World Academy of Science, Engineering and Technology
International Journal of Biomedical and Biological Engineering
Vol:12, No:11, 2018

Finite Element Analysis of the Anaconda Device: Efficiently Predicting the Location and Shape of a Deployed Stent

Authors : Faidon Kyriakou, William Dempster, David Nash

Abstract : Abdominal Aortic Aneurysm (AAA) is a major life-threatening pathology for which modern approaches reduce the need for open surgery through the use of stenting. The success of stenting though is sometimes jeopardized by the final position of the stent graft inside the human artery which may result in migration, endoleaks or blood flow occlusion. Herein, a finite element (FE) model of the commercial medical device AnacondaTM (Vascutek, Terumo) has been developed and validated in order to create a numerical tool able to provide useful clinical insight before the surgical procedure takes place. The AnacondaTM device consists of a series of NiTi rings sewn onto woven polyester fabric, a structure that despite its column stiffness is flexible enough to be used in very tortuous geometries. For the purposes of this study, a FE model of the device was built in Abaqus® (version 6.13-2) with the combination of beam, shell and surface elements; the choice of these building blocks was made to keep the computational cost to a minimum. The validation of the numerical model was performed by comparing the deployed position of a full stent graft device inside a constructed AAA with a duplicate set-up in Abaqus®. Specifically, an AAA geometry was built in CAD software and included regions of both high and low tortuosity. Subsequently, the CAD model was 3D printed into a transparent aneurysm, and a stent was deployed in the lab following the steps of the clinical procedure. Images on the frontal and sagittal planes of the experiment allowed the comparison with the results of the numerical model. By overlapping the experimental and computational images, the mean and maximum distances between the rings of the two models were measured in the longitudinal, and the transverse direction and, a 5mm upper bound was set as a limit commonly used by clinicians when working with simulations. The two models showed very good agreement of their spatial positioning, especially in the less tortuous regions. As a result, and despite the inherent uncertainties of a surgical procedure, the FE model allows confidence that the final position of the stent graft, when deployed in vivo, can also be predicted with significant accuracy. Moreover, the numerical model run in just a few hours, an encouraging result for applications in the clinical routine. In conclusion, the efficient modelling of a complicated structure which combines thin scaffolding and fabric has been demonstrated to be feasible. Furthermore, the prediction capabilities of the location of each stent ring, as well as the global shape of the graft, has been shown. This can allow surgeons to better plan their procedures and medical device manufacturers to optimize their designs. The current model can further be used as a starting point for patient specific CFD analysis.

Keywords : AAA, efficiency, finite element analysis, stent deployment
Conference Title : ICBEMD 2018 : International Conference on Biomedical Engineering and Medical Devices
Conference Location : Tokyo, Japan
Conference Dates : November 12-13, 2018