Effect of Strut thickness on the Inflation Behaviour of Coronary Artery Stent using FEM
Hashim V.
Associate Professor
TKM College of Engineering Kollam, Kerala
hashimtkmce@gmail.com

Resmi S.L.
Assistant Professor
TKM College of Engineering Kollam, Kerala
hashimtkmce@gmail.com

Dr. Dileep P.N.
Professor TKM College of Engineering Kollam, Kerala
hashimtkmce@gmail.com

Abstract - Stent implantation is one of the most accepted method in treating coronary artery diseases due to their simplicity and faster patient recovery. Coronary stents are highly patient specific and the implantation success greatly relies on the design and clinical procedure. Studies confirm that, performance characteristics like radial strength, maximum expansion, stress concentration, recoil etc. are highly governed by the physical design of the stent. This paper focuses on finite element analysis using ABAQUS package to investigate the effect of varying strut thickness on the expansion behavior of a commercially available coronary stent. Envision Scientific Coronary Stent was adopted for this study. Total of five stent designs, one with actual size, two models with a strut thickness lesser than actual thickness and two with higher values were considered. The expansion behavior of the models under consideration was compared to the model with original dimensions. It may be noted that the thickness is one of important design parameter that affects the expansion and stress directly. With 15% reduction in thickness, the expansion increases by 10%. While, for an increase of 15% in thickness resulted in decrease in expansion by 10%. It may be noted that the percentage of increment in stress is linear and inversely proportional. Whereas, the increment in expansion is non linear and is minimal for low thickness. The present approach can be used for selecting the design parameters of stent to arrive in an optimum design.

Keywords - Coronary stent, FEM, Stress concentration, Strut design.

I. INTRODUCTION
Coronary artery disease is considered as one of the leading cause that claims lives of millions every year worldwide. Development of atherosclerotic plaques (stenosis) within the coronary arteries leads to heart blockage or heart attack. Stent implantation is the most popular non-
surgical method and got wide acceptance, because of its simplicity and high success rate. Coronary stents acts as scaffold to the artery and open the stenosed artery to restore blood flow. The implantation of the stent in a human artery is a complicated process. Often, the type of stent selected or the methods involved in stent implantation determines the success rates [1]. Since, physical test methods are very difficult, Finite element analysis provides a relatively cost-effective and extremely beneficial research tool to optimize the mechanical properties of stents. Numerical modeling of stents has been widely used to investigate the influence of stent design on the stent and artery. Many researchers have used the finite element method to evaluate stent characteristics such as recoil, foreshortening and dog boning effects. S.N. David Chua, et al. and others have investigated the influence of the expansion mechanism on the artery wall on stenosed vessels [2]. N Eshghi et.al. analyzed Palmatz stent with artery and balloon and found that the maximum stress is at the four corners of the cell and at the open end [3]. C Lally et.al. made a computational study to investigate the mechanical behavior of stent and interaction between stent and artery. He has compared the performance of different design and developed a novel stent design [4]. Julian Bedoya et.al. made a study on the effect of stent parameters on normal artery wall mechanism. Strut spacing, the radius of curvature at the crown section and axial amplitude are the stent parameters considered for the study[5]. Arjun R. et al. investigated the behavior of the stent by varying the inflating pressure and realized that the degree of expansion is mainly related with the design and the inflating pressure [6]. In the present study, finite element analysis is carried out to assess the effect of strut thickness on the stress and expansion developed in the coronary stent. The geometry of the commercially available Envision Scientific stent is selected for this study. Finite element analysis software ABAQUS is used to analyze the stress concentration in different strut thickness by keeping the inflating pressure constant

II. FE MODEL OF THE STENT

Most of the stents are of the complicated design and commercially available stents rarely disclose the information regarding their geometrical features, the generation of a solid model of a stent is possible by cutting an actual stent along its longitudinal axis and measuring important geometrical properties such as strut width, thickness, and length using a microscope. Using these measurements, a model of the planar stent geometry can be created accurately defined within an FE package. In this analysis solid models of a stent, balloon and artery were created using SOLID WORKS software. Envision 3.5x28 stent is a ring type stent. There are total 21 rings and rings are connected with straight bridge or connector. The strut is having a straight portion and a crown portion. Straight portion is not uniform width, the width is maximum at the centre and it becomes minimum near the crown. The geometry of the stent is measured by using Leica microscope available at Sree Chitra Institute of Medical Sciences, Thiruvananthapuram. The actual geometry of the stent strut is shown in Fig.2 and the dimensions are shown in table 1.
The Model A has been created with an actual size of the strut with 8 rings (total length is 10mm) and thickness 0.064mm. As there are two connectors in each ring and are diametrically opposite, the stent is cut axially from the middle and only half of the stent is considered by taking the advantage of symmetry for the simulation. The model is shown in Fig. 1 below. Four other models (Model B, C, D, E) were also created by increasing the thickness 15% and 30% of the actual thickness and reducing the thickness 15% and 30% of the actual thickness (strut thickness 0.044, 0.054, 0.074 and 0.084 respectively).

| Stent       | Length (mm) | Outer diameter (mm) | Strut thickness (mm) | Strut width (mm) | Crown radius (mm) | No. of struts |
|-------------|-------------|---------------------|----------------------|------------------|-------------------|---------------|
| Envisio n 3.5x28 | 26          | 1.09 (crimped)      | 0.064                | 0.115/0.068      | 0.057             | 21            |

The Model A has been created with an actual size of the strut with 8 rings (total length is 10mm) and thickness 0.064mm. As there are two connectors in each ring and are diametrically opposite, the stent is cut axially from the middle and only half of the stent is considered by taking the advantage of symmetry for the simulation. The model is shown in Fig. 1 below. Four other models (Model B, C, D, E) were also created by increasing the thickness 15% and 30% of the actual thickness and reducing the thickness 15% and 30% of the actual thickness (strut thickness 0.044, 0.054, 0.074 and 0.084 respectively).

![Fig. 1 FE model of stent](image1)

![Fig. 2 geometry of the strut](image2)
III. METHODOLOGY

In the initial stage, a stent model A is created in Solid Works software. Due to the symmetry of the stent by utilizing the correct boundary conditions, half part was used to simulate the expansion process. An elastoplastic material was assumed as the stent material. Material properties taken were chosen to approximately represent the behavior of Cobalt Chromium L605 and is given in table 2. The plastic properties like plastic strain and corresponding stress were entered by taking values from the stress-strain graph assigned to the section of the model. The step time was given as 1.0ms. Symmetric boundary conditions were imposed on the nodes of the stent in the planes of symmetry where all the nodes perpendicular to y-axis were not allowed to move in the y-direction and all the nodes perpendicular to x-axis were not allowed to move in the x-direction. Both ends of the stent were free to move along its axis so that the expansion and shortening behavior of the stent would be observed.

Polyurethane rubber material was used to represent the balloon. The balloon as a medium to expand the stent was modelled as 11mm length, the outer diameter of the balloon as 0.962 mm and the thickness as 0.03mm. Angioplasty balloons are usually manufactured from materials such as polyethylene, polyamide, and their behavior is typically modelled using linear elastic material models based on data provided by the manufacturer. The material property of the balloon is given in the Table: 2. The boundary condition given for balloon is also same as stent except for the two ends. It was assumed that the ends of the balloon are fully tethered and hence only the expansion, in the radial direction was permitted. Surface to surface algorithm approach was selected in assembly of stent and balloon to cope with the nonlinear contact problem between the two component surfaces.

| Part         | Density (kg/m$^3$) | Young’s modulus (GPa) | Poisson’s ratio |
|--------------|--------------------|-----------------------|-----------------|
| Stent (L605) | $9.1 \times 10^{-5}$ | 243                   | 0.30            |
| Balloon (polyurethane) | $1.07 \times 10^{-5}$ | 0.09                  | 0.49            |

IV. LOADING CONDITIONS

| Part      | Element geometry                  | No. of elements | No. of nodes |
|-----------|-----------------------------------|----------------|--------------|
| stent     | Tetrahedral                       | 28712          | 57159        |
| Balloon   | tetrahedral elements with hybrid formation | 15052          | 29473        |
V. RESULTS AND DISCUSSIONS

During the inflation of the stent, the expansion up to 300KPa is quite negligible as the stent is in crimped state. The expansion from 1.09mm to 3mm was very fast above 300KPa. Because uncrimping happens around 320KPa. The further increment of pressure from 400 to 800KPa the increment in displacement is slow and steady. The stent deforms plastically and yielding takes place at this pressure. Due to strain hardening, again the expansion becomes slow above 700KPa. During expansion, the stress gradually increases. However, stress in some areas of the stent is typically high. These areas are found to be the crown region of the stent. This is because the struts are pulled apart from each other to enable radial expansion and there are only little changes in the cross-section area of the strut. But if the applied pressure exceeds 1000KPa the yielding starts and the weakest part of the stent collapses.

The maximum Von Mises stress obtained at an inflating pressure of 0.8MPa is 1117 MPa and the maximum deformation is 1.127mm. This is compared with the experimental results and the results are validated. The figures 3 and 4 gives the result of expansion analysis of the actual stent with the thickness 0.064mm (model A). Then same analysis was repeated to the other models also by changing the thickness (+15%,+30%,-15%,-30% ). The results of the models B, C, D, and E are given in table 4.
Graphs are plotted against Strut thickness Vs Stress and Strut thickness Vs Deformation as shown in Figs. 5 and 6 respectively. It shows that for a given design the stress induced due to the application of a pressure load depends on the strut thickness and it decreases with increase in thickness of strut. The stent B has maximum stress as its thickness is least among the five models. The yielding starts earlier in the strut with small thickness. Whereas the expansion or displacement of the stent decreases with increase in the strut thickness because yielding or
plastic deformation is delayed when the strut thickness is increased. The stent B has maximum expansion of 1.314mm at a pressure of 0.80MPa. Also, the graph shows that the variation of the stress with thickness is linear and inversely propotional whereas the thickness Vs expansion is nonlinear. Percentage of increase in the expansion is minimal in small thickness compared to larger thickness.

VI. CONCLUSION

The mechanical behaviour of a balloon stent assembly with respect to the changes in strut thickness was analyzed in commercially available FEA software, ABAQUS. According to the results, the maximum stress and expansion are obtained at the crown region of the stent. This is due to the pulling effect during expansion. Moreover it was identified that, minimum thickness is better because with low thicknesses the stress is low and the expansion is high. A linear trend was observed in stress increment and is quite predictable but increment in expansion is nonlinear and is minimal for low thickness. It is concluded that the strut thickness which gives maximum expansion and sufficient strength is to be considered as optimum thickness of the strut. More over the results gives a clear indication that, the inflating pressure to be selected based on the strut thickness and the expansion required. The results can be used for further understanding of stent design and optimisation.

References

1. Francesco Migliavacca,Lorenza Petrini,Maurizio Colombo,Ferdinando Auricchio,Riccardo Pietrabissa “Mechanical behavior of coronary stents investigated through the finite element method”. Journal of Biomechanics 35(2002): 803-811

2. David Chua SN, Mac Donald BJ, Hashmi MSJ. Finite-element simulation of stent expansion. Journal of Materials Processing Technology 2002; 120: 335-40.

3. N.Eshghi, M.H Hotjjati, M Imani ,A.M Goudarzi. “Finite Element Analysis of Mechanical Behaviours of Coronary Stent”. Physics Engineering 10 (2011) :3056-3061

4. C.Lally, F. Dolan, P.J.Prendergast. “Cardiovascular stent design and vessel stresses: a finite element analysis. J.of Biomechanics 38(2005) 1574-1581

5. Julian Bedoya, Clark A Meyer “ Effect of stent design parameters on nominal artery wall mechanics” J. of Biomechanical Engineering, oct 2006, vol 128/757

6. Arjun R., Hashim V.,Dileep P.N., “ Effect of inflating pressure on the expansion behaviour of coronary stent and balloon: A finite element analysis”, International Conference on Aerospace and Mechanical Engineering, 14-16 December 2015.