Computational Fluid Dynamics (CFD) simulation of flow in Micro-Arterial Anastomoses

Adam Kyte[1], Matthew Sharman[1], Christopher Pass[1] and Rory Rickard[2]

1 – DesignFlow Consultancy and Research Group, School of Engineering, Plymouth University
2 – Derriford Hospital Plymouth/Royal Centre for Defence Medicine, Queen Elizabeth Hospital Birmingham

Introduction

Numerical modelling of vascular flow using Computational Fluid Dynamics (CFD) permits detailed analysis of a wide range flow phenomena. However, flaws in CFD studies all too often lead to flawed conclusions.

The current study of anastomosed vessels (diameter approx. 1mm) with size mismatch investigates simulation sensitivity to a range of CFD assumptions. It aims to develop a methodology for reliably predicting the nature of flow in such cases.

Earlier studies by Rickard *et al.* [1,2] produced a rodent model for analysis of anastomosis techniques (Fig. 1); a resin cast of an anastomosed artery from these studies was used to produce a micro-CT scanned digital geometry for CFD input. Fig. 2 shows flow streamlines from the resulting CFD analysis.

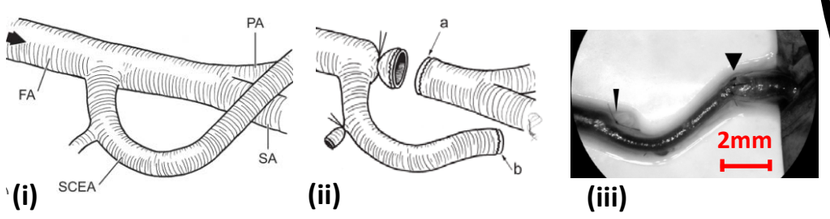

Fig. 1 – Rodent anastomosis model (from Rickard *et al.* [1])
(i) Anatomy of the distal femoral artery
(ii) View prior to anastomosing (a) to (b).
(iii) A completed anastomosis (small arrow: tie around FA; large arrow: sutured anastomosis SCEA to FA)
Key: FA: femoral artery; SCEA: superficial caudal epigastric artery; PA: popliteal artery; SA: saphenous artery.

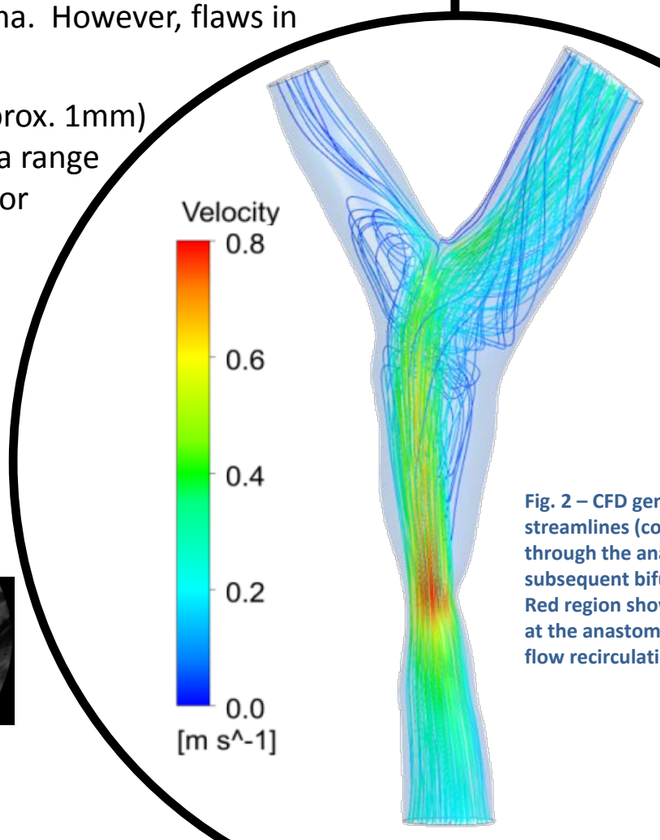

Fig. 2 – CFD generated flow streamlines (coloured with velocity) through the anastomosed section and subsequent bifurcation.
Red region shows accelerated flow at the anastomosis; dark blue shows flow recirculation at the bifurcation

What can CFD potentially tell us?

Microsurgical auto-transplantation of tissues is employed clinically to reconstruct defects following burns, trauma and surgical cancer ablation, and to correct congenital abnormalities. Where anastomoses with large (≈ 3:1) diameter mismatch are necessary, patency rates may be poor. Flow separation and recirculation result in low wall shear stresses, increasing the likelihood of thrombogenesis.

CFD can provide flow visualisation (animations and images such as that in Fig 2) to show flow phenomena such as recirculation. Simulations can also provide valuable metrics for quantitative analysis, allowing comparison between anastomosis techniques.

Wall Shear Stress (WSS), Oscillatory Shear Index (OSI) and Relative Residence Time (RRT) can be calculated as indicators of likelihood of thrombogenesis. Links between these metrics and the generation of vascular disease are widely documented.

CFD results can also be used to highlight regions of high shear stress, in which red blood cell damage can occur.

A key advantage of CFD over experimental approaches is the fact that all flow properties (velocities, pressures, shear rates, etc.) are inherently calculated throughout the entire simulated domain.

Validation Case

Accuracy of CFD simulations are dependent on many factors, some of which are discussed below. The US Food and Drug Administration (FDA) conducted a study [3] in which 28 CFD analysts conducted simulations to compare with experimental results for an idealised geometry (Fig. 3 - a gradual contraction and sudden expansion) featuring flow phenomena typically seen in medical devices (stents, cannulas, etc.). Such phenomena also occur in anastomoses with size mismatch. They considered laminar, transitional and turbulent flows, the laminar case being closest to that interest here.

It is clear from Fig. 4 that the CFD techniques chosen by the participants had a large influence on the results. There was no particular correlation between self-rated expertise of the participants and simulation accuracy. Factors such as turbulence/transition modelling, mesh definition and inlet/outlet boundary conditions were all shown to affect results. Figs 5 & 6 show results from a simulation run as part of the current study, which correlates well with experiment.

This validation case gives us some confidence in our ability to reliably simulate flows in organic geometries containing similar flow features (flow separation/recirculation, jet diffusion, developing flow). More work is now required in other areas (diverging nozzle flow, pulsatile flow, non-Newtonian flow and compliant wall cases).

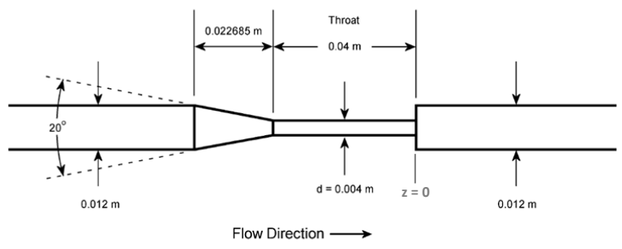

Fig. 3 – The idealised geometry of the FDA's first Computational Inter-laboratory Study (from Stewart *et al.* [3])

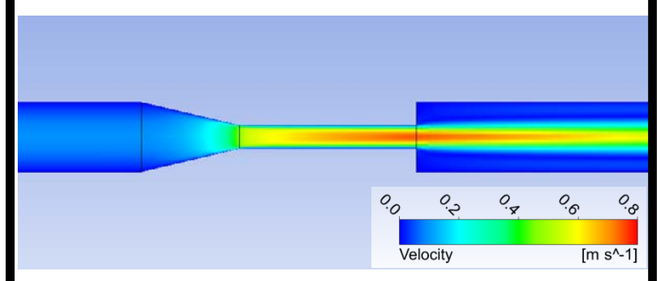

Fig. 5 – CFD velocity prediction (current study) showing flow acceleration in the throat and subsequent jet diffusion

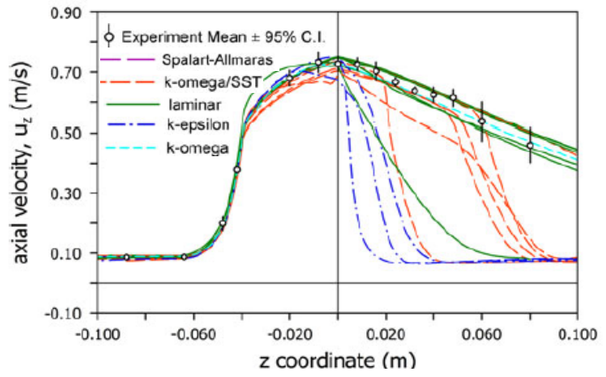

Fig. 4 – Results from the FDA's first Computational Inter-laboratory Study: Axial velocity at the centreline for Re_{throat} = 500 (from Stewart *et al.* [3])

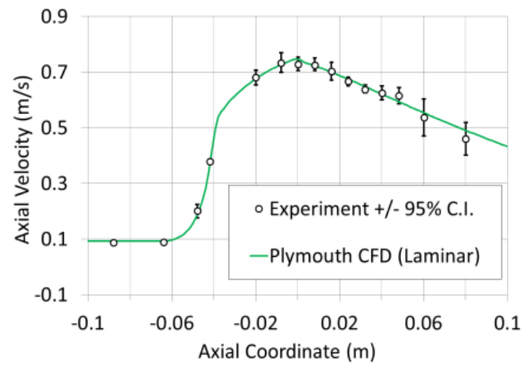

Fig. 6 – Comparison between current study and FDA experimental results: Axial velocity at the centreline for Re_{throat} = 500

Boundary Conditions

The computational expense of simulating all downstream vasculature is prohibitive; instead downstream flow can be represented using a mathematical function based on the electrical analogy (e.g. Fig.7). Simulations are only as accurate as the resistances and capacitances assumed.

Inlet conditions are typically mathematically defined velocity profiles, such as laminar (Poiseuille) flow or transient (Womersley) flow.

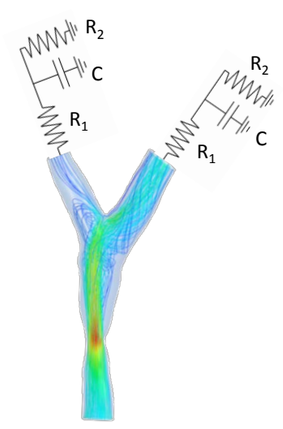

Fig. 7 – The commonly used Windkessel boundary condition applies resistances and capacitances to represent downstream vasculature

Transient Behaviour

The flow of blood is never in steady state; it is transient, ever changing with pulse, respiration, and acute physiological changes.

Assuming a steady flow, for example at peak systole, avoids the considerable additional computational expense of running fully transient simulations but can result in non-physical flow predictions due to neglecting inertial effects of acceleration and deceleration. A transient CFD solver is necessary to overcome this.

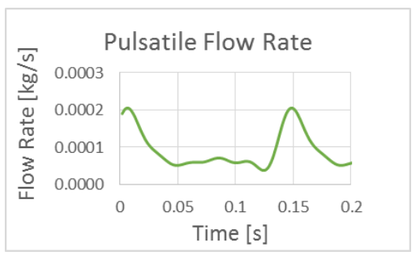

Fig. 8 – The experimentally measured transient flow rate used in [2]

Mesh Definition

CFD reliability depends on the type, quality and resolution of the mesh elements used. Hexahedral, tetrahedral or triangular prismatic elements are commonly used. Tetrahedral meshes are relatively simple to generate but suffer from numerical diffusion (non-physical "smearing" of flow properties). Hexahedral meshes can be more aligned with flow direction (reducing numerical diffusion) and are more computationally efficient.

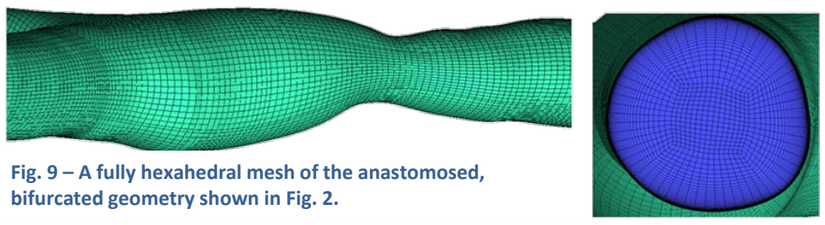

Fig. 9 – A fully hexahedral mesh of the anastomosed, bifurcated geometry shown in Fig. 2.

Turbulence and Transition

Turbulence may occur in large vessels (such as the aorta) or moderately sized vessels with non-natural features (e.g. surgically introduced sudden expansions). In such cases, numerical modelling of turbulence (and its transition from laminar flow) become important. At the (≈ 1mm) vessel size of interest here, flow will be fully laminar. Fig. 4 shows the detrimental effects of inappropriate turbulence modelling in such cases.

Fluid Properties

Blood is non-Newtonian (shear-thinning) and thixotropic, presenting challenges in numerical modelling. Literature suggests that the assumption of Newtonian properties is adequate in relatively straight, uniform diameter vessels above approximately 0.5mm diameter, but in irregular geometries (stenoses, aneurysms, anastomoses, etc.), non-Newtonian effects may be significant for larger vessels.

Wall Behaviour & Compliance

Vessel walls are compliant, orthotropic and non-linear. Physiological factors (e.g. biochemical/neurological response to exercise) also affect shape and structural response. Fluid-Structure Interaction (FSI) methods couple CFD and structural solvers (at great computational expense) to capture this, but accuracy depends on assumed structural properties. We hope to develop new compliant wall modelling methods in future work.

DESIGNFLOW WITH PLYMOUTH UNIVERSITY

References:
[1] R. F. Rickard, J. Wilson and D. A. Hudson, "Characterization of a rodent model for the study of arterial microanastomoses with size discrepancy (small-to-large)," *Laboratory Animals,* vol. 43, no. 4, pp. 350-356, 2009.
[2] R. F. Rickard, C. Meyer and D. A. Hudson, "Computational modeling of microarterial anastomoses with size discrepancy (small-to-large).," *Journal of Surgical Research,* vol. 153, no. 1, pp. 1-11, 2009.
[3] S. F. C. Stewart, E. G. Paterson, G. W. Burgreen, P. Hariharan, M. Giarra, V. Reddy, S. W. Day, K. B. Manning, S. Deutsch, M. R. Berman, M. R. Myers and R. A. Malinauskas, "Assessment of CFD Performance in Simulations of an Idealized Medical Device - Results of FDA's First Computational Interlaboratory Study," *Cardiovascular Engineering and Technology,* vol. 3, no. 2, pp. 139-160, 2012.

Adam Kyte – Lecturer in Mechanical & Marine Engineering Design
Plymouth University (Reynolds Building)
Drake Circus, Plymouth, Devon, PL4 8AA.
+44 (0) 1752 586116
adam.kyte@plymouth.ac.uk