Review

ASIAN CARDIOVASCULAR & THORACIC ANNALS

Asian Cardiovascular & Thoracic Annals
0(0) 1–13

DOI: 10.1177/0218492320957163
journals.sagepub.com/home/aan
SAGE

# Computational fluid dynamics: a primer for congenital heart disease clinicians

Rabin Gerrah[1] and Stephen J Haller[2]

## Abstract

Computational fluid dynamics has become an important tool for studying blood flow dynamics. As an in-silico collection of methods, computational fluid dynamics is noninvasive and provides numerical values for the most important parameters of blood flow, such as velocity and pressure that are crucial in hemodynamic studies. In this primer, we briefly explain the basic theory and workflow of the two most commonly applied computational fluid dynamics techniques used in the congenital heart disease literature: the finite element method and the finite volume method. We define important terminology and include specific examples of how using these methods can answer important clinical questions in congenital cardiac surgery planning and perioperative patient management.

## Keywords

Cardiac surgical procedures, cardiovascular physiological phenomena, computer simulation, heart defects, congenital, hemodynamics, patient-specific modeling

## Introduction

Congenital heart disease (CHD) encompasses many conditions with abnormal flow patterns. For some of these conditions, the defects are corrected primarily by resolving the problem directly (e.g., closure of a ventricular septal defect). In others, new flow patterns, often extraanatomic to sustain life, are established. The extreme and often discussed example of the latter case is the three-stage palliation of hypoplastic left heart syndrome. For hypoplastic left heart syndrome, a series of surgeries create a new system that maintains circulation with only one functional ventricle. For this and many other pathologies, the end results of the treatment process depend on optimal flows along the way. The cardinal question that arises is whether these flows and the subsequent outcomes caused by them can be preoperatively simulated and predicted. Historically, many of these questions have been answered intuitively or by using direct surgical evidence. Examples of these questions are as: what size and in what configuration should a graft connect the inferior vena cava to the pulmonary artery in a Fontan procedure? What is the lowest limit of pulmonary artery diameter for a successful Fontan circulation? What is the ideal aortopulmonary shunt size and configuration that provides the optimal flow and the lowest risk of thrombosis in a specific patient with unique hemodynamic parameters? Opinions in the literature, if they exist, vary and in most cases, solutions to these types of questions are reached intuitively. These and many other questions could be answered objectively with the aid of computational fluid dynamics (CFD).

What is computational fluid dynamics and what can it do?

In the larger picture, CFD provides numeric quantification and qualitative visualization of many useful hemodynamic parameters (Table 1). Qualitatively, streamlines and pathlines provide a visual demonstration of flow through 3-dimensional (3D) structures and are the mainstay option in CFD of utmost importance for surgeons to visualize changes in flow patterns caused by procedures that alter local anatomy. Quantitatively, CFD provides flow velocities and pressures that can be used to aid in objective

[1]Stanford University, Samaritan Cardiovascular Surgery, Corvallis, OR, USA
[2]University of Nebraska Medical Center, Omaha, NE, USA

**Corresponding author:**
Rabin Gerrah, Good Samaritan Regional Medical Center, 3640 NW Samaritan Drive, Suite 100B, Corvallis, OR 97330, United States.
Email: rgerrah@gmail.com

**Table 1.** Hemodynamic variables studied in computational fluid dynamics and their clinical importance.

| Parameter | Description | Clinical significance/example |
|---|---|---|
| Flowrate | Quantified amount of blood volume moving in time unit | A general parameter useful in evaluation of all anomalies and repairs |
| Pressure | A surrogate for the energy generated by heart in the circulation | A valuable parameter important in all anomalies and repairs |
| Velocity | Speed of flow, a surrogate for cross section area of flow | A valuable parameter especially in assessment of flow in valve and vessels stenosis |
| Wall shear stress | The force applied on a structure parallel to flow direction | Thrombosis of aortopulmonary shunts and Conduits |
| Energy loss | The portion of circulation driving power that is lost | Single ventricle, Fontan procedure, Heart Failure |
| Streamlines | Lines that visualize the flow path | Valuable in visualizing all anomalies and their repairs |
| Oxygen delivery | Calculated value to quantify oxygenation | Systemic and end organ oxygen saturation following any procedure |

decision-making. For example, in clinical applications, velocity carries significant importance and is often used as a surrogate to quantify flow, pressure, or the cross-sectional area of a structure through which blood flows through, based on Bernoulli's equation.[1] For instance, a higher velocity means passage of blood through a narrower area. Conversely, CFD allows direct calculation of pressure and velocity simultaneously, with the ability to visualization of these dynamics in a 3D structure view of a patient-specific model. CFD provides a superior method for visualizing flow dynamics and allows calculation of other parameters such as wall shear stress (WSS) and energy loss, which are difficult to measure directly. Furthermore, pressure fields are a surrogate for oxygen supply and velocity fields are indicative of many physical properties. Some very useful clinical parameters such as systemic and end-organ oxygen delivery, most importantly cerebral and myocardial, can be evaluated by CFD rather than by invasive procedures.[2] These parameters are fundamental to the functions of the cardiovascular system and critical in yielding optimal circulation. For instance, perturbations in WSS are associated with thrombosis.

Calculated WSS by CFD can thus guide specific surgical modifications that reduce the risk of thrombosis, a high-risk complication in a aortopulmonary shunts.[3,4] Also of importance in the field of congenital heart disease is the assessment of energy loss. Energy loss occurring during the flow becomes an important factor in single-ventricle physiology, where the entire driving power of the circulation is provided by only one ventricle. Minimizing the energy loss in this type of circulation has significant clinical implications, as in Fontan circulation.[5,6] A specific configuration of the Fontan procedure can be planned to minimize the energy loss accordingly.[7,8]

For these and many other reasons, CFD has become a mainstay in studying cardiovascular diseases. The number of papers utilizing CFD to investigate hemodynamics has increased markedly in recent years. This proliferation is largely attributable to improvements in computing power, which has made CFD more assessable to clinical researchers. Indeed, CFD enables many important clinical questions to be investigated noninvasively and at a level of detail not previously possible. In fact, much of the knowledge so far attained and applied to optimize the surgical outcomes has originated from studies utilizing CFD (Table 2). Some successful clinical examples are as follows. In the Fontan operation, CFD data recommend using a 16- or 18-mm graft as the optimal size, whereas counterintuitively, usage of a larger graft is associated with suboptimal hemodynamics.[15] Based on CFD research that was correlated with clinical studies, in patients with pulmonary valve regurgitation after repair of tetralogy of Fallot, the optimal timing of pulmonary valve replacement to preserve right ventricular function could be determined by specifically set right ventricular systolic and diastolic volumes.[20] As another example of CFD application, CFD analysis has been used to simulate and predict the hemodynamics and outcome of an intervention during virtual surgery.[18] Because the process of CFD analysis is mainly based on imaging and other hemodynamic data and is often time-consuming due to large data process time, its application has been mainly in preoperative assessment, planning, and predication rather than intra- or postoperative management.

In this primer, we aim to explain CFD at a level for and relevant to the clinical audience unfamiliar with computational modeling. It is important to note that CFD is a general term that encapsulates a large collection of numerical methods. Here, we focus on the two most common methods used to model blood flow in CHD patients: the finite element method (FEM) and finite volume method (FVM). Our goal is to assist clinicians in the understanding, application, and

**Table 2.** Studies using computational fluid dynamics models to address clinical problems in congenital heart diseases and surgeries.

| Congenital heart disease or surgery | CFD parameters | Hypothesis or clinical conclusion | Author |
|---|---|---|---|
| Univentricular heart | Pressure, flow, WSS, energy loss, oxygen saturations | Clinical decision-making in univentricular hearts based on CFD simulations | Hsia[2] |
| HLHS | Pressure, Energy loss, WSS | Using CFD with pulsatile simulations to optimize surgical treatment | Qian[9] |
| TOF, shunt | WSS | Shunt geometry: a direct shunt, rather than the central oblique, or right pulmonary artery shunts is preferred. | Piskin[10] |
| HLHS | WSS, energy loss | Creation of a large anastomotic space and a smooth aortic arch angle reduced wall shear stress and energy loss, and should improve long-term cardiac performance after the Norwood procedure. | Itatani[11] |
| HLHS, hybrid procedure | Pressure, energy loss and wall shear stress | Pulmonary artery banding at 50% provided a balanced pulmonary and systemic circulation with adequate coronary flow but without extra energy losses incurred. | Shuhaiber[12] |
| HLHS, hybrid approach | Flows, velocities, oxygen saturations | Effects of pulmonary artery banding diameter and retrograde aortic arch hypoplasia or obstruction on hybrid stage 1 circulation | Baker[13] |
| Fontan circulation | Energy loss | Using CFD to optimize geometry with improving the efficiency and therefore the clinical outcome | Rijnberg[14] |
| Fontan graft size | Energy loss | Larger size conduits for extracardiac Fontan showed redundant spaces, thus 16 and 18 mm conduits were optimal. | Itatani[15] |
| Fontan procedure | Flow studies | Designing surgical procedure based on computational fluid dynamics results | de Leval[16] |
| Fontan procedure | Pressure, energy loss | A small pulmonary artery causes a high-pressure gradient and a high energy loss. The lower limit of pulmonary artery index, considering the exercise tolerance, was $110 mm^2/m^2$. | Itatani[17] |
| Flow simulations in virtual cardiac surgery | Hepatic flow distribution, energy loss | Using CFD to improve the quality of surgery using virtual cardiac surgery | Siallagan[18] |
| Blood flow in shunt | Pressure, flow, wall shear stress | Graft angulation presents a risk for shear stress-induced, platelet- mediated thrombosis, which is more likely to occur in elongated central than in Sano shunts. | Ascuitto[3] |
| Right ventricle-related pathologies (TOF, DORV) | Energy loss | Review of the causes of right ventricular-pulmonary circulation failure and the limitation of current clinical parameters to quantify its dysfunction. | Lee[19] |
| TOF | Power loss | In patients with pulmonary valve regurgitation after repair of TOF, pulmonary valve replacement is recommended before the critical value of 139 $mL{\cdot}m^{-2}$ RVEDV and 75 $mL{\cdot}m^{-2}$ RVESV to preserve right ventricular function. | Fogel[20] |

CFD: computational flow dynamics; DORV: double-outlet right ventricle; HLHS: hypoplastic left heart syndrome; RVEDV: right ventricular end-diastolic volume; RVESV: right ventricular end-systolic volume; TOF: tetralogy of Fallot.

interpretation of studies utilizing these techniques by providing a basic framework contextualized in general CHD blood flow applications. We start with a basic overview of the theory of fluid dynamics, then walk through the common FEM and FEV workflow from clinical imaging to solution analysis. Brief definitions of the most important terms used in clinical studies using FEM and FVM are included in Table 3. Additionally, the application of CFD in CHD research is briefly visited.

## Fundamental principles of CFD

Continuum mechanics underlies the theoretical basis of CFD for most macroscopic flow problems. Instead of modeling fluid as a collection of individual molecules, fluid is abstracted as a continuous substance completely filling the space it occupies. This space is called the fluid domain. Conservation laws (e.g., conservation of mass and conservation of momentum) are then applied to the fluid domain to derive generally applicable equations. Conservation of mass leads to the mass continuity equations. Conservation of momentum leads to the Navier-Stokes equations. Together, these equations form the governing equations of fluid flow CFD methods aim to solve. Although many CFD methods exist, some based on different governing equations (e.g., the Lattice Boltzmann method),[21] not all are commonly used in the CHD literature and are thus beyond the scope of this primer. The two most commonly used CFD methods applied in CHD research (FEM and FVM) share many workflow similarities and will be our focus here.

**Table 3.** Brief glossary of the most important terms used in clinical studies using computational flow dynamics techniques.

| Fluid domain | Continuous 2- or 3-dimensional space in which fluid flow is simulated |
|---|---|
| Mass continuity equations | Governing equations derived from the conservation of mass |
| Navier-Stokes equations | Governing equations derived from the conservation of momentum |
| Newtonian fluid | Fluid that exhibits a linear relationship between shear stress and strain rate through a proportionality constant called viscosity |
| Non-Newtonian fluid | Fluid that exhibits a non-linear relationship between shear stress and strain rate through a more complicated function |
| Velocity field | Vector field with velocity defined at every point |
| Pressure field | Scalar field with pressure defined at every point |
| Fluid structural interaction | Fluid structure interface simulations link computational flow dynamics with solid mechanic simulations |
| Mesh (surface) | Discrete representation of the fluid domain outer boundary |
| Mesh (volumetric) | Discrete representation of the fluid domain inner volume |
| Nodes | Points within the fluid domain |
| Elements | Sets of nodes linked together forming regular sub-domains making up the mesh |
| Steady state | Solution is time-independent |
| Transient | Solution is time-dependent |
| Initial conditions | Properties defined in the fluid domain at time zero |
| Boundary conditions | Properties defined at the fluid domain boundary (e.g., inlet velocities or outlet pressure) |
| Pathlines/ streamlines | Technique for visualizing flow paths in the fluid domain |

## Mass continuity equations

To better illustrate the principles of continuum mechanics, derivation of the mass continuity equations will be briefly discussed. These equations are easy to conceptualize and provide good orientation for understanding the basic principles and logic behind modeling fluid flow. The mathematical setup is shown to illustrate a conceptual point only. Understand that similar logic can be applied to momentum and energy to derive other governing equations (e.g., Navier-Stokes equations).

Consider a volume of space (fluid domain). Fluid can enter this space and it can leave this space, but it can neither be created nor destroyed. As such, the difference between the rates of mass entering and leaving this space is the rate of mass accumulation (net mass addition or subtraction over time). Thus, for a compressible fluid like air, where the amount of mass per unit volume (density) can change, the conceptual translation of the mass continuity equation would be equation 1 as depicted in Figure 1a.

$$\text{Equation 1: input} - \text{output} = \text{accumulation}$$

For an incompressible fluid like blood however, where density can be assumed constant, equation 1

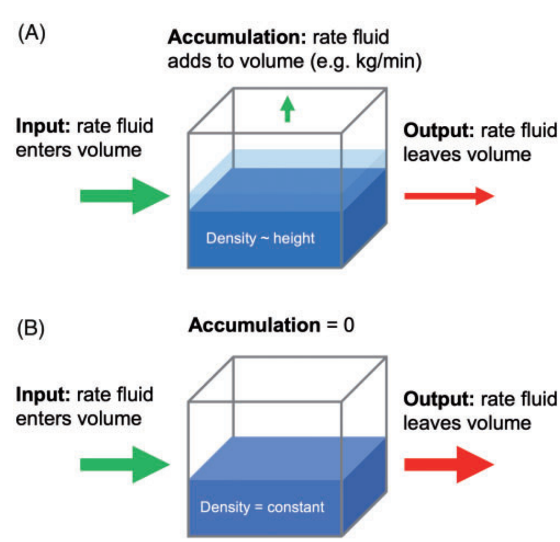


**Figure 1.** The concept of mass continuity, see text for description.

simplifies to equation 2, as depicted in Figure 1b, because non-zero accumulation mandates a change in density given constant volume.

$$\text{Equation 2 : input} - \text{output} = 0$$

Thus, mass continuity is just accounting with certain rules imposed; a balance of stuff in and stuff out. Just looking at these two equations, it should be appreciated that an incompressible fluid is easier to model than a compressible fluid, as equation 2 is simpler than equation 1. Considerations like this are an important part of CFD, because different assumptions can dramatically affect problem complexity and thus simulation runtime.

## Navier-Stokes equations

The mass continuity equations discussed above are not sufficient by themselves to describe real-world flow. Additional knowledge of the viscous behavior or rheology of the fluid itself is required. For example, a Newtonian fluid exhibits a linear relationship between strain rate (the rate of deformation between adjacent volumes of fluid) and shear stress (the force exerted per unit area between adjacent volumes of fluid in the direction of flow) through a proportionality constant called viscosity. It is this viscous behavior, combined with the conservation of momentum and the concept of pressure, that yields the Navier-Stokes equations. Although derivation of the Navier-Stokes equations is too complicated for this brief primer, conceptually, the equations can be thought of as a balance of forces. Here, the local acceleration of fluid is balanced by the distributions of stress and pressure within the fluid plus any external body forces (e.g., gravity). Pressure, viscosity, and mass continuity assumptions based on physical observation thus constrain these general balance equations into their final useful form (Navier-Stokes equations). It is important to understand that the Navier-Stokes equations apply generally to fluid domains of arbitrary shape and describe the flow at every point within these domains. Thus, a vector velocity field and scalar pressure field are simultaneously obtained by solving the Navier-Stokes equations.

## Numerical methods: FEM and FVM

The generalized nature of the Navier-Stokes equations is what makes them particularly useful for solving real-world flow problems. Unlike Poiseuille's Law, which describes fluid flow under very specific conditions (fully developed laminar flow through circular tubes), the Navier-Stokes equations describe the flow of viscous fluids under a wide variety of conditions. This is particularly useful when dealing with the complex anatomies encountered in CHD research. With this geometric complexity however, the Navier-Stokes equations become too difficult to solve analytically and instead must be approximated numerically using computers. This is the motivation behind FEM and FVM. First, the continuous fluid domain is broken into discrete sub-domains called mesh elements or simply elements. The Navier-Stokes equations are then defined over the fluid domain using these elements as basic volumes linked together to approximate the complex fluid domain and obtain approximant solutions. Although many variants of FEM and FVM have been applied in the CHD literature, the specific mathematical details are mostly beyond the scope of this primer. Understand that FEM and FVM yield approximant solutions, the accuracy of which depends on how the problem is setup. Below, we summarize the FEM and FVM workflows common to basic blood flow modeling contextualized in CHD. Our goal is to provide clinicians unfamiliar with CFD a basic roadmap of the process and highlight areas where data acquired from clinicians is crucial.

## Computational fluid dynamics workflow

In practice, CFD can be divided into three main steps: pre-processing, processing, and post-processing. The details of each step are largely problem-specific but share many common features. The discussion below will focus on aspects common to FEM and FVM in modeling basic blood flow, while highlighting some unique considerations relevant to CHD research. The overall goal is to provide the reader with a basic framework for understanding the FEM and FVM workflow. Figure 2 provides an overall summary of the basic workflow. This is not meant to be a detailed how-to guide or comprehensive literature review. For the interested reader, more detailed reviews on CFD and its novel applications in the CHD literature are provided.[22,23]

## Pre-processing

The goal of pre-processing is to fully define the fluid problem. Decisions must be made about the nature of the problem and what assumptions are appropriate. For example, will flow be modeled in 2D or 3D? Although 3D models are the gold standard in CHD research, 2D models are easier to solve and may provide preliminary information. This is particularly true if the problem's geometry possesses symmetry that allows for 2D representation.[24,25] Because CFD is a computationally intensive task, with single problems taking hours or days to solve, simplifying assumptions

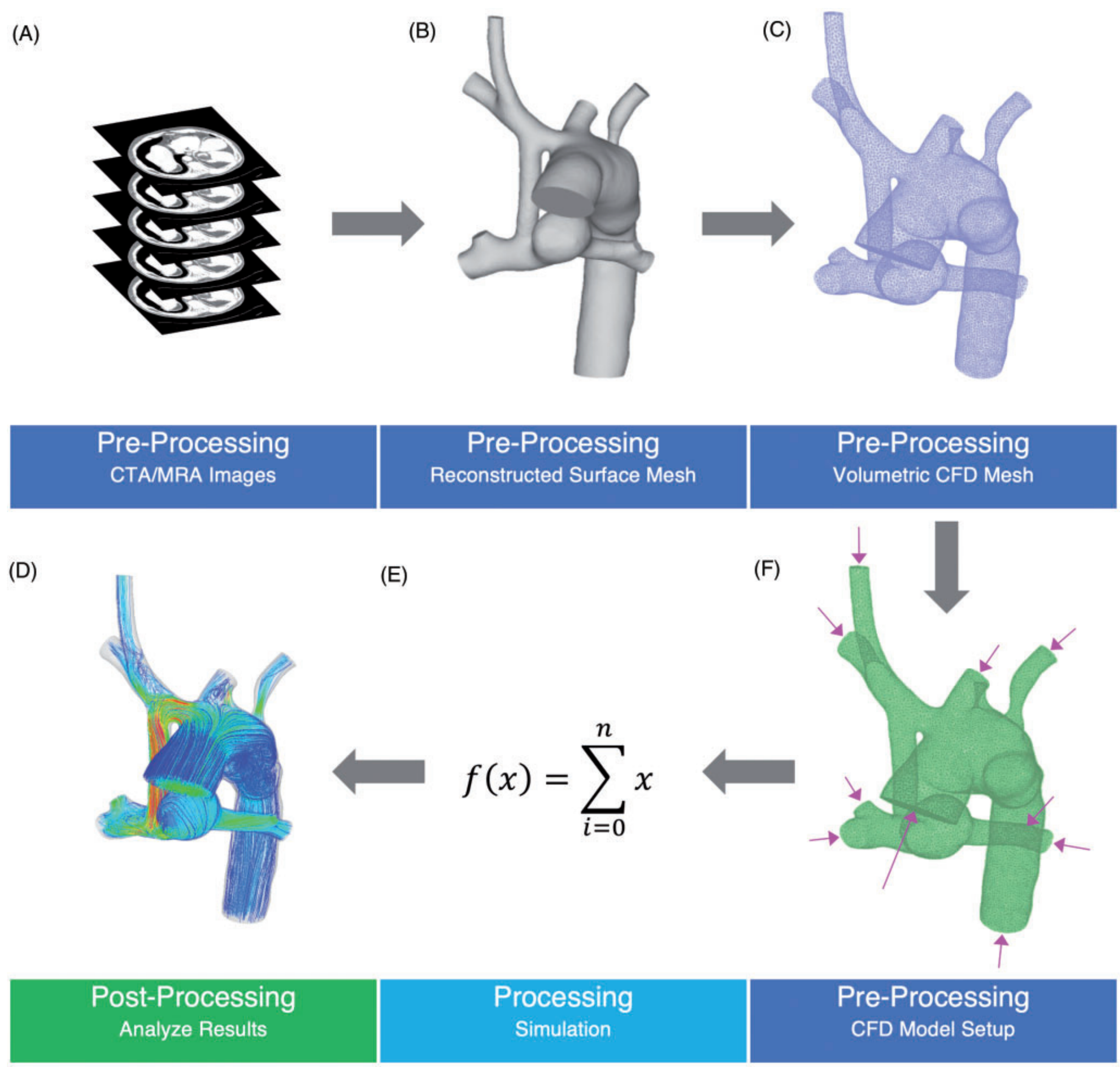


**Figure 2.** Typical computational fluid dynamics workflow illustrating pre-processing, processing, and post-processing. (A) Clinically obtained images (e.g., computed tomography angiography or magnetic resonance angiography) are sectioned so that (B) a 3-dimensional surface mesh can be reconstructed. (C) A 3-dimensional volumetric mesh is then created and (D) boundary/initial conditions are applied. (E) The simulation is processed to produce velocity/pressure fields and (F) results are analyzed for relevant information.

like this are an important part of pre-processing. Below are additional aspects that must be considered.

## Geometry reconstruction

Precise geometric representation of the fluid domain is required for FEM and FVM. Geometries may be created de novo using computer-assisted design software as idealized anatomies or reconstructed from clinical imaging as patient-specific anatomies. With regards to patient-specific anatomies, clinical imaging techniques should provide volumetric data and employ contrast for better delineation of vessel/lumen boundaries (e.g., computed tomography angiography and magnetic resonance angiography). Proper anatomical description is a key area where clinicians can have a profound impact on the CFD workflow. It is therefore important for clinicians to understand the overall process and commonly encountered difficulties.

Once images are obtained, a variety of reconstruction techniques exist, but they all rely on essentially the same principle. First, determine the vessel/lumen interface. Second, construct a surface mesh that represents the vessel/lumen interface (i.e., fluid domain boundary). Third, subdivide the volume bound by the surface mesh (i.e., fluid domain) into a volume mesh composed of elements (see below). While many commercial and noncommercial software packages can expedite this process (e.g., SimVasular, Vascular Modeling Toolkit, or Crimson software), they often require a fair degree of user intervention to generate quality results suitable for FEM and FVM. Depending on the complexity of the anatomy, this can be a difficult process that can take hours per model because low image resolution on small structures introduces ambiguity on where to define boundaries. Indeed, geometry reconstruction remains a major obstacle for conducting large-scale studies and clinical application using patient-specific anatomies. Nevertheless, high-resolution imagining can save time and enable better results through more automated workflows. Clinicians providing patient imaging should thus carefully consider the source of imaging and its practically as a starting point for CFD.

Another important point to understand for many CFD studies is that reconstruction of the vessel wall itself is not necessary unless the solid mechanics of the wall are to be included in the simulation. For many investigations, this level of detail is not necessary and a rigid wall assumption is made. For some investigations, such as modeling heart valves, including the solid mechanics of the valve may be unavoidable. Simulations that couple CFD with structural mechanics equations are referred to as fluid structural interaction problems and are more difficult to solve. Although the remainder of this primer will focus on fluid simulations, knowing fluid structural interaction exists is helpful when thinking about CHD research questions. For the interested reader, reviews on fluid structural interaction applications in modeling cardiovascular disease are included.[25–27]

## Meshing

Meshing is the process through which the continuous fluid domain is subdivided into a finite number of subdomains called mesh elements or simply elements. Each element is composed of a set of points called nodes, linked together to form basic geometric shapes that are easy to define mathematically. Two commonly used types of 3D elements are tetrahedrons and hexahedrons (Figure 3a, 3b). Due to the continuous nature of the fluid domain, elements can share common nodes such that elements may have common edges or faces (Figure 3c). The overall idea in creating a volumetric mesh is to break up the complex geometry of interest (fluid domain) into a discrete number of simpler geometries the Navier-Stokes equations can be more easily solved on. Generally speaking, the more elements incorporated, the more accurate the solution. However, important details exist in the different types of elements

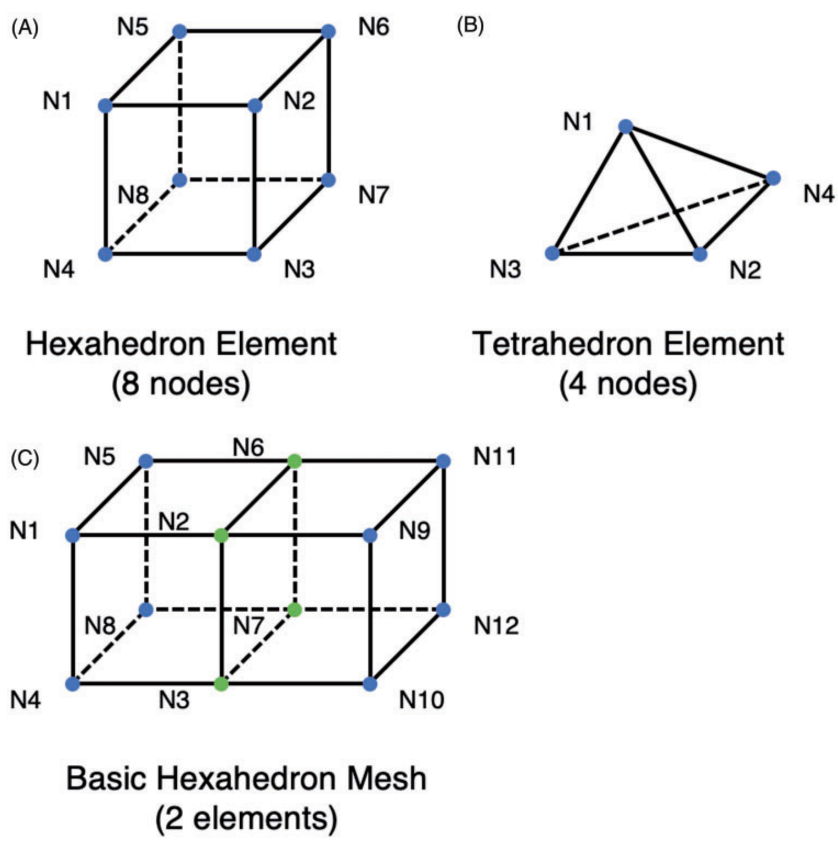


**Figure 3.** Basic computational fluid dynamics elements. (A) A hexahedron element with 8 nodes. (B) A tetrahedron element with 4 nodes. (C) Connectivity showing common nodes from element overlap.

and how these elements are used, which can affect solution accuracy. For example, elements are used differently between FEM and FVM in terms of where values of interest are defined (e.g., velocity). In FEM, velocities are defined at nodes and approximated inside the elements to varying degrees of accuracy when obtaining solutions. The accuracy of these approximations depends on the type of element and interpolation methods used. For example, hexahedrons are generally more accurate than tetrahedrons because they use higher-order interpolation methods and therefore fewer hexahedron elements may be required. Conversely, in FVM, velocities are defined at element centers and interpolated between adjacent elements to compute flows across element faces. Similarly, the accuracy of flows approximated across elements can vary based on the interpolation scheme used. Indeed, creating quality meshes for FEM and FVM can pose specific challenges and should be validated prior to conducting final simulations. So-called mesh sensitivity studies should be performed, where solutions from meshes of varying fidelity are compared for solution convergence. This is particularly important for the complex geometries regularly encountered in CHD research. Indeed, building "clean" meshes from clinically derived anatomies quickly and reliability represents one of the major challenges in making CFD a widespread clinical tool. Although automated FEM and FVM meshing tools generally work well, these tools are highly dependent on well-constructed surface meshes obtained during geometry reconstruction.

## Fluid model

The fluid model or rheology describes the viscous properties and density of the fluid in question; in the case of CHD research, blood. As alluded to previously, a Newtonian fluid exhibits a linear relationship between shear stress and strain rate through a proportionality constant called viscosity. Non-Newtonian fluids exhibit nonlinear relationships and are more difficult to simulate. Although blood is technically a suspension composed of cells, platelets, and proteins floating in plasma that exhibits non-Newtonian behavior (i.e., viscoelastic shear thinning properties),[29,30] blood is often simulated as a Newtonian fluid for simplicity. Additionally, blood is almost always assumed to be incompressible and therefore of constant density. Assumptions like these greatly reduce problem complexity while still providing reasonable accuracy, given other limitations.[31] Nevertheless, there is still some debate in the literature concerning the use of Newtonian versus non-Newtonian fluid models in simulating hemodynamics.[31–33]

## Transient and steady-state flow

Although physiologic blood flow is pulsatile (i.e., time-dependent or transient), modeling steady-state flow may yield sufficient information for certain problems. Steady-state flow is generally easier to model than transient flow because the time components of the Navier-Stokes equations can be removed (such as how the accumulation term is removed from the mass continuity equation when assuming an incompressible fluid). However, it is important to understand the limitations of a steady-state assumption and when it is applicable. Pulsatility index (PI), defined as:

$$\mathrm{PI} \equiv \frac{u_{\mathrm{systolic}} - u_{\mathrm{diastolic}}}{u_{\mathrm{mean}}}$$

where u is velocity, can assist in determining the validity of a steady-state assumption. Conceptually, PI quantifies the periodic time-dependent change in velocity. Systems with low PI values have minimal time dependence and can thus be reasonably approximated as steady state. However, transient models are typically considered the gold standard, particularly in CHD research, where the regions of interest are within or very close to the heart which generates high PI values. Nevertheless, steady-state simulations still have valid uses in CHD research and high PI flows, particularly for calculating pressure drops.[34,35]

## Initial conditions

To model time-dependent transient blood flow, initial conditions must be specified. As the name implies, initial conditions define the system state (e.g., velocity and pressure fields) at the beginning of the simulation. Because transient simulations are time-dependent, the initial state of the fluid domain influences how the simulated flow progresses and thus affects subsequent states. A common method is to initialize the velocity field as zero at all points and allow the flow to develop over a set number of cardiac cycles.[9,36] The first cycles are discarded and the final stable cycles are retained for analysis. Alternatively, initial conditions can be set based on known information such as pervious simulations or clinical data.[37]

## Boundary conditions

Boundary conditions further define the fluid problem and are an essential component of CFD. As the name implies, these conditions are defined at the boundaries of the fluid domain (e.g., walls, inlets, outlets). For example, how does blood flowing at the vessel/lumen interface behave? What is the velocity profile of blood entering a vessel? What is the pressure of blood leaving a vessel? In FEM and FVM, the need to specify these types of conditions becomes clear when considering elements at the surface of the fluid domain. For these elements, at least one node or face will not be linked to another element. Thus, values of interest at these boundaries must be defined explicitly because there is no adjacent element to mathematically link to. Common examples of boundary conditions include the no-slip boundary condition that is often used for the vessel/lumen interface, defining blood flow velocity at the wall as zero. A rigid wall assumption and thus constant-sized fluid domain is also common, although compliant vessels and/or contracting heart chambers can also be modeled.[37,38] Inlet and outlet conditions can be set as normal tractions (i.e., pressure), parabolic or flat velocity profiles based on clinical measurements,[39,40] or linked to resistance networks to simulate flow-dependent peripheral vascular resistance.[10,41] Indeed, because it is impractical to simulate the entire cardiovascular system, any segment that is simulated will have artificial boundaries. Clinically determined pressures and velocities at model boundaries or physiologic-based parameters for modeling blood flows outside the fluid domain in the case of lumped parameter models thus represent additional areas clinicians can greatly contribute to the CFD workflow.

## Processing

The goal of processing is to solve the fluid problem defined in pre-processing. Although many variants of FEM and FVM exist, the mathematical details are mostly beyond the scope of this primer; each method has its pros and cons. For example, traditional FEM does not guarantee local conservation and may destabilize under convection-dominated flow. FVM solves these problems due in part to its cell-centered approach, but introduces new complexities absent from FEM, most of which are not visible to the user. To reiterate, the key difference between FEM and FVM are the locations at which values of interest (e.g., velocity) are computed and stored (nodal vs. cell-centered). Indeed, these locations are reflective of the underling discretization schemes employed by FEM and FVM to approximate the continuous governing equations of fluid flow. In FEM, nodal velocities are interpolated and integrated such that elemental basis functions are satisfied. Conversely, in FVM, elemental control volumes with cell-centered velocities are integrated such that fluxes between adjacent elements are balanced. Overall, both methods are employed by commonly used CFD codes and serve as valid options for CHD research applications. Below are some important practical points to keep in mind.

FEM and FVM are both computationally intensive tasks. It is not uncommon for simulations to take hours or days to complete. This is because CFD is an iterative process; solvers converge on approximant solutions within specified tolerances by calculating the next solution based on the previous solution. As alluded to previously, simplifications are often made when constructing problems to make them easier to solve. The tradeoff is between accuracy and time. Proper assumptions are thus essential in obtaining meaningful CFD results in a timely manner. Additionally, meshes of unsuitable quality can cause simulations to fail. Thus, processing and pre-processing require some back and forth to yield accurate results (mesh sensitivity studies). From a computing perspective, virtually all FEM and FVM programs benefit from parallel computing. However, access to a supercomputer is not required. With recent advances in modern computing, a high-end desktop workstation can be used successfully for many CFD applications. The major advantage in using a computing cluster is running multiple simulations simultaneously or solving more complex problems by distributing the problem over multiple computers.

## Post-processing

The final step in the CFD workflow is post-processing. Post-processing encompasses a variety of techniques used to analyze solutions and extract meaningful information. The solution files obtained from CFD solvers essentially contain a list of coordinates located within the fluid domain, with velocity and pressure values specified; these lists represent velocity and pressure fields, respectively. These data may be visualized directly using special programs, or additional information may be calculated and rendered. It is essential that all results obtained from CFD simulations be validated to ensure they accurately reflect the real-life clinical scenarios they are meant to model. This is of critical importance if future clinical translation is to be achieved.[42,43]

## Additional parameters and visualization

A variety of techniques exist for visualizing CFD solutions. These techniques often involve the use of color maps and/or vectors to represent variables of interest (e.g., velocity, pressure, etc.). The entire fluid domain may be visualized in 3D as a volumetric rendering (Figure 4a, 4b), 2D slices may be taken to view certain regions of data (Figure 4e, 4f), and transient solutions can be viewed over time as animations. Additionally, any parameter(s) dependent on the velocity and/or pressure fields may be calculated and subsequently visualized. Additional parameters of interest in CHD research that may be calculated include WSS, WSS

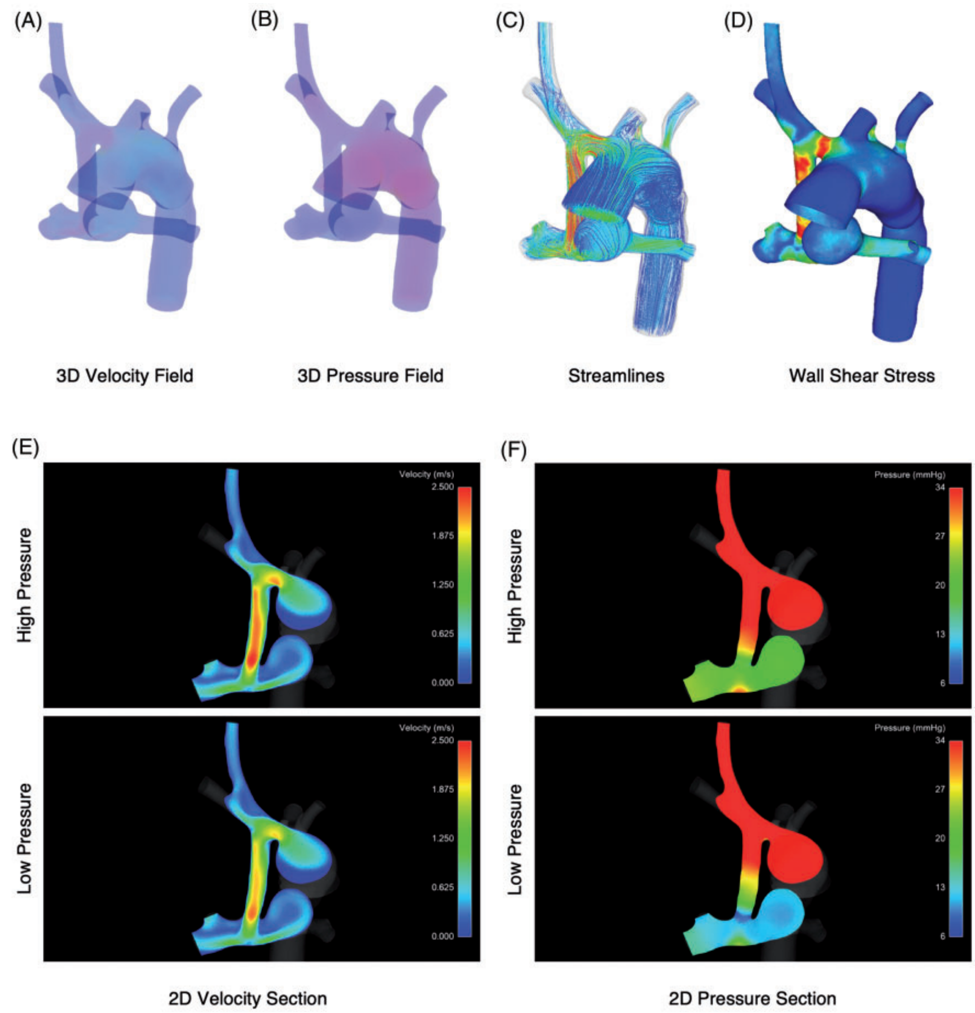


**Figure 4.** Example illustrations of common computational fluid dynamics results. (A, B) Three-dimensional volumetric renderings representing velocity and pressure magnitudes at each point in the fluid domain. (C) Streamlines with color representing velocity. (D) Wall shear stress with color representing the magnitude on the wall surface. (E, F) Two-dimensional sections of velocity and pressure with different pressure loading conditions, respectively.

gradient, and oscillatory shear index. These parameters are commonly used as surrogate markers of endothelial cell function, hemolysis, and platelet activation as part of the coagulation cascade.[44,45] These values can be mapped on to the vessel wall surface to visualize where high shear regions are expected (Figure 4d). Residence times may also be calculated to determine areas of flow stagnation, which are associated with thrombus development.[46] Peak velocities, average velocities, pressure drops, and anything within the scope of the solution can be investigated. This flexibility and ease of querying is what makes CFD particularly useful in comparison to measuring real-world systems. This is especial true for CHD research where making physical measurements on patients often inflicts damage and high costs. Many commercial and non-commercial programs exist for data visualization and with interchangeable support for most features (e.g., Visualization Toolkit, ParaView).

## Pathlines and streamlines

To better visualize flow, it is often helpful to plot the path virtual particles would take through the velocity field. Pathlines serve this purpose. Virtual particles are introduced into the velocity field and their instantaneous trajectories are iteratively computed as they are simulated to travel through the fluid domain. These trajectories are then combined to form continuous lines (Figure 4c). Calculating pathlines is a relatively simple computational task. Thousands may be calculated in the order of seconds to minutes.

The distinguishing feature between pathlines and streamlines is that pathlines represent the virtual paths particles would take through a time-dependent velocity field. Conversely, streamlines represent the instantaneous paths massless particles would take at any given instance. Under steady-state flow, pathlines and streamlines are synonymous. Within highly transient velocity fields however, pathlines will record the flow history while streamlines will only display instantaneous velocities. Additionally, given particle mass, pathlines can account for the inertial effects real particles display. However, for low mass particles like red blood cells inertial effects are minimal. Thus, pathlines provide a physically based model for further investigating CFD solutions, while streamlines provide a convenient visualization tool for assessing instantaneous flow. A major application of pathlines includes the investigation of flow pattern effects on thrombi/emboli formation.[47,48]

## Limitations

Despite its power and utility, CFD is not without its limitations. Simulation results are only as good as the assumptions made, which may not capture all the subtleties present in vivo. Simulating true turbulent flow (as defined by the Reynolds number) with the Navier-Stokes equations also remains a challenge because the numerical methods employed can become unstable under these conditions. Additionally, practical limitations exist in applying CFD to clinical research questions. Although computers are more powerful today than even before, due to the computational demands of FEM/FVM and complexity of CHD problems, studies continue to be limited by computing power. As such, simplifying assumptions remain necessary, with researchers choosing the most important aspects of problems to model. Multidisciplinary teams comprised of clinicians, engineers, and scientists are often the most successful because they can collectively construct the best models, accounting for the most important details. Although this reality necessitates collaboration, it is also one of the obstacles faced by clinicians seeking to employ CFD. Indeed, the introductory level at which CFD was discussed in this primer was done so intentionally to give clinicians unfamiliar with CFD a general understanding. Nevertheless, CFD is much more complex and mathematically rich once one delves deeper.[49,50]

## Conclusions

FEM and FVM has greatly expanded researchers' ability to investigate the complex flows inherent to CHD. Looking forward, we anticipate CFD continuing to grow as an important research tool that will eventually gain more widespread clinical translation. Aside from the obvious technical limitations, perhaps the greatest barrier to widespread clinical incorporation of CFD is the lack of familiarity and comfort with CFD techniques, including FEM and FVM, the two most commonly used methods. Clinicians looking to incorporate CFD into their research and clinical programs should thus gain a basic understanding of the capabilities and limitations of these tools. Basic understanding of the governing theory and workflow will also help facilitate necessary conversations integral to multidisciplinary collaboration, a key feature of most successful CFD applications in the medical field. Ultimately, CFD has the potential to empower the next generation of CHD treatments and move towards realizing the dream of quantitative patient-specific predictions and procedure planning.

### Acknowledgement

The authors would like to thank Dr. Sandra Rugonyi for her assistance with the preparation of the images in this manuscript.

### Declaration of conflicting interests

The author(s) declared no potential conflicts of interest with respect to the research, authorship, and/or publication of this article.

### Funding

The author(s) received no financial support for the research, authorship, and/or publication of this article.

### ORCID iD

Rabin Gerrah https://orcid.org/0000-0002-8089-8396

### References

1. Gerrah R, Haller SJ and George I. Mechanical concepts applied in congenital heart disease and cardiac surgery. *Ann Thorac Surg* 2017; 103: 2005–2014.
2. Hsia TY and Figliola R; Modeling of Congenital Hearts Alliance (MOCHA) Investigators; Modeling of Congenital Hearts Alliance MOCHA Investigators. Multiscale modelling of single-ventricle hearts for clinical decision support: a Leducq Transatlantic Network of Excellence. *Eur J Cardiothorac Surg* 2016; 49: 365–368.
3. Ascuitto R, Ross-Ascuitto N, Guillot M and Celestin C. Computational fluid dynamics characterization of pulsatile flow in central and Sano shunts connected to the pulmonary arteries: importance of graft angulation on shear stress-induced, platelet-mediated thrombosis. *Interact Cardiovasc Thorac Surg* 2017; 25: 414–421.
4. Celestin C, Guillot M, Ross-Ascuitto N and Ascuitto R. Computational fluid dynamics characterization of blood flow in central aorta to pulmonary artery

connections: importance of shunt angulation as a determinant of shear stress-induced thrombosis. *Pediatr Cardiol* 2015; 36: 600–615.
5. Honda T, Itatani K, Takanashi M, et al. Quantitative evaluation of hemodynamics in the Fontan circulation: a cross-sectional study measuring energy loss in vivo. *Pediatr Cardiol* 2014; 35: 361–367.
6. Tang E, Wei ZA, Whitehead KK, et al. Effect of Fontan geometry on exercise haemodynamics and its potential implications. *Heart* 2017; 103: 1806–1812.
7. Trusty PM, Wei Z, Tree M, et al. Local hemodynamic differences between commercially available Y-grafts and traditional fontan baffles under simulated exercise conditions: implications for exercise tolerance. *Cardiovasc Eng Technol* 2017; 8: 390–399.
8. Yang W, Feinstein JA, Shadden SC, Vignon-Clementel IE and Marsden AL. Optimization of a Y-graft design for improved hepatic flow distribution in the fontan circulation. *J Biomech Eng* 2013; 135: 011002.
9. Qian Y, Liu JL, Itatani K, Miyaji K and Umezu M. Computational hemodynamic analysis in congenital heart disease: simulation of the Norwood procedure. *Ann Biomed Eng* 2010; 38: 2302–2313.
10. Piskin S, Unal G, Arnaz A, Sarioglu T and Pekkan K. Tetralogy of Fallot surgical repair: shunt configurations, ductus arteriosus and the circle of Willis. *Cardiovasc Eng Technol* 2017; 8: 107–119.
11. Itatani K, Miyaji K, Qian Y, Liu JL, et al. Influence of surgical arch reconstruction methods on single ventricle workload in the Norwood procedure. *J Thorac Cardiovasc Surg* 2012; 144: 130–138.
12. Shuhaiber JH, Niehaus J, Gottliebson W and Abdallah S. Energy loss and coronary flow simulation following hybrid stage I palliation: a hypoplastic left heart computational fluid dynamic model. *Interact Cardiovasc Thorac Surg* 2013; 17: 308–313.
13. Baker CE, Corsini C, Cosentino D, et al. Effects of pulmonary artery banding and retrograde aortic arch obstruction on the hybrid palliation of hypoplastic left heart syndrome. *J Thorac Cardiovasc Surg* 2013; 146: 1341–1348.
14. Rijnberg FM, Hazekamp MG, Wentzel JJ, et al. Energetics of blood flow in cardiovascular disease: concept and clinical implications of adverse energetics in patients with a Fontan circulation. *Circulation* 2018; 137: 2393–2407.
15. Itatani K, Miyaji K, Tomoyasu T, et al. Optimal conduit size of the extracardiac Fontan operation based on energy loss and flow stagnation. *Ann Thorac Surg* 2009; 88: 565–572.
16. de Leval MR, Dubini G, Migliavacca F, et al. Use of computational fluid dynamics in the design of surgical procedures: application to the study of competitive flows in cavo-pulmonary connections. *J Thorac Cardiovasc Surg* 1996; 111: 502–513.
17. Itatani K, Miyaji K, Nakahata Y, Ohara K, Takamoto S and Ishii M. The lower limit of the pulmonary artery index for the extracardiac Fontan circulation. *J Thorac Cardiovasc Surg* 2011; 142: 127–135.
18. Siallagan D, Loke YH, Olivieri L, et al. Virtual surgical planning, flow simulation, and 3-dimensional electrospinning of patient-specific grafts to optimize Fontan hemodynamics. *J Thorac Cardiovasc Surg* 2018; 155: 1734–1742.
19. Lee N, Taylor MD and Banerjee RK. Right ventricle-pulmonary circulation dysfunction: a review of energy-based approach. *Biomed Eng Online* 2015; 14(Suppl 1): S8.
20. Fogel MA, Sundareswaran KS, de Zelicourt D, et al. Power loss and right ventricular efficiency in patients after tetralogy of Fallot repair with pulmonary insufficiency: clinical implications. *J Thorac Cardiovasc Surg* 2012; 143: 1279–1285.
21. The lattice Boltzmann method: principles and practice. New York, NY: Springer Berlin Heidelberg, 2016.
22. DeCampli WM, Argueta-Morales IR, Divo E and Kassab AJ. Computational fluid dynamics in congenital heart disease. *Cardiol Young* 2012; 22: 800–808.
23. Doost SN, Ghista D, Su B, Zhong L and Morsi YS. Heart blood flow simulation: a perspective review. *Biomed Eng Online* 201625; 15: 101.
24. Horn JD, Maitland DJ, Hartman J and Ortega JM. A computational thrombus formation model: application to an idealized two-dimensional aneurysm treated with bare metal coils. *Biomech Model Mechanobiol* 2018; 17: 1821–1838.
25. Wald S, Liberzon A and Avrahami I. A numerical study of the hemodynamic effect of the aortic valve on coronary flow. *Biomech Model Mechanobiol* 2018; 17: 319–328.
26. Hirschhorn M, Tchantchaleishvili V, Stevens R, Rossano J and Throckmorton A. Fluid-structure interaction modeling in cardiovascular medicine – A systematic review 2017-2019. *Med Eng Phys* 2020; 78: 1–13
27. Marsden AL and Feinstein JA. Computational modeling and engineering in pediatric and congenital heart disease. *Curr Opin Pediatr* 2015; 27: 587–596
28. Sotiropoulos F and Borazjani I. A review of state-of-the-art numerical methods for simulating flow through mechanical heart valves. *Med Biol Eng Comput* 2009; 47: 245–256.
29. Baskurt OK and Meiselman HJ. Blood rheology and hemodynamics. *Semin Thromb Hemost* 2003; 29: 435–450.
30. Fedosov DA, Noguchi H and Gompper G. Multiscale modeling of blood flow: from single cells to blood rheology. *Biomech Model Mechanobiol* 2014; 13: 239–258.
31. Lee SW and Steinman DA. On the relative importance of rheology for image-based CFD models of the carotid bifurcation. *J Biomech Eng* 2007; 129: 273–278.
32. Good BC, Deutsch S and Manning KB. Hemodynamics in a pediatric ascending aorta using a viscoelastic pediatric blood model. *Ann Biomed Eng* 2016; 44: 1019–1035.
33. Cheng AL, Pahlevan NM, Rinderknecht DG, Wood JC and Gharib M. Experimental investigation of the effect of non-Newtonian behavior of blood flow in the Fontan circulation. *Eur J Mech B Fluids* 2018; 68: 184–192.
34. Weese J, Lungu A, Peters J, Weber FM, Waechter-Stehle I and Hose DR. CFD- and Bernoulli-based pressure drop estimates: a comparison using patient anatomies from heart and aortic valve segmentation of CT images. *Med Phys* 2017; 44: 2281–2292.

35. Sacco F, Paun B, Lehmkuhl O, et al. Left ventricular trabeculations decrease the wall shear stress and increase the intra-ventricular pressure drop in CFD simulations. *Front Physiol* 2018; 9: 458.
36. Migliavacca F, Balossino R, Pennati G, et al. Multiscale modelling in biofluidynamics: application to reconstructive paediatric cardiac surgery. *J Biomech* 2006; 39: 1010–1020.
37. Gundelwein L, Miro J, Gonzalez Barlatay F, Lapierre C, Rohr K and Duong L. Personalized stent design for congenital heart defects using pulsatile blood flow simulations. *J Biomech* 2018; 81: 68–75.
38. Schenkel T, Malve M, Reik M, Markl M, Jung B and Oertel H. MRI-based CFD analysis of flow in a human left ventricle: methodology and application to a healthy heart. *Ann Biomed Eng* 2009; 37: 503–515.
39. Goubergrits L, Mevert R, Yevtushenko P, et al. The impact of MRI-based inflow for the hemodynamic evaluation of aortic coarctation. *Ann Biomed Eng* 2013; 41: 2575–2587.
40. Itu L, Sharma P, Ralovich K, et al. Non-invasive hemodynamic assessment of aortic coarctation: validation with in vivo measurements. *Ann Biomed Eng* 2013; 41: 669–681.
41. Baretta A, Corsini C, Yang W, et al. Virtual surgeries in patients with congenital heart disease: a multi-scale modelling test case. *Philos Trans A Math Phys Eng Sci* 2011; 369: 4316–4330.
42. Wei ZA, Sonntag SJ, Toma M, Singh-Gryzbon S and Sun W. Computational fluid dynamics assessment associated with transcatheter heart valve prostheses: a position paper of the ISO Working Group. *Cardiovasc Eng Technol* 2018; 9: 289–299.
43. Steinman DA and Migliavacca F. Editorial: Special issue on verification, validation, and uncertainty quantification of cardiovascular models: towards effective VVUQ for translating cardiovascular modelling to clinical utility. *Cardiovasc Eng Technol* 2018; 9: 539–543.
44. Yu H, Engel S, Janiga G and Thevenin D. A review of hemolysis prediction models for computational fluid dynamics. *Artif Organs* 2017; 41: 603–621.
45. Gimbrone MA, Jr, Resnick N, Nagel T, Khachigian LM, Collins T and Topper JN. Hemodynamics, endothelial gene expression, and atherogenesis. *Ann N Y Acad Sci* 1997; 811: 1–10.
46. Bosi GM, Cook A, Rai R, et al. Computational fluid dynamic analysis of the left atrial appendage to predict thrombosis risk. *Front Cardiovasc Med* 2018; 5: 34.
47. Prather R, Seligson J, Ni M, Divo E, Kassab A and DeCampli W. Patient-specific multiscale computational fluid dynamics assessment of embolization rates in the hybrid Norwood: effects of size and placement of the reverse Blalock-Taussig shunt. *Can J Physiol Pharmacol* 2018; 96: 690–700.
48. Clark WD, Eslahpazir BA, Argueta-Morales IR, Kassab AJ, Divo EA and DeCampli WM. Comparison between bench-top and computational modelling of cerebral thromboembolism in ventricular assist device circulation. *Cardiovasc Eng Technol* 2015; 6: 242–255.
49. Formaggia L, Quarteroni A, Veneziani A, editors. Cardiovascular mathematics: modeling and simulation of the circulatory system. Volume 1. Springer Series MS&A, 2009.
50. Quarteroni A, Dede' L, Manzoni A and Vergara C. *Mathematical modelling of the human cardiovascular system: data, numerical approximation, clinical applications*. Cambridge University Press, 2019.