Enhancing Undergraduate Understanding of Transport Phenomena via Application of CFD

Jennifer Sinclair Curtis
University of Florida

AIChE Annual Meeting
November 6, 2007

CFD

- "Computational Fluid Dynamics"
  - Numerical solution of differential momentum balance and continuity equation (typically finite-volume technique)
  - Sometimes accompanied by the solution of turbulent transport equations, differential energy balance, and/or species continuity equations
  - Non-Newtonian flows, multiphase flows - may need to supply additional constitutive models or closure relations

Increase in Application of CFD

- Due to advances in computer memory and speed, CFD is now a common tool in industry
  - Aerospace
  - Automotive
  - Biomedical
  - Chemical Processing
    - Single and multi-phase flow
    - Heat and mass transfer
    - Chemical reaction
  - Energy

- In industry, often used for design or a flow diagnostic tool
  - More cost effective testing than experimental testing
  - Useful when measurements are difficult
  - Easy to explore different designs and operating conditions
  - Scale-up

CFD in Education

- In academic setting, CFD often used as a research tool and sometimes in graduate education
- Huge potential to use CFD as to enhance undergraduate transport education
  - CFD has many potential applications in a typical fluid flow or transport course
  - CFD can be easily incorporated into a typical fluid flow or transport course

CFD in Undergraduate Education

- CFD is still largely underutilized in the undergraduate curriculum
  - time constraints in the curriculum
    - fitting it into the syllabus
    - time to train the students
  - lack of knowledge of CFD tools available
  - lack of knowledge of the possible uses of CFD for enhancing teaching
  - training of faculty an issue, need a “champion” in the department

Why Use CFD in Education?

- Enhance understanding of concepts through visualization
  - Students can see developing profiles, boundary layer development, regions of recirculation
  - Graphics bring life to the lecture or homework assignment
  - Computer laboratory - explore the effects of changes in fluid properties, system geometry, and operating conditions
- Increases student interest in transport

Why Use CFD in Education?

- Investigate fluid flow problems beyond those involving one-dimensional, steady state, fully developed flow (ones that can be solved analytically)
  - Without knowledge of CFD, students can easily get the impression that if the flow situation is complex, empiricism is needed to analyze the process

Why Use CFD in Education?

- Introduction to the numerical solution of the continuity equation and momentum balance
  - Finite-Volume Technique, Grid, Iteration, Residuals, etc.
- Learn a bit about turbulence modeling
  - Move beyond Prandtl mixing length
  - Strengths and weaknesses of turbulence modeling
- Students become familiar with a tool for design/scale-up/optimization of flow processes

Why Use CFD in Education?

- Compare CFD results to:
  - Analytical solutions discussed in class
  - Results from empirical correlations
  - Their own laboratory data or published data
- Evaluate turbulence models, closure relations, grid density, etc.

CFD Codes

- In CFD, modeling equations are discretized into algebraic equations using numerical methods (finite volume technique). The system of algebraic equations is solved in an iterative fashion.
- Numerical solution includes:
  1. Grid generation
  2. Discretization method
  3. Solvers
  4. Post-processing

CFD Codes for Education

- Training on use of code
  - At least several lecture periods

- Time to set-up problem - students are very involved in problem set-up and numerical solution strategy
  - Generation of grid
    - For complex geometries need to import the database of the geometry into CFD software
    - Advantage – flexibility
  - Physical model - inlet and boundary conditions, closures needed, etc.
  - Decisions concerning time step, simulation domain size, etc. related to solver

- Time to reach a converged solution

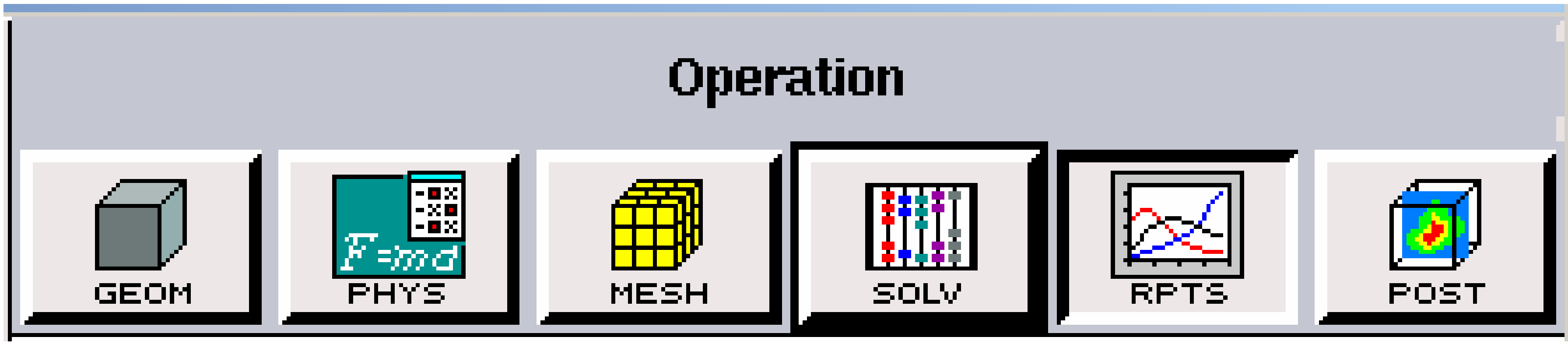
CFD Code - FlowLab

- "Standard geometries" in FlowLab
  - Flexibility in...
    - Geometry size, relative dimensions
    - Fluid properties
    - Fluid velocity
- Physical model – some options
- Discretization method/solver – mostly hidden from user
- Wide range of post-processing options and graphics

CFD Code - FLOWLAB

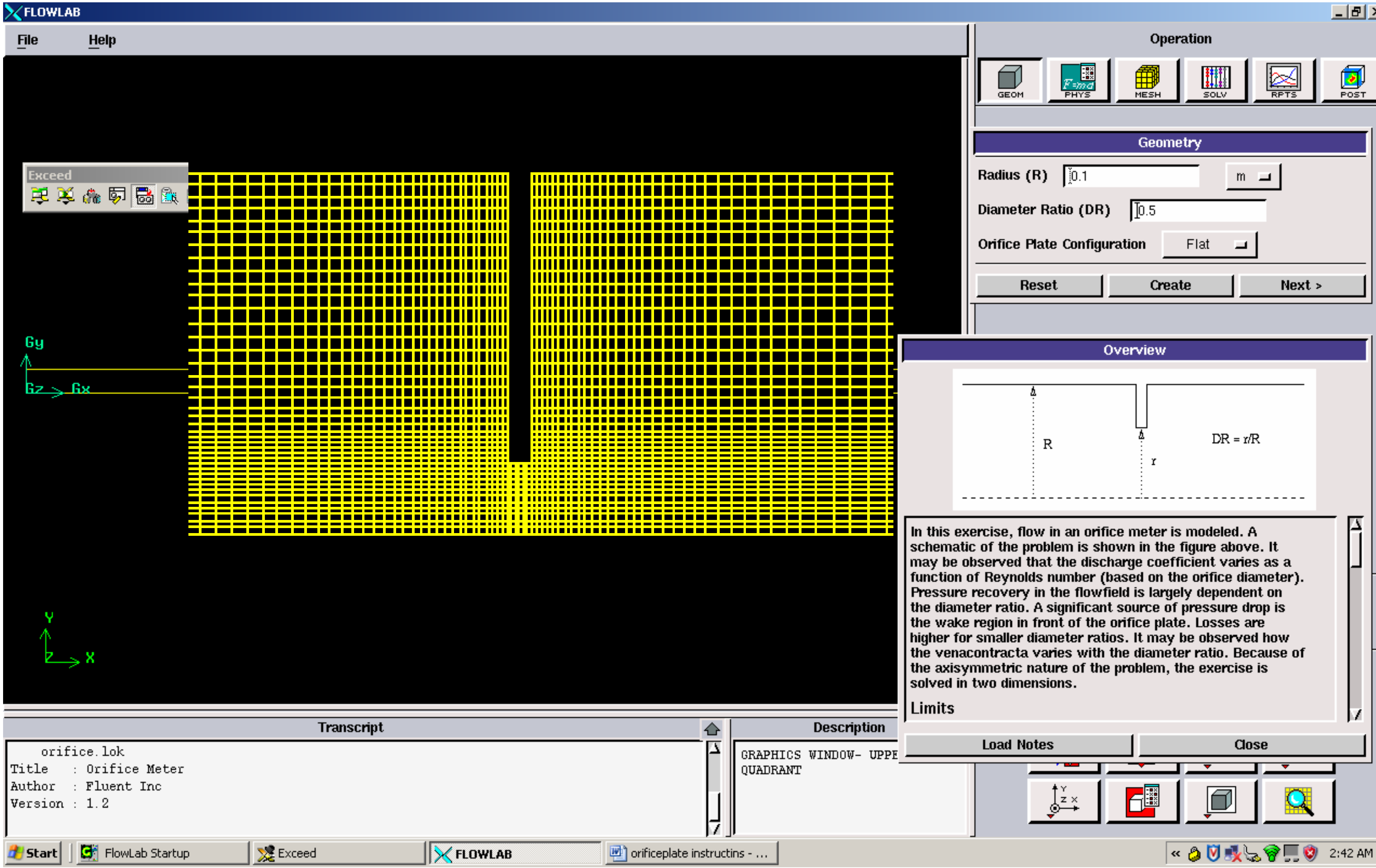
- Very user-friendly
- Minimal start-up time
  - Class time
  - Training for students
  - Training for faculty

CFD Process

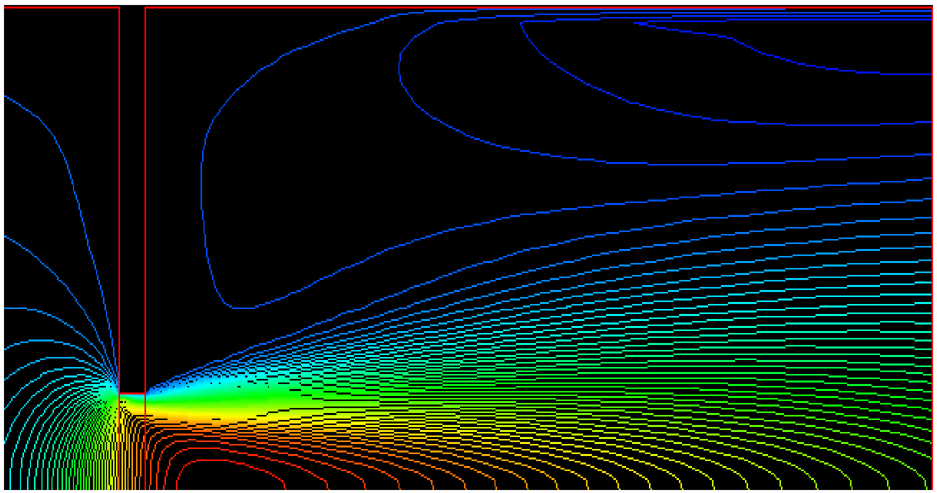


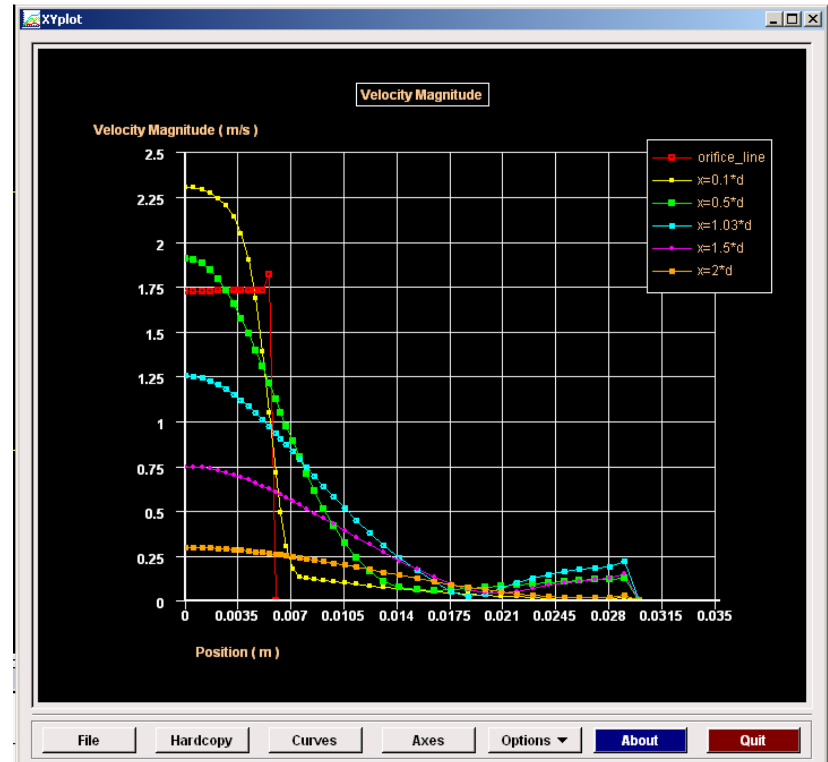
Geometry
Physics
Mesh
Solve
Reports
Post Processing

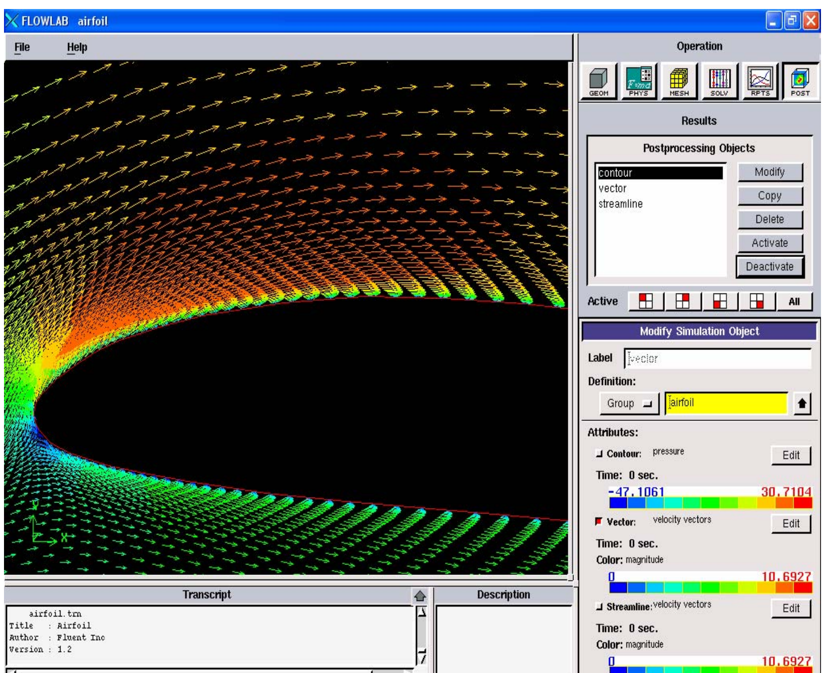
Interface


FLOWLAB
File
Help
Operation
GEOM
PHYS
MESH
SOLV
RPTS
POST
Geometry
Radius (R)
0.1
m
Diameter Ratio (DR)
0.5
Orifice Plate Configuration
Flat
Reset
Create
Next >
Exceed
Gy
Gz
Gx
Y
X
Overview
R
r
DR = r/R
In this exercise, flow in an orifice meter is modeled. A schematic of the problem is shown in the figure above. It may be observed that the discharge coefficient varies as a function of Reynolds number (based on the orifice diameter). Pressure recovery in the flowfield is largely dependent on the diameter ratio. A significant source of pressure drop is the wake region in front of the orifice plate. Losses are higher for smaller diameter ratios. It may be observed how the venacontracta varies with the diameter ratio. Because of the axisymmetric nature of the problem, the exercise is solved in two dimensions.
Limits
Load Notes
Close
Transcript
orifice.lok
Title : Orifice Meter
Author : Fluent Inc
Version : 1.2
Description
GRAPHICS WINDOW- UPPE
QUADRANT
Start
FlowLab Startup
Exceed
FLOWLAB
orificeplate instructins - ...
2:42 AM

Examples of Post-Processing







- Contours
- Velocity Vectors
- Streamlines

- Export Data

CFD in an Undergraduate Transport Course

- One CFD lecture
  - Introduction to discretization of governing equations
  - CFD terminology
    - Residuals
    - Consistency – How accurately does numerical solution reflect solution to differential equation? Grid size, time step, accuracy of numerical method
    - Convergence – Are residuals decreasing with increasing number of iterations?
    - Convergence Limit – Residuals for all variables must be below this value
  - Discrepancies between CFD results, analytical results and empirical results
- CFD in-class demonstrations/visualizations
- Distribute CFD manuals for pipe flow and flow through an orifice
- CFD homework exercise each week which complement traditional lecture material - compare CFD solutions with theory, data, empirical correlations

FlowLab Templates

Laminar and Turbulent Flow
Internal and External Flow
With and Without Heat Transfer

- Flow through an Orifice
- Developing Flow in a Pipe
- Sudden Expansion in a Pipe
- Flow over a Heated Plate
- Flow over a Cylinder
- Conduction
  - Steady and Unsteady
  - Series and Parallel
- Airfoil

CFD Exercises

- Flow in Conduits
  - Laminar and Turbulent Flow
    - Development of the Velocity Profile
    - Development Length
    - Pressure Gradient
    - Vary Pipe Diameter, Fluid Viscosity, Inlet Velocity

CFD Exercises

- Flow in Conduits with Heat Transfer
  - Wall flux or wall temperature conditions
  - Vary Pr
  - Thermal entrance length
  - Developing Nu number

CFD Exercises

- Flow past immersed objects
  - Drag
  - Stagnation zones, wakes

- Flow in Sudden Expansions and Contractions
  - Pressure Drop, Loss Coefficients, Regions of Recirculation

Learning Outcomes

- Students knowledge of fundamental concepts in fluid mechanics is enhanced via CFD
- Students gain a basic understanding of the principles of CFD
- Students can analyze comparisons between CFD solutions and results of analytical solutions or empirical data
- Students gain an understanding of what problem-solving tools are available to address complex, real-world problems