CFD MODELING OF WATER FLOW THROUGH SUDDEN CONTRACTION AND EXPANSION IN A HORIZONTAL PIPE

V.V.R. Kaushik, S. Ghosh, G. Das, and P.K. Das
IIT Kharagpur • 721302, India

Over the last few decades, there has been a growing interest in Computational Fluid Dynamics (CFD) in addition to experimental and analytical methods, for the solution and analysis of fluid flow problems. Several commercial CFD software packages (Fluent, Star-CD, and CFX) are used in industry to design and analyze different types of flow situations. CFD has also been accepted widely in engineering education. Traditionally in academia, CFD methods have been taught at the graduate level. There are two different approaches to teaching CFD in current engineering education. The first is the traditional approach that emphasizes the numerical methods with little or no attention given to the use of commercial software. The second approach is to introduce a CFD software in the class without any emphasis being given to learning the software.[2] Such software products allow students to get started immediately without proper knowledge of geometry and mesh-creation skills. Students can generate the results just by the click of a button after setting the problem parameters.

The traditional CFD education has been aimed at rigorous numerical training with the major emphasis on algorithms and code development. In this type of teaching, students require a strong mathematical background as well as visualization power to appreciate the class. Lack of imagination may cause difficulty in understanding several concepts of computational fluid dynamics. This is the major lapse of the traditional way of teaching CFD at the graduate level.

In the second type, students have an exposure to teaching software in addition to the theoretical concepts of CFD.[3-7] In this approach, however, since students can get the results mechanically by the click of a button after setting the problem parameters, they often treat the software as a "black box," just to produce "colorful" results. It is more desirable that graduate students, with their understanding of the basic concepts, be able to validate the results and test the accuracy of any simulation. It is also important for the students to understand the limitations of these software products in solving different problems. Otherwise the "colorful" results of CFD can lead students to draw wrong or unrealistic predictions. The other major concern of this type of approach is that teaching software products are much-simplified versions of commercial CFD software used in industry. Therefore, students will need further training to use commercial software products after this type of course. Recently some courses on CFD were also developed that use commercial software products in the curriculum.[8-12] This type of course prepares students who have both theoretical concepts of CFD as well as a working knowledge of commercial software.

The objective of the current course is to teach students the fundamentals of computational fluid dynamics as well as to enable them to handle commercial software indepen-

V.V.R. Kaushik *obtained his Bachelor of Technology degree in chemical engineering from the Indian Institute of Technology, Kharagpur, India. At present he is working as graduate engineering trainee in an oil company.*

S. Ghosh *received her Master's of Technology degree from the Department of Cryogenic Engineering, Indian Institute of Technology, Kharagpur, in 2006 and is continuing there as a doctoral student. Her research interests are experimental and numerical techniques of multiphase flow. She has published three articles in well-recognized international journals to date.*

G. Das *is a professor in the Department of Chemical Engineering at Indian Institute of Technology, Kharagpur, India. Her research interests include multiphase flow, two-phase instrumentation, and computational fluid dynamics. She has published more than 60 articles in reputed journals and conference proceedings. She has several patents from her research work. She is involved with a number of research projects and provided consultancy to various public and private industries.*

P.K. Das *is a professor in the Department of Mechanical Engineering, Indian Institute of Technology, Kharagpur. His research activities cover various topics of thermo fluids engineering, such as multiphase flow; extended surface heat transfer; analysis, optimization, and dynamic simulation of heat exchangers; thermal hydraulics of natural circulation loops; rewetting of hot solids; heat transfer in nano fluids; and hydraulic jump. He has published several book chapters and around 100 refereed international journal papers. He also has several patents. He has executed a number of projects funded by different funding agencies and provided consultancy to various public and private industries. He is a fellow of the Indian National Academy of Engineering.*

dently. This enables the students to simulate any realistic flow situation and also to interpret the data, analyze the data, and contribute to the design of the system. Incorporation of commercial software into the curriculum can expose students to the same or similar software they may be expected to use as professionals in industry. To prevent students from treating the software as a black box, the supplemented projects/assignments are defined so that the use of CFD becomes a meaningful learning experience that can be applied to real-life engineering situations. This paper describes two such assignments, namely flow-through contraction and expansion.

COURSE DESCRIPTION

This course, entitled CFD Application in Chemical Engineering, was introduced in the autumn semester of 2008 in the Department of Chemical Engineering at IIT Kharagpur. It is designed for final-year undergraduate and first-year graduate students. As noted earlier, the objective of the course is to develop theoretical concepts of computational fluid dynamics as well as have an exposure to commercial CFD software (FLUENT). Because it is an elective course, the number of students is not fixed each year. It varies between 30–35 students/semester.

The course comprises four hours of lecture class (three theories and one tutorial) per week. Thus, in one semester, the students are exposed to 56 hours of classroom lecture. The syllabus includes detailed discussion on governing equations, discretization schemes, staggered grid and pressure velocity coupling, turbulence modeling, and introduction to multiphase modeling. Apart from class lectures, the following assignments are given in the class.

a) *To develop a 3-D model to simulate the turbulent flow of water (Re=5,000 to 20,000) through a straight horizontal. The simulation results are validated against textbook information.*
b) *To develop a 3-D model to simulate the turbulent flow of water through an annulus. The simulation results are validated against Bird, et al.*[13]
c) *To develop a 3-D model to simulate turbulent flow of water through sudden contraction with diameter ratio (ratio of smaller to larger diameter, designated as β) varying from 0.3 to 0.9 for the same range of Reynolds (5,000 to 20,000) number and develop empirical correlation to predict the loss coefficients.*
d) *To develop a 3-D model for turbulent flow of water through sudden expansion for the same range of β and Reynolds number as in the previous case and develop empirical correlation to predict the loss coefficients.*
e) *To develop a 3-D model for turbulent flow of water through return bends and develop empirical correlation to predict the loss coefficients.*

In all the cases, Reynolds number is defined based on inlet pipe diameter. Hence, to keep the same range of Reynolds number in assignments c and d, the inlet velocity of water is adjusted accordingly.

Among the assignments, the first two are common to all students. The last three are treated as class projects. Students are required to execute the assignments in the department's centralized computer laboratory, in which FLUENT is installed in several PCs. The students are used to handling this software under the supervision of the course instructor during the tutorial session. Initially, emphasis is given on proper understanding of the design software GAMBIT. After that, students generally solve some simple fluid-flow problems (assignments a and b), the results of which can be checked and validated against available analytical solutions. After this, students are divided into groups and assigned different projects. They solve the problems using FLUENT and also critically examine the data or analyze the data to generate some fruitful results.

These exercises allow students to visualize and apply many of the concepts they have learned in the traditional CFD as well as Fluid Mechanics classes. These projects constitute 30% of the total grade. The gradation is based on the level of execution of the project by the group. Each project is divided into five parts. Completion of each part earns 20% of the project marks. The students who can complete the project, validate the results, and develop the required correlations are awarded an excellent grade (above 90% mark).

Course objectives

The course has the following objectives

a) *To understand the basic concepts of CFD*
b) *To familiarize students with CFD software FLUENT*
c) *To verify theoretical concept by numerical simulation*
d) *To develop an insight to the design of some complex fluid-flow problems*
e) *To critically examine the results and data*

STEP-BY-STEP IMPLEMENTATION OF THE ASSIGNMENTS

This section describes step-by-step implementation of two projects (assignments c and d). Initially the students are introduced to grid-generation software GAMBIT. Mesh models are explained and demonstrated using tutorials and the user guide. In the first one to two tutorial classes students are asked to go through the user guide and solve the tutorial problems to get accustomed to the software. Next they are given a demonstration, using one of the tutorial problems of FLUENT, to show how to handle the software. The working of the software and several concepts such as continuum, operating, and boundary are explained. Students are asked to go through the user guide for understanding the different models, how to choose the appropriate one, and general problems associated with operating the software.

After five introductory and practicing sessions students are assigned the first two problems as described in the previous section. They are instructed to check the grid independency of the results. With this exercise they get familiar with the post-processing tools (*e.g.*, velocity contours, x-y plots, path lines) to visualize and analyze the flow fields. Students verify the results for axial velocity profile and pressure drop with analytical results. They usually get excited upon noting the good agreement with text for both the cases. To complete these initial assignments a total time of three weeks has been allotted. For completion of course projects six to seven weeks have been allotted. Execution of these course projects is carried out in parts; first the students are asked to construct the geometries and generate the meshes. Next, they need to select the model, execute the program, and generate profiles of velocity, pressure, and pathlines as well as perform the trend matching of these profiles with textbooks. The progress of each group in

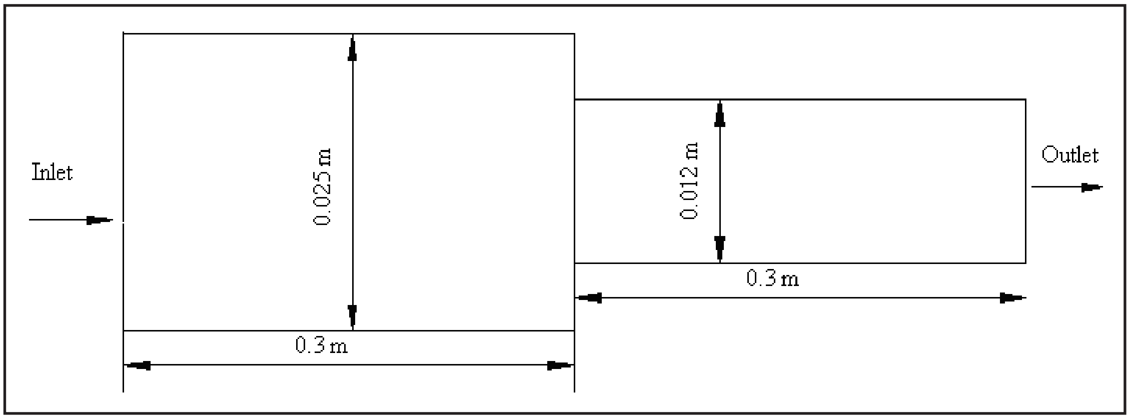

Figure 1. *Schematic of the geometry considered for modeling sudden contraction.*

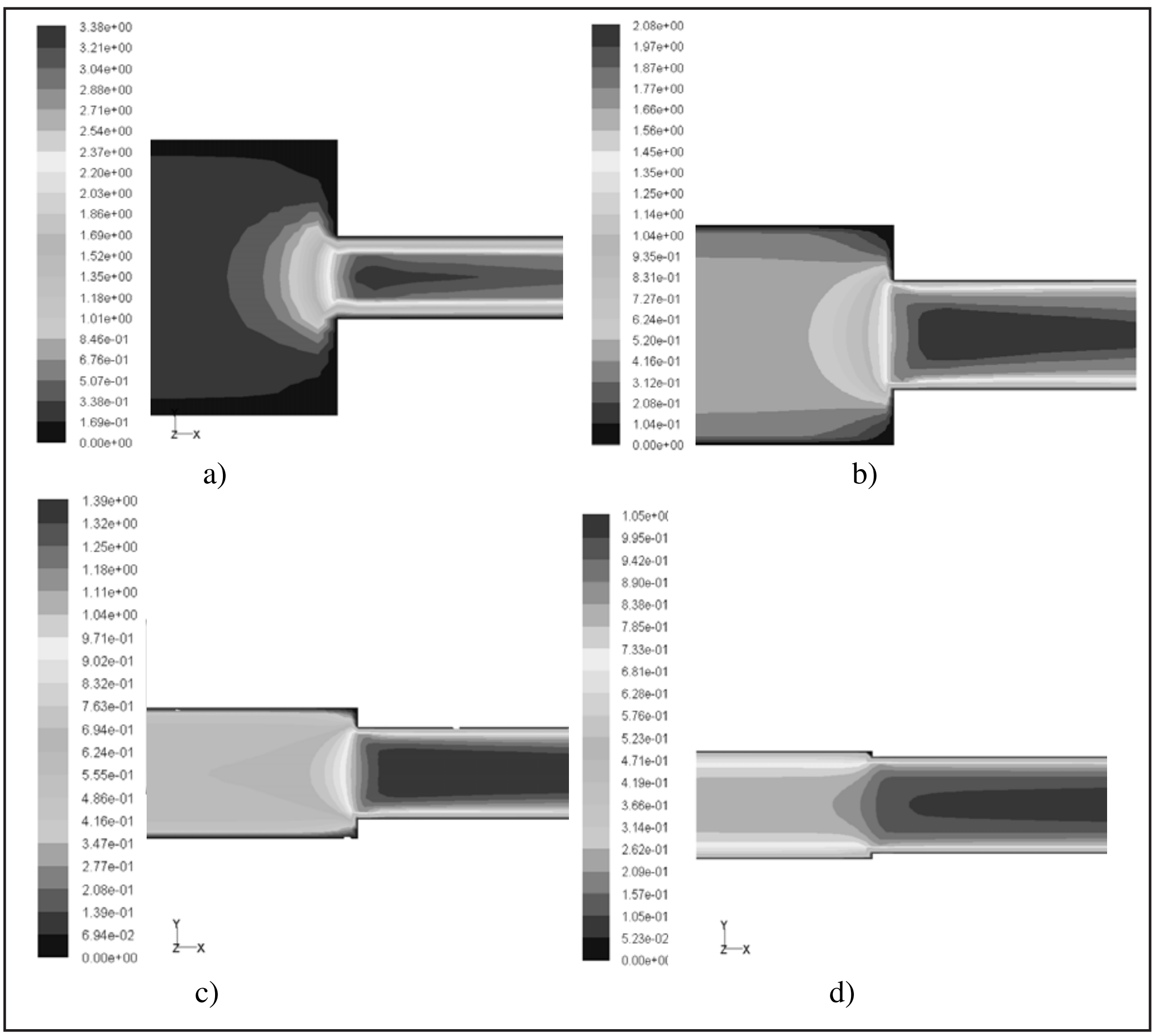

Figure 2. *Velocity contours of sudden contraction at Re=10,000 with a) ß=0.3, b) ß=0.5, c) ß=0.7, d) ß=0.9.*

its project is discussed during tutorial sessions, and after the group members successfully match the velocity and pressure profiles they are introduced to the concept of loss coefficients and encouraged to develop correlation to predict it. Two such course projects are discussed in this section.

a) Assignment: Model development and analysis of data for sudden contraction:

One of the course projects is to develop 3-D models to simulate turbulent flow through sudden contraction. Figure 1 schematically represents the flow geometry. The length of each section is 0.3 m. The smaller tube diameter is kept constant at 0.012 m and the larger pipe diameters are varied from 0.014 to 0.0423 m. such that the ratio of smaller-to-larger pipe diameter (β) ranges from 0.3 to 0.9. First-order upwind method is used for discretization of momentum equation. The turbulent kinetic energy and dissipation rate equations are also discretized by this method. For pressure velocity coupling, SIMPLE[14] is used. At the inlet, velocity of the fluid is specified. The velocity is normal to the inlet plane. A stationary no-slip boundary condition is imposed on the wall of the pipe. Pressure outlet boundary is used at the outlet. After developing the model the students analyze the velocity profile (Figures 2a-d) and then the pressure profiles (Figure 3). The trend of the profile agrees well with that reported in textbooks.

This analysis enables students to have a clear understanding of the effect of sudden contraction on these two variables. Further, they also visualize boundary-layer separation during sudden contraction (Figure 4). This enhances their understanding of fundamental concepts of fluid mechanics that have been taught earlier.

b) Assignment: Model development and analysis of data for sudden expansion:

In this assignment students develop a 3-D model to simulate flow of water through sudden expansion for the same range of the Reynolds number. A schematic of the flow geometry is shown in Figure 5. The diameter ratio and length of the sections as mentioned in Figure 5 are the same as that of contraction. Figures 6a-d depict the velocity profile at expansion for the diameter ratio of 0.3 to 0.9 and Reynolds number of 10,000. Figure 7 depicts the pressure profile in expansion. Its trend is matching well with the textbooks. The students also generate the path lines to visualize the circulating zone near the plane of area change (Figure 8).

c) Theoretical analysis:

Finally, after successful completion of model development and trend matching, students are asked to generate additional information that can be useful for designing such flow systems. This is done to encourage students to handle and analyze the information available from a model. Flows through such fittings are characterized by a significant loss of mechanical energy at the plane of area change due to boundary-layer separation. Proper knowledge of contraction and expansion frictional energy loss is important in design as well as control of the flow system. Consequently, frictional energy losses are expressed in terms of loss coefficient as:

$$h_f = k_c \frac{U^2}{2} \text{ for contraction} \tag{1}$$

and

$$h_f = k_e \frac{U^2}{2} \text{ for expansion} \tag{2}$$

where U is the velocity in the smaller pipe, h_f is the frictional energy loss per unit mass of flowing fluid, and k_c and k_e are the contraction and expansion loss coefficients, respectively. Past researchers have studied loss coefficients[15-18] as a function of diameter ratio and estimated k_c using the empirical formula based on area ratio given in White[17] viz.:

$$k_c = 0.42\left(1-\beta^2\right) \tag{3}$$

Similarly, in case of expansion the Borda-Carnot equation as described by Massey[18] is widely used to predict the expan-

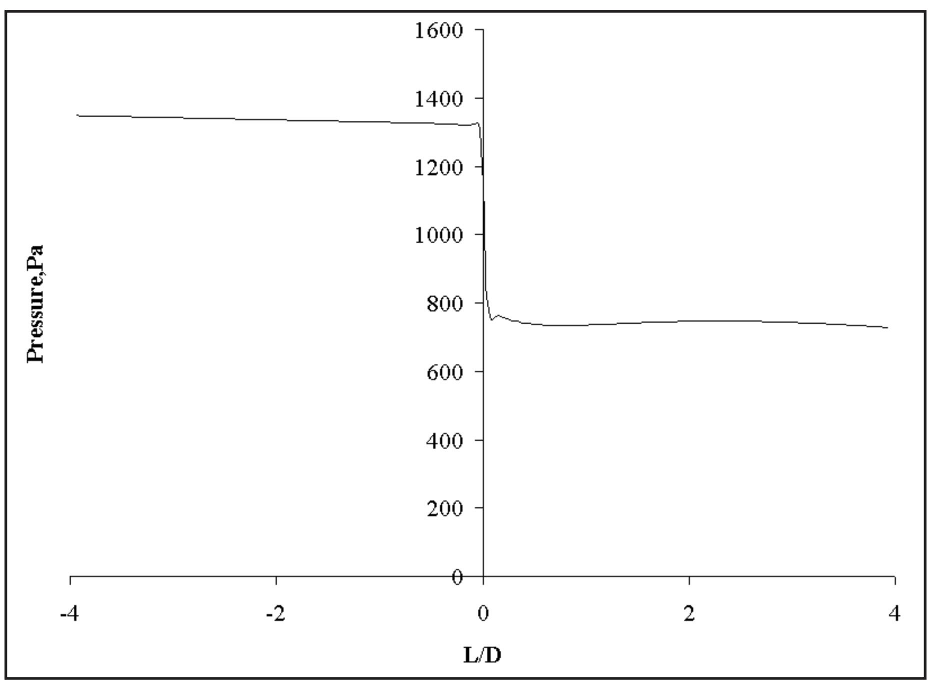

Figure 3. *Pressure profile of sudden contraction at Re=10,000 and ß=0.5.*

sion loss coefficient as:

$$k_e = \left(1-\beta^2\right)^2 \tag{4}$$

Students calculate the loss coefficients using the pressure drop data obtained from the simulation at the contraction-expansion plane. Next, the effects of inlet Reynolds number (based on the inlet pipe diameter) and smaller-to-larger pipe diameter ratio (β) on loss coefficient are analyzed. Figure 9a shows the variation of k_c with diameter ratio for different Reynolds numbers and compares it with the available literature data. Figure 9b represents the same for expansion. A good agreement is observed for Idelchik[16] in case of contraction. A deviation is noted for the larger-diameter ratio in case of expansion.

Finally, students have developed two empirical correlations using software lab-fit for both contraction and expansion loss coefficient in the turbulent regime.

$$k_e = 0.0000045\,\mathrm{Re} - 1.039\beta + 1.11 \tag{6}$$

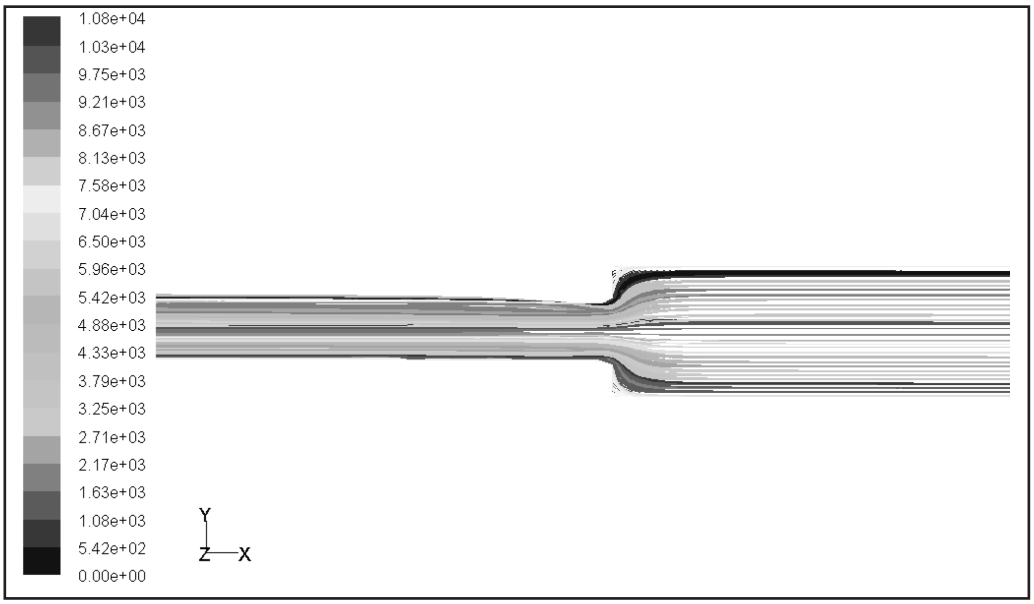

Figure 4. *Contours of path lines of sudden contraction at Re=10,000 and ß=0.5.*

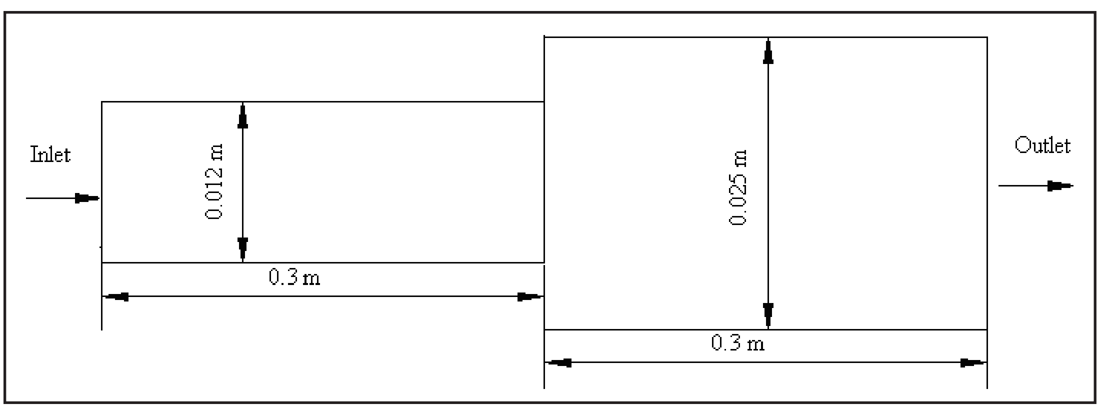

Figure 5. *Schematic of the geometry considered for modeling sudden expansion.*

CONCLUSION

This paper discusses the structure of a computational fluid dynamic course in the Department of Chemical Engineering at Indian Institute of Technology, Kharagpur. With the combination of classroom teaching, tutorial sessions, and course projects, we have introduced CFD as a design tool to students.

We observed that students could quickly obtain enough knowledge of CFD while investigating fluid-flow problems under the supervision of the instructor. They could generate geometry and appropriate meshes in GAMBIT, chose physical models, solve numerical problems, and visualize and analyze the flow field with post-processing tools available in FLUENT.

It has been noted that similar types of course structures are proposed by many researchers.[3,5-7] Most of these courses use various teaching software with little or no emphasis given on geometry-creation and mesh-generation steps. As a result, students get the results mechanically by the click of a button after setting the problem parameters. In the present course, students are first exposed to drawing software GAMBIT. They learn to create geometry and mesh it. They are even encouraged to perform grid-independent tests so that they are accustomed to all aspects of CFD after the course. It may be noted that assignments of the present course can also be demonstrated by using teaching software like Flow Lab. Thus, the present course provides an opportunity for students to apply their fundamental knowledge and contribute to the design issues of such systems apart from just visual observation of velocity and pressure distribution.

Assessment of students is done on the basis of level of completion of each step of the tutorial class as well as on the full completion of the course project. Student improvement is documented by conducting a test at the end of the semester. Frequent discussions are undertaken with students on the effectiveness of the course contents and assignments. The course is upgraded by regular feedback from students. From the feedback, it is observed that the purposes of this course are fulfilled, which is to help students gain understanding of CFD application in industry design and the internal structure and operation of CFD solvers, build up their knowledge of fluid mechanics, and interpret and validate CFD results as well as understand some advanced concepts. Students also preferred FLUENT as a teaching tool used in the class because FLUENT is a commercial package used in industry. Hence an exposure to FLUENT may help them in securing a good job.

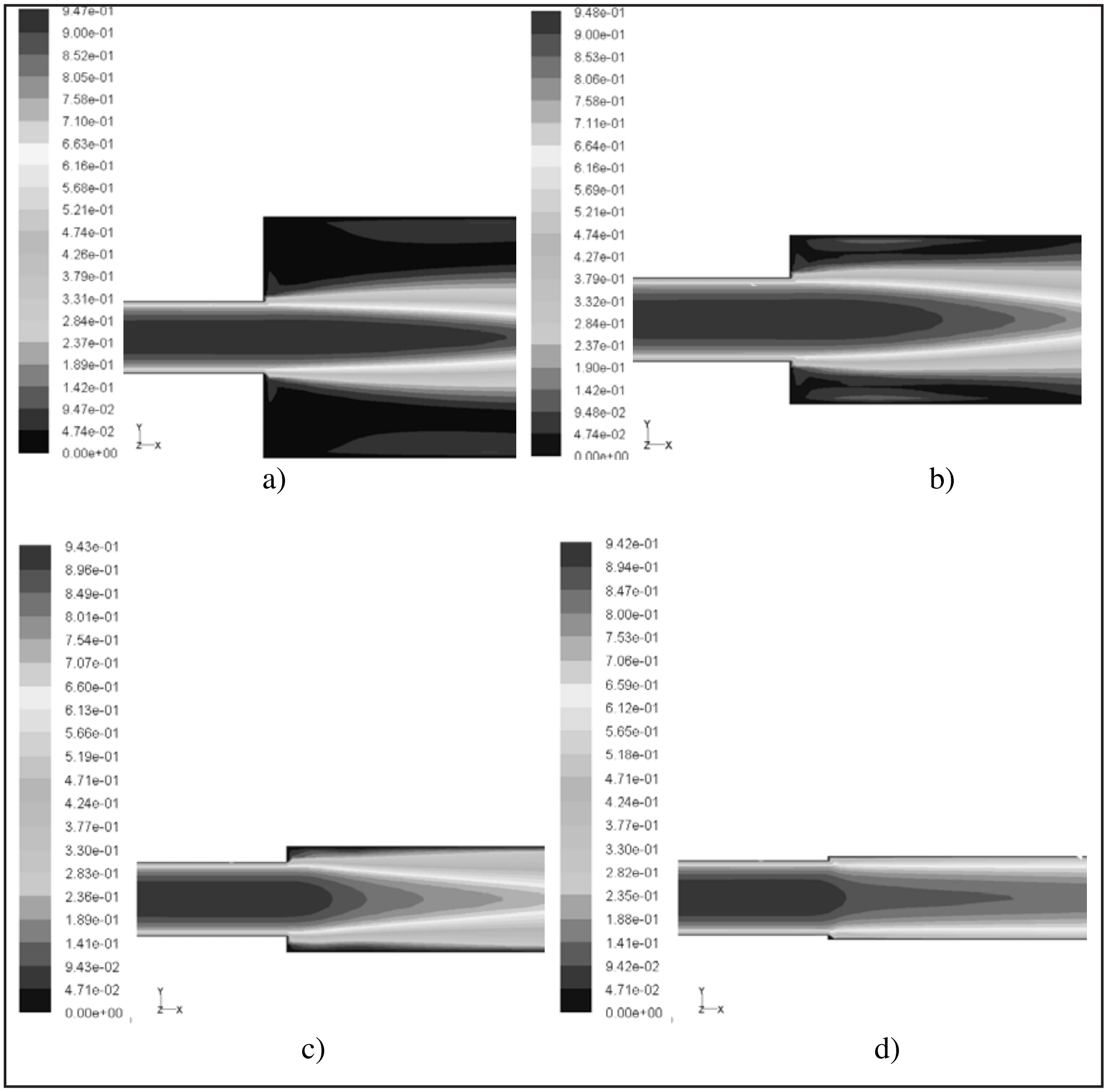

Figure 6. *Velocity contours of sudden expansion at Re=10,000 with a) ß=0.3, b) ß=0.5, c) ß=0.7, and d) ß=0.9.*

REFERENCES

1. Fluent 6.2 User's Guide, 2005. Fluent Inc., Lebanon, NH
2. Hu, J., L. Zhang, and X. Xiong, "Teaching Computational Fluid Dynamics (CFD) to Design Engineers," ASEE *Proceedings* (2008)
3. La Roche, R.D., B.J. Hutchings, and R. Muralikrishnan,

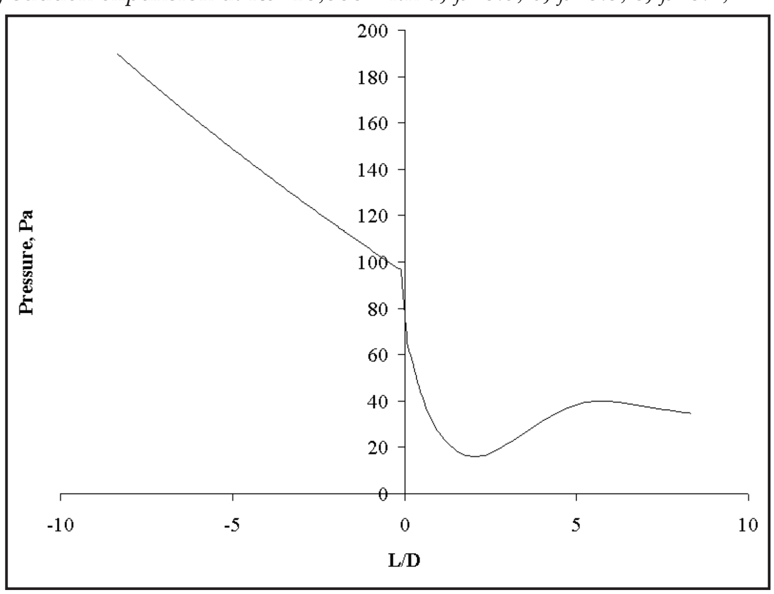

Figure 7. *Pressure profile of sudden expansion at Re=10,000 and ß=0.5.*

Flow Lab: Computational Fluid Dynamics (CFD) Framework for Undergraduate Education, ASEE/SEFI/TUB Colloquium (2002)
4. Hailey, C.E., R.E. Spall, and D.O. Snyder, "Computational Fluid Dynamics Presented in the Undergraduate Engineering Curriculum," *Computers Eng. J.*,**11,** 2-8 (2001)
5. Blekhman, D., "Lessons Learned in Adopting a CFD Package," ASEE *Proceedings* (2007)
6. Sert, C., and G. Nakiboglu, "Use Of Computational Fluid Dynamics (CFD) in Teaching Fluid Mechanics," ASEE *Proceedings* (2007)
7. Stern, F., T. Xing, D. Yarbrough, A. Rothmayer, G. Rajagopalan, S.G. Otta, D. Caughey, R. Bhaskaran, S. Smith, B. Hutchings, and S. Moeykens, "Development of Hands-On CFD Educational Interface for Undergraduate Engineering Courses and Laboratories," *Proceedings* ASEE Annual Conference & Exposition, June, Salt Lake City, Utah (2004)
8. Navaz, H.K, B.S. Henderson, and G. Mukkilmarudhur, "Bring Research and New Technology Into the Undergraduate Curriculum: A Course in Computational Fluid Dynamics," ASEE *Proceedings*, June, Seattle (1998)
9. Guessous, L., R. Bozinoski, R. Kouba, and D. Woodward, "Combining Experiments With Numerical Simulations in the Teaching of Computational Fluid Dynamics," ASEE Annual Conference & Exposition *Proceedings*, June, Nashville, TN (2003)
10. Aung, K., "Design and Implementation of an Undergraduate Computational Fluid Dynamics (CFD) Course," ASEE Annual Conference & Exposition *Proceedings*, June, Nashville, TN (2003)
11. Pines, D., "Using Computational Fluid Dynamics to Excite Undergraduate Students about Fluid Mechanics," ASEE Annual Conference & Exposition *Proceedings*, June, Lake City, Utah (2004)
12. Bhaskaran, R., and L. Collins, "Integration of Simulation into the Undergraduate Fluid Mechanics Curriculum using FLUENT," ASEE Annual Conference & Exposition *Proceedings*, June, Nashville, TN (2003)
13. Bird, R.B., W. Stewart, and E.N. Lightfoot, *Transport Phenomena*, 2nd Ed., Wiley, New York (2001)
14. Patankar, S.V., and D.B. Spalding, "A Calculation Procedure for Heat, Mass, and Momentum Transfer in Three-Dimensional Parabolic Flows," *Int. J. Heat and Mass Transfer*,**15**, 1787 (1972)
15. Benedict, R.P., N.A. Carlucci, and S.D. Swetz, "Flow Losses in Abrupt Enlargements and Contractions," *Trans. ASME, J. Eng for Power* **88**,73-81 (1966)
16. Idelchik, I.E., *Handbook of Hydraulic Resistance*, U.S. Atomic Energy Comm., AEC-tr-6636 (1966)
17. White, F.M., *Fluid Mechanics*, McGraw-Hill, New York (1987)
18. Massey, B., *Mechanics of Fluids*, Nelson Thrones Ltd., U.K. (2001) ❐

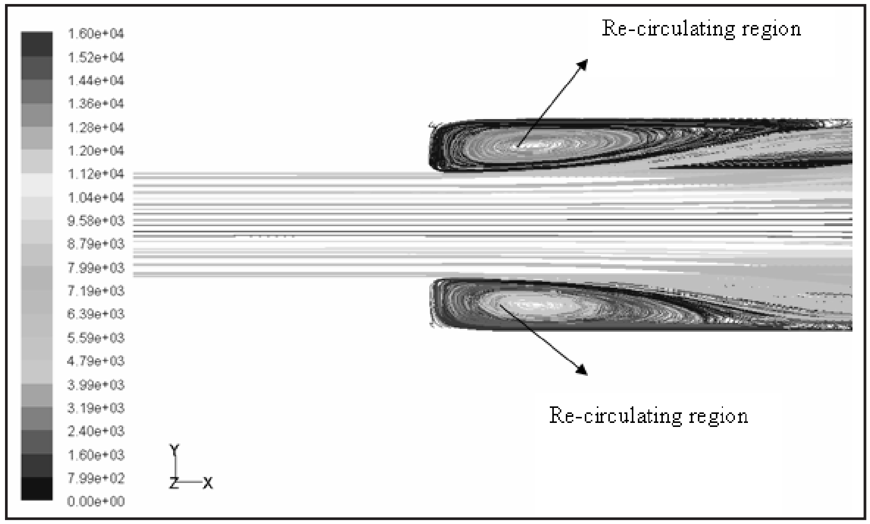

Figure 8. *Path lines sudden expansion at Re=10,000 and ß=0.5.*

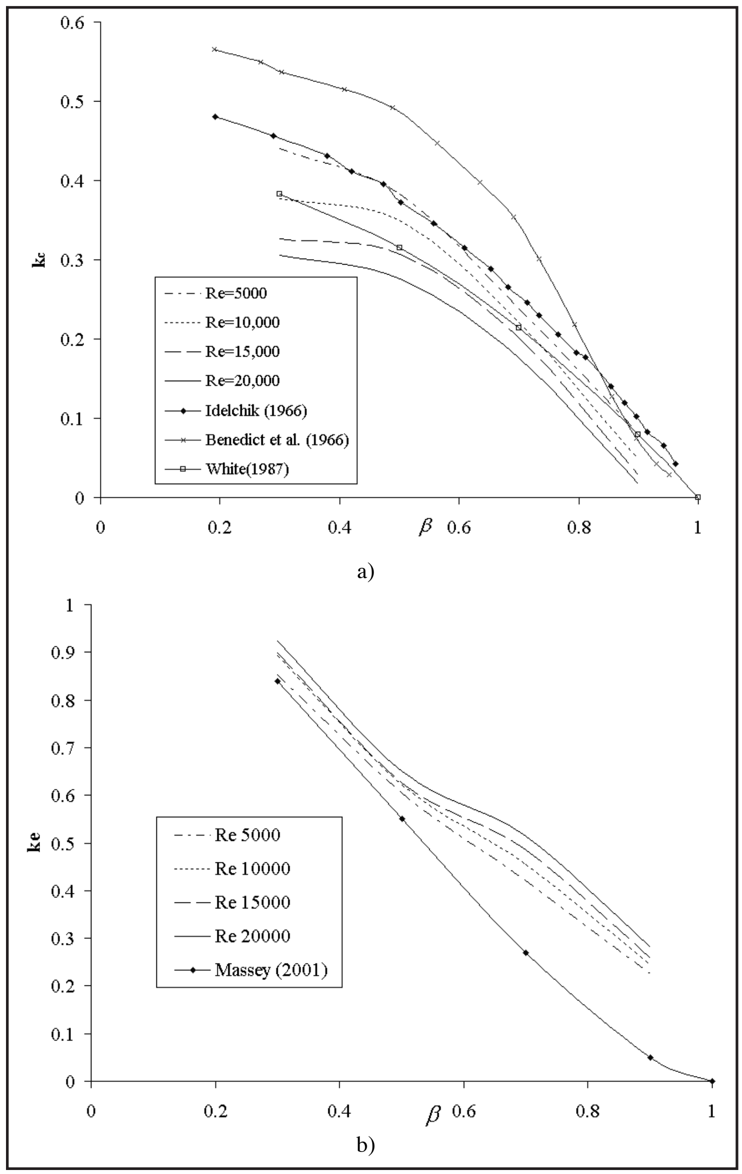

***Figure 9.** Comparison of numerically predicted loss coefficients with literature, (a) Contraction and (b) Expansion.*