2nd International Conference on Mathematical and Computational Biomedical Engineering - CMBE2011
March 30 - April 1, 2011, Washington D.C, USA
P. Nithiarasu and R. Löhner (Eds.)

Patient-specific Finite Element Analysis of Carotid Artery Stenting: Impact of constitutive vessel modeling on vessel wall stress distribution

F. Auricchio[⋆], **M. Conti**[⋆◇], **A. Ferrara**[⋆], **S. Morganti**[⋆], **A. Reali**[⋆]
[⋆]Structural Mechanics Department, Università degli Studi di Pavia, Via Ferrata 1, 27100 Pavia, Italy
[◇]michele.conti@unipv.it

Key Words: *Finite Element Analysis (FEA), Carotid Artery Stenting (CAS), Patient-specific modeling.*

1 Introduction

Carotid artery stenting (CAS) is emerged as a safe and cost-effective treatment of carotid artery (CA) stenosis [1]. However the long-term efficacy of CAS is under clinical evaluation and, in particular, the role of in-stent restenosis (ISR) is not clear. More data and dedicated studies are necessary to elucidate the mechanisms of ISR and their relation with the novel carotid stent designs. Experimental evidence shows that ISR is related to vessel wall injury and numerical simulations can play a key role supporting the assessment of the vessel wall distribution after stent implant. In this scenario, the modeling of the vessel wall mechanics is an important issue which can influence the stress calculation. In this study, we evaluate the impact of constitutive vessel modeling on the vessel wall stress distribution computed through patient-specific finite element analysis (FEA).

2 Materials and Methods

2.1 CA Model

We base the CA model on DICOM images of a neck-head Computed Tomography-Angiography (CTA) performed on 70 years-old female patient. The CTA scan is performed at IRCCS San Matteo in Pavia, Italy, using a Somatom Sensation Dual Energy scanner (Siemens Medical Solutions, Forchheim, Germany). We elaborate the images using OsiriX (see figure 1-a); we focus on the left CA highlighting both the bifurcation lumen and the calcific plaque as shown in figure 1-b. We then export both lumen and plaque geometry as stereolithography (STL) file format. To create the CA finite element mesh, we follow a procedure, implemented by Matlab (Matlab, Mathworks Inc., Natick, MA) and characterized by three main steps: i) definition of vessel wall inner profile elaborating the lumen surface derived from the DICOM images; ii) definition of vessel wall outer profile enlarging, in an appropriate way, the inner profile; iii) generation of the mesh between the inner and outer profile; iv) inclusion of the plaque in the vessel wall model. In particular, through a dedicated subroutine implemented by Matlab, we create the plaque model by detecting the elements of the vessel wall mesh enclosed in the 3D surface of the plaque geometry obtained from the CTA.

2.2 Stent model

We consider a open-cell stent creating the corresponding finite element model from an high resolution micro-CT scan. The stent has a straight configuration with diameter of 9 mm and strut thickness 0.190 mm.

2.3 FEA of CAS

To investigate the interaction between the stent and the patient specific carotid artery model, we perform a two-step simulation procedure [2] where the stent deformation is driven by the change of configuration of the catheter. The simulation is performed using Abaqus/explicit (Dassault Systémes Simulia Corp., Providence, RI, USA) as finite element solver since the numerical analysis is characterised by non-linearity due to the material properties, large deformations and complex contact problems.

2.4 Constitutive modeling

To model the mechanical behaviour of the vessel wall, we choose two constitutive models, i.e. isotropic and anisotropic hyperlelastic. For the isotropic model, we use a second-order polynomial strain energy function (SEF):

$$U_I = \sum_{i+j=1}^{2} C_{ij}(\bar{I}_1 - 3)^i(\bar{I}_2 - 3)^j + \sum_{i=1}^{2} \frac{1}{D_i}(J^{el} - 1)^{2i} \tag{1}$$

where C_{ij} and D_i are material parameters; $\bar{I}_1$ and $\bar{I}_2$ are respectively the first and second deviatoric strain invariants; J^{el} is the elastic volume ratio.
For the anisotropic model, we use the SEF proposed by Holzapfel et al. [3] and Gasser et al. [4]:

$$U_A = C_{10}(\bar{I}_1 - 3) + \frac{1}{D}\left(\frac{(J^{el})^2 - 1}{2} - ln(J^{el})\right) + \frac{k_1}{2k_2}\sum_{\alpha=1}^{N}\left\{exp\left[k_2\left\langle \bar{E}_\alpha \right\rangle^2\right] - 1\right\} \tag{2}$$

where

$$\bar{E}_\alpha \equiv k(\bar{I}_1 - 3) + (1 - 3k)(\bar{I}_{4(\alpha\alpha)} - 1) \tag{3}$$

and C_{01},D,k_1,k_2 are material coefficients, while N is the number of families of fibers; in our case $N = 2$ as we assume two family of fibers; k represents the fiber dispersion, in our case we assume $k = 0$ which corresponds to a full anisotropic behaviour; $\bar{I}_1$ and $\bar{I}_{4(\alpha\alpha)}$ are invariants defined as specified in literature [3, 4]. Starting from the above described two SEFs, we consider the five model variants: 1) SEF U_I with the coefficients reported by Creane et al. [5] (model HI_1); ii) SEF U_I with the coefficients reported by Lally et al. [6] (model HI_2); iii) SEF U_A calibrated on data reported by Sommer et al. [7] for the intact wall (model HA_1). Both SEFs are already implemented in the material model library of Abaqus. In case of model HA_1, we implement in Matlab a procedure to define local coordinate system for each mesh element in order to assign fiber-orientation.

2.5 Post-processing

We evaluate the impact of stenting evaluating the von Mises stress distribution in the post-stenting vessel as a measure of potential injury induced by the stent apposition to the vessel wall. To neglect peak values of von Mises stress, due to local concentration, we consider the 99 percentile with respect to the pre-stenting vessel volume (i.e. only 1% of the volume has stress above this value).

3 Results

The result of the CAS simulation for vessel model HA_1 is reported in figure 2-a, while in figure 2-b the von Mises stress (99 percentile) for each model is reported. It is possible to note that isotropic models HI_1 and HI_2 provide similar results while the use anisotropic model HA_1 provides a lower stress value.

4 Conclusions

The system under investigation is very complex; to simplify the analysis we neglect axial pre-stretch, residual stresses and arterial blood pressure. Keeping the highlighted limitations in mind, we believe that the present study represents a further step towards a quantitative assessment of the relation between the complex mechanical features of a given stent design and a given patient-specific anatomy, which could be useful for both procedure standardization and stent design evaluation. Clearly, the numerical analysis should be validated and integrated with clinical and biological considerations based also on the surgeon experience which plays a primary role for the optimal CAS outcomes.

5 Acknowledgments

This work is funded by PhD grant from MIUR (Ministero dell'Istruzione, dell'Università e della Ricerca, Italian Ministry of Education, University and Research) and Cariplo Foundation through the Project no. 2009.2822. The authors would acknowledge: Dr. R. Dore, Prof. A. Odero and Dr. S. Pirrelli of IRCCS Policlinico S. Matteo, Pavia, Italy for support on medical aspects related to the present work; Eng. D. Van Loo of Ghent University (UGCT), Ghent, Belgium for micro-CT scanning of stent samples and Eng. C. Ferrazzano (Unipv) for medical image elaboration.

References

[1] CREST committee, Stenting versus endarterectomy for treatment of carotid-artery stenosis, New England Journal of Medicine 363 (2010) 11–24.

[2] F. Auricchio, M. Conti, M. De Beule, G. De Santis, B. Verhegghe, Carotid artery stenting simulation: from patient-specific images to finite element analysis, Medical Engineering & Physics, in press. DOI: 10.1016/j.medengphy.2010.10.011.

[3] G. Holzapfel, T. Gasser, R. Ogden, A new constitutive framework for arterial wall mechanics and a comparative study of material models, J Elast 61 (2000) 1–48.

[4] T. Gasser, R. Ogden, G. Holzapfel, Hyperelastic modelling of arterial layers with distributed collagen fibre orientations, Journal of the Royal Society Interface 3 (2006) 15–35.

[5] A. Creane, E. Maher, S. Sultan, N. Hynes, D. Kelly, C. Lally, Finite element modelling of diseased carotid bifurcations generated from in vivo computerised tomographic angiography, Computers in Biology and Medicine 40 (2010) 419–429.

[6] C. Lally, F. Dolan, P. Prendergast, Cardiovascular Stent Design and Vessel Stresses: a Finite Element Analysis, Journal of Biomechanics 38 (2005) 1574–1581.

[7] G. Sommer, P. Regitnig, L. Koltringer, G. Holzapfel, Biaxial mechanical properties of intact and layer-dissected human carotid arteries at physiological and supraphysiological loadings, Am J Physiol Heart Circ Physiol 298 (2009) H898–912.

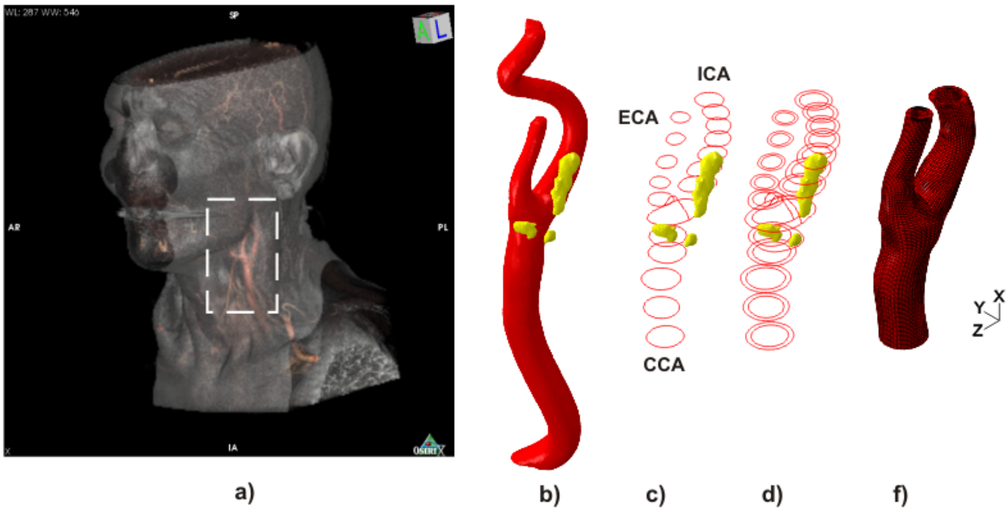

Figure 1: Elaboration of CTA DICOM images: a) whole 3D reconstruction of neck-head district highlighting the region of interest; b) 3D reconstruction of both lumen of left CA bifurcation and calcific plaque (depicted in white).

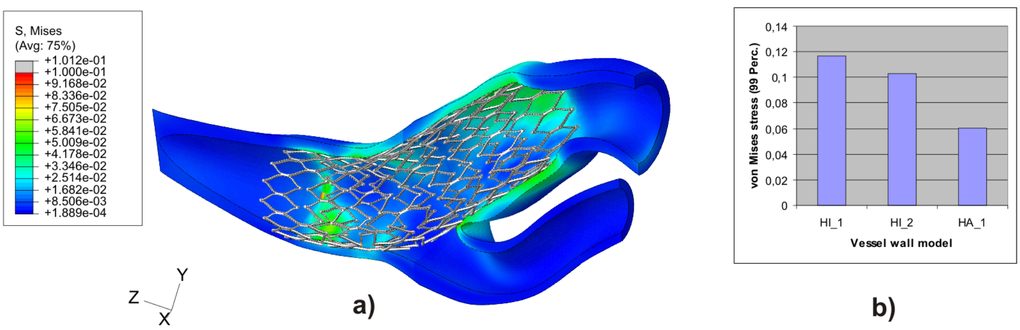

Figure 2: a) contour plot of the von Mises stress distribution in the CA wall after the stent apposition, the results refer to model HA_1; b) histograms depicting the maximum von Mises stress value (99 Perc.) as function of the vessel constitutive model. For both figures, MPa is the stress unit measure.