10th International Workshop on Innovative Simulation for Healthcare
18th International Multidisciplinary Modeling & Simulation Multiconference

ISSN 2724-0371 ISBN 978-88-85741-65-2 © 2021 The Authors.
doi: 10.46354/i3m.2021.iwish.002

Influence Of Cannula Angle For Cardiopulmonary Bypass Using Computational Fluid Dynamics

Gionata Fragomeni[1,*]

[1]Department of Medical and Surgical Sciences Magna Graecia University, Campus "S. Venuta", Catanzaro, 88100, ITALY

[*]Corresponding author. Email address: fragomeni@unicz.it

Abstract

Effects of perfusion during extracorporeal circulation with different angle of arterial cannulae on aortic hemodynamics were assessed using a patient-specific Computational fluid dynamics (CFD). Three different models were made, based on the angle of the cannula from 0° to 30° and the effects on the hemodynamics were studied.

Keywords: CFD; Cardiopulmunary Bypass; Cannula

1. Introduction

In the last years cardiac surgery techniques and approaches have had a very important evolution, which has been made possible thanks to the growth of applications of extracorporeal circulation. Cardiopulmunary bypass (CPB) is essential for almost all cardiac surgery. More than 150,000 CABG (Coronary artery Bypass Graft) interventions are performed all over the world every year (Morrow et al. 2008 and Niles et al. 2001).

The main function of cardiopulmonary bypass is to replace the pump action of the heart and to ventilate the lungs, thus allowing perfect perfusion to organs while the heart is stopped to carry out the surgery. To do this, there is a need to connect the patient's circulation with the cardiopulmonary bypass circuit through cannulae. Venous blood is channeled towards the heart-lung machine thanks to the use of a venous cannula, which can be positioned in different locations. Before returning to the patient's circulation via an arterial cannula, the blood passes through pumps, filters and oxygenator.

As for the arterial cannula, it can be positioned in:

- ascending aorta
- supraortic trunk
- axillary artery
- femoral artery
- left ventricular apex

The position of cannulae is of great importance and can affect the success of cardiac surgery. There are a lot of physiological effects that are associated with Cardiopulmonary Bypass, like for example thrombocytopenia, activation of "complement factors", immunosuppressant, inflammatory responses, etc . Manipulation in ascending aorta during cannulation and clamping may induce the formation of embolisms that cause brain damage, for instance (Niles et al. 2001). Therefore, it is advisable to reduce all possible causes of problems to the patient. This work studies the effects of the cannula angle on hemodynamics with respect to the aorta. To do this, Computational Fluid Dynamics (CFD) tools have been used. CFD simulations allow to get some very important information on surgical techniques and on devices in a non-invasive way (Condemi et al 2017, Gaudio et al. 2016, Caruso 2015).

2. Materials and methods

2.1. Geometrical model and Boundary Conditions

A 3D real aorta model was generated from Computer Tomography images using segmentation and reverse engineering techniques (de Moraes et al. 2011, Pham and Hieu 2008) (Figure 1).

The assumption of identical boundary conditions was chosen for the different simulations in order to analyse the only effects of cannula's angle.

A 22 Fr cannula was modeled and placed in ascending aorta as in clinical practice, above the Sino-Tubular junction (Figure 2), perpendicular to the aortic wall.

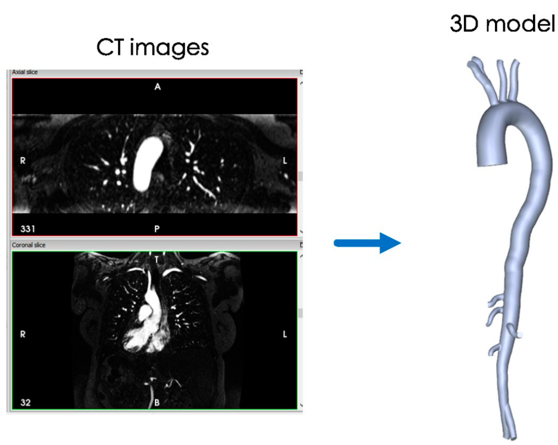

Figure 1. Aorta Model

Three different models were made based on the angle of the cannula (Figure 1):

- Case A: 0° angle;
- Case B: 15° angle;
- Case C: 30° angle.

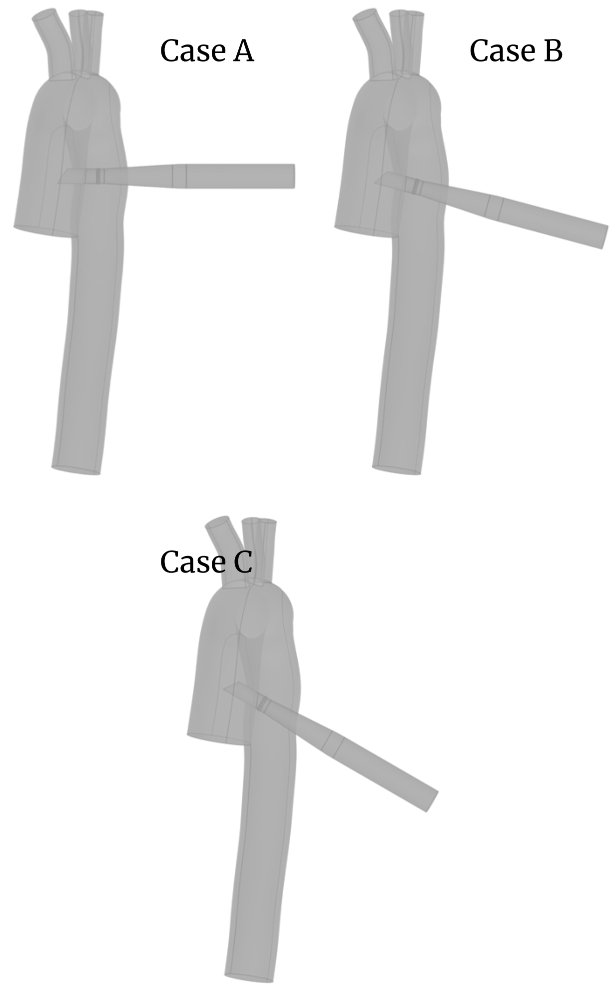

Figure 2. Model geometry, Case A, Case B and Case C

Since the aorta is a large vessel, blood can be approximated as a Newtonian and incompressible fluid, with a density of 1,060 kg/m^3 (Cutnell and Johnson 1998) and a viscosity of 0.0035 Pa·s (Caruso et al. 2016). Moreover, the flow can be numerically described by means of 3D Navier-Stokes equations (Gramigna et al. 2015).

As a boundary condition, the same mean inlet flow of 3 L/min was applied to the cannula in the two cases, whereas zero-pressure conditions were set as outlets in all vessels, as in similar comparative studies (Caruso et al. 2017, Karmonik et al. 2012, Lawford et al. 2008). For the outputs open boundary conditions were chosen. The no-slip boundary condition was applied to the fluid-vessel interface. From the natural entrance of the aorta there is no flow, as it is clamped during the cardiopulmonary bypass procedure.

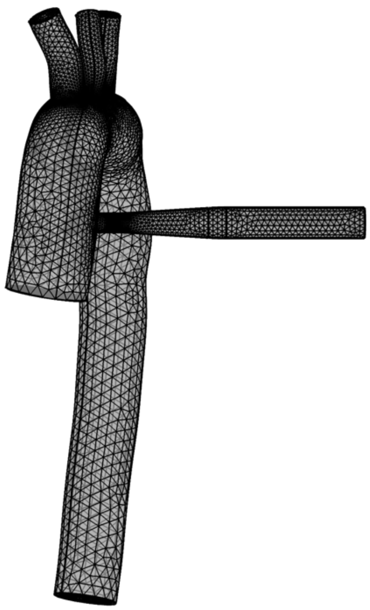

Figure 3. Mesh for case A geometry

2.2. Simulation details

The Computational Fluid Dynamics simulations were carried out using COMSOL 5.6 (COMSOL Inc, Stockholm, Sweden). The meshes had four boundary layers and tetrahedral elements, with a changeable total number of elements according to the geometry was optimized by analyzing the error trend as a function of simulation time (Figure 3).
The Pardiso solver was employed to solve the Navier-Stokes equations, choosing the P1-P1 discretization and a step of 0.001. The flow was considered as laminar, and 3D Navier-Stokes equations were used as governing laws (Gramigna 2015).
The incompressible condition gives:

$$\nabla . u = 0 \qquad (1)$$

The governing equation used to solve the laminar model is:

$$\rho \frac{\delta u}{\delta t} + \rho(u.\nabla)\mathrm{u} = \nabla.\{-pI + \mu[\nabla \mathrm{u} + (\nabla \mathrm{u})^T]\} \qquad (2)$$

where ρ is the fluid density, u is the fluid velocity, p is the pressure, I is the unit diagonal matrix and μ is the viscosity (Caruso 2016).

3. Results and Discussion

CFD analyses were performed in order to obtain the numerical results affecting blood velocity, starting from Finite Elements Models of aorta reconstructed geometries.

Table 1 shows the flow trend in the thoracic aorta. It can be seen that as the angle changes, the flow decreases.

Table 1. thoracic flow [%]

Case #	Thoracic Flow [%]
Case A	32,0
Case B	31,6
Case C	29,4

The same considerations can be made by referring to figure 4. Moreover, from the analysis of the streamlines, it can be seen that in Case A there is a stagnation area near the clamping area, whereas the angle variation produced an orderly flow pattern in the supra-aortic vessels.

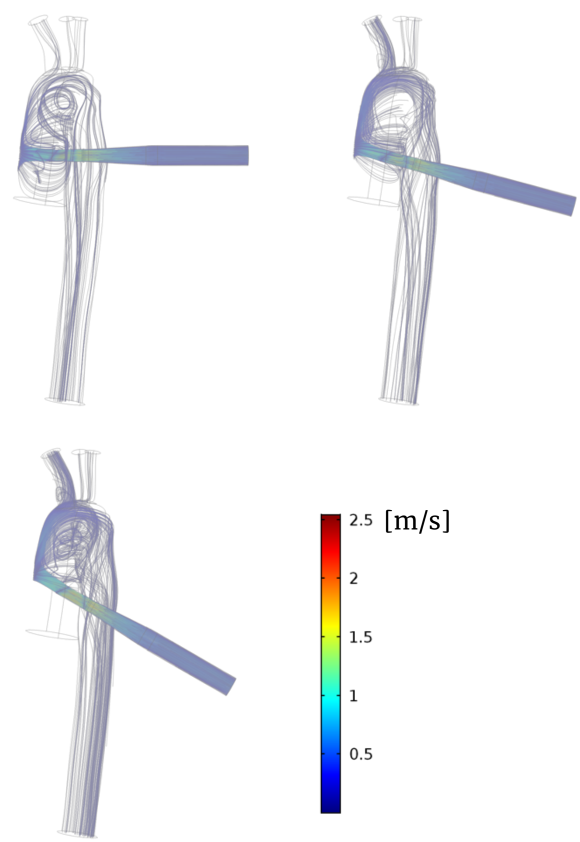

Figure 4. Streamlines for Case A, Case B and Case C

Furthermore, results highlight that the cannula can generate a swirling flow with stasis in ascending aorta downstream anastomosis and flow separation in supra-aortic vessels, whereas the flow from the cannula produces recirculation in ascending aorta and orderly pattern in supra-aortic vessels.

4. Conclusions

In conclusion, this non-invasive assessment demonstrates how the insertion angle of the cannula causes significant changes in aortic hemodynamics during Cardiopulmunary bypass. The position of the cannula seems to generate a steadier flow pattern with good recirculation also in the ascending aorta. On the other hand, even though it seems potentially more thrombo-genic offered an enhanced flow output selectively to the supra-aortic vessels.

References

Caruso, M. V., De Rosa, S., Indolfi, C., & Fragomeni, G. (2015). Computational analysis of stenosis geometry effects on right coronary hemodynamics. Paper presented at the *Proceedings of the Annual International Conference of the IEEE Engineering in Medicine and Biology Society, EMBS, 2015-November* 981-984. doi:10.1109/EMBC.2015.7318528

Caruso, M. V., Gramigna, V., Renzulli, A., Fragomeni, G. (2016). Computational analysis of aortic hemodynamics during total and partial extra-corporeal membrane oxygenation and intra-aortic balloon pump support. Acta of Bioengineering and Biomechanics, 18(3), 3-9.

Caruso, M. V., Renzulli, A., Fragomeni, G. (2017). Influence of IABP-Induced Abdominal Occlusions on Aortic Hemodynamics: A Patient-Specific Computational Evaluation. ASAIO Journal, 63(2), 161-167.

Condemi, F., Wang, D., Fragomeni, G., Yang, F., Zhao, G., Jones, C., Zwischenberger, J. B. (2016). Percutaneous double lumen cannula for right ventricle assist device system: A computational fluid dynamics study. *Biocybernetics and Biomedical Engineering, 36*(3), 482-490. doi:10.1016/j.bbe.2016.04.002

Cutnell J. D. and Johnson, K.W., 1998. Physics. Wiley, New York.

de Moraes T.F., Amorim P.H., Azevedo F.S., da Silva J.V., 2011. InVesalius: An open-source imaging application. Computational Vision and Medical Image Processing: VipIMAGE 2011, 405.

Gaudio, L. T., De Rosa, S., Indolfi, C., Caruso, M. V., Fragomeni, G. (2017). Hemodynamically non-significant coronary artery stenosis: A predictive model. Paper presented at the 6th International Workshop on Innovative Simulation for Health Care, IWISH 2017, Held at the International Multidisciplinary Modeling and Simulation Multiconference, I3M 2017, 26-30.

Gramigna, V., Caruso, M. V., Rossi, M., Serraino, G. F., Renzulli, A., & Fragomeni, G. (2015). A numerical analysis of the aortic blood flow pattern during pulsed cardiopulmonary bypass. Computer methods in biomechanics and biomedical engineering, 18(14), 1574-1581.

Karmonik C, Partovi S, Loebe M, et al: Influence of LVAD cannula outflow tract location on hemodynamics in the ascending aorta: A patient-specific computational fluid dynamics approach. ASAIO J 58: 562–567, 2012

Lawford P.V., Ventikos Y., Khir A.W., Atherton M., Evans D., Hose, D.R., ... Hollis A.P ., 2008. Modelling the interaction of haemodynamics and the artery wall: current status and future prospects. Biomedicine & Pharmacotherapy, 62(8), 530-535.

Morrow DA, Gersh BJ. Chronic coronary artery disease. Libby P, Brawnwald E, editors. Brownwald's heart disease: a textbook of cardiovascular medicine. Philadelphia: saunders/ Elsevier; 2008. Pp. 1353-1417.

Niles NM, Mc Grath PD, Malenka D, et al. Survival of patients with diabetes and multivessel coronary artery disease after surgical or percutaneous coronary rivascularization: results of a large regional prospective study. Northern New England Cardiovascular Disease Study Group. J Am Coll Cardiol 2001; 37: 1008- 1015.

Pham D.T. and Hieu L.C., 2008. Reverse engineering hardware and software. Reverse Engineering, 33-70.