

Finite Element Analysis of a Coil Spring RFID Antenna Embedded in a Rubber Patch

Steven M. Cron

Michelin Americas Research & Development Corp.
Greenville, South Carolina

One of the many powerful capabilities of Abaqus is its ability to easily model a structure embedded inside of a host structure. This technique has been used to analyze an RFID chip and helical antenna embedded in a rubber patch for use in tire applications. The main objective of the analysis of the chip/antenna/patch assembly was to determine the stress concentration in the antenna caused by its interaction with the relatively rigid chip. In the course of conducting this analysis, a high degree of sensitivity was observed in the interaction between the embedded helical antenna and the host rubber patch. This paper will show how adequate refinement of the host structure becomes crucial if accurate results are to be obtained with the embedded element technique. It will also present results of the antenna stress concentration analysis.

Keywords: Embedded Element, Mesh Refinement, RFID, Coil Spring.

1. Introduction

The use of RFID chips is now commonplace in a broad range of applications. To facilitate communication with the RFID device, an antenna is sometimes required. In the case of RFID chips used in tires, one proposed solution for the antenna is to employ a coil spring which, along with the RFID chip, is embedded in a rubber patch.

Abaqus provides a powerful tool for analyzing such structures with the Embedded Element technique wherein a mesh representing a given structure can be embedded within an independent mesh representing a host structure. In the course of using this technique to analyze stresses in an embedded RFID coil spring antenna, a significant degree of non-physical stiffening was observed, depending on the host mesh refinement. This paper will explore the parameters that influenced this observed stiffening and provide some ideas for how to assess the quality of results when modeling complex structures embedded in relatively low modulus host structures. A brief summary of the stress analysis results for a practical implementation will also be presented.

2. Test Case Modeling

2.1 Strategy

The original motivation for this effort was to understand the stress concentrations produced in a steel coil spring when used as an antenna soldered to an RFID chip assembly and embedded in a deformable rubber host structure. The analysis strategy employed was to first analyze a simple structure involving a representative spring model embedded in a representative host model. This allowed for a quick assessment of the effect of various parameters influencing the interactions caused by the embedding constraint.

A test spring model that yields reasonably accurate stress and stiffness results will first be presented. Subsequently, a simple cylindrical rubber test model will be evaluated. The next series of steps will involve analysis of the combined structure where the spring mesh will be held constant and the host mesh varied.

2.2 Spring Test Model

The spring test model is shown in Figure 1. Note that for a practical implementation the antenna would be much longer, up to 50 mm.

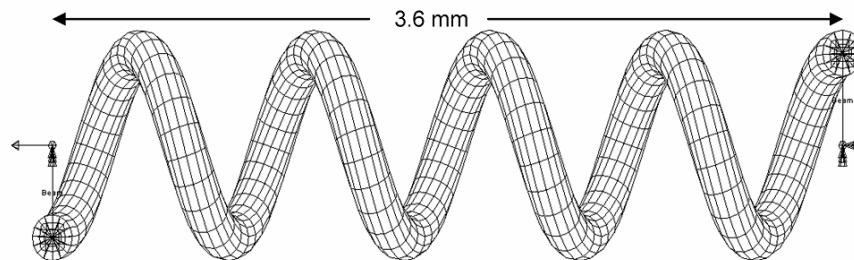


Figure 1. Test Model of Coil Spring.

Catia V5 was used to generate the geometry and Abaqus CAE was used to import the geometry (in STEP format) and generate the finite element model. The mesh is comprised of linear hexahedral elements with reduced integration (C3D8R).

To evaluate the quality of the spring model, the structure was stretched by 0.1 mm (2.7%) along its centerline. The reader should note that the load was applied along the spring axis using beam connectors which allowed the spring to both elongate and rotate freely.

Figure 2 shows a section of the deformed spring with the peak shear stress highlighted.

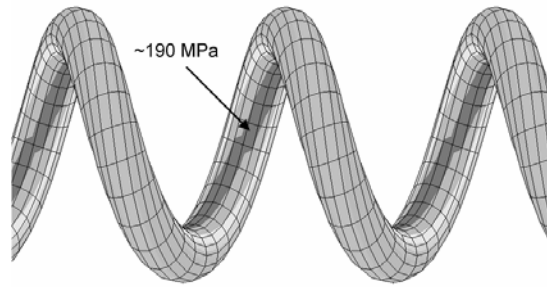


Figure 2. Maximum Computed Shear Stress (2.7% Stretch).

As indicated, the peak shear stress computed with Abaqus was approximately 190 MPa. To check the quality of the model, the Abaqus result was compared to the theoretical value given by Shigley [1]:

$$\tau_{\max} = \varepsilon \frac{Gd^2}{PD^3\pi} \left(\frac{D}{d} + 0.5 \right) = 191.3 \text{ MPa} \quad (1)$$

Where,

τ = Shear stress

ε = Spring stretch expressed as strain = 0.027 mm/mm

G = Shear modulus of steel = 76,923 MPa

P = Spring coil pitch = 1.2 turns per mm

D = Mean spring coil diameter = 0.8 mm

d = Spring wire diameter = 0.2 mm.

The spring stiffness can also be compared to the stiffness predicted by the following well known expression,

$$K = \frac{Gd^4}{8N_a D^3} = 6.68 \text{ N/mm} \quad (2)$$

Where N_a is equal to the number of “active” coils which for this configuration is 4.5. The predicted stiffness computed with Abaqus is 6.13 N/mm.

Given the level of approximation involved, the agreement between the theoretical values and numerically computed values seems reasonable.

2.3 Test Model of Antenna Embedded in Rubber Host

With a satisfactory model for the spring in hand, we next turn our attention to modeling of the host rubber test structure into which the antenna model was embedded. The simple cylindrical test model shown in Figure 3 was used to evaluate the interactions between the spring mesh and the rubber mesh. The cylinder diameter is 1.5 mm and the length is just slightly longer than the total spring at 3.95 mm.

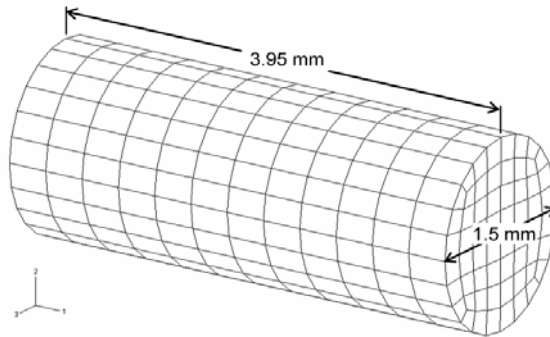


Figure 3. Test Host Cylinder Model.

Since our intention was to model the spring embedded in an incompressible rubber host, the elements in this initial model are linear (Q1) hybrid hexahedral elements. The rubber material constitutive behavior was modeled with the Mooney-Rivlin law with $C10 = 0.5$ MPa and $C01 = 0.0$. This will give a Young's modulus of approximately 3 MPa.

Before embedding the spring, this model was also stretched by 0.1 mm (2.5%) along its axis to establish a baseline stiffness. Under this loading condition, the finite element model predicted a stiffness of 1.29 N/mm vs. a theoretical stiffness of 1.34 N/mm based on the Young's modulus given above. It should be noted that the model was loaded in uniaxial tension by applying a $U1 = 0$ condition on the left end and $U1 = 0.1$ mm on the right end. Transverse deformations were not constrained except to eliminate the rigid body translations.

At this point we can say that, individually, the models for the spring and cylinder are reasonably accurate for the prediction of stress and load under prescribed deformation conditions. It would also seem safe to make an initial estimate of the combined stiffness of the model shown in Figure 4 to be the sum of the two individual FE derived stiffnesses: $6.13 \text{ N/mm} + 1.29 \text{ N/mm} = 7.42 \text{ N/mm}$. This estimate assumes that the two structures act as springs in parallel.

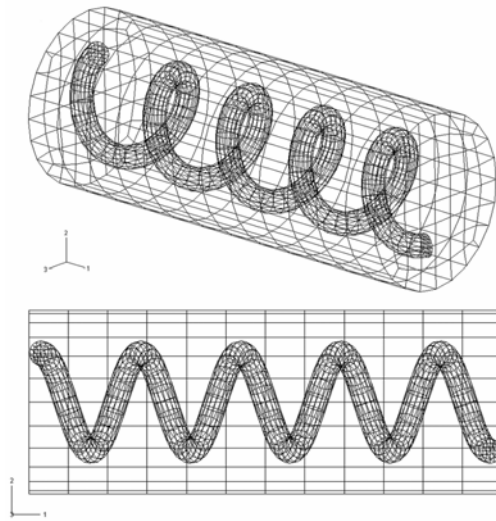


Figure 4. Test Host Model with Embedded Spring Model.

Upon application of a 0.1mm elongation to the combined model of Figure 4, however, a very different result was obtained. In fact the predicted stiffness was 578 N/mm! The deformed model is shown in Figure 5. Note that the spring model was embedded in the host using the default Abaqus/Standard embedded constraint (***EMBEDDED ELEMENT**).

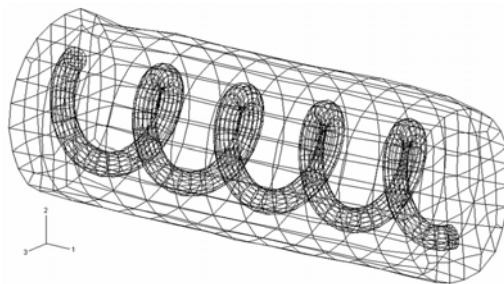


Figure 5. Deformed Test Model.

At this point we must conclude that either a) there is a huge error introduced by the interaction between the host and embedded models or b) we are observing a non-obvious physical interaction between the spring and the incompressible host material. Most likely the observed stiffening was erroneous.

Our interest at this point was to explore the causes of this stiffening behavior and to satisfy ourselves that we had an accurate result. To that end we explored the two most likely causes for

the observed behavior: constitutive law (including incompressibility) and mesh refinement (including element order).

The first variation to check was the effect of changing the material from a hyperelastic constitutive law (Mooney-Rivlin in this case) to a simple linear elastic law with a modulus of 3 MPa and a Poisson's ratio of 0.333. The element type was also changed from hybrid to a default linear element. This change reduced the stiffness from 578 N/mm to 27 N/mm. Clearly, incompressibility is a major source of the observed stiffening behavior but the difference with respect to the initial estimate of 7.42 N/mm is still nearly 400%. In addition, given that the analysis goal was to model the spring embedded in a rubber host material that is in fact nearly incompressible, this change to the model is instructional but not very practical.

Returning to a hyperelastic constitutive law, the next step taken in exploring the stiffening behavior was to adjust mesh refinement. The reader will note that the mesh of Figure 5 is divided into 12 elements along its length. In this next step the number of elements along the length was varied from the initial value of 12 up to a value of 30. Stiffness is plotted as a function of this parameter in Figure 6.

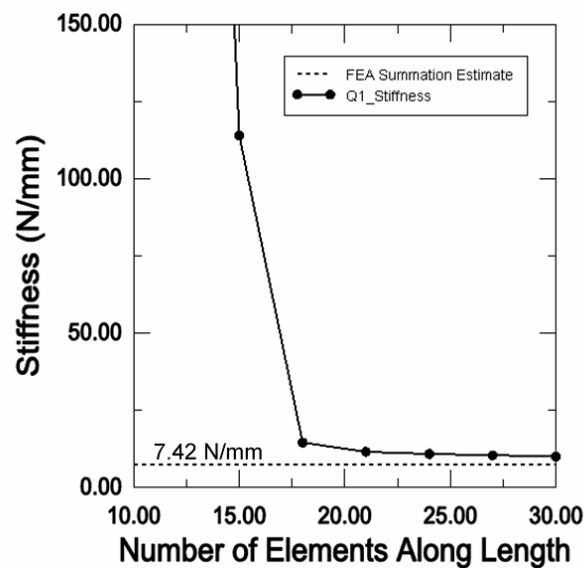


Figure 6. Stiffness as a Function of Mesh Refinement in Cylinder Axial Direction.

Clearly the overall stiffness is strongly dependant on this mesh refinement. With 18 elements along the cylinder axis, the difference between the value computed with the FE model and the summation estimate of 7.42 N/mm becomes relatively small.

We can now easily check the effect of element order by simply converting the models used to generate the Q1 stiffnesses in Figure 6 to quadratic (Q2) elements. Figure 7 shows the Q1 results

along with Q2 results for 12, 18 and 30 elements in the axial direction along with the FE summation estimate.

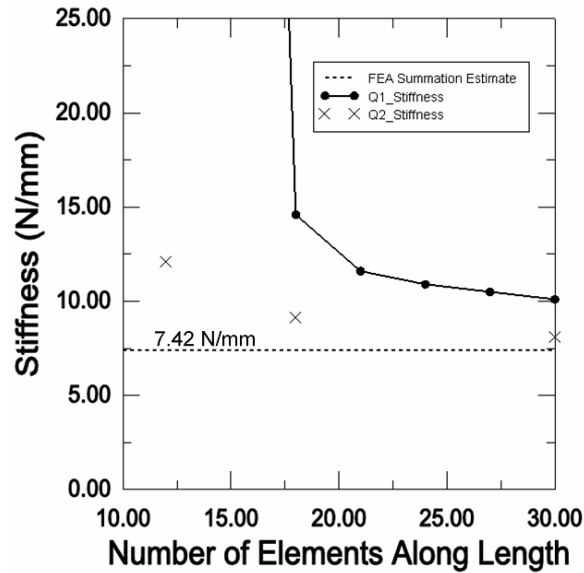


Figure 7. Stiffness Effect of Using Q2 Elements.

Again, the use of Q2 elements greatly reduces the tendency of the model to stiffen even using the relatively coarse 12-element mesh.

To this point we have only addressed refinement of the host mesh along the length of the cylinder. For completeness, a quick check of the effect of cross-section mesh refinement was also made. Figure 8 shows the cross-section mesh of the reference model (of Figure 4) and a refined test mesh.

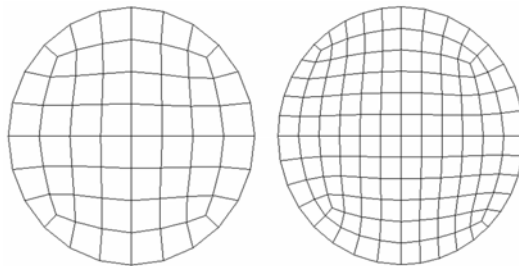


Figure 8. Reference and Refined Cross-Section Meshes.

Table 1 compares the stiffnesses over the range of lengthwise mesh refinements for the two cross-section meshes shown above.

Table 1. Comparison of Stiffness for Two Cross-Section Meshes (Q1 Elements).

Number of Elements Along Length	Stiffness (N/mm)	
	Reference Cross-Section Mesh	Refined Cross-Section Mesh
12	578	773
15	114	159
18	14.6	12.2
24	10.9	9.73
30	10.1	9.15

What is interesting to note here is that the adverse effect on stiffness is exacerbated by the finer cross-section mesh when combined with the coarser lengthwise meshes.

The last effect that should be explored is that of changing the geometry of the spring, in particular the number of coil turns per unit length. For this test we embedded an equal-length spring of 2.5 turns vs. the reference (of Figure 4) spring with 4.5 turns. All other parameters were held constant. The new combined model is shown in Figure 9. From Equation 2, the theoretical spring stiffness is 12 N/mm. Adding the baseline rubber cylinder stiffness of 1.29 N/mm gives us an estimated stiffness of 13.3 N/mm for the combined structure.

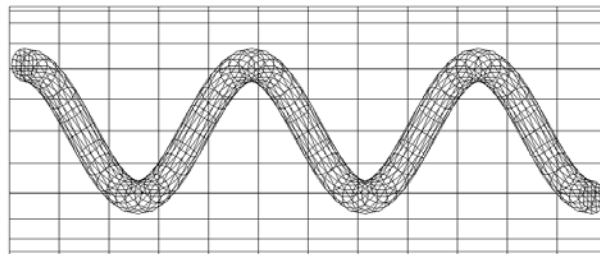


Figure 9. Model of Coil Spring with 2.5 Turns.

A comparison of the stiffness as a function of lengthwise Q1 mesh refinement is again shown in Figure 10. What we see now is there is no longer the dramatic increase in stiffness as the lengthwise refinement is reduced. Specifically, the difference between the reference cylinder model plus embedded spring with 12 elements along the length and the theoretical summation is only 30% whereas the equivalent configuration with the 4.5 turn coil showed a difference of a factor of nearly 8.

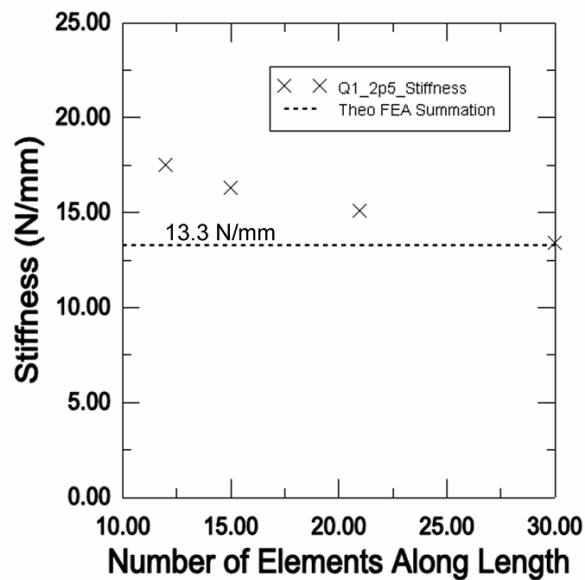


Figure 10. Stiffness as a Function of Mesh Refinement in Cylinder Axial Direction with 2.5 Turn Spring (Q1 Elements).

What then is the critical parameter that controls this stiffening behavior? One hypothesis would be that the number of elements spanning the gap between spring coils must be above some minimum threshold level. Looking back to Figure 6 we see that a huge reduction in artificial stiffening was observed (for the Q1 models) when the number of elements along the length was at or above 18. If we divide 18 by the number of turns, 4.5, we get a value of 4 elements per spring coil. We can test this hypothesis with the 2.5 turn model of Figure 9. Given that the spring has 2.5 turns we would expect to see a large rise in artificial stiffening with any number of elements along the length less than 10. In fact, when the number of elements along the length is reduced to 8, the stiffness value jumps to 50 N/mm as expected. In the limit, as the spring becomes a straight rod, the sensitivity to lengthwise mesh refinement disappears.

While a “4 element rule” might be satisfactory for modeling embedded springs, a more general diagnostic technique would certainly be more useful for assessing the quality of results where the embedded structure is of an arbitrary shape. One possible technique would be to look at elements grouped by maximum distortion. For our purposes, the maximum shear strain is taken as a measure of total element distortion. Thus, the first step would be to calculate the maximum shear strain (max principal strain – min principal strain) for each element using the “Create Field Output from Field” function within Abaqus/CAE. The second step would be to use Display Groups to eliminate from the display all elements above an arbitrary maximum shear strain level. As this arbitrary level is reduced, more and more host elements will disappear. At some point, elements from the embedded body will also begin to disappear. At the level where embedded elements begin to disappear, the following assessment can be made. If the remaining host elements are

nicely grouped around the embedded body as in Figure 11 below, the analysis is probably good. If, on the other hand, the remaining host elements tend to “bridge” the gaps in the embedded body as in Figure 12 below, the results are probably not to be trusted. The rationale for this technique is based on the assumption that the host structure is of considerably lower modulus than the embedded structure and should, therefore, have much greater distortion than the host structure at points away from the embedded structure. Obviously this technique would become less and less valid as the modulus of the embedded structure approached that of the host.

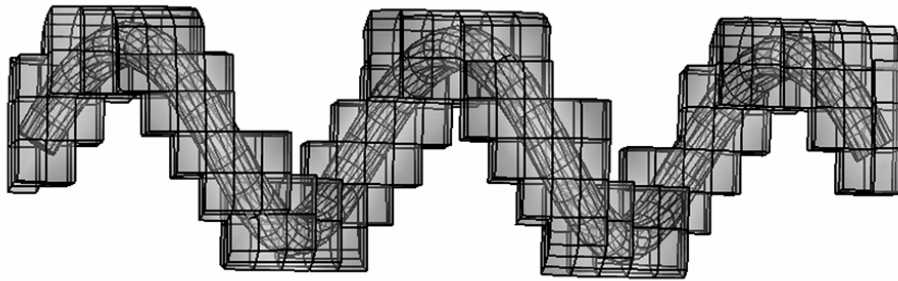


Figure 11. Example of Host Elements “Clustered” Around Embedded Structure.

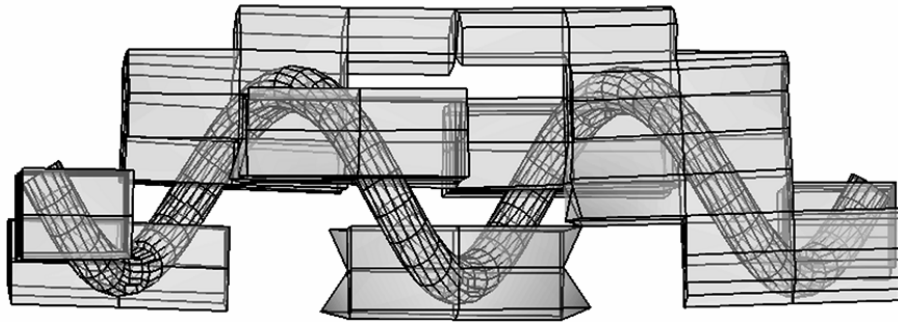


Figure 12. Example of Host Elements “Bridging” Gaps in Embedded Structure.

3. Modeling of RFID Chip and Antenna Embedded in Rubber Patch

3.1 Introduction

Confident that we understand how to model a spring embedded in a rubber cylinder, we can move on to look at the results for an actual application. Our objective will be to determine the stress concentration factor seen by a steel coil spring antenna when it is soldered to an RFID chip assembly, embedded into a generic patch of rubber, and then stretched by 4%. Symmetry is assumed so that only half of the structure need be modeled.

3.2 Model of Chip, Solder and Base

The half-model of the chip, solder and base is shown in Figure 13. Figure 14 shows the same model with the antenna model in its position embedded in the solder joint.

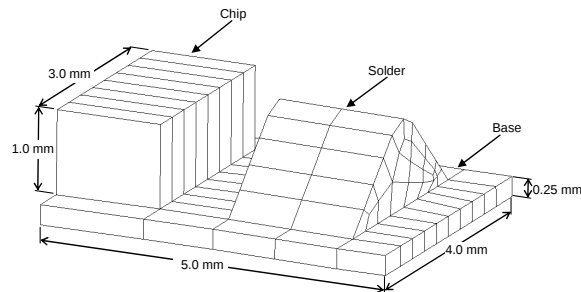


Figure 13. Chip, Solder and Base Model

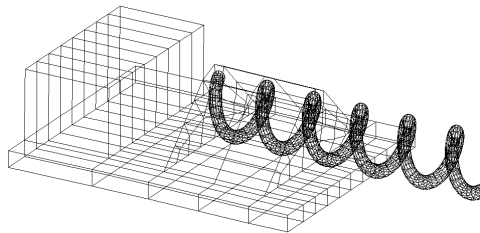


Figure 14. Chip, Solder and Base Model with Antenna Embedded

This model is intended to constrain the deformation of the coil antenna and provide a reasonable representation of the discontinuity caused in the patch rubber. The elements of the chip/solder/base are linear hexahedral (C3D8R) and the material is taken to be linear elastic with a modulus of 3000 MPa.

3.3 Combined Model

The combined model of the patch, chip/solder/base assembly, and antenna coil is shown below in Figures 15, 16, and 17. It should be apparent that the model represents half of the entire structure. As such, symmetry boundary conditions were enforced as indicated in Figure 15.

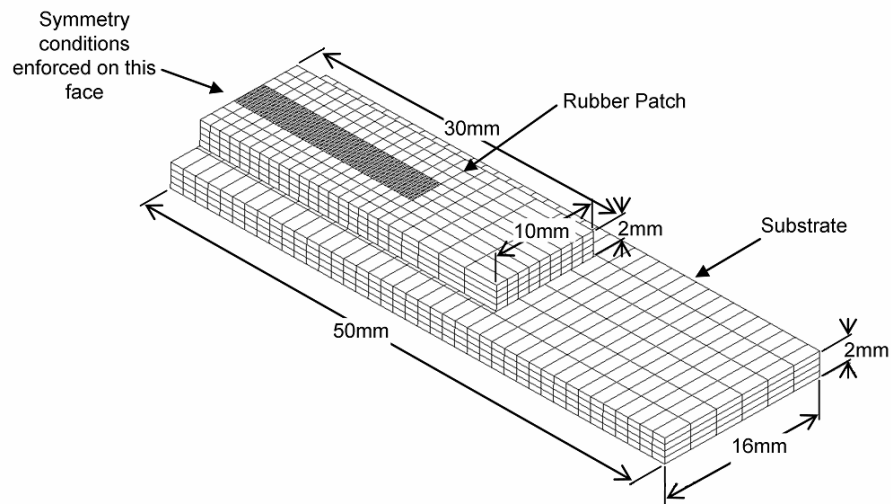


Figure 15. Combined Model of Substrate, Patch, Coil and Chip

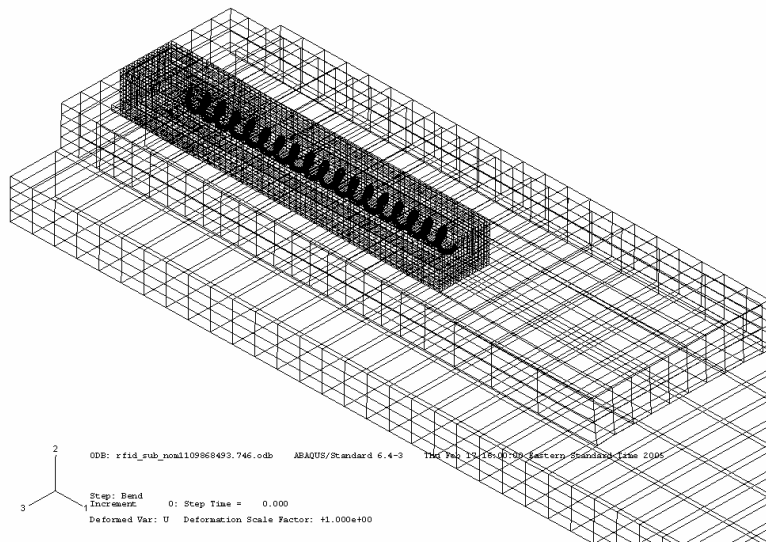


Figure 16. Zoom of Combined Model of Substrate, Patch, Coil and Chip

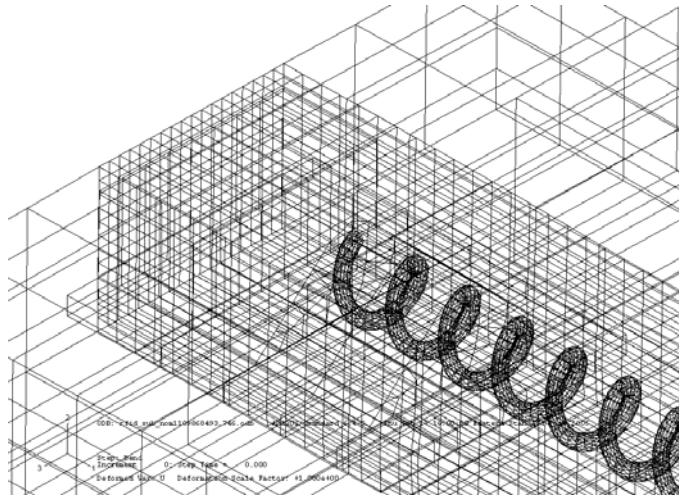


Figure 17. Zoom of Region where Coil is Embedded into Solder and Patch

The mesh of the substrate region and the coarsely meshed portion of the patch were comprised of C3D20RH elements. The fine mesh into which the chip assembly and antenna were embedded was comprised of C3D20H elements. The patch rubber was again modeled as a Mooney-Rivlin material with $C10=0.5$. The substrate was modeled as a linear elastic material with a modulus of 3000 MPa. The model contained approximately 259,000 d.o.f.

4. Results

4.1 Results for Stretch Loading Condition

As mentioned above, the intended deformation of the patch rubber structure was to be 4% elongation. To evenly distribute this deformation into the patch, the relatively stiff substrate material was stretched along the axis of the spring such that the entire structure experienced a nominal imposed elongation of 4%. Figure 18 shows the strain in the 11-direction (LE11).

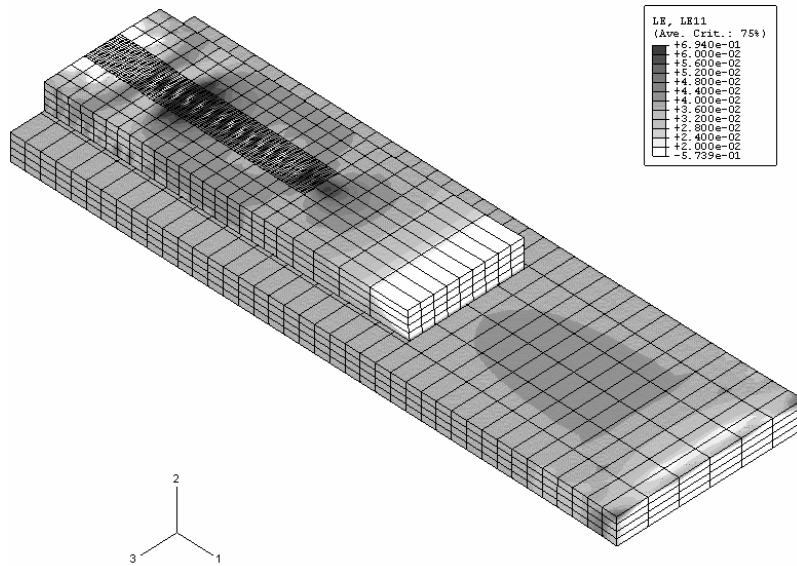


Figure 18. 11-Strain Due to 4% Elongation in 11-direction

In Figure 19 we isolate just the portion of the patch rubber surrounding the chip assembly and the antenna and see the strain field at this level.

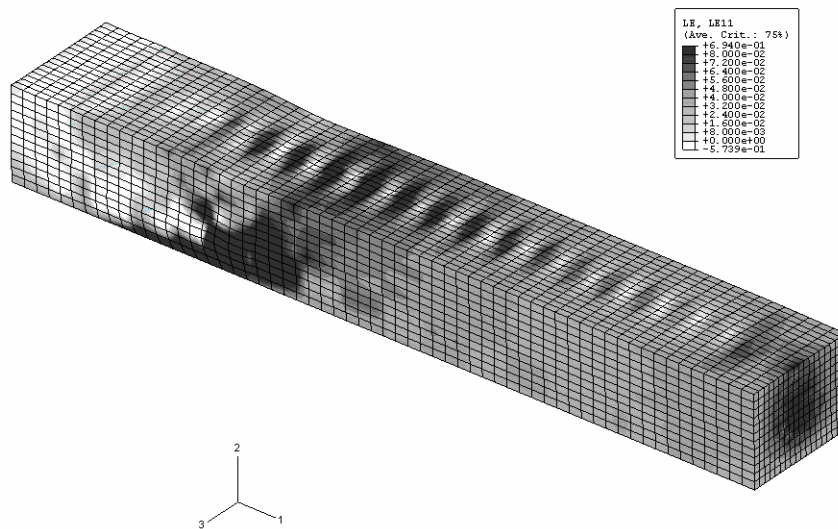


Figure 19. 11-Strain in Patch Rubber Surrounding Chip Assembly and Antenna

Finally, in Figure 20 we have isolated the chip assembly and the antenna to see the effect on the stresses in the antenna. Note that this view is from below the chip assembly base looking up.

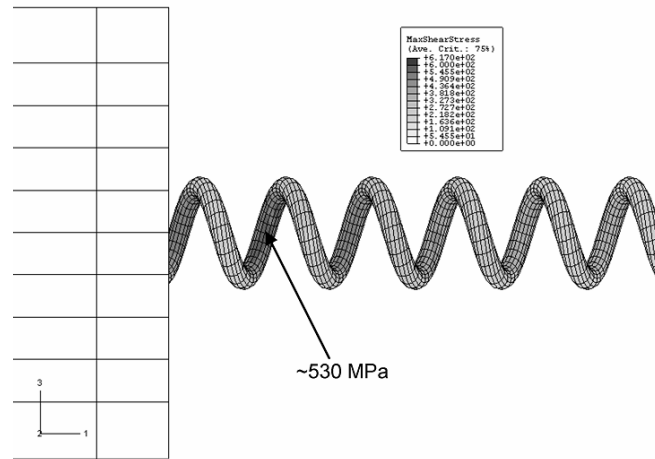


Figure 20. Maximum Shear Stress Due to Stretch

The peak shear stress is approximately 530 MPa. Referring back to Equation 1, we can calculate the theoretical shear stress at 4% spring elongation to be 286.9 MPa. Furthermore, we can say that the stress concentration due to the antenna and chip being embedded in the patch rubber is roughly $530/286.9 = 1.85$.

5. Conclusions

The first, and probably most significant finding of this work is the fact that modeling of embedded structures that have complex geometries must be done with some care. It has been shown that enormous amounts of artificial stiffness can be generated when host mesh refinement (or element order) is too low. Remedies for this behavior include increasing the order of the host elements as well as increasing mesh refinement. A proposed technique for evaluating the quality of results has also been presented based on a qualitative measure of the degree to which elements of relatively low distortion are grouped around the embedded structure.

The analysis of the chip/solder/base assembly embedded in a low modulus rubber patch indicates that there is significant effect on the patch rubber strain field in the vicinity of connection between the antenna and the RFID chip. For a nominal elongation of 4% the net effect on the stress concentration in the antenna has been found to be 1.85.

6. References

1. Shigley, J.E., "Mechanical Engineering Design", Third Ed., McGraw Hill, 1977.