

INTRODUCING CFD THROUGH A CARDIOVASCULAR APPLICATION IN A FLUID MECHANICS COURSE

ALEX J. APOSTOLIDIS, ANTONY N. BERIS, AND PRASAD S. DHURJATI
University of Delaware • Newark, DE 19716

We have developed an innovative project within our required undergraduate fluid mechanics course at the University of Delaware (UD) that integrates advanced research concepts into the curriculum by way of course projects. Traditional fluid mechanics courses consist mostly of lectures that tend to be fairly mathematical, while the projects are usually introduced through artificial examples that hamper the enthusiasm and motivation of students. Our course at Delaware is focused on explaining concepts in the context of contemporary applications and project work.

The current investigators were major contributors to the preparation and teaching of the novel junior-level fluid mechanics course. The class, which consisted of 79 students, was introduced to both macroscopic and microscopic balances. Upon completion of each section, students had to work on a month-long group-effort project, which a) facilitated the revision and understanding of the previously taught material, b) offered students the opportunity to practice with state-of-the-art software, and c) introduced them to modern, research-based topics that reinforced their interest in the class material.

The first project emphasized macroscopic fluid mechanics and required the students to use the AFT Fathom software to design a commercial-scale pipeline system (such as the Trans-Alaskan Pipeline or the Keystone Pipeline). The second project, which is the subject of this manuscript, focused on applying computational fluid dynamics (CFD) to obtain the solution of the full Navier-Stokes equations to demonstrate concepts in microscopic fluid mechanics. We applied CFD to a contemporary cardiovascular problem, the blockage (stenosis) in a very common artery of the heart—the first bifurcation of

the left coronary artery (LCA). The CFD project was driven by recent work on cardiovascular research conducted within our department. As part of this project, students performed CFD simulations using the ANSYS FLUENT 12.1 software.

The student response to the project-based introduction of CFD in the course was overwhelmingly positive. Based on the very supportive evaluation comments that we received, we believe that other universities could also benefit by adopting our approach to teaching advanced concepts and integrating research ideas within class projects. The intended use of this paper is to communicate our experience on how to prepare and organize such a research-originated fluid mechanics problem as a hands-on introduction to a CFD project.

Alex J. Apostolidis is a graduate student of chemical and biomolecular engineering at the University of Delaware. He received his B.S. in chemical engineering from the Aristotle University of Thessaloniki in 2009. His NSF-supported research is in blood flow modeling and simulations, focusing on the rheological characterization of blood as a complex fluid.

Antony N. Beris is a professor of chemical and biomolecular engineering at the University of Delaware also affiliated with the interdisciplinary program of biomedical engineering. He received his Ph.D. in chemical engineering from the Massachusetts Institute of Technology in 1985. His research focus is in the modeling and simulation of complex flows, non-equilibrium thermodynamics, and transport phenomena.

Prasad S. Dhurjati is a professor of chemical and biomolecular engineering at the University of Delaware with a joint appointment in the Department of Mathematical Sciences. He received his Ph.D. in chemical engineering from Purdue University in 1982. His current research focus is in the area of systems biology and modeling of engineering systems.

COURSE STRUCTURE

The majority of chemical engineering processes are conducted in the fluid phase. Examples of such operations can be found in the chemical, biochemical, pharmaceutical, polymer, and energy industries. Therefore, building a strong foundation in fluid mechanics is vital for future chemical engineers. One of the goals of the course is to equip students with the necessary knowledge to work or conduct research in this field.

As outlined in Table 1, we structured the material of the course in two parts: the study of macroscopic or large-scale phenomena and that of microscopic or small-scale phenomena. For the first part, in which we devoted 12 of the 25 (in total) class lectures, we closely follow the material of Wilkes,^[1] our adopted textbook, while for the microscopic analysis we partially used the textbook and to a larger extent additional material. The completion of each section of the course was followed by a project assignment for which students had to work in groups of four. This provided the students with the opportunity to reinforce the previously taught concepts within practical problems of contemporary interest, to work on research-level projects that give them an appreciation of the chemical engineering profession, and to improve their teamwork skills.

A key feature of the teaching methodology that was followed in this junior-level course was also the additional input provided by guest lecturers. Taking advantage of the long-standing collaborations between the University of Delaware and DuPont, we found that researchers from the chemical company responded positively in contributing to the teaching of the course. Daniel Woods, a DuPont expert on pumps, contributed with lectures on this particular subject and also provided extensive training on AFT Fathom, a software used by DuPont (among other companies) for pipe-flow analysis and modeling. Upon completion of the macroscopic fluid mechanics section, students worked on a pipeline project where—pretending to act as consultants in a pipeline design company—they had to redesign, using AFT Fathom, the Trans Alaskan pipeline.

The majority of the microscopic fluid mechanics section was taught by another engineering consultant and researcher of DuPont, Dr. James Tilton. Dr. Tilton is an expert on fluid mechanics and an affiliated faculty member at the Department of Chemical and Biomolecular Engineering at UD; he also has had important contributions to the fluid mechanics section of Perry's *Chemical Engineering* handbook. The prospect of having “people from the industry” teach portions of the course is something that motivated students and which they eagerly anticipated. Upon completion of this section of the course, students were introduced to CFD which, as elaborated later on, is needed in order to deal with research-level problems involving microscopic fluid dynamics.

There have been several significant efforts to incorporate CFD in the undergraduate chemical engineering curriculum.^[2-6]

The important contribution of our approach stems from the use of a contemporary biomedical application based on current research to better engage and motivate the students. Sinclair^[2] describes in detail the benefits of CFD modeling and illustrates the use of FLUENT via a fluid-particle-flow case study. Lawrence, *et al.*^[3] describe the incorporation of CFD into a junior-level course where students, with the use of CFX, had to predict the single-reaction conversion of a species in a non-ideal reactor. In a more recent work Kaushik, *et al.*^[4] show how CFD was introduced into a course to model the flow of water through sudden contraction and expansion in a horizontal pipe. In that particular course, students were also introduced to the drawing software GAMBIT, which is used to create the problem geometry and mesh it. Smith^[5] describes in detail a more advanced, senior-level CFD course that emphasizes the understanding of the numerical techniques involved. A better understanding of the underlying techniques ensures that CFD software is not used as a black box. However, this requires advanced coding that may be difficult to introduce at an undergraduate level. Most recently, Nijdam^[6] describes a one-dimensional transient diffusion problem that students were asked to solve analytically and numerically, using their own finite volume code. This represents a compromise between a course with a focus on code development and a course that teaches students how to use commercial CFD software, by focusing more on the fundamentals beneath numerical methods.

In all of the previously mentioned instances, the emphasis was placed on CFD rather than on the fluid mechanics fundamentals on which CFD relies. As a result, CFD occupied a very significant part, if not all, of the course. In our case, we introduce CFD into a standard fluid mechanics chemical engineering undergraduate course. For that to be possible, certain components such as the mesh generation had to be skipped. Furthermore, to still actively engage and motivate the students, the application needs to be carefully selected. In the following we show how this is possible based on current research performed in the department, in a contemporary issue of general interest, such as the cardiovascular arterial diseases. This introduction of research-based CFD application into teaching seems to fall in line as the best approach to integrate CFD into the chemical engineering undergraduate curriculum.^[7] Exactly as suggested in Reference 7, we implemented it here through the coordination of the instructor and a faculty member, with a graduate student responsible for the CFD research.

An in-depth introduction of a CFD software requires a significant amount of time. In the Fluid Mechanics course taught in our department, the goal is to familiarize students with the use of ANSYS FLUENT through a contemporary research example that can trigger their interest for further CFD research in the future. In this case, however, where CFD constitutes only a part of the course rather than the main

focus of it, emphasis must be placed on the correct use of the software and the prevention of “black box” simulations. We devoted one lecture of the course to introduce students to ANSYS FLUENT, the CFD software that has been used for the needs of this course, and to show its basic features. The teaching assistant held a separate three-hour session for a more elaborate demonstration of the capabilities and the features of the software. In addition, students were given a handout with the basic instructions on how to use ANSYS FLUENT, while an elaborate e-tutorial focusing on specific features of the software, such as results processing and visualization, was also created. Finally, in a homework problem that was assigned prior to the CFD project, students were asked to simulate a simple Poiseuille flow problem. This assignment served two purposes: a) having students practice with ANSYS FLUENT before proceeding with the more demanding CFD project, and b) preventing the “blind use” of ANSYS FLUENT, as students were required to compare the simulation results to their analytical solution of the problem. Finally, the students required approximately 3-4 hours for the pre-assignment and 15-20 hours for the CFD project.

In the following sections we provide the context within which this CFD project has been placed. First, we describe, through an overview of the basic fluid mechanics principles, how the need to perform CFD simulations arises. Then, we present some of the basic features of ANSYS FLUENT 12.1, the CFD software that has been used for the needs of this course. The third section includes a short description of the pre-project assignment. In the fourth section, we introduce the CFD project, which includes a) background information on the study of pathological conditions of the coronary as well as other arteries, b) the problem statement for the particular CFD project, c) the specific questions that students were asked, and d) a generic analysis of the problem. Finally, in the last section of this investigation we provide some of the course evaluation comments that we received from students.

FROM MACROSCOPIC BALANCES TO CFD

The starting point for any type of problem in the microscopic examination of an incompressible fluid flow is the combination of the mass continuity equation to the Cauchy momentum equation, which describes the momentum transport in any continuum medium. For an incompressible fluid the mass continuity reduces to the divergence-free condition:

$$\nabla \cdot \underline{v} = 0 \quad (1)$$

whereas the Cauchy momentum equation can be written as:

$$\rho \frac{D\underline{v}}{Dt} = \nabla \cdot \underline{\underline{\sigma}} + \underline{f} \quad (2)$$

where \underline{v} is the velocity vector field, $\underline{\underline{\sigma}}$ is the total stress tensor, \underline{f} contains all of the body forces per unit volume, and

TABLE 1
Course Structure

Part I – Macroscopic Fluid Mechanics	
1.	Introduction to Fluid Mechanics
2.	Mass, Energy, and Momentum Balances
3.	Fluid Friction in Pipes
4.	Flow in Chemical Engineering Equipment
5.	Pipeline Project
Part II – Microscopic Fluid Mechanics	
6.	Differential Equations of Fluid Mechanics
7.	Navier Stokes – Creeping Flow Examples
8.	Inviscid, Irrotational, and Porous-Media Flows
9.	Non-Newtonian Fluids
10.	CFD Project

$\frac{D\underline{v}}{Dt}$ is the material or substantial time derivative defined as:

$$\frac{D\underline{v}}{Dt} = \frac{\partial \underline{v}}{\partial t} + \underline{v} \cdot \nabla \underline{v} \quad (2a)$$

Based on the earlier description, the total stress tensor is split into pressure and the extra stress tensor:

$$\underline{\underline{\sigma}} = -p\underline{\underline{I}} + \underline{\underline{\tau}} \quad (3)$$

where $\underline{\underline{I}}$ is the identity tensor and $\underline{\underline{\tau}}$ is the extra stress tensor.

Eqs. (2)-(3) are still in need of additional information for the extra stress to be able to find a solution (open equations). All of the closed momentum-conservation equations, such as the Navier-Stokes equation, are obtained from those, by substituting the stress tensor, $\underline{\underline{\tau}}$, through a constitutive relationship, in terms of the velocity field and its derivatives. For example, for the Navier-Stokes equation Newton’s law of viscous fluids is used, which for incompressible fluids reads:

$$\underline{\underline{\tau}} = \eta \cdot (\nabla \underline{v} + \nabla \underline{v}^T) \quad (4)$$

where η is a non-negative scalar material parameter (the viscosity) and $\nabla \underline{v}$, $\nabla \underline{v}^T$ represent the velocity gradient and the velocity gradient transpose, respectively. It is this dependence of the extra stress on the velocity field characteristics that makes fluid velocity evaluation a necessity in fluid mechanics problems.

The problems addressed in the fluid mechanics textbooks are those that are solved analytically based on rough approximations. In most cases the fluids are assumed to be Newtonian, thus the solution is obtained through the Navier-Stokes (N-S) equations. Even with the assumption of a Newtonian fluid, however, most analytical solutions in use are for laminar, unidirectional, or axisymmetric flows. However, such conditions can only be met for flows in very simple geometries, for instance flow in a tube long enough to exclude any entrance effects, or flow around a sphere. For more complicated geometries the problems need to be solved numerically, employing state-of-the-art CFD software, such as FLUENT.

ANSYS FLUENT 12.1

The CFD software used for the purposes of this project is ANSYS FLUENT 12.1. A brief introduction of the basic components of FLUENT to the students is necessary to familiarize them with its user interface and with the basic setup and solution procedures.

To facilitate this task, a tutorial was prepared and handed out to the students, demonstrating the required steps toward the completion of a CFD simulation for given meshes (as was the case here, the mesh structures had been prepared by the TA). The basic build-up and execution processes include five generic steps, from launching the software and inputting the mesh files to the postprocessing of the results. The main steps are summarized in Table 2.

For the needs of this project the mesh files were created in ICEM ANSYS. Developing the mesh files for complicated geometries is arguably one of the most demanding parts of CFD. This is why for this project all the files have been generated by the TA and were made available to the students. These files, which essentially communicate the geometry of the problem to the CFD software, are used as input to ANSYS FLUENT.

Once the mesh file has been read, the user can perform the mesh-related activities and choose a solver. The main actions

1.	Launching ANSYS FLUENT and reading the existing mesh file
2.	Defining the geometry and fluid properties
3.	Setting the material properties and boundary conditions
4.	Initiating the calculations and calculating the solution
5.	Visually examining the output results using the post-processing tools of ANSYS FLUENT

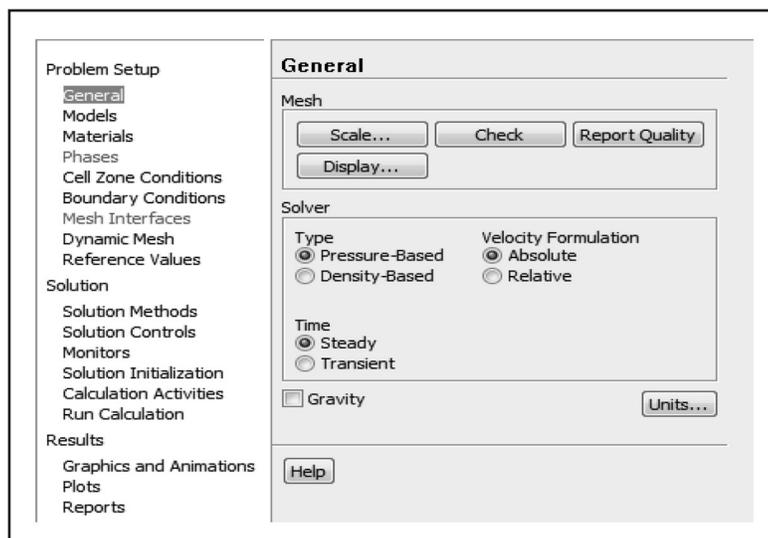


Figure 1. ANSYS® FLUENT® 12.1 interface.

are the scaling, if necessary, of the geometry and checking for errors in the input mesh file.

Figure 1 shows the ANSYS FLUENT interface, once the mesh file has been read. From the “General” tab (highlighted in Figure 1) the user can perform the mesh-related activities, such as rescaling and checking of the mesh quality, and choosing a solver. Next, depending on the flow conditions and the Reynolds number, the user has to choose the appropriate model. By default, FLUENT assumes a viscous-laminar model that is appropriate for laminar flow conditions. In the Problem Setup stage, the user has to define the material properties and the boundary conditions to be used. Once the Problem Setup section is complete the user has to provide an initial guess for the solver. This is done in the Solution Initialization tab. Only then can the simulation be executed, from the Run Calculation tab, under Solution.

One of the advantages of ANSYS FLUENT is the plethora of ways by which one can visualize or process the results. For instance, in the Reports tab one can view the mass, volumetric flow rate, or velocity for all inputs and outputs of the simulated geometry. Alternatively, using the Plots tab, one can obtain in an XY plot the centerline velocity or even view the velocity profile at the outlet. Probably the most recognizable feature of any CFD software, however, is the contour visualization. From the Graphics and Animations tab, the user can construct 2D or 3D contours for velocity, pressure, wall shear stress, etc. These basic features of ANSYS FLUENT are adequate for students to respond to the needs of the CFD project.

PRE-PROJECT ASSIGNMENT

Once the ANSYS FLUENT software was introduced and a demo illustrated, the students were asked in a pre-project exercise to work out a simple Poiseuille entrance flow problem. In this assignment, and for a given mass flow rate, students were asked to: a. plot the velocity profile and the wall shear rate (wall skin-friction coefficient in ANSYS FLUENT) at the outlet of a defined pipe geometry; b. compute the distance, from the inlet of the pipe, at which the flow becomes fully developed; and c. compare the results of questions a. and b. to their analytical solution of the problem.

This was a pre-project homework assignment and as such a report was required. The mesh files for this problem were prepared by the TA for the course.

THE CFD PROJECT

Upon completion of the pre-project homework assignment, the actual CFD project was distributed. The students worked in groups of four for the CFD project. The project statement was the same for all groups and copying was not an issue, as each group chose slightly different boundary conditions for the

solution of the problem. The project has three parts involving background material, problem statement, and questions. Those are described in detail below, followed by an analysis of the most important results.

Background information

The heart is an extraordinary organ that pumps blood via the blood vessels to provide nutrients to the trillion cells of the body. The work of feeding cells is of paramount importance to the sustainment of life and therefore this process can never stop. In that sense, the heart can be viewed as an extraordinary pump, as it operates continuously for an average lifetime of 78 years and it circulates an average of 5 liters per minute of blood.

To perform this vital work the heart needs to be fed. The supply of blood to the heart is done through the coronary arteries. There are two major coronary arteries that branch off from the aorta: the left coronary artery (LCA), also known as the left main artery (LM), and the right coronary artery (RCA); these feed blood to the left and right sides of the heart, respectively. The LM bifurcates into the left anterior descending artery (LAD) and the left circumflex artery (LCX) (Figure 2).

The vital role of coronary arteries for the physiological heart function also implies severe pathological conditions in case of damage to the arteries. Coronary artery disease (CAD) constitutes the most common type of heart disease and cause of myocardial infarctions (heart attacks). CAD occurs when the cells lining the arteries are damaged from plaques (fatty materials) accumulating on the vessel walls (Figure 3, next page). This buildup causes the arteries to become hardened and narrowed and as a result blood flow to the heart can be limited. This condition is known as atherosclerosis, while the narrowing of the arteries is also referred to as stenosis.

Coronary artery disease is the leading cause of death in the United States contributing an annual cost of \$108.9 billion. Almost one of every six deaths in the United States in 2009 was a result of a coronary heart disease. More than 385,000 people are killed every year from this type of disease, while this year alone ~635,000 Americans are expected to experience a new coronary attack.^[8] Projections show that by 2030, the prevalence of cardiovascular heart disease (CHD) will increase by ~18% from 2013 estimates.^[8] These devastating statistics justify the enormous amount of research being conducted on cardiovascular diseases and CAD in particular.

Concepts from fluid mechanics and hemodynamics are commonly used to determine and quantify the factors that promote atherosclerosis and therefore CADs. Various studies in the 1980s established wall shear stress (WSS) as a key biophysical force regulating endothelial cell and hence vascular function.^[9-12] The work of Friedman, *et al.*^[13,14] suggested that vessel geometry might be an independent risk factor for atherosclerosis due to the flow dynamics developed in certain configurations. Modeling the carotid bifurcation, Bharadvaj,

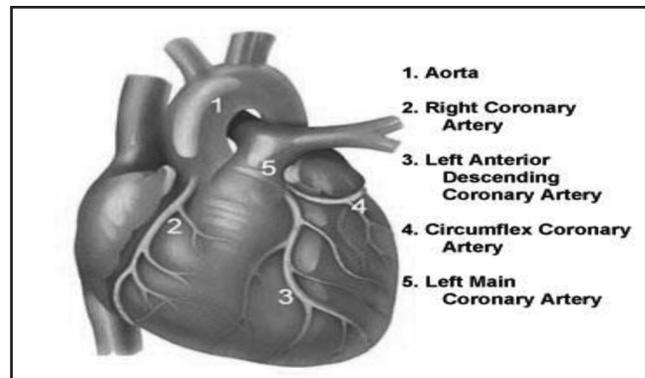


Figure 2. Topology of arteries in the heart <<http://www.froedterthealth.org>> (accessed 05/2013).

et al.^[15] identified a correlation between intimal thickening and oscillatory stress. This evidence, along with the work of Davies, *et al.*^[16]—which suggests an increased endothelial cell turnover in the presence of low shear regions—establishes the low and oscillatory WSS as a key player in the initiation and development of atherosclerosis. Subsequent *in vitro* studies show that low WSS (<1Pa) and oscillatory flow directly affect genetic expression within endothelial tissue, encouraging the expression of genes involved in atherogenesis, the early stages of atherosclerosis.^[17-19]

CFD simulations offer a valuable alternative to experimental studies and an easy way to obtain the flow dynamics and the WSS distribution in any vascular geometry. Asakura and Karino^[20] examined the flow patterns and distribution of atherosclerotic lesions in coronary arteries and found that the flow is either slow or disturbed, with the formation of secondary flows at the bifurcation sites. Their results further support the hypothesis of low fluid velocity and low shear stress dictating the localization of atherogenesis. He and Ku^[21] studied the average conditions in the LCA bifurcation and found the time-averaged mean WSS to vary from 0.3 Pa to 9.8 Pa. In both studies pulsatile flow conditions were employed, while blood was treated as a Newtonian fluid. Johnston, *et al.*^[22] simulated the steady flow of blood in four different right coronary arteries, using five non-Newtonian models as well as the Newtonian approximation. Their results show that, even though the pattern of WSS distribution is similar in all models, the magnitude of WSS is influenced by the model used. Fraunfelder, *et al.*^[23] assessed hemodynamic parameters, such as the mass flow and WSS, with CFD in coronary arteries using patient-specific data from computed tomography (CT) angiography. The flow was modeled as laminar, Newtonian, and pulsatile.

Problem statement

The CFD project regards the investigation of atherosclerosis in the left coronary artery (LCA), see Figure 2. The goal of this project is to familiarize students with ANSYS

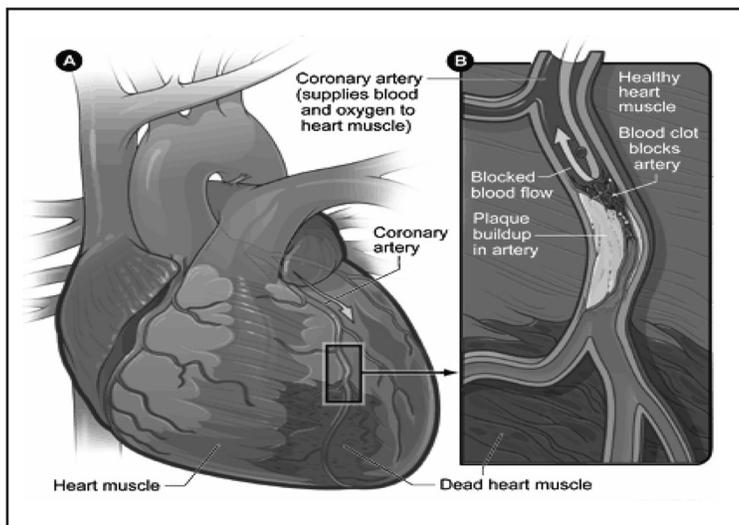


Figure 3. Coronary arteries of the heart. A: Topology, B: Manifestation of plaque buildup in the coronary artery <<http://www.nhlbi.nih.gov>> (accessed 05/2013).

FLUENT while offering them the opportunity to work on a contemporary problem based on real data that is also a subject of extensive research, as described in the previous sections. The students are asked to evaluate the impact of a stenosis on the physiological operation of the LCA (or LM) as well as to examine the possibility of restoring the physiological function via a bypass surgery by performing a CFD analysis of the problem, under steady state conditions.

For the system under investigation we use flow data and anatomically accurate dimensions as reported in the literature.^[24] The heart processes typically 5 l/min of blood, approximately 5% of which, under resting conditions, is used to feed the heart muscle. Under the assumption of equal distribution of this flow between the LM and RCA, the inlet average flow rate to the LM, $Q_{LM}=Q_0$, is readily available. This flow input is distributed from the LM to LAD and LCX and from there all the way down to the capillaries, the smallest vessels in the human body, where the pressure, P_c , can be assumed constant at 2000 Pa. The average pressure at the LM entry can be taken as ~ 100 mmHg ($P_{LM}=P_0=100$ mmHg). The dimensions and lengths for each artery are also known. All data are gathered in Table 3.

The volumetric flow rate, or equivalently the velocity, and the pressure at the outlets (LAD and LCX), Q_{LAD} , Q_{LCX} , P_{LAD} , and P_{LCX} , are unknown. These variables are the output of the CFD simulations.

The system under investigation, the left coronary artery LM and its first bifurcation (LCX and LAD), is only a small part of the network of vessels that transfer blood to the heart. Although with sophisticated software we can only focus on a limited component of the whole arterial network, nevertheless we also need to somehow account, even approximately,

for the impact that the rest of the network has on the flow under investigation. In particular, the pressure drop that results from the large number of vessels that exist between the LCX/LAD and the capillaries needs to be taken into consideration. This can be done using a resistance-model approach.

Just as in the case of an electric circuit, where the resistance R of an object is defined as the ratio of voltage across it, V , to current through it, I , we define the steady state resistance of a network of vessels as the ratio of pressure drop between the outlet of the artery (e.g., LAD), P_{LAD} , and the capillaries, P_c , to the volumetric flow rate through the specific network, Q_{LAD} . This quantity, in principle, can be determined by evaluating the individual resistances to the flow added by all vessels of arterial network, as we have demonstrated in a previous work^[25] where Non-Newtonian effects have also been taken into account. However, for simplicity in this project, we asked the students to approximate the resistances with an estimate based on the pressure and flow data of the physiological case. Thus, R_1 describes the average resistance in the flow of blood between LAD and the capillaries, while R_2 describes the average resistance between LCX and the capillaries. Even though the geometries of the mesh files consist only of the arteries LM, LAD, and LCX (and not the capillaries or any other vessels in between), the output of the simulation (Q_1, P_1, Q_2, P_2) should always satisfy the conditions set by R_1 and R_2 . The impact of the rest of the network is thus taken into consideration.

$$R_1 = R_{LAD} = \frac{|\Delta P|}{Q} = \frac{|P_c - P_{LAD}|}{Q_{LAD}} = \text{const.};$$

$$R_2 = R_{LCX} = \frac{|\Delta P|}{Q} = \frac{|P_c - P_{LCX}|}{Q_{LCX}} = \text{const.} \quad (5)$$

Questions

a. Using ANSYS FLUENT 12.1 and the mesh file physiological_case.msh (corresponding to the physiological case) find the resistances R_1 and R_2 for the flow between the LAD and the capillaries and the LCX and the capillaries, respectively.

Hint: Use velocity for inlet boundary condition (BC) and pressure for the outlet BCs. For the outlet BCs, set both pressures, P_1 and P_2 , equal to zero (relative values). Run the simulation. Estimate the absolute pressures, P_1 and P_2 . Based on those, calculate R_1 and R_2 .

Comments:

1. The pressure BCs are relative pressures with respect to an internal reference pressure, typically taken to be the LAD outlet pressure. Using relative pressures with respect to a zero internal reference pressure ($P_1=0$) is the best way to conduct FLUENT simulations in order to avoid instabilities and

excessive numerical errors. One can always do that as the flow is incompressible and only relative pressure difference matters to the flow.

2. While we are using equal outlet pressures ($P_1=P_2=0$) for the simulation, in real physiological data this would not be the case. This is a working approximation used for simplicity. Another approximation is the steady state examination, as in reality the blood flow in the heart is pulsatile.

b. For given resistances in the two networks use ANSYS FLUENT 12.1 and the mesh file stenosis.msh to estimate the impact of the stenosis on the flow (the stenosis of this mesh file corresponds to 90% occlusion). Answer the following:

- i. Is the impact of the stenosis significant? What is the decrease (%) in the flow rate through LAD that is caused by the stenosis?
- ii. You are a surgeon. Your patient has a 90% stenosis in the LAD. The decrease (%) of the flow rate through his LAD is the one you calculated in part i. Would you proceed with a bypass operation? Explain.

Hint: Use a velocity inlet BC and pressure outlet BCs. Set one of the two BCs to zero. Run the simulation and check whether the resistance criteria are met. The resistances are constant and therefore you must aim at getting the same R_1 and R_2 that you got in question a. This is the convergence criterion. Trial and error are needed, each time varying the non-zero valued outlet BC and potentially the inlet BC as well, until the simulation output results in the desired R_1 and R_2 .

c. The patient has decided to opt for the bypass operation, regardless of your feedback. For given resistances use FLUENT and the mesh file bypass.msh to compute the fluxes (Q_{LAD} , Q_{LCX}) and the pressures (P_{LAD} , P_{LCX}) of the two outlets. To what extent is the flow restored after the bypass operation?

Analysis of a typical project solution

At first, the students need to calculate R_1 and R_2 , which correspond to the resistances between LAD and the capillaries and the LCX and the capillaries, respectively. Since the

resistance is defined as $\frac{|\Delta P|}{Q}$, one needs the pressure at LAD and LCX, along with the corresponding flow rates.

In the pressure contours of Figure 4 the relative pressure from either one of the two outlets is displayed. The pressure at the outlets has therefore been set to zero (dark gray box at the bottom of color bar in Figure 4) as boundary condition (BC), as described in the previous section. For zero valued pressure BCs at the outlets, the simulation

shows that the maximum pressure appears at the inlet and is ~ 105 Pa. Since we know from the problem statement that the absolute pressure at the entrance of LM is 100 mmHg (~ 13330 Pa) we can therefore deduce the absolute pressure at LAD and LCX (e.g., the absolute pressure at LAD would be, in this case, $13330 - 105 = 13225$ Pa) and with this information compute R_1 and R_2 . These should not change in all three cases we examine. Having run the simulation for the healthy case, one can easily gather the results presented in Table 4, for all three arteries.

When running the simulations for the other two cases—the stenosis and the bypass geometries—one has to make sure that the resistances that are computed in those cases, based on the FLUENT results, are identical to the resistances of the physiological case. A modification of the boundary conditions

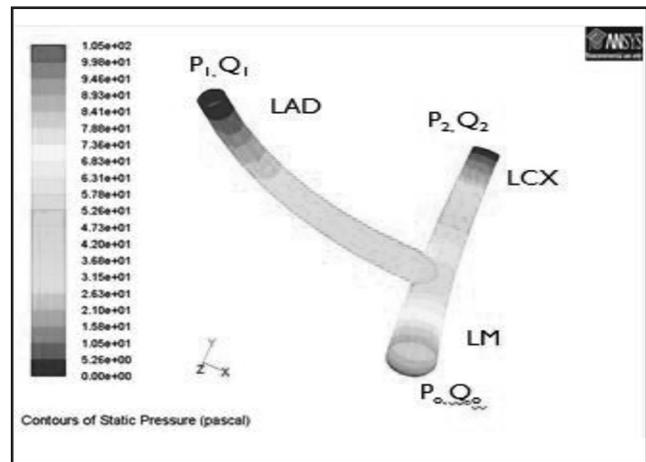


Figure 4. Pressure contours (in Pa).

	LM	LAD	LCX
Length, L (cm)	0.9	2.7	1.06
Diameter, D (mm)	3.8	2	2.3
Volumetric flow, Q (l/min)	0.125	-	-
Pressure, P (mmHg)	100	-	-

Location	LM	LAD	LCX
Relative Pressure	105 Pa	0 Pa (BC)	0 Pa (BC)
Volumetric Flow Rate	2.080×10^{-6} m ³ /s	5.589×10^{-7} m ³ /s	1.522×10^{-6} m ³ /s
Velocity	0.1837 m/s (BC)	0.1779 m/s	0.3364 m/s
Reynolds Number	211.4	107.8	255.2
Blood Pressure	13332 Pa	13227 Pa	13227 Pa
Resistance	--	1.983×10^{10} kg/m ⁴ s	7.280×10^9 kg/m ⁴ s

would therefore be required to achieve that. For instance, the pressure BC at the LAD should be altered due to the presence of the stenosis. We expect a lower value compared to that of the physiological case, since in this case, due to the added resistance to the flow from the stenosis, the approximation of equal pressures at LAD and LCX would not be appropriate.

Some converged boundary conditions for the simulation of the “stenosis” case are shown in Table 5. For simulation stability reasons, one of the pressures has to be set equal to zero. This implies that the LAD pressure BC should be a negative number. Students are expected to try different predictions until the resistance criterion is met to within some reasonable tolerance, say 1%. Once this has been achieved

Location	LM	LAD	LCX
Relative Pressure	--	-750 Pa	0 Pa
Velocity	0.1837 m/s	--	--

then the students have all the information needed to answer questions b. and c. of the problem statement.

Finally, for some more in-depth analysis students can use the contour predictions for the wall shear stress. Some typical results are shown in Figure 5. We know from theory that one of the phenotypes for atherogenesis, the early stages of atherosclerosis, is a low WSS in the arteries. Thus, from the WSS contour of the physiological case in Figure 5, one can conclude that the most susceptible region for fatty acid buildup and subsequent stenosis is the very beginning of the LAD artery, right after the bifurcation, where the WSS is less than 1 Pa. Similarly, the stenosis and bypass contours of WSS can be used to evaluate the risk of having a new stenosis developed.

COURSE EVALUATIONS

Student feedback on the course and project was very positive. At the end of the semester, the students filled out a questionnaire asking them to respond to the following statements about their experience in the course. The options given were 1 = Definitely False, 2 = More False than True, 3 = In Between,

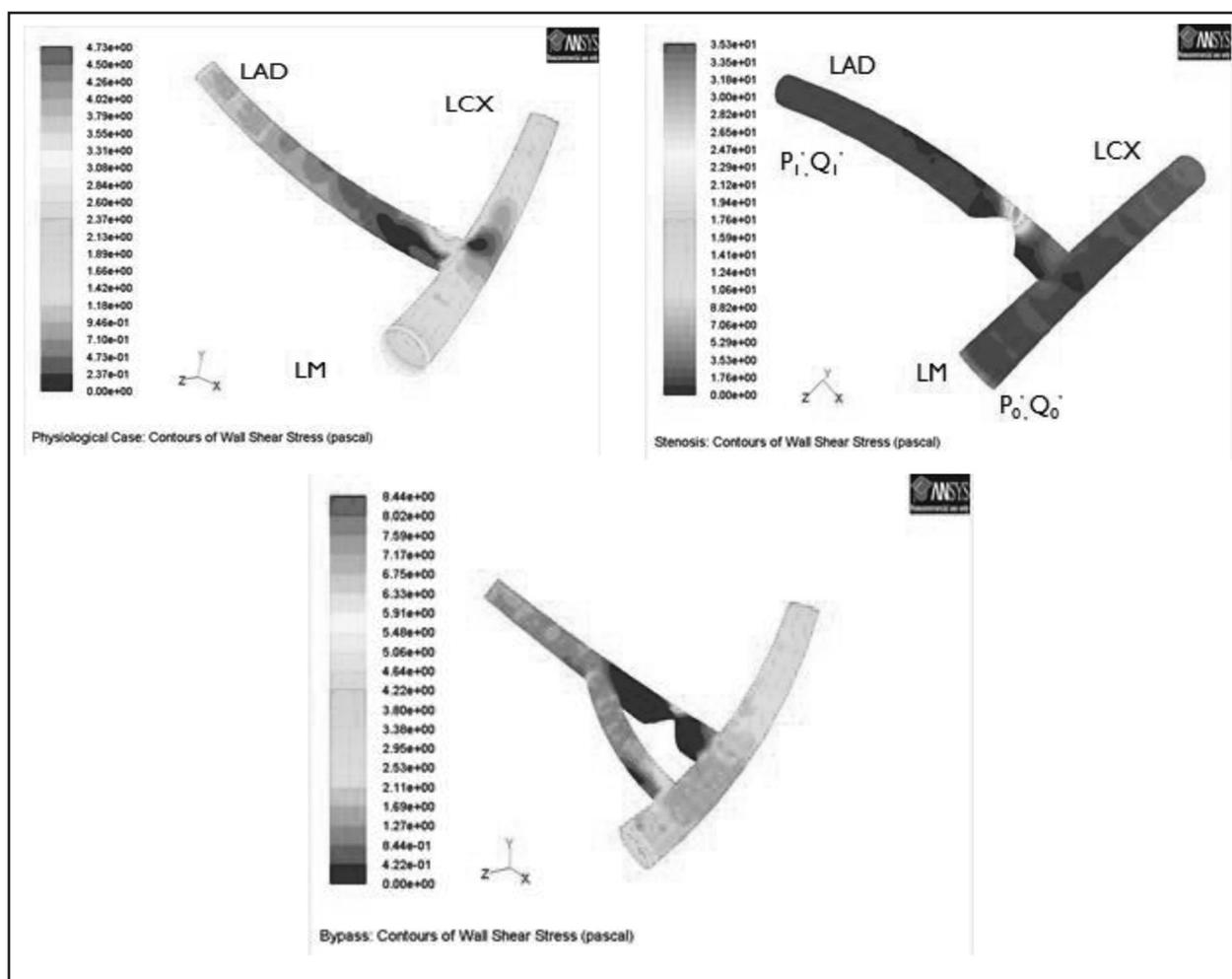


Figure 5. Wall shear stress (WSS) contours for all three geometries. (WSS units: Pa.)

4 = More True than False, and 5 = Definitely True. Some questions relevant to the CFD project were:

- *Q1: The projects were a good complement to the class lectures and enhanced my understanding of fluid mechanics.*
- *Q2: The following objective was achieved: “Working effectively in teams by optimal distribution of workload and achieving common objectives within time constraints (e.g., Pipeline project, CFD project, etc.)”*
- *Q3: The CFD project helped me learn advanced computational software (ANSYS FLUENT) and use it to numerically solve the Navier-Stokes Equations for a problem of practical interest.*
- *Q4: The following ABET objective was achieved: “Develop quantitative problem-solving skills by applying knowledge of mathematics (e.g., Differential equations, vector calculus) to model and analyze fluid flow.”*

The results from the questionnaire are included in Table 6.

The total number of students enrolled in the Fluid Mechanics class, CHEG 341, was 79. The class was split into two sections, CHEG 341-010 with 35 students and CHEG 341-011 with 44 students.

As can be seen from Table 6, the students’ opinions of the course were highly favorable from both sections. The strongest agreement appeared to be in response to Q2, regarding teamwork and collaboration promoted through the project. The CFD project was welcomed by the students and the opportunity to work on a problem of practical, contemporary interest was appreciated. Many positive comments were made by the students on the structure of the course and the assignment of practical project assignments. A few of those comments are presented below:

- *“I’m fairly certain that fluid mechanics would still be fairly abstract to me if the projects were not part of this course.”*
- *“I loved the CFD project and enjoyed exploring a medical application to chemical engineering.”*
- *“Projects were excellent in cementing concepts.”*
- *“Projects were interesting, thought-provoking, and made us use our knowledge of what we learned.”*
- *“Definitely emphasized the application of the concepts and there was always a feeling of importance in what we were doing. The projects were relevant and enjoyable.”*
- *“I liked the projects for the course and how they related to the material in class. I think they helped me to learn*

			Q1	Q2	Q3	Q4
CHEG 341-010	Fall 2012	Mean	4.68	4.84	4.5	4.66
35 students		Std Dev	0.65	0.45	0.57	0.83
			Q1	Q2	Q3	Q4
CHEG 341-011	Fall 2012	Mean	4.78	4.65	4.5	4.7
44 students		Std Dev	0.48	0.68	0.88	0.62
			Q1	Q2	Q3	Q4
Overall	Fall 2012	Mean	4.74	4.73	4.5	4.68
		Std Dev	0.6	0.5	0.7	0.7

the material well and made the material more interesting by relating it to real-life situations.”

- *“Projects were rewarding—real-world stuff!”*
- *“It was interesting and moved at a good pace. Loved the CFD project, helped solidify the material. It is good that we got the chance to learn a new software.”*

CONCLUSIONS

This paper reports on how an innovative CFD project was incorporated into a junior-level fluid mechanics course. The project was based on a contemporary problem that is currently the subject of active research in the department. The project helped motivate the students into both appreciating the usefulness of microscopic fluid mechanics as well as encouraging them to overcome the activation energy required to learn and use CFD software. We believe that this is essential for a well-rounded chemical engineering education.

NOMENCLATURE

CFD	Computational Fluid Dynamics
BC	Boundary Condition
CAD	Coronary Artery Disease
CHD	Coronary Heart Disease
LAD	Left Anterior Descending
LCA	Left Coronary Artery
LCX	Left Circumflex
LM	Left Main Coronary Artery
N-S	Navier Stokes Equation
RCA	Right Coronary Artery
WSS	Wall Shear Stress

ACKNOWLEDGMENTS

We would like to thank Dr. James Tilton and Dr. David Johnson for their important contributions to the organization, teaching, and completion of this project-based course. We also thank Dr. Natalie Germann for her assistance in preparing the ANSYS FLUENT tutorial, and Dr. Minye Liu for his help on questions related to ANSYS FLUENT. Finally, we would like to thank the students for actively participating in

the course and providing their feedback. This material was based upon work (of Alex Apostolidis and Antony Beris) supported by the National Science Foundation under Grant No. CBET 1033296. Any opinions, findings, and conclusions or recommendations expressed in this material are those of the author(s) and do not necessarily reflect the views of the National Science Foundation.

REFERENCES

1. Wilkes, J.O., *Fluid Mechanics for Chemical Engineers*, 2nd Ed., Prentice Hall (2010)
2. Sinclair, J.L., "CFD Case Studies in Fluid-Particle Flow," *Chem. Eng. Ed.*, **32**(2), 108, (1998)
3. Lawrence, B.J., J.D. Beene, S.V. Madihally, and R.S. Lewis, "Incorporating Nonideal Reactors in a Junior-Level Course Using Computational Fluid Dynamics (CFD)," *Chem. Eng. Ed.*, **38**(2), 136 (2004)
4. Kaushik, V.V.R., S. Ghosh, G. Das, and P.K. Das, "CFD Modeling of Water Flow Through Sudden Contraction and Expansion in a Horizontal Pipe," *Chem. Eng. Ed.*, **45**(1), 30 (2011)
5. Smith, M.K., "Computational Fluid Exploration As an Engineering Teaching Tool," *Int. J. Eng. Ed.*, **25**(6), 1129 (2009)
6. Nijdam, J.J., "Mesh and Time-Step Independent Computational Fluid Dynamics (CFD) Solutions," *Chem. Eng. Ed.*, **47**(4), 191 (2013)
7. LaRoche, R.D., "Integration of Computational Fluid Dynamics (CFD) Into the Chemical Engineering Undergraduate Curriculum. A Report From CACHE CFE Task Force," AICHE Annual Meeting & Centennial Celebration (2008)
8. Go, A.S., et al. "Heart Disease and Stroke Statistics-2013 Update: A Report From The American Heart Association," *Circulation*, **127**(1), e6 (2013)
9. Langille, B.L., and S.L. Adamson, "Relationship Between Blood Flow Direction and Endothelial Cell Orientation at Arterial Branch Sites in Rabbits And Mice," *Circ. Res.*, **48**(4), 481 (1981)
10. Nerem, R.M., M.J. Levesque, and J.F. Cornhill, "Vascular Endothelial Morphology As an Indicator of the Pattern of Blood Flow," *J. Biochem. Eng.*, **103**(3), 172 (1981)
11. Langille, B.L., and F. O'Donnell, "Reductions in Arterial Diameter Produced by Chronic Decreases in Blood Flow Are Endothelium-Dependent," *Science*, **231**(4736), 405 (1986)
12. Zarins, C.K., M.A. Zatina, D.P. Giddens, D.N. Du, and S. Glagov, "Shear Stress Regulation of Artery Lumen Diameter in Experimental Atherogenesis," *J. Vasc. Surg.*, **5**(3), 413 (1987)
13. Friedman, M.H., G.M. Hutchins, C.B. Barger, O.J. Deters, and F.F. Mark, "Correlation Between Intimal Thickness and Fluid Shear in Human Arteries," *Atherosclerosis*, **39**(3), 425 (1981)
14. Friedman, M.H., O.J. Deters, F.F. Mark, C.B. Barger, and G.M. Hutchins, "Arterial Geometry Affects Hemodynamics. A Potential Risk Factor for Atherosclerosis," *Atherosclerosis*, **46**(2), 225 (1983)
15. Bharadvaj, B.K., R.F. Mabon, and D.P. Giddens, "Steady Flow in a Model of the Human Carotid Bifurcation. Part I-Flow Visualization," *J. Biomech.*, **15**(5), 349 (1982)
16. Davies, P.F., A. Remuzzi, E.J. Gordon, C. Forbes Dewey Jr., and M.A. Gimbrone Jr., "Turbulent Fluid Shear Stress Induces Vascular Endothelial Cell Turnover *In Vitro*," *Proc. Natl. Acad. Sci. USA*, **83**(7), 2114 (1986)
17. Dewey, C.F., S.R. Bussolari, M.A. Gimbrone Jr., and P.F. Davies, "The Dynamic Response of Vascular Endothelial Cells to Fluid Shear Stress," *J. Biochem. Eng.*, **103**(3), 177 (1981)
18. Resnick, N., and M.A. Gimbrone Jr., "Haemodynamic Forces Are Complex Regulators of Endothelial Gene Expression," *FASEB J.*, **9**(10), 874(1995)
19. Nagel, T.E., N. Resnick, C. Forbes Dewey Jr., and M.A. Gimbrone Jr., "Differential Responses of Endothelial Cells to Uniform and Disturbed Laminar Shear Stress," *J. Vasc. Interv. Radiol.*, **10**(7), 952 (1999)
20. Asakura, T., and T. Karino, "Flow Patterns and Spatial Distribution of Atherosclerotic Lesions in Human Coronary Arteries," *Circ. Res.*, **66**(4), 1045 (1990)
21. He, X., and D.N. Ku, "Pulsatile Flow in the Human Left Coronary Artery Bifurcation: Average Conditions," *J. Biomech. Eng.*, **118**(1), 74 (1996)
22. Johnston, B.M., P.R. Johnston, S. Corney, and D. Kilpatrick, "Non-Newtonian Blood Flow in Human Right Coronary Arteries: Steady State Simulations," *J. Biomech.*, **37**(5), 709 (2004)
23. Frauenfelder, T., E. Boutsianis, T. Schertler, L. Husmann, S. Leschka, D. Poulidakos, B. Marincek, and H. Alkadhi, "In-Vivo Flow Simulation in Coronary Arteries Based on Computed Tomography Datasets: Feasibility and Initial Results," *Eur. Radiol.*, **17**(5), 1291 (2007)
24. Johnson, D.A., U.P. Naik, and A.N. Beris, "Efficient Implementation of the Proper Outlet Flow Conditions in Blood Flow Simulations Through Asymmetric Arterial Bifurcations," *Int. J. Numer. Meth. Fluids*, **66**(11), 1383 (2011)
25. Johnson, D.A., J.R. Spaeth, W.C. Rose, U.P. Naik, and A.N. Beris, "An Impedance Model For Blood Flow in the Human Arterial System. Part I: Model Development and MATLAB Implementation," *Computers in Chem. Eng.*, **35**(7), 1304 (2011) □