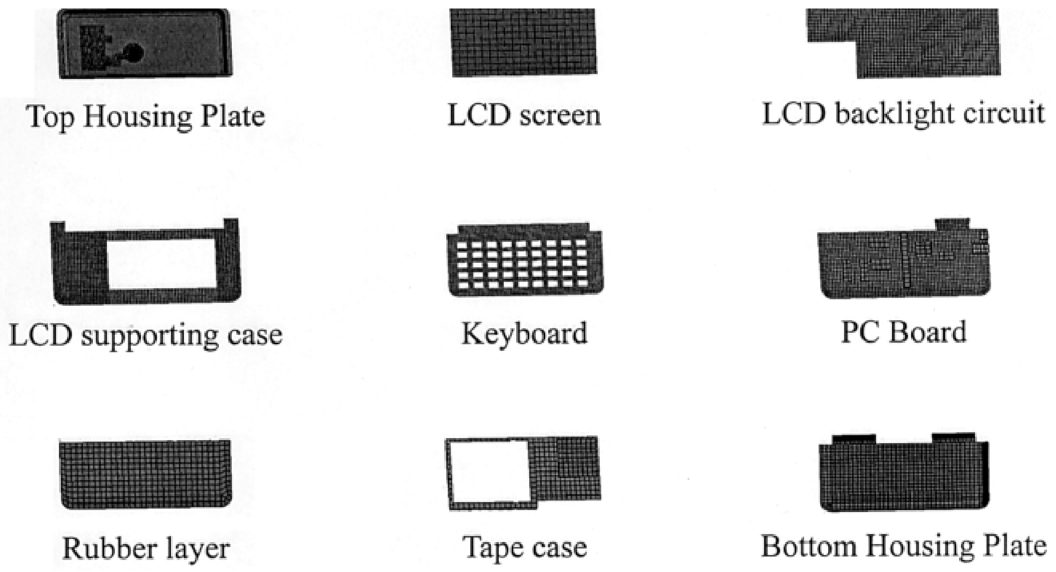


Figure 3 FEM for nine components of CD-66.

by a uniform-weight plate attached with up to eight major IC components modeled as solid elements. In addition, there are 21 contact surfaces defined for the possible drop contact area. The drop speed when the model touches the contact surface is assumed to be 4.89 m/s, which is equivalent to a free drop condition from a 1.2-m height to the ground. Table I(a,b) lists the number of elements, material properties, and weight and thickness of the associated components, both of ABS and PC materials, used for the simulations. Details of the simulation methodology and numerical FEM were reported elsewhere.[11]

SIMULATED RESULTS AND DISCUSSIONS

Impact stress analysis: simulation accuracy and efficiency with respect to weight distribution

Stress distributions in each component of the CD-66 are shown in Figure 4 and the corresponding maximum stress values, both of ABS and PC material, are listed in Table II. Figure 5(a,b) gives the impact velocity and force responses of the contacting point at the top housing. It was found that both the top housing and the bottom housing are subjected to a greater stress level upon dropping as compared with other

TABLE I
Model Properties: ABS and PC

Part	Component weight (g)	*E* (GPa) Young's modulus	*ν* Poisson ratio	Thickness (mm)	No. elements
(a) ABS					
Top housing	17.4	1.72	0.38	1.6	1786
LCD screen	47.8	65	0.2	2	209
LCD backlight circuit	28.5	14	0.2	1	1116
LCD supporting case	12.77	1.72	0.38	1	1145
Keyboard	22.31	1.72	0.38	2	1566
PCB	30.19	14	0.2	1	1882
Rubber layer	2.54	2.4	0.3	0.5	720
Tape case	36.29	1.72	0.38	1	422
Bottom housing	27.30	1.72	0.38	2	934
(b) PC					
Top housing	19.8	2.8	0.38	1.6	1786
LCD screen	47.8	65	0.2	2	209
LCD backlight circuit	28.5	14	0.2	1	1116
LCD supporting case	12.77	2.8	0.38	1	1145
Keyboard	22.31	2.5	0.38	2	1566
PCB	30.19	14	0.2	1	1882
Rubber layer	2.54	2.4	0.3	0.5	720
Tape case	36.29	2.8	0.38	1	422
Bottom housing	31.08	2.8	0.38	2	934

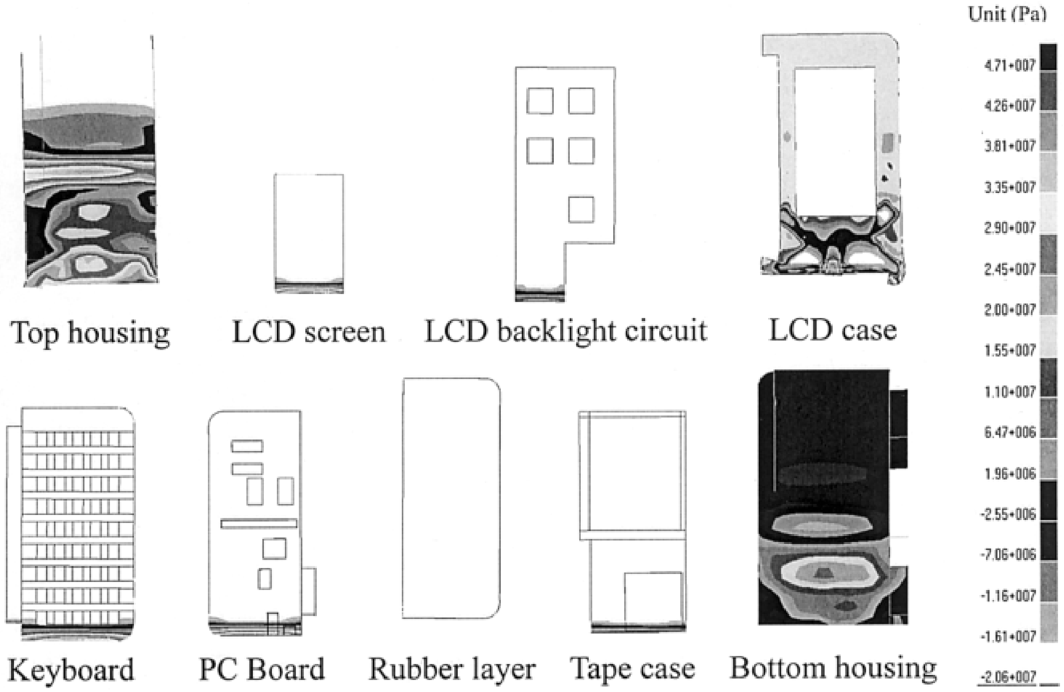


Figure 4 Stress distributions in nine components of CD-66 (at 3.3 ms after impact).

interior components. The simulation also indicates that stress decays significantly as the distance from the impact location increases. Comparison between the two different materials indicates that ABS has a higher impact stress than has PC; on the other hand, it is apt to initiate damage due to lower mechanical stiffness.

In a complete CD-66 model, several major IC components along with a socket and numerous small electrical components are attached to a uniform-weight plate to model the PCB and LCD backlight circuit. Small electrical components are made of different materials with different complex geometries, which will definitely increase the FEM time. A simplified modeling alternation is to use different numbers of IC components while keeping the total weight, mass distribution, and location of the gravity-centroid of the PCB/LCD backlight circuit unchanged. The same simulation process is now applied to the LCD backlight circuit for changing the mass distribution and location of the gravity-centroid of the LCD backlight circuit. Based on the unbalance weight evaluation, the numerical simulations reveal that the maximum stress value of the PCB model under a uniform weight distribution assumption, that is, a uniformed plate model, shows about a 3.6% difference for the ABS material and 1.3% for the PC material, as compared with that of the complete model. This simplification of the PCB model, however, only causes about a 4.3% difference for the ABS material and a 2.3% difference for the PC material, as compared with its complete model in the prediction of maximum stresses for both top housing and bottom housing (Table III).

In the case of model simplification for the LCD backlight circuit, the effect of the unbalance weight distribution modeling on the impact stress on the LCD backlight circuit is even trivial for both the top/bottom housing and the LCD backlight circuit, as shown in Table IV. Because of the light weight of the IC components, they exhibit less effect on the stress level of the top housing and the bottom housing. The present investigation demonstrates that the simplification in FEM for the PCB and LCD backlight circuit is acceptable when the structure performance of the top housing or bottom housing is the major design concern. However, the simplification of the PCB and LCD

TABLE II
Maximum Stresses (MPa) in Major Components of CD-66 (Full Model)

Part	MPa	
	ABS	PC
Top housing	46.84	41.67
Bottom housing	46.75	41.53
LCD screen	12.63	12.41
LCD backlight circuit	15.20	15.04
LCD supporting case	32.15	30.02
Keyboard	33.24	31.75
PC board	16.59	16.37
Tape case	22.61	21.39
Rubble layer	1.07	1.07

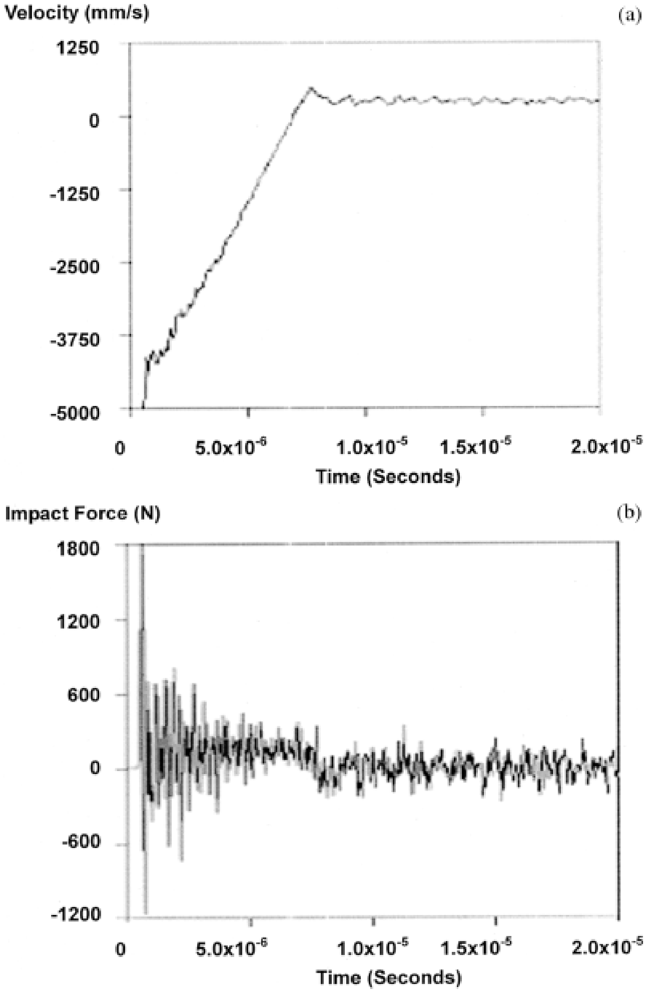


Figure 5 (a) Impact velocity response of contacting point at top housing; (b) impact force response of contacting point at top housing.

backlight circuit model is not suitable to detect the disconnection and the breakage of the concerned small components.

Structural design: maximum bending stress analysis

Structural stiffness is considered to be a critical design parameter when a component develops into thin-wall, lightweight product. The influential factors for structural performance may result from the criteria for part design, materials selection, and mold design, verification of CAE simulations, as well as the effect of processing conditions on the part properties. To thin the housing without a great loss of part strength, design guideline should be established, and verification from CAE simulation provides an excellent engineering investigation into these design criteria. Assuming two line-support and two line-load sources as shown in Figure 6, the maximum stress was found to be around the corner along the side. As a consequence, the sidewall structure holds more external stress than does the middle structure of the panel. From the above observation, the top plate thickness and sidewall thickness can be considered as two separate design parameters as the product structural strength criteria are considered. The percentages in maximum stress under a given four-point bending loading are listed in Table V, assuming that different thicknesses for the top housing plate and the sidewall are redesigned. It is noted that if the whole part thickness was reduced to 1 mm the maximum stress will increase by about 55% under the same load. However, if the plate thickness is reduced to 1 mm whereas the sidewall is kept as 1.6 mm thick, the maximum stress level increases only by about 4%. The analytic results demonstrate that such a thinning structure design is plausible if the part thickness can be reduced at the region where the structure loading is not critical. Based on such a design strategy, a product engineer is able to redesign the thinning structure through CAE simulation, at the same time keeping the part in a desired strength performance.

TABLE III
Maximum Impact Stresses (MPa) for PCB under Different Weight Distributions

	MPa (% difference)[a]					
	ABS			PC		
Model	Top housing	Bottom housing	PCB board	Top housing	Bottom housing	PCB board
Full model (plate + 1 socket + 8 IC components)	46.84	46.75	16.59	41.67	41.53	16.37
Uniformed plate	48.78 (4.1%)[a]	48.74 (4.3%)	17.18 (3.6%)	42.63 (2.3%)	42.47 (2.3%)	16.58 (1.3%)
Plate + 1 socket	47.63 (1.7%)	47.68 (2.0%)	17.05 (2.8%)	42.15 (1.2%)	42.28 (1.8%)	16.74 (2.3%)
Plate + 1 socket + 2 IC components	46.98 (0.3%)	46.92 (0.4%)	16.94 (2.1%)	41.76 (0.2%)	41.74 (0.5%)	16.65 (1.7%)
Plate + 1 socket + 4 IC components	46.88 (0.1%)	46.80 (0.1%)	16.76 (1.0%)	41.68 (0.0%)	41.64 (0.3%)	16.50 (0.8%)
Plate + 1 socket + 6 IC components	46.84 (0.0%)	46.75 (0.0%)	16.63 (0.2%)	41.67 (0.0%)	41.53 (0.0%)	16.39 (0.1%)

[a] Compared with full-model value.

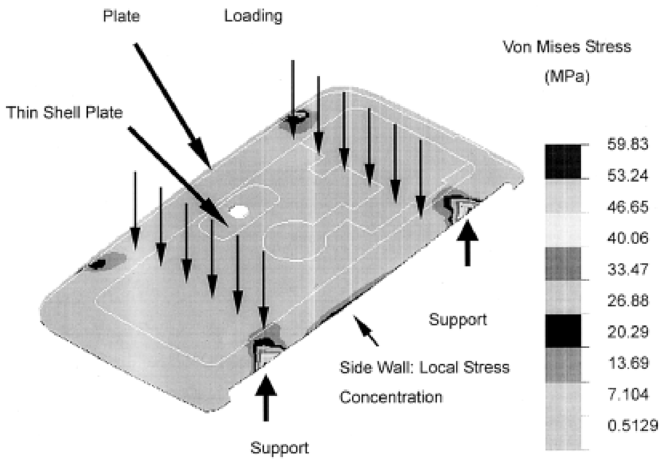


Figure 6 The Von-Mises stress analysis by FEM simulation with a thickness of 1.6 mm for both plate and sidewall.

Structural design versus maximum impact stress variation

Following the above design philosophy, the simulated stress values with different thickness designs are listed in Tables VI and VII for the top housing of PC and ABS plastics. Drop orientations at the impact moment, which play an important role on the stress level for housing, are shown in Figure 7(a,b). Six different drop orientations for the top plate surface (top face of the CD-66 housing) and side surface are simulated. The percentage increases in maximum impact stresses are also listed in Tables VI and VII. In the current work, only the impact stress level on the top and bottom housings is considered.

It is noted that if the whole part thickness (PC case) was reduced to 1 mm the impact stress level will increase about 40% more in various drop orientations, resulting in a weak impact performance. However, if only the plate thickness is reduced to 1 mm whereas the sidewall thickness is retained as 1.6 mm, the maximum stress level only increases less than 5%. This leads to the design criterion that part thickness can be reduced more at the region where the structure loading is not critical.

TABLE V
Increasing Percentage of Stress Under Different Thickness Designs (PC)

	Percentage increase in maximum stress		
	Sidewall thickness (mm)		
Plate thickness (mm)	1.6	1.2	1.0
1.6	0	—	—
1.2	2.8(%)	31.5(%)	—
1.0	4.1(%)	33.8(%)	54.9(%)

CONCLUSIONS

The advantages of computer simulation are that an engineer can predict the product performance prior to production, thus reducing manufacturing cycling time, improving product reliability, and combining more complicated engineering technologies into future development. Drop/impact simulation plays an important role in the determination of the product design for 3C thin-wall products. Experimental validation, robust modeling, analysis skill, specification of typical inspection cases, and material property databases are the key factors to a future application. Using the CD-66 as an illustrated application, the present study carried out preliminary bending and drop-impact simulations. Simulation accuracy and efficiency regarding model simplification, particularly of interior components, were investigated. Investigation regarding wall thickness reduction of the CD-66 while retaining its structural performance was also employed. From the simulated results, the following was found:

1. From the full model analysis, the top housing and bottom housing, where impact contact was initiated, are subjected to the greatest stress level. The impact stress wave propagates inside the top and bottom housings and decays quickly as the distance from the contact location increases. The inner components exhibit less impact stress upon dropping if those link components are not properly modeled (not modeled in the current study).

TABLE IV
Maximum Impact Stresses (MPa) for LCD Backlight Circuit Under Different Weight Distributions

	Impact Stress MPa (% difference)[a]					
	ABS			PC		
Model	Top housing	Bottom housing	LCD backlight circuit	Top housing	Bottom housing	LCD backlight circuit
Full model (plate + 5 IC components)	46.84	46.75	15.20	41.67	41.53	15.04
Uniformed plate	46.88 (0.1%)[a]	46.78 (0.1%)	15.31 (0.7%)	41.65 (0.0%)	41.55 (0.0%)	15.09 (0.3%)
Plate + 3 IC components	46.84 (0.0%)	46.75 (0.0%)	15.24 (0.3%)	41.63 (0.1%)	41.53 (0.0%)	15.06 (0.1%)

[a] Compared with full-model value.

TABLE VI
Maximum Impact Stresses (MPa) Versus Different Drop Orientations with Various Combinations of Thickness Design Dropped at Different Orientations (Top Housing, PC)

	Orientation					
Design (Plate/sidewall)	I	II	III	IV	V	VI
(1.6/1.6 mm) Original design	41.67	37.74	41.26	35.41	40.43	49.65
(1.0/1.0 mm)	60.15	48.39	59.74	46.28	58.66	75.52
Maximum stress increase versus original design (%)	44.3%	28.2%	44.8%	30.7%	45.1%	52.1%
(1.0/1.6 mm)	43.13	38.81	42.81	36.46	42.04	51.78
Maximum stress increase versus original design (%)	3.5%	2.83%	3.75%	2.96%	3.98%	4.29%

TABLE VII
Maximum Impact Stresses (MPa) Versus Different Drop Orientations (Top Housing, ABS)

	Orientation				
Design (Plate/sidewall)	I	II	III	IV	V
(1.6/1.6 mm) Original Design	46.84	42.59	46.44	40.37	45.56
(1.0/1.0 mm)	66.4	55.3	68.12	53.8	66.51
Maximum stress increase versus original design (%)	41.7%	30.0%	46.6%	33.7%	46.0%
(1.0/1.6 mm)	48.85	44.03	48.66	41.94	47.88
Maximum stress increase versus original design (%)	4.3%	3.4%	4.8%	3.9%	5.1%

Different modelings in the gravity-centroid location of the IC components (unbalanced weight distribution) result in a slight influence on the impact stress of the exterior housings.

2. The simplified models of the PCB and LCD backlight circuit do not lose simulation accuracy significantly in the housing impact stress analysis. This indicates that one can neglect the detailed

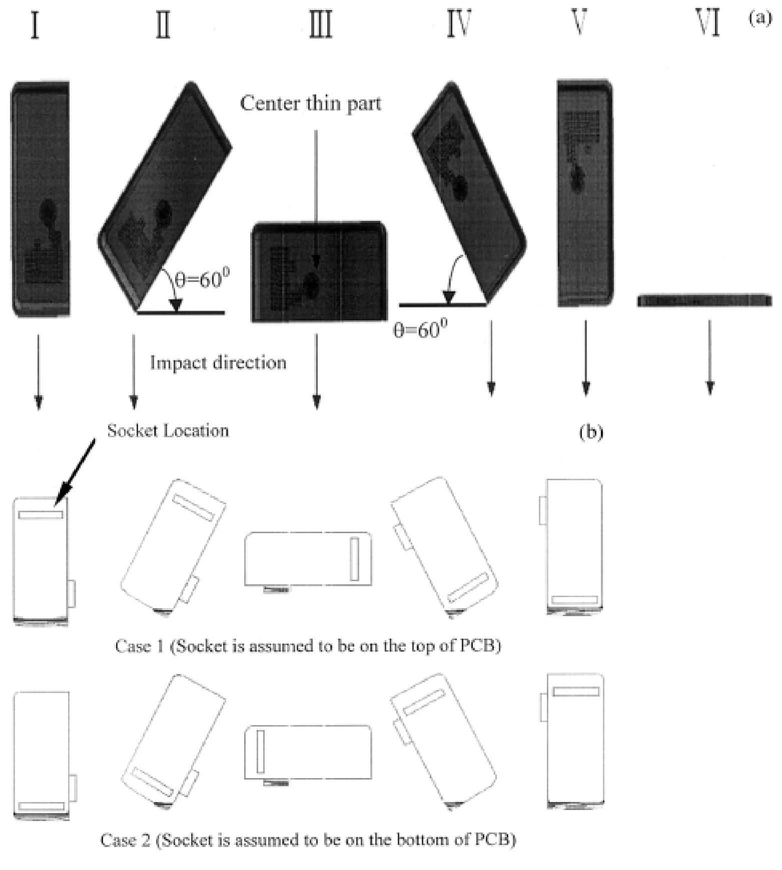


Figure 7 (a) Drop impact under various orientations; (b) stress distributions in PCB under various orientations (at 3.3 ms after impact).

modeling for interior components of the circuit boards, while saving much modeling cost, particularly in the preliminary product design stage. During model simplification, if material property averaging and mass distribution assignment are reasonably assumed, simulation differences can be significantly reduced.

3. A design strategy of the thin-wall product is well established through CAE simulation for structural performance analysis, such as an indication for the critical region of the stress concentration effect and quantitative prediction of the structural redesign performance.
4. If only the top plate thickness of the CD-66 was reduced whereas the sidewall thickness was retained, the drop-impact performance of the top housing showed an insignificant change.
5. Comparison between two different housing plastics, PC and ABS, indicates that PC has a superior antishock capacity compared to that of ABS. All simulated impact stresses for ABS are higher than are the results for PC.

This research was supported by the National Science Council (NSC Grant 89-2622-E194-001) and a Distinguished Research Grant of Chung Yuan Christian University.

References

1. Kessler, S. In Symposium Sponsored by ASTM Committee D-20 on Plastic, March 1985.
2. Lei, W.; Yao, M. Eng Fract Mech 1996, 55, 25.
3. Lei, W.; Yao, M. Eng Fract Mech 1996, 55, 645.
4. Raymon, M. B. Mechanical Impact Dynamics; Wiley: New York, 1991.
5. Wu, J. In 4th International LS-DYNA3D Conference, Minneapolis, Sept., 1996.
6. Wu, J. In 97 Summer Advanced Mechanical Technology of Motorola, Chicago, July 1997.
7. Wu, J. In 97 Winter Advanced Mechanical Technology of Motorola, Phoenix, Jan. 1997.
8. Wu, J.; Song, G.; Yeh, C. P.; Wyatt, K. In International Society Conference on Thermal Phenomena, 1998; p 330.
9. Wu, J. Global and Local Coupling Analysis for Small Components in Drop Simulation; Mechanical Technology Center, PCS, Motorola, 2000.
10. MSC® Dytran User Manual, 2, Version 4.7, 1999.
11. Chang, C.-T.; Chen, S.-C.; Chen, J.-P.; Peng, H.-S.; Huang, L.-T. In 24th National Conference on Applied and Theoretical Mechanics, Taiwan, ROC, 2000; p Q108.