Purdue University
Purdue e-Pubs

ASEE IL-IN Section Conference

Introducing CFD Numerical Analysis in Fluid Dynamics to Junior Engineering Students

Khaled Zbeeb
Western Illinois University

Blair McDonald
Western Illinois University

Il-Seop Shin
Western Illinois University

Prathivadi Ravikumar
Western Illinois University

Introducing CFD Numerical Analysis in Fluid Dynamics to Junior Engineering Students

Khaled Zbeeb, Blair McDonald, Il-Seop Shin and Prathivadi Ravikumar

Western Illinois University, Moline Il 61265

Abstract

To enhance the students' analytical capability with fluid dynamics problems, Western Illinois University engineering faculty introduce ANSYS workbench during Fluid Dynamics, a junior level core-engineering course in many engineering programs. Traditionally, advanced analytical software is not introduced until the senior year or in graduate courses. However, since the methods of teaching engineering have evolved dramatically toward using advanced technological tools and software, the use of ANSYS workbench software in the junior year is now quite natural. Using advanced numerical software provides students with better understanding and visualization of a flow field. The current generation of students is accustomed to watching videos and animations to grasp a concept or an idea. The animations, contours and figures generated using a CFD numerical analysis program provide X university's engineering students with a greater understanding of flow behavior in all but the simplest dynamic fluid problems. As in most programs, physical laboratory experiments are conducted in the fluid dynamics class. Then the students model the experiments using CFD simulations. Consequently, both the experimental and numerical results are able to be compared and validated. The decision to use advanced CFD software in the fluid dynamics class has produced a positive impact on the students' overall knowledge of fluid mechanics. The students are excited to use state of the art analysis techniques and demonstrate greater enthusiasm in class.

Introduction

The study of fluid has been around for millennium, dating back to ancient Greece, but their understanding did not go beyond what they needed to know to run aqueducts and other waterworks [1]. Da Vinci further pursued the topic during the Renaissance observing waves and free jets. Even Newton studied fluids. The topic did not mature until people like Bernoulli and Euler investigated it and developed equations that were later named after them. The Euler equations were further modified by Claude Louis Marie Henry Navier and George Gabriel Stokes to create the Navier-Stokes equation [2]. These men laid the groundwork that would be the foundation of computational fluid dynamics.

Computational Fluid Dynamics (CFD) is a technology based on a fast and reliable computational methodology for solving complex fluid flow and heat transfer problems [3, 4]. CFD enables the product design team to reduce their risks of potential design failures, optimize their engineering design, and, can provide them with that illusive competitive advantage in the marketplace [5, 6]. That is required for constant success.

CFD Numerical simulations of fluid flow are applied in various applications. For example, CFD analysis enable

- architects to design comfortable and safe living environments
- designers of vehicles to improve the aerodynamic characteristics
- chemical engineers to maximize the yield from their equipment
- petroleum engineers to devise optimal oil recovery strategies
- surgeons to cure arterial diseases (computational hemodynamics)
- meteorologists to forecast the weather and warn of natural disasters
- safety experts to reduce health risks from radiation and other hazards
- military organizations to develop weapons and estimate the damage
- CFD practitioners to make big bucks by selling colorful pictures

Figures 1 and 2 show examples of CFD applications.

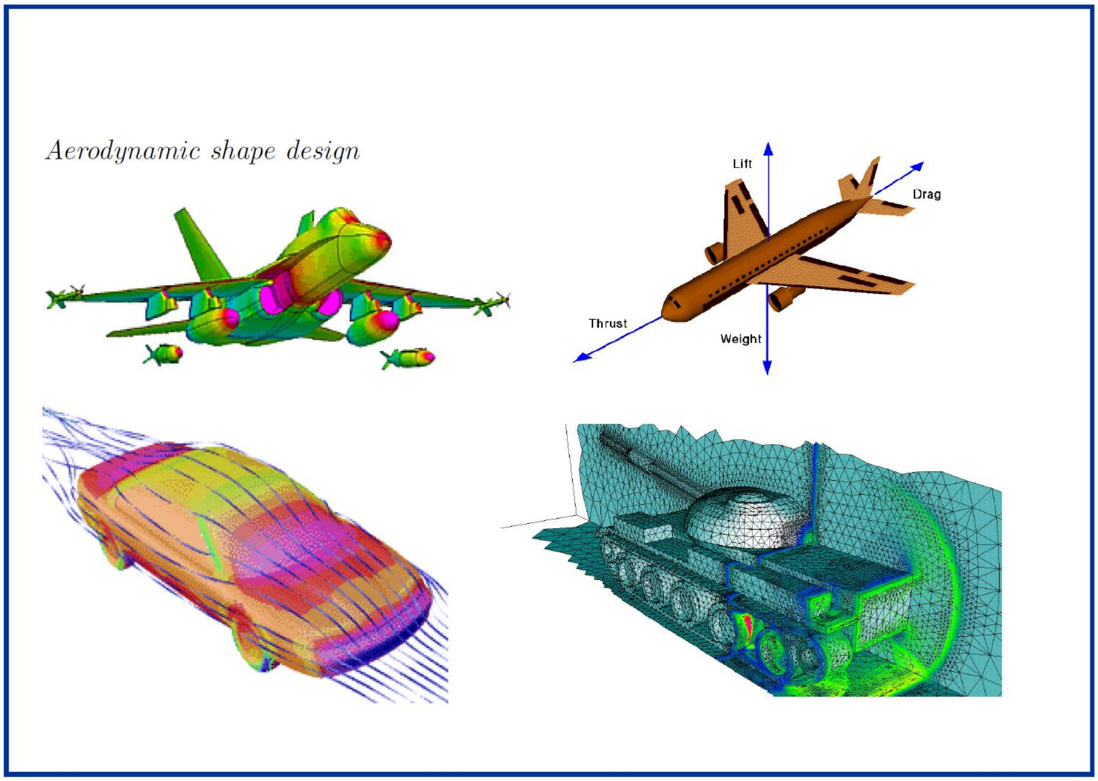


Figure 1: Examples of CFD applications [7]

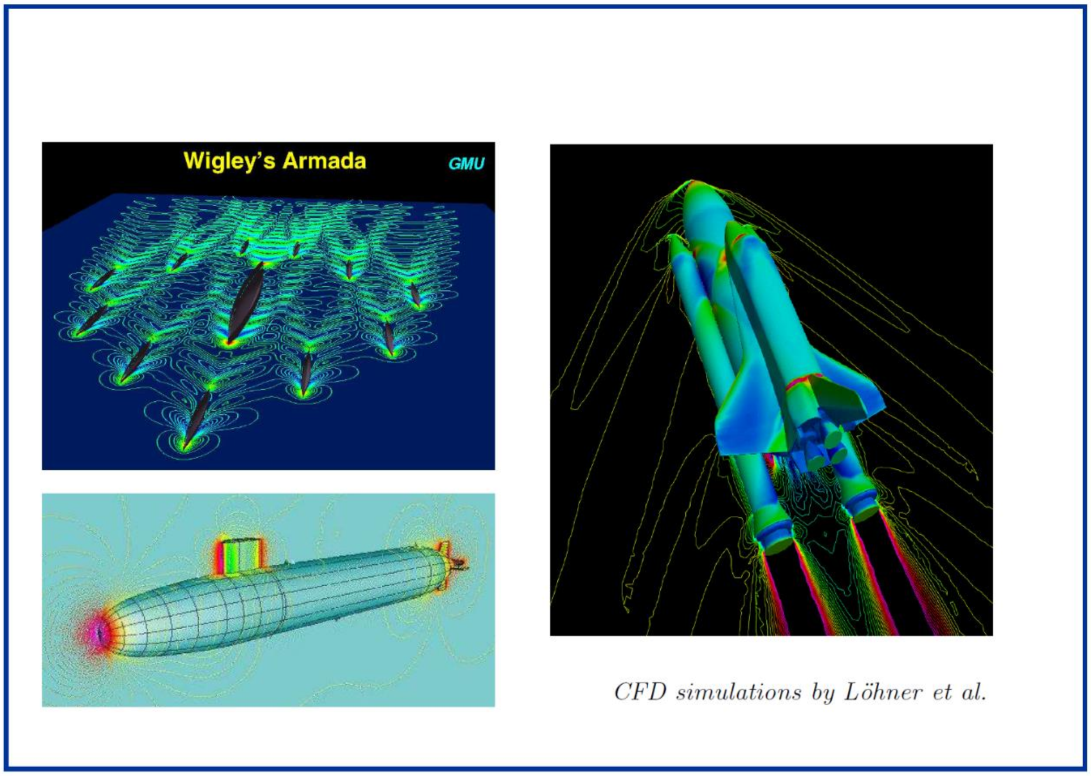


Figure 2: Examples of CFD applications [8]

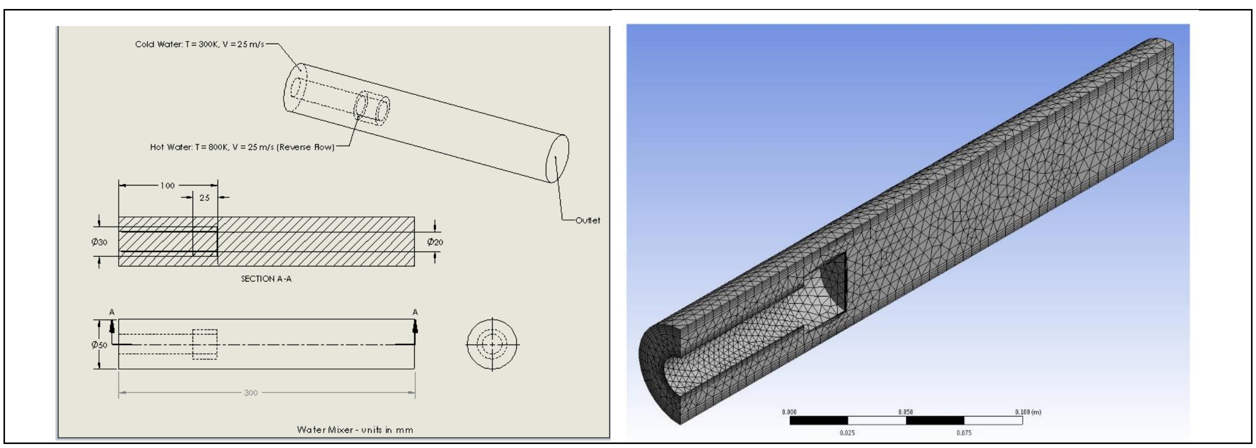
At Western Illinois University, CFD simulation techniques are usually introduced to engineering students in their senior year and CFD software (ANSYS) is taught in the Intermediate Thermo-Fluids course (ENGR 410) which is a senior level course. Besides covering the fundamentals of compressible flow, open channel flow, turbomachinery and chemical reactions, this course introduces the scientific principles and practical engineering applications of CFD. Although it provides an overview of some of the fundamental mathematical equations governing the fluid flow and heat transfer phenomena, its emphasis is to teach the theory behind the technology and to help the participants apply the knowledge gained into practical use of educational CFD codes, particularly ANSYS Fluent®. Most of the students who were taking ENGR 410 in their senior year, showed a lot of interest in learning CFD simulations using ANSYS, and have mentioned to the instructor that performing CFD analysis and flow visualization could have been useful to them in their Fluid Dynamics class (ENG 310). They also thought that CFD simulations have improved their understanding of the flow behavior and helped them grasp the fluid dynamics concepts. Based on the students' suggestions, it was decided to include CFD simulations in the Fluid Dynamics class, and ANSYS workbench was introduced to the ENGR 310 class in the spring of 2014. The course has been taught using CFD since then.. The CFD simulations were performed in class to show the students the simplicity and power of using numerical computation in analyzing flow fields, and let them experience the effectiveness of using the CFD software to solve the fluid dynamics equations (conservation of mass, momentum and energy equations) numerically.

CFD teaching Strategy

As it was mentioned earlier, the aim was to introduce basic CFD numerical analysis to the students to help them understand the flow behavior. The CFD demos that were done in class were mostly simple CFD examples like a water mixer, converging-diverging nozzles and basic combustion problems. There were no assigned CFD projects or homework problems for the students, except for honors students. Honors students were expected to work on a term project that involved CFD simulations.

The in-class CFD demos provide detailed explanations of how to set up, run and interpret the results of CFD models for different case studies. The demonstrations and examples cover all the necessary theoretical background for industrial applications of Computational Fluid Dynamics (See Figures 3 to 5). The students are taught to work through the following systematic instructions for each case study:

- Establish or import the model geometry using ANSYS design modeler
- Perform the model meshing and define boundary conditions (Inlet, Outlet, Symmetry and Wall)
- Set up the most appropriate CFD model (in terms of boundary conditions, material properties, solution control parameters, solution monitor, etc.) for the problem in hand
- Set up the most appropriate turbulence model for their particular applications
- Explain how to conduct both Steady state and Transient (time dependent) fluid flow simulations
- Explain how to solve for both isothermal and non-isothermal thermo-fluid applications, by including all the necessary modes of heat transfer i.e. conduction, convection and radiation, in their CFD model set up
- Explain how to solve for both Incompressible and Compressible fluid flow applications
- Initialize solution and perform numerical iterations
- Describe how and extract the required results and plots from the wealth of information available at the solution stage (Contours, Plots and Pathlines)



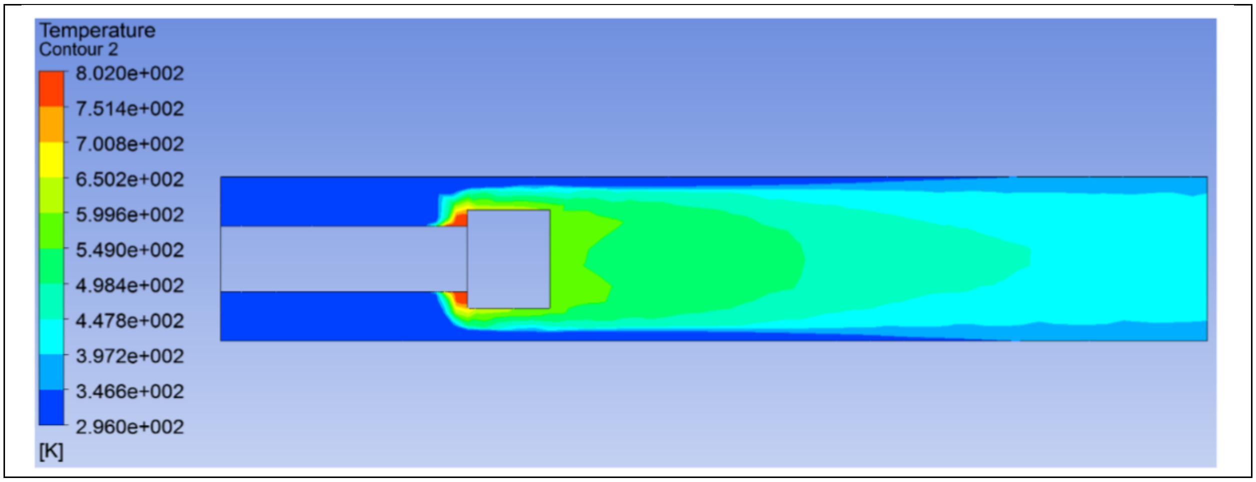


Figure 3: CFD Simulations (Water Mixer)

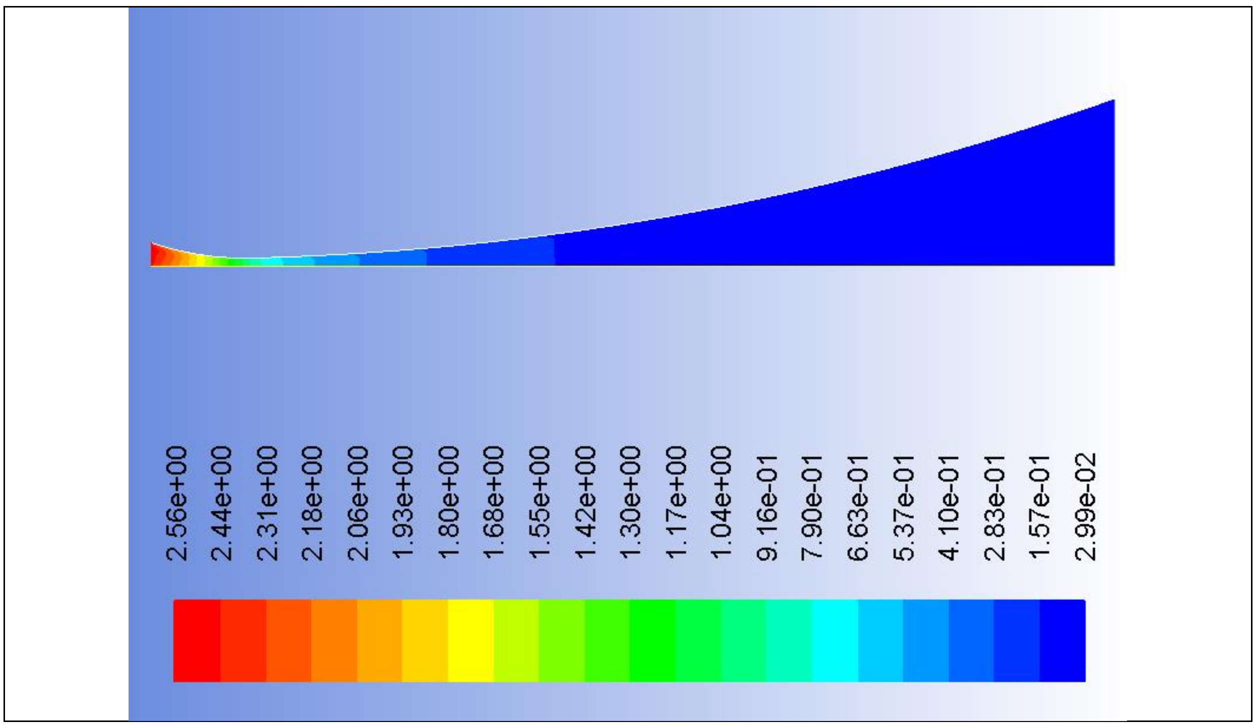


Figure 4: CFD Simulations (Converging Diverging Nozzle)

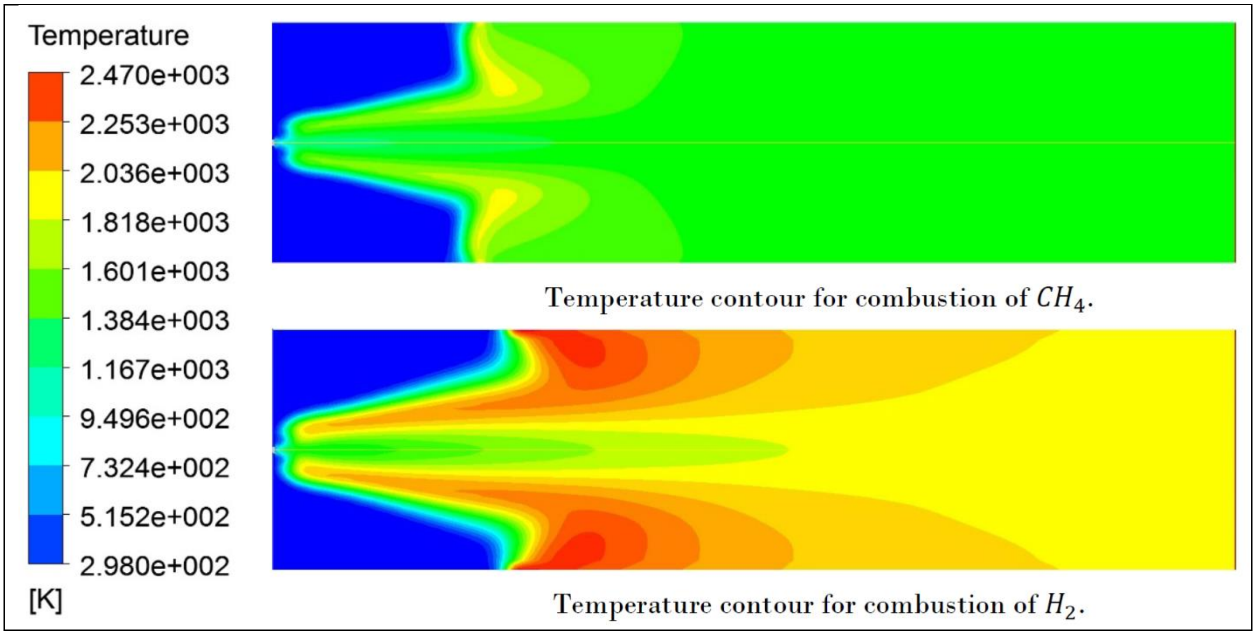


Figure 5: CFD Simulations (Combustion)

Impact of the early introduction of CFD analysis on the senior level courses

In the senior year, several courses that use ANSYS workbench simulations are offered at WIU school of engineering. One of these courses is ENGR 410 (Intermediate Thermo-Fluids). The use of the CFD numerical simulations is essential in this course, and many assigned projects deal with compressible flow, turbomachinery and combustion. Because of the early introduction of ANSYS in their Fluid Dynamics class (ENGR 310), the students feel more comfortable in using the software, and tend to be more creative by exploring many features of the CFD software. A second course that uses ANSYS workbench is Heat Transfer (ENGR 411). In this class, students are expected to run steady and transient heat transfer simulations using ANSYS workbench. Even though Heat transfer is a different subject than CFD, the ANSYS workbench knowledge that they gained from ENGR 310 helps the students navigate through the software easily and comfortably set up their heat transfer models using similar steps that are used in CFD simulations. Similar to ENGR 411, Finite Element Analysis (ENGR 481) is also another course that uses ANSYS workbench extensively. In this class, students are taught to run structural simulations using ANSYS software. The steps to setup an ANSYS structural model are also similar to the CFD and Heat transfer models, with minor changes in the setup section of the ANSYS workbench. In addition to formal course work, most senior design projects involve CFD, structural or Heat transfer analysis that can be performed using ANSYS workbench. Consequently, the CFD numerical analysis experience that students gain in ENGR 310 has enhanced their ability to use ANSYS software in many of the senior year courses.

Impact of the early introduction of CFD analysis on the local Engineering industry

The engineering program at WIU is offered in the Quad Cities campus, and it is positively driven by the local industry. Many local engineering companies (John Deere, Alcoa, Kone, Caterpillar, Siyver Steel, Cobham and many others) are always looking for interns. These companies employ many of our sophomore, junior and senior students. A great number of these local companies use ANSYS or other CFD software to run their CFD analysis, and the knowledge that WIU students gain from learning CFD numerical analysis during their junior year makes them very attractive to hire as interns. Consequently, the students have greater opportunities in getting full time positions at these companies after graduation. The advisory committee for the mechanical engineering porgram has constantly praised the efforts of the WIU Faculty in introducing CFD knowledge to WIU students at early stages in their education. The members of the mechanical engineering advisory committee are mostly directors and managers from local industry, and their feedback on evaluating WIU engineering program has affected program growth positively.

Assessment of students' performance

Although there is currently no direct measure to assess the impact of the early introduction of CFD analysis on the performance of WIU engineering students, the students, faculty members and local industry leaders have shown great enthusiasm and interest in introducing this knowledge at early stages in WIU engineering program. Most of the WIU senior engineering students' feedback was very positive, and they have constantly talked about their comfort in using ANSYS in their senior year courses and at the job sites. When teaching the senior level courses, the WIU engineering faculty members have also noticed a great boost in the students' knowledge of running ANSYS workbench for CFD, structural and Heat transfer analysis. Lastly, most of the local industry has repeatedly praised the WIU engineering faculty efforts in introducing and teaching CFD analysis to junior students. The greatest expression of their appreciation is by their hiring these students as interns and full time employees.

Future Study

An assessment system needs to be establish to assess the impact of the early introduction of CFD analysis on the performance of WIU engineering students. This assessment system should include surveys conducted by students, faculty members and local industry leaders. Moreover, a comparison study of students' performance in their senior level courses before and after the early introduction of CFD analysis would be beneficial. A great amount of data is needed to conduct such a study. The WIU engineering program is quite young, and the number of senior engineering students is currently too small to draw statistical data to establish a comparison study. There is great hope among the faculty and the administration at the WIU School of Engineering that the enrollment of both the General Engineering and Mechanical Engineering programs will grow in the next couple of years due to the introduction of the Mechanical Engineering program in 2017. Consequently, in the future more data will be available to assess the impact of the early introduction of CFD analysis on the performance of WIU engineering students.

Conclusion

Educators always desire to enhance the way they conduct classes and deliver messages to students. The aim of the WIU engineering program was essentially to optimize the teaching methods to fit the needs of this generation of students. This generation of students are driven by technology and they are accustomed to using computer software to learn and advance in their fields. The use of ANSYS workbench to conduct CFD simulations is a great method to help engineering students analyze and visualize flow fields. This technological tool was not available in the past. For years, Instructors have relied on experiments and the limited use of calculus to solve the flow field equations (continuity, momentum and conservation of energy). With advancements in technology and computing, the numerical computation of these equations is now easier than ever, and engineering students should take advantage of and use this technology at early stages in their education. The use of advanced CFD software in the fluid dynamics class has provided the students with a powerful learning tool to analyze and visualize flow fields to enhance their overall knowledge of fluid mechanics. The students are excited to use state of the art analysis techniques and demonstrate greater enthusiasm in class. Moreover, faculty members and local industry leaders have shown great interest in introducing this knowledge at early stages in the WIU engineering program.

References

1. CFD-Wiki http://www.cfd-online.com/Wiki/Main Page

2. C. Cuvelier, A. Segal and A. A. van Steenhoven, Finite Element Methods and Navier-Stokes Equations. Kluwer, 1986.

3. J.H. Ferziger and M. Peric, Computational Methods for Fluid Dynamics.Springer, 1996.

4. C. Hirsch, Numerical Computation of Internal and External Flows. Vol. II and I. John Wiley & Sons, Chichester, 1990.

5. P. Wesseling, Principles of Computational Fluid Dynamics. Springer, 2001.

6. S. Turek, Efficient Solvers for Incompressible Flow Problems: An Algorithmic and Computational Approach, LNCSE 6, Springer, 1999.

7. R. L¨ohner, Applied CFD Techniques: An Introduction Based on Finite Element Methods. John Wiley & Sons, 2001.

8. J. Donea and A. Huerta, Finite Element Methods for Flow Problems. JohnWiley & Sons, 2003.