# Advanced Modeling and Drop Simulation With New Features of LS-DYNA

Jason Wu
familyofwu@hotmail.com

## Abstract

*Phone drop simulation is an important application of LS-DYNA in electronics industry. The application has been widely used in all leading companies to produce electronic handsets, like Nokia, Motorola, and Samsung. Since more new features complemented in 970 version of LS-DYNA, phone modeling and drop simulation strategy has changed a lot to obtain more accurate and quick results. This paper introduces new methods, and recent development of phone modeling and drop simulation, based on author's experiences in recent years. The paper is focused on using right element type in LS-DYNA element library and some new strategies to establish better phone model and drop simulation. The paper provides benchmarks to verify the modeling strategies. The paper still gives an example for advanced application of LS-DYNA in multiply impact simulation.*

## Introduction

Since computer simulation technique was applied into cellular phone drop in 1996, computer technology and engineering analysis codes have obtained great progress. Over 100 material models, new contact types, and 10-node tetra elements have been introduced in 970 version of LS-DYNA. Affordable servers equipped with Intel Itanium or AMD Opteron CPU, can finish drop simulation of a complex phone in several hours. The progress of simulation tools gives wide space for modeling upgrade. Now, a model with various element types, consisting of 200 - 300 thousands elements are practicable for drop simulation, instead of the previous model in dozens of thousands elements, with shell as major element type. As computer becomes faster and faster, the model size turns to bigger and bigger. How to efficiently organize a model to obtain more accurate mechanical behavior still is a challenging subject in front of FEA performers. This paper presents some new strategies to establish a phone model, based on author's practice and study on phone drop simulation in recent years.

## Element type selection in phone modeling

A typical phone model usually has 200K-300K elements divided in 50 – 70 parts. Almost all element types in LS-DYNA will be used into the modeling, based on their geometrical characteristics and deformation modes. Tetrahedral elements are used in modeling of housing and other parts in complex geometry. With the type of elements, user can take advantage of auto-meshing to handle complex geometry. Hexahedral and pentahedral elements are used in the parts in relatively simple geometry, and subjected to 3D complex stress. The brick elements have better performance in explicit codes, but need more manual effort to create mesh. In shell element types, thin shell and thick shell are all recommended. Thin shell element is priority for thin-wall parts in bending dominate deformation, for instance, PCB and thin-wall covers. Thick shell is still useful in phone modeling. It's suitable for the bending parts attached by another part on surface, for instance LCD glasses with polarizer attachment. Thick shell has good bending performance, and easy to attach another part at its face without the problem of existing a gap to middle plane of a thin shell. Beam element plays important role for simplified solder-joint modeling, which is the only choice to include the tiny structure into a system level model. And the representation is proven effective and enough accurate. 1D rigid link is necessary in parts attachment and model assembly. It's a favor method to simplify screw, clip and snap-in attachment with the rigid link. Finally, spring elements are very useful for part connection. It is a reliable alternative of contact in some cases, and replacements of rigid link to a rigid body connection. With pre-defined extension/compression, spring element can supply a pre-force to structures.

The performance of different element types in LS-DYNA library is very important for element selection in modeling. The performance includes accuracy and efficiency as well as meshing cost. As CPU time is always a concern when a drop simulation is submitted to run, the efficiency exhibited by each element type is important in performance benchmark.

In figure 1, a typical monoblock-type phone model and its element types are demonstrated, where S represents shell element, H means hex/penta element, and T is tetra element.

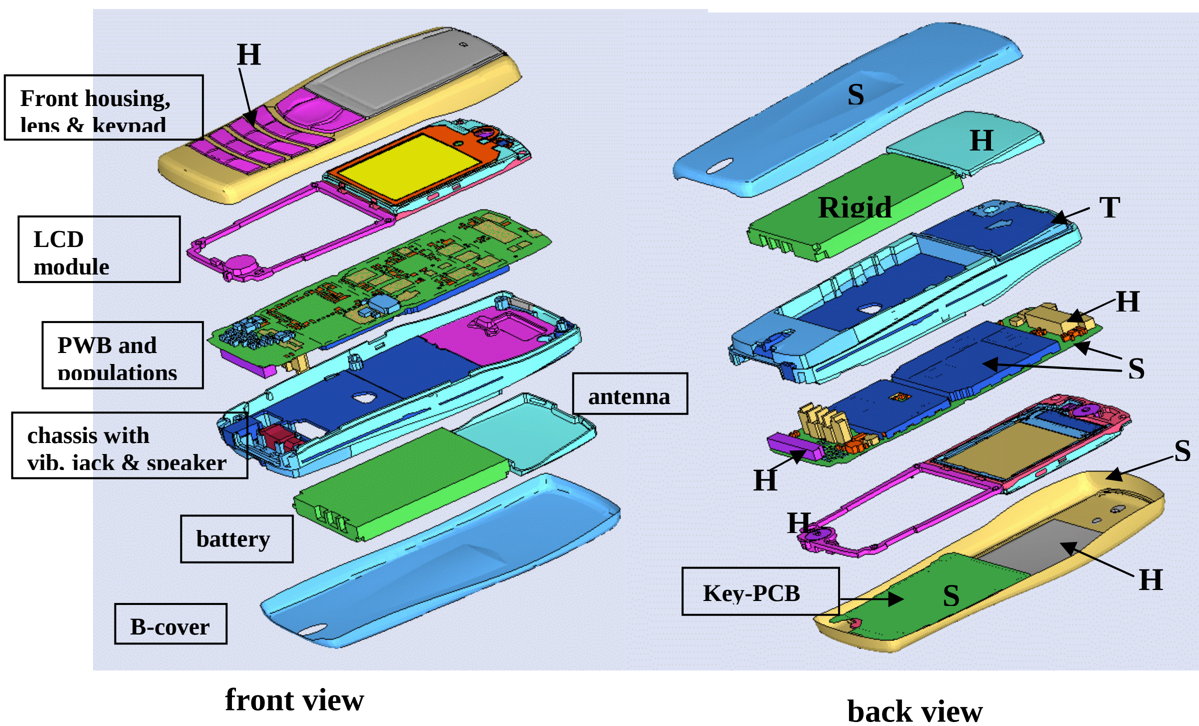



**Figure 1: phone model structure and element types**

Two benchmark cases are accomplished to check element performance. The first example checks performance of bending deformation. A JESD PCB (Printed Circuit Board) attaching a chipset at central location is subjected to base shock – a half-sine inciting acceleration at 1500 g and 0.5 ms duration. The board is fixed at shock table by four screws, shown in Figure 2. Eight element types are used to model PCB. Besides thin and thick shells, three types of hex elements are tested. They are hex element type 1, 2 and 3 in section definition. Type 1 (Hex-1) uses reduced integral formulation. The element has very weak bending stiffness. It must use multiply-layer of elements to reach right bending stiffness. Here four-layer is adopted. Type 2 (Hex-2) and type 3 (Hex-3) are all in full-integral algorithm. Only one-layer of elements is used in the benchmark. Three available tetra element types in LS-DYNA are benchmarked here. They are tetra type 4

(Tetra-4, the second order, one point integral element), type 10 (Tetra-10, the first order, one point integral) and type 16 (Tetra-16, 10-node in second order tetra element). The results are quoted in Figure 3 and Figure 4. Figure 3 gives time-history of PCB deflection at central position, and Figure 4 shows the first principal strain at PCB center. The maximum deflection, strain and CPU time are listed in Table 1.

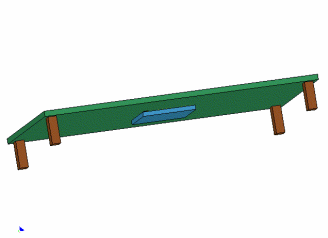


**Figure 2: JESD drop simulation**

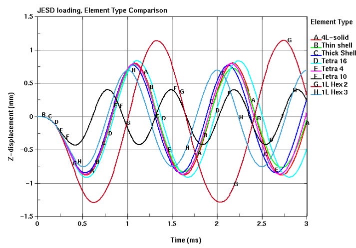



**Figure 3: deflection**

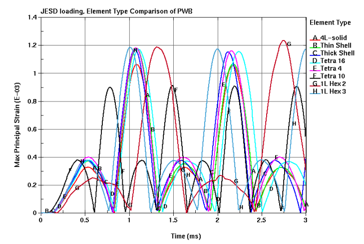



**Figure 4: principal strain**

From the results, we can obtain the conclusion that except tetra 10 and Hex-2, the other element types can be used to simulate bending deformation without significant error. Checking deformation mode, tetra 10 exhibits too stiff, Hex-2 in full-integral formulation looks too soft, but the similar element type – Hex-3 performs much better. In consideration of CPU cost, thin shell runs most fast. Thick shell needs double CPU time, and four-layer Hex-1 needs triple time. And Hex-3 spends over five times of running time. In tetra element side, in consideration of element number correction in comparison with hex and shell elements, Tetra-4 spend double CPU time of thin shell, but only 42% of Tetra-16 running time.

**Table 1: Element Type Comparison**

| Element type | 4-L Hex-1 | Thin shell | Thick shell | Tetra-16 | Tetra-4 | Tetra-10 | 1L-Hex 2. | 1L-Hex3 |
|---|---|---|---|---|---|---|---|---|
| Element # of PWB | **4x10,965** | **10,965** | **10,965** | **15004** | **15004** | **15004** | **10,965** | **10,965** |
| Max deflection (-) mm | **0.876** | **0.865** | **0.841** | **0.911** | **0.861** | **0.426** | **1.288** | **0.7503** |
| Max deflection (+) mm | **0.807** | **0.787** | **0.775** | **0.840** | **0.800** | **0.408** | **1.146** | **0.6974** |
| Max strain (x0.001) | **1.065** | **1.180** | **1.170** | **1.174** | **1.183** | **0.916** | **1.236** | **1.186** |
| CPU time (min) | **35** | **11** | **25** | **84** | **35** | **9** | **22** | **61** |

The second benchmark is a cylinder compression case to check solid element performance in tension/compression deformation. The cylinder is supposed made of polycarbonate material, E=2.5 GPa, in 10 mm diameter and 10 mm height, see Figure 5. The bottom end of the cylinder is fixed, and top end is subjected to a 6.4 MPa uniform pressure. The benchmark is for Hex-1, Tetra-4, Tetra-10 and Tetra-16 elements. Figure 6 shows axial displacement of the cylinder. The result indicates Hex-1 model is very close to exact solution. Tetra-16 exhibits stiffer. Tetra-4 and Tetra-10 have similar behavior, more stiff than Tetra-16.

The conclusion in combination of above results comes that 1) thin shell is priority for thin-wall parts in bending dominate deformation; 2) multiply layer brick element is preferred for the parts in 3D complex stress state; 3) tetra element is used in the parts, with complex geometry and in 3D deformation mode. At current point, Tetra-4 shows a little bit better behavior than Tetra-16 in bending case, but a little bit worse in compression. Considering running speed, Tetra-4 saves half CPU time of Tetra-16 spends.

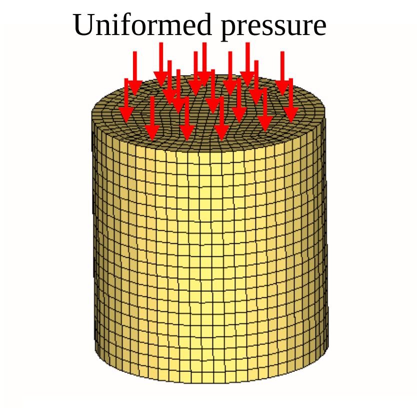



**Figure 5: cylinder under pressure**

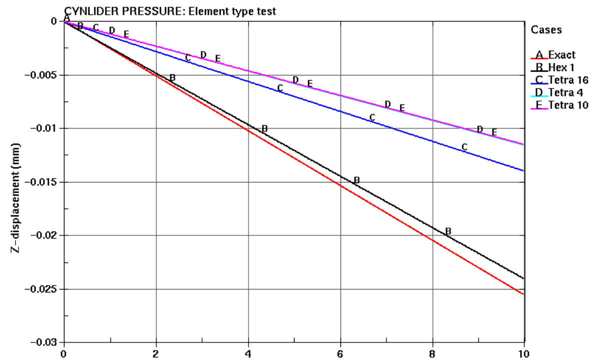



**Figure 6: displacement**

In the following paragraphs, modeling strategy for major parts of cellular phone is described.

## Keymat modeling

Keymat is a major part in cellular phone. Usually it consists of keymat, light-guide and a piece of PCB. The part assembly itself has no drop performance concern. Its role played in drop simulation is only 'cushion' effect in front drop. The key mat is made of silicone. The soft material will reduce impact to interior in front drop. Based on the consideration, two points are impotent in keymat modeling. At the first, it's not worth to model detailed features in the parts. Reasonable simplification to keep its compression characteristics is enough in drop simulation. The modeling can save a lot of modeling and computer running time. Secondly, the material property of silicone is important for its right 'cushion' effect. In practice, Fu-Chang rate-dependant material model is the best candidate for the silicone material. For numerical stability and efficiency, brick element is strongly recommended to model the part, rather than tetra elements. Figure 7 gives modeling examples. The tetra element model on the left hand has 103.7K 10-node tetra element, vs. a simplified version of 3,668 brick elements at the middle. Another hex element model in more fine meshing on the right hand consists of 18.6K Hex-1 elements. How much difference will be introduced when the detailed features are neglected, and element number reduces 80%, even more? Let's check following benchmark for keymat simplification in Figure 8. A contact pressure is subjected on single key. The first model is detailed meshed, with 31,324 10-node tetra elements, and the second model has only 139 Hex-1 elements – tremendously simplified, detailed feature neglected. Figure 9 displays difference of their displacement and stiffness. The difference is less than 10%. The stiffness of the key mainly comes from central plunger block. The error is expected to further reduce if the section of the plunger block is modeled more close to the real key. The CPU time to run the simplified model is only 15 seconds, vs. the full model running time of 5 hours and 17 minutes. The simplification works, and worth to do.

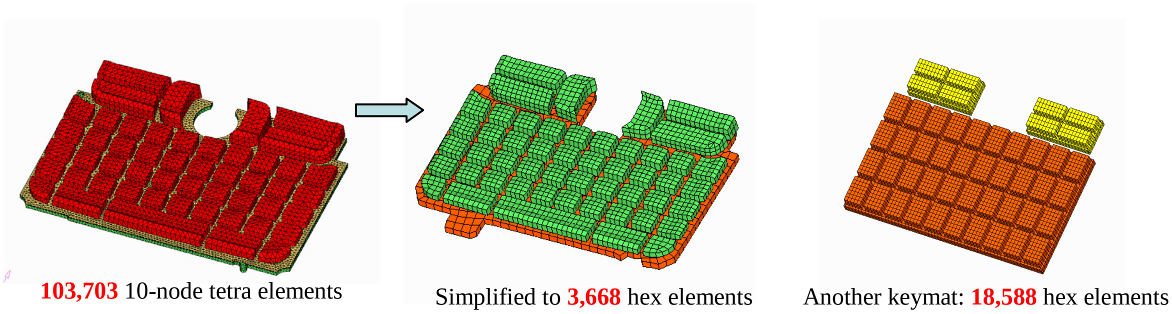



**Figure 7: Keymat modeling comparison**

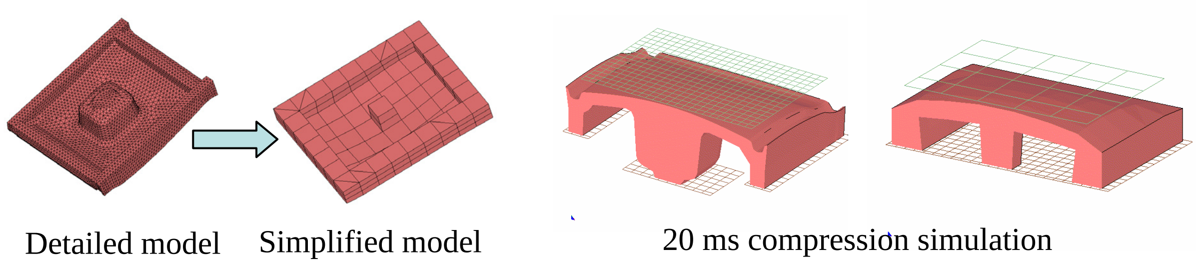



**Figure 8: Keymat simplification**

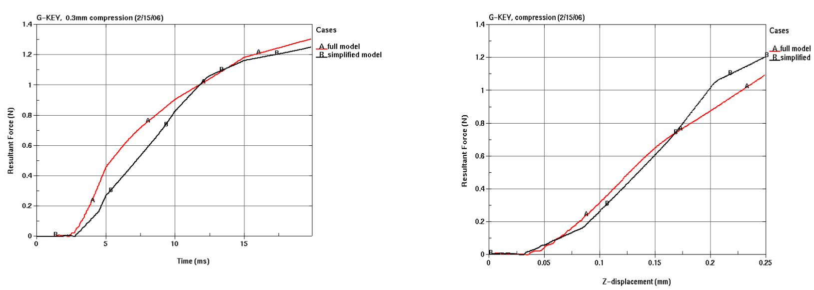



**Figure 9: comparison of displacement (the left) and stiffness (the right)**

## LCD module modeling

LCD (Liquid Crystal Display) module is one of the most important parts to check drop induced failure. The first issue is glass breaking/cracking. Figure 10 shows a model of LCD parts. As importance of LCD module in drop simulation, mesh quality must be strictly controlled for right simulation results and failure prediction. The elements in the example are all shell, Hex-1 and Hex-3. Tetra elements are not recommended in the LCD modeling for the reasons: 1) Some of LCD parts get out-of-plane deformation as major modes, likes glasses, light-guide, back sheet and front metal shield. The parts should be modeled with thin or thick shell. Frame, holder and gasket can be modeled with hex/penta elements, no particular difficulty. Those element types have better performance than tetra elements, in another word, more stable and accurate in LS-DYNA analysis. 2) Don't want any additional mass at LCD module to change its dynamic characteristics and simulation result, when mass-scaling is adopted in whole phone model. That means the element size of LCD module should satisfy time-step requirement. Particularly, there should be no ill-shaped elements, which are the major source to induce mass-scaling on themselves. Using auto-meshing to create tetra elements often leads to the ill-shaped elements due to loss of meshing quality control.

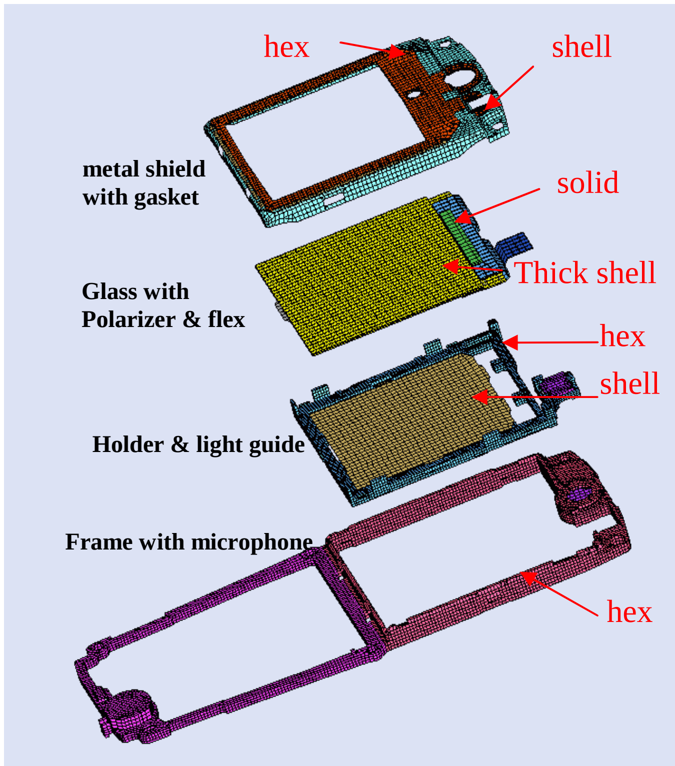



**Figure 10: modeling of LCD parts**

In a LCD model, some very thin sheets, thinner than 0.1 mm, like adhesive, tape and film, are negligible to succinct modeling. Flex is negligible either. The flex modeling is only to fill the space for precise contact among the surrounding parts. Polarizer, about 0.2 – 0.3 mm thickness, is adhered at surface of LCD glasses. Thick shell element is preferred to model the glasses, rather than thin shell elements. The thick shell element is defined in 3D configuration as solid element. The polarizer can be directly attached onto glasses with common nodes. Otherwise, gap and attachment must be specially treated when use thin shell element.

Another point in LCD modeling is adoption of one-piece or two-piece modeling of glasses. The real structure of LCD glasses is shown in Figure 11. Two pieces of glasses are bound in peripheral zone, leaving 0.01 mm space between the two pieces of glasses, where small balls are distributed to keep the gap. Liquid crystal is filled in the gap. In point of view of structural mechanics, a question is the deformation of the LCD glasses like one-piece plate, or separate two-piece plate? The different deformation modes are shown in Figure 12. To check difference of their dynamic response, a jig drop case is simulated. LCD glasses are clamped at a jig in Figure 13. Six different LCD glasses models are tested to check their deflection and stress. Case 1 and 3 are one-piece models, thin shell and 4-layer hex elements, respectively. Case 2 and 4 are two-piece thin shell and thick shell models, respectively. Case 5 and 6 are two-piece 4-layer and

8-layer hex element models. In two-piece models, spotweld is used to connect the glasses nodes in two rows at edges.

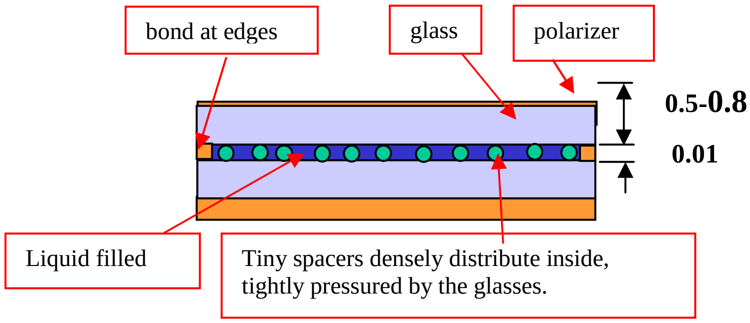



**Figure 11: LCD structure**

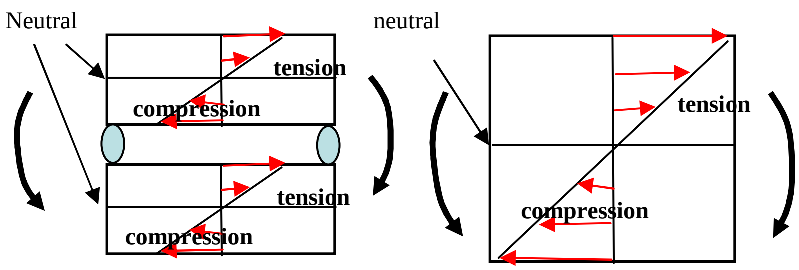



**Figure 12: bending modes in one-/two-piece models**

Figure 13 demonstrates the time-history curve of deflection and surface strain at central elements. The results can be divided into two groups – one-piece group and two-piece group. In the same group the curves are quite close with each other, no matter what element type used. In comparison of the two groups, one-piece model has a half cycle time and a half deflection of two-piece model. With the half deflection in one-piece model, its maximum strain at surface has the similar value with two-piece model, as its double thickness in bending mode.

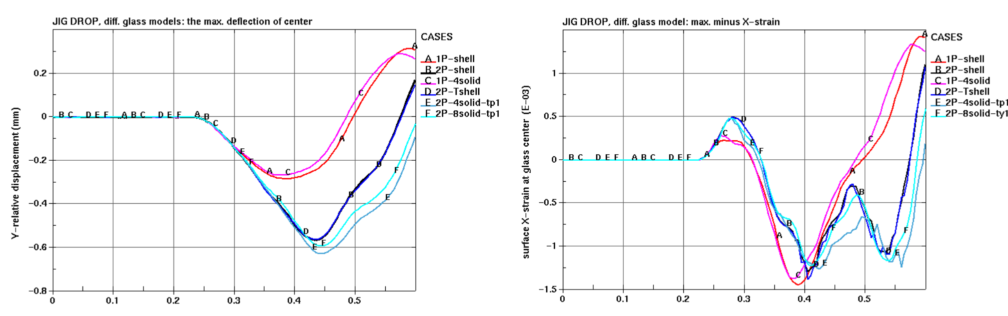



**Figure 13: deflection and strain of LCD glasses**

No available test data of deflection to verify which model is close to the real situation. If it's only strain inspection, the two kinds of modeling have no significant difference. But, in phone drop simulation, double deflection in two-piece model would cause heavier impact to neighbor parts. The two-piece of thick shell model, with peripheral rigid link, is preferred in LCD modeling, as

Figure 14. The two rows spotweld link will provide rotation restriction as well as common translation.

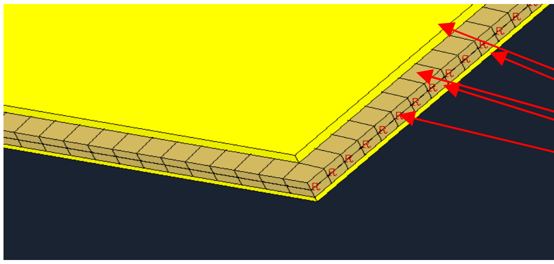



**Figure 14: LCD glasses modeling**

## PCB and its population modeling

PCB drop performance is another emphasis in drop simulation. Solder joints of BGA (Ball Grid Array) connects major CSP (Chip Set Package) to PCB. Reliability of BGA is the most important checking point of phone drop performance. Besides BGA, board-to-board connector and some of other component connection still need the failure check. In another hand, PCB and its populations is a very complex structure. A complete PCB can contain 100K surfaces defined in its CAD data. How to quickly create PCB model, and keep their intrinsic, structural characteristics is a challenging task for FE analysts.

To right prediction of BGA failure risk, we want to setup PCB model as precise as possible. Solder joint behavior is very locally determined. It requires not only right mass distribution and global stiffness of PC board, but also right local mass and stiffness which varies by attached components and metal shields. Simplification is indispensable when facing the huge CAD data for PCB description. The key point is what is negligible and what details must be kept in modeling.

**Beam representation of solder joint:** As a checkpoint of drop simulation, solder joint must be modeled. A solder ball is a cylinder in 0.3-0.6 mm height, and 0.2-0.4 mm diameter. The modeling by solid elements in real geometry will cause too small time step. A whole phone drop simulation can't afford to the CPU cost in the modeling. The solution is use of equivalent beam element to represent a solder joint. The following benchmark in Figure 15 is to verify correctness of this representative**.** On one side, it is a cylindered solder joint modeled with 1224 hex elements. On another side, it's a one beam element in the same geometrical definition as the solid element model. They are subjected to the same boundary condition – fixed at bottom end, and free rotation or rotation restricted at top end. A given lateral displacement of 0.1 mm is subjected to the top end. The shear force is compared in Figure 15 and Table 2. In another compression comparison, 0.1 mm compression is given, and axial force is listed in Table 2. From the benchmark, the beam representative is acceptable in system-level (whole phone) drop simulation.

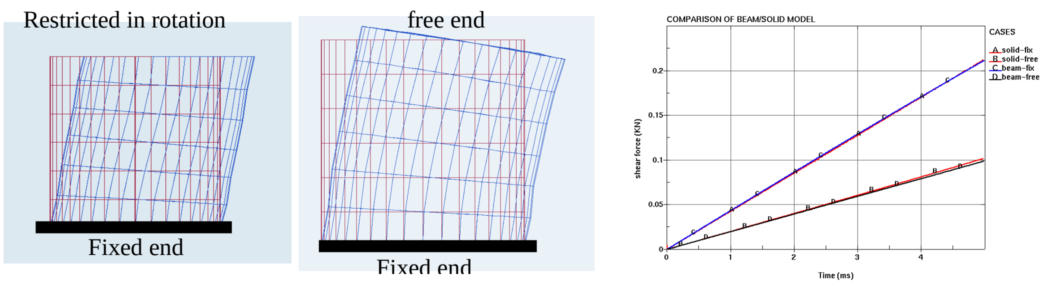



**Figure 15: solder joint model and shear force comparison to one-beam model**

**Table 2: correlation of beam model with solid element model**

| Force (kN) | Solid model | One-beam model | Error (%) |
|---|---|---|---|
| Shear, rotation restricted | 0.2128 | 0.2122 | -0.3 |
| Shear, free rotation | 0.1022 | 0.0992 | -2.9 |
| Compression | 1.0219 | 0.9817 | -3.9 |

Figure 16 shows the modeling. Mesh should match pitch of solder joints, usually 0.5 mm or 0.8 mm. One beam represents one solder ball. Cracking or breaking always happens at corner solder joints, which stress/deformation are output to check.

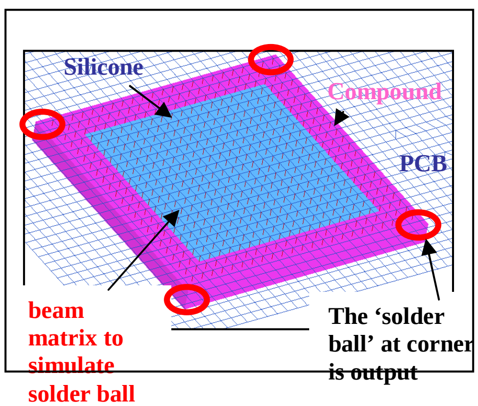



**Figure 16: solder-beam model**

There are two methods to check solder ball failure based on the whole phone drop information. The simple one is to check stress/strain directly from the system-level simulation output. LS-DYNA can output beam stress. But keep in mind that the stress is defined at Gaussian point, different form the stress calculated from resultant force and resultant moment, which could be the maximum stress at surface. Another method is to use global-to-local analysis to obtain details of solder joint deformation. It requests a detailed solder ball model. Dynamic analysis in the local model, still with LS-DYNA, is recommended. The time history data of resultant force/moment or relative displacement of the beam ends will input to local model. An example of local model, including 19K hex elements, is shown in Figure 17. In the demonstration, only 0.03 mm lateral displacement is considered. Elastic-plastic material property is used. Plastic strain under the load is shown in the figure.

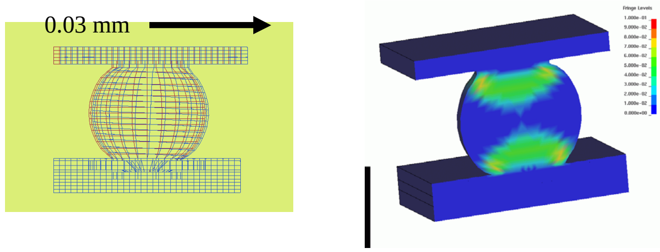



**Figure 17: local model of solder joint and plastic strain contour**

**PCB and CSP modeling:** After decision of beam element to represent solder joints, the PCB and CSP modeling must match this strategy. PCB can be modeled with multilayer solid element, thick shell or thin shell. But, only thin shell is the right choice to connect solder beams for its rotation degree-of-freedom. Only in thin shell model, PCB element can provide right rotating stiffness to the joint node with solder beam. The node of solid element and thick shell element has no rotation degree-of-freedom. The connected node to solder beam is lack of right rotation restriction. The model will cause reduced stress in solder joint. A benchmark provides the verification. A JESD drop is simulated as Figure 2. Stress of solder beam at corner is output to check PCB modeling that is formed by thin shell, thick shell, tetra and 4-layer hex elements. The axial stress is exhibited in Figure 18. The stresses in all other models than thin shell have the similar value, but much lower than the stress in thin shell model. A correction is suggested to adhere a very thin shell (0.01mm) on the surface of solid elements to provide the rotation degree-of-freedom on their nodes, and no influence to PCB stiffness. The measure can't change the fact because rotation stiffness from the shell contribution is too weak to right rotation restriction. When using thin shell, it will leave a gap between PCB middle plane and solder beams. The gap can be filled by spotweld element in LS-DYNA.

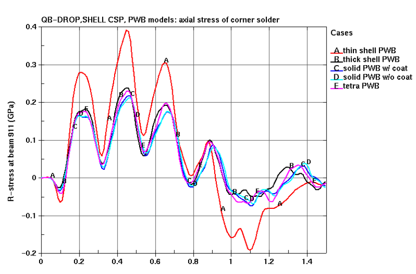



**Figure 18: benchmark for LCB modeling**

Chipset modeling faces the same problem. It must be thin shell to connect solder beam to provide right rotation stiffness. Three modeling alternatives are demonstrated in Figure 19. The first scheme is modeling pad/substrate layer of chipset by thin shell, which directly connects to solder joint. Then, the rest part of chipset is modeled by thick shell or solid element attached at the shell elements. The other two methods simplify the whole chipset as a shell, or composite shell. One is to locate the shell at bottom position of the chipset, another one is to put it at the middle position. The benchmark in Figure 19 indicates quite different stress level at solder beam of the three models. No available test evidence, up to now, can prove which one is more close to real situation. The first modeling is chosen by author, just for it gives the stress in middle level between the other two models.

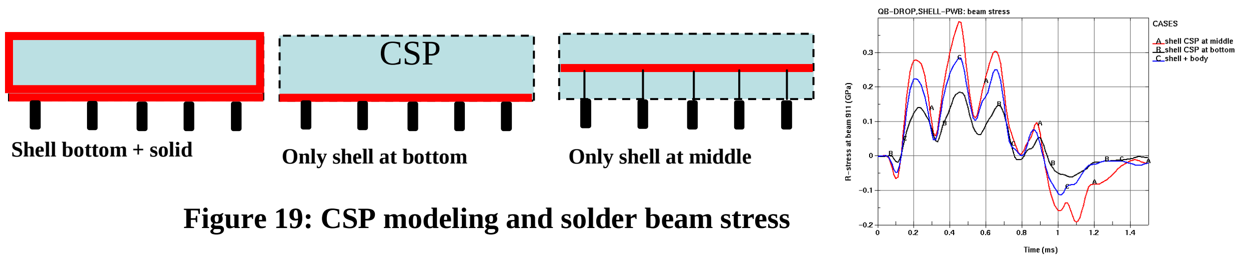
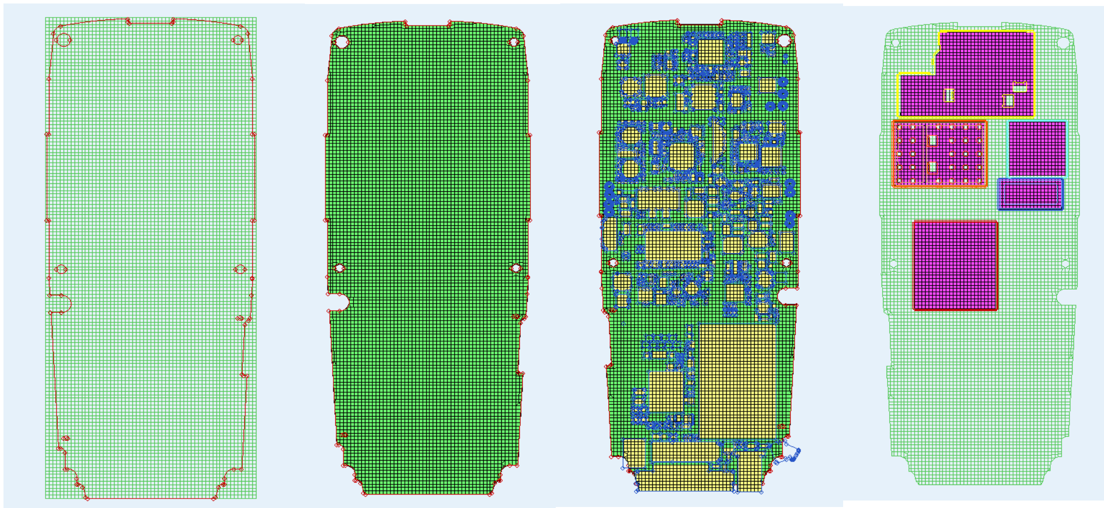



**Figure 19: CSP modeling and solder beam stress**

**Chess-board method:** Facing dozens of thousands surfaces in PCB design data, it's still a big challenge to quickly create PCB assembly mesh with reasonable simplification and without loss mechanical intrinsic. Here, a 'chess-board' method is recommended as a solution. The basic idea of its solution is to give up exact following original geometry in CAD data, but create mesh based on a chess-board in reference of original geometry. The method has four steps:

1) Decide element size firstly. Usually using BGA pitch of major CSP, for instance 0.5 mm. Then put a chess-board with 0.5 mm by 0.5 mm regular mesh on the PCB geometry.
2) Trim the board edge to match PCB geometry. A few of elements will be cut off, or changed to triangle. Some nodes need to move a little to match PCB edge.
3) Create BGA and components at PCB. For general components, simply drag shell element at the chess-board to form 3D solid element, in reference of component location in 3D mentor picture of CAD data. Don't care exact shape and location of the components, but follow the chess-board mesh. It will introduce error within half of chess-board mesh in location and shape. The drag starts from big components to smaller, until the rest components are considered too tiny to PCB mechanical behavior. The same way for the chipsets with BGA, but separate CSP bottom from PCB and lift it to leave space for solder beam modeling. For precision of BGA modeling, the chess-board mesh under BGA should be locally adjusted a little bit to match its exact location.
4) Drag and create metal shields. At metal shield location, copy chess-board elements and move them to shield top. Then, create side wall elements by one-to-one mapping from chess-board to shield top.

In the chess-board method, the all components are created based on the chess-board mesh, not original geometry. The only correction of chess-board mesh is PCB edge, and BGA location, which is expect to keep precision. The other components are created in approximate shape and location. But the approximation is good enough for the PCB assembly model. Figure 20 gives pictures to explain the method. An overview of PCB model created in chess-board method is demonstrated in Figure 21, with its CAD data comparison. With the method, a PCB assembly model can be finished in 1 – 2 days, by use of right tool and skill.



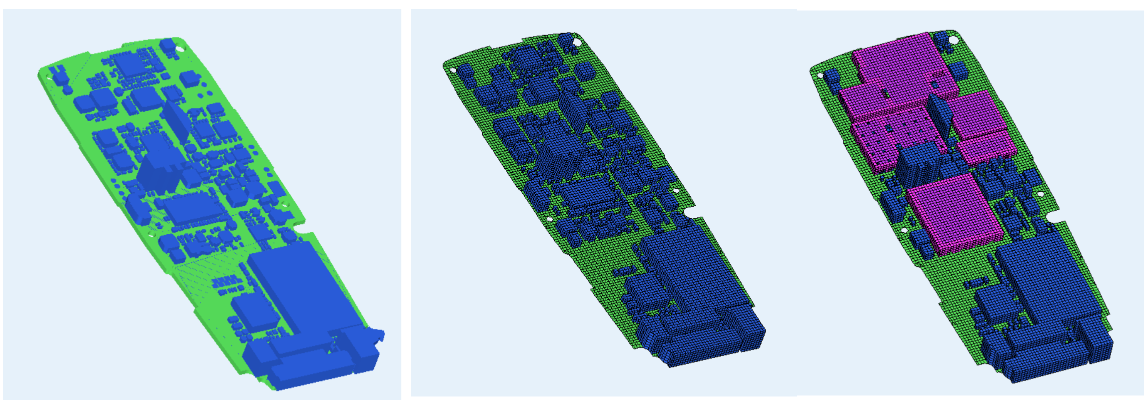
Step 1: Create net covering PCB

Step 2: trim net edge matching PCB

Step 3: drag to form components

Step 4: drag to form metal shields

**Figure 20: chess-board method to create PCB model**



**CAD data**

**FE model**

**Figure 21: Comparison of CAD picture and FEA model**

## Advanced modeling and simulation

A whole phone drop simulation usually runs 1.5 -2 ms of simulation time. In the model, leave about 1 mm gap between dropping phone and target, and define initial velocity to the phone. The dynamic response will attenuate soon after impact to the target. Most of information of maximum deformation, stress, strain, and other parameters can be picked up within 1 ms simulation after impact. A typical phone model, with 260K elements, including 25% 10-node tetra elements, needs around 3 hours of CPU time to finish its 1.5 ms drop simulation at dual-CPU Opteron machine. The fast computer and upgraded LS-DYNA provide bright prospect for wider application of drop simulation. An extension is to simulate multiply impact in phone drop. The multiply impact phenomenon has been noted for many years. That is when a phone drops in certain angle, the phone will rotate after the first impact, contact target at one end, then at another end, so called ‘clattering’. The following impact is doubted to get heavier impact, and have more risk for phone damage. The phenomenon can be simulated now.

The following example is a monoblock-type phone drop at 30 degree obliquity, in 24 ms simulation time. It spends about 9 hours of CPU time at dual-CPU Opteron machine. In the simulation, a special feature of LS-DYNA is used - conversion between deformable and rigid-body modes. A rigid-body model can change to deformable when an appointed contact force is detected. After 2.5 ms duration, the deformation model changes back to rigid-body model without the contact force action. The conversion keeps structural deformation at impact, and save computer time with rigid-body mode when the phone is flying in the air. In the 24 ms simulation, 7.5 ms is deformable mode at its three times impact. The rest simulation time, the phone keeps rigid-body status. Figure 22 give the CPU time distribution.

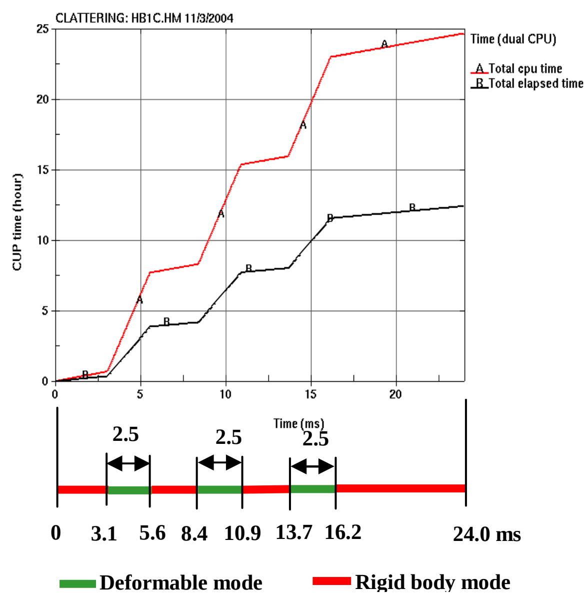



**Figure 22: CPU time in convertible model**

The simulation clearly shows ‘one-two-three’ clattering in its moving mode. And the second impact gets the highest contact force, due to drop velocity plus rotation moving. Figure 23 shows the model and contact forces compared among convertible model, rigid-body model and deformable-body model. Rigid-body model makes big difference of contact force from others, and moves at ‘one-two’ clattering mode. The behavior between convertible model and deformable model agrees with each other very well.

Besides exploration of the clattering phenomenon, the simulation can help us to improve modeling. Some modeling defects might not cause divergence in a short time simulation, but it will emerge and damage numerical analysis in a long time simulation. In another word, the long time simulation requires more robust and reliable modeling.

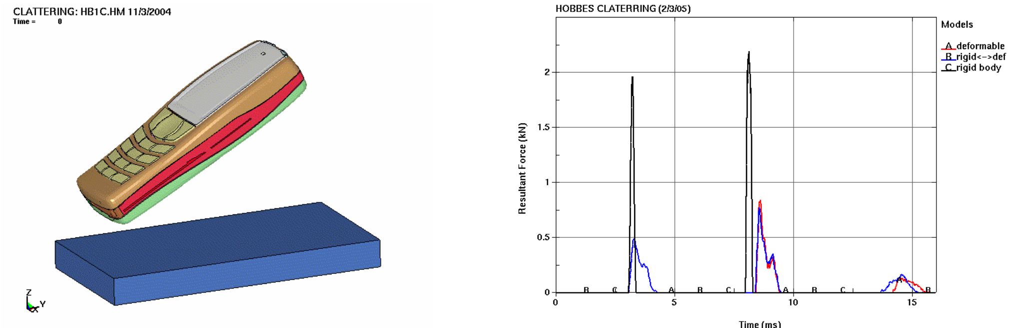



**Figure 23: Clattering simulation and contact force comparison**

## References


1. Foundations of Solid Mechanics, Y. C. Fung; Prentice-Hall, Inc. 1965;
2. The Finite Element Method (The 3rd edition), O.C. Zienkiewicz, McGraw-Hill, 1977
3. Bathe, KJ and Wilson E.L. (1976). "Numerical Methods in Finite Element Analysis" Prentice-Hall, Inc.
4. Ready, J.N. and Rasmussen M.L. (1982) "Advanced Engineering Analysis". Wiley-Interscience Publication
5. Theoretical Manual for DYNA3D, J. O. Hallquist, LLNL, 1985;
6. J. Wu, et al: Drop/Impact Simulation and Test Validation in Motorola, 5th LS-DYNA User Conference, 1998
7. J. Wu, et al: Drop/Impact Simulation and Test Validation of Telecommunication Products, ITHERM98, 1998
8. J. Wu, et al: Global-and -Local Analysis for Small Components in Drop Simulation, 6th LS-DYNA User conference, 2000