7th International Congress of Serbian Society of Mechanics
Sremski Karlovci, Serbia, June 24-26, 2019

FEM ANALYSIS OF CORONARY STENT DEPLOYMENT

Nevena A. Aranđelović[1], Buljak V. Vladimir[1]

[1] Faculty of Mechanical Engineering,
The University of Belgrade, Kraljice Marije 16, 11120 Belgrade 35
e-mail: narandjelovic@mas.bg.ac.rs, vbuljak@mas.bg.ac.rs

Abstract:

As stent technology moves forward, thereôs a growing need for fast and accurate stent deployment visualizations. To that end, the Finite Element Method was used for quasi-static analysis of a balloon expandable stent. A model was created, which includes a stent, balloon, artery and a plaque. In order to achieve faster computation, various approximations have been included, especially considering the balloon, artery and plaque. Some non-linear problems are assessed, like the contact between different parts of the model. Arterial tissue and plaque are modeled as isotropic hyperelastic materials. The balloon was specially designed and discretized to avoid the computational difficulties during inflation. Thanks to the complexity of this model, useful results can be obtained in a reasonable time frame. The software used for FEM analysis was ABAQUS.

Key words: stent, FEM, ABAQUS, mechanical analysis, angioplasty, plasticity, contact problems

1. Introduction

Coronary stents have a major role in treating atherosclerosis, a common disease which leads to narrowing and blocking of various blood vessels. This occurs due to the formation of a plaque in the arterial wall. A plaque is a hard bump consisted of cholesterol, dead white cells and minerals. As a result, the blood flow is restricted, which puts a great burden on the heart. If the plaque ruptures, blood cloths are formed inside the damaged artery and may be carried off to the brain or heart, which often leads to a stroke, or a heart attack.

The preferred treatment is angioplasty, a very non-invasive procedure thatôs proved to be a much better option in comparison to the traditional bypass surgery. A catheter carrying a stent is navigated through the blood system, so that the stent can be positioned and deployed in the narrowed artery. During the deployment the stent is expanded radially, so that it comes in contact with the plaque, and creates a pressure which makes the artery wider.

Introducing a foreign object inside the human body is very complicated in itself, and many problems may occur. This leads to various restrictions which have to be overcome during the design process. The dimensions, proportions and geometry of the stent are ñtunedò to prevent accidental damaging of healthy tissue during the medical procedure. Every change requires new calculations. Thus the importance of creating such models, which can give adequate results in a short period of time. In this way, not only time, but valuable investments and equipment are spared. The idea is for the actual experiment to be the last resource, which is used for the final validation.

The created numerical model includes a stent, balloon, artery and a plaque. The geometry of the stent is what, above all, makes one stent different from the other. Hence, this information is protected as intellectual property and is not available. The geometry created for this model has all of the general characteristics of stents, but itôs simplified and regular. The artery and plaque have an idealized form, since their purpose in the model is only to present a certain obstacle during the expansion of the stent.

Special attention was dedicated to the geometry of the balloon. In reality, the balloon is folded on the catheter in a very precise way, in order to achieve uniform and gradual inflation. The folded configuration of the balloon is also a manufacturerôs secret, so a simplified configuration was created.

FEM was used to calculate stresses and deformations, as well as contact pressures. This resulted in a frame sequence which provides a good visualization of stent deployment.

2. Model

The parts and the assembly were made in ABAQUS and SOLIDWORKS. The approximate dimensions and guidelines, as well as material properties can be found in [1], [5] and [8]. The assembly is shown in Fig. 1, while the geometry of the stent can be seen in detail in Fig. 2. Only a quarter of the model is used because the entire model is symmetrical. The stent is supposed to be made of stainless steel 316L, which is commonly used in manufacturing of stents. The artery is partitioned into 3 layers, with different material properties as suggested in [5]. The material of the artery and plaque is considered to be isotropic hyperelastic, with specific properties given in [5].

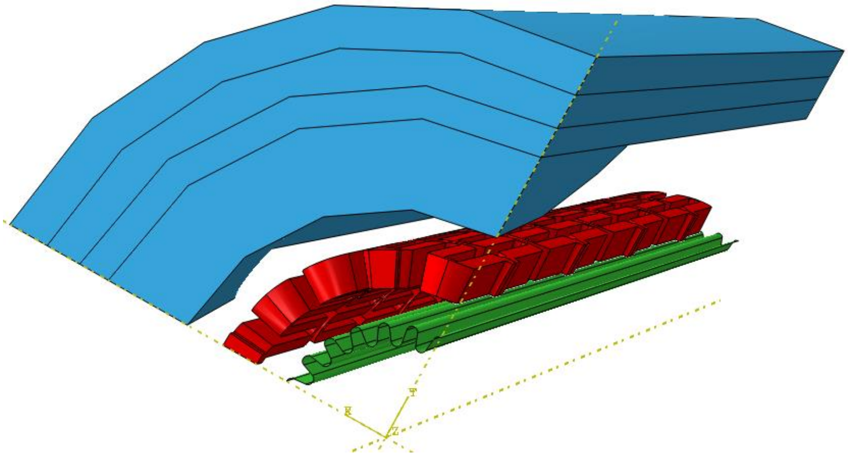

Fig. 1. Assembly, a quarter of the model

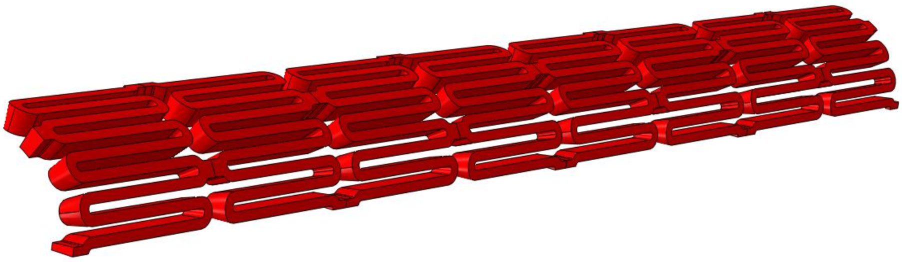

Fig. 2. A symmetrical quarter of the stent

The folded configuration of the balloon is designed as shown in Fig. 3. The material itôs made of is a type of nylon, so itôs obvious that, in reality, it would not be possible for the balloon to retain this configuration around the catheter. However, this is not an issue, since the only purpose it has in this model is to convey and evenly distribute the load on the stent. The only difference in the results will be the absence of ñdog boningò ï the occurrence of initial disproportional widening of the ends of the stent, which can result in damaging of the arterial wall. The nylon is supposed to have linear elastic and isotropic properties.

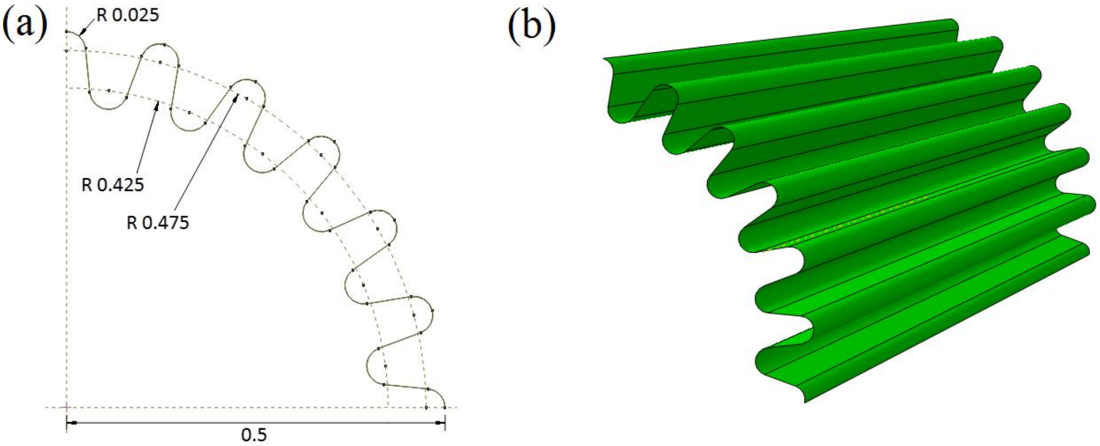

Fig. 3. Cross section of a quarter of the balloon (a) and a detailed view of the 3D model (b)

3. Calculation parameters

The stent was discretized using C3D8 brick elements, which are recommended for strain and stress analysis in such cases when non-linearity occurs [13]. A finer mesh is defined through the thickness of the stent. Linear geometric order of elements is chosen to avoid volumetric locking thanks to selective reduced integration [13]. The created mesh is consisted of 101710 elements.

The balloon has in-plain stiffness only, which causes it to act as a membrane. Despite this fact, shell elements were used. This was done to avoid the snap-through effect which would occur due to the specific geometry of the balloon coupled with membraneôs zero bending stiffness. 51840 S4 elements with linear geometric order were created.

The artery and plaque were discretized using C3D8H brick elements with linear geometric order and hybrid formulation. This is recommended [13] because of the hyperelastic properties of the material. The mesh consists of 21032 elements.

Contact was defined between the balloon and the stent, as well as between the stent and the artery and plaque. In order to do so, *Surface to surface* discretization was performed between:

- the inner surface of the stent and the outer surface of the balloon and
- the outer surface of the stent and the inner surface of the artery and plaque.

Recommendations [13] were followed while defining the properties of these contacts. *Finite sliding* was chosen in the case of the first pair, and *Small sliding* in the second. The surface of the stent was selected as the *Slave surface* in both contacts. Tangential behavior between the contact surfaces was defined as recommended in [9]. Normal behavior was defined as exponential *Pressure-Overclosure*.

The symmetry boundary conditions were used to account for the absence of three quarters of the model. Uniform pressure was applied to the inner surface of the balloon. A multiple-step analysis was performed, in order to remove this pressure when the original radius of the artery is restored.

4. Results

The analysis was performed on a PC with the maximum CPU speed of 2.5 GHz and 8 GB of RAM. The computation lasted around 8 hours. The obtained results are in accordance with the ones in existing publications [4], [6], [9] and [12].

The deformation sequence can be seen on Fig. 4 and Fig. 5. As the balloon inflates, it comes into contact with the stent and causes its expansion (Fig. 4.b and Fig. 5.b). This leads to the contact of the stent and the plaque which is pushed in a radial direction, and the previous radius of the artery is restored (Fig. 4.c and Fig. 5.c). Due to large deformations of the stent, it undergoes plastic deformation. As the balloon deflates, elastic deformation of the stent is removed, which results in a small relapse of the radius, both of the stent and of the artery (Fig. 4.d and Fig. 5.d). Thanks to the plastic deformation of the stent, the blood flow is restored through the damaged artery as the circulatory path remains open.

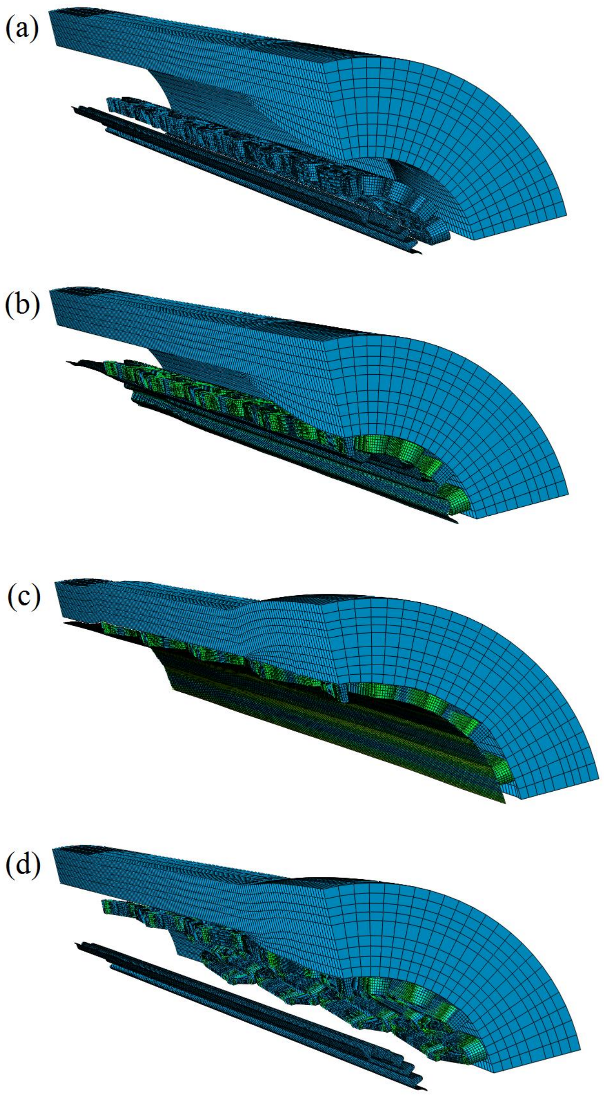

Fig. 4. Deformation sequence, side view

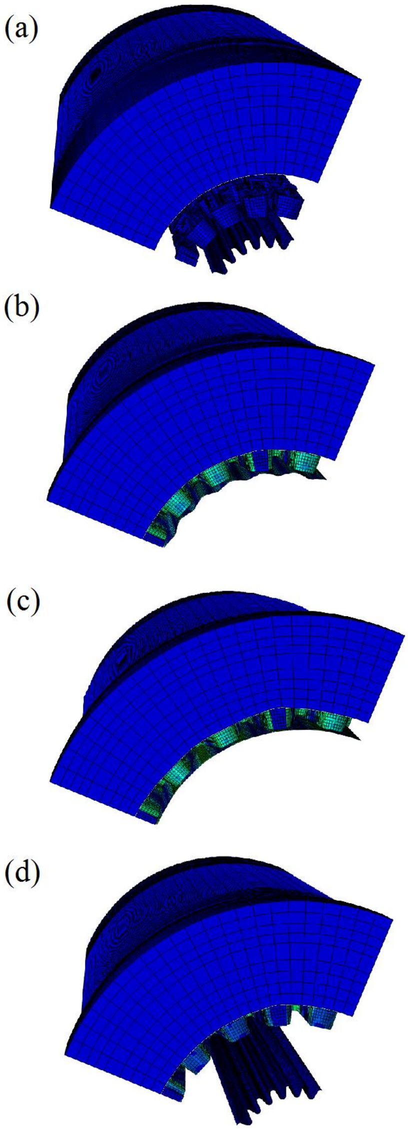

Fig. 5. Deformation sequence, front view

The sequential deformation of the stent itself can be seen on Fig. 6. The wire-like structure of the stent allows for its characteristic expansion and contraction. In Fig. 6.b the stent is at its maximum radius (the balloon is completely inflated). After the deflation of the balloon and the removal of elastic deformation, the residual plastic deformation can be seen (Fig. 6.c).

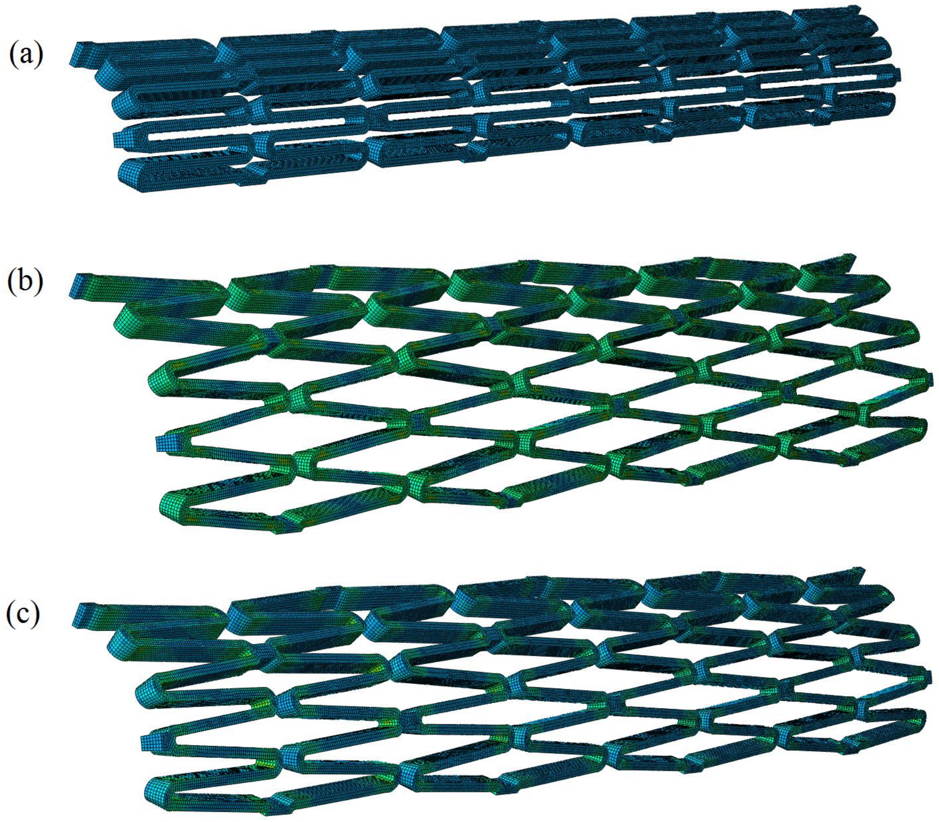

Fig. 6. Deformation sequence of the stent

Contact pressure to the inner surface of the artery and plaque is presented in Fig. 7.

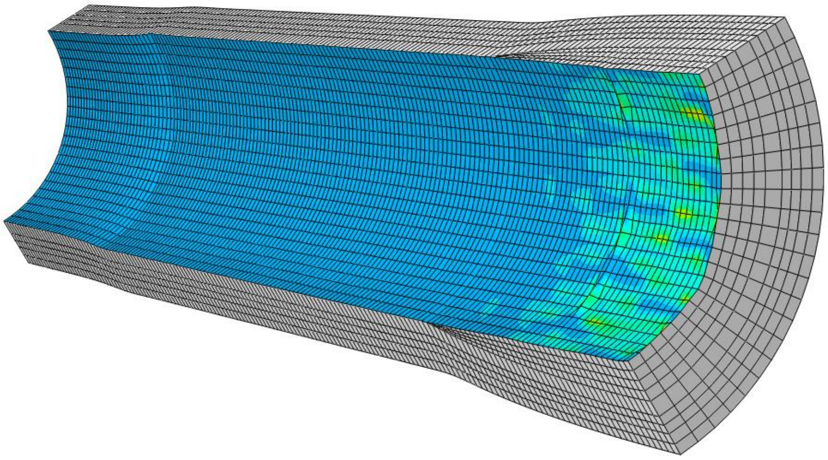

Fig. 7. Contact pressure visualization

5. Conclusions

The described multi-body and multi-step FEM analysis of stress, deformation and contact pressure, has a relatively small computation period and yields some very useful results. This model provides a good visualization of stent deployment, even though many approximations were included. It can be assumed that, if a realistic geometry of a stent was present, the results would have high resemblance to the true process of deformation.

However, if the true configuration of the folded balloon was introduced, the model would advance in terms of being able to result in some of the finer details of deformation, such as the dog boning phenomenon. Other materials used for manufacturing of stents, such as shape memory alloys, should also be considered in further work.

6. Acknowledgements

This research was supported under Grants Nos. ON174001 and TR35006 by the Ministry of Education, Science and Technological Development of the Republic of Serbia. This support is gratefully acknowledged.

References

[1] Bosignore, C., *Open Stent Design,* 2012, doi: 10.6084/M9.FIGSHARE.95614

[2] Chua, S., N., D., Mac Donald, B., J., Hashmi, M., S., J., *Finite element simulation of stent and balloon interaction,* Journal of Materials Processing Technology, Vol. 143-144, 591-597, 2003.

[3] Gasser, T., C., Holzapfel, G., A., Stadler , M., *Changes in the mechanical environment of stenotic arteries during interaction with stents: Computational assessment of parametric stent designs*. Journal of Biomechanical Engineering , Vol. 1(127), 166-180, 2005.

[4] Hao, P., Enoki , M., Sakurai, K., *Finite element analysis of tensile fatigue behavior of coronary stent*, Materials Transactions, Vol. 53(5), 959-962, 2012.

[5] Holzapfel, G., Sommer, G., Gasser, C., Regitnig, P., *Determination of layer-specific mechanical properties of human coronary arteries with nonartherosclerotic intimal thickening and related constitutive modeling*, American Journal of Physiology - Heart and Circulatory Physiology, Vol. 289(5), H2048-58, 2005.

[6] Liang, D., K., Yang, D., Z., Qi, M., Wang, W., Q., *Finite element analysis of the implantation of a balloon-expandable stent in a stenosed artery*, International Journal of Cardiology, Vol. 104(3), 314-318, 2005.

[7] Liu, G., R., Quek, S., S., *The Finite Element Method: A Practical Course*, Elsevier Scientific, Linacre House, Jordan Hill, Oxford OX2 8DP, 2003.

[8] Loree, H., Grodzinsky, A., Park, S., Gibson, L., Lee, R., *Static circumferential tangential modulus of human arteriosclerotic tissue*, Journal of Biomechanics, Vol. 27, 195-204, 1994.

[9] Martin, D., Boyle, F., *Finite element analysis of balloon-expandable coronary stents: Influence of stent and vessel deformation*, International Journal for Numerical Methods in Biomedical Engineering, Vol. 29, 1161-1175, 2013.

[10] Migliavacca, F., Petrini, L., Colombo, M., Auricchio, F., Pietrabissa, R., *Mechanical behavior of coronary stents investigated through the finite element method*, Journal of Biomechanics, Vol. 35(6), 803-811, 2002.

[11] Mortier, P., De Beule, M., Carlier, S., G., Van Impe, R., Verhegghe, B., Verdonck, P., *Numerical study of the uniformity of balloon-expandable stent deployment*, Journal of Biomechanical Engineering, Vol. 130, 021018-021018-7, 2008.

[12] Schievano, S., Petrini, L., Migliavacca, F., Coats, L., Nordmeyer, J., Lurz, P., Khambadkone, S., Taylor, A., M., Dubini, G., Bonhoeffer, P., *Finite element analysis of stent*

deployment: Understanding stent fracture in percutaneous pulmonary valve implantation, Journal of Interventional Cardiology, Vol. 20(6), 546-554, 2007.

[13] Dassault Systèmes' SIMULIA. Getting started with abaqus. Accessed: 2018-06-10.