The cardiovascular diseases constitute one of the main causes of death worldwide. One of the main diseases is atherosclerosis, which causes narrowing of the arterial lumen. In order to solve this condition, several treatments have been developed; and Percutaneous Transluminal Coronary Angioplasty (PTCA) with the placement of stent have gained relevance. Many of these devices are currently coated to increase the biocompatibility and to decrease the restenosis risks. The biomechanical studies of the stent-coating interface behavior are necessary given the associate risks to possibility of failures or breakages of the coating during stent deployment. In this study the possible occurrence of coating delamination during stent deployment and the influence of parameters as the thickness and material were studied. The study starts by obtaining a geometric model of a stent unit of the Sirius Carbostent stent for the further processing by the Finite Element Method. The simulation was developed by applying restrictions so that the modeled stent hinge simulates his behavior during stent deployment. Considering these aspects was possible to evaluate the coating integrity. With this model was possible to predict the occurrence of delamination during stent deployment and to determine that the delamination risks increases with increasing the coating thickness. Finally, it was obtained a general function that allows to determine the maximal contact stress for a stent hinge with U shape.

Keywords: stent, finite element method, delamination.

References:


Presenting author’s email: ernestico9023@gmail.com