Preliminary of finite element analysis (FEA) investigation on the stent expansion

Zulfaqih Lazim, Al Emran Ismail¹, Ishkrizat Taib
Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia (UTHM), Batu Pahat 86400 Johor, Malaysia.

E-mail : emran@uthm.edu.my / al_emran@hotmail.com

Abstract. The requirement of stent expansion is important to stent deployment on the inner of the defected blood vessel. This defected blood vessel decreases in lumen diameter size because of accumulation of fat or plaque and caused the reduction of blood flow and oxygen supply to overall body. According to clinical guidelines, blood vessel that blockage by 70% need to be stenting. Stent expand past it elastic limit and plastically deform until the targeted diameter is achieved. Explicit dynamics Finite Element Analysis (FEA) was used in this paper to analyse the stent expansion process. The knowledge of stent radial displacement could prevent the injury of inner surface of artery while understanding on longitudinal foreshortening that aided the stent placement on targeted artery location. The controlled variable was geometry, material and boundary condition of stent. The manipulated variable was the speed of pressure application. The outcomes of this stent expansion simulation are maximum radial expansion, the uniform of radial displacement and longitudinal foreshortening. This simulation results help to improve stent installation on human body for balloon expandable stent and increase the understanding of explicit dynamics structure simulation of finite element in biomedical application.

1. Introduction
Stent is metal tube mesh structure designed to regain more blood flow at blockage blood vessel area. Mostly the blockage of 70% from original blood vessel diameter required a stent to be implanted to open up the lumen area. The stent is crimped inside catheter and guided to the location of stent placement through inside of the blood vessel, the stent then expands to the required diameter to regain normal blood flow. The increasing of fatty substances contributes to total blockage of blood vessel and lead to heart attack [1]. Therefore, it’s become normal nowadays for medical doctor to treat blockage artery with stent mechanisms [2-3]. Another method of regaining normal blood flow from blockage artery is through bypass open surgery. Traditionally angioplasty is used to open up blockage artery by inflating tiny balloon in coronary artery [4]. However, re-narrowing of artery occurs after the angioplasty procedure at the defected artery location. Stent is introduced to overcome this problem and hold artery based on radial strength after nonlinear stent expansion. The stent will not recoil because it expands to plastic deformation region since it is experienced a permanent shape. However, some structural damages can be happened during the expansion process if the stent is not properly designed. In this paper, a preliminary investigation is conducted to design and analysis of the stent performances

¹ To whom any correspondence should be addressed.
2. Geometrical model, material and boundary condition

Stoeckel et al. [6] estimated that more than 100 of stent design model exist in biomedical device market. There are many aspects of design variability for example in term of strut surface area, geometry, fabrication and material of the stent. The stent geometrical model of FEA simulation in this study was adopted from stent geometrical model by Chua et al. [7]. The stent model consists of 3mm outer diameter, 2.9mm inner diameter, 20mm in length and 66 slots. The dimension and slot arrangement are visualized in Figure 1. The cross-section shape of the model stent strut was chosen as rectangular shape. Figure 1(a) shows a unit cell of stent where the distance between two horizontal lines is 0.214 mm and the gap between struts is 0.40 mm. On the other hand, Figure 1(b) reveals the complete in-plane stent design before it is wrapped over the solid cylindrical bar.

Figure 1: (a) Strut’s bridge and width dimension 0.400mm and 0.214mm on 2D drawing, (b) Stent wrap feature in SOLIDWORK for 3D stent model generation.

The slot or cell shape is also rectangular shape when view normal to stent surface. This stent design having similar size of cell that distributed evenly on stent surface area. Full stent surface area is 101.71mm² while full cylinder surface area without stent cells is 188.4956mm². This stent design having 53.96% surface area from full cylinder surface area with the same diameter and length. Stent material mostly made from stainless steel. It was least expensive material compare to other stent material type such as Nitinol. Stainless steel provides good radial strength for stent material [8]. The stent material assign in this Finite Element model for FEA stent expansion simulation was stainless steel 304 (SS304). Hiramoto et al. [9] also used SS304 material on stent analysis of the effect of magnetic resonance imaging. The SS304 material assignment consists of density 7.86kg/mm³, young’s modulus 193MPa, Poisson’s ratio 0.27, yield strength 207MPa and tangent Modulus 692MPa. A bilinear isotropic hardening on plasticity was assumed as stent material model. Symmetry by half model was choose for boundary condition for simulation of stent expansion.

Chua et al. used half stent symmetry [7] model and quarter stent symmetry model [10] for boundary condition. Observing from this two works [7 & 10], the symmetry is used for stent model while number of planes are selecting based on geometrical cell arrangement of the stent geometrical model. The selection of symmetry depending on orientation of stent bridge and strut arrangement together with number of cells. The number of cells determine the surface area of stent. Figure 2 show half stent model by symmetry in ANSYS design modeler for FEA. It is showed the stent expansion process in blockage artery. Figure 2 reveals the geometry of the stent used in this study. Figure 2(a) shows the complete or full stent model consisting of 66 slots with 0.5mm strut thickness. Since the model is symmetry, only half of the model is used as shown in Figure 2(b). This symmetry control stent simulation to expands radially and shrink by longitudinal direction on stent expansion process. However, a proper boundary condition should be aligned with stent geometrical design to avoid inaccurate FEA. Sensitivity analysis was conducted to find proper meshing size [11-12] for further
stent expansion simulation. The comparison was made between nine simulations with different mesh body sizing. Other parameters treated as constant variable such as geometry, material assignment and speed pressure application (SPA). The material used for this sensitivity analysis was SS304, while speed pressure application taken as 400MPa/ms. The simulation for sensitivity analysis objective to find suitable mesh size for further stent expansion simulation. Figure 3 shows the enlarged area of the stent when the model in Figure 2 is meshed using ANSYS finite element software. Figure 4 shows the expansion inside the blood vessel governed by ballooning mechanism.

Figure 2: (a) Stent deployment in defected artery, (b) 3D stent model, and (c) Symmetry half stent boundary.

Figure 3: (a) Mesh by 0.02mm body sizing consist of 411942 elements, and (b) Mesh by 0.075mm body sizing consist of 9854 elements.

Figure 4: Stent expansion mechanisms inside the blood vessel.

Figure 5 show the visualization of mesh for sensitivity analysis, (a) for body sizing consist of 411942 elements, and (b) for 0.075mm body sizing consist of 9854 elements. The 411942 elements take simulation run time by 854.685 minutes approximately 14 hours for this sensitivity analysis study. In FEA, quality and cost are important consideration for simulation work, quality in term of
accuracy required more mesh element that increase complexity however it cost more simulation time [13]. Makino et al. [14] studied simulation of car model crash analysis on the effect of fine mesh model and produced a 10-million shell elements car model to achieve the good accuracy in crash analysis. Table 1 shows element size as an input of simulation and simulation time, pressure input stent expansion and percentage error as an output of the simulation.

| No | Element Size (mm) | Number of elements (x1000) | Simulation time (ms) | Pressure Input (MPa) | Percentage Error (%) |
|----|-------------------|-----------------------------|----------------------|---------------------|----------------------|
| 1  | 0.020             | 411.942                     | 0.026703             | 10.6812             | 0.00                 |
| 2  | 0.030             | 123.252                     | 0.028219             | 11.2876             | 5.37                 |
| 3  | 0.040             | 68.398                      | 0.029391             | 11.7564             | 9.15                 |
| 4  | 0.050             | 21.582                      | 0.027531             | 11.0124             | 3.01                 |
| 5  | 0.060             | 15.855                      | 0.027907             | 11.1628             | 4.31                 |
| 6  | 0.075             | 9.854                       | 0.027315             | 10.9260             | 2.24                 |
| 7  | 0.100             | 6.662                       | 0.029439             | 11.7756             | 9.29                 |
| 8  | 0.500             | 0.685                       | 0.032396             | 12.9584             | 17.57                |
| 9  | 0.750             | 0.420                       | 0.027917             | 11.1668             | 4.35                 |

The element size of 0.075mm body sizing with 9854 number of elements was selected for further FEA simulation. This selection based on result of percentage error. The lowest percentage error gives more accurate result compared to other element size with own percentage error as shown in Table 1. Reference mesh for percentage error is highest elements which is 411942 elements and 0.02mm body sizing because high elements number claimed [14] to produce more accurate result. High mesh produces accurate result however taking more than half day of simulation time for this stent expansion simulation is very time consuming for eight speed pressure application testing to failure for stent expansion. From the sensitivity analysis in Figure 5, the mesh size is reduced from 0.02mm body sizing to 0.075mm body sizing which produce almost similar results and decrease the cost of simulation time.

3. Effect of pressure speed

The speed pressure application (SPA) was taken as loading in this simulation. There are 8 different value of speed pressure which are 400, 350, 300, 250, 200, 150, 100 and 11.3 MPa over 1ms in this paper. The value of speed pressure application was referred from Chua et al. [7] that was used only two speed pressure application for stent expansion simulation which is 400MPa/ms and 11.3MPa/ms.
Present study used higher than SPA, more element size and with sensitivity analysis to define better accurate mechanical stent structure response from different speed input variability by FEA. The stent structure undergoes stress deformation from elastic to plastic and continue until reaching material ultimate tensile stress (UTS) that considered failed [7]. The application of pressure on stent internal surface shown on Figure 6.

4. Results and discussion

There are several analysis of stent expansion by FEA. Output of this result can be used for future reference of stent speed pressure input or stent design optimization. The stent expands until its failure, the failure based on Ultimate Tensile Strength (UTS) 517MPa [7].

Figure 7: (a) Stent expansion testing to failure 517MPa with undeformed geometry of stent, (b) Closer view of maximum stress at edge stent for 400MPa/ms SPA, and (c) Closer view of maximum stress at middle stent for 11.3MPa/ms.

Maximum Von-Mises Stress on stent structural are visualized in Figure 7(a). It is showed half of stent structure expand to failure on stent expansion process, the radial displacement and
foreshortening were analyse based on eight speed pressure application. From the figure 7(a), it can be clearly seen expansion of stent resulting increasing in diameter and decreasing in longitudinal dimension. These two output parameters referring as radial displacement and foreshortening of the stent. Stent structure consider fail after reaching ultimate tensile strength 517MPa [7]. For high SPA the stent maximum stress occurs at the edge shown by Figure 7(b) and for low SPA the maximum stress occurs at the middle shown by Figure 7(c) explaining the effect of input from low SPA which is 11.3MPa/ms producing high stress occur at four corners of the cells and low stress at the middle of the strut. Material structure erosion or fracture potentially happen at the maximum stress area. The possible crack location and fatigue fracture of stent have been reviewed by Lazim et al. [15] that mostly happen at bridge tip region similar location on this result visualization showed on Figure 7(c). Design optimization by geometrical stent structure could be focus in this area. The maximum input pressure just before failure following speed pressure application from SPA1 to SPA 8 was 10.93MPa, 9.96MPa, 8.97MPa, 7.92MPa, 6.82MPa, 5.63MPa, 4.33MPa and 1.36MPa. Result by SPA8 with 11.3MPa/ms speed pressure gives 1.36MPa maximum pressure input show 0.7% different from simulation done by Chua et al. [7] on similar speed input pressure, which is 1.35MPa of maximum pressure input. Figure 8 indicate high SPA resulting low strain and low SPA resulting high strain on nonlinear plastic region at the end of simulation, and the result is identical following SPA1 to SPA8 from high speed to low speed of pressure application on stent inner surface area.

![Figure 8: Stent against strain of eight speed pressure variation for stent expansion](image)

There are two significant area defining the deformation of the stent under increasing of pressure over time, which are elastic region and plastic region. This region divides by Yield Strength of SS304 which is 207MPa. The stent behaves similar on stress and strain curve by eight variation of SPA on elastic region shown on figure 8. After the structure achieving Yield strength, under plastic region the graph indicates larger deformation from low SPA with the increasing of stress on the biomedical structure. The lowest speed pressure is curve SPA8 have the higher strain therefore indicate the largest deformation on the speed pressure setting.
Figure 9: Radial Displacement against Pressure for 8 different speed pressure application.

Figure 9 shows the radial displacement against Pressure for 8 different speed pressure application. The lowest pressure speed denote by SPA8 has the largest radial displacement. This radial displacement data explaining on stent radial displacement selecting as surface body of stent outer surface based on polar coordinate. Data results from simulation on pressure speed SPA1 which is 400Mpa/ms gives lowest radial displacement of 3.3740mm while pressure speed SPA8 which is 11.3Mpa/ms give highest output of radial displacement of 7.0851mm. Other SPA is identical based on graph and the results are in between SPA1 and SPA8. Stent gives highest radial displacement on lower SPA. The injury on blood vessel inner surface exist on improper stent expansion process, therefore higher SPA is proposed for stent expansion based on result data. This is because low pressure speed has resulted higher radial displacement over time which can damage or rupture the artery from uncontrollable high radial displacement. The detail analysis of radial displacement is displayed on 3D view of point node and graph at end of simulation time for lowest and highest SPA illustrated in Figure 10(a) and (b). There are 9-point nodes that selected along the longitudinal axis or z-axis on polar coordinate system used in this analysis. Polar coordinate system setting z-axis for foreshortening of the stent, and x-axis for radial displacement while y-axis represent theta that indicate torsion which is not included for analysis of stent expansion in this paper. The point nodes are selected align between each other in z-axis while assuming the constant expansion in radial deformation over time. The uniform of radial displacement is investigated and showed at Figure 10(a) and (b). Uniform radial displacement is displayed by graph of SPA1 which is 400MPa/ms on figure 10(b) while non-uniform radial displacement showed by graph 11.3MPa/ms. Stent expansion require uniform radial displacement from uniform speed pressure application input to avoid inner tissue arterial surface injury.
Figure 10: (a) Nodal point for radial displacement analysis on 3D stent half model, and (b) Radial displacement against longitudinal nodal location for speed pressure application of 400 and 11.3 MPa/ms.

The uniform radial displacement resulting from high speed pressure input while non uniform radial displacement resulting from low speed pressure input. Low speed pressure input has higher simulation time before structure failure and the deformation process occur differently between the middle portion of stent structure and the edge of stent structure compare to high speed pressure input. Therefore, result from the graph indicate high SPA is preferable for stent expansion application. Figure 11(a) show visualization of half stent foreshortening at the end of simulation time for 11.3MPa/ms speed pressure. It showed that large displacement on z-axis direction while the outputs of the results are controlled to give deformation only in z-axis on polar coordinate setting. The largest foreshortening showed by graph SPA8 on Figure 11(b). The simulation of stent expansion simulated by Explicit Dynamics ANSYS workbench using AUTODYN solver.

Stent foreshortening at the end of the simulation gives 4.1124mm by SPA8 and 0.8249mm by SPA1 while the other graph of SPA gives identical results in between of SPA1 and SPA8. The lower foreshortening is more favourable to assist greater stent placement inside defected blood vessel. Therefore, based the results, it indicates that larger speed of pressure application produced shorter foreshortening thus the speed pressure input by 400MPa/ms is favourable for stent expansion application.
Figure 11: (a) Half stent model foreshortening on expansion process for 11.3MPa/ms, and (b) Foreshortening against Pressure for eight different pressure application.

5. Conclusion
Based on the preliminary finite element analysis and investigation, there are several conclusions can be drawn:

1. Insignificant effect of different element size and number of elements on the simulation time.
2. During the expansion processes, the maximum stresses generally occurred at the joints between the structs where once the expansion reached the maximum level, the stress is beyond that the ultimate strength.

Acknowledgement
Authors acknowledge Kementerian Pendidikan Malaysia for financially sponsoring this paper through Fundamental Research Grant Scheme (FRGS) vot. 1592.

References
[1] Marple AF, Elliot EM, Hand MM, 2006, Modern Treatment for Heart Attacks Opening Blocked Arteries Quickly, Circulation, 114(20), 578-580.
[2] Edward H. Livingston, MD; Cassio Lynm, MA, 2012, Stents to treat coronary artery blockages, JAMA, 308(17) 1824
[3] Dangas G, Kuepper F, 2002, Restenosis: Repeat Narrowing of a Coronary Artery, Circulation, 10(22) 586-2587.
[4] Morrison D, Sacks A, Grover JF, Hammermeiste KE, 1995, Effectiveness of percutaneous transluminal coronary angioplasty for patients with medically refractory rest angina pectoris and high risk of adverse outcomes with coronary artery bypass grafting, Am. J. Cardiol., 75 (4), 237–240.
[5] H. Li et al., 2017, Design optimization of stent and its dilatation balloon using kriging surrogate model, Biomed. Eng. Online,16, 13.
[6] Stoeckel D, Bonsignore C and Duda S, 2002, A survey of stent designs, Minim. Invasive Ther. Allied Technol., 11(4)137–147.
[7] Chua SND, Mac Donald BJ, Hashmi MSJ, 2002, Finite-element simulation of stent expansion, *J. Mater. Process. Technol.*, 20(1–3) 335–340.

[8] He D, Liu W, Zhang T, 2015, The Development of Carotid Stent Material, *Interv. Neurol.*, 3(2), 67–77.

[9] Hiramoto JS, Reilly LM, Schneider D, Skorobogaty BHJ, Chuter TAM, 2007, The effect of magnetic resonance imaging on stainless-steel Z-stent–based abdominal aortic prosthesis, *J. Vasc. Surg.*, 45(3), 472–474.

[10] David Chua, Mac Donald, SN, Hashmi, MSJ, 2003, Finite element simulation of stent and balloon interaction, *J. Mater. Process. Technol.*, 143–144(1), 591–597.

[11] McCurry MR, Evans AR, McHenry CR, 2015, The sensitivity of biological finite element models to the resolution of surface geometry: a case study of crocodilian crania, PubMed, 3, 988.

[12] Rozakis, L. *Schaum’s Quick Guide to Writing Great Research Papers*. New York: McGraw-Hill, 2007.

[13] Liu, Y, Glass, G, 2013, Effects of Mesh Density on Finite Element Analysis, *SAE Technical Paper*.

[14] Makino, M, 2008, The Performance of 10-Million Element Car Model by MPP Version of LS-DYNA on Fujitsu PRIMEPOWER, *10th International LS-DYNA Conference*, 16, 7–10.

[15] Lazim Z, Ismail AE, Taib I, Atan BAM, 2017, A review of stent’s failure on patent ductus arteriosus, *IOP Conf. Ser. Mater. Sci. Eng.*, 165 (1), 12007.

[16] Paisal MSA, Adnan SFS, Taib I, Ismail AE, Abdullah MK, Nordin N, Seri SM, Darlis N, 2017, Flow Characteristics Near to Stent Strut Configurations on Femoropopliteal Artery, *IOP Conference Series: Materials Science and Engineering*, 226 (1), 012147.

[17] Mohd Atan BA, Ismail AE, Taib I, Lazim Z, 2017, A review on fracture prevention of stent in femoropopliteal artery, *IOP Conference Series: Materials Science and Engineering*, 66 (1), 012006.

[18] Amir Paisal MS, Taib I, Ismail AE, 2017, Computational Analysis on Stent Geometries in Carotid Artery: A Review, *IOP Conference Series: Materials Science and Engineering*, 166 (1), 012003.

[19] Z. Lazim, A.E Ismail, I. Taib and B.A Mohd Atan, 2017, A review of stent's failure on patent ductus arteriosus, *IOP Conference Series: Materials Science and Engineering*, 166 (1), 012007.

[20] Asrizan Kaha, Ishkrizat Taib, Muhammad Sufyan Amir Paisal, Ahmad Mubarak Tajul Ariffin, Al Emran Ismail, Norzelawati Asmuin, Reazul Haq Abdul Haq, Surapong Chatpun, Takahisa Yamamoto, Kahar Osman, 2018, Lumped Parameter Modelling in Femoral Popliteal artery for Normal and Severe Conditions, *International Journal of Integrated Engineering: Special issue 2018: Mechanical Engineering*, 10 (5) 193-203.

[21] Hamdan A, Mustapha F, Ahmad KA, Mohd Rafie AS, Ishak MR, Ismail AE, 2016, The Effect of Customized Woven and Stacked Layer Orientation on Tensile and Flexural Properties of Woven Kenaf Fibre Reinforced Epoxy Composites, *International Journal of Polymer Science*, 2016, 6514041.

[22] Ismail AE, Hassan MA, 2014, Low velocity impact on woven kenaf fiber reinforced composites, *Applied Mechanics and Materials*, 629, 503-506.