CFD as a Visualization Tool in Undergraduate Fluid Mechanics

David Calamas and Gyunay Keten
Department of Mechanical Engineering, Georgia Southern University

Abstract

One of the most important discoveries in fluid mechanics was a result of the use of flow visualization. With the widespread availability of Computational Fluid Dynamics (CFD) software the ability to visualize flow behavior in a lecture setting has vastly improved. While undergraduate engineering students are often perfectly capable of solving problems out of a textbook they often miss out on the physical significance of the equations that they are using. It is critically important for students to be able to relate the results of an equation they have solved to physical flow behavior in the real world. A commercially available CFD software was used in an undergraduate fluid mechanics course to help students understand fluid transport phenomena. Quantitative and qualitative results from the following CFD simulation topics were used to increase student understanding of flow behavior: flow over a flat plate and flow past a sphere.

Keywords

Fluid Mechanics, CFD, Visualization, Learning Modules

Introduction

Undergraduate fluid mechanics contains numerous topics that require effective visualization to convey the physical significance of the various fluid transport processes taking place. As experimental visualization techniques may be unavailable, or impractical to use due to class sizes, it may be desirable to utilize Computational Fluid Dynamics (CFD) to illustrate flow behavior. It is not expected, however, that undergraduates will have experience with CFD. Even introductory CFD courses are often first offered at a graduate level. As such, the learning modules presented in this paper do not require a background in CFD. Students will be assigned pre-configured learning modules to help them better understand flow behavior. These learning modules have already been validated with theoretical and/or experimental solutions and have been verified for grid independence. The mesh in the learning modules has already been generated and the appropriate boundary conditions imposed. In addition, all post-processing (e.g. contour plots, vector plots, etc.) has been completed prior to student interaction. Interaction with the CFD software will be limited to visualization and changing of pre-defined input parameters to study various flow fields. Several learning modules will be utilized in undergraduate fluid mechanics throughout the curriculum. Two of the learning modules are shown for brevity. The CFD examples can be found in an introductory CFD tutorial textbook by Matsson[1] and accompany theoretical solutions found in an introductory fluid mechanics text by Cengel and Cimbala[2].

© American Society for Engineering Education, 2016

Student and Course Background

Fluid Mechanics is offered during the fifth semester of the undergraduate Mechanical Engineering program at Georgia Southern University. Thus, the course is composed primarily of first semester juniors. Engineering Mechanics I (statics), Calculus III, and Differential Equations are pre-requite courses. Students must pass pre-requisite courses with a grade of C or better. The textbook utilized for the course is *Fluid Mechanics: Fundamental and Applications* by Cengel and Cimbala[2]. McGraw-Hill Connect is used for electronic book access as well as for homework assignments.

Learning Module 1: Flow over a Flat Plate

The concept of a boundary layer is first introduced in undergraduate fluid mechanics. Boundary layers are typically introduced in textbook chapters which cover approximate solutions of the Navier-Stokes equation. A boundary layer is a thin region of flow near a solid surface where viscous forces and rotationality cannot be ignored. Students are traditionally introduced to the concept of a boundary layer by studying a uniform stream flowing parallel to a long flat plate. The boundary layer thickness, δ, is defined as the distance away from the solid surface at which the velocity component parallel to the wall is 99 % of the fluid velocity outside of the boundary layer. The higher the free-stream velocity, V, the thinner the boundary layer. For laminar flow, the boundary layer thickness is a function of the Reynolds number, Re. The Reynolds number is the ratio of inertia to viscous forces and characterizes the type of flow. External flow over a flat plate is considered to be laminar for Reynolds numbers less than 5×10^5. The local Reynolds number, Re_x, is dependent on position, x, velocity, V, and kinematic viscosity, ν and is defined as

$$Re_x = \frac{Vx}{\nu} \tag{1}$$

Once the Reynolds number is known the boundary layer thickness can be determined. The theoretical expression for the thickness of the boundary layer for laminar flow is given as

$$\delta = \frac{4.91x}{\sqrt{Re_x}} \tag{2}$$

While students can easily calculate the boundary layer thickness they often lack an appreciation of the significance and scale of a boundary layer. While the shape of a boundary layer profile can be obtained experimentally through the use of various flow visualization techniques students may not have access to appropriate experimental visualization facilities. Instead, CFD can be utilized to visual boundary layers.

An overview of the geometry used to computationally analyze boundary layer development over a 1m long flat plate can be seen in Fig. 1. In the computational model all fluid properties were assumed to be constant and evaluated at standard temperature and pressure (293 K and 101,325 Pa respectively).

© American Society for Engineering Education, 2016

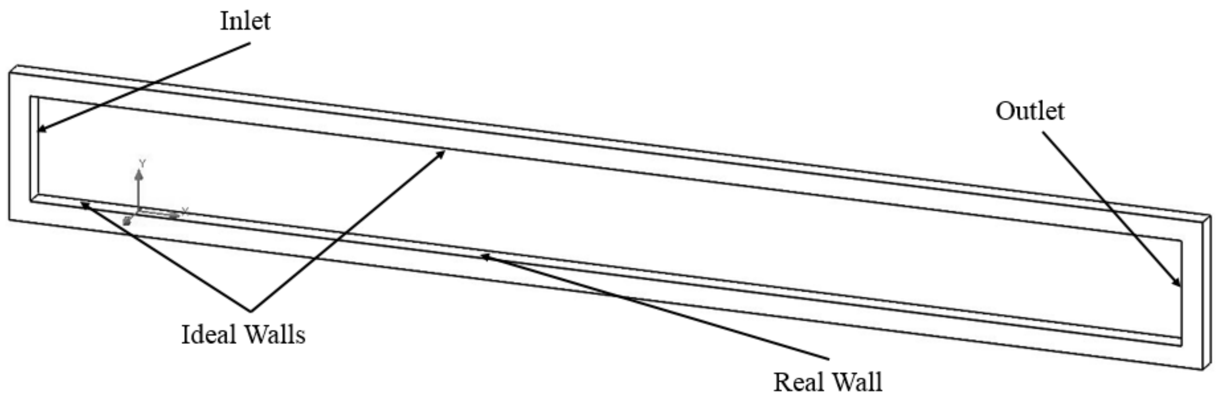

Figure 1. Computational Domain and Boundary Conditions for External Laminar Flow over a Flat Plate

The boundary conditions utilized in the computational model can be seen in Table 1.

Table 1. Inlet, Outlet and Wall Boundary Conditions

Boundary Condition	Value
Inlet	Uniform velocity of 5 m/s
Outlet	Static pressure of 101,325 Pa
Ideal wall	Frictionless
Real wall	No-slip condition

By opening the pre-configured simulation students are able to visualize various aspects of boundary layer flow. For example, students can visualize a contour plot of the velocity field or pressure field over a flat plate as seen in Figures 2 and 3 respectively. Students can visualize the thickness of the boundary layer as well as the stagnation point at the leading edge of the plate.

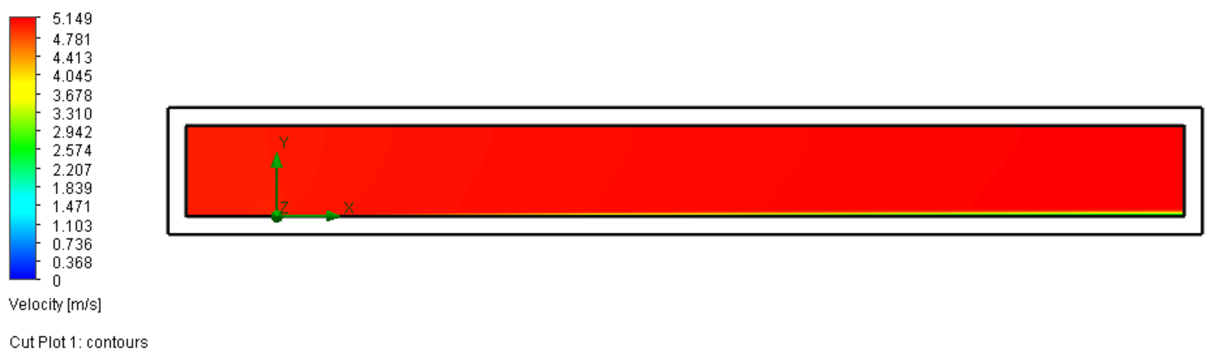

Figure 2. Velocity Field for External Laminar Flow over a Flat Plate

© American Society for Engineering Education, 2016

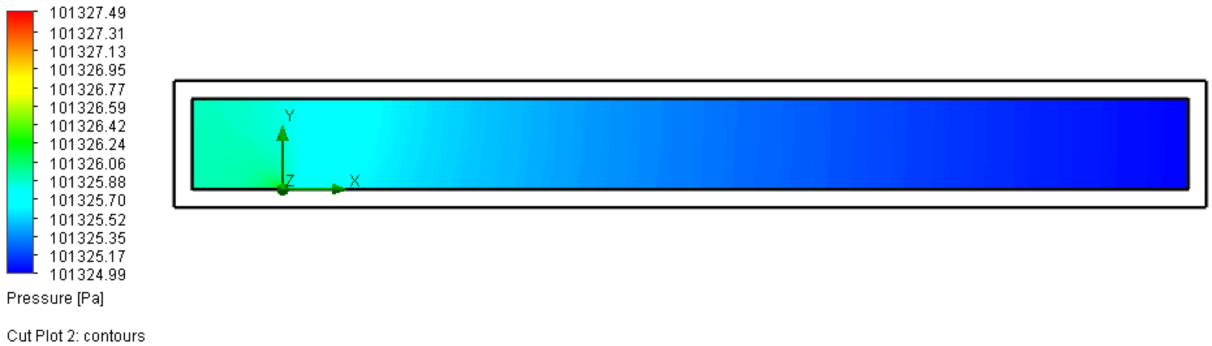

Figure 3. Pressure Field for External Laminar Flow over a Flat Plate

Students may probe the velocity field to determine the velocity, and thus Reynolds number, at various locations along the flat plate. Students may then compare the theoretical boundary layer thickness from the boundary layer approximations to the computational boundary layer thickness as shown in Table 2.

Table 2. Comparison of CFD and Theoretical Boundary Layer Thickness

x (m)	Re_x	δ_{CFD}(mm)	δ_{theory} (mm)	$\%_{difference}$
0.2	66,181	3.81	3.82	0.19
0.4	132,362	5.33	5.40	1.27
0.6	198,544	6.45	6.61	2.44
0.8	264,725	7.38	7.63	3.33

Students are then encouraged to alter the inlet velocity boundary condition (and thus Reynolds number) and re-run the simulation to observe the effect of inlet velocity on boundary layer thickness. It is hoped that through CFD visualization students will have a better understanding of the physical significance and scale of a boundary layer.

Boundary Layer Concept Survey

The objective of the learning modules presented are to increase student understanding of conceptual content and the physical significance of flow behavior in undergraduate fluid mechanics. As such, a concept survey will be given before and after each learning module to assess if the learning modules increased student understanding. The following Fundamentals of Engineering (FE) Exam style concept questions obtained from Cengel and Cimbala[2] will be assigned before and after the first learning module that demonstrates boundary layer development over a flat plate:

1. A very thin region of flow near a solid wall where viscous forces and rotationality cannot be ignored is called the
   a. inviscid region
   b. irrotational region

© American Society for Engineering Education, 2016

   c. boundary layer
   d. outer flow region
2. What fluid property is responsible for the development of the velocity boundary layer?
   a. Density
   b. Viscosity
   c. Surface tension
   d. Specific gravity
3. For a laminar boundary layer growing on a horizontal flat plate, the boundary layer thickness is not a function of
   a. velocity
   b. distance from the leading edge
   c. fluid viscosity
   d. gravitational acceleration
4. At a given location, if the Reynolds number were to increase, the boundary layer thickness would also increase.
   a. True
   b. False
5. As the fluid viscosity increases the boundary layer thickness increases.
   a. True
   b. False

Learning Module 2: Flow past a Sphere

Flow over spheres is often encountered in various sports (e.g. baseball, golf, soccer, tennis, etc.). Flow over spheres is typically introduced in textbook chapters which cover external flow, drag, and lift. As with flow over a flat plate the type of flow past a sphere is characterized by the Reynolds number. However, the characteristic length for flow past a sphere is the external diameter, D. Thus, the Reynolds number is calculated as

$$Re_D = \frac{VD}{\nu} \tag{3}$$

The critical Reynolds number for flow over a sphere is 2×10^5 below which the flow is laminar. The point at which the boundary layer detaches from the surface and forms a wake region behind a sphere is highly dependent on the Reynolds number. Thus the flow behavior past a sphere strongly affects both friction drag and pressure drag. The drag coefficient, C_D, for external flow past a sphere can be determined if the total drag is known. The drag coefficient is a function of the drag force, F_D, the upstream velocity, V, the density of the fluid, ρ, and the cross-sectional area of the sphere, A_c, and is defined as

$$C_D = \frac{F_D}{\frac{1}{2}\rho V^2 A_c} \tag{4}$$

For laminar flow the drag coefficient can also be calculated from a curve-fit of experimental data as

© American Society for Engineering Education, 2016

$$C_D = \frac{24}{Re} + \frac{6}{1+\sqrt{Re}} + 0.4 \quad (5)$$

As with the first learning module students will utilized a pre-configured CFD simulation to visualize the flow behavior past a sphere. An overview of the geometry used to computationally analyze flow past a 50 mm diameter sphere at a velocity of 0.003 m/s can be seen in Fig. 4. In the computational model all fluid properties were assumed to be constant and evaluated at standard temperature and pressure (293 K and 101,325 Pa respectively).

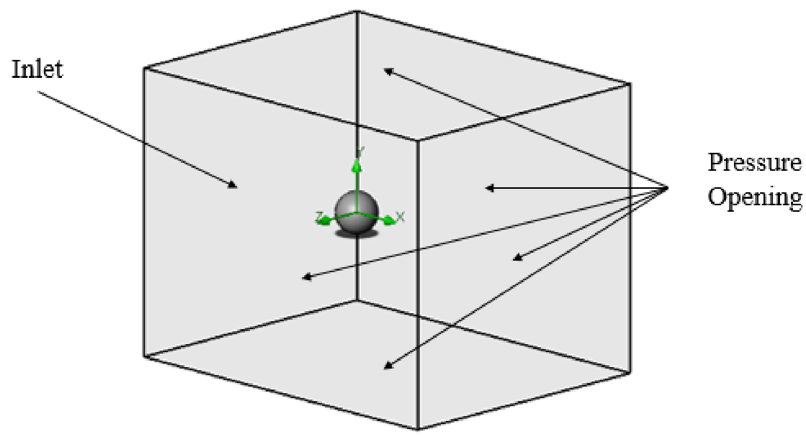

Figure 4. Computational Domain for External Flow past a Sphere

The boundary conditions utilized in the computational model can be seen in Table 3.

Table 3. Inlet, Boundary Conditions

Boundary Condition	Value
Inlet	Uniform velocity of 0.003 m/s
Pressure Openings	Static pressure of 101,325 Pa

By opening the pre-configured simulation students are able to visualize various aspects of flow past a sphere. For example, students can visualize velocity contours and observe the effect of flow separation on the velocity field as seen in Fig. 5. In addition, students can visualize the size of the wake region for various Reynolds numbers.

© American Society for Engineering Education, 2016

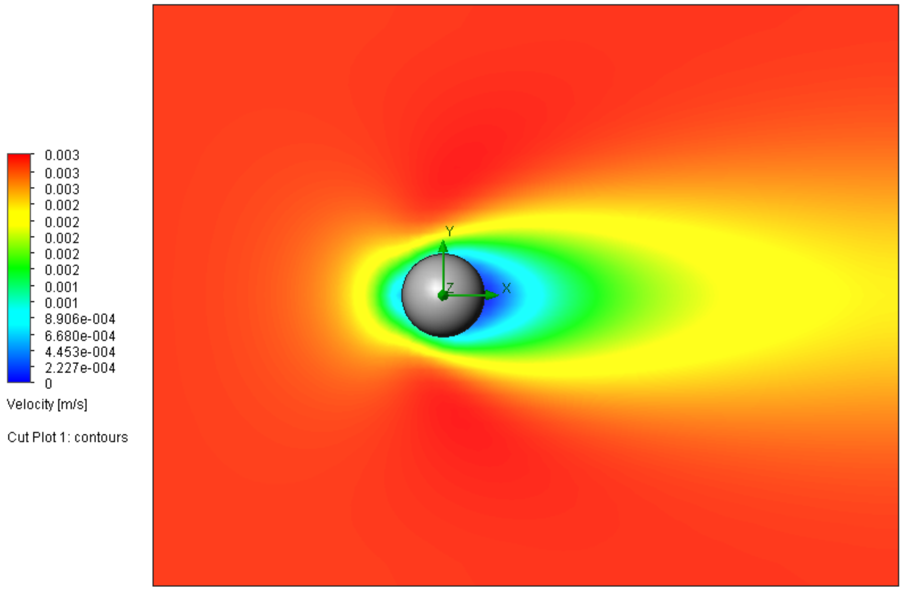

Figure 5. Velocity Field for External Laminar Flow past a Sphere

Students may compare the drag coefficient calculated by the CFD simulation to the experimental drag coefficient for laminar flow past a sphere as seen in Table 3.

Table 4. Comparison of CFD and Experimental Drag Coefficient

Re_D	$C_{D,CFD}$	$C_{D,experimental}$	$\%_{difference}$
9.93	4.11	4.26	3.54

Students are then encouraged to alter the inlet velocity boundary condition (and thus Reynolds number) and re-run the simulation to observe the effect of inlet velocity on the location of flow separation and resultant wake region. It is hoped that through CFD visualization students will have a better understanding of the physical significance of flow separation and its impact on drag.

External Flow past a Sphere Concept Survey

The following Fundamentals of Engineering (FE) Exam style concept questions obtained from Cengel and Cimbala[2] will be assigned before and after the second learning module that demonstrates external flow past a sphere:

1. The region of flow trailing the body where the effects of the body are felt is called the
    a. Separated region
    b. Rotational region
    c. Wake region
    d. Irrotational region

© American Society for Engineering Education, 2016

2. In flow over bluff bodies, friction drag is
    a. due to shear stress at the surface.
    b. due to the pressure differential between the front and back sides of the body because of the wake that is formed.
    c. much greater than pressure drag.
    d. dependent on the fluid density alone.
3. An important consequence of flow separation is
    a. a reduction in pressure drag due to the presence of a wake region.
    b. an increase in friction drag due to the presence of a wake region.
    c. the formation and shedding of circulating fluid structures, called vortices, in the wake region.
    d. the formation of a high-pressure region behind the body where recirculating and backflows occur.
4. As the Reynolds number increases flow separation is delayed.
    a. True
    b. False
5. As the Reynolds number increases the size of the wake region increase.
    a. True
    b. False

Conclusion

Computational Fluid Dynamics (CFD) can be utilized in undergraduate fluid mechanics to help students visualize various fluid transport processes. In this paper two learning modules were presented to help students visualize boundary layer development over a flat plate and flow separation over a sphere. The learning modules presented do not require prior CFD knowledge or simulation experience. The modules are intended to help students understand the relationship between what they are solving for in a problem and the physical significance of the result they obtained. Pre-learning module and post-learning module concept quizzes were developed and will be implemented to assess the effectiveness of the modules.

References

1 Matsson, J., An Introduction to SolidWorks Flow Simulation 2015, SDC Publications, Mission, KS, 2015.
2 Cengel, Y., and J. Cimbala, Fluid Mechanics: Fundamentals and Applications, 3rd Edition, McGraw-Hill, New York, NY, 2014.

David Calamas

David Calamas is an Assistant Professor in the Mechanical Engineering Department at Georgia Southern University. He received a BS in Mechanical Engineering from Clemson University in 2010. He received a MS and PhD in Mechanical Engineering from The University of Alabama in 2012 and 2013 respectively. He currently teaches heat transfer, fluid mechanics, and an energy science laboratory. His research interests are in the in the areas of biologically-inspired heat transfer, fluid and thermal transport phenomena, active and passive thermal management, solar energy, as well as engineering education.

Gyunay Keten

Gyunay Keten is a Graduate Research Assistant within the Department of Mechanical Engineering at Georgia Southern University. He is currently pursuing a Master of Science in Applied Engineering. His research is on passive and active thermal management of electronic devices using biologically-inspired fractal geometries.

© American Society for Engineering Education, 2016