FINITE ELEMENT ANALYSIS OF AUTOMOBILE CRASH SENSORS FOR AIRBAG SYSTEMS

By

Christopher A. Leger & Claude A. Bryant
Breed Technologies, Inc.
Lakeland, Florida

FINITE ELEMENT ANALYSIS OF AUTOMOBILE CRASH SENSORS FOR AIRBAG SYSTEMS

Christopher A. Leger & Claude A. Bryant
Breed Technologies, Inc.
Lakeland, FL 33807

ABSTRACT

Automobile spring bias crash sensor design time can be significantly reduced by using finite element analysis as a predictive engineering tool. The sensors consist of a ball and springs cased in a plastic housing. Two important factors in the design of crash sensors are the force-displacement response of the sensor and stresses in the sensor springs. In the past, sensors were designed by building and testing prototype hardware until the force-displacement requirements were met. Prototype springs need to be designed well below the elastic limit of the material. Using finite element analysis, sensors can be designed to meet force-displacement requirements with acceptable stress levels. The analysis procedure discussed in this paper has demonstrated the ability to eliminate months of prototyping effort.

MSC/ABAQUS has been used to analyze and design airbag crash sensors. The analysis was geometrically nonlinear due to the large deflections of the springs and the contact between the ball and springs. Bezier 3-D rigid surface elements along with rigid surface interface (IRS) elements were used to model ball-to-spring contact. Slideline elements were used with parallel slideline interface (ISL) elements for spring-to-spring contact. Finite element analysis results for the force-displacement response of the sensor were in excellent agreement with experimental results.

INTRODUCTION

An important component of an automotive airbag system is the crash sensor. Various types of crash sensors are used in airbag systems including mechanical, electro-mechanical, and electronic sensors. An electro-mechanical sensor (see Figure 1) consisting of a ball and two springs cased in a plastic housing is discussed in this paper. When the sensor experiences a severe crash pulse, the ball pushes two springs into contact completing the electric circuit allowing the airbag to fire. The force-displacement response of the two springs is critical in designing the sensor to meet various acceleration input requirements. Stresses in the sensor springs must be kept below the yield strength of the spring material to prevent plastic deformation in the springs. Finite element analysis can be used as a predictive engineering tool to optimize the springs for the desired force-displacement response while keeping stresses in the springs at acceptable levels.

In the past, sensors were designed by building and testing prototype hardware until the force-displacement requirements were met. Using finite element analysis, the number of prototypes built and tested can be significantly reduced, ideally to one, which substantially reduces the time required to design a sensor. The analysis procedure discussed in this paper has demonstrated the ability to eliminate months of prototyping effort.

MSC/ABAQUS [1] has been used to analyze and design airbag crash sensors. The analysis was geometrically nonlinear due to the large deflections of the springs and the contact between the ball and springs. Various contact elements were used in this analysis including rigid surface interface (IRS) elements, Bezier 3-D rigid surface elements, parallel slide line interface (ISL) elements, and slide line elements. The finite element analysis results were in excellent agreement with experimental results for various electro-mechanical sensors studied in this paper.

PROBLEM DEFINITION

The key components of the electro-mechanical sensor analyzed are two thin metallic springs (referred to as spring1 and spring2) which are cantilevered from a rigid plastic housing and a solid metallic ball as shown in Figure 1. The plastic housing contains a hollow tube closed at one end which guides the ball in the desired direction. The ball is held in place by spring1 at the open end of the tube. When the sensor is assembled, spring1 is initially displaced by the ball which creates a preload on spring1. The ball is able to travel in one direction only in this sensor and this direction will be referred to as the x-direction (see the global coordinate system shown in Figure 2) in this paper. Once enough acceleration in the x-direction is applied to overcome the preload on spring1, the ball displaces the spring. As the acceleration applied continues to increase, spring1 is displaced until it is in contact with spring2. Once

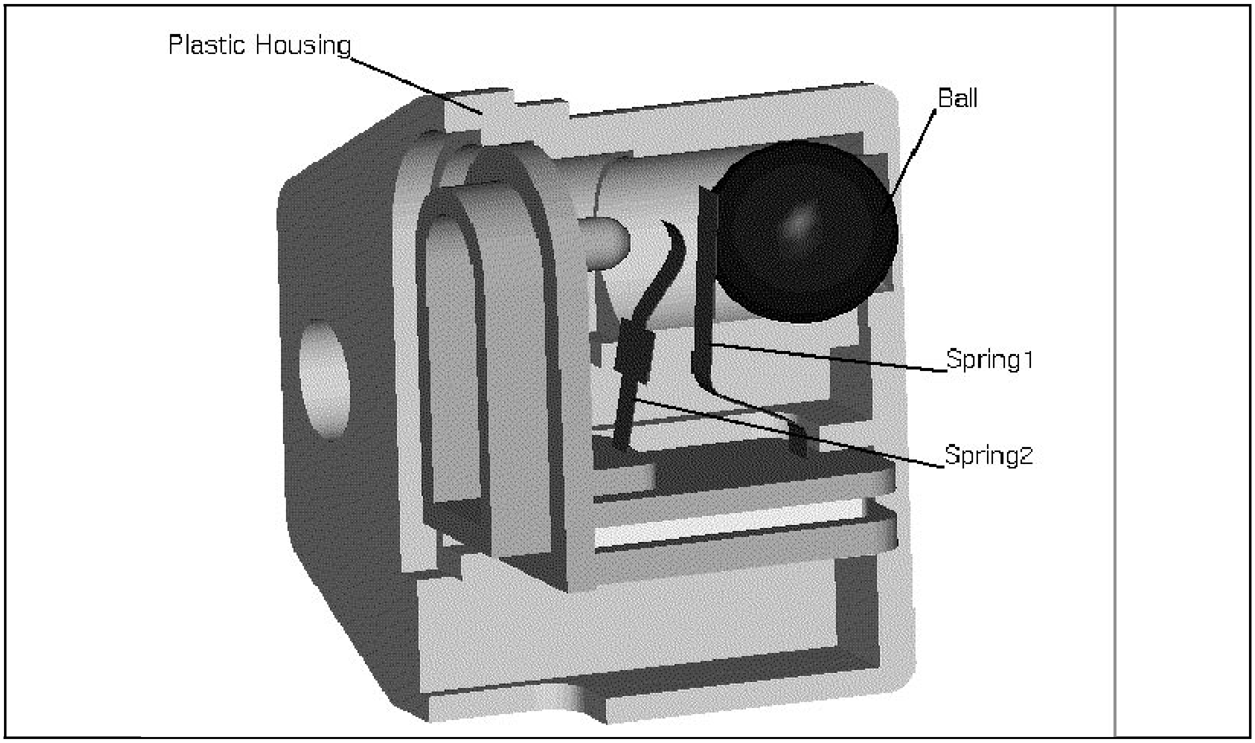

Figure 1. Electro-mechanical automobile crash sensor.

contact is made between spring1 and spring2, an electric circuit is completed allowing the sensor to perform its function within the airbag system.

FINITE ELEMENT ANALYSIS METHODOLOGY

When creating a finite element representation of the sensor, the following simplifications can be made. The two springs can be fully restrained at their bases implying a perfectly rigid plastic housing. This is a good assumption when comparing the flexibility of the thin springs to the stiff plastic housing. The ball can be represented by a rigid surface since it too is very stiff as compared to the springs. Rather than modeling the contact between the plastic housing and the ball, all rotations and translations are fully restrained except for the x-direction on the rigid surface representing the ball. These restraints imply that the housing

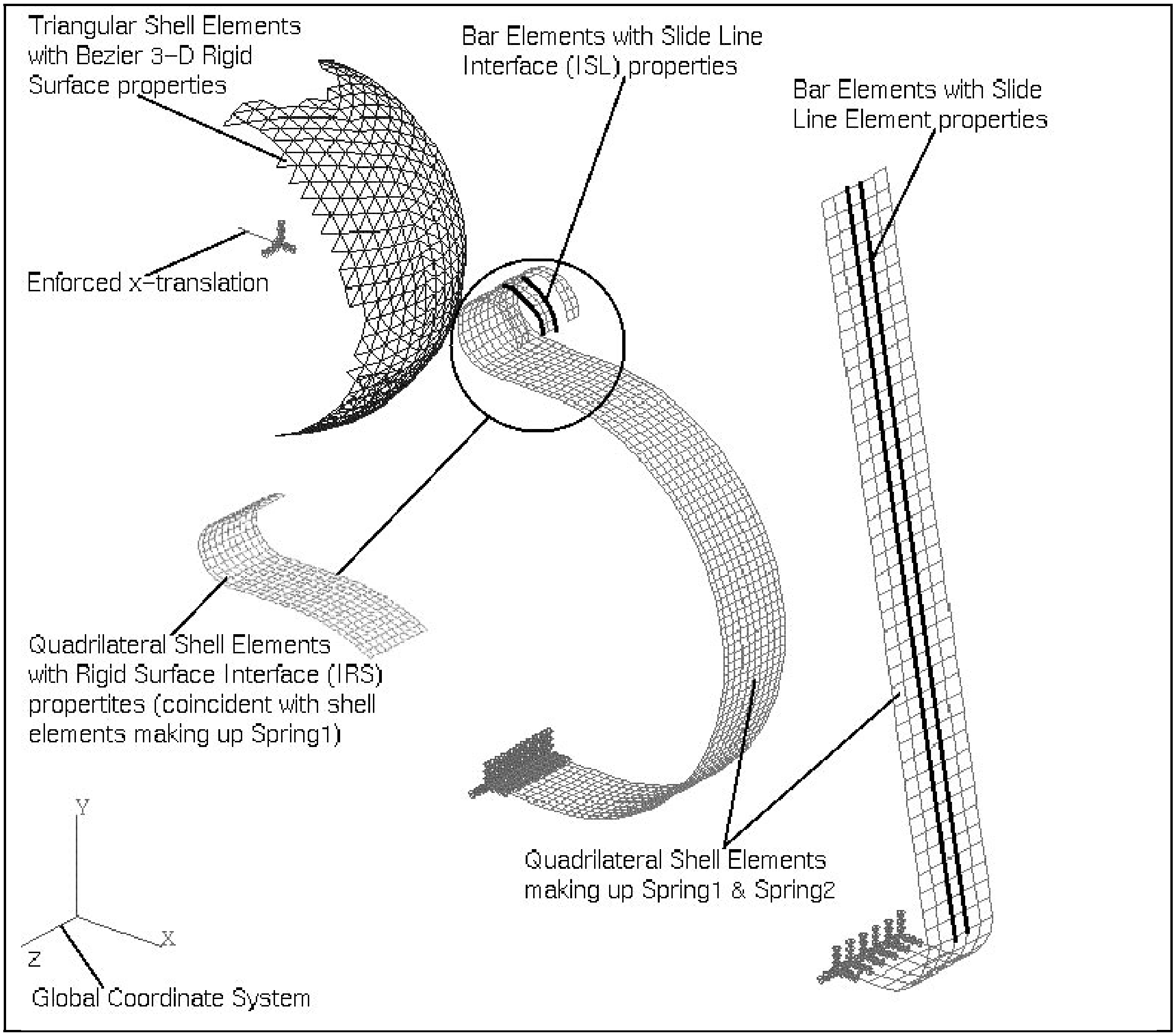

Figure 2. Electro-mechanical sensor finite element mesh.

will have no significant deformation due to contact with the ball. These restraints also ignore any gaps due to tolerances between the ball and the housing. The effect of friction between the ball and plastic is negligible in this analysis.

The sensor can be analyzed by applying an enforced displacement in the x-direction to the rigid surface representing the ball to simulate the full displacement of the ball. Contact between the ball and springs is modeled with various contact elements as discussed in the following section. A nonlinear static analysis is sufficient to capture the force-displacement response of the sensor versus using a more expensive and time consuming nonlinear transient analysis. Although the sensor is designed with a ball mass and spring stiffness that gives the desired response to a given acceleration, there is no mass associated with the ball in this static analysis. The mass of the ball can be determined by dividing the force required to deflect the springs by the acceleration input into the sensor.

Mesh

The finite element mesh for the sensor was constructed using MSC/PATRAN [2]. The solver used to analyze the sensor was MSC/ABAQUS. The finite element mesh including the contact elements is shown in Figure 2. The plastic housing was assumed to be rigid in this analysis and was not modeled. Both springs were modeled with linear quadrilateral shell elements with thin shell physical properties. The ball was assumed to be rigid and was modeled with linear triangular shell elements with Bezier 3-D rigid surface properties.

To model contact between the ball and spring1, rigid surface interface (IRS) elements were used in conjunction with the Bezier 3-D rigid surface elements making up the ball. Linear quadrilateral shell elements with IRS physical properties were placed on spring1 and had coincident nodes with the quadrilateral shell elements making up spring1. The IRS elements were used only in the region of ball contact.

To model contact between spring1 and spring2, parallel slide line interface (ISL) elements were used in conjunction with slide line elements. Linear bar elements with ISL physical properties were placed on spring1 and had coincident nodes with the shell elements on spring1. Linear bar elements with slide line physical properties were placed on spring2 and had coincident nodes with the shell elements making up spring2.

Material

Both spring1 and spring2 were thin metallic springs modeled with a linear elastic material model. No material properties were required for the contact or rigid surface elements.

Boundary Conditions

Both springs were assumed to be fully restrained at their base to simulate a rigid plastic housing. An enforced displacement in the x-direction was applied to the ball. The ball was fully restrained in all rotational and translational directions with the exception of the x-direction translation. Boundary conditions for the springs and ball are shown in Figure 2.

DISCUSSION

Typical results of interest for an electro-mechanical sensor would be the deflected shape of the springs, the force-displacement response of the sensor, and the stress levels in the springs. Results from an analysis of the electro-mechanical sensor shown in Figure 2 will be used as

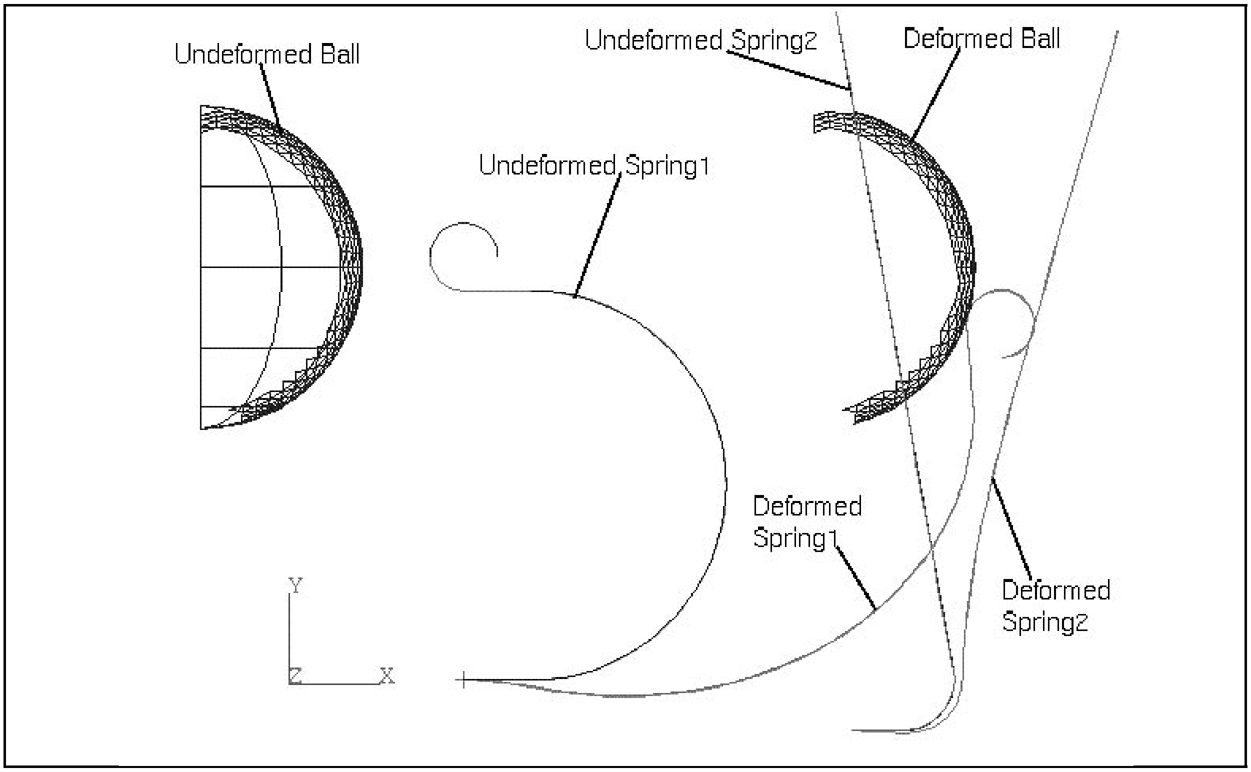

Figure 3. Electro-mechanical sensor deflected shape.

an example for this paper. The deflected shape of this sensor is shown in Figure 3 for full ball travel. Looking at the deflected shape of the springs can provide insight into the performance of the sensor as well as aid in the design of the sensor housing. Stresses in the springs are important results in this analysis to ensure stress levels in the springs are at acceptable levels. Desired components of stress can be examined through various means including color contour plots. One of the most important results from the analysis is the force-displacement response for the sensor shown in Figure 4. From this force-displacement response, the force required to push spring1 into contact with spring2 can readily be determined. This force requirement can be used with a given acceleration to determine the mass required for the ball. Based on these results, one or more variations of several variables such as spring width, spring thickness, ball diameter, and ball material can be updated until the force-displacement requirements are achieved within a desired accuracy.

A prototype of the sensor shown in Figure 2 was constructed and tested to determine its actual force-displacement response. Figure 4 shows the MSC/ABAQUS results along with the experimental results for the force-displacement response of the sensor. There was an excellent correlation between finite element and experimental results for this sensor as well as

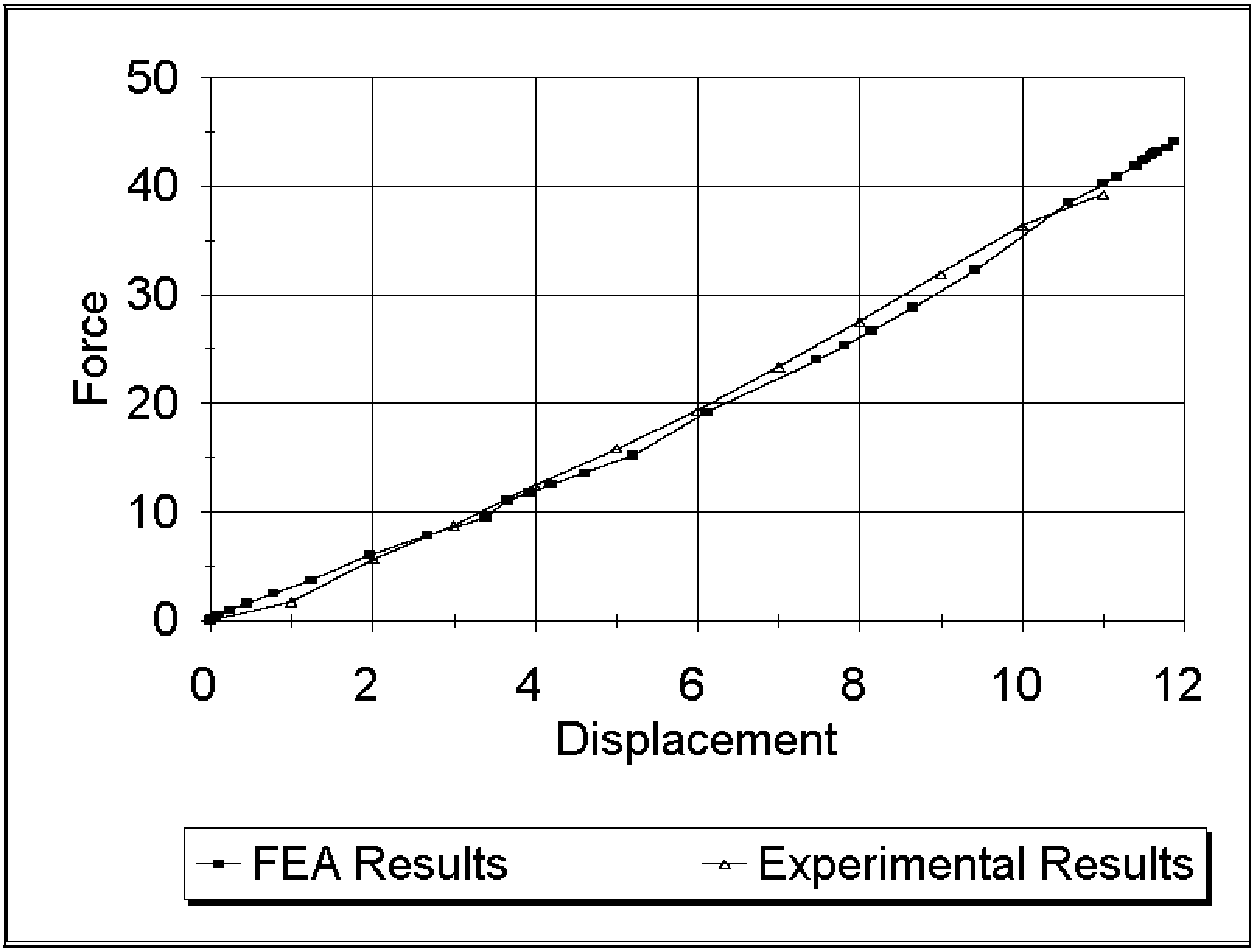

Figure 4. Electro-mechanical sensor force-displacement response.

for several other sensors examined. Table 1 shows the difference in percent between finite element and experimental results including force at preload on spring1, force at spring1-to-spring2 contact, and force at full ball travel for two sensor configurations. Sensor A in Table 1 is shown in Figure 1. Sensor B in Table 1 is based on the sensor shown in Figure 2.

The sensor model analyzed in this paper was also analyzed with parabolic quadrilateral and bar elements to ensure convergence of the solution. Force-displacement results converged to less than 1% using linear elements. The stresses in the springs for this sensor converged to within 10% for the linear elements. The parabolic elements increased solve time by more than an order of magnitude over the linear elements. With more complex spring shapes, a denser linear mesh or parabolic elements used locally in areas of stress concentrations would be necessary to obtain more accurate stresses in the springs.

	%Difference Between FEA and Experimental Results		
Sensor [1]	Force at Preload	Force at spring1-to-spring2 contact	Force at full ball travel
A	+2.0	+1.0	- [2]
B	+1.6	-0.5	+1.0

Table 1. Comparison of FEA versus Experimental Force-Displacement Responses.

Notes:
1. Sensor A results are based on 1 prototype manufactured and tested. Sensor B experimental results are based on the average of 20 prototypes manufactured and tested.
2. No experimental data for force at full ball travel for Sensor A.
3. %Difference=(FEA Result - Experimental Result)/Experimental Result

CONCLUSIONS

MSC/ABAQUS has been used to analyze and design airbag crash sensors. The finite element analysis results were in excellent agreement with experimental results for several electro-mechanical sensors for which prototypes were built and tested. Using finite element analysis, sensors can be designed to meet force-displacement requirements with acceptable stress levels. The analysis procedure discussed in this paper has demonstrated the ability to eliminate months of prototyping effort. This paper has demonstrated the power of finite element analysis as a predictive engineering tool even with the use of multiple contact element types.

REFERENCES

(1) MSC/ABAQUS User's Manual, Version 5.4, The MacNeal-Schwendler Corporation, Los Angeles, CA, 1995.

(1) MSC/PATRAN User's Manual, Version 1.4, The MacNeal-Schwendler Corporation, Los Angeles, CA, 1995.