

Computational Fluid Dynamic Analysis to Prevent Aortic Root and Valve Clots during Left Ventricular Assist Device Support

Michael Poullis, BSc(Hons), MBBS, MIEEE, FRCS(Cth)

Liverpool Heart and Chest Hospital, Liverpool, United Kingdom

Presented at the 50th International Conference of the American Society of Extracorporeal Technology, Orlando, Florida, March 30, 2012.

Abstract: Aortic root and valve clots are rare but well described in patients on maximal left ventricular assist device (LVAD) support. We performed a theoretical analysis using computational fluid dynamic analyses in two dimensions to try and ascertain if inflow cannula design/orientation/placement affect aortic root flow dynamics. Two-dimensional computational fluid dynamics using easy CFD-G was performed. The effect of a curved inflow cannula, a straight cannula, and one with a hole in the outer curve was analyzed. In addition, the effect of inflow conduit angulation on the ascending aorta was studied. Computational fluid dynamic (CFD) analysis predicts that stagnant blood exists in the aortic root when little or no cardiac ejection is taking place. Coronary flow is too small to

affect the root flow streamlines. A hole on the root side of a curved inflow aortic cannula increases the flow in the aortic root and may decrease the incidence of root and valve thrombosis. The angle of the inflow conduit attachment to the ascending aorta was also found to be crucial with regard to aortic root blood stasis. In addition, a baffle at the tip of the inflow cannula may prove to be beneficial. Theoretical analysis using the technique of CFD predicts that inflow cannula position and design may affect the incidence of aortic root thrombosis during LVAD support when minimal cardiac ejection is occurring. **Keywords:** cardiopulmonary bypass, mechanical support, computational fluid dynamics, thrombosis. *JECT. 2012;44:210–215*

Aortic root and valve clots are well described in patients on left ventricular assist device (LVAD) support (1–4). The reduced flow in the ascending aorta is usually thought to be the major contributing factor responsible for this. The presence of a mechanical valve and heparin-induced thrombocytopenia probably increases the risk but these are not proven etiological factors.

Computational fluid dynamic (CFD) analysis was introduced by the National Aeronautics and Space Administration in the early 1960s to avoid expensive model testing (5). Modern computer power has revolutionized this area and uses concepts proposed by some of

the most famous names in science: Archimedes, Leonardo da Vinci, Newton, Bernoulli, Euler, Navier, Stokes, Reynolds, Prandtl, and Von Karman.

If the aortic geometry, aortic flow rates, blood viscosity, and temperature are known, then CFD is able to calculate flow patterns, pressures, and wall shear stress within the aorta. This enables multiple different scenarios and solutions to be analyzed very quickly without resorting to in vitro or vivo testing.

We used CFD analysis in two dimensions to try to ascertain if inflow cannula design, orientation, and position are important factors predisposing to root thrombosis. In addition, we try to demonstrate the role of CFD analysis to provide a possible solution to the problem.

METHODS

Basic Computational Fluid Dynamic Model

A normal computed tomographic (CT) scan was used to construct a two-dimension model of the root,

Received for publication March 21, 2012; accepted September 20, 2012.
Address correspondence to: Michael Poullis, BSc(Hons), MBBS, MIEEE, FRCS(Cth), Consultant Cardiothoracic Surgeon, Liverpool Heart and Chest Hospital, Thomas Drive, Liverpool, L14 3PE, UK. E-mail: mike.poullis@lhch.nhs.uk

The senior author has stated that authors have reported no material, financial, or other relationship with any healthcare-related business or other entity whose products or services are discussed in this paper.

ascending, and proximal arch of a hypothetical patient. Flow through the LVAD inflow cannula was assumed to be 5 L/min. The pressure within the aortic root was assumed to be 80 mmHg.

Scenarios Studied

Various scenarios were modeled for (Figure 1):

1. Height of inflow cannula above the aortic valve; and
2. Angle of inflow cannula to ascending aorta.

All analyses had velocity measured in m/sec.

Widget

In this article, a widget is defined as a geometric object placed or part of the inflow of the inflow cannula and the aorta.

Solutions Studied

Various possible solutions were modeled for (Figure 2) an inflow widget at level of inflow cannula and aortic wall (Figure 3), a modified curved inflow cannula, double inflow cannula, and pulsatile flow. The results of the CFD analysis are shown in Figure 4.

Computational Fluid Dynamics

The description of the blood flow can be derived from the time-dependent, two-dimensional, incompressible

Navier-Stokes equations (6). This system of equations was provided with boundary and initial conditions. At the flow entrance, fully developed velocity profiles were assumed. The arterial wall was considered to be rigid and the no-slip condition was applied. The numerical solution of the Navier-Stokes equations was performed by using EasyCFD-G (University of Coimbra, Coimbra, Portugal). EasyCFD-G software has previously been validated against Fluent (7).

Boundary Conditions: For a CFD study to be performed boundary, inlet and outlet conditions need to be defined. Steady boundary conditions were used with a flow rate of 5 L/min.

Computational Fluid Dynamic Settings: Blood was modeled as an incompressible Newtonian fluid (a common assumption in the medical literature) with a density of 1060 kg/m^3 , laminar Prandtl 25, and a dynamic viscosity of 25 Ns/m^2 .

Computational Fluid Dynamic Mesh: To analyze a given situation, CFD breaks down the geometric shape into little boxes (discrete cells). Together the boxes look like a mesh. The more boxes, for a given geometry, the smaller they are, and thus the more accurate the analysis is. Increased accuracy comes at the price of increased

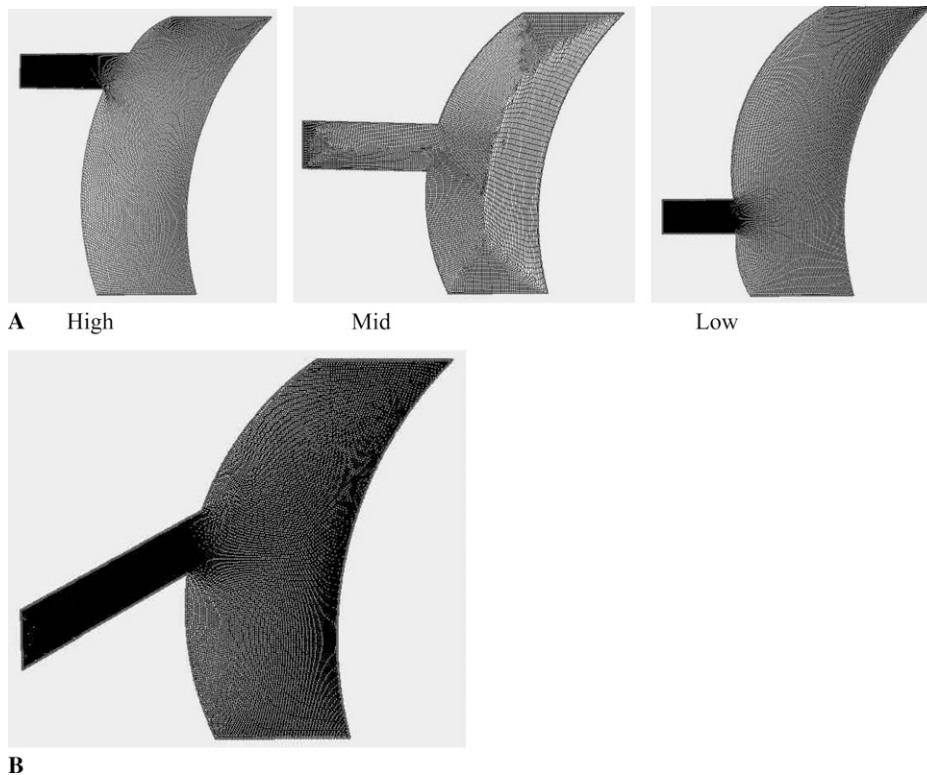


Figure 1. Scenarios modeled to determine factors affecting aortic root thrombosis: (A) height of inflow cannula above the aortic valve; (B) angle of inflow cannula to ascending aorta.

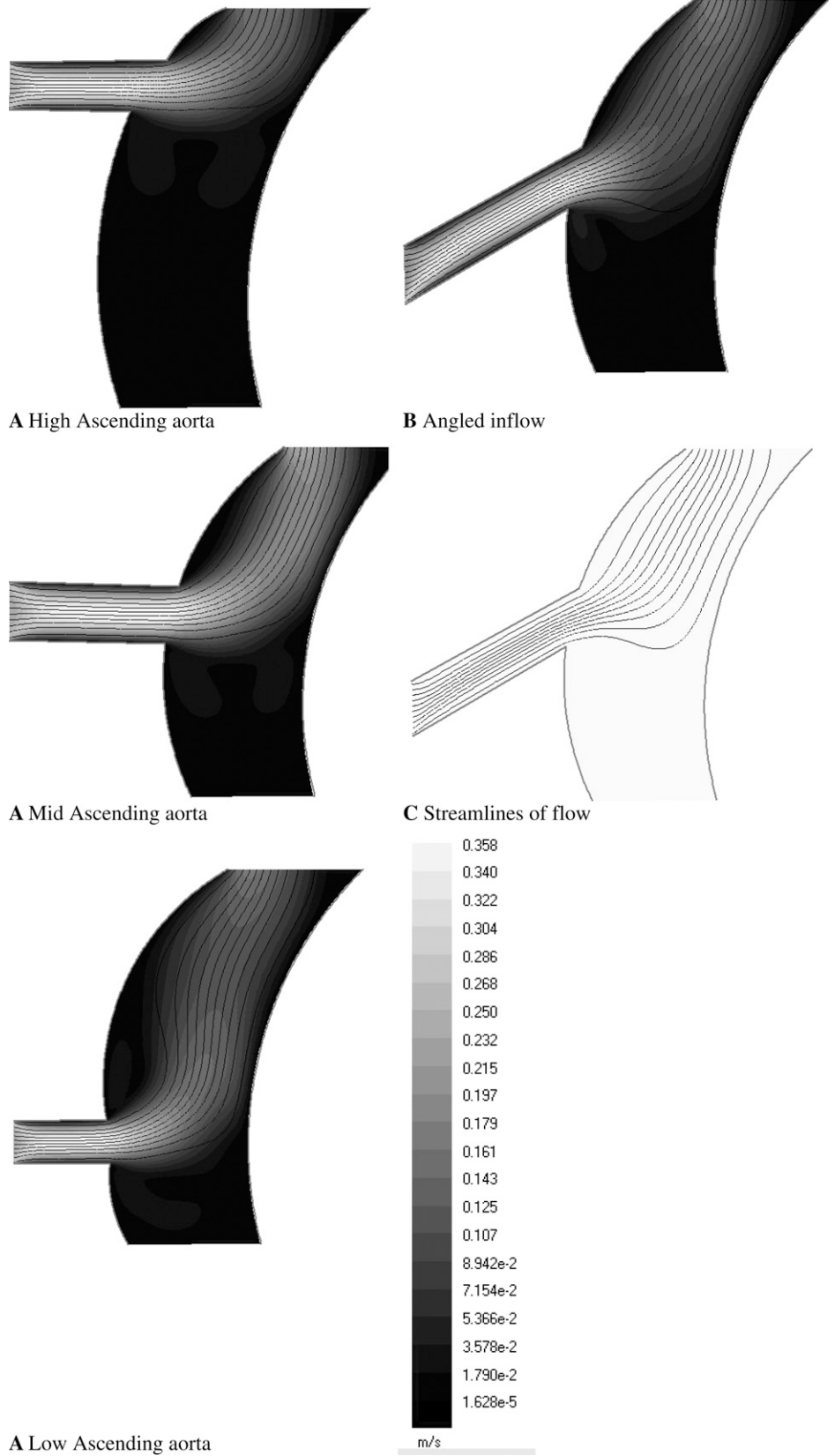


Figure 2. Scenarios modeled to determine possible solutions to aortic root thrombosis: (A) height of cannula (high, mid, low) above the aortic valve; (B) angled inflow cannula relative to the ascending aorta, velocity profile of flow; (C) angled inflow cannula relative to the ascending aorta, streamlines of flow demonstrated.

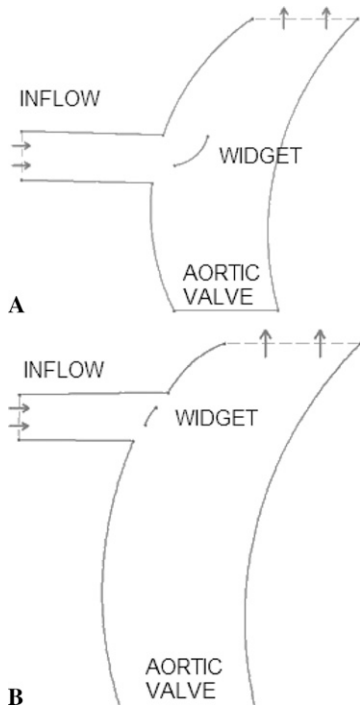


Figure 3. Widget used in this article: (A) curved portion of indwelling cannula; (B) baffle at level of aortic wall.

computational power needed to solve the mesh. A compound mesh, typically between 10,000 and 30,000 elements, was used and simulation was continued with EasyCFD-G until convergence criteria were met.

For all CFD analysis, flow was in m^3/s , and velocity was in m/sec .

RESULTS

Aortic Flow

Typical aortic flow velocity is in the region of .1 m/sec . This figure is highly variable secondary to the large variation in aortic geometry, cardiac output, and the pulsatile nature of the flow. It can be seen from all the figures that an aortic cannula produces a jet effect with very high flow. This is the result of the same flow as normal cardiac output but a very much reduced cross-sectional area of flow, i.e., cannula cross-sectional area. In all the figures, the lower the velocity of flow within the aorta, the darker the color representing it.

Aortic Valve Flow

This was assumed to be zero for all the scenarios modeled; we have modeled the worst case scenario. The higher the flow, the less the chance of aortic valve thrombosis.

Scenarios Studied

Figure 3 demonstrates the effect of height of inflow cannula above the aortic valve, angle of inflow cannula to ascending aorta, and the effect of a straight or curved inflow cannula.

Height of Inflow Cannula Above the Aortic Valve

It can be seen that little flow occurs below a horizontal inflow path. The lower the inflow cannula height above the aortic valve, the more flow in the root resulting from mixing and frictional drag within blood.

Angle of Inflow Cannula to Ascending Aorta

It can be seen that an oblique angle for blood inflow into the ascending aorta is associated with aortic root blood stasis.

Solutions Studied

Figure 4 demonstrates the effect of an inflow widget at the level of the inflow cannula and aortic wall, a modified curved inflow cannula, and the use of a double inflow cannula on root thrombosis.

An Inflow Widget at the Level of Inflow Cannula and Aortic Wall

The geometry of the widget determines the flow streamlines in the ascending aorta. It can be seen that splitting on the inflow so that recirculation of the relative blood occurs in the root reduces root stasis.

Modified Curved Inflow Cannula

A fenestration in the curved indwelling portion of the aortic cannula results in better recirculation in the aortic root secondary to splitting the inflow stream.

Double Inflow Cannula

If a smaller tube graft such as a saphenous vein is anastomosed low in the aortic root, recirculation in the root occurs reducing stasis and hence risk of thrombosis.

Pulsatile Flow

Pulsatile flow is difficult to model accurately secondary to the unpredictable nature of turbulence. Pulsatile flow produces complex flows in the aortic root that are associated with less stasis (higher velocity flows) (Figure 5).

DISCUSSION

Theoretical modeling through the technique of CFD analysis indicates that cannula position, angulation, and geometry all contribute to varying flow profiles in the aortic root and potentially the risk of aortic valve and root thrombosis during LVAD support. A number of options exist to reduce the incidence of thrombosis; however,

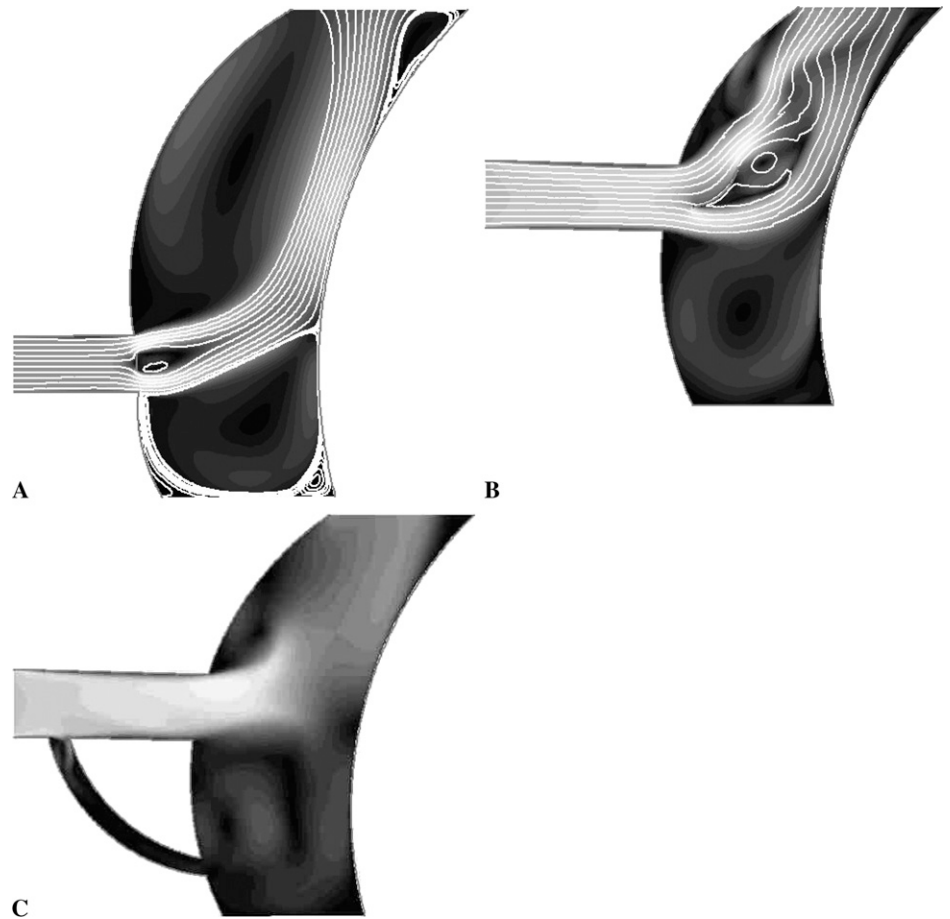


Figure 4. Computational fluid dynamic (CFD) solutions: (A) an inflow widget at level of inflow cannula and aortic wall; (B) modified curved inflow cannula; and (C) double inflow cannula. White lines demonstrate flow streamlines.

some may not be practical in the clinical setting. If possible the cannula should be as low as possible. A widget at the level of the inflow aortic wall junction may be of benefit to direct blood toward the aortic root to prevent stasis. If an indwelling cannula is used, a curved one with a fenestration on the outer curvature should be used. A very simple possible solution is to use a vein graft anastomosis very low on the aortic root.

Virchow's triad (flow, thrombogenicity, and hypercoagulability) is central to our understanding of root thrombosis during LVAD support (8). Unfortunately, quantification of the risk of thrombosis secondary to stasis and relative stasis is impossible. CFD analysis is unable to model for the effect of anticoagulation status, hematocrit, and platelet function.

Cannulation of the left ventricle apex will reduce the chance of aortic valve and root thrombosis to a minimum; however, this is not always practical and may paradoxically increase the risk of left ventricular and atrial clots secondary to stasis.

Pulsatile flow is known to increase the incidence of aortic valve opening and closing, potentially reducing the chance

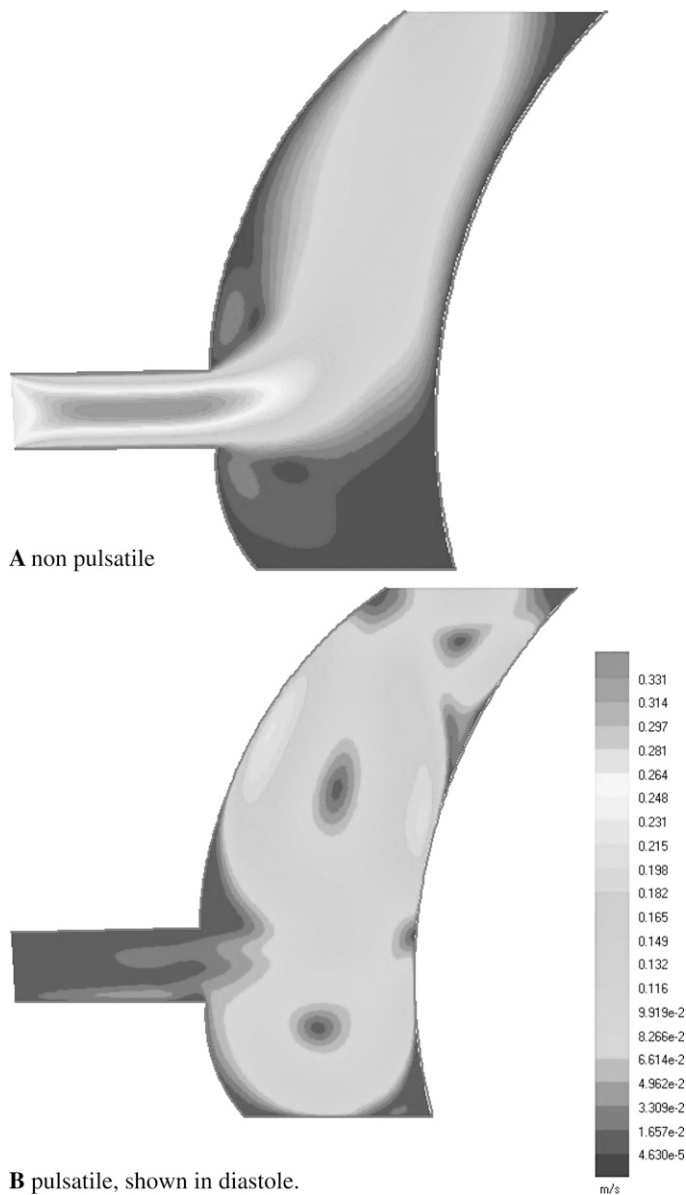
of thrombus formation (9). Turbulent flow secondary to pulsatile flow may be advantageous in the aortic root.

CFD analysis has previously been used to determine the optimum position for inflow cannula position on the ascending aorta to reduce the incidence of cerebral embolism during cardiopulmonary bypass; however, they did not study root stasis (10).

A widget at the level of the inflow cannula and aortic wall alters the flow streamlines in the root. The exact geometry is not important in this concept paper because in clinical practice, a three-dimensional widget would need to be used.

Our analysis has been in two dimensions so as to simplify analysis but presents relevant concepts. Should this concept be applied in clinical practice, the unique patient's aortic geometry would need to be imported from their CT scan, the flow from the left anterior descending would need to be known, and a three-dimensional analysis would have to be performed: patient-directed perfusion. In the current world this can all be performed on a laptop; however, calculations may take in excess of 60 minutes.

Color Doppler on transesophageal or transthoracic echocardiography is a simple technique that is readily available



A non pulsatile

B pulsatile, shown in diastole.

Figure 5. Demonstration of the effect of pulsatile flow on aortic root flow: (A) nonpulsatile; (B) pulsatile.

to clinicians to evaluate the turbulence in the aortic root to confirm or refute the CFD findings or predictions.

Aortic root and valve thrombosis may not be completely preventable during LVAD support as a result of the complex nature that exists in this difficult patient group. CFD analysis may assist in determining the optimal configurations for a LVAD cannula that may reduce the incidence of clot formation if applied on a case-by-case basis. Pulsatile inflow is an immediate easily adoptable partial solution.

We present this article as a concept.

REFERENCES

1. Crestanello JA, Orsinelli DA, Firstenberg MS, Sai-Sudhakar C. Aortic valve thrombosis after implantation of temporary left ventricular assist device. *Interact Cardiovasc Thorac Surg.* 2009;8:661–2.
2. Freed BH, Jeevanandam V, Jolly N. Aortic root and valve thrombosis after implantation of a left ventricular assist device. *J Invasive Cardiol.* 2011;23:E63–5.
3. Nawata K, Nishimura T, Kyo S, et al. Thrombotic occlusion of the ascending aorta after Toyobo LVAD implantation with descending aortic perfusion in a post-Bentall patient. *J Artif Organs.* 2010;13:228–31.
4. Rose AG, Connelly JH, Park SJ, Frazier OH, Miller LW, Ormaza S. Total left ventricular outflow tract obstruction due to left ventricular assist device-induced sub-aortic thrombosis in 2 patients with aortic valve bioprosthesis. *J Heart Lung Transplant.* 2003;22:594–9.
5. Malkawi A, Augenbroe G. *Advanced Building Simulation.* New York, NY: Routledge; 2004.
6. Axner L, Hoekstra AG, Jeays A, Lawford P, Hose R, Sloot PM. Simulations of time harmonic blood flow in the mesenteric artery: comparing finite element and lattice Boltzmann methods. *Biomed Eng Online.* 2009;8:23.
7. Ting TS, Prakash M, Cleary PW, Thompson MC. Simulation of high Reynolds number flow over a backward facing step using SPH. *ANZIAM J.* 2006;47:C292–309.
8. Bagot CN, Arya R. Virchow and his triad: A question of attribution. *Br J Haematol.* 2008;143:180–90.
9. Myers TJ, Bolmers M, Gregoric ID, Kar B, Frazier OH. Assessment of arterial blood pressure during support with an axial flow left ventricular assist device. *J Heart Lung Transplant.* 2009;28:423–7.
10. Osorio AF, Osorio R, Ceballos A, et al. Computational fluid dynamics analysis of surgical adjustment of left ventricular assist device implantation to minimise stroke risk. *Comput Methods Biomech Biomed Engin.* 2011 Dec 21 [Epub ahead of print].