

Drop Test Simulation of a Cordless Mouse

Summary

Portable, hand-held electronic devices have become commonplace due to their small size and light weight. It is inevitable that such devices will occasionally experience the shock loading associated with being dropped. Accounting for this loading scenario in the design process, both analytically and experimentally, allows for the development of more durable products. The ability to simulate drop-type loading reliably reduces the dependency on experimental testing. Abaqus/Explicit has been used extensively to examine the behavior of electronic devices experiencing mechanical shock loading.

Background

It is not uncommon for personal electronic devices to be subjected to mechanical shock loading, particularly that associated with being dropped from the hand or falling from a table. The ability to withstand such loading is critical to the design of a successful product.

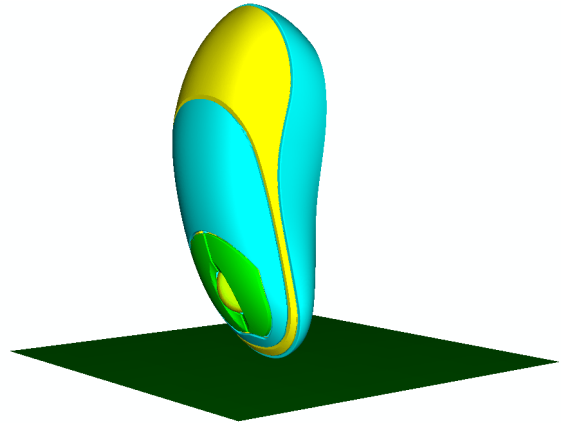
Assessing the response of a prototype electronic device to shock loading typically involves a combination of laboratory testing and analytical simulation. Simulations have clear advantages over physical tests in that they offer repeatable results, provide information (stress, strain, acceleration, etc.) at any point in the model at any time during the analysis, can be performed at less cost, and can be included at any stage in the design process.

The analysis considered in this technology brief is a simulation of a 1-meter drop test of a cordless, optical computer mouse. The potential malfunction of electronic components is the primary focus of the analysis.

Finite Element Analysis Approach

The mouse model consists of housing components, a circuit board with electronic components, a wheel, and two buttons. The case components, buttons, and wheel are meshed with modified 10-node tetrahedral elements. These elements provide a robust solution during large-deformation and contact analyses and exhibit minimal volumetric and shear locking tendencies. They are also advantageous in that they can be used with automatic tetrahedral mesh generators, thus easing the task of meshing the complex molded geometry typical in electronic components.

The circuit board and electronic components are meshed with hexahedral elements. The electronic components are modeled as rigid bodies, connected to the circuit



Key Abaqus Features and Benefits

- Robust explicit solution method for efficient simulation of short duration, highly dynamic, nonlinear events.
- General (“automatic”) contact algorithm for simplified contact definitions.
- Modified 10-node tetrahedral elements that facilitate the meshing of complex geometry and provide accurate contact results.
- Variable mass scaling to increase computational efficiency.
- Mesh-independent fasteners that use connector elements as the basis for point-to-point connections, allowing for complex constitutive and mechanical behavior, including failure.

board with surface-based tie constraints. The tie-type constraint rigidly bonds surfaces together, and these bonds cannot break during the analysis. The advantage of using tie-type constraints is that the meshes to be joined are not required to match, allowing the electronic components to have relatively coarse meshes. The components making up the mouse are shown in Figures 1–3.

All the deformable parts use linear elastic material descriptions. This is typical for analyses of this type, where the strain in the components is expected to be small. An actual computer mouse is held together with screws, snap-fit clips, or other connection methods. While complete, detailed modeling of such features is possible, the present analysis focuses on the loading experienced by the electronic components.

Therefore, all mechanical joints between components are approximated with mesh-independent fasteners or surface-based tie constraints. Tie constraints are used to join the circuit board to the base housing and the buttons to the upper housing.

Mesh-independent fasteners allow two or more bodies to be joined at arbitrary points; coincident nodes are not required at the location of the connection. Further, the actual connection can be modeled with connector elements. This capability allows complex behavior such as material nonlinearity, friction, and failure to be included in a point-to-point connection.

The base, middle, and upper housings of the mouse are joined using connector-based mesh-independent fasteners. Connectors of Cartesian and Cardan type are combined to capture beam-like behavior. The location of the fastener connectors is shown in Figure 4.

Mesh independence is achieved by distributing the attachment to several nodes on the surfaces being joined. The nodes are automatically selected and coupled to the attachment points. The attachment points are then used to define the connector element. As an example, Figure 5 shows the fastener (viewed from inside) that joins the base and middle housings at the front of the mouse. The connector element and the nodes to which the element is coupled are highlighted in red.

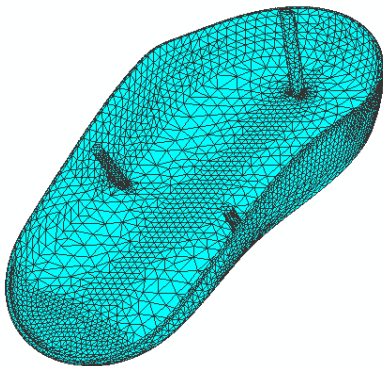


Figure 1: Base housing

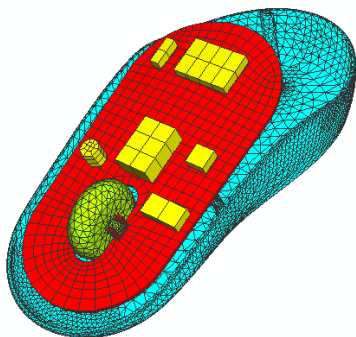


Figure 2: Circuit board, electronics, and wheel

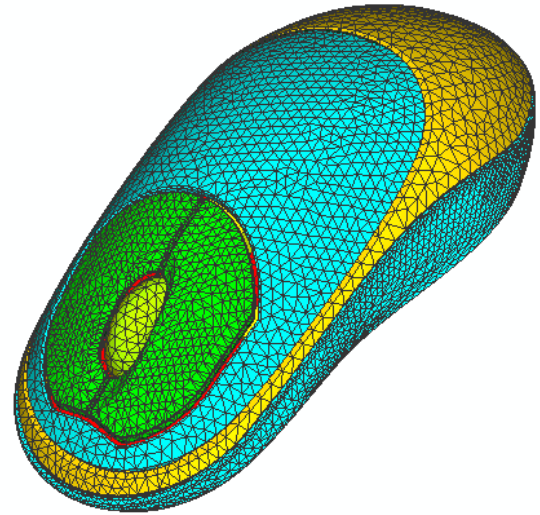


Figure 3: Complete model

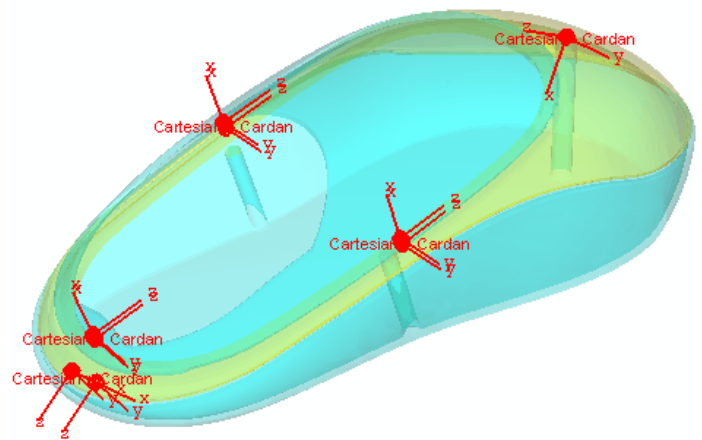


Figure 4: Location of mesh-independent fasteners

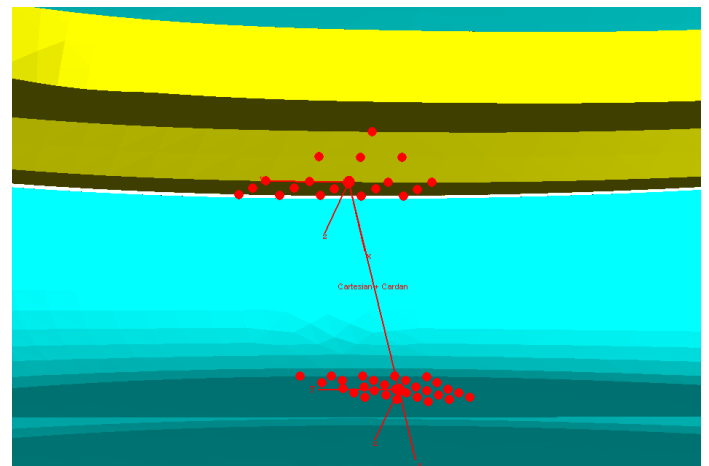


Figure 5: Cutaway view showing forward base-to-middle housing fastener and coupling nodes

While not included in the current analysis, material failure could also be used to model damage of the housings. For electronic devices that have either a screen or a lens, this type of modeling would allow the survivability of these vulnerable parts to be assessed more accurately.

The general (or “automatic”) contact algorithm simplifies the definition of contact in that individual surface pairs do not need to be defined explicitly. Rather, the entire assembly can be included in a contact domain with a single interaction definition. The floor is modeled as rigid, and gravity loading is applied to all the mouse components. Initial velocity conditions consistent with a 1-m free fall are also prescribed at every node.

The analysis simulates a 5-millisecond drop event. To reduce the required runtime for this analysis, variable mass scaling is used. With this option, Abaqus automatically performs mass scaling at discrete time points during the solution so that a user-specified minimum time increment is maintained.

Analysis Results and Discussion

A contour plot of the Mises stress in the case immediately after impact is shown in Figure 6. As expected, stress concentrations are shown at the impact area and also at the base of the lug that supports the circuit board. The stresses in these areas could be investigated in further detail using a submodeling technique in which the solution obtained from a global model is used to drive a smaller, more detailed model of a local region.

A typical fastener might have pieces that break if a relative motion component, force, or moment becomes too large. Abaqus/Explicit allows for the specification of an individual component of relative motion that will break or the specification that all components may break. The component of relative motion on which the failure criterion is based can also be selected.

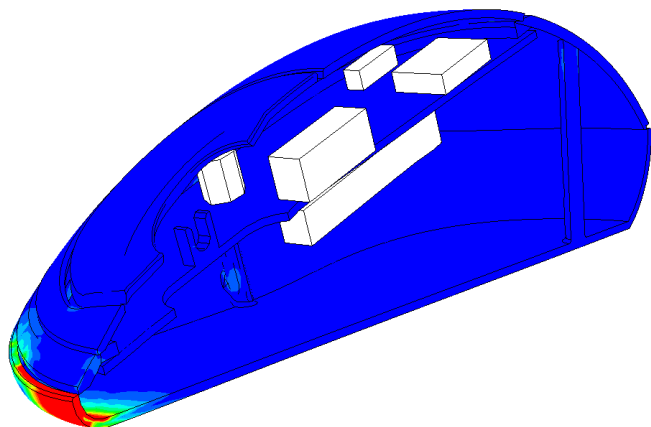


Figure 6: Mises stress in case immediately after impact

For each of the connector elements, failure force in the axial direction was specified. In addition, all components of relative motion were released upon connector failure. The configuration of the model at the end of the analysis is shown in Figure 7. The connectors fail at approximately 0.08 milliseconds, causing the housing components to become completely separated.

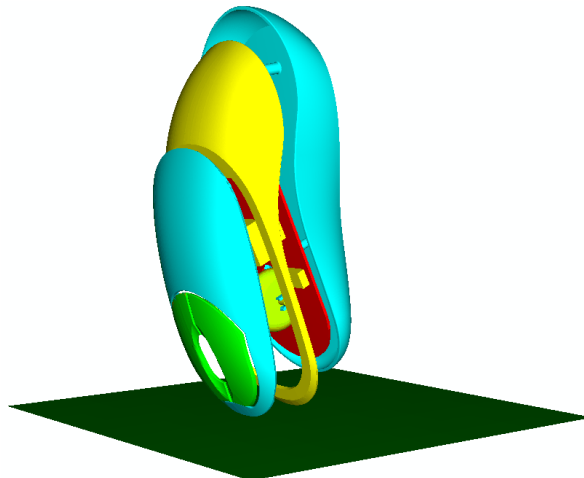


Figure 7: Mouse model at 5 milliseconds, showing separation of case components

The total forces in the forward base-to-middle housing fastener (shown in Figure 5) are plotted in Figure 8. The local X- and Z-components are shown; these correspond to axial and lateral shear forces, respectively. The local Y-component is zero due to symmetry. An additional curve is shown for each component for the case in which the failure of the connector was prevented.

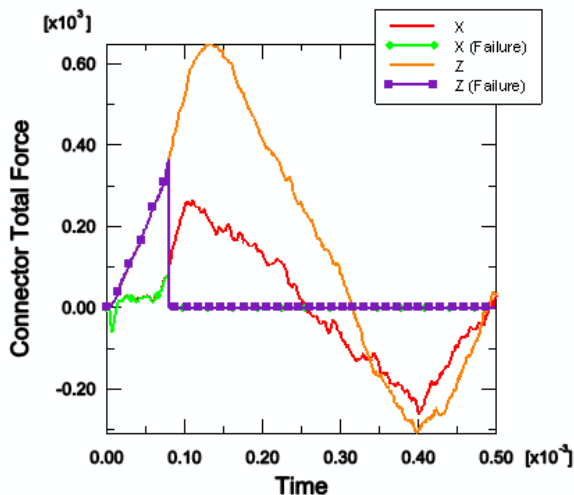


Figure 8: Total forces in forward base-to-middle housing fastener connector

The model's internal and kinetic energy histories are shown in Figure 9. At the beginning of the simulation the mouse is in free fall; consequently, the kinetic energy accounts for the entire energy content. The initial impact deforms the case, transforming the kinetic energy into internal energy. Two additional curves are shown, one for each energy component for the case in which failure of the connector was not included.

The purpose of the analysis is to assess the potential for damage to the electronic components. The acceleration histories of the circuit board components offer direct information about the severity of the loading. The responses are measured at the center of mass of each component; the respective node locations are shown in Figure 10. In Figure 11 the vertical acceleration history of component 3001 is plotted. The results are again replicated for the case in which failure of the connectors is not included.

The raw acceleration results contain noise, so the curves shown in Figure 11 have been filtered. Element, nodal, contact, integrated, and fastener interaction history output can be prefiltered during the solution process before it is written to the output database. In the current analysis a Butterworth filter with a cutoff frequency of 5 kHz was chosen. Excessive accelerations during impact may damage the electronic components, even though they may remain attached to the circuit board. To assess survivability, peak accelerations of the electronic components could be checked against experimentally determined limits.

Other methods for determining survivability include examining detailed stress and strain histories, specifically in the circuit board near the electronic components. Limiting values of these quantities can be used to estimate failure of solder joints.

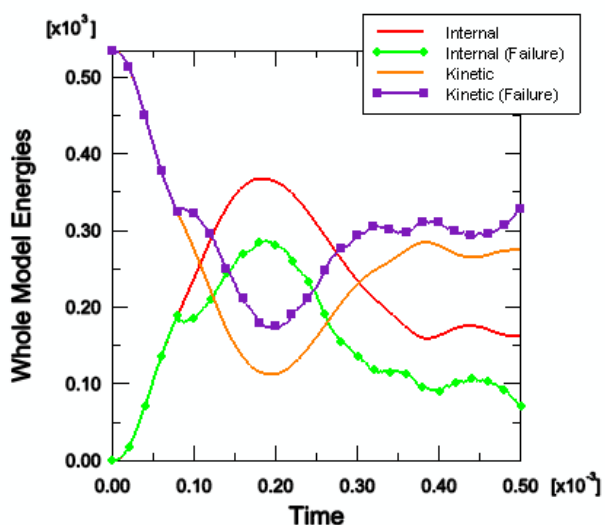


Figure 9: Whole model internal and kinetic energies

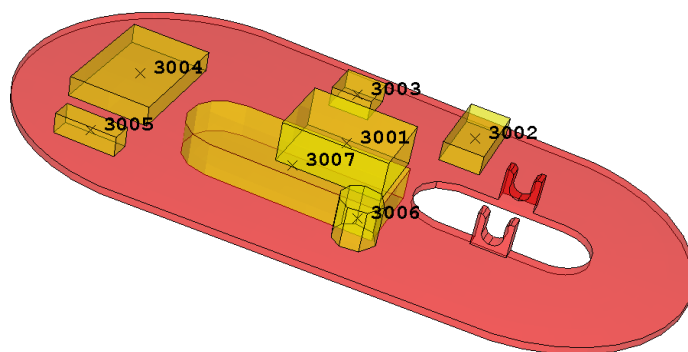


Figure 10: Labels of electronic component centers of mass

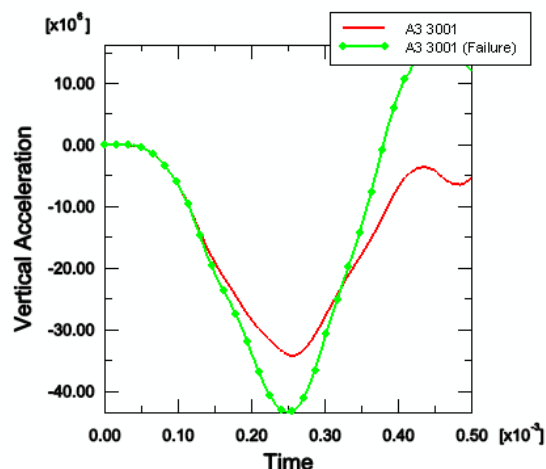


Figure 11: Vertical acceleration of electronic component 3001

Conclusion

Analytical simulation of mechanical shock loading has become an indispensable tool in the design of personal electronics. The advanced features of Abaqus/Explicit allow this loading scenario to be easily included in the design process.

References

- Wang, J., C. Ho, and E. Wong, "Modeling Solder Joint Reliability of BGA Packages Subject to Drop Impact Loading Using Submodeling," Proceedings of the 2002 Abaqus Users' Conference.
- Hirai, M., M. Tanoue, M. Mukai, and Y. Yokono, "Simulation of Notebook Personal Computer Drop Test," Proceedings of the 2001 Abaqus Users' Conference.
- Nagaraj, B., D. Carroll, and T. Diehl, "Ball Drop Simulation on Two-Way Radio Lens Using Abaqus/Explicit," Proceedings of the 1999 Abaqus Users' Conference.

Abaqus References

For additional information on the Abaqus capabilities referred to in this brief, see the following Abaqus 6.12 documentation references:

- Analysis User's Manual
 - "Output to the output database," Section 4.1.3
 - "Explicit dynamic analysis," Section 6.3.3
 - "Mass scaling," Section 11.6.1
 - "Solid (continuum) elements," Section 28.1.1
 - "Mesh tie constraints," Section 34.3.1
 - "Mesh-independent fasteners," Section 34.3.4
 - "Defining contact interactions," Section 35
- Example Problems Manual
 - "Knee bolster impact with general contact," Section 2.1.9
 - "Cask drop with foam impact limiter," Section 2.1.12
 - "Oblique impact of a copper rod," Section 2.1.13

About SIMULIA

SIMULIA is the Dassault Systèmes brand that delivers a scalable portfolio of Realistic Simulation solutions including the Abaqus product suite for Unified Finite Element Analysis, multiphysics solutions for insight into challenging engineering problems, and lifecycle management solutions for managing simulation data, processes, and intellectual property. By building on established technology, respected quality, and superior customer service, SIMULIA makes realistic simulation an integral business practice that improves product performance, reduces physical prototypes, and drives innovation. Headquartered in Providence, RI, USA, with R&D centers in Providence and in Suresnes, France, SIMULIA provides sales, services, and support through a global network of over 30 regional offices and distributors. For more information, visit www.simulia.com

The 3DS logo, SIMULIA, Abaqus and the Abaqus logo are trademarks or registered trademarks of Dassault Systèmes or its subsidiaries, which include ABAQUS, Inc. Other company, product and service names may be trademarks or service marks of others.

Copyright © 2007 Dassault Systèmes

