Towards Real-Time Interventional Simulation of Balloon Angioplasty and Stenting

Delorme, Sebastien; Laroche, Denis; Holzapfel, Gerhard; Stadler, Michael; Buithieu, Jean; DiRaddo, Robert

Towards Real-Time Interventional Simulation of Balloon Angioplasty and Stenting

Sebastien Delorme[1], Denis Laroche[1], Gerhard Holzapfel[2], Michael Stadler[2], Jean Buithieu[3], and Robert DiRaddo[1]
[1]Industrial Materials Institute (IMI), Boucherville, QC, Canada, sebastien.delorme@cnrc-nrc.gc.ca
[2]Graz University of Technology, Graz, Austria
[3]McGill University Health Center, Montreal, QC, Canada

INTRODUCTION

Percutaneous Transluminal Coronary Angioplasty (PTCA) consists of deploying a polymer balloon, with or without a metallic stent, at a controlled pressure until it contacts, deforms and unblocks a stenosed artery. PTCA induce vascular wall fractures or dissections, and achieves apparent luminal enlargement by a combination of plaque compression, plaque redistribution and vascular wall expansion, depending on the original composition of the atherosclerotic lesion.

The numerical prediction of luminal area and vascular wall injury for an angioplasty on specific high-risk patients could help select a balloon size, stent type and balloon inflation pressure that would minimize vascular injury and reduce the risk of clinical complications, such as restenosis. A finite element model has been developed to predict deformation and stresses in an homegeneous artery during balloon angioplasty (Laroche 2003, Delorme 2004). The goal of this study is to demonstrate the feasibility of the approach for stent implantation in a layer-specific artery.

METHODS

A finite element modeling software developed at IMI for the analysis of large deformations of soft materials was improved to solve angioplasty mechanics. One of the most important improvement to the software was the capability to properly model the contact and slip between deformable bodies using a slave-node/master-surface technique.

An artery geometrical model (Holzapfel 2002) was obtained from high-resolution magnetic resonance imaging of a human artery. Seven distinct layers were traced on the scans, correlated with histological analysis, and meshed in 3D with 8-node hexahedrons (Figure 1). A stent similar to a commercial type and shortened to fit the artery was modeled in 3D and meshed with 8-node hexahedrons.

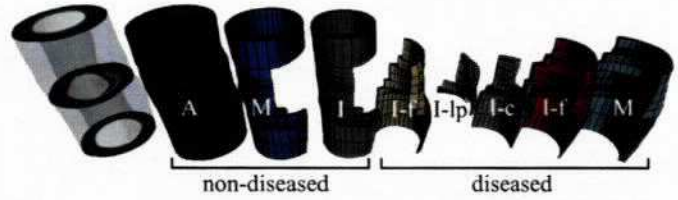

Figure 1: Exploded view of the multi-layer geometrical model of an artery. A: adventita; M: media; I: intima; f: fibrous; lp: lipid pool; c: calcification.

The stent and artery materials were considered incompressible and were modeled as hyperelastic materials with the two-parameter *Mooney-Rivlin* constitutive equation. The Mooney-Rivlin parameter values for the stent, the artery and the plaque components were calculated from published shear moduli (Holzapfel 2002).

A gradually increasing pressure was applied on the inner surface of the stent. The nodes at the bottom and top of the artery were fixed.

RESULTS AND CONCLUSION

The deformed meshes at the end of simulation of stent deployment are represented in Figures 2 and 3. Stretch ratios up to 1.6 and stresses higher than failure limits were predicted in some regions.

The presented finite element approach is capable of predicting artery deformations and stresses during stent implantation. Further work in underway to combine folded balloon, stent and artery in one single simulation.

REFERENCES

Delorme, S. et al (2004) *Annual Technical Conference (ANTEC) of the Society of Plastic Engineers*, Chicago, IL.
Holzapfel, G.A. et al. (2002) *J Biomed Eng*, **30**, 753-767.
Laroche, D. et al (2003) *ASM Materials & Processes for Medical Devices Conference*, Anaheim, CA.

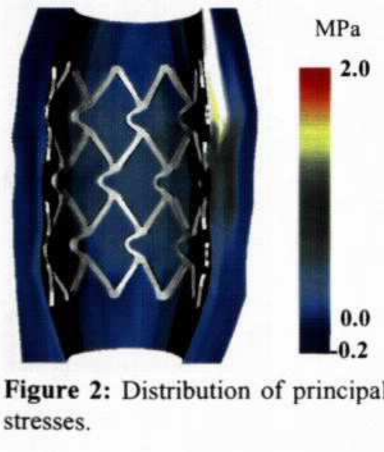

Figure 2: Distribution of principal stresses.

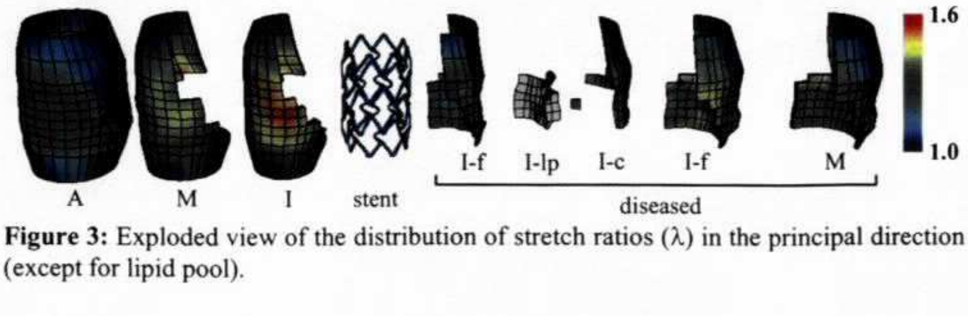

Figure 3: Exploded view of the distribution of stretch ratios (λ) in the principal direction (except for lipid pool).