# THE ROLE OF CASE STUDIES IN CFD EDUCATION

**Gordon D. Stubley[1] and Brad R. Hutchinson[2]**
*stubley@uwaterloo.ca, brad@asc.on.ca*
[1]*Professor, Mechanical Engineering, University of Waterloo*
[2]*Vice-President, Mechanical Engineering, AEA Technology Engineering Software*

**Abstract**

The user-friendliness of commercial CFD software has improved dramatically in the last ten years. The potential now exists for engineers with an undergraduate education to use the software for guiding design decisions. To realize this potential, educators must ensure that graduating students have a solid understanding of algorithm operation, modelling limitations, and the fluid mechanics of complex flows.

The present work shows how case studies can be used by instructors of undergraduate level CFD courses to develop and reinforce student understanding of complex flow physics. The case studies are presented as mysteries that incite the students' interest. With CFD visualization, the case studies and related concepts can be developed with relatively little lecture time.

## INTRODUCTION

Significant improvements in the robustness and usability of computational fluid dynamics (CFD) software is increasing the access of these analysis and design tools in the engineering community. Whereas ten years ago the typical users of CFD had graduate level training in both engineering fluid mechanics and CFD, the typical user today has undergraduate level training. A disparity is growing between the power of CFD software and the knowledge base of the typical user.

For example, one of the benefits of CFD analysis is that complete simulations of flow fields, along with all of the corresponding physical phenomena, are generated. The CFD tools provide a capability and power of analysis that is significantly beyond the understanding of an undergraduate student who has completed one or two courses in fluid mechanics. Undergraduate courses aimed at developing good practitioners of CFD in engineering design and analysis must address this issue. Unfortunately, it is a challenge to design a course curriculum that will develop a mature understanding of complex flow physics while also covering the range of concepts essential for successful use of CFD.

Textbooks provide one mechanism to meet curriculum design challenges. Traditional textbooks on CFD, such as those by Ferziger and Peric [1], Patankar [2], and Tannehill et al [3], put a significant emphasis on the development and implementation of CFD algorithms. Implicit in these textbooks, is the premise that the student will have a solid understanding of the fluid mechanics of complex flows. While these textbooks make excellent references for graduate level courses and for engineers developing CFD algorithms, they do not address the issues that are directly pertinent to CFD users.

Several authors have written CFD textbooks directed at undergraduate students and practicing engineers. The textbook by Abbott and Basco [4] was one of the first directed at undergraduate students. This textbook is written on the same model as Patankar's [2] classic text. With a strong focus on algorithms, it does little to improve a student's understanding of complex flows. The textbook by Shaw [5] focuses directly on the use of commercial software. This textbook describes and provides background on the steps a user completes in a CFD simulation. Again, this textbook does little to develop understanding of complex flows. The recent textbook by Versteeg and Malalasekera [6] is also written on the Patankar model, but provides practical advice and some background on flow physics. However, in an effort to keep the text brief and to cover a wide range of topics, the coverage of flow physics is insufficient for the needs of undergraduate students.

A premise of the present paper is that additional mechanisms are required to meet the challenges of designing effective CFD courses for undergraduates. In particular, the paper shows how CFD can be used to develop maturity in the analysis of complex flows through a series of case studies. Before presenting the goals and format of the case studies along with an example case, the paper reviews the context of CFD education in which these case studies have been developed.

## FLUIDS ENGINEERING CURRICULUM AT WATERLOO

At the University of Waterloo most undergraduate students complete two courses in engineering fluid mechanics before their fourth year. These introductory courses develop the students' abilities to carry out and understand one-dimensional analyses (Bernoulli's equation, conservation of mass, momentum, and angular momentum, and flow losses) of both incompressible and compressible flows, to use experimental correlations in analyses such as piping systems and pump operation, and to understand simple two dimensional boundary layer phenomena. The student will be aware of the difference between laminar and turbulent flow but will only have a primitive understanding of the physics of turbulent flow. This level of understanding does not include a strong mathematical perspective or maturity.

ME566, CFD for Engineering Design, is offered as a fourth year elective course. The goal of the course is to develop engineers who can use CFD effectively to guide engineering design decisions. The focus of the course is on incompressible laminar and turbulent single phase flows.

The course begins with a brief overview of CFD and the essential components of all CFD simulations: model formulation, discretization, equation solution, and analysis of results and uncertainties. In the second week of the course, the students are introduced to a commercial CFD package, CFX-TASCflow©, and are led through a tutorial exercise so that they become aware of the logistical operation of CFD software. CFX-TASCflow is a well-established package that operates stably on student level computing workstations and is well developed for mechanical engineering applications such as turbomachinery.

After the introductory portion of the course, the curriculum continues with lectures on three-dimensional fluid mechanics and the physics of turbulent flows, on the finite volume method and discretization techniques and errors, on iterative solution algorithms, their monitoring and operation, and on the modelling of boundary conditions, fluid properties, and turbulent stresses. The lectures conclude with a section on grid generation.

A set of pre-computed simulations are made available to the students and used for homework assignments throughout the term. Homework assignments involve analyzing a complex flow, assessing the influence of grid resolution and discretization method on the accuracy of CFD simulations, and the role of grid quality on solution algorithm performance. In all of these assignments, students use CFX-TASCflow to explore and analyze the pre-computed simulations.

The course culminates with a student project. Each student uses CFD to evaluate possible design changes to a piece of fluids engineering hardware such as a dump combustor. All facets of the course concepts, including complex flow physics, control of CFD modelling errors, and efficient use of resources, are required for successful completion of the project.

Approximately 20% of the students will take a graduate level course in CFD. The emphasis in the graduate level course is on understanding, implementing, and developing CFD algorithms. Also, approximately 30% of the students will use CFX-TASCflow/TurboGrid in ME563, Turbomachines, which is offered in the second term of fourth year.

## MYSTERIES OF FLUIDS ENGINEERING

One of the characteristics of the curriculum for ME566, as outlined above, is that a broad set of loosely related topics are covered. For example, during the sections on discretization methods and solution algorithms, it is common for students to stop progressing in their understanding of complex flow physics. One of the challenges in presenting the course is to continually reinforce the material on complex flow physics so that students have sufficient time for their understanding in this area to mature.

This challenge is met by using a case study approach for a small portion of the lecture time each week to discuss a *mystery of fluids engineering*. Each case study or mystery is presented as a question or apparent contradiction to provoke the students. While viewing a set of flow visualizations, the students discuss the flow field characteristics and analyze the dynamics of the flow. In this way, the students continually exercise and develop their knowledge of complex flows.

Besides a set of flow visualizations, the teaching materials for each case includes a discussion of engineering applications in which the flow phenomena may occur, references for further reading, additional flow visualizations for further exploration, and questions for student self-study and exploration.

The format of each mystery is best seen in the abbreviated example given in the next section. While not presented here, the complete teaching material for the example case study also contains full details of the methods used to obtain the simulated flow fields, analysis of the equations of motion, and bibliographies.

## FATE OF SINKING TEA LEAVES

### The Mystery

Take some tea leaves and add them to a tall clear glass of still water. After a short time, most of the leaves sink to the bottom which indicates that the soggy leaves are denser than water. When the water is stirred in the glass, the leaves move towards the glass centre line instead of moving outwards towards the glass walls. Why doesn't the *centrifuge effect* carry the tea leaves outwards?

### Model Flow 1

To understand the flow in the glass of water, a simpler flow is considered. This model flow is near a stationary flat plate lying in the $x-y$ plane with the fluid far above the plate rotating at a constant angular velocity about the $z$ axis. The simulated flow field is calculated from the semi-analytical solution presented by Schlichting [7]. The velocity vector plot on the $x-z$ meridional plane, Figure 1, shows

- the tangential velocity components due to the solid body rotation,
- the effect of the no-slip condition at the plate,
- the radial velocity component towards the axis of rotation, and
- the weak axial velocity component away from the plate.

The pressure field on the meridional plane, Figure 1, shows that the pressure only varies in the radial direction. The resulting radial pressure gradient balances the radial component of the centrifugal acceleration in the flow field far above the plate. As the plate is approached, the fluid speed decreases, thereby decreasing the radial centrifugal acceleration component. The imbalance between the radial pressure gradient and centrifugal acceleration near the plate creates a radial acceleration towards the axis of rotation as illustrated in Figure 2.

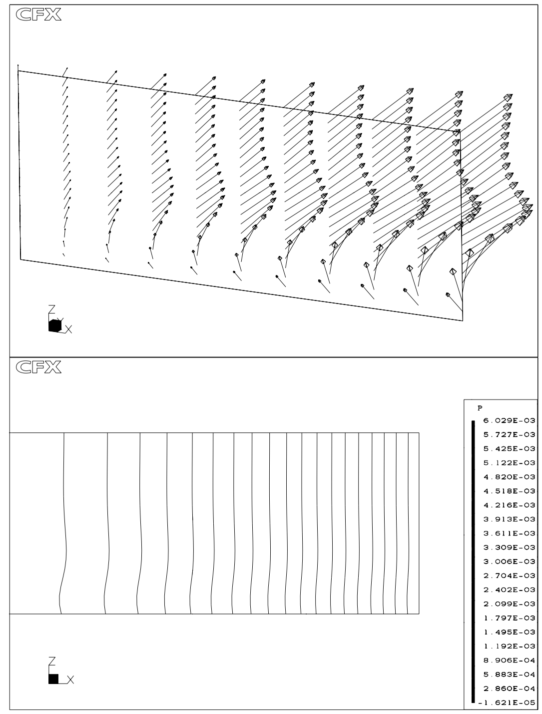


Figure 1: Features of rotating flow near a stationary flat plate plotted on the meridional plane. The upper portion shows the three-dimensional velocity field and the lower portion shows the pressure contours.

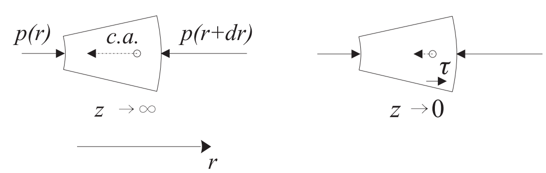


Figure 2: Illustration of the radial momentum balance in the far field and near plate regions. Relevant surface pressure forces, $p(r)$, centrifugal accelerations ($c.a.$), and net surface shear friction forces, $\tau$, are shown.

### Model Flow 2

Another simple flow, similar to that in the glass of stirred water, is the laminar flow in a cylindrical

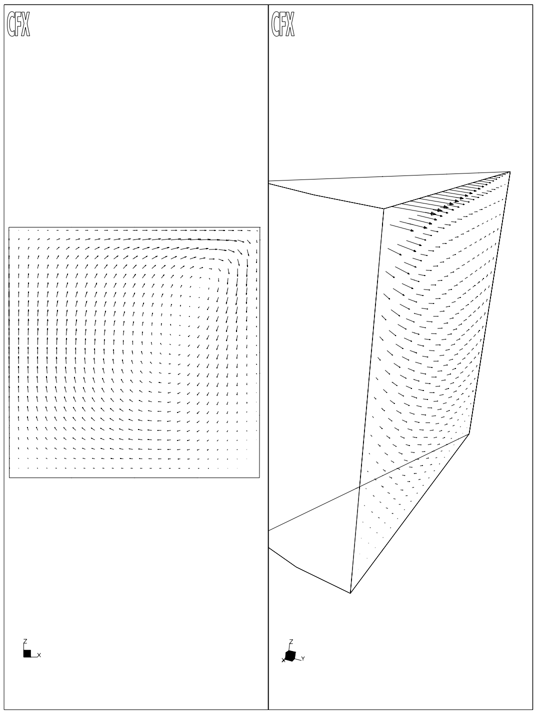


Figure 3: Velocity vectors on the $x-z$ meridional plane. The right view shows all three vector components and the left view shows the axial and radial components.

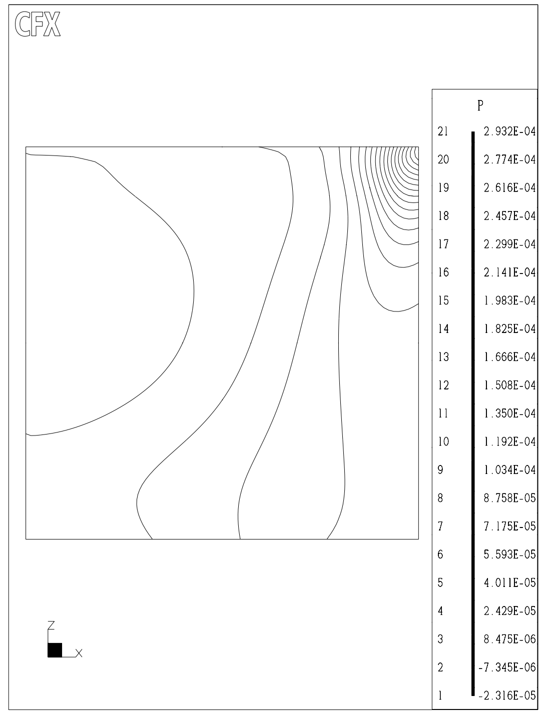


Figure 4: Pressure contours on the $x-z$ meridional plane for fluid in a container with a spinning lid.

container with a spinning lid. The Reynolds number, $Re \equiv \frac{\omega R^2}{\nu}$, is set to 400 where $\omega$, $R$, and $\nu$, are the lid rotation rate, container radius, and fluid kinematic viscosity, respectively. The simulated flow field is obtained with CFX-TASCflow. The three-dimensional and meridional velocity vectors, shown on the meridional plane in Figure 3, illustrate that the velocity speed drops rapidly with distance from the spinning lid, the speed also drops close to the container wall, and there is a single closed circulation cell in the meridional plane circulating about a point near the upper right corner of the plane. The pressure field, Figure 4, in the meridional plane shows that the pressure gradient at the lid is not sufficient to balance the centrifugal acceleration at the lid, thereby creating an outwards radial flow. There is also a radial pressure gradient at the bottom of the cylinder that drives a weak flow towards the centre line of the container, and an axial pressure gradient at the centre line that drives a weak vertical axial flow at the centre line.

**Further Explorations**

The flow field that moves the tea leaves towards the centre of the glass and those in the two model flows are examples of secondary flows. In each case, there is a main flow pattern of fluid rotating about an axis with a secondary flow pattern in the meridional plane. Similar secondary flows are common in rotating machinery. For example, Daily and Nece [8] present experimental measurements of the flow in a rotating disk cavity similar to the cavity behind the back face of centrifugal pump impellers. Other applications where similar secondary flows occur are the flow in duct and pipe bends, and the atmospheric flows around high and low pressure centers.

Neither of the two model flows exactly replicates the flow field in the glass of stirred water. Careful observation of the tea leaves indicates that they do not move all the way to the centre line but form a ridge a small distance away from the centre line. Studying the flow in the cylindrical container at higher Reynolds numbers shows that as the Reynolds number increases an instability forms on the center line and eventually multiple cells form in the meridional plane. When the flow becomes turbulent, there are multiple cells in the meridional plane. A stagnation point occurs at the point where the two counter-rotating cells meet on the bottom surface. It is at this point where the ridge of tea leaves forms.

Escudier [9,10] gives a thorough explanation of secondary flow fields in rotating flows along with five examples in which vortex and secondary flow motion play a significant role in fluid machinery. The two textbooks by Lugt [11, 12] give a complete presentation of the physics of rotating flows.

## SUMMARY

The case study outlined in the previous section can be used to reinforce many important concepts and skills:

- use of flow visualization to determine the kinematics and dynamics of a flow field,
- three-dimensional velocity and pressure fields,
- the three-dimensional action of pressure to establish a mass conserving velocity field through the three components of the pressure gradient in the momentum equations,
- the role of the no-slip condition in creating regions of viscous shear action,
- the variation of flow properties with increasing Reynolds number, and
- flow instabilities.

These concepts are significantly more complex than the concepts introduced in introductory fluid engineering courses. However, they are concepts which are applicable to a wide range of engineering flows and which must be understood by practitioners of CFD.

Other examples that provide interesting and stimulating fluid flow mysteries include:

- the creation of a weak oblique shock wave on a flat plate that is parallel to a supersonic flow,
- the mechanism responsible for the formation of wing tip vortices,
- the secondary flow created in rectangular ducts by turbulent stress anisotropy, and
- the role of streamline curvature in augmenting and suppressing flow instabilities.

One of the issues that arises in the design of undergraduate CFD courses is the role that commercial software should play. The concern is often raised that commercial software is so complex and finely tuned that students become isolated from the real physics of fluid flows and the errors inherent in CFD simulation. In other words, the software becomes a black box serving no long-term educational purpose.

The present paper attempts to show that this need not be true. Indeed, commercial software has been used to help develop essential concepts. These concepts are universal in fluids engineering and will arise with the use of any CFD package. One of the unique features of the present project is the degree of cooperation between the University of Waterloo and AEA Technology: Engineering Software Limited to develop resources for the teaching of CFD at the undergraduate level. AEA is making completed case studies available on its world-wide web page, http://www.software.aeat.com/cfx/.

Given the positive reception of the *mysteries* from students, future work will consider the use of case studies for other purposes. For example, it is easy to imagine a set of case studies that would illustrate good practices of CFD, such as appropriate placement of boundary conditions, grid resolution, fluid property models, turbulence models, and geometry modelling. Each case study could provide simulations obtained with good practice and poor practice along with comparisons to experimental results.

As these and other resources for teaching CFD at the undergraduate level become available, the capabilities of students can be increased to realize the potential of CFD to guide design decisions.

### Acknowledgements

Discussions with Dr. P.J. Zwart, and Professors F-S Lien, G.D. Raithby, and G.E. Schneider have improved the quality of the present work. The feedback and interest of the many ME565 students at the University of Waterloo is also gratefully appreciated.

## REFERENCES

[1] Ferziger, J.H. and Peric, M. 1996, *Computational Methods for Fluid Dynamics*, Springer-Verlag, Berlin, Germany.

[2] Patankar, S.V., 1980, *Numerical Heat Transfer and Fluid Flow*, Hemisphere Publishing Corporation, New York, USA.

[3] Tannehill, J.C, Anderson, D.A., and Pletcher, R.H., 1984, *Computational Fluid Mechanics and Heat Transfer, Second Edition*, Taylor-Francis, Washington DC, USA.

[4] Abbott, M.A. and Basco, D.R., 1989, *Computational Fluid Dynamics*, Longman Scientific and Technical, Essex, England.

[5] Shaw, C.T., 1992, *Using Computational Fluid Dynamics*, Prentice Hall, Hertfordshire, England.[1].

[6]Versteeg, H.K. and Malalaskera, W., 1995, *An Introduction to Computational Fluid Dynamics: The Finite Volume Method*, Addison Wesley Longman Ltd, Essex, England.

[7] Schlichting, H., 1979, *Boundary-Layer Theory, 7th. Edition*, pp. 225-230, McGraw-Hill, New York, USA.

[8] Daily, J.W. and Nece, R.E., 1960, Chamber Dimension Effects on Induced Flow and Frictional Resistance of Enclosed Disks,*ASME J. of Basic Engineering*, Vol. 82, pp. 217-232.

[9] Escudier, M., 1987, Confined Vortices in Flow Machinery, *Annual Review of Fluid Mechanics*, Vol. 19, pp. 27-52.

[10] Escudier, M., 1988, Vortex Breakdown: Observations and Explanations, *Progress in Aerospace Sciences*, pp. 189-229.

[11] Lugt, H.J., 1983, *Vortex Flows in Nature and Technology*, John Wiley and Sons, New York.

[12]Lugt, H.J., 1996, *Introduction to Vortex Theory*, Vortex Flow Press, Potomac, Maryland.

[1]This textbook is out of print but is available on the Internet at http://www.eng.warwick.ac.uk/staff/cts/cfdbook/