AC 2007-1560: USE OF COMPUTATIONAL FLUID DYNAMICS (CFD) IN TEACHING FLUID MECHANICS

Cuneyt Sert, Middle East Technical University

Cuneyt Sert received his B.S. and M.S. degrees from the Mechanical Engineering Department of Middle East Technical University (METU), Ankara, Turkey and his Ph.D. degree from Texas A&M University. He is currently working as an Asst. Prof. at METU. His current research interests include numerical simulation of thermofluidic transport problems and development of active/visual software for the use of engineering education.

Gunes Nakiboglu, ROKETSAN Missiles Industries Inc.

Gunes Nakiboglu received his B.S. degree from the Mechanical Engineering Department of Middle East Technical University (METU), Ankara, Turkey. He is currently involved with the Virtual Flow Lab project as a masters student in the same department. He is also working full time as a member of the Propulsion System Design Department of ROKETSAN Missiles Industries Inc., Ankara, Turkey.

Use of Computational Fluid Dynamics (CFD) in Teaching Fluid Mechanics

Abstract

Computational Fluid Dynamics (CFD) is a tool that allows the solution of fluid flow problems numerically by the use of computers. Its development is derived mainly by i) its use in the academia and research institutions for the inevitable need for understanding complicated flow phenomena where experimental and theoretical approaches either are not possible or do not provide enough insight, ii) its use in the industry via commercial software for an economical speed up of the design process. The use of CFD in engineering education is mostly limited to graduate level courses where the mathematical background necessary to write CFD programs is taught. Although recent undergraduate level fluid mechanics books involve CFD related chapters, references indicating the use of CFD as an undergraduate level teaching aid are limited.

This paper is about the possible use of CFD in teaching undergraduate level fluid mechanics. In the first part, the topics of a typical fluid mechanics course that may be supported with CFD are investigated. In the second part the tools necessary and suitable for the efficient use of CFD in teaching fluid mechanics are examined and a CFD software called Virtual Flow Lab developed by the authors is introduced and its capabilities and potential use for educational purposes are discussed.

Introduction

In many engineering departments students take their first fluid mechanics course in their second or third year. Although for some of the departments a single semester course is enough, usually undergraduate level fluid mechanics is taught as a two semester course. Typical outline of a two semester fluid mechanics course is given below

1. Introduction to Fluid Mechanics and Fluid Properties
2. Fluid Statics
3. Integral Analysis of Fluid Motion (conservation of mass, momentum and energy)
4. Bernoulli Equation
5. Fluid Kinematics
6. Differential Analysis of Fluid Motion
7. Similitude and Dimensional Analysis
8. Viscous Flows in Pipes and Channels
9. Flow over Immersed Bodies
10. Introduction to Compressible Flow
11. Introduction to Turbomachines

Due to the inherent complexity of fluid motion, fluid flow problems require a different viewpoint compared to solid mechanics problems. Understanding the topics like continuum assumption and its validity, proper comprehension of the field concept such as the velocity or the pressure field, making the switch from the classical Lagrangian approach, which is taught in earlier statics and dynamics courses, to the Eulerian approach, establishing the link between these two different view points, mathematical and physical understanding of the convective derivatives are some of the challenges that the students face with when they begin studying fluid mechanics.

Other than the above mentioned mathematical modeling related difficulties, students also get confused due to the simple fact that it is hard to observe and comprehend the behavior of fluids in everyday life. For example students often do not question the way an I-beam deflects under the action of some bending forces, but they can easily get confused in a simple pipe flow problem, where the existence of viscous forces causes a pressure drop but not a velocity drop. Or they comfortably take apart a complicated solid structure into its simple elements by drawing free-body-diagrams with proper reaction forces and moments, but it is not that easy for them to work with an imaginary control volume that is open to mass, momentum and energy transfer. Or for example it is difficult to mentally visualize the way the properties, such as viscosity, of a gas changes while it is being heated. In addition students get quite puzzled when they learn that almost all practically important fluid flow applications involve turbulence, which is considered to be one of today's most challenging physical phenomena.

Another major difficulty in learning fluid mechanics is the necessity of proper simplification of a given problem. Fluid flow problems of engineering importance can be so complex that one almost always needs to make a number of simplifying assumptions in order to be able to approach it by analytical means. For example the analytical solution of conservation of linear momentum, in other words the Navier-Stokes equations, is only possible for a few very simple problems. Neglecting the viscous affects, these equations reduce to the Euler equations, which are still quite difficult to solve. Another simplification comes when we consider the Euler equations along a streamline, which leads to the Bernoulli equation. Other than these one for example might need to consider if the compressibility of the fluid or the unsteadiness of the flow is of importance or not. It is not easy to get comfortable with the use of these different levels of simplifications.

Visualizing the fluids in action is a very informative tool that helps overcome the above mentioned difficulties to a certain degree. Carefully designed educational experimental studies are important in this respect. Many of the recently published fluid mechanics textbooks come with discs that include movies of many interesting fluid flow phenomena[1]. It is also possible to freely download excellent series of educational fluid mechanics films, such as the ones prepared by NCFMF[2] and IIHR[3], from the internet. Today another alternative is the use of CFD simulations as an educational tool. CFD enables us to solve fluid flow problems numerically by the use of computers. One simple advantage of this is its power in attracting the attention of today's computer oriented students[4]. But the actual benefit is that students feel comfortable when they see that the governing equations, which are known to be impossible to solve analytically in most cases, are actually solved with an acceptable engineering accuracy. The importance of such a numerical study is quite different than the importance of performing experiments. Experimentation is very valuable in understanding the underlying physics of a certain problem. This is necessary in establishing and validating a mathematical model. But then we naturally feel the need to solve that mathematical model. If we can not do that, our model is not very useful. If, for example, we can not solve the Navier-Stokes equations by any means, then being able to derive these equations is not very valuable. Today CFD is the only possible way to test the validity of our mathematical models for almost any type of flow problems. In this paper the use of CFD in undergraduate fluid mechanics education will be considered.

Sample CFD Simulation Ideas to be Used in an Undergraduate Fluid Mechanics Course

The outline of a typical undergraduate level fluid mechanics course was mentioned in the previous part. In this part the level of support that CFD simulations might provide in understanding the topics of this outline will be discussed.

Fluid statics is easy to learn since it involves no fluid motion. CFD simulations can still be used to explain the pressure distribution in a static fluid. Students can perform a number of numerical experiments with different shaped cups or tubes to see how the pressure increases linearly in a direction opposite to the gravitational acceleration independent of the shape of the container.

In integral analysis of fluid motion, students can check the mass conservation inside a box with multiple inlets and exits. They can be asked to use the numerical data of a CFD simulation of external flow over a body to calculate the drag and lift forces acting on the body. Comparing the forces calculated with a standard integral approach that uses the numerically obtained velocity profiles against the forces directly calculated by the differential approach of a CFD simulation could be quite instructive.

The Bernoulli equation is one of the most useful but also the most misused equations of fluid mechanics. It can be demonstrated by the simulation of fluid leaving a tank through a number of openings at different elevations. The concepts of static, dynamic and total pressure, which are difficult to grasp, can be explained by examining the result of a converging diverging duct simulation at different locations. CFD simulations are excellent ways to discuss the use of Bernoulli equation in flowrate measuring devices, such as an orifice meter or a venturi meter.

While discussing fluid kinematics, movies of rotational and irrotational flow simulations can be used. Results of the simulation of a developing flow that enters a channel uniformly can be used to demonstrate the effect of viscosity and shear forces in the creation of rotational motion inside the boundary layers. Deformation of a stragiht line or square shaped fluid element can be visualized in different flow fields to understand the type of deformation that they go through.

About the differential analysis of fluid motion, inviscid and viscous simulations inside a number of different geometries can be considered. Analytical solutions of Couette, Poiseulle and Hagen-Poiseulle flows can be compared with the numerical ones. Possible sources of differences between numerical and analytical results can be discussed along with the limitations of numerical simulations.

CFD simulation of a carefully designed piping system with different sized pipes and a number of bends, expansions, contractions, etc. will be very valuable in understanding the major and minor frictional losses and related pressure drops, as well as the application of extended Bernoulli equation for the calculation of these quantities.

Lectures about external flows over immersed bodies can be supported by a simple boundary layer growth demonstration over a flat plate and comparing the results with the analytical ones. It is also possible to visualize the solution of flow fields around streamlined and blunt bodies and discuss the affect of streamlining a body through the inspection of numerically calculated shear and pressure drag forces. The difficulty of the proper solution of separating flows can be demonstrated. A discussion about the D'alambert's paradox can be made by comparing inviscid and viscous flow simulations.

CFD can also be used to enhance the understanding of compressible flow lectures. Simulations of a number of flows inside a converging-diverging de Laval nozzle can be used to see different flow regimes, formation of shocks and choking.

Of course the above problem list is quite long and it is not possible to make use of all of them in a single or even a two semester fluid mechanics course. Each problem should be considered one by one from an educational standpoint. Here the interest is not just being able to solve these problems numerically, but they have to be designed in a way that they clarify the issues that are harder to understand through other means. Depending on the CFD software available, templates can be prepared for these problems and step by step instructions can be provided to the students to perform relatively easy simulations. For many of them results of the simulations can directly be provided and the students can be asked to only perform the postprocessing step and then discuss the results.

Virtual Flow Lab Software

Today it is possible to find a number of commercial CFD software in the market that can successfully perform all the above mentioned simulations. However as mentioned in the previous part, when it comes to using CFD as an educational tool, a software tailored specifically for this purpose would be a better choice. One advantage of using such a software will be its cost. Due to the relative simplicity of the problems that such a software is expected to be used for, its capabilities would be limited compared to a full featured CFD software resulting in a more economical product. Also the price would be affected by the fact that the software would be used for educational purposes and it should be affordable by the students. Such educational software should also be easy to learn and use. It should support the use of templates, problems that are parametrically designed and created by the instructor to be used in a series of numerical experiments. Templates could be prepared in as much detail as the instructor wants. For example to study the drag characteristics of a certain object placed in a flow field, a template can be created based on the parameters such as the upstream fluid velocity, viscosity of the fluid and the dimensions of the object. Students can use such a template to easily perform a number of numerical tests to see the effect of different parameters. Mesh generation, solution and postprocessing steps can all be predefined in the template, which means that the student can obtain the results by the click of a button after setting the problem parameters to the desired values. For an undergraduate level fluid mechanics course the details of mesh generation and the selection of solver parameters should be hidden from the students as much as possible.

When the literature about the use of CFD for educational purposes is investigated it is easy to see that the software alternatives are very limited. Most of the studies used FlowLab[5,6], which is a tailored version of Fluent designed to be used in teaching both fluid mechanics and CFD itself. CFD Studio[7] is another example of CFD software written to be used as an educational tool. In this part of the paper another software called Virtual Flow Lab, designed and written by the authors will be presented.

Virtual Flow Lab is an open source software. It is distributed under the GPL license[8], which means that it can freely be downloaded from the internet[9]. The source codes can also be downloaded and if necessary they can be modified by the users according to their specific needs. It is written in C++ language. User interfaces are designed with Qt[10], a C++ user interface library freely available from Trolltech. It is a platform independent software which can be compiled and run under different computer architectures without no or very little source code change. It is successfully tested under Linux and Windows operating systems. It is a multi-lingual software, which means that the user interface can be switched to the desired language with the click of a button. Using the tools provided by Trolltech, it is possible to translate the whole user interface from one language to another in just a few hours. The Virtual Flow Lab project is still under development and the capabilities of the current version of it will be presented in the following paragraphs.

Virtual Flow Lab is written to be a complete CFD package. Most of the CFD software written as a part of an academic study includes not more than a solver part. In such an approach the user first needs to draw the problem geometry and prepare the mesh using a mesh generation software. Then an input file that includes the solver parameters, boundary and initial conditions, etc. is prepared. The generated mesh and this input file is fed into the solver to obtain the numerical results. These results need to be analyzed using a visualization software. This is not a very user friendly approach. It is also not economical due to the need of separate pre- and post-processing software. Virtual Flow Lab tries to include relatively simple but functional pre-processing, solver and post-processing tools in a single software.

Screenshots of Virtual Flow Lab taken during the geometry definition and mesh generation steps are given in Figures 1 and 2. The user can draw the geometry of the problem using elementary geometrical entities such as lines, circular arcs, splines, etc. Then the mesh points on the boundaries of the problem should be selected. A multi-block structured mesh can then be generated using either algebraic of differential mesh generation methods[11]. Figure 3 shows a screenshot taken during the specification of boundary and initial conditions.

Specification of solver parameters and the convergence monitoring capability of the software can be seen in Figure 4. The incompressible flow solver is based on the SIMPLE algorithm of Patankar[12]. Its variations such as SIMPLER and SIMPLEC are also available as alternative solver schemes. Space discretization can be made using one of Central, Upwind, Power Law or Hybrid choices. It is possible to perform steady or unsteady simulations on staggered or collocated mesh arrangements. Work is in progress to extend the incompressible flow solver to an all-speed solver so that compressible flow simulations can also be performed[13].

As seen in Figure 5 it is very simple to create external flow fields by selecting proper blocked cells in a Cartesian mesh, which will be treated as walls in a simulation. Other than the real-time convergence plot, it is also possible to select the desired number of control points and visualize the convergence of the solution by following the plots of variations of the flow properties at these points. After the convergence of the solution the results can be post-processed. Although the post-processing module of the software is currently under development, an early screenshot showing a contour plot is given in Figure 6.

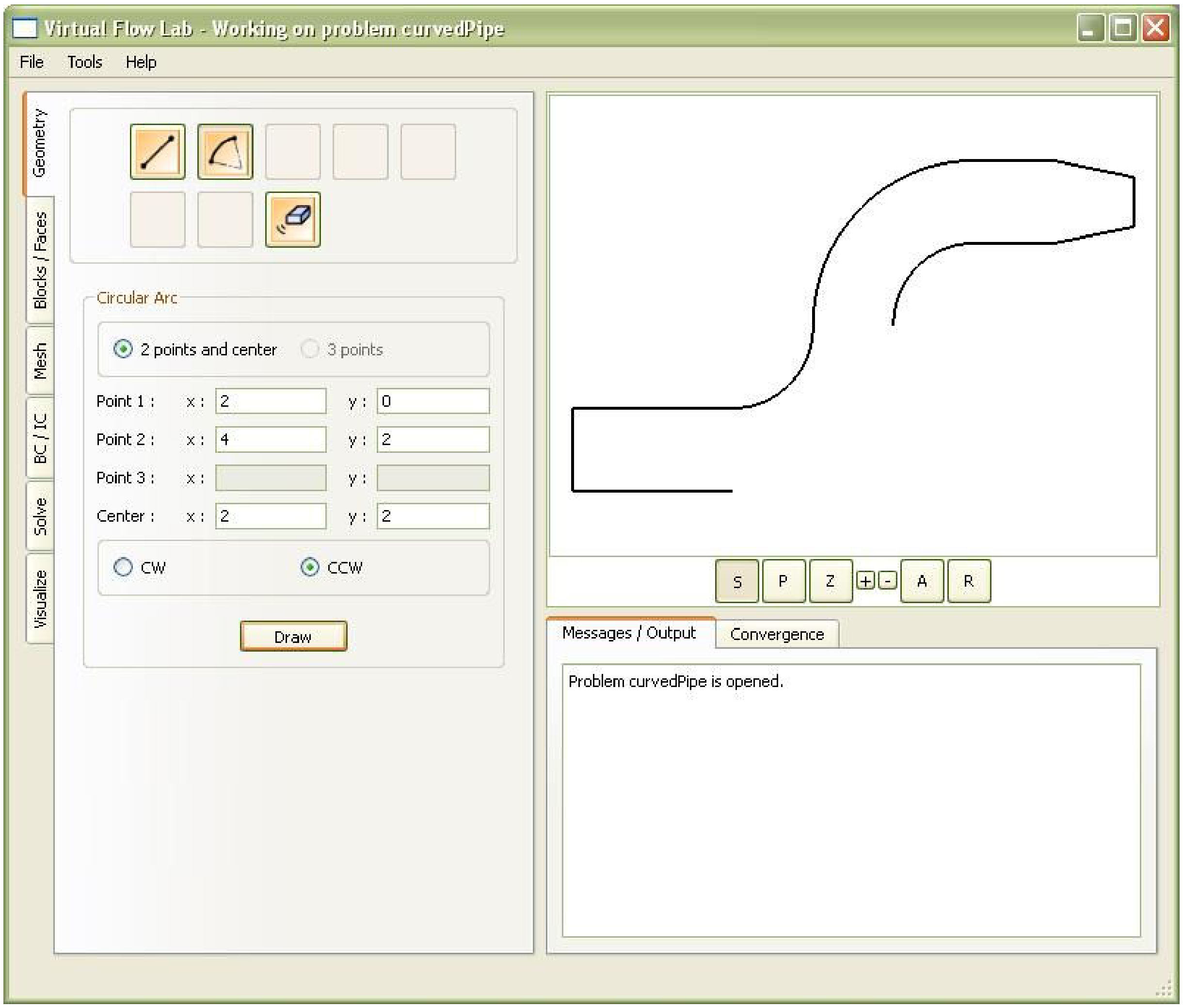

Figure 1. A screenshot of Virtual Flow Lab taken during the creation of a problem geometry

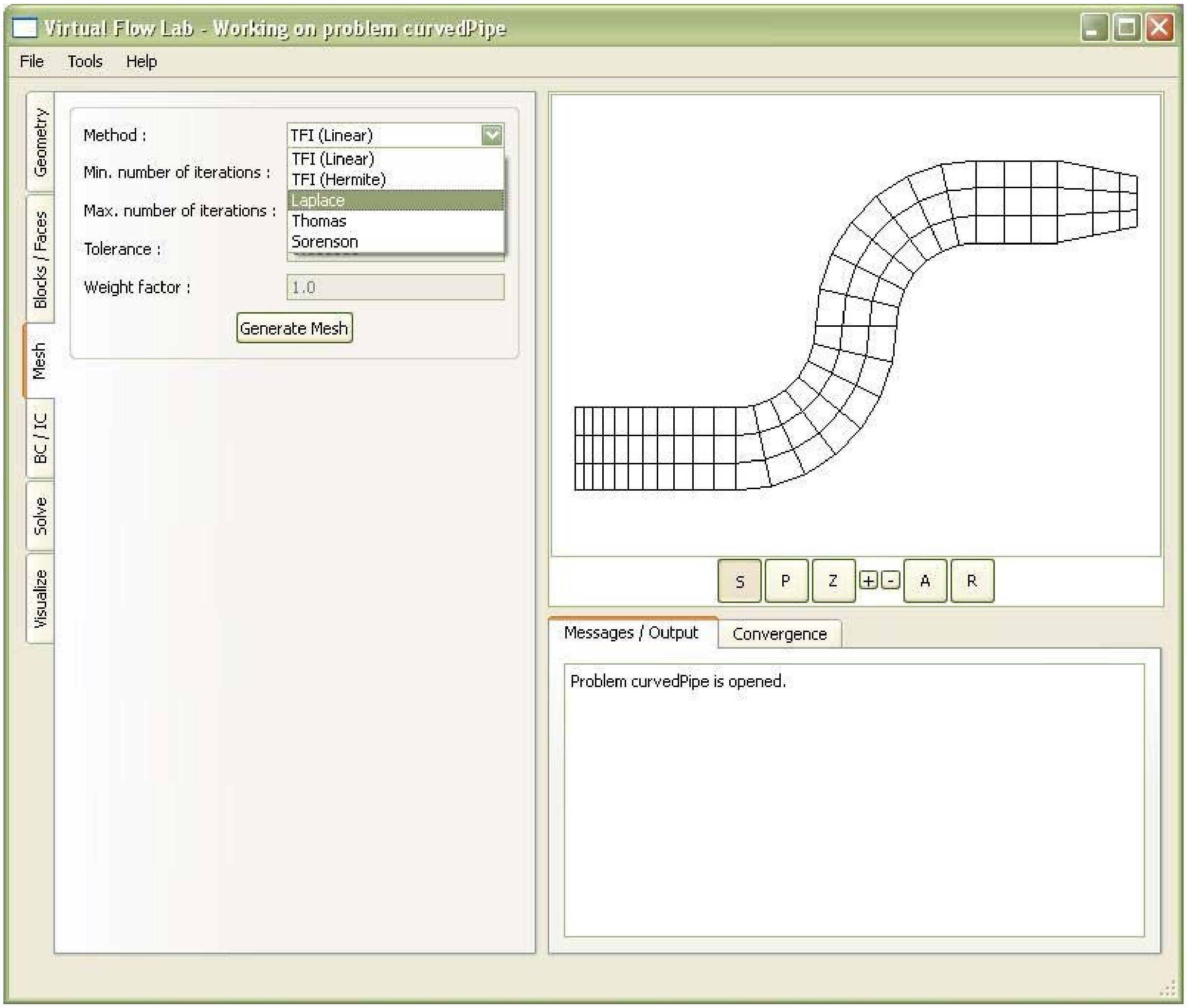

Figure 2. A screenshot of the Virtual Flow Lab taken during mesh generation.

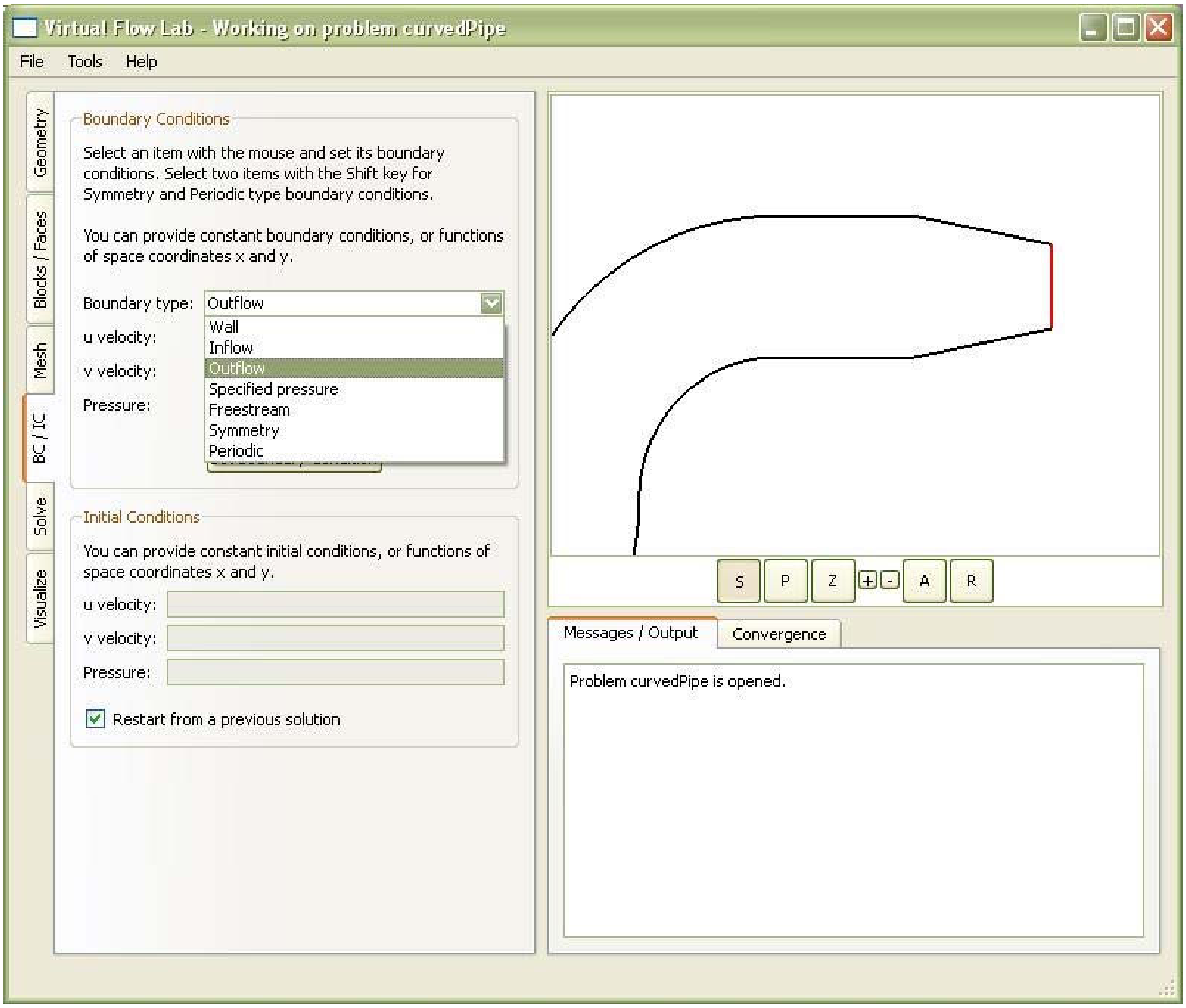

Figure 3. A screenshot of Virtual Flow Lab taken during the specification of boundary and initial conditions

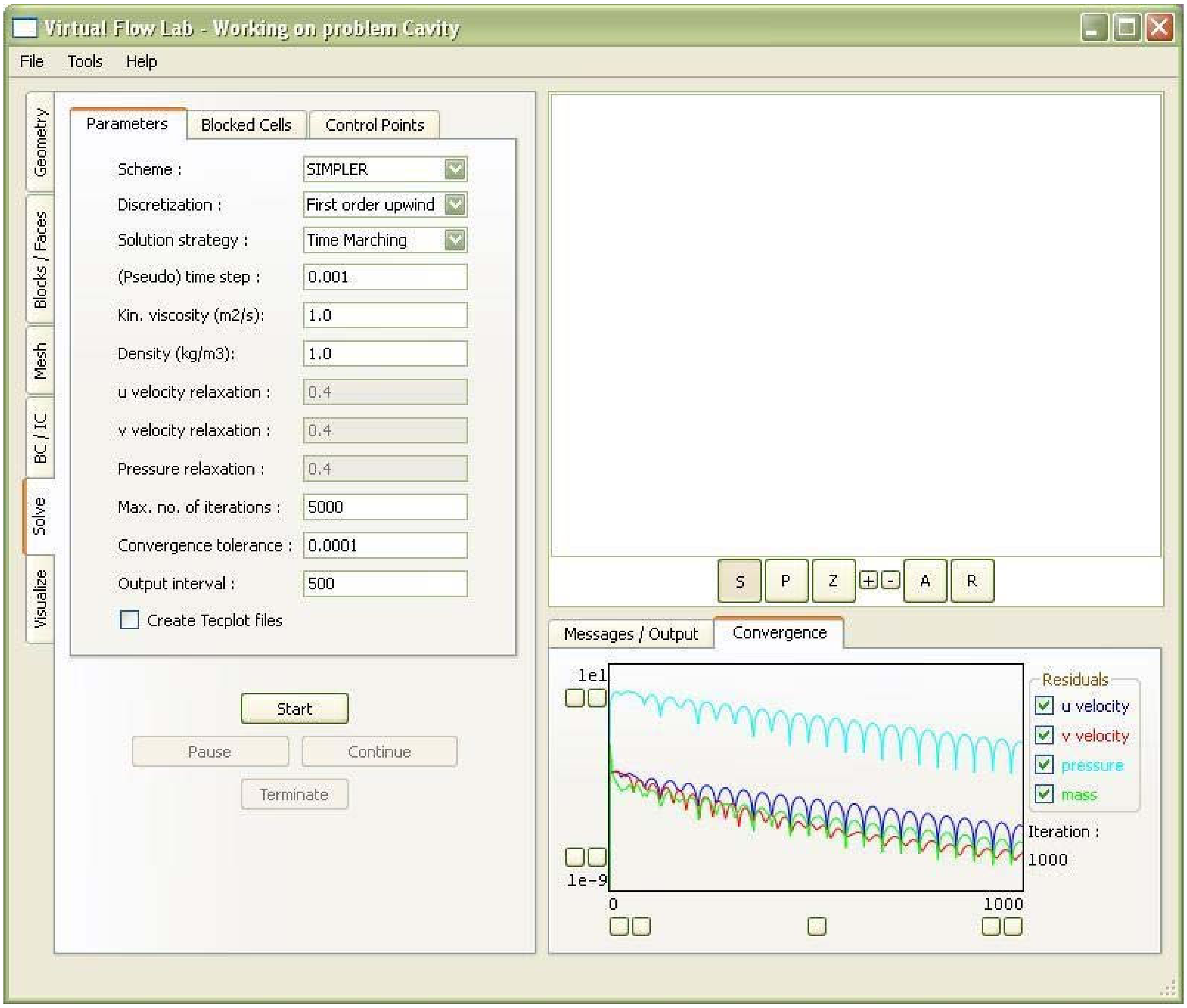

Figure 4. A screenshot of Virtual Flow Lab showing the convergence monitoring of a running simulation

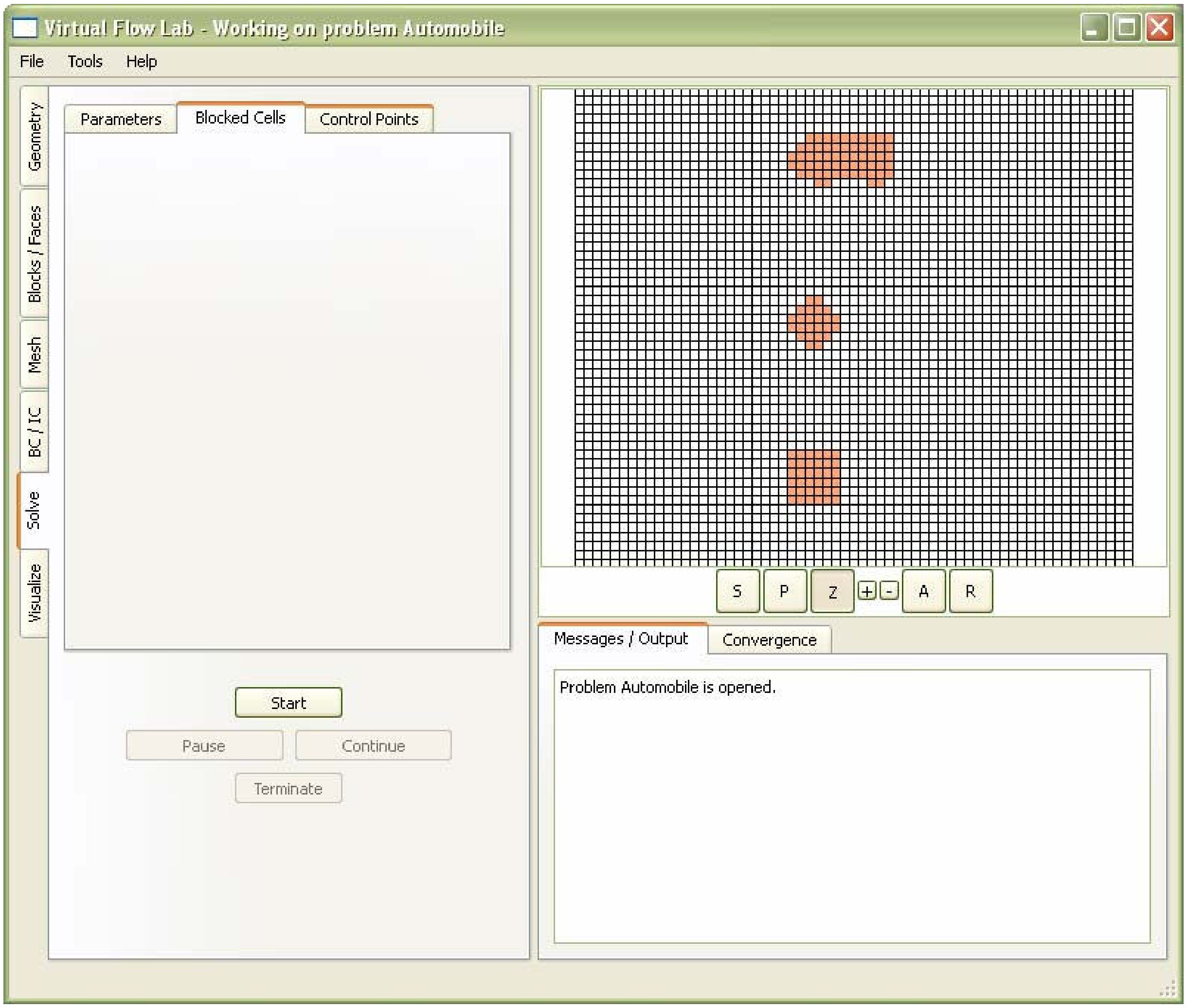

Figure 5. A screenshot of Virtual Flow Lab taken during the selection of blocked cells.

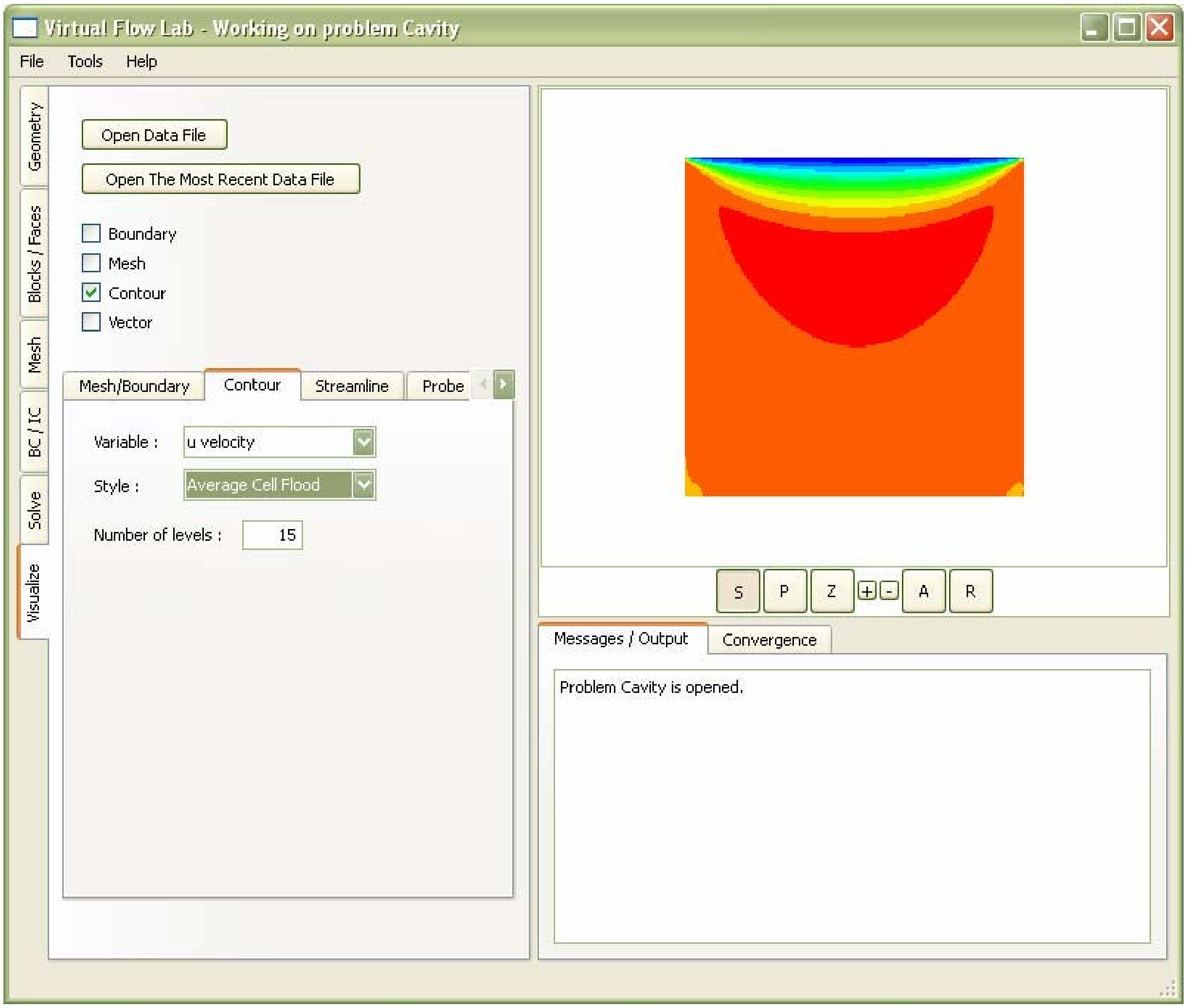

Figure 6. A screenshot of Virtual Flow Lab showing the postprocessing step

Conclusions

In this study the need and the possible use of Computational Fluid Dynamics (CFD) in undergraduate level fluid mechanics courses is discussed. It is believed that carefully selected problems will help a lot in understanding the relatively harder topics of fluid mechanics. CFD is also useful in getting the attention of students to fluid mechanics. The selection of the software that will be used for this purpose is important. Ease of learn and use and the accessibility are the most important features of such an educational software. A freely available, open source software developed by the authors to be such an educational tool is introduced. Although a long list of CFD simulation ideas is presented in this paper, a future study should be based on the testing of these ideas one by one to see the support they can provide from an educational standpoint. Only after such a careful study it is possible to select the appropriate simulations to be included as supporting teaching material of a fluid mechanics course.

Bibliography

1. Y. A. Cengel, J. M. Cimbala, *Fluid Mechanics Fundementals and Applications*, McGraw-Hill, New York, 2006.
2. NCFMF Fluid Mechanics Films, http://web.mit.edu/fluids/www/Shapiro/ncfmf.html
3. Hunter Rouse (IIHR) Fluid Mechanics Films, http://www.iihr.uiowa.edu/products/dhrm.html
4. D. Pines, Using Computational Fluid Dynamics to Excite Undergraduate Students about Fluid Mechanics, ASEE Annual Conference and Exposition, Session: 1055, 2004.
5. R. D. LaRoche, B. J. Hutchings, R. Muralikrishnan, FlowLab: Computational Fluid Dynamics (CFD) Framework for Undergraduate Education, ASEE Annual Conference and Exposition, Session: 1258, 2002.
6. R. Bhaskaran, L. Collins, Integration of Simulation into the Undergraduate Fluid Mechanics Curriculum using FLUENT, ASEE Annual Conference and Exposition, Session: 1637, 2003.
7. R. A. Pieritz, R. Mendes, R. F. A. F. Da Suva, C. R. Maliska, CFD Studio: An Educational Software Package for CFD Analysis and Design, Computer Applications in Engineering Education, vol. 12 (1), pp. 20-30, 2004.
8. GNU General Public License (GPL), http://www.gnu.org/copyleft/gpl.html
9. Homepage of Virtual Flow Lab software, http://www.me.metu.edu.tr/cuneyt/VirtualFlowLab
10. J. Blanchette, M. Summerfield , *C++ GUI Programming with Qt 4*, Prentice Hall, 2006.
11. J.F. Thompson, B. Soni, N.P. Weatherrill, *Handbook of Grid Generation*, CRC Press, 1998.
12. H.K. Versteeg, W. Malalasekera, *An Introduction to Computational Fluid Dynamics*, Addison Wesley Longman, 1995.
13. F. Moukalled, M. Darwish, A High-Resolution Pressure-Based Algorithm for Fluid Flow at All Speeds, J. Comp. Phys., vol. 168, pp. 101-133, 2001.