Applications and Impact of the new Beam to Beam Contact Capability to Industries

Peter Zhang[1],Xiaoliang Qin[2],George Ang[2], Wei Gan[1], and Subham Sett[2]

[1] Medtronic CRDM and [2] Dassault System SIMULIA Corporation

Abstract: In this paper, we discuss applications and impacts of the new beam to beam contact capability from Abaqus V6.13 to industries – in particular to the medical device and automotive industries. The applications that we interested in the medical device industry are bending simulations on the implantable leads, specially, bending simulations on coils and multi-strand cables. The applications in the automotive industry include simulations on bending of electrical cables, wires, and wire bundles. Being able to accurately and fast model the stress or strain to the coil filar and cable strand level for the problems above are important for design engineers to predict the field performances and reliabilities of the products in both medical device and automotive industries.

Without the beam to beam contact, the simulation problems mentioned above are either intractable or take very long computational time. With the new beam to beam contact simulation capability, the intractable problems could become tractable, and the computational time of the problems could be reduced by orders of magnitude. Because of the significant reduction of the computational time, we could potentially model problems that were not possible to do such as exploring the design space and create response surfaces or creating automated interactive modeling tools that can be accessed by design engineers.

1. Introduction

In the medical device industry, implantable leads connect the signal or power generator device to therapy delivery sites. The therapies include both cardiac and neurological stimulations. For cardiac rhythm devices in Figure 1, the cardiac leads connect the pacing or defibrillation generator to the heart. The construction of a cardiac implantable lead is illustrated in Figure 2. It consists of metal coils or cables as conductors and polymer tubing as insulators.

Due to patients' body motions, implantable leads experience a great deal of repetitive motions. One common loading condition on the leads is bending. To simulate bending of a lead and to obtain detailed stress and strain of each coil filar and cable strand in the lead could be time consuming and sometime the problem could be so large that become intractable. To speed up the simulations, component abstraction could be employed to reduce the size of the problem. For this case, the coil filars and cable strands can be abstracted as beam components because of the small ratio of the cross-section radius to the length. By using beam elements, the sizes and time of the simulation problems could be reduced significantly. With shorter simulation time, more simulations can be performed to thoroughly explore the design space of a product. However, to solve the bending problem correctly, beam to beam contacts have to be included in the simulations. The beam to beam contact needs to be a cross section profile to profile contact (or outer surface to outer surface contact), not just centerline to centerline contact.

The need of having beam to beam contact is not unique to the medical device industry. The beam to beam contact simulation capability is also needed in the automotive industry. For simulating the fatigue performance of the wire bundle for the door harnesses, the beam to beam contact is also needed. In fact, one of the authors requested to add the beam to beam contact capability to Abaqus from his previous job in the automotive industry about 10 years ago. After moving to the medical device industry, the same author found the similar needs of beam to beam contact in the medical device industry, and a similar improvement of functions request on beam to beam contact was filed from the medical device industry.

Before Abaqus V6.13, there is no beam to beam contact in Abaqus/Standard. In Abaqus V6.13, the new functionality of beam to beam contact will be introduced. As part of the beta test, we use medical lead body bending as a test case. In this paper, we report the results of beam to beam contact on the case of medical device lead body bending.

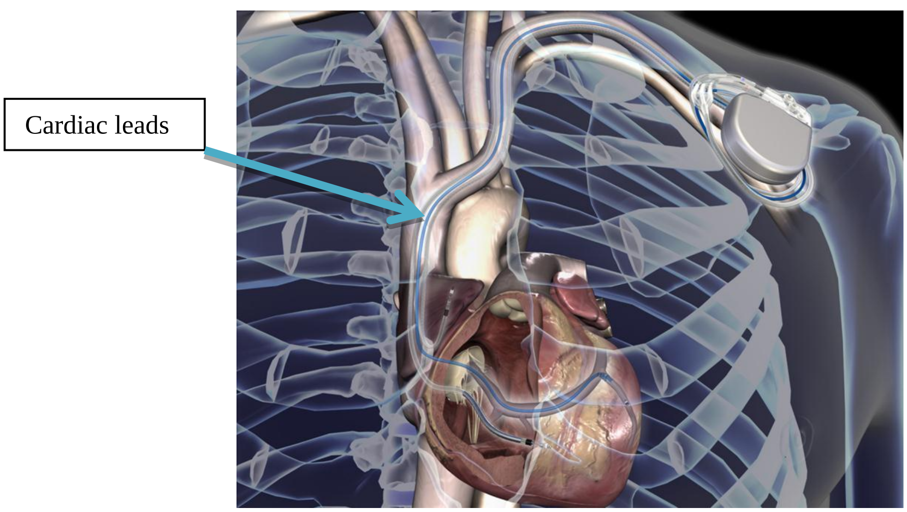

Figure 1. Cardiac implantable leads connect the device with stimulation sites in the heart.

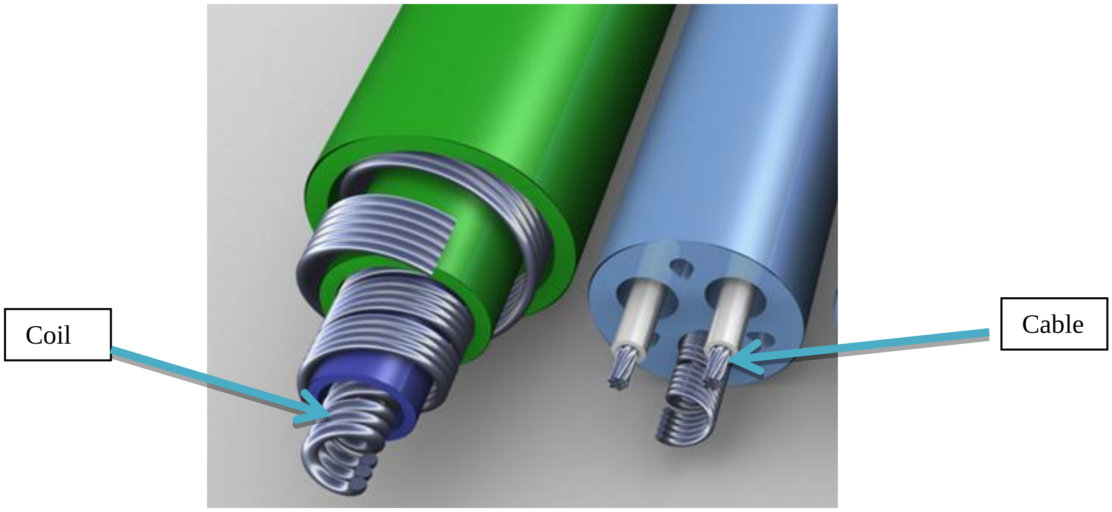

Figure 2. Representative cardiac lead constructions: Multilayer construction with coil and tubing (left). Multi lumen construction with both cable and coil (right).

2. Model Description

To test the new beam to beam contact capability, a generic medical implantable lead is employed in our modeling study. Figure 3 shows the lead's geometry model. The model consists of the exterior polymer tubing and the five-filar metal coil inside the tubing. The tubing is about 0.15in long cylinder with exterior radius of 0.017in and internal radius of 0.014in. The wire for the coil has a radius of 2e-3in. There are 0.5e-3in initial gap between two filars. The tubing and the metal coils are constrained to move together at both ends using kinematic coupling. The contacts for the model include the coil filar to coil filar contacts and coil filar to tubing contacts.

Three cases of modeling the contact between tubing and the coils are studied in this paper, as shown in Table 1. The first case is to model both of the tubing and the coil using solid elements; the second case is to model the tubing using solid element but the coil using beam elements; the third case is to model both of the tubing and the coils using beam element. The meshes of these three cases are shown in Figure 4.

To apply the bending boundary conditions, coupling constraints are applied to both ends of the lead. The boundary conditions for the bending simulations are fixing the reference point of the coupling constraint at one end and rotating the reference point of the coupling constraint at the other end.

	Tubing	Coils
Case 1	Solid element	Solid element
Case 2	Solid element	Beam element
Case 3	Beam element	Beam element

Table 1. Modeling cases

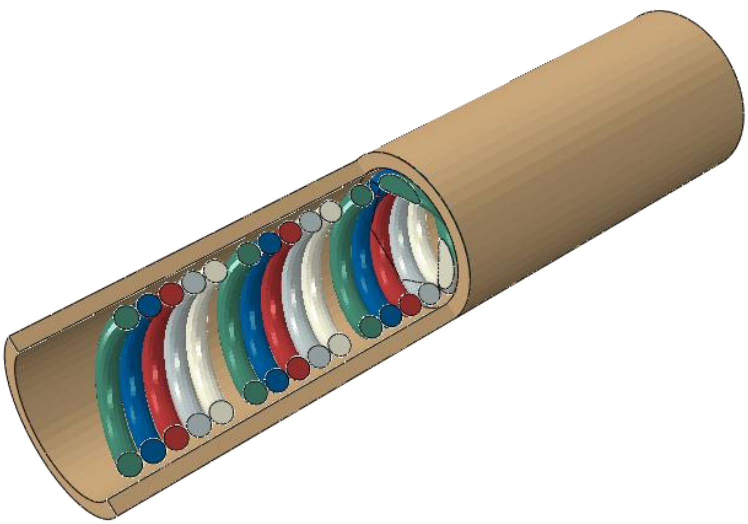

Figure 3. Lead’s geometry model

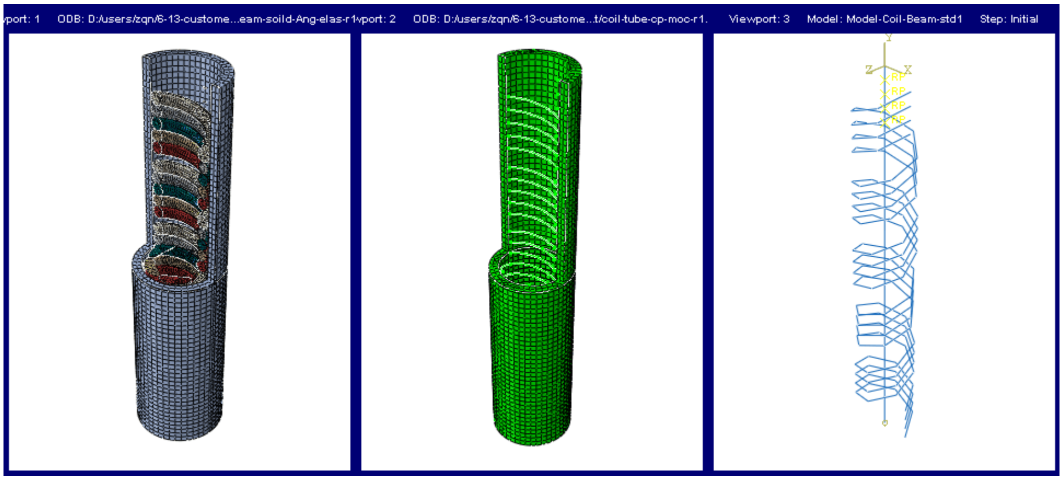

Figure 4. Lead meshes for the three cases in the modeling study

3. Results and Discussion

The deformed configurations of the three cases of lead bending models are shown in Figure 5. Since solid element and the contact between the solid elements are very well developed in ABAQUS, Case 1 is the most accurate model and is used as the results comparison baseline in this study. For case 1 and case 2, since the tubing is modeled as solid elements, it is able to deform freely. Thus, the tubing has local buckling on the compression side for both case 1 and case 2. For case 3, because the tubing is modeled as beam elements with tube profile, no local buckling is allowed for this case.

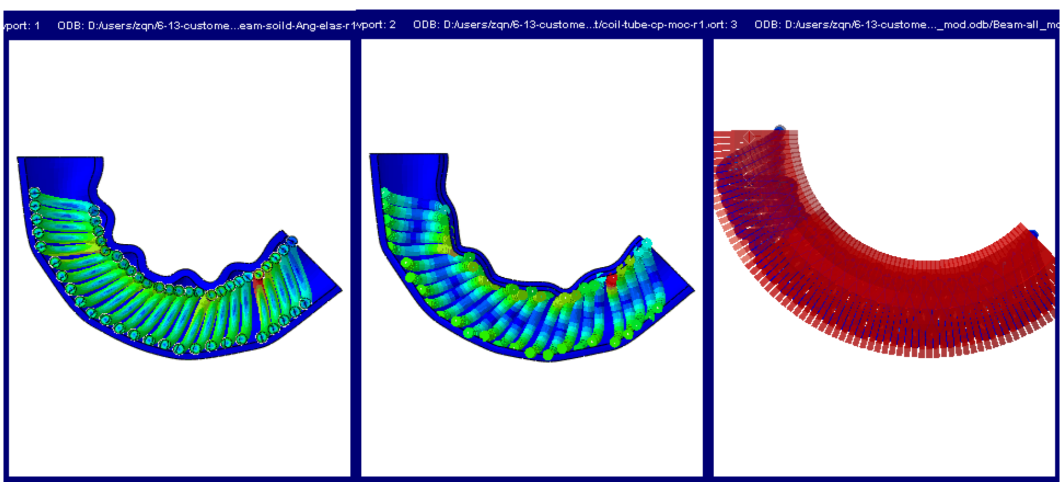

Figure 5. Leads model with mesh

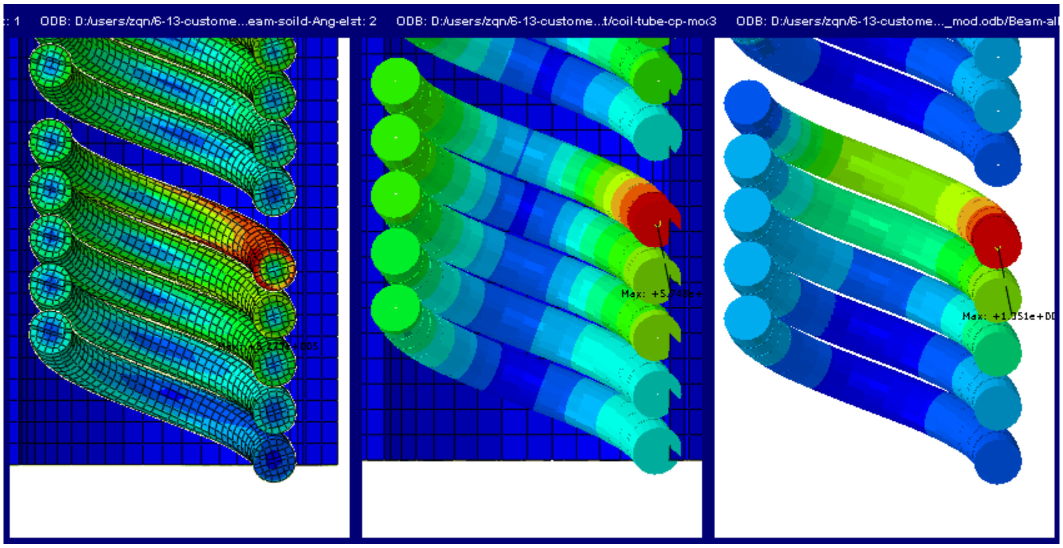

Figure 6. Maximum Von Mises stress in nondeformed configuration

Figure 6 shows the maximum Von Mises stress in the coil. The maximum Von Mises stress are at the same location of the coils for all of the three cases. The displacement and stress history are further studied at the maximum Von Mises stress location (location A) in Figure 7 and Figure 8. In Figure 7, Case 2 and Case 3 have closed displacement in X direction at $0<t<0.5$. After $t>0.5$, the X displacement for Case 2 matches with the X displacement in Case 3. The similar behavior can also be observed in for the Von Mises in Figure 8. The Von Mises stress for Case 2 is a little bit larger than Case 3 during $0<t<0.5$; it becomes smaller than Case 3 during $t>0.5$; the stress for case 1 and 2 are close at $t=1$. The reason is that the coil by beam elements is more "stiff" than the coil by solid elements, because the cross section of the beam is not allowed to deform. The Von Mises for Case 3 is much higher than Case 1 and Case 2. It is because the tube is rigid in cross section, and constrains the deformation of the coil.

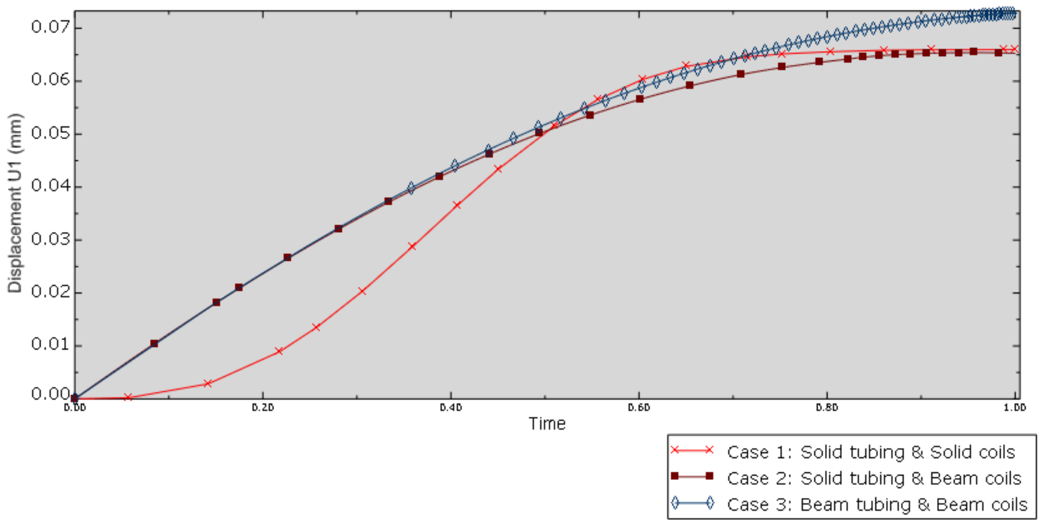

Figure 7. U1 displacement at location A

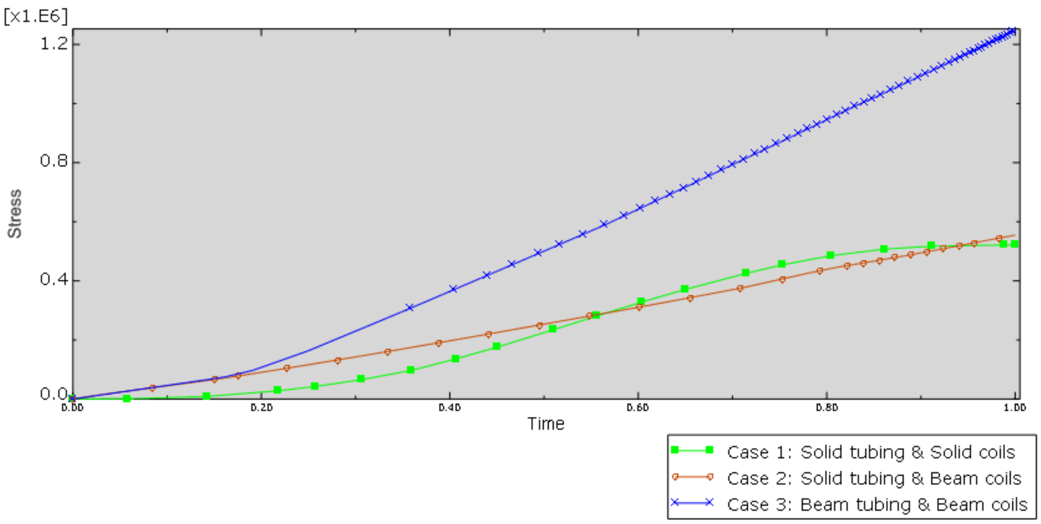

Figure 8. Von Mises stress at location A

4. Summary

Abaqus V6.13 introduces the new beam to beam contact feature; this new feature is a long waited and a welcome addition to the strong existing Abaqus simulation features. This new beam to beam contact feature has many applications in many industries particularly in the medical device and automotive industries. This paper compares the simulation results using the new beam to beam contact feature to simulation results using existing solid element contacts with cardiac implantable lead as the test case. The study shows the following results:

- The maximum stress locations using beam coil contacted with both solid tubing and beam tubing are similar to the full solid to solid contact simulation results.
- The maximum stress values from beam coil to solid tubing contact simulation are close to the result of solid coil to solid tubing contact simulation.
- The maximum stress values from beam coil to beam tubing contact simulation are deviated from results of the solid coil to solid tubing contact simulation because the beam tubing couldn't represent the local buckling of the tubing at higher bending curvature.

Since the beam to beam contact is a new feature, it needs to be further tested and improved with a wider range of applications from various industries.