Presented at the COMSOL Conference 2009 Boston

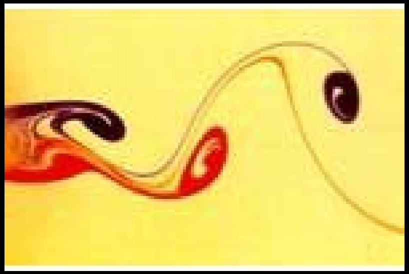
The Use of CFD Simulations in Learning Fluid Mechanics at the Undergraduate Level

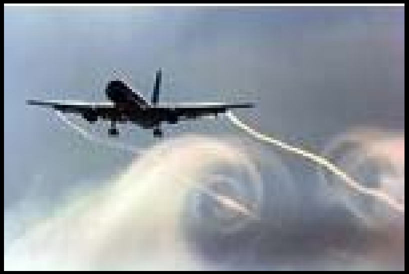
Marc K. Smith
Woodruff School of Mechanical Engineering
Georgia Institute of Technology
Atlanta, GA 30332-0405

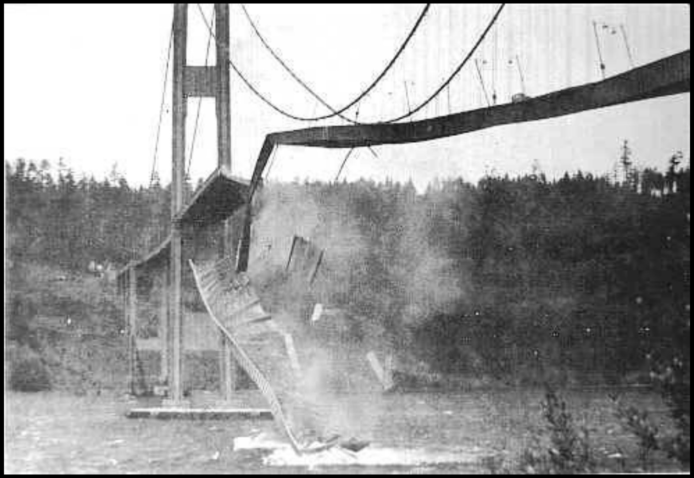
COMSOL Conference, Boston, Oct 8–10, 2009

Fluid Mechanics

- Fluid flows can be beautiful, amazing,







- and destructive.

Source: eFluids.com

A Traditional CFD Course

- Basic methods:
  - Finite difference, volume, or element.
  - Volume of fluid, level-set, phase field, etc.
- Gridding or meshing.
- Pressure-velocity coupling.
- Upwind differencing, artificial diffusion.
- Concentrate on numerical technique.
- Write your own code.

CFD for Fluid Exploration

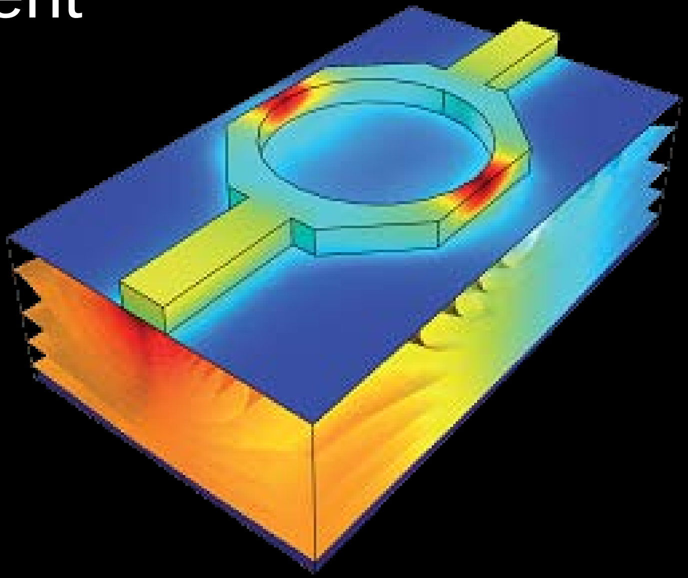
- Design a senior elective course that explores fluid flows using CFD as a tool.
- Use COMSOL Multiphysics as a PDE solver, but not a black box.
- Learn to properly pose the mathematical model.
- Learn to find and assess an accurate and reasonable solution.
- Explore the difference between modeling and reality.
- Develop a feel or intuition for the physics.

Course Content

- Three credit-hour semester course.
- Two lectures (as needed) & one lab per wk.
- FEM lectures (two weeks).
  - One programming assignment on FEM.
- COMSOL tutorial (one week).
- Seven labs on different fluid flows (1 or 2 weeks each).
- One final project (one month).

Laboratory Manuals

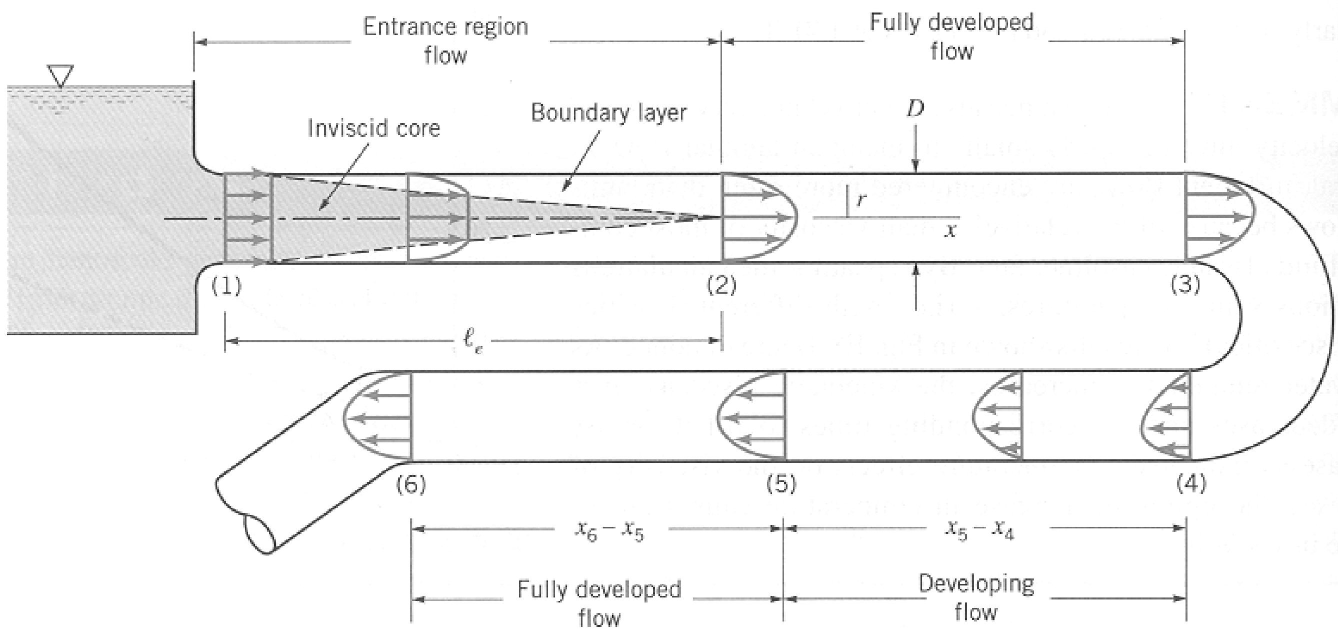
- Introduction and use of COMSOL software.
  - Lab 0: Steady electric current and heat generation in an aluminum film on a silicon substrate.
    - Thank you, COMSOL!



- All labs written in a tutorial form with decreasing level of detail from Labs 1 to 7.

Developing Flow in a Channel

- Commonly studied in most introductory fluid mechanics courses.



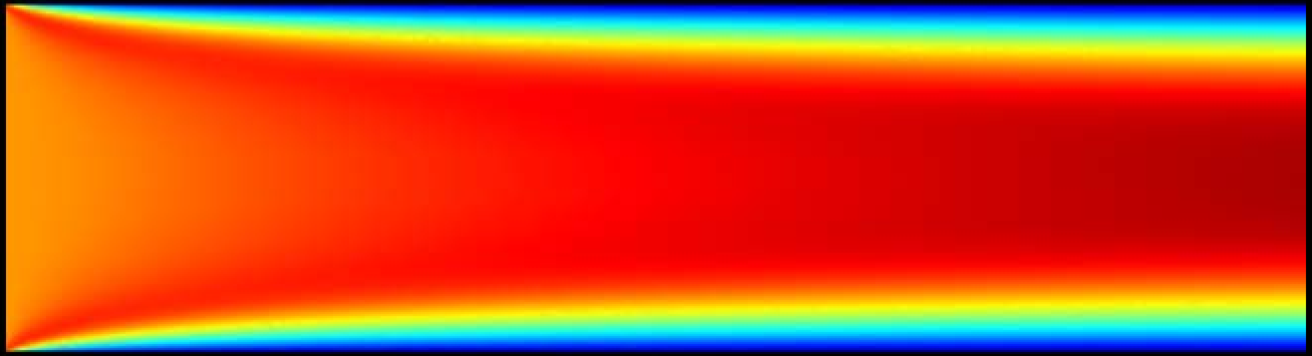
■ FIGURE 8.5 Entrance region, developing flow, and fully developed flow in a pipe system.

"Fundamentals of Fluid Mechanics," Munson, Young, Okiishi, and Huebsch, 6th ed.

Developing Flow in a Channel – 2

- Examine developing and fully developed velocity profiles.
- Measure entrance length: $\frac{L_E}{h} = f\left(\frac{\rho U_0 h}{\mu}\right) = 0.06\,Re$
- Find inviscid core flow with Bernoulli's eq.
- Compare to theory and experiment.

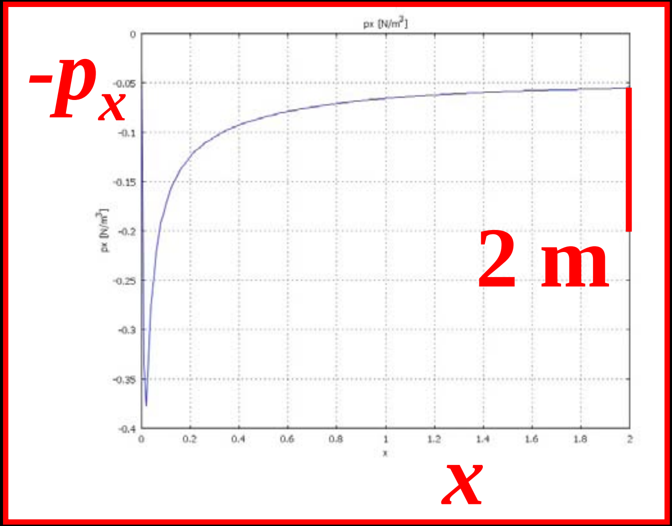
Surface plot of velocity magnitude at channel entrance.



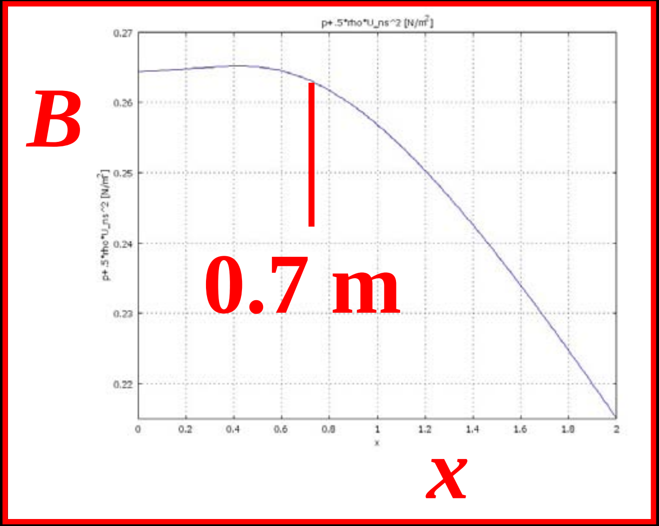
Developing Flow in a Channel – 3

- Student results: U_{avg} = 0.4 m/s, h = 4 cm

Pressure Gradient



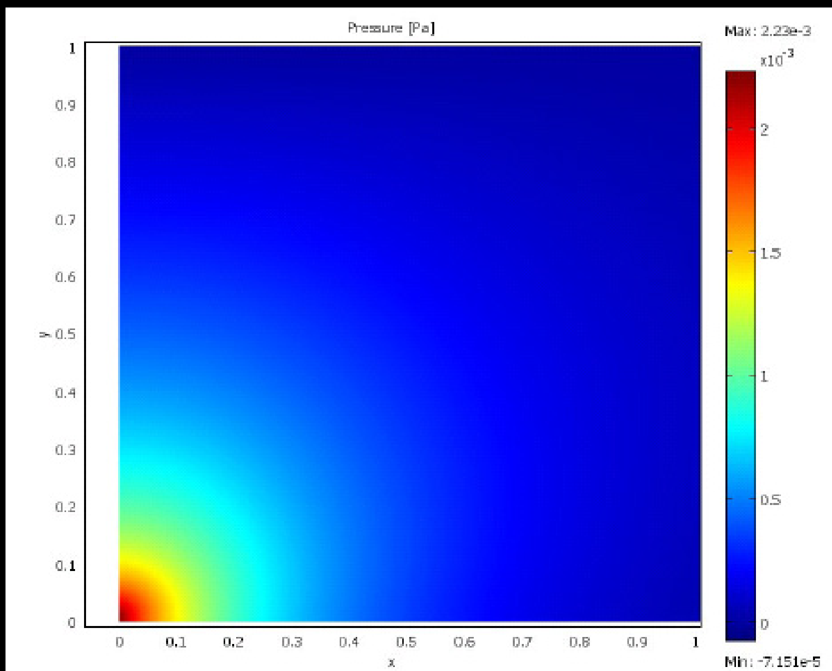
Bernoulli's Equation



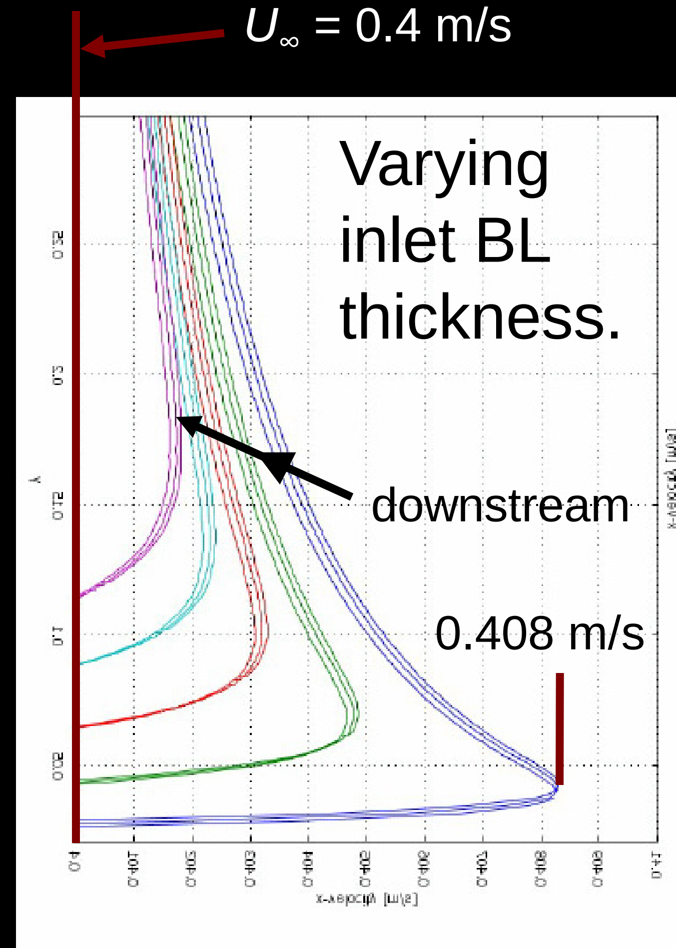
$$B = p + \tfrac{1}{2}\rho V^2$$

Developing Boundary Layer

Uniform flow past
a sharp-edged flat plate.



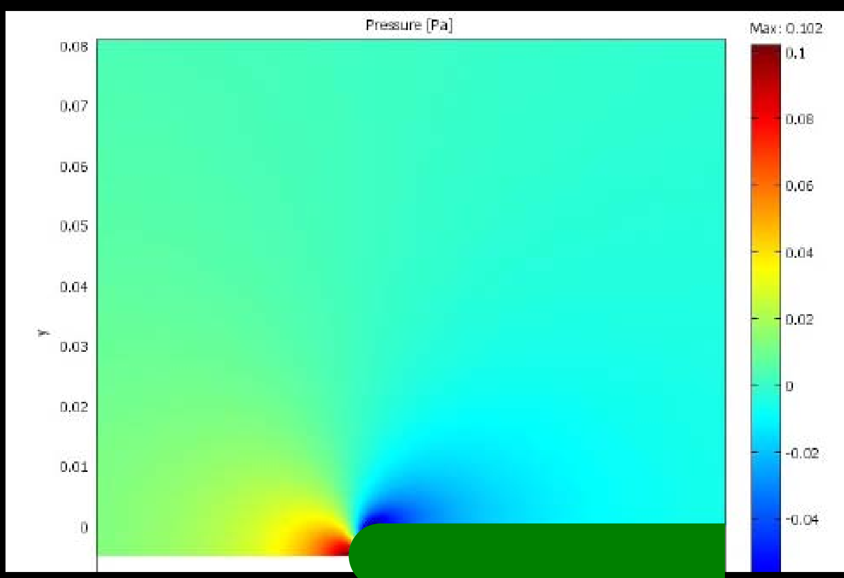
Pressure near leading edge



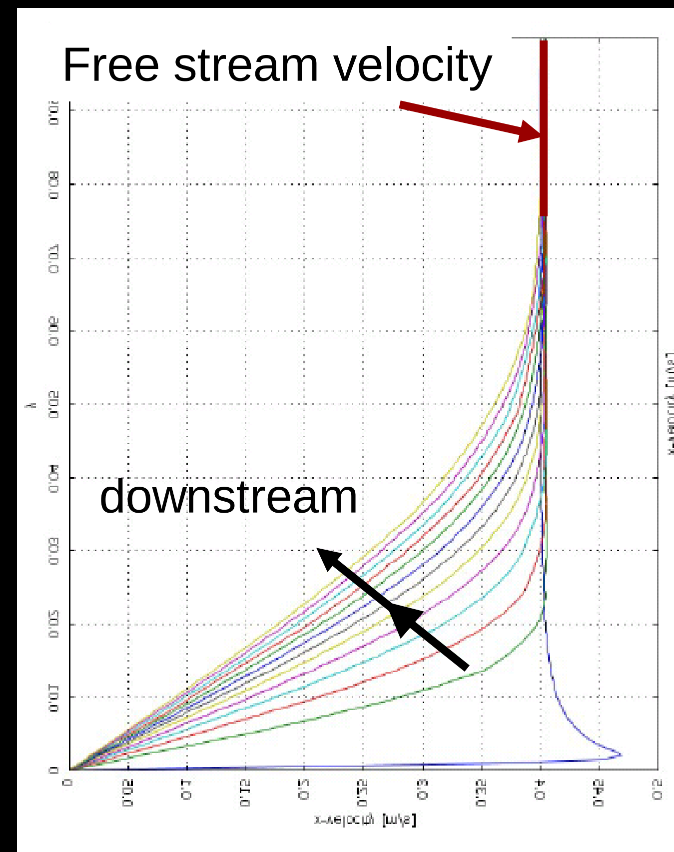
Horizontal velocity

Developing Boundary Layer – 2

Uniform flow past a round-edged flat plate.



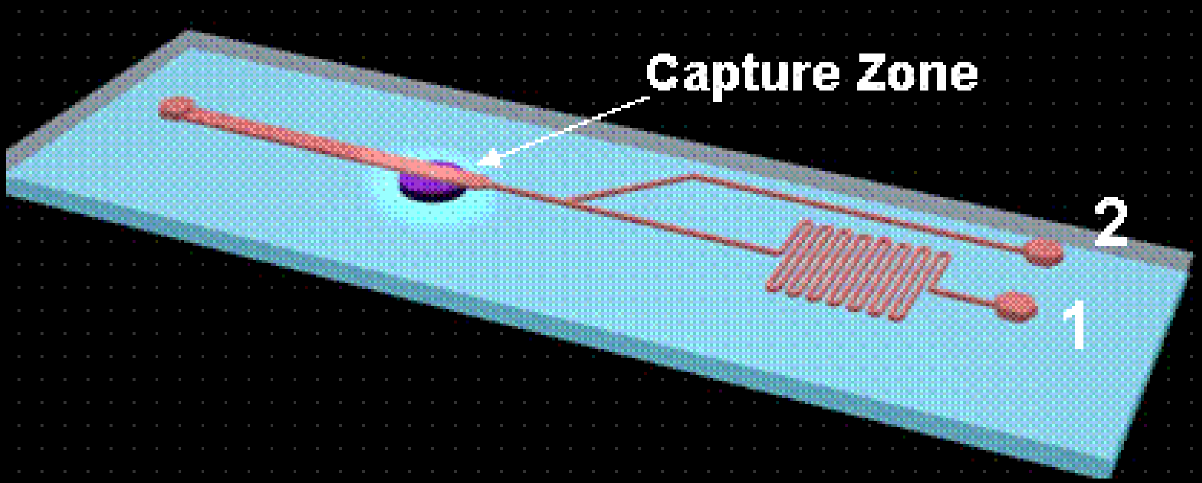
Pressure near leading edge



Horizontal velocity

Microfluidics

- Current technology of great interest.



A microfluidic fluorescence-based biosensor from IBI, Inc.
http://www.ibi.cc/microfluidics.htm

Tea Cup Experiment

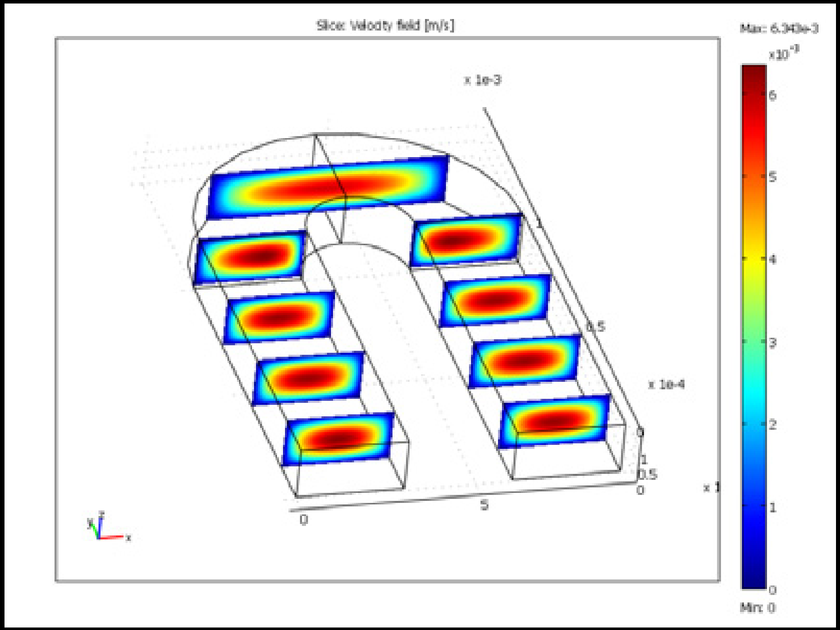
- Simple classroom demonstration.

Why does this happen?

Stirred by Hilda Elizabeth Tippette Nix

Microfluidics – Low *Re*

- Flow of water in a 180° bend.
- Cross-section: 300 μm x 150 μm.
- Uniform inlet: 0.007 m/s, ***Re*** = 0.94.

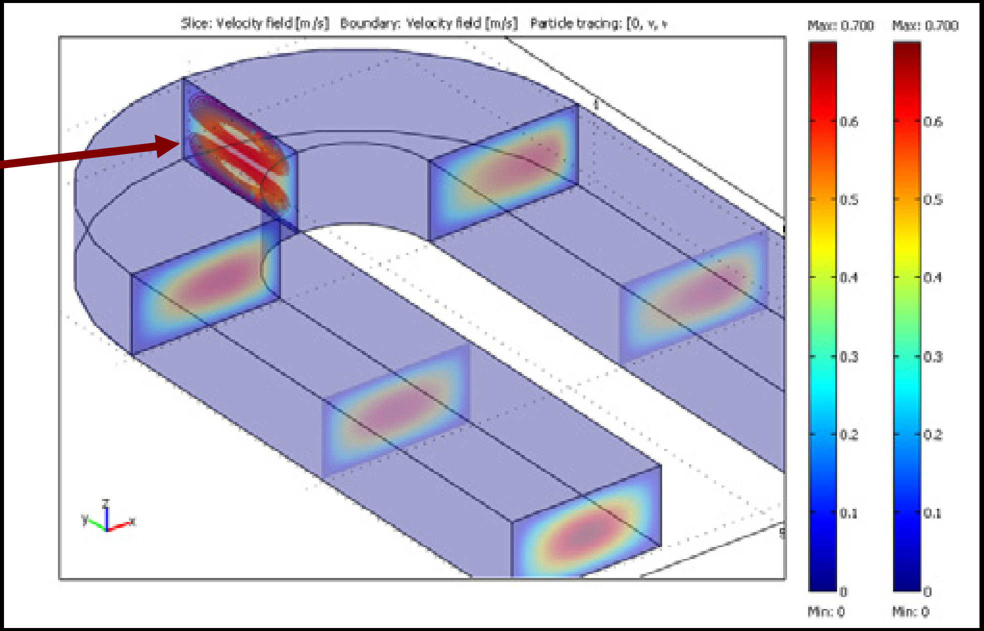


Velocity field

Microfluidics – Secondary Flow

- Flow of water in a 180° bend.
- Cross-section: 300 μm x 150 μm.
- Uniform inlet: 0.7 m/s, $\boldsymbol{Re}$ = 94.

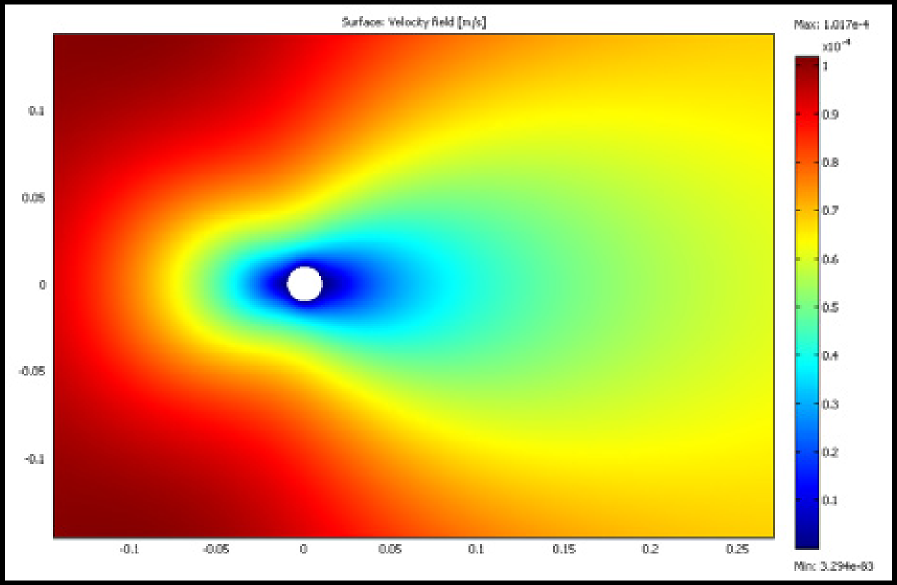
Particle traces in the plane



Velocity field

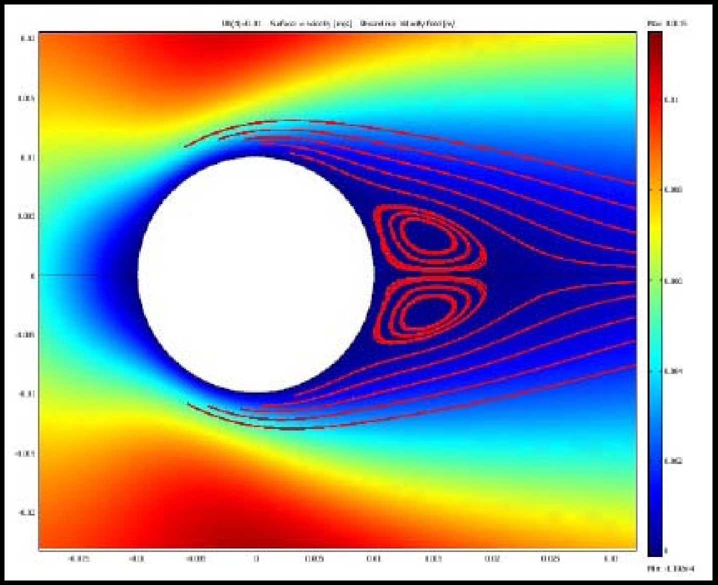
Flow Past a Cylinder

$Re = 1.37$



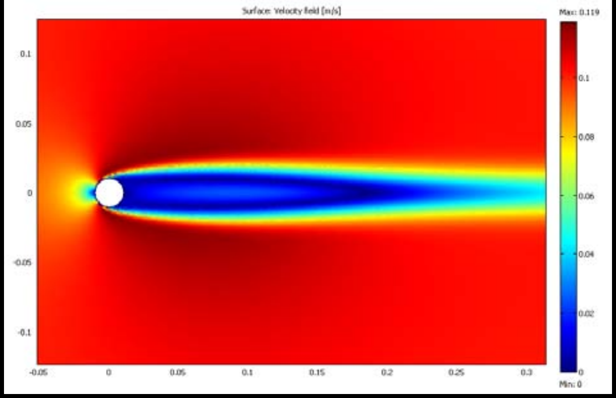
Bound vortices

$Re = 13.7$

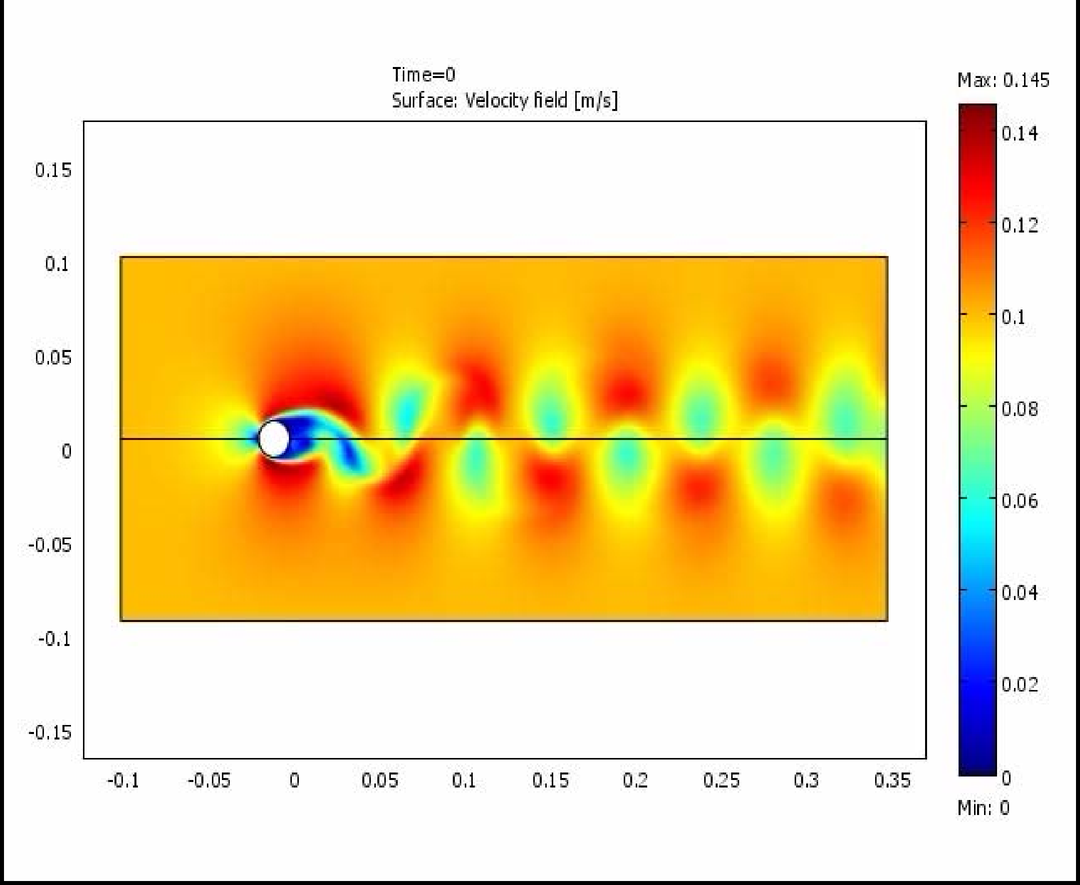


Flow Past a Cylinder – 2

Steady
Re = 137

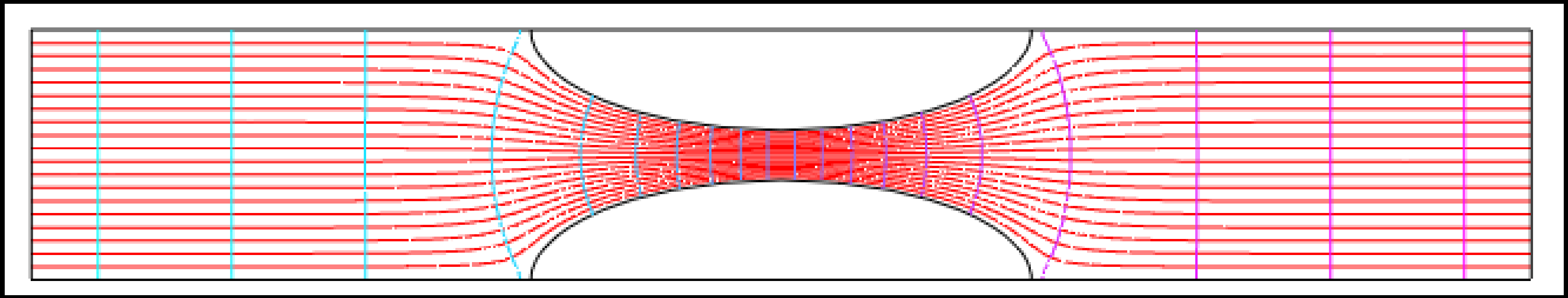


Unsteady, *Re* = 137



Potential Flow

- Potential flow in a Venturi.



Contours: potential Streamlines: velocity

- Compare flow rate to 1-D theory.
- Measure pressure at throat and upstream using point integration coupling variables.

Potential Flow – 2

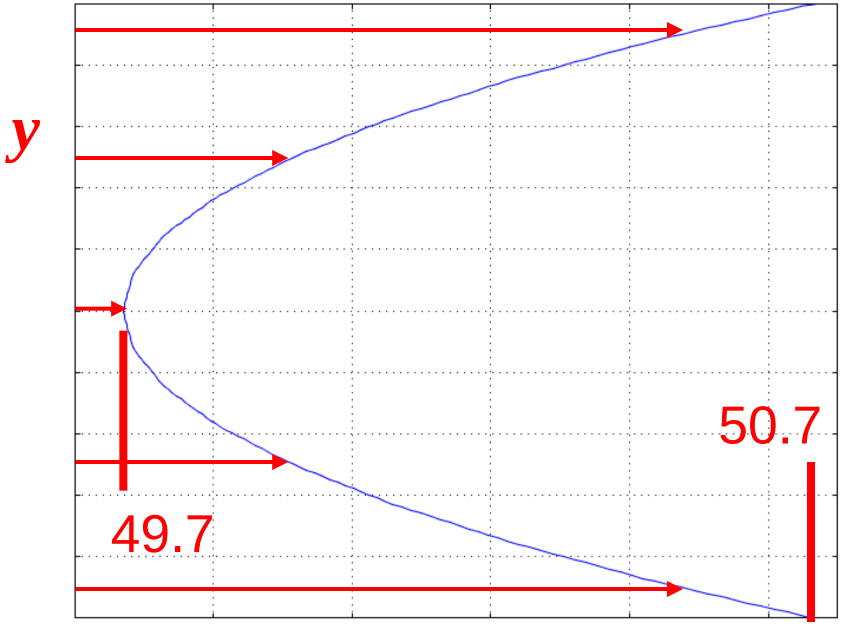
- Compare the flow rate Q to inviscid 1-D flow theory.

$$Q = \left(\frac{2\Delta p / \rho}{D_u^2 - D_t^2} \right)^{1/2} D_u D_t$$

- The relative error in Q is 1.4% too high.
- Why?
- Answer: *It's a 1-D modeling error!*

Potential Flow – 3

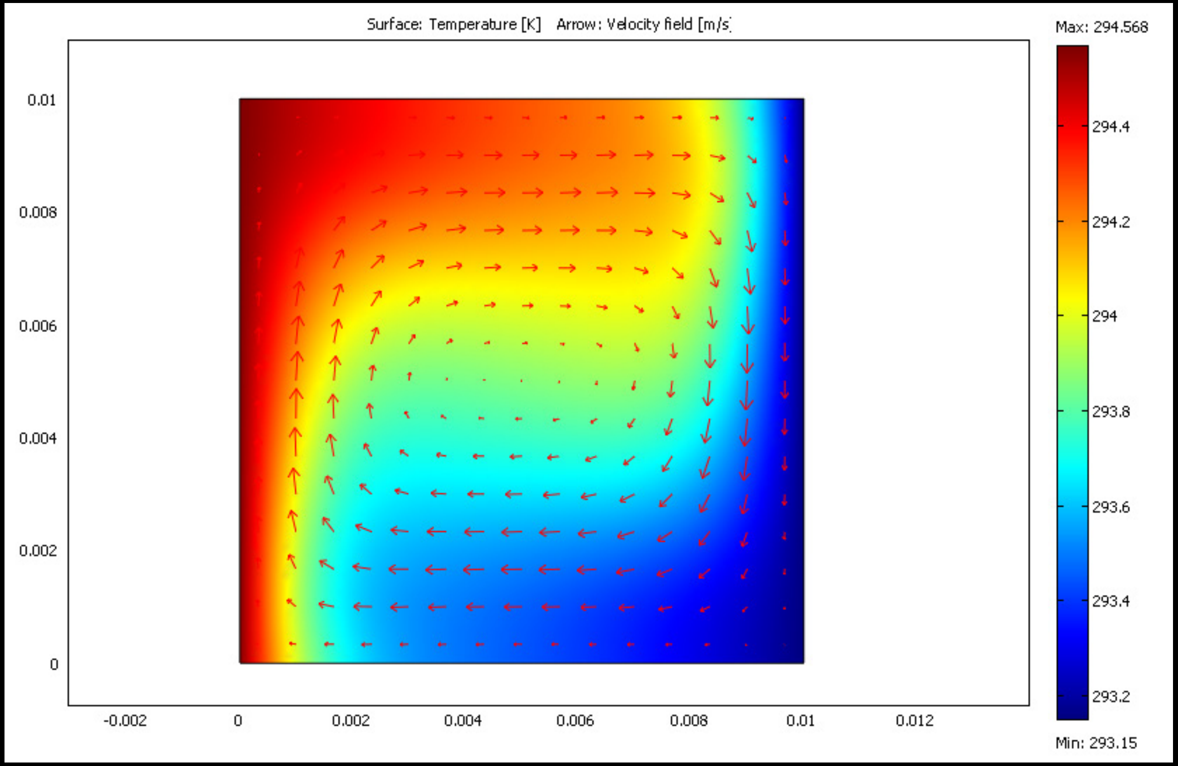
- Why? — The flow is not one-dimensional.



Horizontal velocity at throat

Thermal Convection

- Side heating: Boussinesq approximation and temperature-dependent properties.



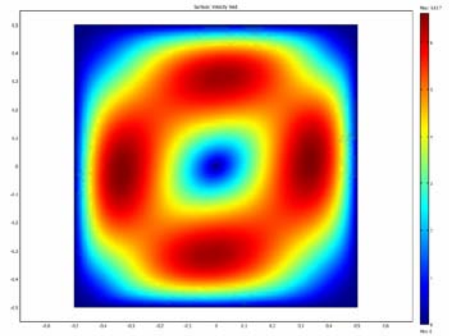
Color: temperature, Arrows: velocity

Thermal Convection – 2

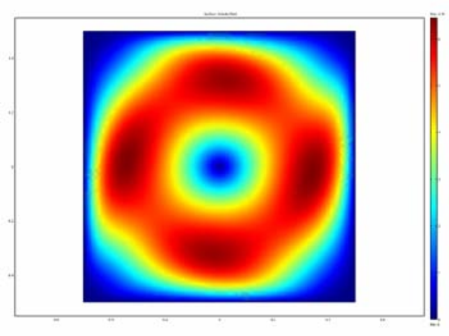
- Effect of Prandtl number

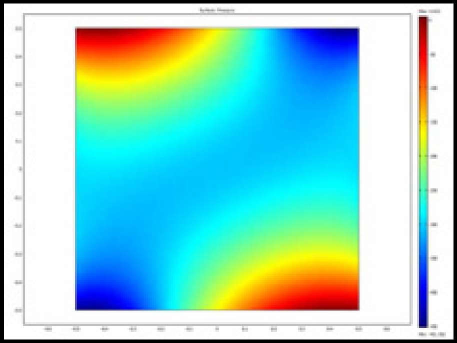
$Ra = 2000, \quad Pr = 7$

$Ra = 2000, \quad Pr = 0.1$

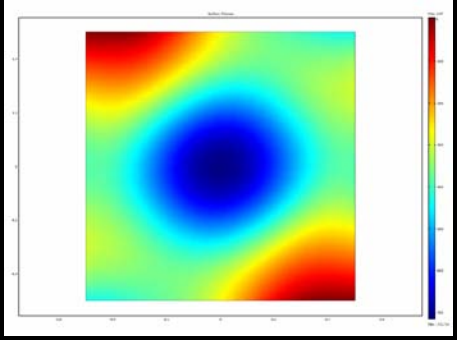


Velocity magnitude





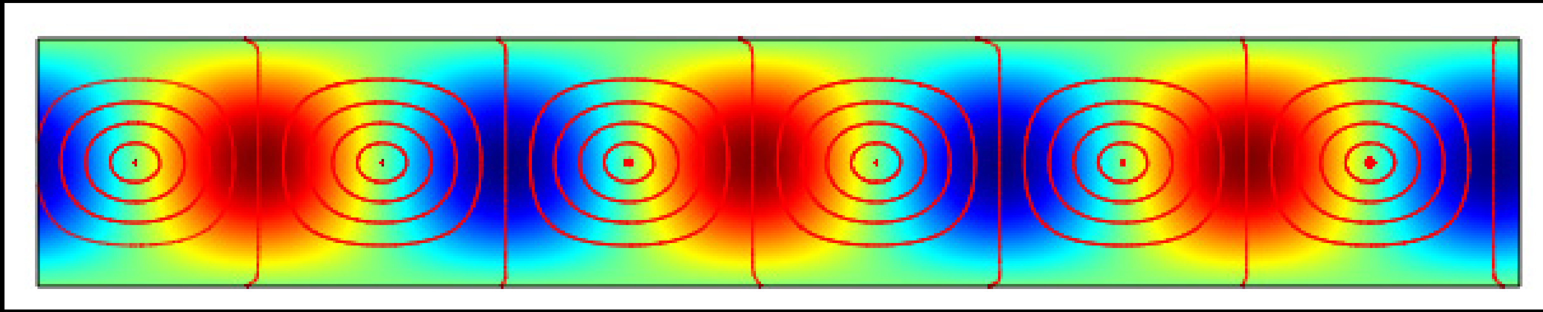
Pressure



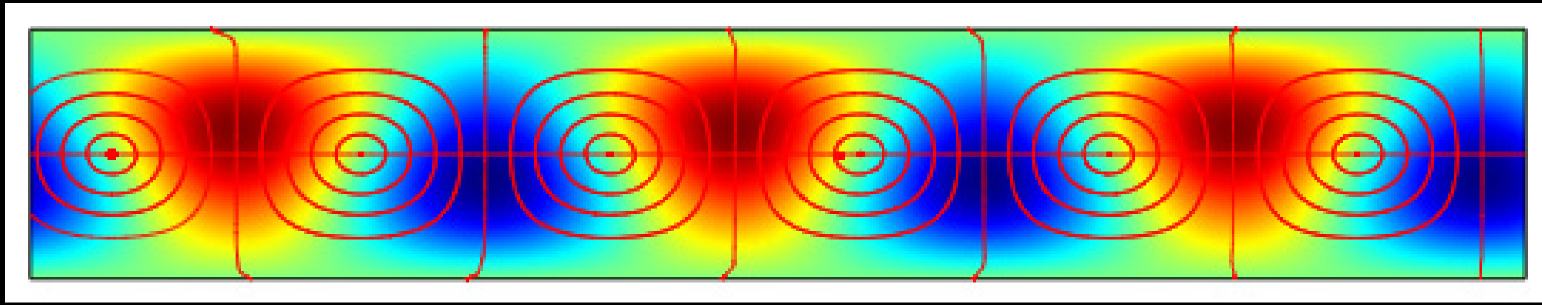
Thermal Convection – 3

- Liquid layer heated from below.

$Ra = 1709$



$Ra = 2000$



Color: temperature,   Streamlines: velocity

Conclusions

- Students: Learned to investigate, interpret, and understand fluid flows. They became very competent with COMSOL.
- Professor: Lots of work, lots of fun.
- Needs: Better or easier tools.
  - Solution quality: element quality, residuals, global mass, force, and energy balances.
  - Particle-tracing tool.
  - Projected streamlines in a plane for 3-D flows.