



## Integration of CFD and EFD for Experiential Learning in Fluid Mechanics

**Dr. Quamrul H. Mazumder, University of Michigan, Flint**

Dr. Quamrul Mazumder is currently a professor of mechanical engineering at University of Michigan-Flint. His areas of research include computational fluid dynamics, multiphase flow, quality in higher education, metacognition, motivation, and engagement of students. He is a Fulbright scholar travelled around the world to promote quality and globalization of higher education.

**Mr. Mohammed ASLAM, University of Michigan - Flint**

Mohammed Aslam Biography: Mechanical Engineer, Graduated with master's in mechanical engineering from Wayne state university in 1981 • Currently a Part time faculty at UM Flint Campus • Recently retired from Delphi as senior staff engineer • 30 years of automotive experience in product design and development • Recipient of various patents in pump technology and presented paper in SAE symposium • Board member of Flint Islamic Center and school board member of Genesee Academy

**Fardeen Mazumder, University of Michigan-Flint**

Undergraduate Research Assistant, University of Michigan-Flint, USA

# **Integration of CFD and EFD for Experiential Learning in Fluid Mechanics**

## **Abstract**

Computational Fluid Dynamics (CFD) has become an essential tool for the solution and analysis of fluid mechanics and heat transfer problems over the past few decades. CFD simulation can provide valuable insight into fluid flow behavior and proven to be a feasible tool for modeling complex fluid flow phenomena with a better understanding of the flow characteristics. CFD analysis uses physical laws to provide solutions for fluid flow problems in the form of partial differential equations. The undergraduate fluid mechanics curriculum is primarily based on analytical fluid mechanics (AFD) where students are required to solve problems using conservation of mass, momentum and energy equations. Solutions of differential and integral equations required for analysis of fluid flow characteristics may pose challenges as students are not be able to fully comprehend the concepts and may become disinterested in the subject. The integration of CFD into fluid mechanics curriculums is important as it can provide in-depth knowledge and understanding of flow behavior using visual images and graphical user interfaces. A commercial CFD code FLUENT was used to introduce concepts and applications of computational fluid dynamics in an undergraduate fluid mechanics course. Three different analyses were performed to study fluid flow using CFD: blood flow through an artery, airflow over the airfoil, and particulate multiphase flow in elbow geometry.

To validate the CFD analysis results, the experimental fluid dynamics (EFD) approach was introduced to analyze flow over an airfoil and flow inside an elbow. A wind tunnel was used to perform experiments to measure the flow over an airfoil. Using all three approaches (AFD, CFD, and EFD) in the undergraduate fluid mechanics course provided better knowledge and understanding of different fluid flow phenomenon. Students were able to visualize the flow characteristics using both Eulerian and LaGrangian methods. A survey was conducted to assess student learning and their perception of the integration of CFD and EFD. The results of the survey showed positive feedback with an improved understanding of fluid flow characteristics. The motivations of students were also increased along with their skills in using approaches to solve complex fluid flow problems.

## **Introduction**

Computational Fluid Dynamics (CFD) uses a wide variety of numerical methods developed for solving complex fluid flow problems using computer simulation with graphical outputs. This method was developed as a result of early approaches to solve the Navier-Stokes equation. In more recent history, CFD has become an essential tool in engineering analysis due to the rapid increase in computer processing power and technology. CFD can be used in the analysis of different types of fluid flow problems that are complex and cannot be solved using classical equations. It also provides an easy to understand the visual image of fluid flow characteristics such as pressure, velocity, temperature profiles, etc. To effectively integrate CFD in the undergraduate fluid mechanics course, three different examples were used 1) Blood flow through an artery (Biomedical), Flow over an airfoil (Aeronautical), Erosion in elbow

(Mechanical/Civil). Arteries are used to transport blood using the pressure developed by the pumping action of the heart. The pulse, which can be felt over an artery lying near the surface of the skin, results from the alternate expansion and contraction of the arterial wall as the beating heart forces blood into the arterial system via the aorta.

Blood flow through an artery is laminar due to low velocity, which means that blood flows in parallel layers with no disruption between the layers. In some cases, the blood flow may be turbulent due to restrictions and blockages in the arteries when vascular conditions disrupt the flow and increase the velocity. In the human circulatory system, turbulent flow is seen in the aorta, in the region of vascular bifurcations, and distal to areas of stenosis [1]. The difference between laminar flow and turbulent flow is illustrated in Figure 1.



Figure 1: Depiction of laminar and turbulent flows.

To perform CFD analysis of the artery, the geometry file was provided to the students and all the conditions were defined. The CFD analysis result showed flow characteristics of an artery's pressure, temperature, and velocity as shown in figure 2.

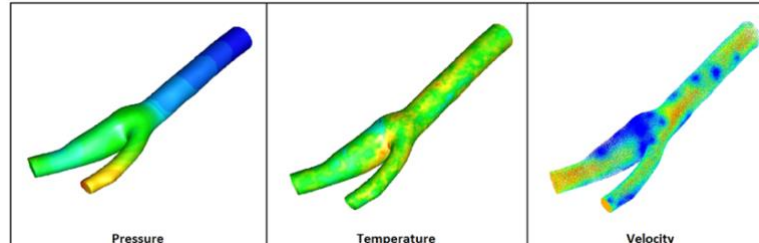


Figure 2: Contours of pressure, temperature, and velocity along walls of artery.

A CFD analysis was performed in a NACA 2415 airfoil shown in Figure 3. The geometry of an airfoil is defined in terms of the thickness, chord length, and the detailed coordinates of the upper and lower surfaces. The rounded edge at the front of the wing, and the pointed edge at the end of the wing, are referred to as the leading and trailing edges. The chord is an imaginary straight line connecting the leading and trailing edges of an aero foil. The straight line drawn from the leading to trailing edges of the airfoil is called the chord line which divides the airfoil into an upper surface and a lower surface. If the points that lie halfway between the upper and lower surfaces are plotted, a mean camber line is obtained which is an important feature of the airfoil geometry as cambered wings lift more easily than flat wings. Manipulation of the camber shape can greatly affect the overall performance of the airfoil by changing the values of pressure and velocity of the air passing over the wing surfaces.

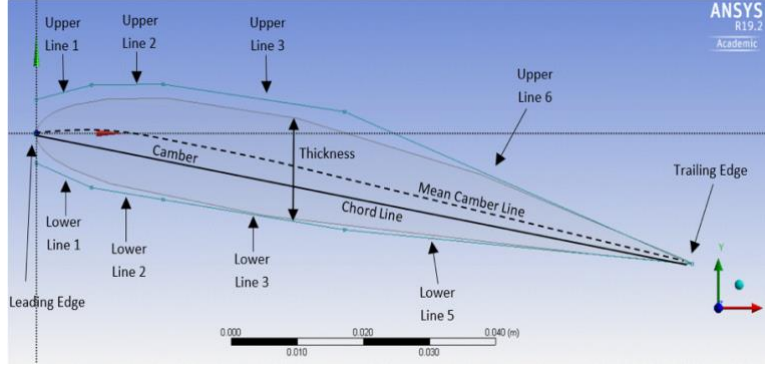


Figure 3: NACA 2415 Airfoil Geometry

In most industrial processes, fluids are used as a medium for material transport in pipelines or hoses. Complete knowledge of the principles and phenomena involving fluids transportation leads to a more efficient system. However, in many industries, such as petroleum, chemical, oil, and gas industries, two-phase or multiphase flow is frequently observed [2]. Multiphase flow involves a flow of solid, liquid and/or gas phases in the fluid. Although multiphase flow is common in industrial applications, there is no classical equation or analysis available to analyze the flow characteristics. Due to this limitation, multiphase flow is analyzed using experimental or simulation methods. In a liquid-solid or gas-solid multiphase flow in an elbow, the entrained solid particles impinge in the inner wall of the elbow resulting in material loss called erosion. The concentration of solid particles in the analysis was 2% and therefore the gas or liquid phases were dominant phases or continuous phases. The solids are called discrete phases. Flow simulation of the continuous fluid (carrier fluid) is the first step of the CFD-based erosion prediction procedure [3]. CFD analysis uses governing equations of motion, fluid flow and opera the fundamental laws of fluid mechanics. The governing equations are the conservation of mass equation:

$$(\partial \rho / \partial t) + \nabla \cdot (\rho \mathbf{V}) = 0$$

and the conservation of momentum equation:

$$\rho(\partial \mathbf{V} / \partial t) + \rho(\mathbf{V} \cdot \nabla) \mathbf{V} = -\nabla p + \rho \mathbf{g} + \nabla \cdot \tau_{ij}$$

These equations, along with the conservation of energy equation, form a set of coupled, nonlinear partial differential equations. Since CFD is primarily based on Navier-Stokes equations, the results are displayed in graphical images with different color contours of pressure, velocity, and other flow parameters, these results can be used in engineering design and application of different systems and can be used to throughout the design process [4].

The accuracy of the analysis, and the computational time needed to perform these calculations, is determined by the element size (mesh density). CFD models that use a finer mesh (small element size) provide the most accurate results but take longer computational time. In contrast, the use of a coarse mesh (large element size) will yield less accurate results in a small amount of time [5]. Mesh sensitivity analysis must be performed to determine the optimum mesh size for the analysis to attain accurate results. In mesh sensitivity analysis mesh refinements are performed to observe any effect of mesh size on analysis results.

Students were provided detail instructions and tutorials about the CFD simulations and the software to be used to perform the analysis. After the CFD simulations were performed, students were able to develop conceptual understanding, importance, and how it can be used to perform complex fluid mechanics analysis. The importance of validation of the CFD results through experiments were emphasized during the process. Due to limitations of the experimental facilities and time allowed during the course, the experiment was conducted to validate the flow over an airfoil. Experimental Fluid Dynamics (EFD) is the physical testing of concepts learned in the classroom and through analytical fluid dynamics (AFD) and/or CFD. EFD provides higher confidence in the CFD results where experimental results are compared to CFD results. The variations between CFD and EFD results were discussed as the uncertainties associated with different parameters during both simulation and experimental investigations. Due to the limited scope of this study, a detail uncertainty analysis was not covered.

### Case Study One: Blood Flow Through Artery

Computational fluid dynamics (CFD) were used to analyze blood flow through an aortic artery as shown in Figure 4. A commercial CFD code Ansys FLUENT was used to perform the analysis. Pressure and velocities at seven different locations were investigated: the large and small inlets, the outlet, the intersection of the large and small artery branch, and locations 1 and 2. The characteristic flow properties were observed by visualizing the pressure and velocity contour plots in these cross-sectional areas.

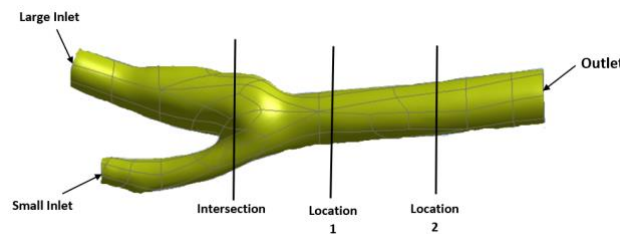


Figure 4: Artery geometry description.

The blood flow was modeled as laminar flow with a density of  $1060 \text{ kg/m}^3$ , the specific heat of  $3513 \text{ J/Kg-K}$ , the thermal conductivity of  $0.44 \text{ W/m-K}$ , and a viscosity of  $0.003 \text{ Kg/m-s}$ . The blood entered through the two branches of the inlet at  $0.3 \text{ m/sec}$  velocity and left through the large main branch of the artery outlet. Mesh sensitivity analysis revealed the optimum mesh configuration with 139,202 elements and 27,309 nodes. The meshed artery is presented in Figure 5.

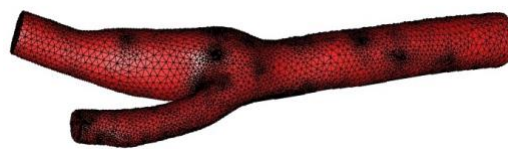


Figure 5: Depiction of fine mesh configuration

Blood flow refers to the movement of blood through a vessel, tissue, or organ and is initiated by the contraction of the ventricles of the heart. Ventricular contraction ejects blood into the major arteries, resulting in flow from regions of higher pressure to regions of lower pressure [6]. This phenomenon was observed by tracking the blood flow streamlines from inlet to outlet and observing the variations of pressure as presented in pressure contour plots of the seven cross-sections. As the blood travels through the branches, it flows toward the larger artery towards the artery outlet. At the intersection of two smaller arteries, pressure loss can be observed as the blood enters the larger artery as shown in Figure 6. As the flow exits through the outlet, a uniform, concentric velocity contour is observed with a higher velocity at the center of the artery.

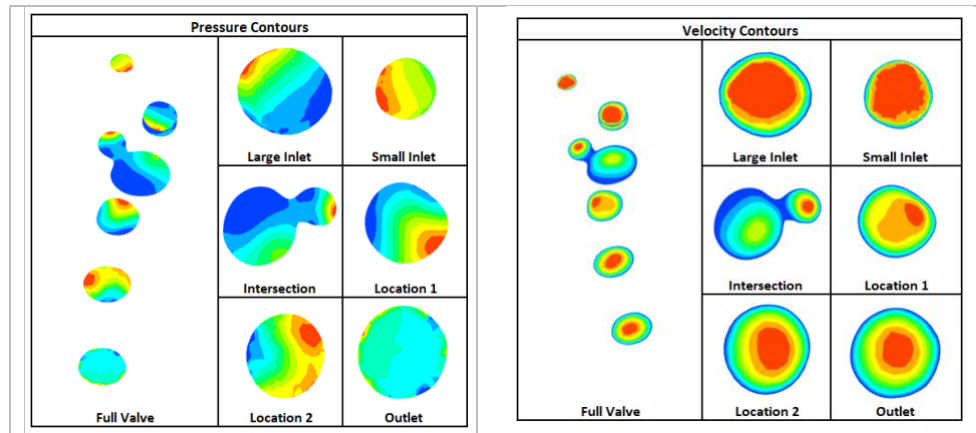


Figure 6: Pressure & velocity contours at different locations in the artery.

The rate of blood flow is proportional to the total cross-sectional area of the blood vessels. As blood flows through the vessel, it encounters resistance or a force that opposes the flow of a fluid. In blood vessels, most of the resistance is due to vessel diameter. As artery diameter decreases, the resistance increases and blood flow decreases [6].

As blood enters the artery through the inlets at a high velocity, the velocity decreases as it travels through the artery due to frictional head losses in the artery walls. As the flow converges from two inlet arteries to a larger artery, the velocity decreases further. The maximum velocity in the artery is observed at the center of the artery that is typical for laminar flow in pipes. Streamline plots of flow through the artery as shown in Figure 7 shows highest velocity at the downstream of the intersection of two smaller inlet arteries. The maximum streamline velocity is displayed in red in the figure with the minimum velocity regions in blue.

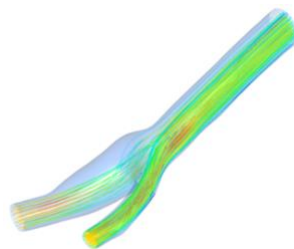


Figure 7: Depiction of velocity streamline flow through artery.

## **Study 2: External Flow Over an Airfoil**

The objective of this CFD analysis to simulate the velocity and pressure distributions over a NACA 2415 airfoil. The CFD analysis results were validated using experiments using a wind tunnel. Ansys Fluent software was used to perform CFD analysis. During the simulation, the airfoil model was positioned at an angle of attack of 10 degrees and 75 ft/sec inlet air velocity.

Before the CFD analysis, the NACA 2415 airfoil geometry file was provided to the students along with a tutorial describing the CFD analysis process. A three-dimensional external cube was drawn around the airfoil model to analyze the pressure and velocities in the cube around the airfoil.

The XY-plane was used as the reference plane because during simulation when analyzing how velocity and pressure affect the model. The inlet and outlet regions were defined for the direction the fluid flow over the airfoil with walls were established around the airfoil to specify the area to be analyzed. Air with standard properties was used as the fluid around the airfoil. After completion of the CFD analysis, post-processing of the results showed static pressure, the magnitude of velocity, as well as the pressure and velocity, streamline lines as the fluid, flows over the airfoil.

A commercial CFD code FLUENT was used to simulate airflow at 75 ft/s (22.86 m/s) over the airfoil profile 2415. The pressures at different locations on the upper and lower surfaces of the airfoil were analyzed. The lowest pressure on the upper surface was observed downstream of the stagnation point. As air flows over the airfoil, the upper surface pressure increased from the leading edge to the trailing edge. The lower surface pressure was higher than the upper surface that generates lift in the airfoil.

The contour plots showed the highest pressure at the stagnation point. The streamlines displayed the lowest pressure on the upper surface of the airfoil and higher pressures on the lower surface of the airfoil. Analysis of velocity profiles showed the highest velocity at 0.005 m from the leading edge at approximately 32m/s (104.987 ft/s) with the lowest pressures.

### **Validation of CFD using Experimental Fluid Dynamics (EFD):**

Experiments were conducted in the airfoil model using a wind tunnel as shown in Figure 8. A Comparison of CFD and EFD results is plotted in Figure 9 showing similar trends. However, there were differences between CFD and EFD results due to uncertainties associated with the experimental methods and CFD results. The CFD analysis results were plotted relative to the X-direction, whereas the wind tunnel data acquisition software utilizes sensors to calculate pressures on the upper and lower surfaces.

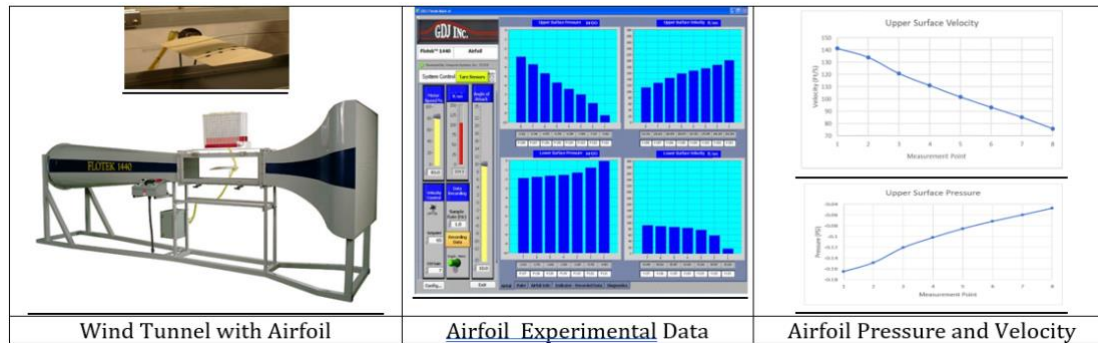


Figure 8: Experimental Fluid Dynamics Results

During the CFD experiment, several uncertainties affected the accuracy of the data. One of the issues was due to the assumption used in CFD simulation as Z-coordinate set at 0 that is different than the experimental data. The line/rake points were set at different X-coordinates from each other. After adjustment of the X-co-ordinates, the results were found to be in a better agreement with each other.

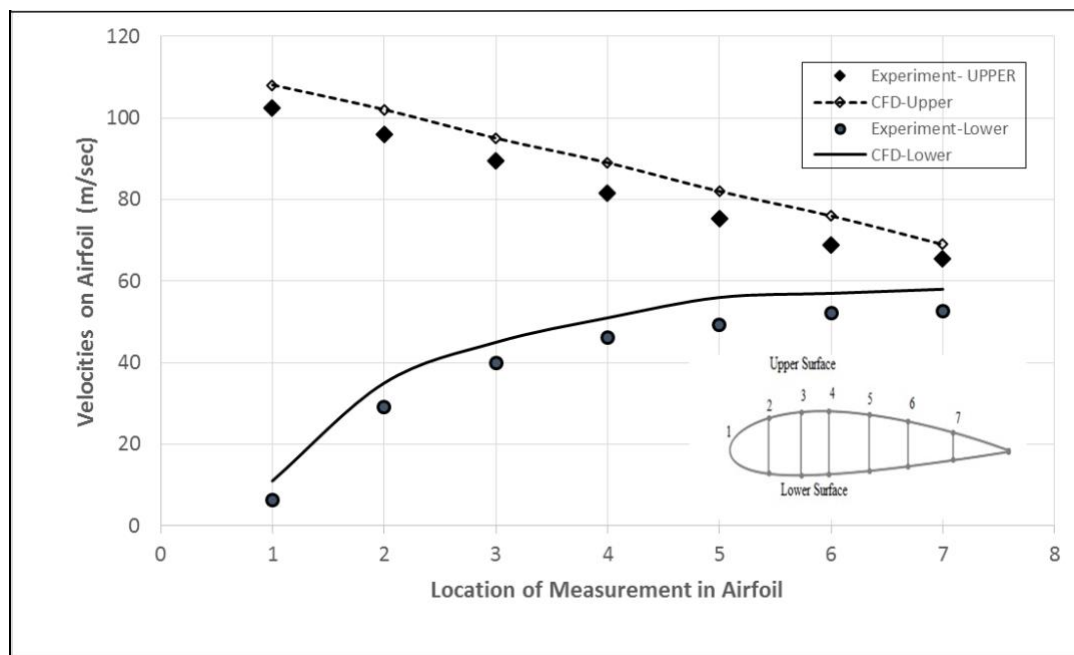


Figure 9A: CFD and Experimental Velocities on Airfoil Surfaces



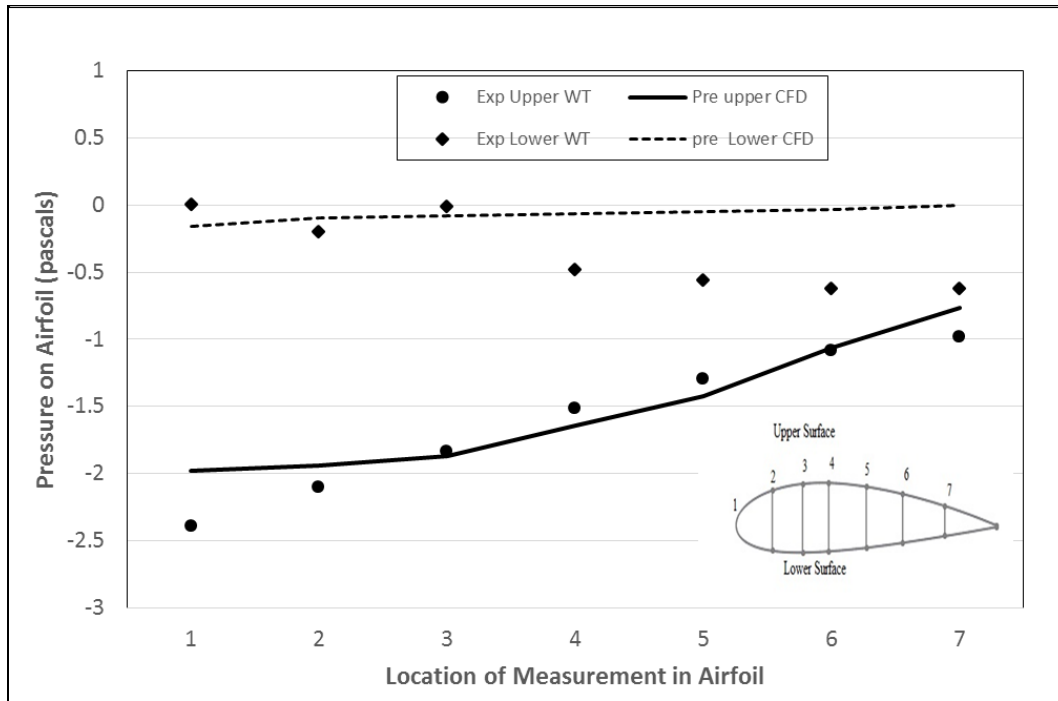


Figure 9B: CFD and Experimental Pressures on Airfoil Surfaces

Comparison velocities and pressure on airfoil surfaces are presented in Figures 9A and 9B. The experimental data showed reasonably good agreement with CFD results for both upper and lower surfaces of the airfoil. The CFD results are higher than experimental results as CFD analysis did not consider some of the losses in velocities and pressure during the experiment. Comparing CFD with EFD validated the CFD analysis results.

### Study 3: Multiphase Flow Through a 90-Degree Elbow

CFD analysis was performed for fluid flow through a 90-degree elbow to determine the location of maximum erosion. Erosion is a micro-mechanical, physical process that refers to the gradual removal of solid material from the wall due to repeated impact of the entrained particle in multiphase flow [7]. Solid particle erosion is a common cause of failure in many fluid equipment and systems; therefore, the study of erosion is important to avoid failure in the pipe walls. As erosion occurs in internal surfaces of the piping equipment, it is very difficult to detect until the failure occurs. A common method for determining the effects of erosion in a pipe is by analyzing the rate at which erosion will occur under a given condition using CFD software. In this analysis, the rate of erosion in a 90-degree elbow was determined by creating a model of elbow geometry as shown in Figure 10. The water-solid multiphase fluid was used in the simulation using Discrete Particle Model (DPM) available in FLUENT CFD software.



Figure 10: Isometric view of 90-degree elbow.

The inlet, outlet, and wall boundary conditions were defined; with the y-face signifying the inlet and the x-face signifying the outlet. The fluid was modeled as a viscous-laminar flow, evaluated using the K-omega equation which is often used to predict turbulence using two partial differential equations; known as the Reynolds-averaged Navier-Stokes equations [7]. The fluid was specified as water with an inlet velocity of 1m/s. Finally, the solver was set to utilize the pressure-based coupled algorithm which obtains a more robust and efficient single-phase implementation for steady-state flows [8].

Mesh sensitivity analysis was performed to determine optimum mesh size for accurate CFD results. The analysis was repeated three times and the average of the results was used. The analysis results for three different mesh sizes are presented in Table 1 showing significant differences in results that are dependent on mesh sizes. While the velocity remained similar, the maximum erosion rate and pressure drops (difference between pressures) varied significantly with different mesh configurations.

Table 1: CFD analysis results for different Mesh

Mesh	Number of Nodes	Number of Elements	Mesh Size (m)	Max. Erosion Rate (kg/m <sup>2</sup> ·s)	Max. Velocity (m/s)	Max. Pressure (Pa)	Min. Pressure (Pa)
Fine	12246	11498	0.0016	2.13E-25	1.3	345.5	-340.4
Medium	1520	1170	0.6	6.13E-25	1.34	469.6	-267.2
Coarse	280	190	125	8.83E-25	1.31	514.5	-117.9

There are different element types, and their usage depends on the geometry and analysis to be performed [5]. For this analysis, quadrilateral elements were used. Of the three elements sizes, the fine mesh appears to generate more accurate results due to the larger number of nodes and elements in the model. A node is a coordinate location where the degrees of freedom (DOFs) are defined. The DOFs for each point represent the possible movement of this point due to the loading or particle impacts at these various points within the structure. The DOFs also represent which forces and moments are transferred from one element to the next. The results of a finite element analysis are usually given at the nodes. It follows that more nodes will yield more refined results [5].

In the post-processing of the CFD results, the results were plotted of the RNS equations in the form of contour maps. The location of maximum erosion was found to be at 41.9 degrees from the inlet at the outer wall of the elbow. Cross-sectional pressure and velocity contours at different locations in the elbow are presented in Figure 11.

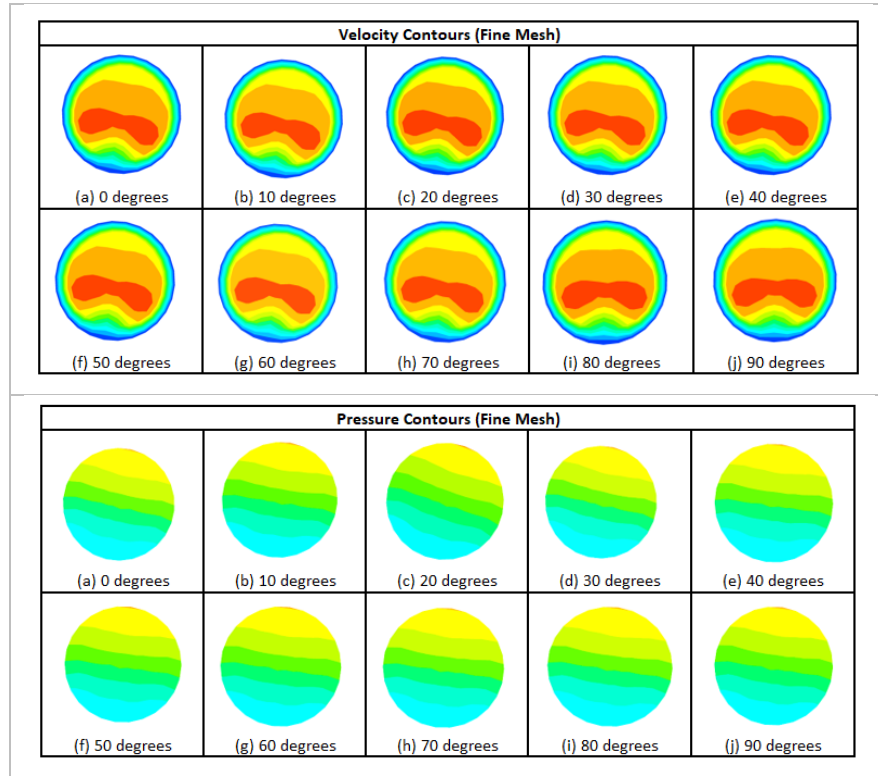


Figure 11: Velocity & Pressure contours through the 90-degree bend.

From Figure 12 it can be observed that the maximum velocity through the bend is at the inner wall of the bend. The maximum pressure was along the outer walls of the bend. The location of maximum erosion was found to be at approximately  $41.9^\circ$  from the inlet of the elbow.

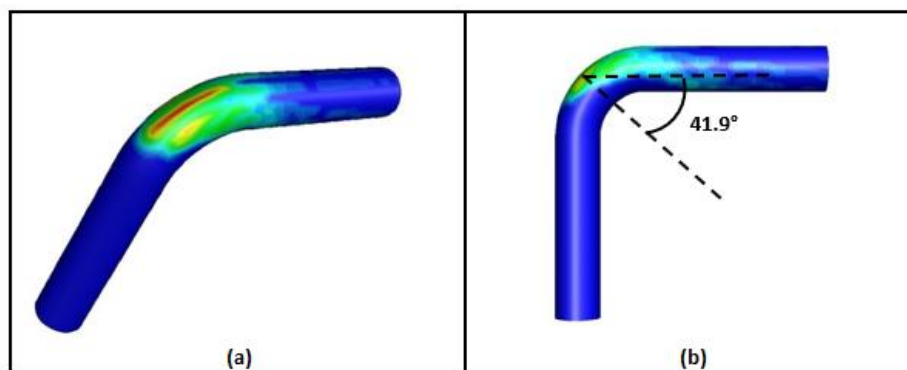


Figure 12: Location of maximum erosion in the elbow

To validate the CFD results, experimental investigations are currently being conducted using a particle image velocimetry (PIV) system. Comparison of CFD and PIV results will be presented in the future work.

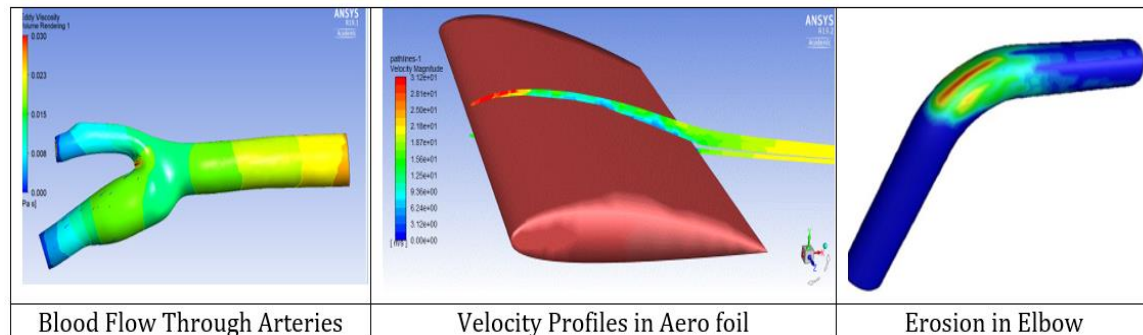


Figure 13: CFD Analysis Results

### Assessment of learning

A survey was conducted to assess students' level of understanding, interest, motivation and learning effectiveness of CFD and EFD. The assessment plan for the Fluid Mechanics course was developed considering the objectives and clear outline for student learning. The survey used 16 questions with a five-point Likert scale and the overall responses were analyzed to determine the effectiveness of the integration. The results of the survey are summarized below:

- There was a consensus in the support of our use of the software
- The support is readily available from the students to integrate CFD, EFD, and AFD with the general theoretical lectures and coursework in Fluid Mechanics.
- The results indicate that students will not only benefit their learning and improve their general overall understanding but can impact the student's potential job opportunity prospects.

Table 2: Integration of CFD and EFD in Fluid Mechanics

<b><u>Integration of Computational (CFD) and Experimental (EFD) in Fluid Mechanics Course</u></b>					
<b>Course Name and Number: EGR350: Fluid Mechanics</b>				<b>Date: _____</b>	
Questions	Strongly Agree	Agree	Disagree	Strongly Disagree	No Opinion
1) I have taken Fluid Mechanics course and developed basic understanding of different fluid flow characteristics	4	4			
2) The lecture and projects in CFD improved my knowledge, understanding and motivation towards fluid mechanics.	2	11	2		
3) The experimental project helped me develop in depth understanding of aerodynamics and flow over objects.	1	11	2	1	
4) The CFD and EFD projects provided supplemental knowledge to the fluid flow theory	2	11	1	1	
5) The CFD and EFD projects motivated me to become a better learner and prepared me for professional job as an engineer.	2	8	3		2
6) I can present results from EFD laboratories in written and graphical form.	2	8	2	1	2
7) I can conduct experiments in modern facilities such as pipe stands and wind tunnels	1	12	1		1
8) I can use EFD data for validation of CFD results.	1	10	1	1	2
9) I can analyze EFD results to gain increased understanding of fluid physics.	1	12		1	1
10) I can relate EFD results to fluid physics and classroom lectures.	2	7	2		4
11) I have a basic understanding of CFD methodology and procedures.		12		1	2
12) I can use FLUENT for solving laminar and turbulent pipe flow and inviscid and viscous airfoil flow.		13	2		
13) I can present results from CFD simulations in written and graphical form.	3	10	1	1	
14) I can evaluate iterative convergence criteria and analysis of solution residuals.	1	7	4		3
15) I can compare the computational results with the experimental data and analyze the differences.	1	13	1		
16) I can relate the CFD results to fluid physics presentations in written materials and the classroom lectured.	1	12	1		1
17) Recommendation for Improvement in the Integration Process					

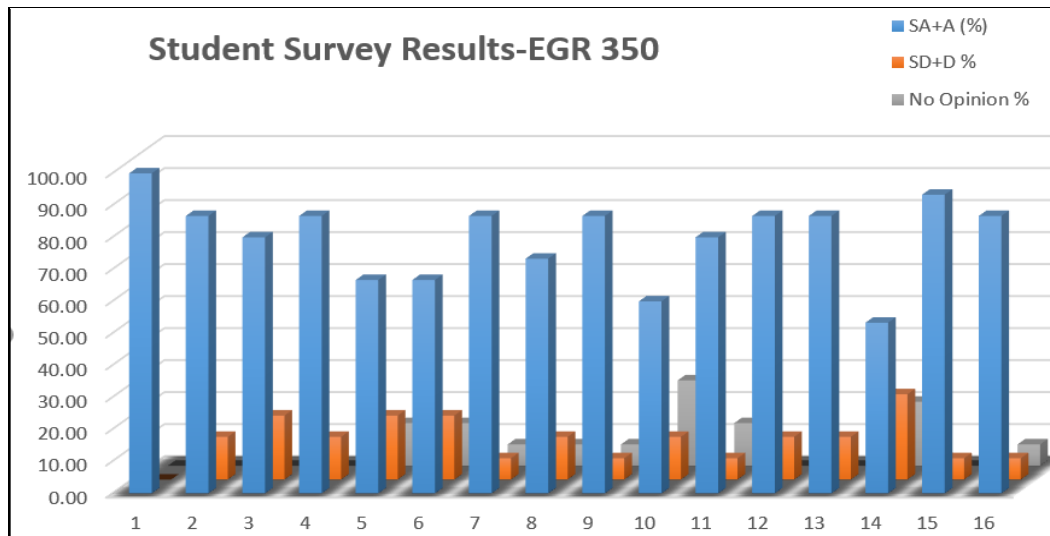


Figure 14: Student survey Results graph

- A control group survey of students in the same course who had not learned CFD was then conducted.
- The survey consisted of the same questions and the comparison of the two surveys shows much better understanding of Fluid Mechanics by the class that had taken CFD.

## SPSS Analysis

A composite score is calculated based on students' responses to their self-reported survey. For each question, a score ranging from 0-2 is created, with 2 given for agreeing, 1 for disagree, and 0 for no opinion. The scores from each of the 16 questions were added together and divided by the total possible score, creating a ratio which is then scaled by 10. The composite score is designed to represent the student's self-measurement of their ability in CFD and its usefulness in engineering courses.

The one-way analysis of variance (ANOVA) is used to determine whether the mean of a dependent variable is the same in two or more unrelated, independent groups. However, it is typically only used when you have three or more independent, unrelated groups, since an independent-samples t-test is more commonly used when you have just two groups. If you have two independent variables you can use a two-way ANOVA. Alternatively, if you have multiple dependent variables you can consider a one-way ANOVA.

The One-way ANOVA was conducted to compare the effect of taking a CFD course on the student's composite score. The analysis of variance showed that the effect of taking a CFD course on student's composite score was Significant,  $F(1, 27) = 46.16$ ,  $P = .00$ . Also, it can be further seen that students who have taken the CFD course show significantly higher scores than those who did not use the plot of the student's composite score.

Table 3: Result of one-way ANOVA

	Sum of Squares	df	Mean Square	F	Sig.
Between Groups	256.451	1	256.451	46.159	0
Within Groups	150.008	27	5.556		
Total	406.459	28			

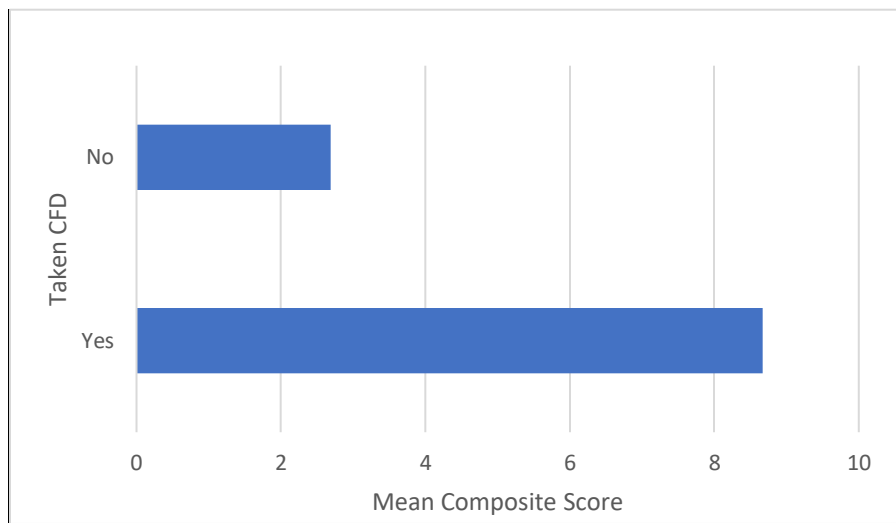


Figure 25: Mean plot of composite score between CFD course takers and non-takers

## Conclusion

CFD analysis performed on this small section of an aortic artery is only a small sampling of the powerful and wide applicability of CFD modeling in the field of medicine. As computer processing power has increased, the time necessary to simulate actual human systems has become more manageable and easier to do. Modeling blood-flow through vascular systems, like arteries and veins (which include moving boundary conditions), as well as modeling non-Newtonian fluids like blood, is now becoming possible.

Upon completion of the airfoil CFD simulation, it was apparent that the software verifies the data gathered from the previous wind tunnel experiment; just displayed from a different vantage point. Bernoulli's equation represents how velocity and pressure are inversely proportional to one another and the data acquired from both the CFD analysis and wind tunnel experiment confirm this. Simulating path lines of the fluid of study as well as displaying pressure and velocity gradients produces a visual representation of the hot spots found on the airfoil model.

EFD using the wind tunnel provided invaluable laboratory experience and better insight into fluid mechanics in the aerodynamics industry.

CFD analysis demonstrated the effects of various mesh sizes on the accuracy of results concerning fluid flow through a 90-degree elbow. It was found that smaller element sizes resulted in a finer mesh configuration which improved the accuracy of the analysis. The solution of the Reynolds-averaged Navier-Stokes equations allowed the Ansys Fluent software to provide a physical depiction of fluid flow in the form of contour maps. These contour maps show that as fluid flows through the bend, the maximum flow velocity occurs along the inner portion of the bend, while the maximum pressure occurred along the outer walls of the bend. This pressure causes erosion along the outer wall of the elbow, with the maximum erosion occurring at a location of approximately  $41.9^\circ$ .

CFD software allows for the analysis of different prototypes without the additional labor, production, and raw material cost associated with creating minuscule changes in new designs. Analytical Fluid Dynamics (AFD) is important in the classroom as students need to learn the concepts and science behind fluid dynamics, but the integration of CFD into those classes will better prepare those students for the real world as CFD is beginning to become more prevalent in the industry today. In addition to better preparation, students may also become more engaged in the class as CFD provides a visual for what they are learning. Finally, as the students become more familiar with CFD, it would be advantageous for them to also perform EFD to validate AFD and CFD results. Validation of simulations such as CFD gives the student confidence in the precision and accuracy of the program they use and will make them a well-rounded and valuable asset to many companies.



## References:

- [1] Britannica, The Editors of Encyclopaedia. "Artery." Encyclopædia Britannica. January 14, 2016. Accessed March 30, 2019. <https://www.britannica.com/science/artery>.
- [2] S. F. Sánchez, R. J. C. Luna, M. I. Carvajal, and E. Tolentino, "Pressure drop models' evaluation for two-phase flow in 90-degree horizontal elbows," *Ingeniería Mecánica Tecnología Y Desarrollo*, vol. 3, no. 4, pp. 115–122, 2010.
- [3] X. Chen, B.S. McLaury, S.A. Shirazi. "Application and experimental validation of a computational fluid dynamics (CFD)-based erosion prediction model in elbows and plugged tees". 1251-1272, 10.1016/j.compfluid.4004.02.003.
- [4] Mazumder, Quamrul H., Siwen Zhao, and Kawshik Ahmed. "Effect of Bend Radius on Magnitude and Location of Erosion in S-Bend." *Modelling and Simulation in Engineering*2015 (December 14, 2015): 1-7. doi:10.1155/2015/930497.
- [5] Liu, Yucheng, and Gary Glass. "Effects of Mesh Density on Finite Element Analysis." *SAE Technical Paper Series*, January 08, 2013. doi:10.4271/2013-01-1375.
- [6] OpenStax. "Blood Flow, Blood Pressure, and Resistance." *Anatomy and Physiology*. March 06, 2013. Accessed March 30, 2019. <https://opentextbc.ca/anatomyandphysiology/chapter/20-2-blood-flow-blood-pressure-and-resistance/>.
- [7] Safaei, M. R., O. Mahian, F. Garoosi, K. Hooman, A. Karimipour, S. N. Kazi, and S. Gharehkhani. "Investigation of Micro- and Nanosized Particle Erosion in a 90° Pipe Bend Using a Two-Phase Discrete Phase Model." *The Scientific World Journal*2014 (October 14, 2014): 1- doi:10.1155/2014/740578.
- [8] "25.9.1 Choosing the Pressure-Velocity Coupling Method." *Ansys Fluent*, September 20, 2006. Accessed March 15, 2019. <https://www.sharcnet.ca/Software/Fluent6/html/ug/node1021.html>.
- [9] John Aguirre. "STUDY OF 3-DIMENSIONAL CO-FLOW JET AIRPLANE AND HIGH-RISE BUILDING FLOW USING CFD SIMULATION". December 2008. University of Miami.
- [10] "Erosion in Steam and Condensate Piping." *TLV A Steam Specialist Company*. <https://www.tlv.com/global/US/steam-theory/piping-erosion.html>.