Investigation of Interaction between Guidewire and Native Vessel Using Finite Element Analysis

Atul Gupta[1], Subham Sett[2], Srinivasan Varahoor[1], and Ben Wolf[1]

[1]Medtronic Inc., [2]Dassault Systemes, SIMULIA Corp.

Abstract: Endovascular aneurysm repair involves insertion of an introductory component called guidewire through native vessels to help with the guidance of the delivery catheter. Guidewire tends to alter the vessel geometry due to its higher stiffness compared to the vessel wall. Very limited data is available to understand such interactions. Investigation of interaction between guidewire and native vessels could provide useful insight into vessel stresses and guidewire deformation in-vivo. This could further help in understanding the initial conditions for delivery catheter and device performance testing to improve deliverability in tortuous anatomy. So, in this study, we evaluated the vessel deformation during guidewire insertion using 3-D finite element analysis with undeformed vessel geometry developed from patient specific Computed-Tomography (CT) scans. Pre-operative CT scans were used to generate 3-D reconstructions of human iliac region using Mimics (Materialise, Ann Arbor, MI). A finite element model was set up to simulate the vessel deformation where a guidewire was introduced from the distal end of the iliac and pushed up to the aneurysm region. The guidewire insertion analysis was run with both Abaqus/Standard and Abaqus/Explicit to compare the performance and to determine advantages of using one procedure over the other. Both Abaqus/Standard and Abaqus/Explicit were able to successfully predict the overall deformed shape of the vessel accurately. It was also observed that the guidewire has a tendency to significantly deform the native vessels specifically around sharp bends in the iliac resulting in reduced tortuosity.

Keywords: Vessel Deformation, Implantable Medical Device, Guidewire

1. Introduction

Aneurysm (Figure 1) is a disease in which the aorta swells abnormally which could lead to rupture of aorta if not treated in a timely manner. Endovascular Aneurysm Repair (EVAR) is a procedure to treat aneurysm in which a delivery device catheter is inserted through femoral iliac artery and a stent-graft is implanted at the aneurysm site (Figure 1). The delivery catheter tends to be very stiff as compared to the human vessel. So, in order to facilitate smooth insertion of delivery catheter through the vessel and to prevent any damage in the vessel due to direct insertion of delivery catheter, a stiff guidewire is inserted first into the vessel. The delivery catheter is then loaded on to the guidewire and pushed in to access the aneurysm site. The guidewire, which is typically made up of 0.035” steel wire, is stiff enough to prevent any kinking when the delivery device is pushed through sharp bends in iliac. Due to its stiffness, the guidewire tends to straighten the iliac thus making it less tortuous for the delivery catheter. Since the performance of delivery catheter depends on the tortuosity of iliac, and the tortuosity depends on the native vessel geometry as well as the straightening due to guidewire, it is important to first understand the interaction between the guidewire and iliac and capture the deformation so that further improvements in delivery catheters can be done to make it easier for physicians to deliver the devices.

Recent advancements in Finite Element Analysis (FEA) techniques have enabled the users to simulate highly complex contact conditions, anisotropic hyperelastic material modeling etc. Therefore, in this study, we used Abaqus (Dassault Systèmes, Simulia Corp., Providence, RI) to investigate the interaction between guidewire and human vessel. Both Abaqus/Standard (dynamic implicit) and Abaqus/Explicit (dynamic explicit) solvers were used separately to model this interaction. The deformed shapes were then compared between these models to select the most suitable technique which could be used to investigate the guidewire interaction for different native vessel geometries.

Figure 1: Thoracic Aneurysm with Talent™ Thoracic stent graft (left) and Abdominal Aneurysm with Talent™ Abdominal stent graft (right).

2. Methods

Pre-operative CT scans were used to generate 3-D reconstructions of the iliac artery region (Figure 2) using Mimics (Materialise, Ann Arbor, MI). Then the iliac surfaces were converted to triangular element mesh in Mimics and exported as Abaqus input file containing the surface mesh (Figure 3). The input file was then imported into Abaqus/CAE. A finite element model was set up to simulate the vessel deformation when a guidewire is introduced from the distal end of the iliac and pushed up to the iliac bifurcation region keeping both ends of the iliac fixed (Figure 3). The vessel was modeled with approximately 26,000 triangular shell elements (S3R) with a shell thickness of one mm and the guidewire was modeled with 150 beam elements (B31). At the beginning of the Abaqus/Explicit analysis, the guidewire was placed concentric to an external rigid guide sheath in order to prevent the guidewire from buckling before it enters the vessel. The rigid guide sheath was held fixed during the entire simulation. In Abaqus/Standard, USDFLD was used to change the stiffness of the guidewire at the iliac entry point from a high value to actual guidewire stiffness to prevent any buckling before entering the vessel.

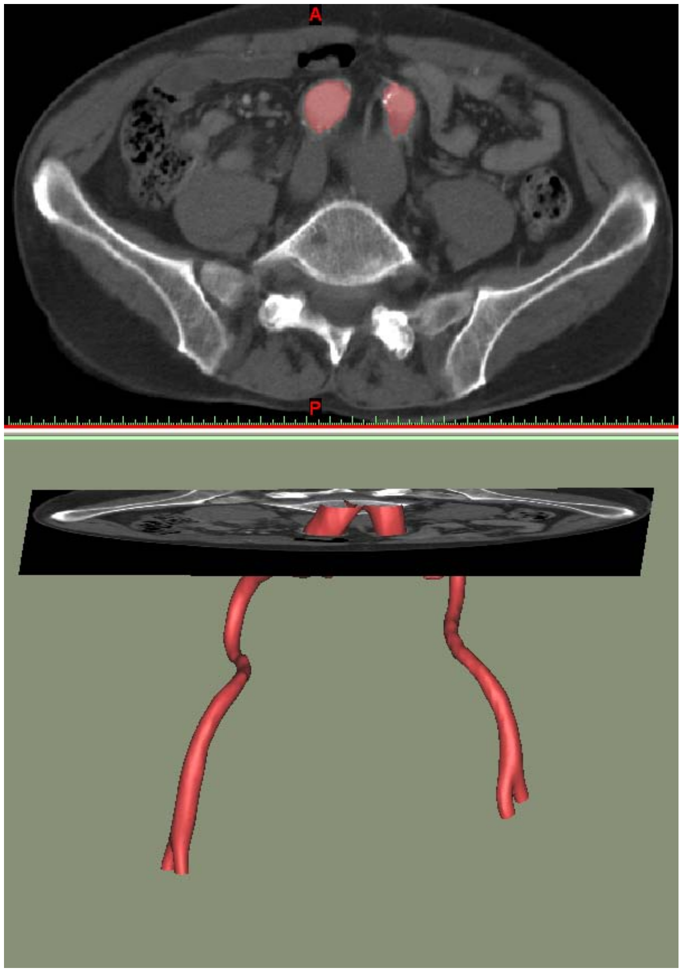


Figure 2: Three-dimensional reconstruction of iliac region (bottom) using CT scans (top) and Mimics.

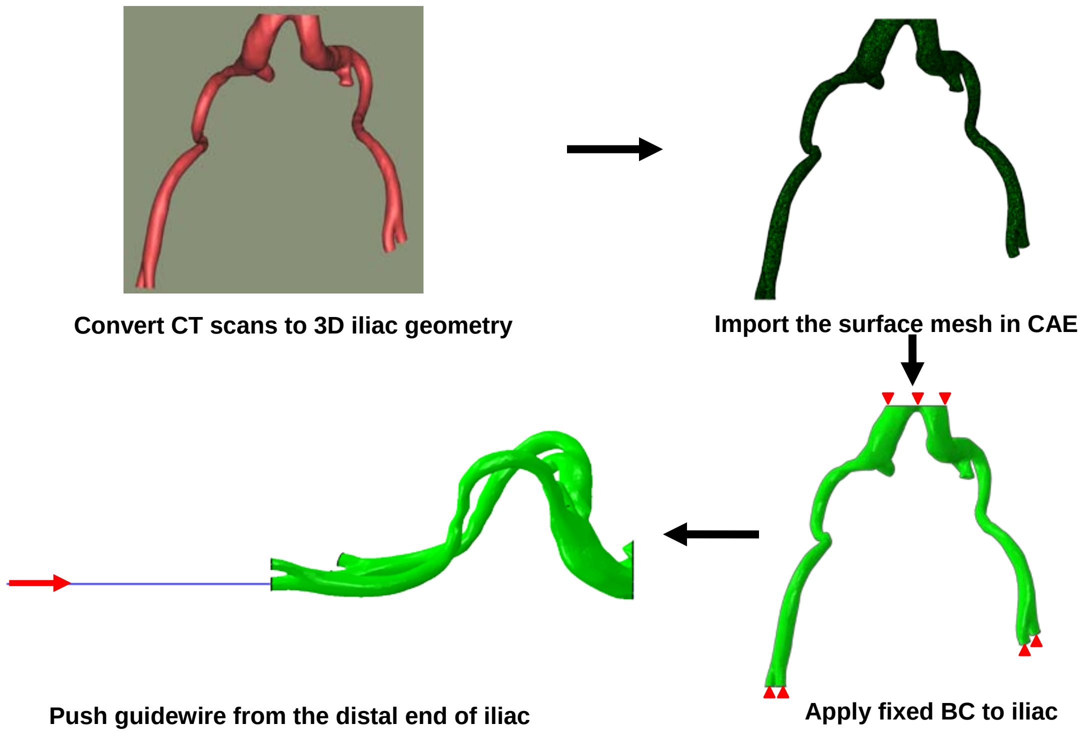


Figure 3: Different stages of FEA model generation.

The guidewire in this model had a circular cross-section with a diameter of 0.035”. Its properties were determined from the tensile and three-point bend tests. The material properties were found to be similar to high strength steel with Young’s Modulus of 200 GPa and Yield Stress of 500 MPa. Poisson’s ration of 0.3 was selected.

Vessel was modeled with Fung’s anisotropic strain energy function with material parameters from published literature (Vorp, 2007) for Abdominal Aortic Aneurysm (AAA) tissue. In Vorp, 2007, the material properties of native vessel were determined in its completely unloaded state i.e. without the effects of internal blood pressure. Therefore, a separate simulation was run on a straight vessel with the anisotropic hyperelastic material property to understand the effect of application of blood pressure to internal walls of the idealized vessel. It was observed that the vessel diameter changes by approximately 15% when the pressure was increased from 0 mmHg to 100mmHg (Figure 4). This resulted in significant stiffening of vessel because of its hyperelastic behavior. Since, this stiffening effect would be important for the iliac-guidewire interaction model it was considered necessary to apply an internal pressure of 100mmHg as the first step of the guidewire insertion analysis. But with anisotropic hyperelastic material properties, the vessel dilated by 15-20% due to pressure application causing the native geometry to change from what was obtained from CT scans. So, in order to simplify the model, the vessel material properties were determined from the idealized vessel model after application of 100mmHg internal pressure on the vessel walls (Figure 4). This resulted in stiffened vessel Young’s modulus of 7 MPa which was used as an isotropic modulus for the vessel-guidewire interaction model. A Poisson's ratio of 0.4 was selected.

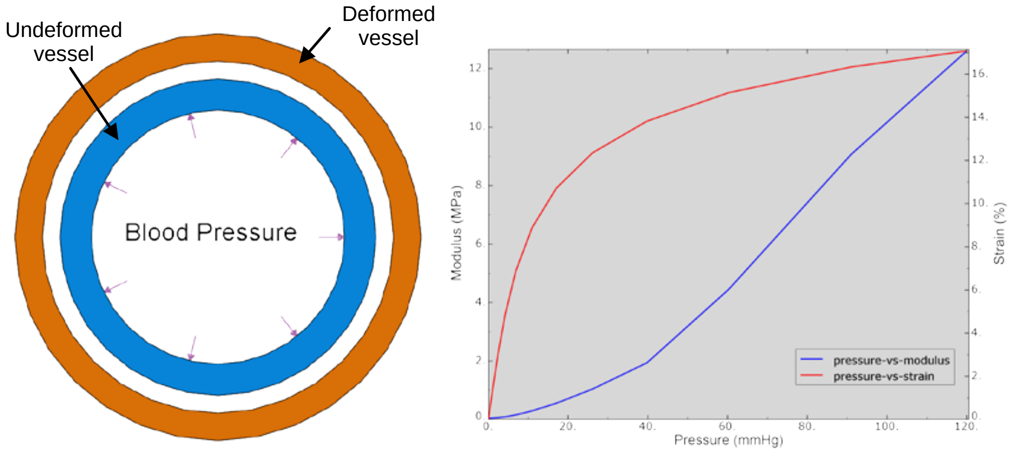


Figure 4: Vessel dilation due to internal blood pressure (left) and change in Young's Modulus and tangential strains with the increase in blood pressure (right).

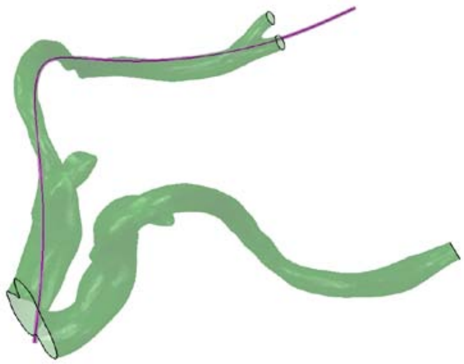
Separate analyses were run with both Abaqus/Standard and Abaqus/Explicit and general contact was used in both for all the contact interactions. Two steps were used in each of these analyses where in the fist step, an internal pressure of 100mmHg was applied to the internal walls of the vessel and in the second step the guidewire was pushed by 400 mm from its distal end along its axis.

In the Abaqus/Explicit analysis, the dynamic, explicit procedure was used and an external viscous pressure was applied to the vessel walls to reduce dynamic effects. Mass scaling was also applied to scale the minimum time increment to 2e-5 in the first step and 1e-6 in the second step. In addition, the guidewire push velocity was scaled up from approximately 10-50 mm/s to 1000 mm/s to reduce the analysis runtime.

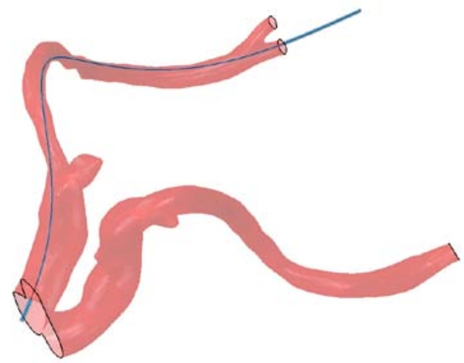
In the Abaqus/Standard analysis, the dynamic, implicit procedure with a newly added capability to perform quasi-static analyses (*DYNAMIC, APPLICATION=QUASI-STATIC) was used to obtain what is essentially a static response. The quasi-static method by default employs a backward Euler time integration method to ensure a quasi-static response as well as good convergence in problems where significant contact is expected. The guidewire push velocity was set to 75 mm/s. Performance gains (mainly contact calculations) were obtained by replacing the guidewire sheath with position dependant stiffness for the guidewire. Using the subroutine USDFLD, the guidewire stiffness and densities were adjusted so it remains very stiff in the region outside the iliac. Once inside the iliac, the guidewire assumed its normal behavior. No instabilities or convergence issues were observed due to this transition. Other techniques employed to further improve solution efficiency such as introducing material damping, switching contact pair definitions to make the guidewire head a slave surface resulted in additional performance enhancements. Since, the element loop operations for the new quasi-static procedure were not parallelized in Abaqus 6.9-EF, beta pre-release of Abaqus 6.10 was used with element loop parallelization for improved performance.

3. Results

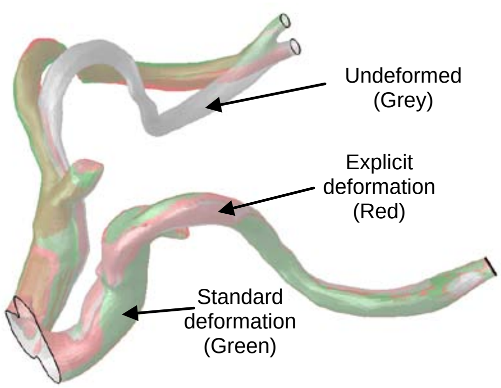
From both Abaqus/Standard and Abaqus/Explicit analyses, it was observed that the guidewire insertion reduced the tortuosity of iliac (Figure 5) specifically around the sharp corners thus making the delivery catheter path less tortuous. A successful run using both the techniques showed that continuous change in contact and significant geometric deformations can be successfully captured in implicit and explicit dynamic procedures of Abaqus. In addition, a comparison of deformed geometries resulting from both analyses showed that there were negligible differences in the deformed shapes of the iliac (Figure 5C). The guidewire deformations were also very similar (Figure 5D). The difference in guidewire deformation at the free end could be attributed to mass scaling in Abaqus/Explicit which resulted in increased inertia of guidewire due to small element size. Also, it was observed that the guidewire deformed beyond its elastic limit causing permanent deformation. Abaqus/Standard solution took 8 hours and Abaqus/Explicit solution took 2 hours to complete on 4 CPUs each of a Linux X86-64 bit cluster.



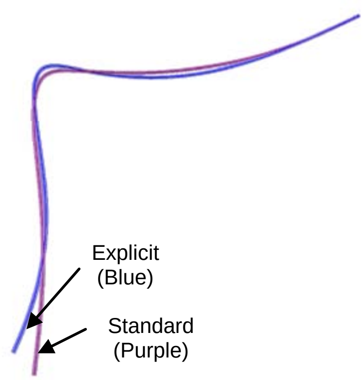
A) Deformed shape Abaqus/Standard



B) Deformed shape Abaqus/Explicit



C) Comparison of deformed vessel



D) Comparison of deformed guidewire

Figure 5: Deformed shapes A and B and their comparison using overlay plots C and D. Figure C compares iliac deformation and Figure D compares guidewire deformation from Abaqus/Standard and Abaqus/Explicit.

4. Discussion

In this study, the effect of guidewire insertion in tortuous iliac was investigated and results from both Abaqus/Standard and Abaqus/Explicit were compared. Both analyses showed that significant straightening occurs when guidewire is inserted into the vessel. So, in order to evaluate the delivery catheter performance in more realistic environment, guidewire straightening should be considered in the hard bench-top models of iliac used to test the delivery devices during their design process.

Also, successful run with both techniques demonstrate robust performance of Abaqus for such complicated models. The Abaqus/Explicit analysis showed some effects of mass scaling on the guidewire deformation which could be reduced by lower mass-scaling. In addition, dynamic, implicit, quasi-static procedure in Abaqus/Standard proved to be a good alternative to running these types of simulations in Abaqus/Explicit which are usually quasi-static but involve complex contact conditions and large geometric deformations. Since the results from both Abaqus/Standard and Abaqus/Explicit were very similar and the analysis run time for the Abaqus/Explicit was less, it can be concluded that the analysis with guidewire and vessel interaction can be run faster in Abaqus/Explicit with the help of appropriate mass-scaling, viscous pressure and increasing the speed of the simulation.

It should be noted that several components of this model may need further investigation to validate this approach against in-vivo deformation of iliac. This includes, more realistic vessel property assignment, effect of surrounding muscle tethering etc.

Next step of this analysis could be modeling the delivery system insertion on the guidewire and performing virtual deployment in a tortuous anatomy to better understand the delivery system and implanted device performance.

5. Conclusion

This study showed that the insertion of a guidewire reduces the tortuosity of the iliacs and both Abaqus/Standard and Abaqus/Explicit are well suited to run such analyses. The deformed iliac geometry obtained from FEA can be used to generate simulated flow models for more realistic evaluation of acute delivery system performance. This approach can be further utilized to study the interactions of the iliac with the delivery system and to design a delivery system suitable for tortuous anatomy.

6. References

1. Vorp, D. A., “Biomechanics of Abdominal Aortic Aneurysm,” Journal of Biomechanics, vol. 40, pp. 1887-1902, 2007.