Automatic stent fitting using CFD estimates of hemodynamic parameters for pulmonary artery stenosis

Xavier Jordi Oller[1], Lina Gundelwein[2], Joaquim Miró[3], and Luc Duong[1]

[1] École de technologie supérieure, Montréal, Canada
[2] Ruprecht-Karls-Universität Heidelberg, Heidelberg, Germany
[3] CHU Sainte-Justine, Montréal, Canada

Abstract. The role of stenting is to restore the blood flow, hence reducing cardiac effort which, in terms of hemodynamics, implies a lower pressure gradient between both sides of the stenosis. Determining the geometry and position of a stent is one of the most sensitive tasks when planning a pulmonary artery stenosis intervention. The goal of this study is to automatically determine the position and geometry of the deployed stent using hemodynamic parameters computed from computational fluid dynamics (CFD). The first step is segmentation and 3D modeling of the pulmonary artery from the patient's computed tomography angiography (CTA) scan. Afterwards, the artery's centerline is extracted and the stenosis location detected. Finally, a set of stenting solutions is generated and the pressure gradient calculated for each of them. The gradient calculations are performed by means of Lattice-Boltzmann CFD simulations, which are well suited for complex geometries and are comparatively efficient in terms of execution time. The minimum gradient value will constitute the optimal stent placement. This study showed that an improvement of 60% and 23% of the pressure gradient could be obtained by placing optimally the stent.

Keywords: Computational fluid dynamics, segmentation, stent placement, pulmonary artery

1 Introduction

The incidence of Congenital Heart Defects (CHD), including pulmonary artery stenosis, is estimated to be between 4 and 50 individuals per 1000 births [9]. Interventions are most commonly performed through catheterization, given its advantages in comparison with open surgery in terms of blood loss, time consumption and treatability of high-risk patients [4]. However, endovascular interventions for treatment of stenosis introduce a new constraint when the placement of a stent is required: it might be useful to estimate the stent position and geometry before the intervention. For this reason, a preoperative software tool would be highly valuable to obtain an optimally placed stent. Multiple methods aiming to

Paper presented at:
Joint MICCAI-Workshops on Computing and Visualization for Intravascular Imaging and Computer Assisted Stenting (CVII-STENT)
Athens, Greece, Oct. 17-21, 2016

support cardiologists in stenosis preoperative planning have been developed in recent years. A common approach is the simulation and visualization of stent expansion and placement for further validation [3]. For instance, Gundelwein et al. [6] proposed a method for automatic stent placement in pulmonary arteries using the artery's volume as an optimization parameter. The artery volume might not fully describe the geometric complexity of the stenosis [2]. Also, hemodynamic data is frequently used in the catheterization room to evaluate the intervention's effectiveness, in particular the translesional pressure gradient. In vivo pressure gradient measurement can be obtained using pressure catheters, but their manipulation in vascular structure remains highly invasive. Several studies have used CFD methods to analyze the behavior of blood flow in the presence of stenosis [13]. Taylor et al. [10] calculate pressure and Fractional Flow Reserve (FFR) in patients' coronary arteries with stenosis, both before and after treatment. However, FFR is not considered for optimization of stent's geometry and CFD simulations are only employed as a technique to assess blood flow at preoperative and postoperative stages. Wall-Shear Stress (WSS) measurements play a major role in the re-stenosis process and might be relevant for stenosis assessment.

2 Methods

The overall workflow of our approach is illustrated in Fig. 1. First, the pulmonary artery is segmented from a CTA volume and the centerline and stenosis is modeled in 3D. Second, an iterative scheme is devised to find the optimal placement using CFD simulations. The diameter of the pulmonary artery is modified at each iteration until the optimal gradient before and after the stenosis is obtained.

Vessel segmentation and modeling Initially, a 3D model of the patient's pulmonary artery is obtained. A semi-automatic segmentation is performed with TurtleSeg software [11], which generates a 3D mesh from a set of user-defined contours on different planes intersecting the CTA volume. Afterwards, the mesh is converted to parametric NURBS with Rhino 5 CAD software (McNeel, Seattle, WA, USA). The mesh's surface is smoothed to reduce irregularities and the number of faces is lowered to reduce computational costs as illustrated in Fig. 2. The inlets and outlets of the artery are generated for the CFD simulations.

Centerline extraction and identification of the stenosis The artery's centerline is extracted from the 3D mesh through Voronoi diagram implemented in VTK. Since the stent will be placed along the artery's centerline in the simulation, the centerline is then smoothed to prevent unrealistic stent deformation. Once the centerline has been obtained, the stenosis location can be automatically detected. First, each artery point is assigned to its closest centerline point. Sets of artery points belonging to each centerline point are formed. The average distance is then calculated for each set of points, resulting in the average radius of the artery's

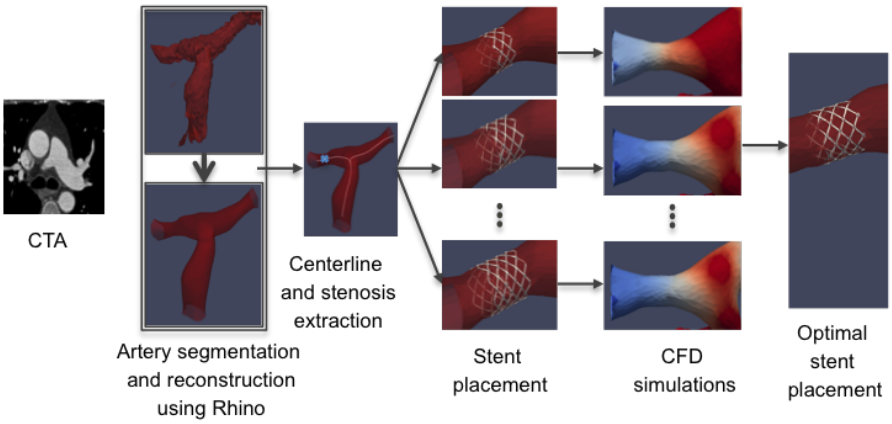

Fig. 1. Workflow of the stent fitting procedure. From left to right: The patient's pulmonary artery is segmented from a CTA acquisition. A 3D model of the pulmonary artery is build from the CTA data, including matching centerline and the location of the stenosis. Finally, stent placements are generated at different expansions and hemodynamic simulations are conducted to determine the optimal placement.

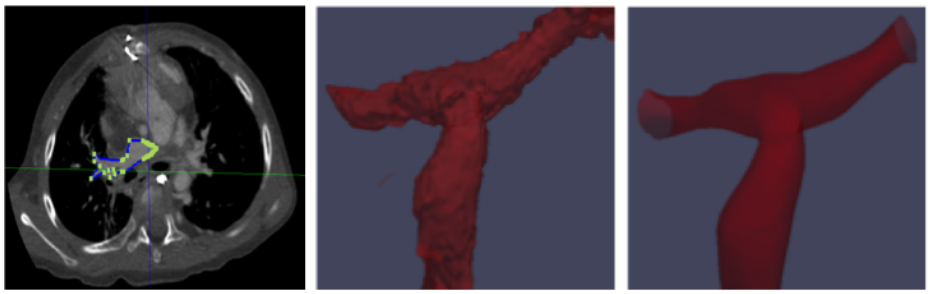

Fig. 2. User-defined contour on a CTA slice in blue, with associated key points from different planes from segmentation in green(left), automatically generated mesh from contours (center) and resulting vessel geometry after NURBS modeling with Rhinoceros (right).

lumen at every point of the centerline. Thus, the stenosis location is defined as the centerline point with the lowest average radius.

Automatic stent placement The following step consists in simulating stent placement at different balloon expansions as described in [6]. Firstly, the stent is scaled to the expansion diameter and the foreshortening of the stent is applied using an interpolation function of the manufacturer's foreshortening values. The expansion diameter is defined as the diameter required for the stent to roughly fit in the stenosis. Secondly, the stent is placed at the stenosis location and curved along the artery's centerline. Each section of the stent is rotated so that its normal vector matches the centerline's tangent vector. The process is done iteratively starting at the stenosis location and moving along each of the centerline points towards both ends of the stent. To determine the rotation angle at each iteration step, a cubic spline of the centerline is calculated and subsequently derived to obtain the tangent vector at each point of the centerline. The last step of the stent placement simulation is the expansion of the artery to match the stent expansion. The followed approach consists in finding all the points whose distance to the centerline is smaller than the artery's expansion and translating them along the radial direction so that they match the stent expansion's radius. The geometric optimization of the stent pose and expansion was performed using a Hill Climbing optimization algorithm and the objective function was designed by considering CFD estimates of hemodynamic measurements.

Hemodynamic assessment from CFD simulation of the stenosis Hemodynamic simulations in the vicinity of the stenosis are performed using the Lattice-Boltzmann method (LBM) for every stent placement. Simulations were conducted using the Palabos C++ library [1]. The interest of using LBM instead of more classical CFD approaches lies on the flexibility of the algorithm, which allows high adaptability to complex and diverse geometries, as is the case with blood vessels. Moreover, LBM is highly parallelizable, which increases efficiency. The approach followed by LBM consists in calculating the collision of particles inside a certain domain to simulate the macroscopic behavior of the fluid. This is done through discretization of the Lattice Boltzmann transport equation [8]:

$$\frac{\partial f}{\partial t} = \frac{\partial f}{\partial t} force + \frac{\partial f}{\partial t} diffusion + \frac{\partial f}{\partial t} collision \tag{1}$$

where f_i is the single particle distribution defining the position and momentum of a set (i.e. particle) of molecules at a given time. A D3Q19 lattice model is used in this study, which discretizes the velocity space in 19 distribution functions (18 velocity directions and a velocity for particles at rest) [5].

In this study, since pulmonary arteries are large vessels [12] blood is considered Newtonian with a cinematic viscosity of 3.7 x 10-6 m^2/s and a density of 1080 kg/m^3. For simplification, the pulsatility of the flow is omitted and the artery's walls are considered rigid. The velocity profile at the inlet is a constant Poiseuille profile with an average velocity of 10 cm/s. A zero pressure boundary condition

is set at the outlets. Having a velocity boundary condition for the inlet and constant pressure boundary conditions for outlets will keep the internal mass constant. The fluid-wall interaction is modeled with a Guo off lattice boundary condition. This kind of boundary condition is well suited for curved geometries and models the vessel wall as fluid particles that follow the same collision step as nodes in the domain [7].

3 Experimental results and discussion

The method has been tested with 2 different stenosis cases and compared to the stenting performed by the interventionist. For the comparison, the stent's length and expansion have been measured from X-ray angiography and 3D models of the expanded arteries have been generated as illustrated in Fig.3. Subsequently, the methodology described above has been applied to the 3D model of each artery at pre-operative state. The CFD studies show gradient improvements in relation to the post-operatory gradient for all stenosis cases.

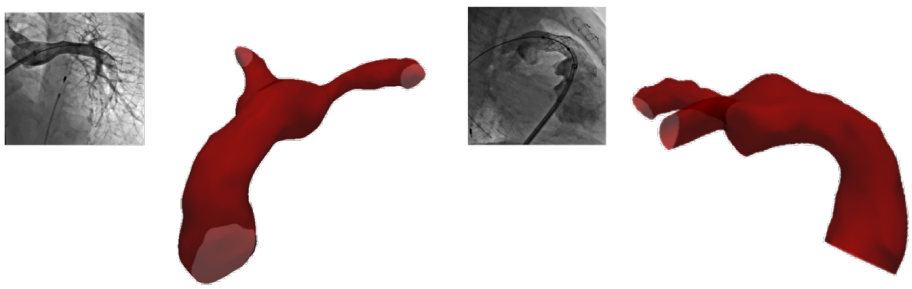

Fig. 3. X-ray angiography of pulmonary artery during stenting, with matching 3D model extracted from CTA.

Table 1. Pressure gradient from CFD simulations, before, after and the optimized solution

Patient	Pre-interv. gradient (mmHg)	Post-interv. gradient (mmHg)	Optimized gradient (mmHg)	Improvement
Patient 1	2.1	2.0	0.6	60.0%
Patient 2	2.1	1.1	1.6	23.0%

As explained in section 1.4, for each expansion diameter a CFD simulation is performed. A sample is presented in Fig.6 for one of the cases. The pressure

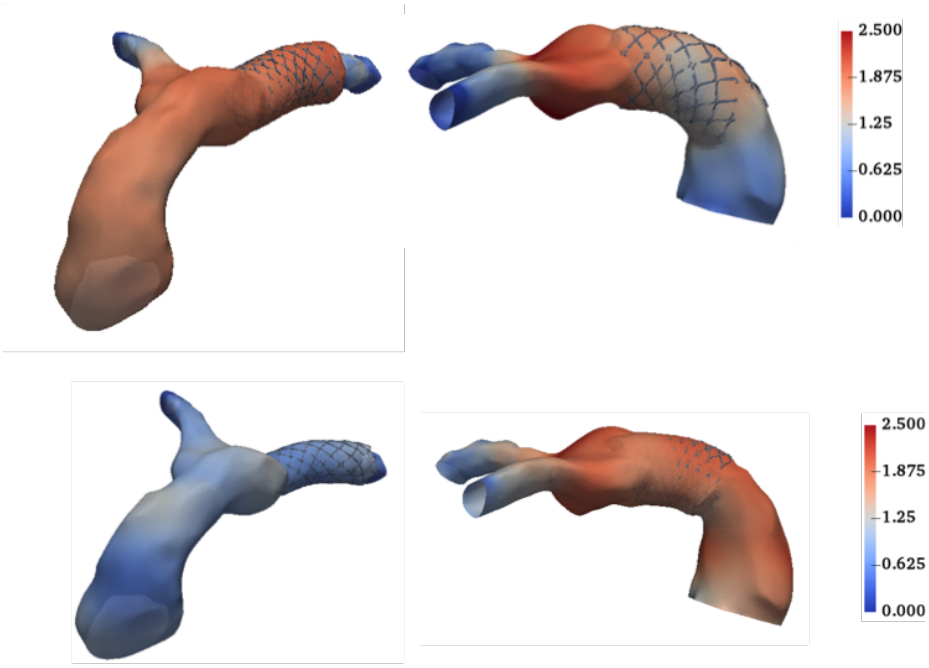

Fig. 4. Visualization of pre- and post-interventional geometries

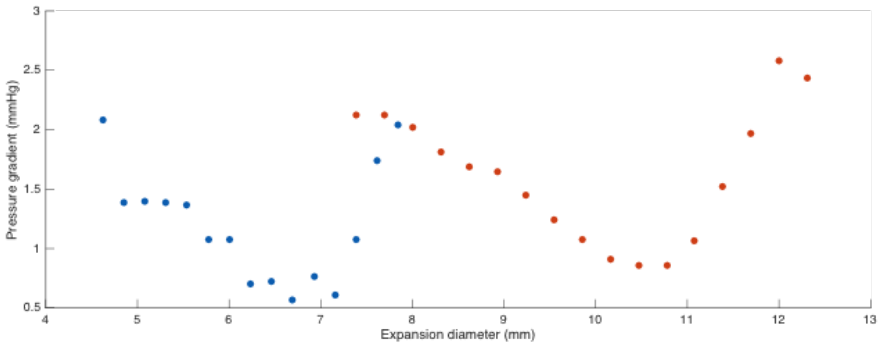

Fig. 5. Pressure gradient at each expansion for 2 patient cases

gradient, in this study, is defined as the average pressure at all points of the inlet minus the pressure at the outlets. The fluid simulations not only allow to determine a minimum pressure gradient, but also enable to observe the impact of stent placement in the patient's hemodynamics. In the present case, it can be observed that, as the gradient is improved locally, the pressure drop concentrates at the beginning of one of the branches, where another case of stenosis is located. It is observed that the pressure gradient is reduced at each expansion step until a minimum is obtained. From this point, further stent expansions imply a growing pressure difference.

In terms of execution, results also show that the number of collision steps to reach stationary state varies significantly depending on the patient and less depending on each expansion. For instance, the number of collision steps to reach stationary state was of 7310 (SD: 1429) and 3957 (SD: 116) for patient 1 and 2 respectively. Stent placement optimization with CFD simulations ranged from 6 to 10 hours for a full pulmonary artery as considered in our work. This may be improved by considering only the location of the stenosis rather than the entire pulmonary artery.

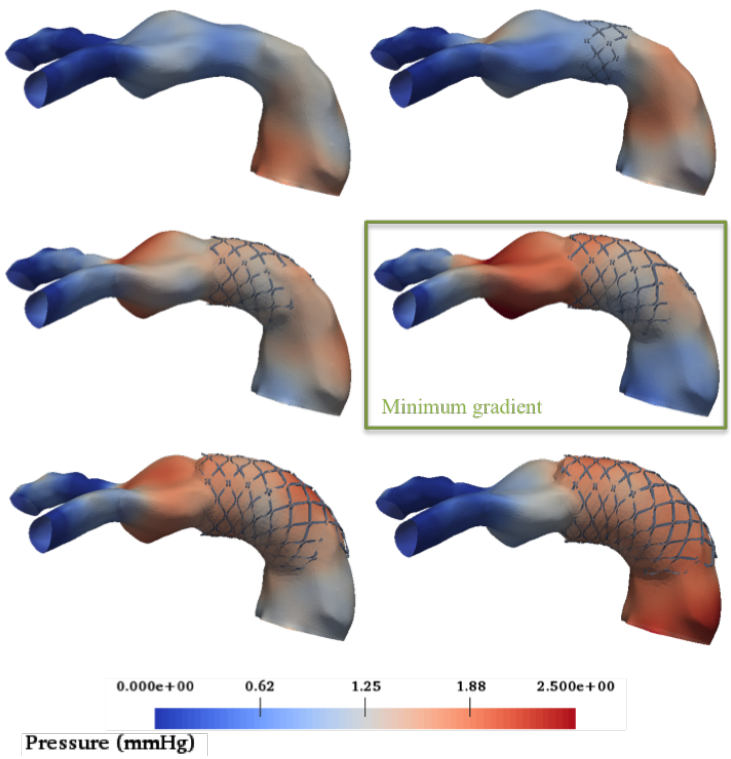

Fig. 6. Simulated stent diameters of 8.0, 9.2, 10.2, 11.1 and 11.7 mm, from left to right and top to bottom

4 Conclusion

This study introduced CFD estimates for optimization of stent fitting. The translesional gradient, used routinely in vivo in the assessment of pulmonary artery stenosis promises as a CFD measurement for the optimization of stent fitting. Other measurements might also be relevant such as FFR and WSS and will be investigated in future works. Further works will also include developing a faster and more accurate 3D segmentation and considering pulsatile flow, the Windkessel effect and the artery temporal deformation associated with cardiorespiratory motion.

Acknowledgments The authors would like to acknowledge the clinical team of Sainte-Justine's catheterization laboratory for their assistance.

References

1. Palabos library. http://www.palabos.org/ (2010–2015)
2. Choi, J., et al.: Pressure distribution and wall shear stress in stenosis and abdominal aortic aneurysm by computational fluid dynamics modeling (cfd). Korean J Chem Eng 31(3), 402–411 (2014)
3. Egger, J., Grosskopf, S., Nimsky, C., Kapur, T., Freisleben, B.: Modeling and visualization techniques for virtual stenting of aneurysms and stenoses. Computerized Medical Imaging and Graphics 36(3), 183 – 203 (2012)
4. Flórez-Valencia, L., Montagnat, J., Orkisz, M.: 3d graphical models for vascular-stent pose simulation. MG&V 13(3), 235–248 (Jan 2004)
5. Goetz, J.: Simulation of bloodflow in aneurysms using the lattice boltzmann method and an adapted data structure. Tech. rep., University of Erlangen-Nuremberg (2006)
6. Gundelwein, L., Miro, J., Duong, L.: Automatic stent placement and stent size selection for preoperative planning of pulmonary artery intervention (2015)
7. Guo, Z., Zheng, C., Shi, B.: An extrapolation method for boundary conditions in lattice boltzmann method. Physics of Fluids 14(6), 2007–2010 (2002)
8. Harris, S.: An introduction to the theory of the Boltzmann equation. Dover publications (2011)
9. Hoffman, J., Kaplan, S.: The incidence of congenital heart disease. Cardiol. 39(12), 1890–900 (2002)
10. Taylor, C.A., Fonte, T.A., Min, J.K.: Computational fluid dynamics applied to cardiac computed tomography for noninvasive quantification of fractional flow reserve: scientific basis. Journal of the American College of Cardiology 61(22), 2233–2241 (2013)
11. Top, A., Hamarneh, G., Abugharbieh, R.: Active learning for interactive 3d image segmentation. In: Proc. MICCAI. pp. 603–610. Springer (2011)
12. Tu, J., Inthavong, K., Wong, K.K.L.: Computational Hemodynamics – Theory, Modelling and Applications. Springer (2015)
13. Zhang, W., Qian, Y., Lin, J., Lv, P., Karunanithi, K., Zeng, M.: Hemodynamic analysis of renal artery stenosis using computational fluid dynamics technology based on unenhanced steady-state free precession magnetic resonance angiography: preliminary results. The International Journal of Cardiovascular Imaging 30(2), 367–375 (2014)