Presented at the COMSOL Conference 2008 Boston

An Analysis of Hiemenz Flow

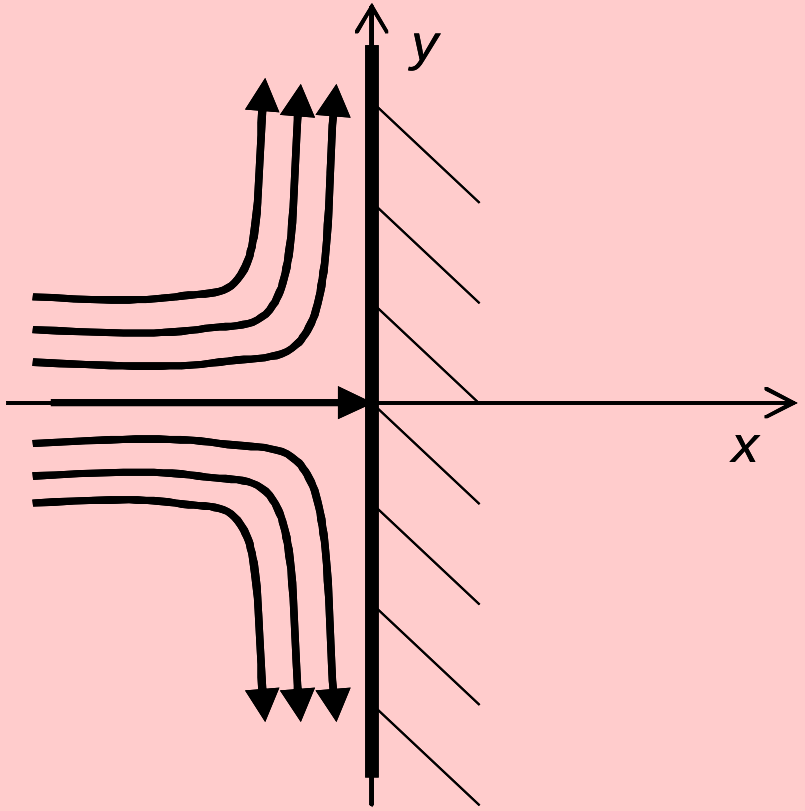
E. Kaufman and E. Gutierrez-Miravete
Department of Engineering and Science
Rensselaer at Hartford

OBJECTIVE

To solve a simple flow field for which an exact solution is available using COMSOL & FLUENT

HIEMENZ FLOW

- Planar
- Laminar
- Viscous
- Incompressible
- Close to a stagnation point



BACKGROUND

- Exact Solution Exists for Hiemenz Flow
- Viscous Solution is Derived from the Inviscid Solution

Inviscid

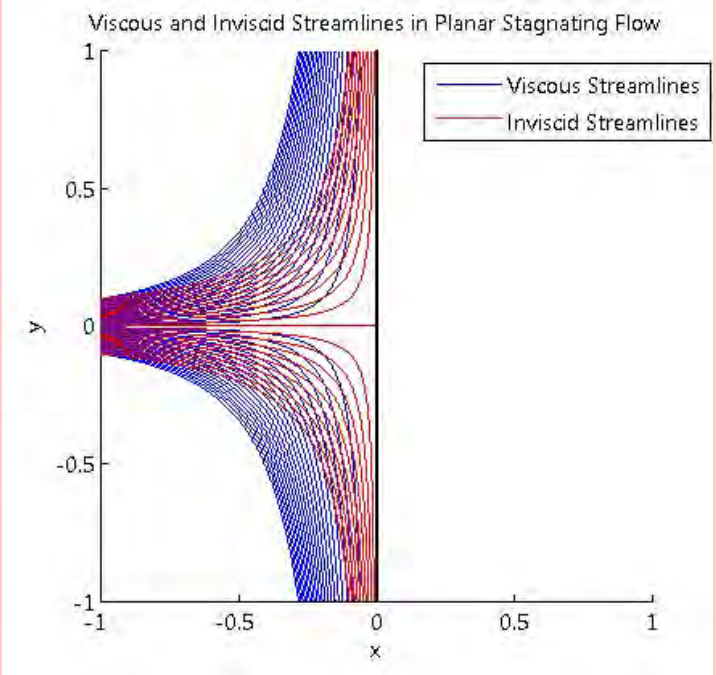
$$\vec{V} = -ax\hat{i} + ay\hat{j}$$

$$p_0 - p = \tfrac{1}{2}\rho a^2\left(x^2 + y^2\right)$$

Viscous

$$\vec{V} = -af(x)\hat{i} + ayf'(x)\hat{j}$$

$$p_0 - p = \tfrac{1}{2}\rho a^2\left[x^2 + F(y)\right]$$



HIEMENZ SOLUTION

- Substitute into 2D Incompressible Navier-Stokes Equations

$$\rho a^2 x\left(f'^2 - ff''\right) = \rho g_x - \frac{\partial p}{\partial x} + \mu a x f''' \qquad \rho a^2 f f' = \rho g_y - \frac{\partial p}{\partial y} - \mu a f''$$

- Similarity Solution Yields Hiemenz Equation

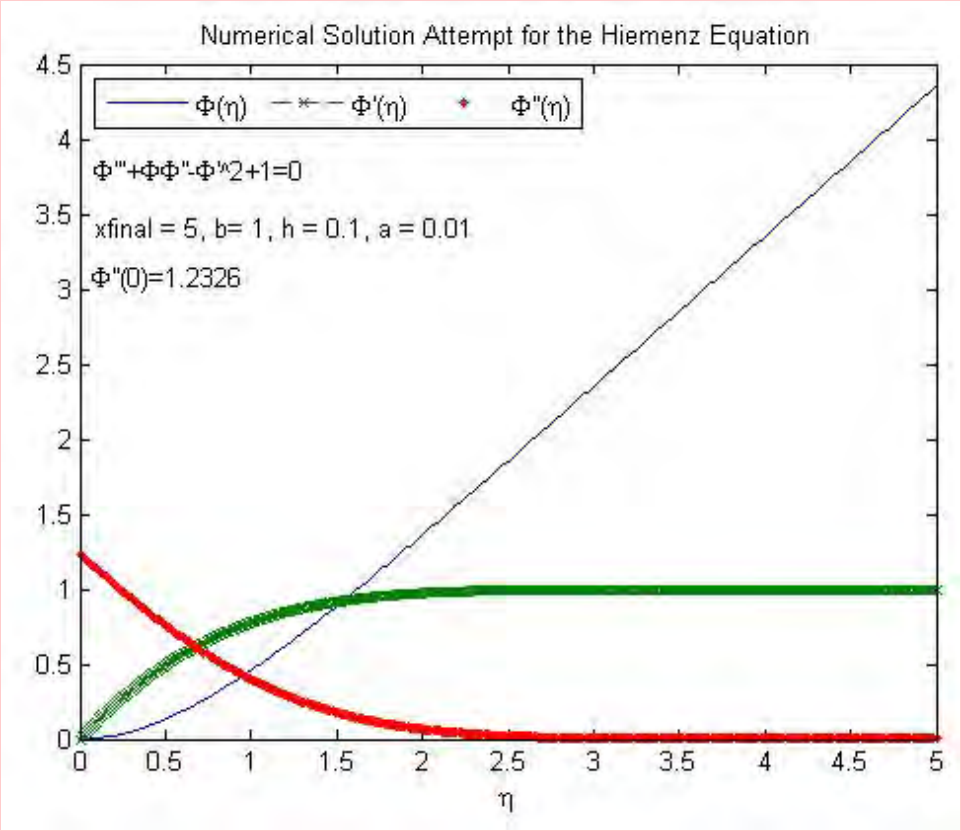
$$\phi''' + \phi\phi'' - \phi'^2 + 1 = 0$$

- Boundary Conditions

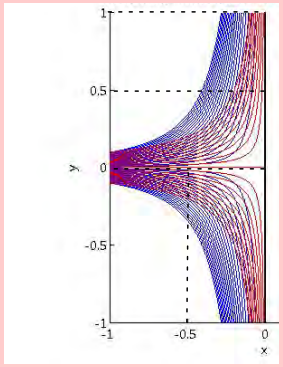
$$\phi(0) = \phi'(0) = 0$$

$$\phi'(\infty) = 1$$

- Solve Using Shooting Method

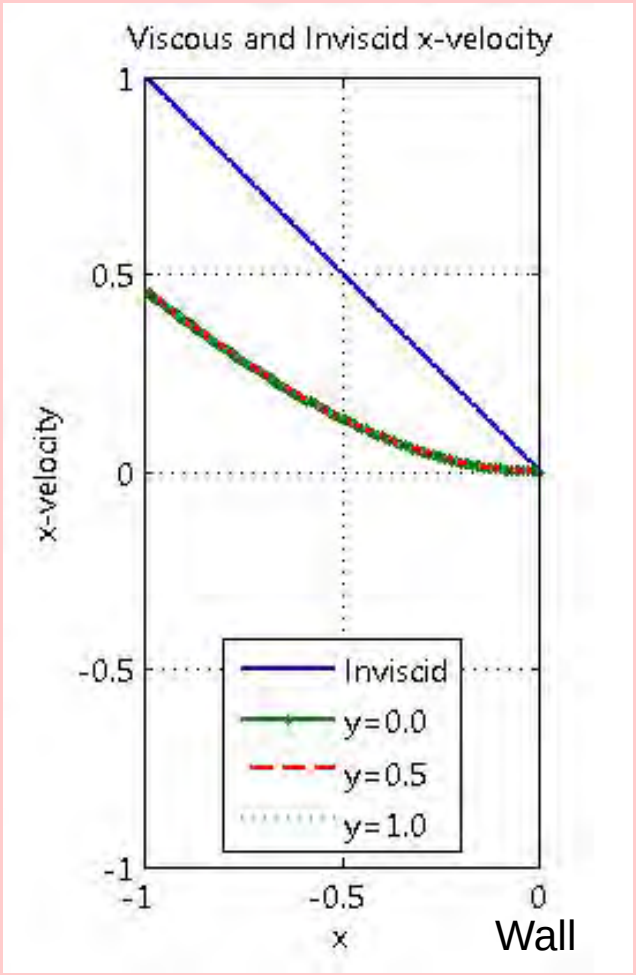


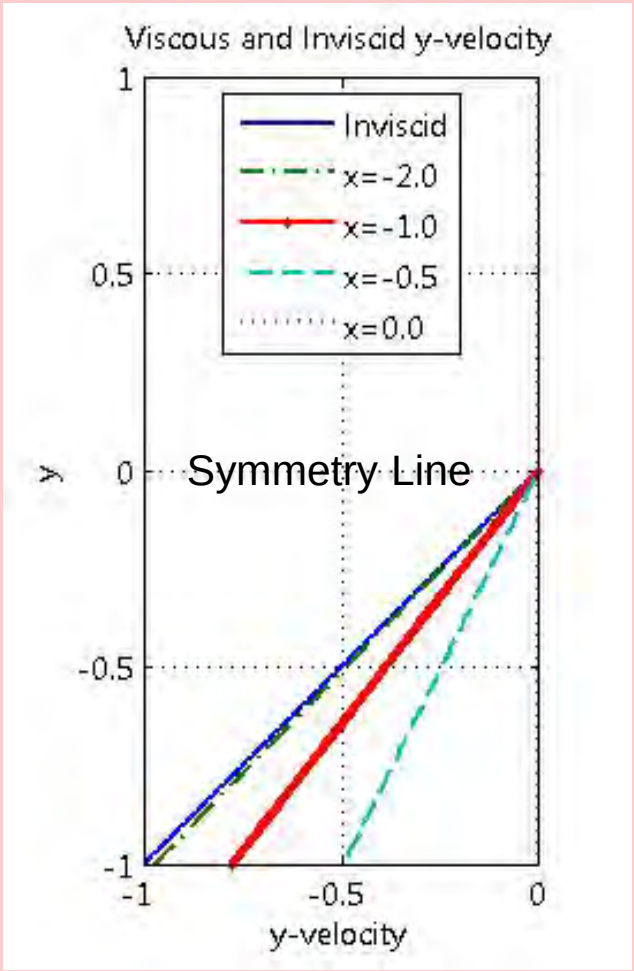
Inviscid and Viscous Hiemenz Flow: Velocity



Flow Direction

Velocity Profiles

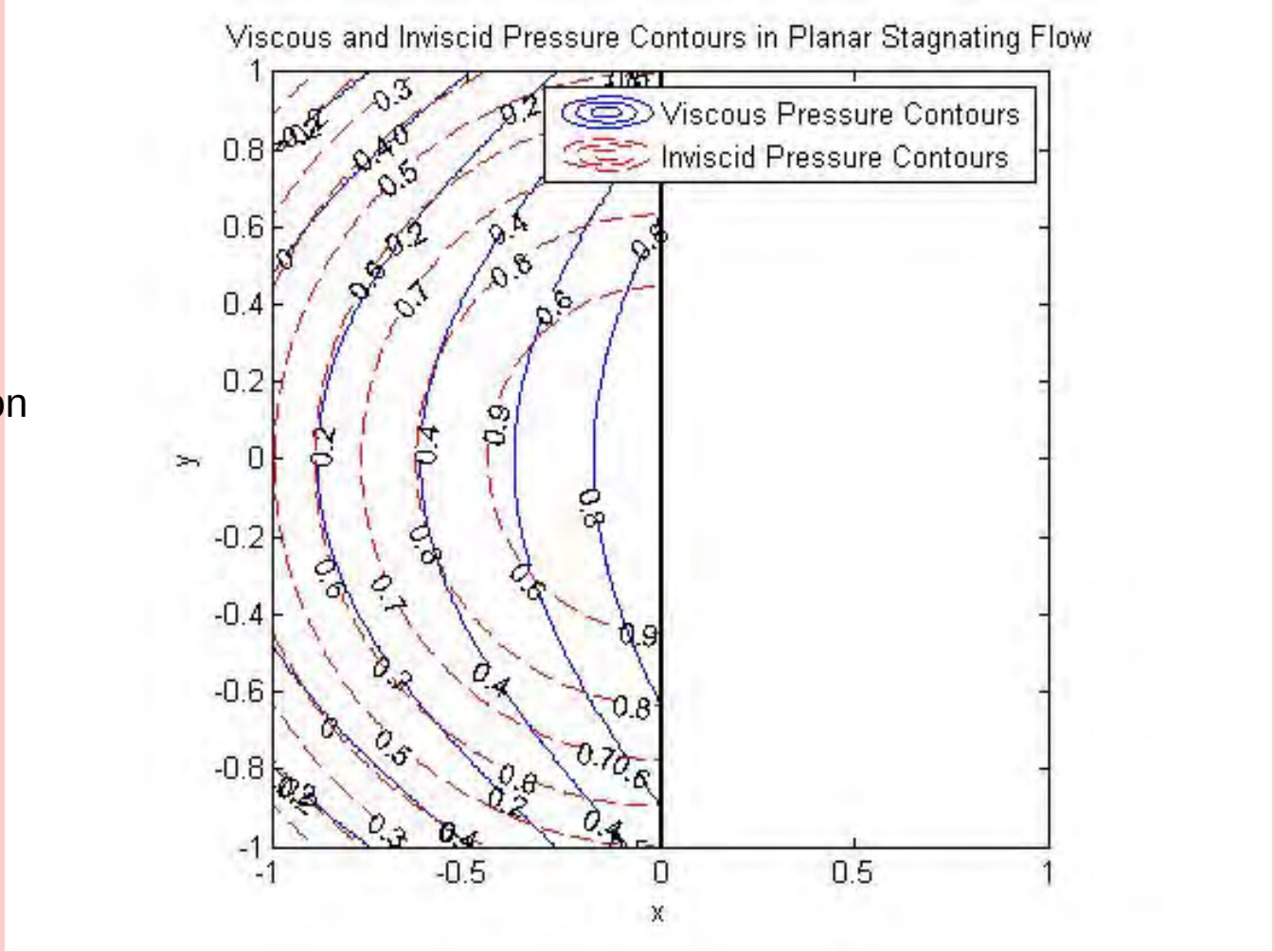




Inviscid and Viscous Hiemenz Flow: Pressure

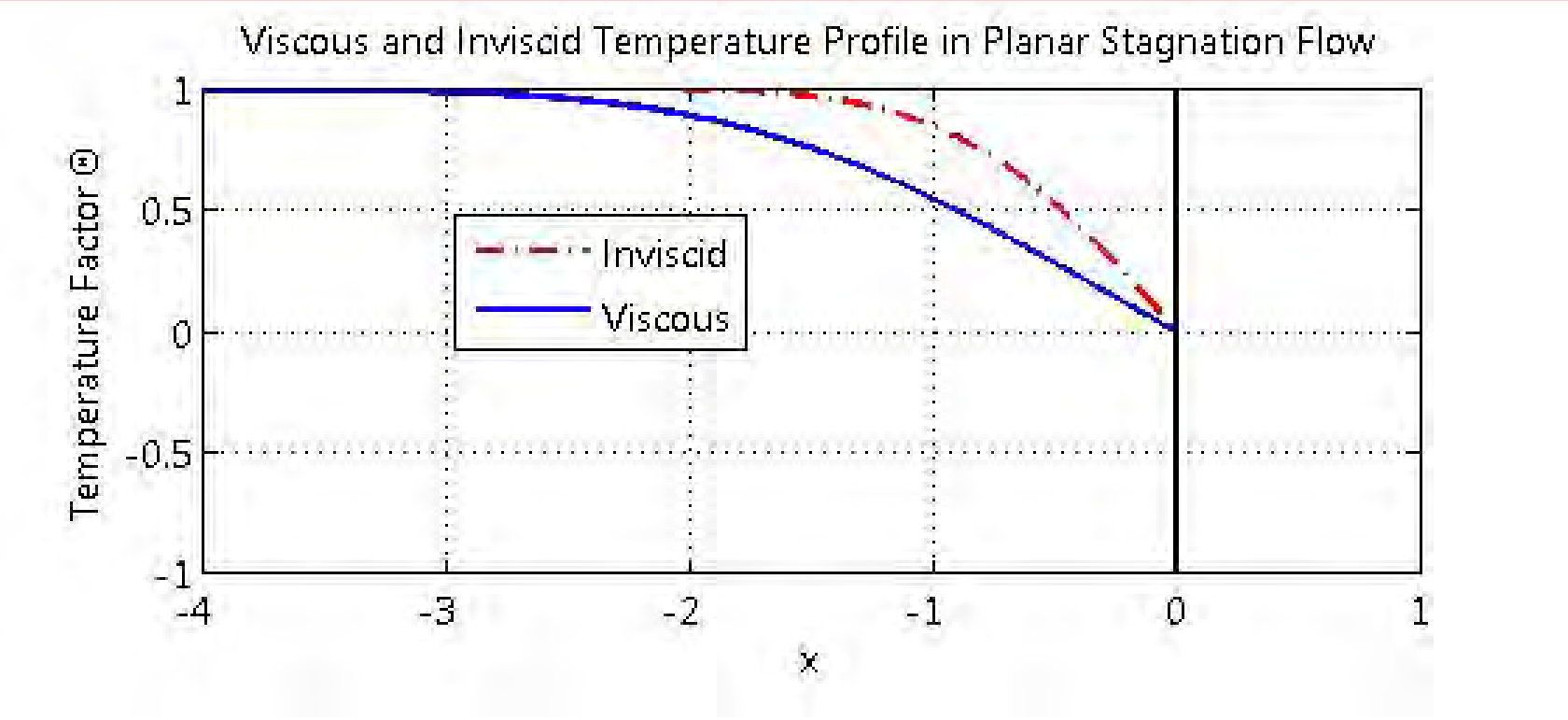
Pressure Contours

Flow Direction



Wall

Inviscid and Viscous Hiemenz Flow: Temperature



- **Inviscid heat flux at the wall is ~ 2X Viscous**

COMSOL & FLUENT: Case Study - Inputs

- **Compare Analytical Solution to Computational Results**
- **Set up Dimensional Problem for Easy Comparison**

Material Properties

- **Density = 1 kg/m^3**
- **Viscosity = 1 kg/ms**
- **Conductivity = 1 W/m^{o}C**

Flow Field Scale Factor

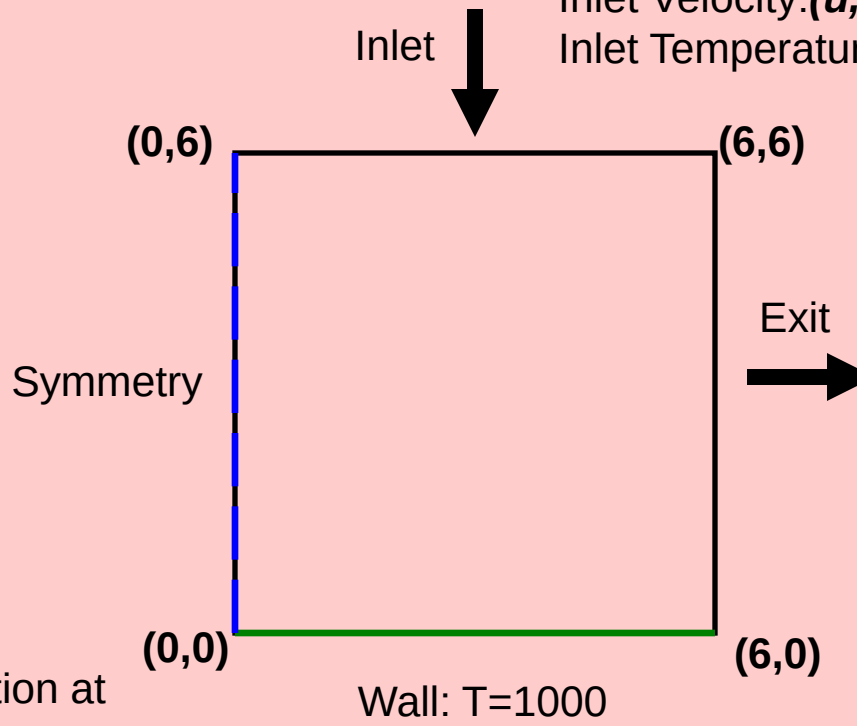
- **a = 1 s^{-1}**

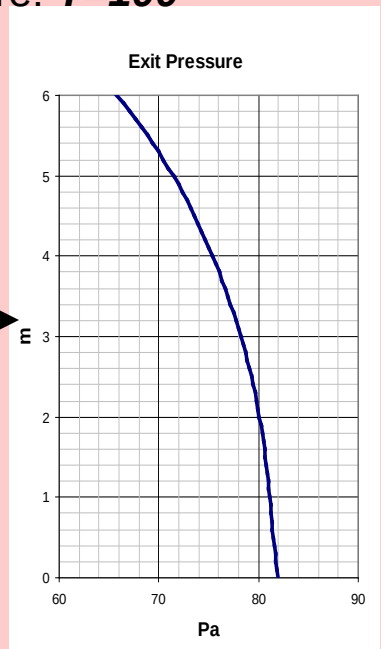
Similarity Factors

- $\eta = y/1$
- $\phi = f/1$

Boundary Conditions

- Consistent with analytical solution at boundaries
- Assume stagnation pressure of 100 Pascals

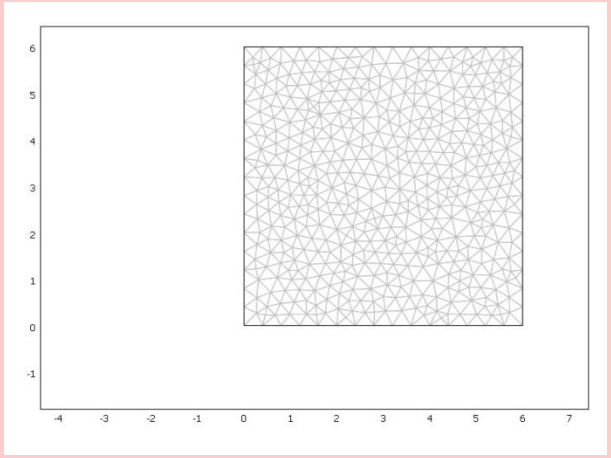




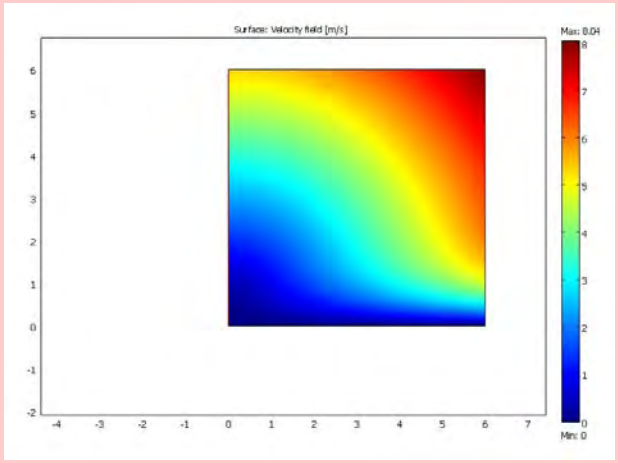
Computational Domain

- **6 meters by 6 meters**

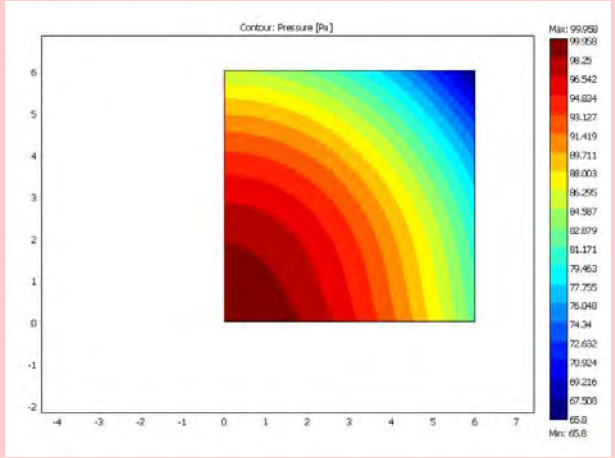
COMSOL & FLUENT: Case Study - Results



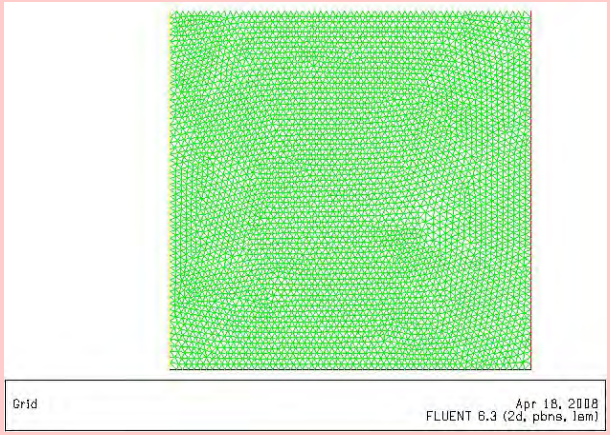
COMSOL Mesh



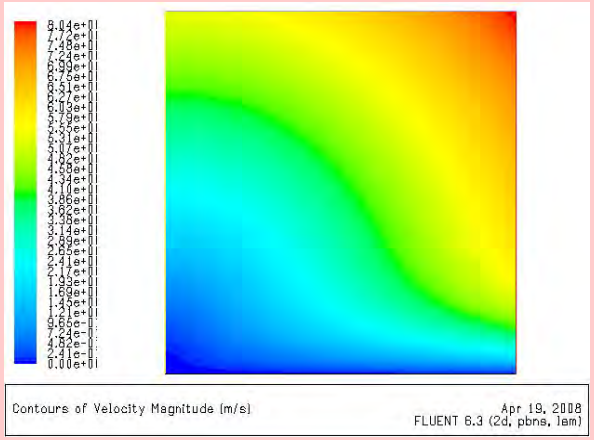
COMSOL Velocity



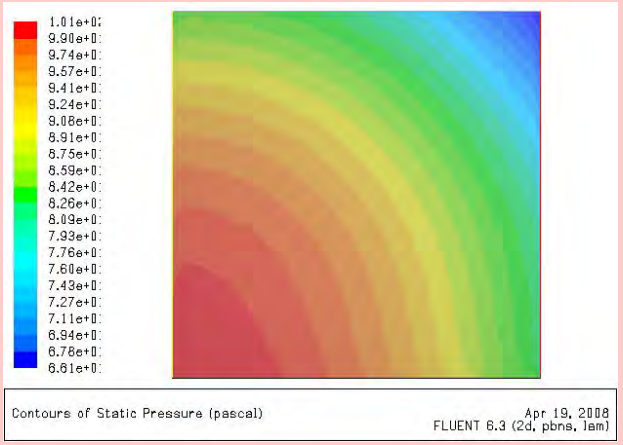
COMSOL Pressure Field



FLUENT Mesh



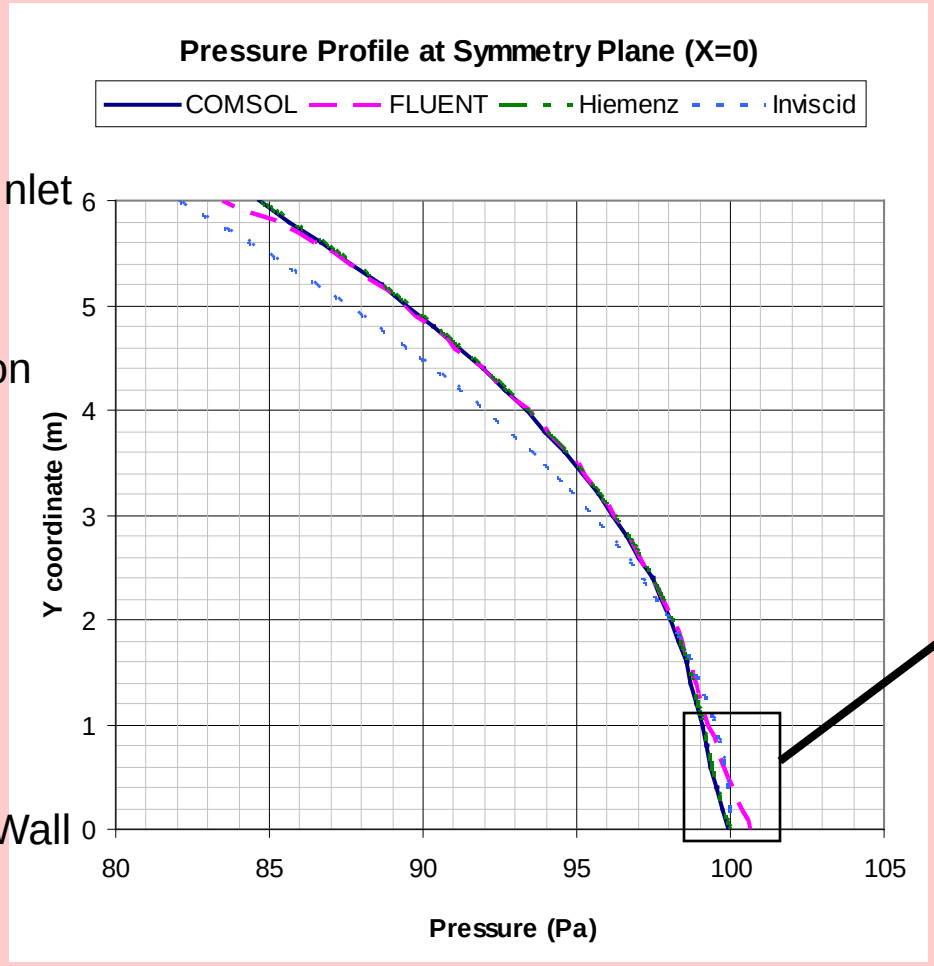
FLUENT Velocity

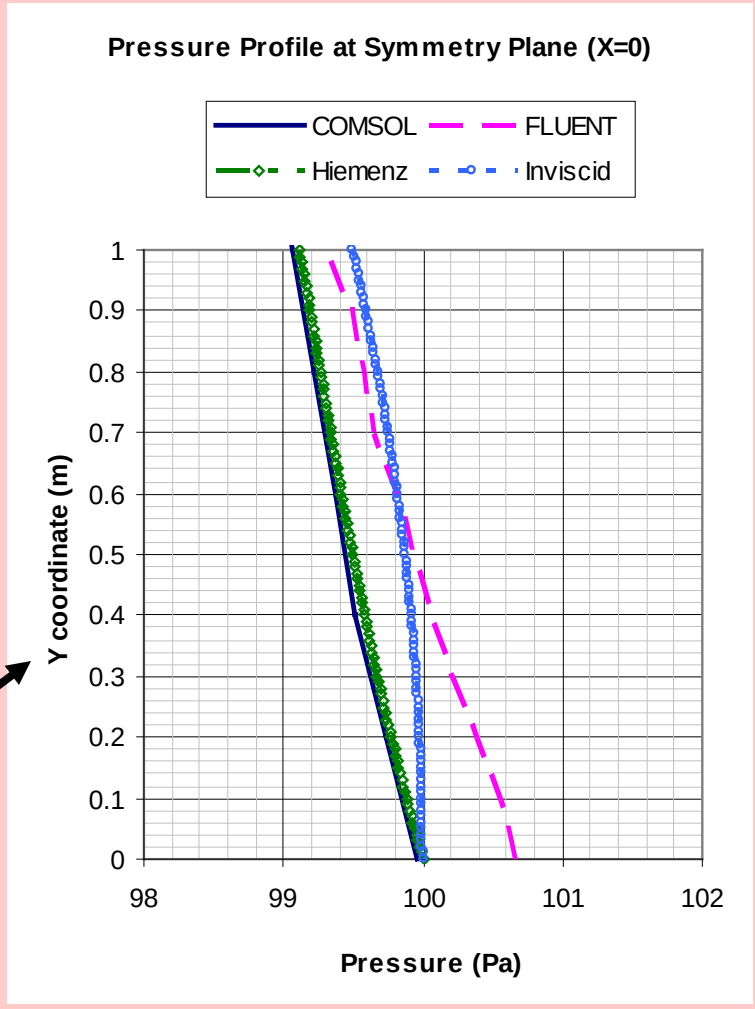


FLUENT Pressure Field

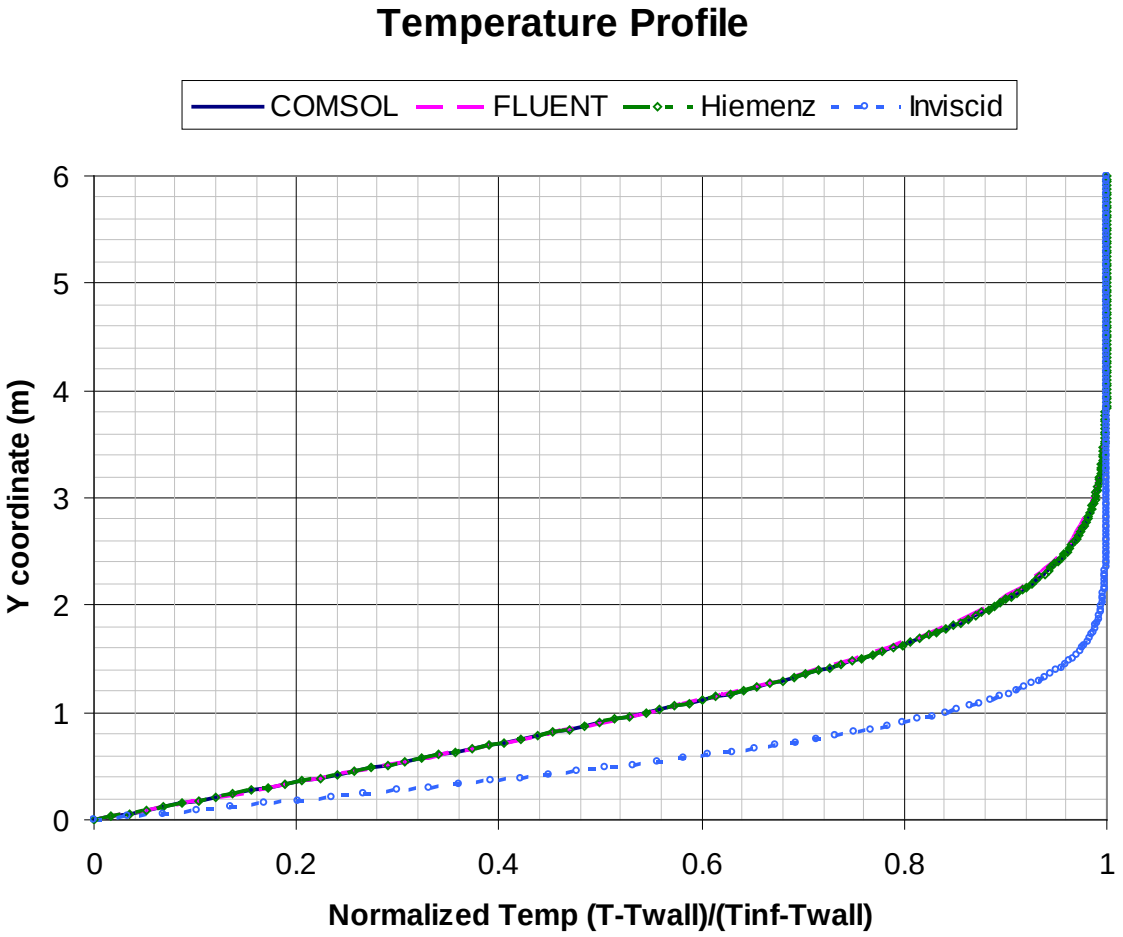
COMSOL & FLUENT: Case Study – Results
Static Pressure Along Symmetry Plane

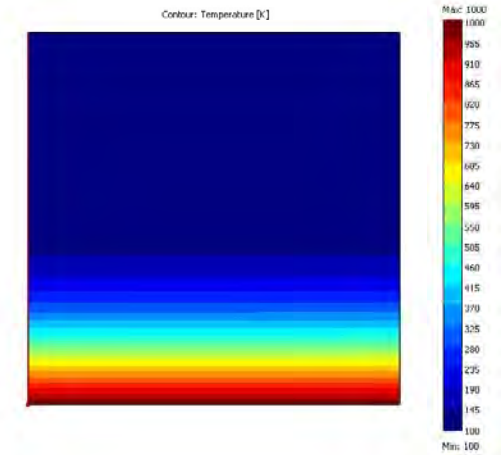
Flow Direction



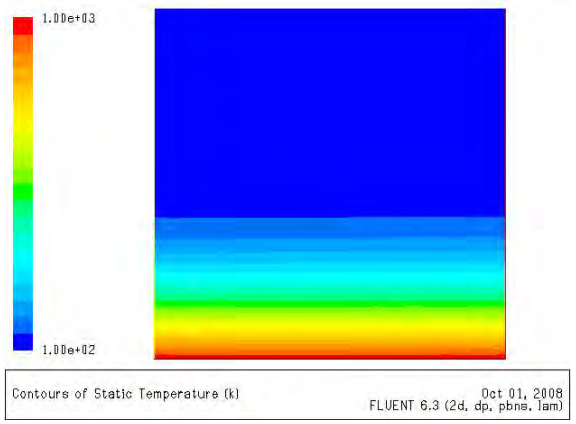


COMSOL & FLUENT: Case Study – Results Temperature





COMSOL Temperature Distribution



FLUENT Temperature Distribution

CONCLUSIONS

- **Both COMSOL and FLUENT have been shown to reliably reproduce the flow field**

- **Velocities, Pressures and Temperatures match the analytical predictions closely**

- **Selection of appropriate boundary conditions and adequate domain size are critical to accurately predict flow field**