FLUENT in Intermediate Fluid Dynamics

Rajesh Bhaskaran

Sibley School of Mechanical & Aersopace Engineering
Cornell University

MOTIVATION

- Computer-based simulation has become an integral part of engineering practice of fluid dynamics.

  $\Rightarrow$ The intelligent and proper use of CFD tools is an important engineering skill to be imparted at the undergraduate level.

- Scientific knowledge is encapsulated in CFD tools.

  $\Rightarrow$ Easy-to-use CFD software offers the opportunity to improve and enrich students' understanding of fundamental concepts.

OBJECTIVES

FLUENT introduced in *M&AE 423/523 Intermediate Fluid Dynamics* in Spring '02. Taught by Prof. Lance Collins.

- Expose students to the intelligent use of an industry-standard CFD tool.

- Use FLUENT as a platform for students to explore flow physics and to reinforce fundamental concepts.

- Use hands-on simulations and a rich visual medium to stimulate student interest and facilitate learning.

FLUENT VS. FLOWLAB

- FLOWLAB: Wrapper around FLUENT which makes it easier for students to run simulations by limiting available options.

- Considered using FLOWLAB in *M&AE 423* but decided FLUENT was more appropriate.
  - More useful for students to be exposed to a full-fledged CFD package at this level.
  - FLOWLAB results were not validated adequately at that point.

Provided the geometry and grid for running FLUENT.

FLOWLAB is more appropriate in an introductory-level class and/or lab.

FLUENT EXAMPLES

The classroom examples used were:

1. Laminar and turbulent flow in a circular pipe

2. Compressible flow in an axisymmetric nozzle

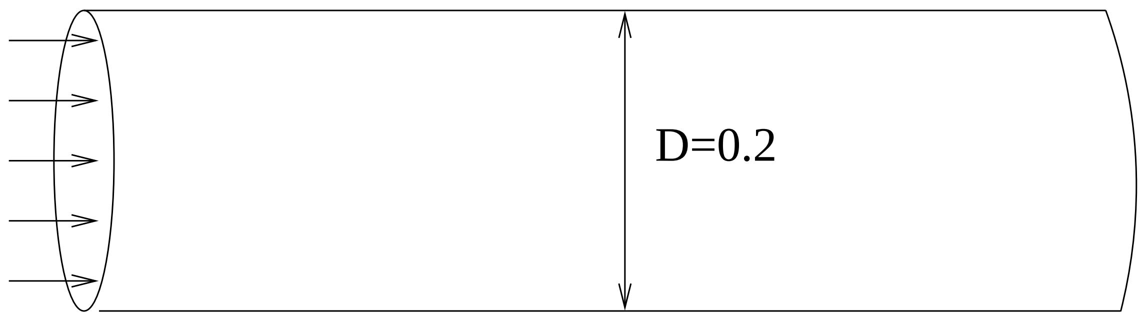
3. 2D flow past an airfoil

Possible additions for next year:

- Jet

- Flow over cylinder or sphere

LAMINAR PIPE FLOW

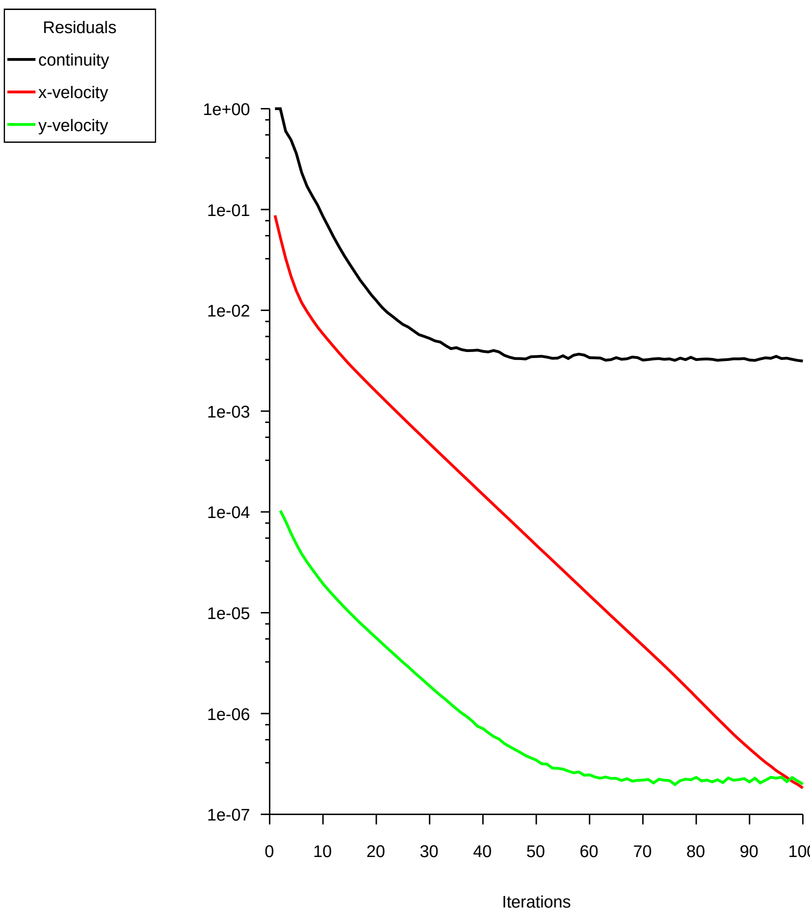


Incompressible, developing flow in a circular pipe.

Inlet velocity = 1 m/s   Exit pressure = 1 atm

$Re_D = \frac{\rho V D}{\mu}$=100 or 200

LAMINAR PIPE FLOW

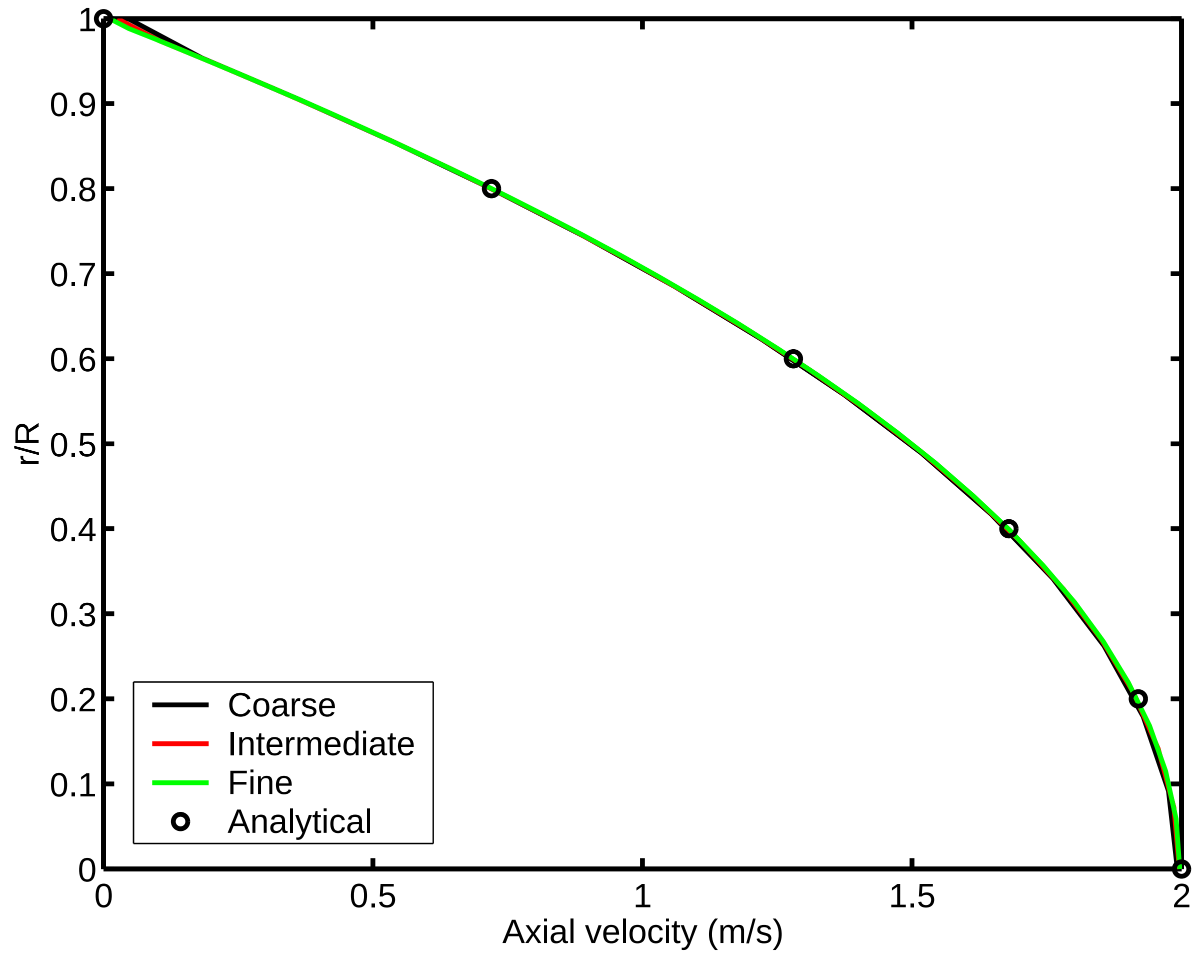


Scaled Residuals

May 30, 2002

FLUENT 6.0 (axi, segregated, lam)

LAMINAR PIPE FLOW



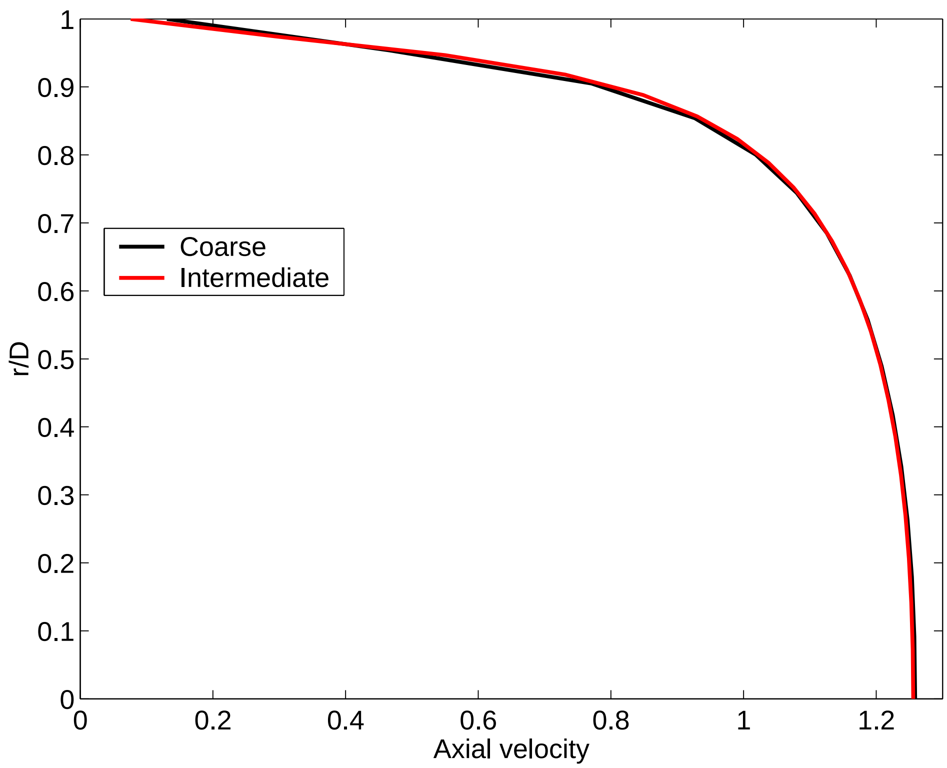
C_f at exit is same as analytical value.

Qualitative discussion of development length

TURBULENT PIPE FLOW

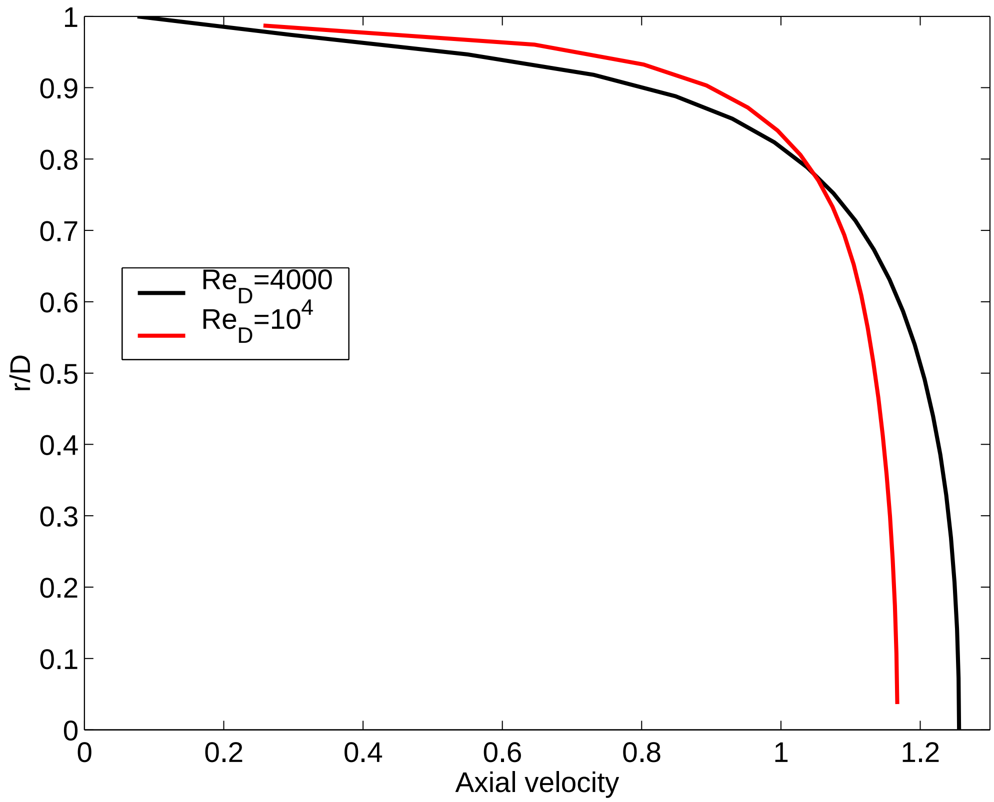
The Reynolds number is changed to 4,000 and 10,000 by changing μ.

Use k-ϵ turbulence model (Standard, RNG and Realizable).



Std. k-ϵ, Intermediate grid, Re_D=4000:
u_{max}=1.26 White u_{max}=1.27 Diff=-0.8%
C_f=0.012 White C_f=0.010 Diff=20%

TURBULENT PIPE FLOW

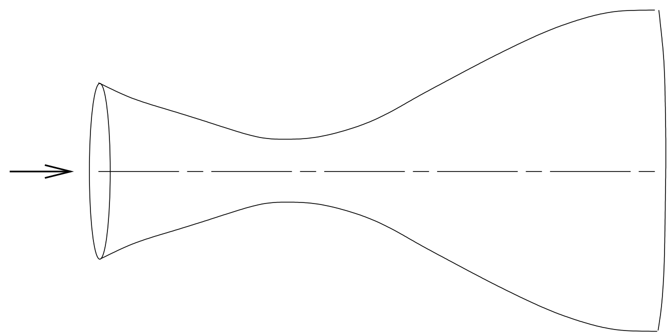


For Re_D=10,000:

u_{max}	White u_{max}	Diff.
1.17	1.23	-4.9%

C_f	White C_f	Diff.
0.0097	0.0077	26%

NOZZLE FLOW



Compressible, inviscid flow through an axisymmetric nozzle

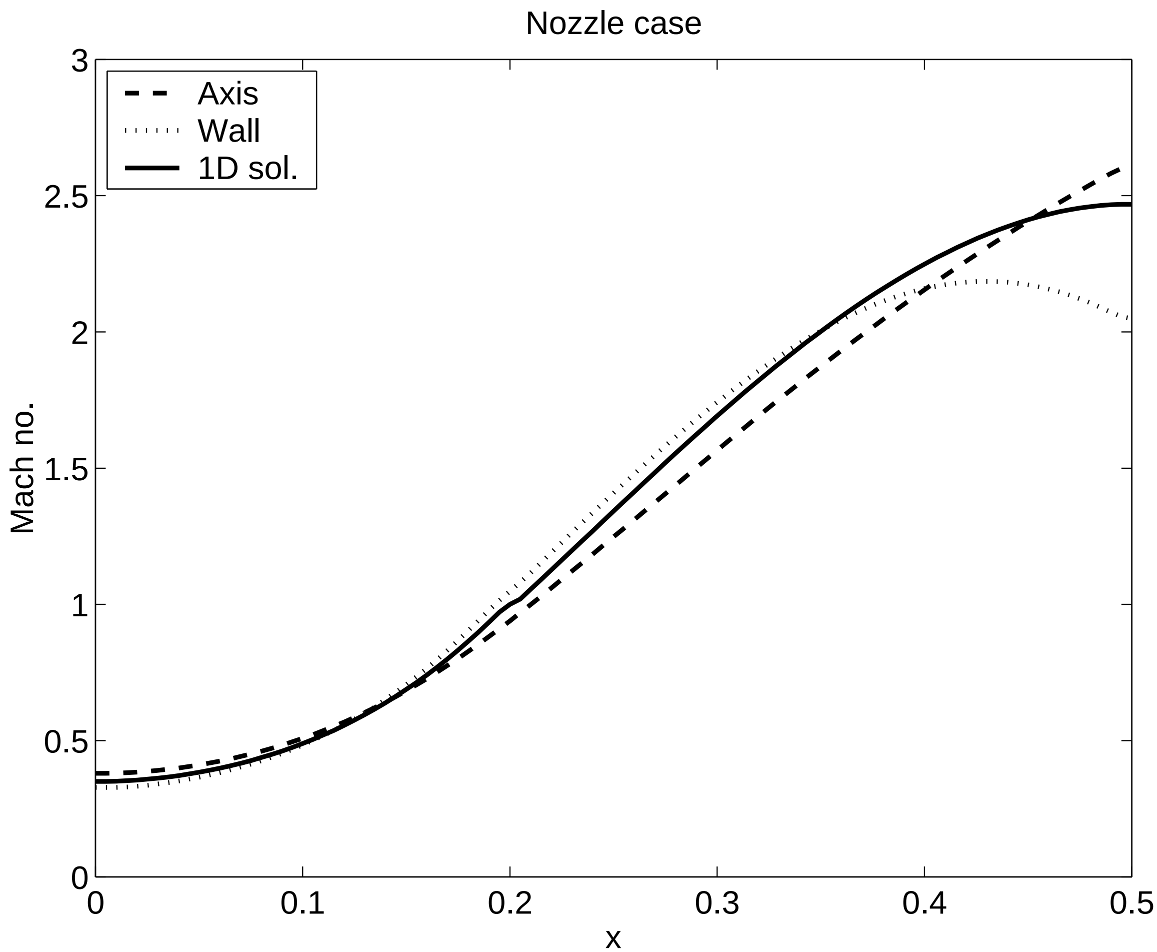
Calculation preceded by discussion of 1D nozzle flow analysis.

Going from incompressible, viscous to compressible, inviscid flow involves a few clicks of the mouse.

At the inlet, p_o=106,439 Pa T_o=300 K
At the exit, p=6,544.7 Pa

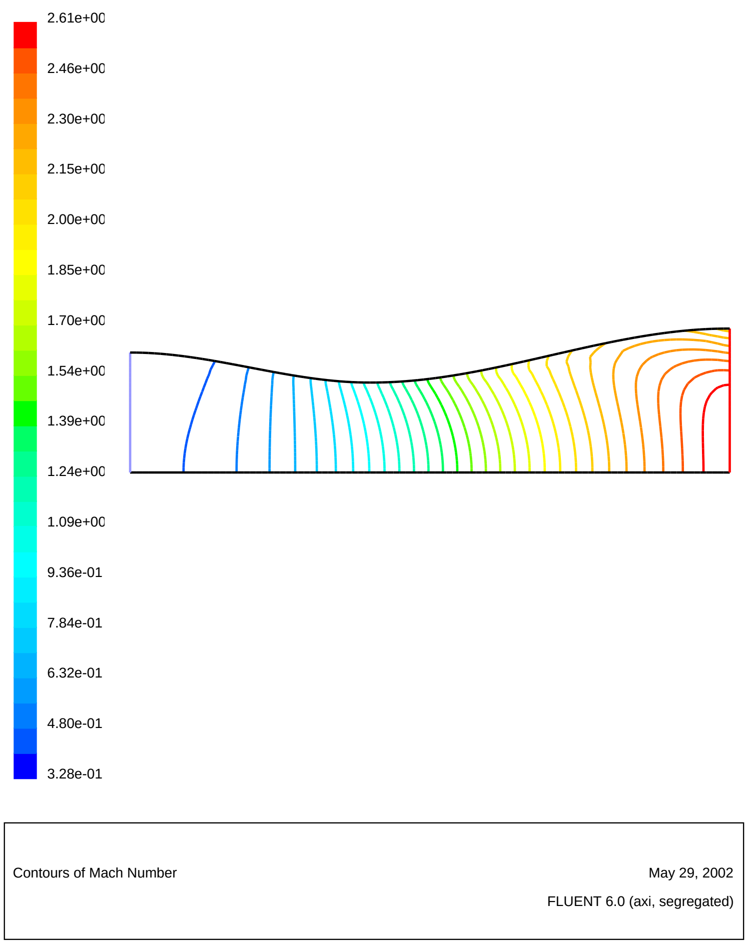
When material is changed to ideal gas, FLUENT automatically turns on the energy equation. Relates to an important physical principle.

NOZZLE FLOW



- Greatest deviation from 1D result: Wall region near the exit

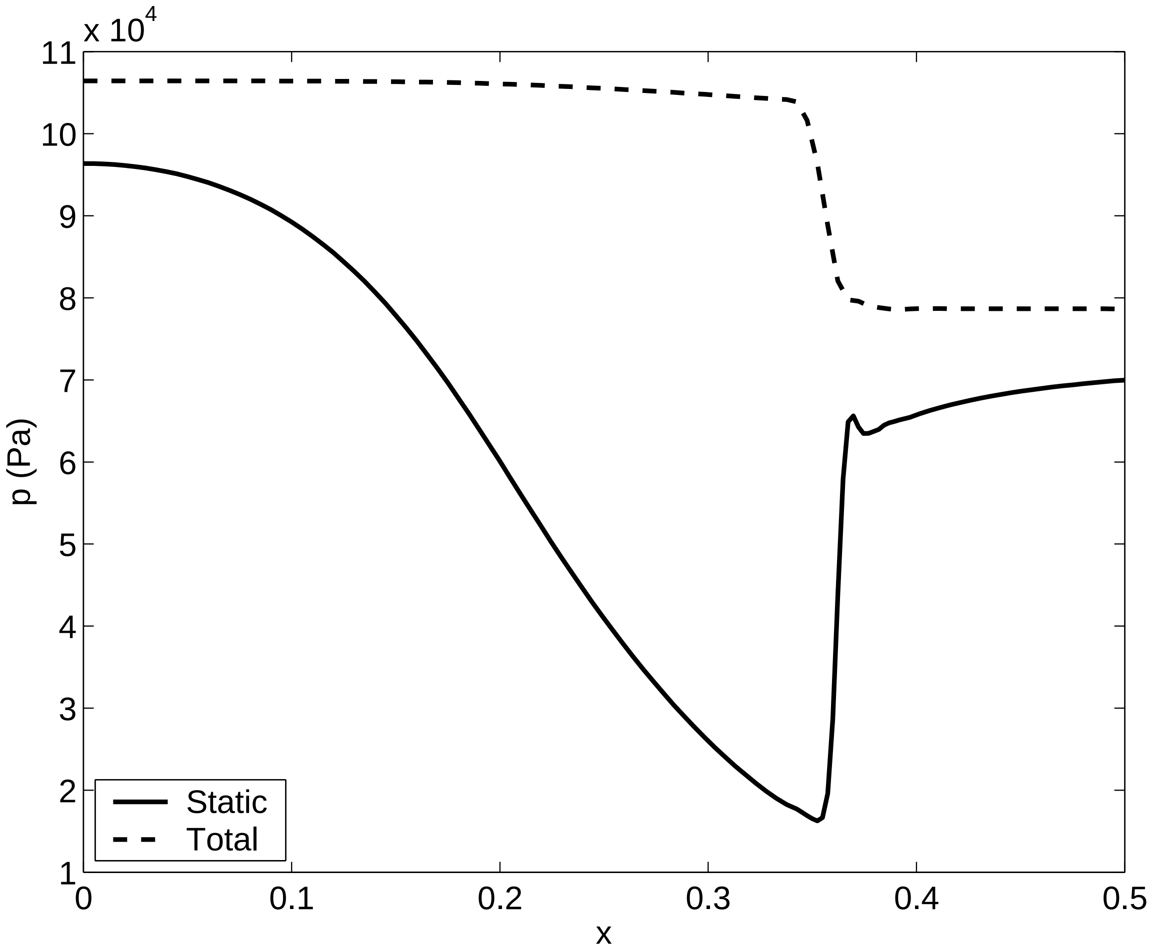
NOZZLE FLOW



- Flow is strongly 2D in the wall region near the exit.

NOZZLE FLOW

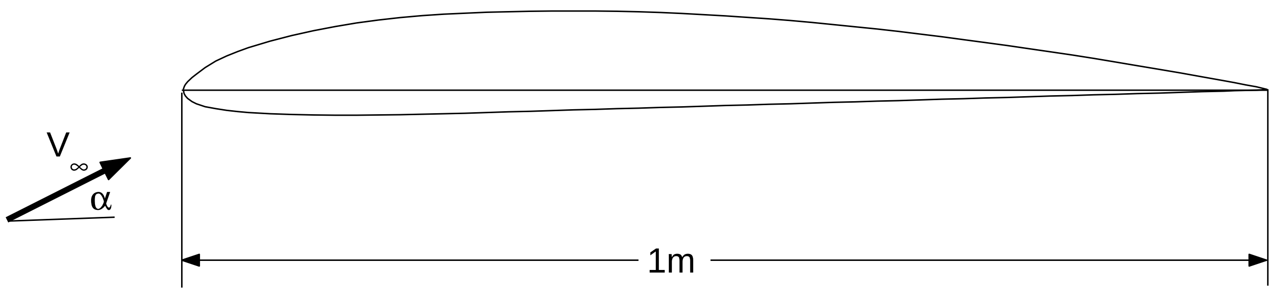
Exit pressure is changed from 6544 Pa to 70,000 Pa.



Shock wave in the diverging section.

Grid adaption is used to better resolve shock wave.

AIRFOIL FLOW



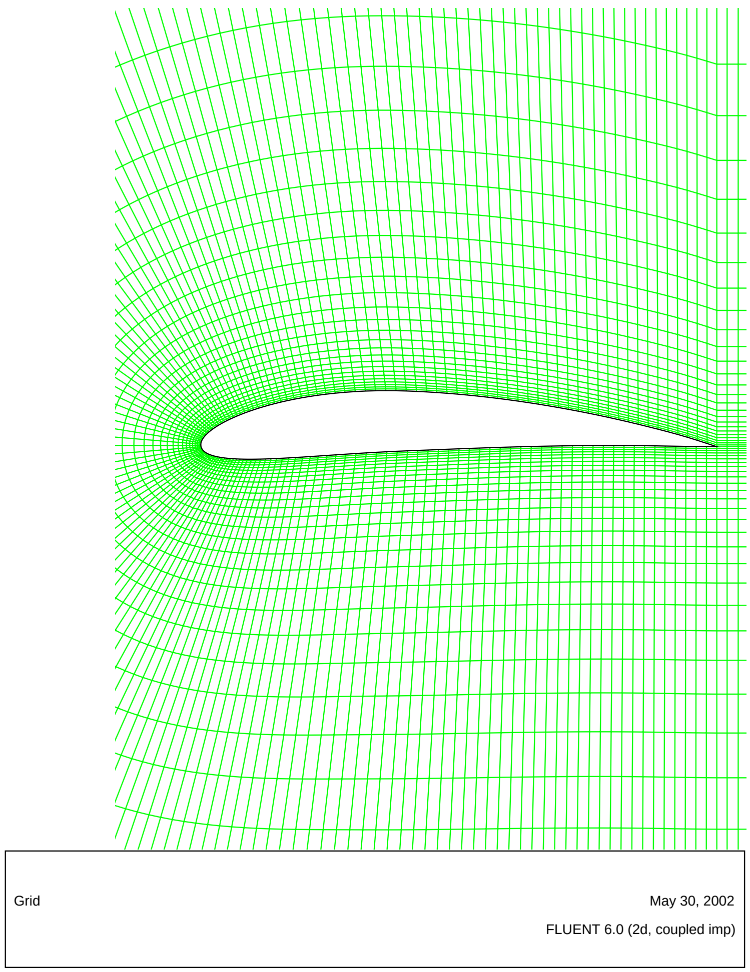
- Incompressible flow over an airfoil

- V_∞=50 m/s Re=3.4x10^6

- Consider both inviscid and viscous cases

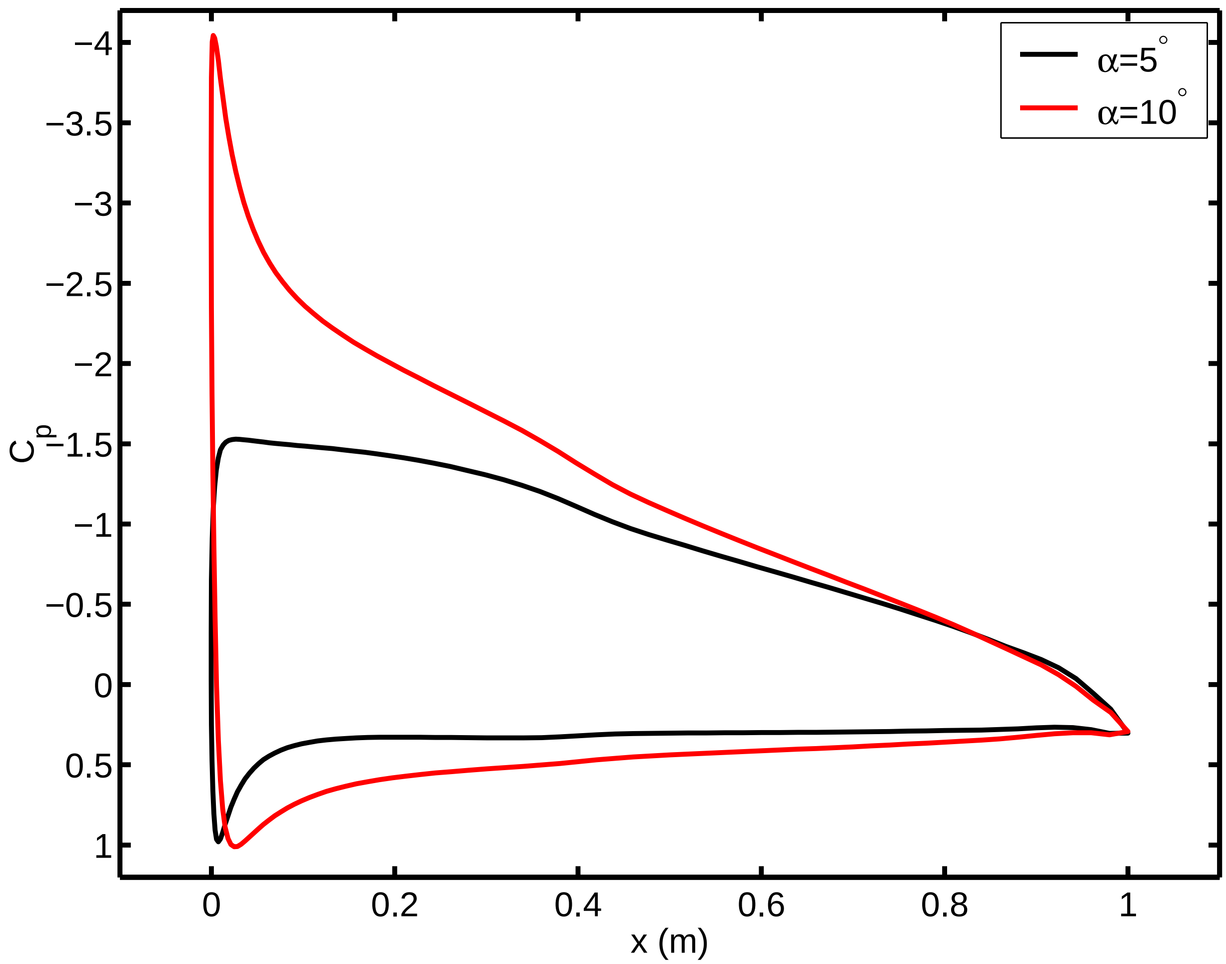
- Viscous case: Std. k-ϵ turbulence model

AIRFOIL FLOW



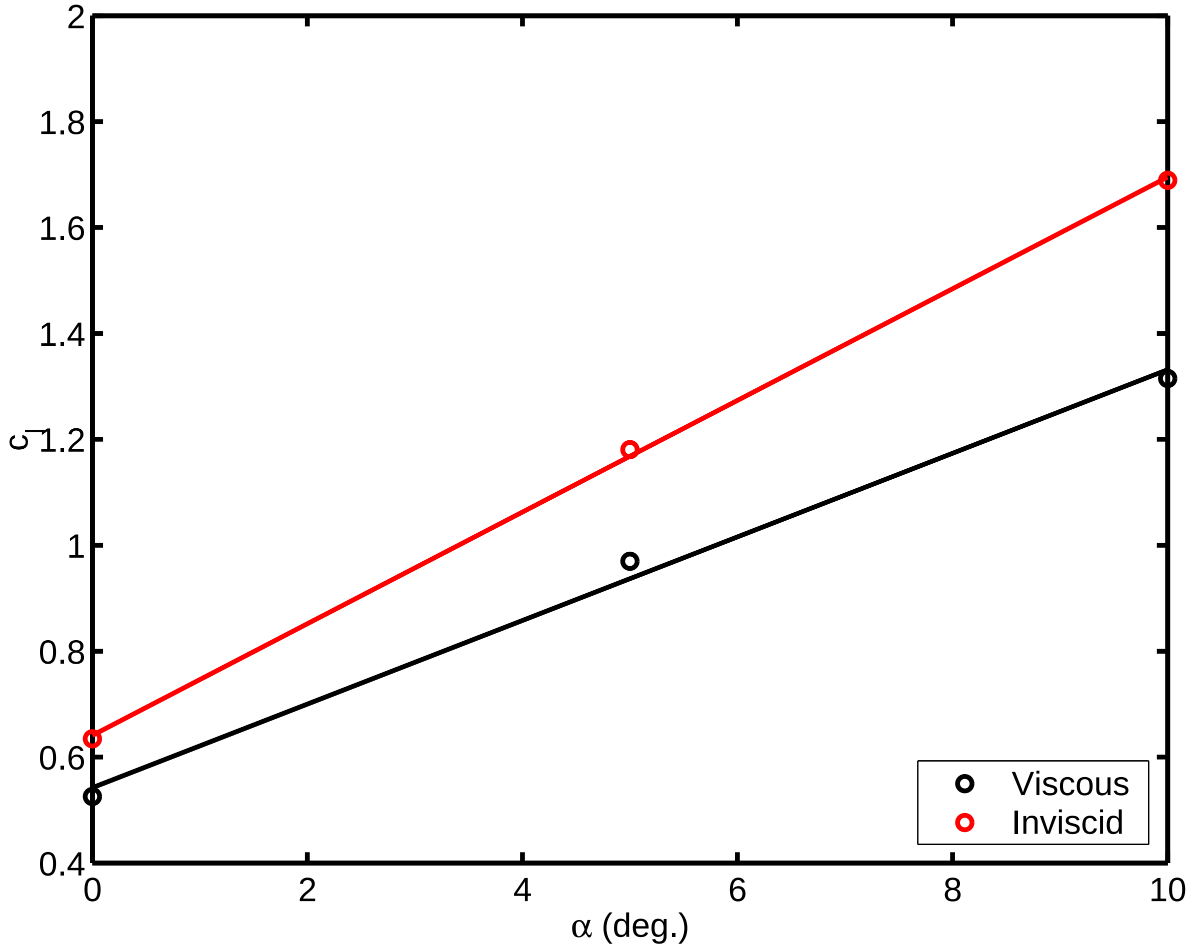
- C-grid with 12,195 cells

AIRFOIL FLOW



- $c_l \simeq$ Area under curve

AIRFOIL FLOW



- Inviscid case: $\dfrac{dc_l}{d\alpha} = 1.92\,\pi$ per radian

- Viscous case: $\dfrac{dc_l}{d\alpha} = 1.44\,\pi$ per radian

AIRFOIL FLOW

- Inviscid case:

α	c_l	c_d
10°	1.689	0.023

- Viscous case:

α	c_l	c_{d_p}	c_f	c_d
10°	1.315	0.066	9.169e-3	0.075

- These results are with the coupled solver
  Segregated solver gives c_d values that are about half of the coupled solver

- c_d values are too high
  Prof. Caughey's solver:
  c_l=1.86 c_d=-6x10^{-4} (α=10°, M=0.1)

CONCLUSION

- Need to look into improving the accuracy of FLUENT results.
  University support unable to fully address these concerns.
  Could use help from a senior Fluent engineer.

- Installation could be simplified.

- Exceed X-server is an additional complication.

- Inexpensive student version would be useful.

CONCLUSION

At the undergraduate level, FLUENT can be a valuable tool in teaching

- The intelligent use of CFD.

- The basic principles of fluid dynamics.

Appropriate curriculum material is needed. This effort is a step towards providing this.

Student project ideas solicited.

Positive buzz from students.

Availability of inexpensive, powerful computers and simplified software interfaces enables a new way to teach fluid dynamics.