

FORCED CONVECTION SIMULATION FOR HT2 EXPERIMENT

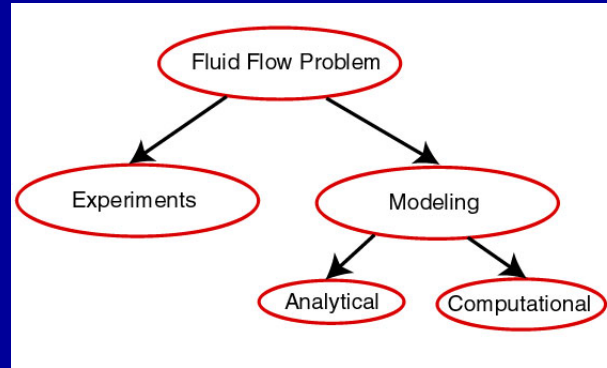
MAE 4272

Fall 2009

Mechanical & Aerospace Engineering

Cornell University

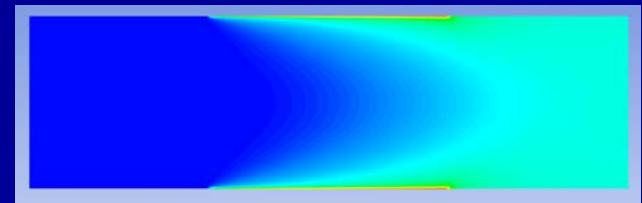
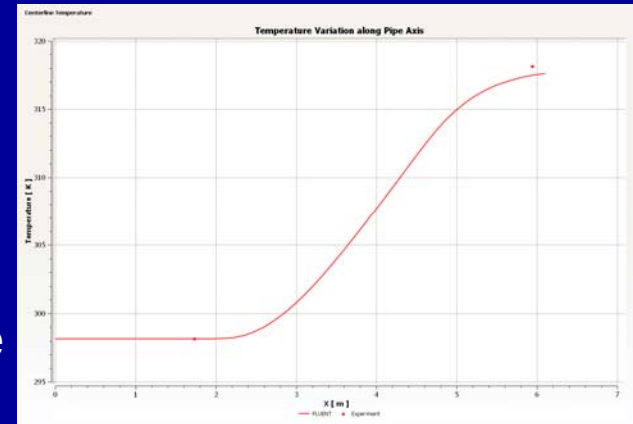
Experiment vs. Modeling



- Computational Fluid Dynamics (CFD): Computational modeling of fluid flow problems.
- Falls under the rubric of “Computer Simulation” or just “Simulation”.
- We’ll use FLUENT software for the HT2 simulation. Popular CFD package used in industry.
- FLUENT will be used to obtain an **approximate** solution to the governing equations.

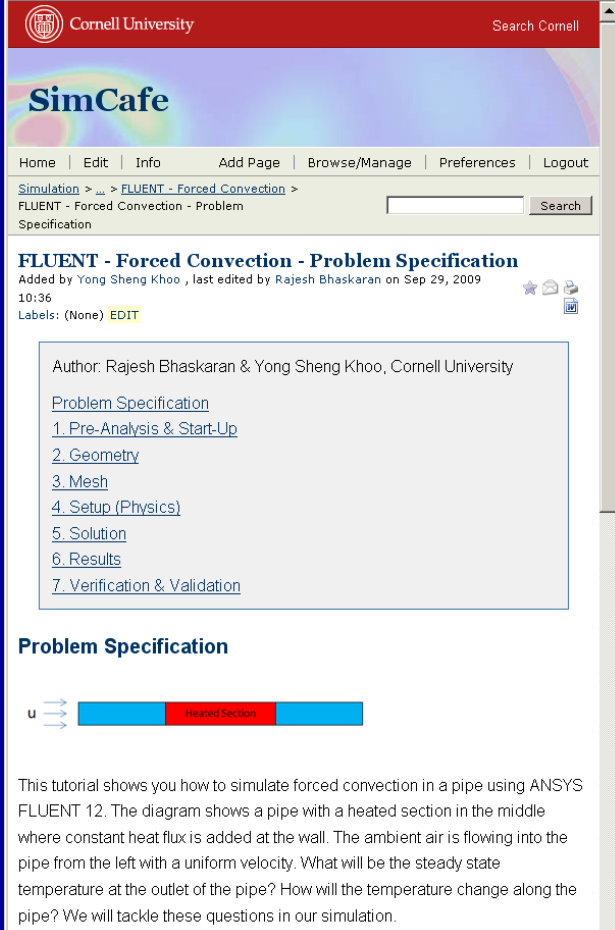
Why Perform the CFD Simulation?

- We can
 - See how simulation can complement experiments.
 - Look “under the hood” using the simulation.
 - Get a better understanding of the flow than is possible from point measurements
 - Check assumptions made in post-processing experimental data
 - Get an overview of the CFD simulation process and its benefits and challenges.
 - “Garbage in, garbage out”



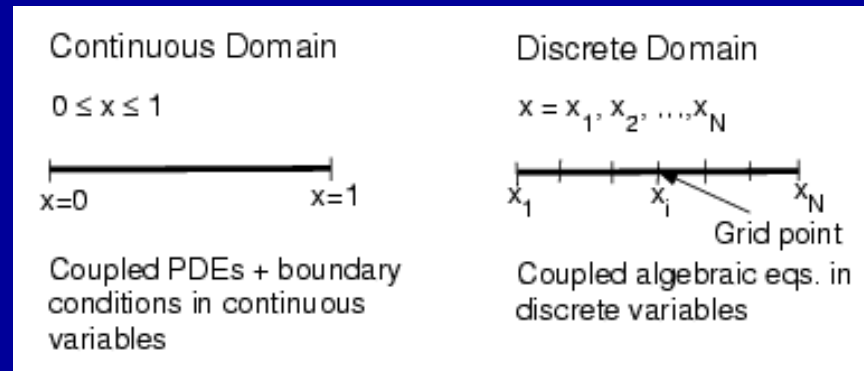
CFD Simulation

- A tutorial on how to apply FLUENT to simulate the forced convection experiment is available at: <https://confluence.cornell.edu/display/simulation/forcedconvection>
- Run FLUENT and tutorial side-by-side.
- Skip geometry and mesh steps (mesh is provided).



The screenshot displays the SimCafe web application interface. At the top, there is a red header bar with the Cornell University logo and a search bar. Below this, the SimCafe logo is prominently displayed. A navigation menu includes links for Home, Edit, Info, Add Page, Browse/Manage, Preferences, and Logout. The main content area shows the breadcrumb trail: Simulation > ... > FLUENT - Forced Convection > FLUENT - Forced Convection - Problem Specification. A search bar is located to the right of the breadcrumb trail. The title of the page is "FLUENT - Forced Convection - Problem Specification", with a subtitle indicating it was added by Yong Sheng Khoo and last edited by Rajesh Bhaskaran on Sep 29, 2009. Below the title, there is a list of labels: (None) and an EDIT link. The author information is listed as "Author: Rajesh Bhaskaran & Yong Sheng Khoo, Cornell University". A table of contents is provided, listing seven sections: 1. Pre-Analysis & Start-Up, 2. Geometry, 3. Mesh, 4. Setup (Physics), 5. Solution, 6. Results, and 7. Verification & Validation. The "Problem Specification" section is currently selected. A diagram illustrates a pipe with a heated section in the middle, indicated by a red bar labeled "Heated Section". The flow direction is indicated by arrows labeled "u". The text below the diagram explains the simulation setup: "This tutorial shows you how to simulate forced convection in a pipe using ANSYS FLUENT 12. The diagram shows a pipe with a heated section in the middle where constant heat flux is added at the wall. The ambient air is flowing into the pipe from the left with a uniform velocity. What will be the steady state temperature at the outlet of the pipe? How will the temperature change along the pipe? We will tackle these questions in our simulation."

Strategy of CFD

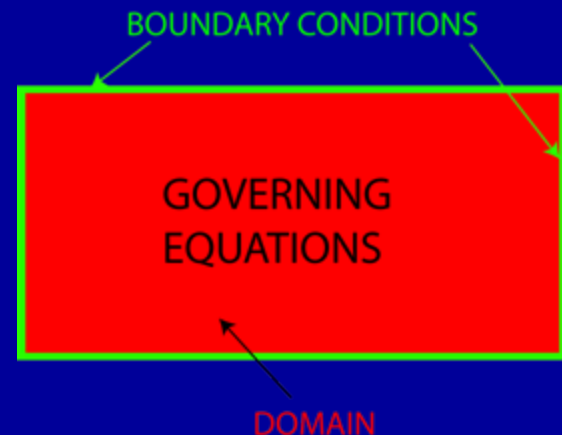


- Eg.: Continuous Domain: $p = p(x)$, $0 < x < 1$
Discrete Domain: $p_i = p(x_i)$, $i=1,2,\dots,N$
- Truncation error introduced. Can be reduced by refining the mesh.
- Mesh refinement study required to assess the level of truncation error.

CFD Simulation

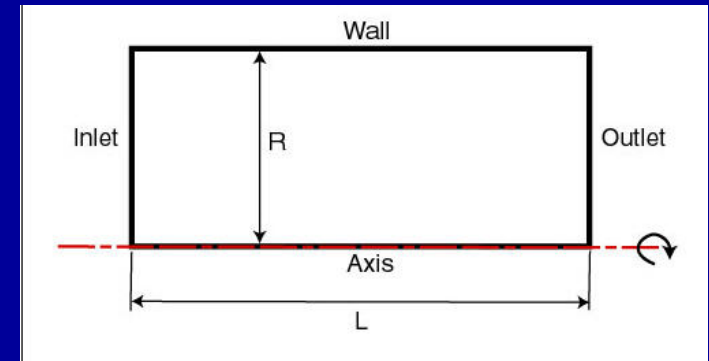
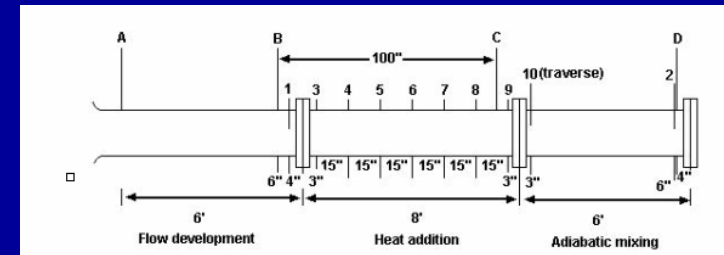
- We'll be using FLUENT to solve a boundary value problem.
- We need to specify the governing equations, boundary conditions and domain in FLUENT.
- FLUENT will obtain an approximate numerical solution to the BVP.

Boundary Value Problem



Domain

- Length of pipe included in the simulation: From A to D.
- Assume flow is axisymmetric. Hence, domain is rectangular.
- Rotate the rectangle 360° about the axis to get the full pipe geometry.
- We'll solve the axisymmetric form of the governing equations.

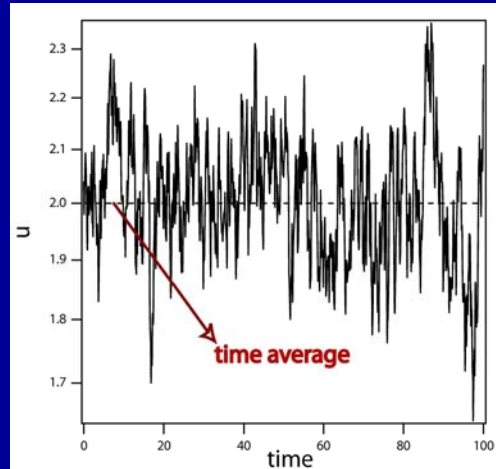


Governing Equations

- Coupled non-linear partial differential equations
- No. of independent variables = 5
- No. of equations = 5
- Need to modify these to account for the effect of turbulence

Equation	Dependent variables
Continuity eq. in cylindrical coordinates	u, v, ρ
Axial momentum eq.	u, v, p, ρ
Radial momentum eq.	u, v, p, ρ
Energy eq. in cylindrical coordinates	u, v, p, T, ρ
Ideal gas law	p, T, ρ

Turbulence



- Cannot resolve rapid fluctuations in turbulent flow
- We solve only for averaged quantities: $u' = u - \bar{u}$
- Average the governing equations → Reynolds Averaged Navier Stokes (RANS) equations.
- RANS equations govern the *mean* velocities, pressure and temperature.

Turbulence

- Problem: Fluctuating quantities appear in the RANS equations
- Example: x-momentum for 2D, incompressible flow

$$\rho \frac{\partial(\bar{u}^2)}{\partial x} + \rho \frac{\partial(\bar{u}\bar{v})}{\partial y} + \rho \frac{\partial(\overline{u'u'})}{\partial x} + \rho \frac{\partial(\overline{u'v'})}{\partial y} = -\frac{\partial \bar{p}}{\partial x} + \mu \left(\frac{\partial^2 \bar{u}}{\partial x^2} + \frac{\partial^2 \bar{u}}{\partial y^2} \right)$$

- Underlined terms: additional momentum fluxes resulting from turbulent fluctuations.
- Called “turbulent stresses” or “Reynolds stresses”.

Turbulence

- Common approach: Relate the Reynolds stresses to the mean velocity gradients through an equation of the form:

$$-\overline{\rho u'v'} = \mu_t \left(\frac{\partial \bar{u}}{\partial y} + \frac{\partial \bar{v}}{\partial x} \right) \quad \mu_t(x,y): \text{“Turbulent viscosity”}$$

- RANS equations end up looking almost like the laminar equations.

$$\rho \frac{\partial(\bar{u}^2)}{\partial x} + \rho \frac{\partial(\bar{u}\bar{v})}{\partial y} = -\frac{\partial \bar{p}}{\partial x} + (\mu + \mu_t) \left(\frac{\partial^2 \bar{u}}{\partial x^2} + \frac{\partial^2 \bar{u}}{\partial y^2} \right)$$

k - ε Turbulence Model

- There are many different semi-empirical “models” to calculate the turbulent viscosity
- All can be useful and all can burn you
- A model that is used a lot is the k - ε turbulence model

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon}$$

- k : kinetic energy contained in the fluctuations
- ε : rate at which k is dissipated into heat

$$k = \frac{1}{2} \left(\overline{u'^2} + \overline{v'^2} \right)$$

$$\varepsilon = \nu \left[\overline{\left(\frac{\partial u'}{\partial x} \right)^2} + \overline{\left(\frac{\partial u'}{\partial y} \right)^2} + \overline{\left(\frac{\partial v'}{\partial x} \right)^2} + \overline{\left(\frac{\partial v'}{\partial y} \right)^2} \right]$$

k - ϵ Turbulence Model

- A semi-empirical transport equation is formulated each for k and ϵ in terms of mean quantities.
- Each of these two equations is a second-order PDE.
- k equation from FLUENT manual:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \epsilon - Y_M + S_k$$

- Will use k - ϵ turbulence model out-of-the-box.

Ideal Gas Law

- Variations in *absolute* pressure are small.
- Use “Incompressible ideal gas” model in FLUENT: Neglects variations in absolute pressure in ideal gas law $\rho = \frac{P_{ref}}{RT}$
- Changes in density are due to changes in temperature.
- Saves on computational work without sacrificing accuracy.
- P_{ref} :FLUENT calls this “operating pressure”.
Input measured ambient value.

Governing Equations: Final Form

- Coupled non-linear partial differential equations
- No. of independent variables = 7
- No. of equations = 7

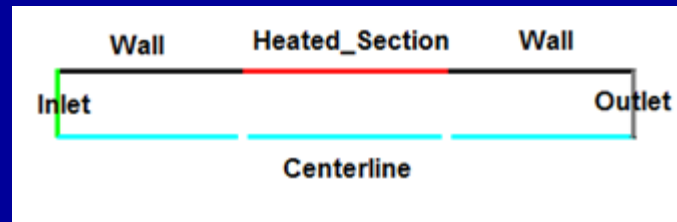
Equation	Dependent variables
Continuity eq. in cylindrical coordinates (Reynolds-averaged)	$\bar{u}, \bar{v}, \bar{\rho}$
Axial momentum eq. (Reynolds-averaged)	$\bar{u}, \bar{v}, \bar{p}, \bar{\rho}$
Radial momentum eq. (Reynolds-averaged)	$\bar{u}, \bar{v}, \bar{p}, \bar{\rho}$
Energy eq. in cylindrical coordinates (Reynolds-averaged)	$\bar{u}, \bar{v}, \bar{p}, \bar{T}, \bar{\rho}$
“Incompressible ideal gas” law	$\bar{p}, \bar{T}, \bar{\rho}$
Transport equation for k	$k, \bar{u}, \bar{v}, \bar{\rho}$
Transport equation for ε	$\varepsilon, \bar{u}, \bar{v}, \bar{\rho}$

Material Properties

- Material properties appearing in the governing equations: μ , C_p , thermal conductivity
- These are functions of temperature for air
- Approximation: Assume these are constant and use average values over temperature range that you get in the experiment

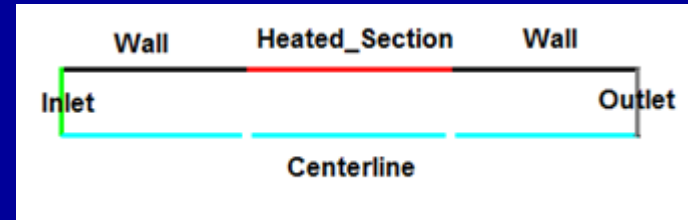
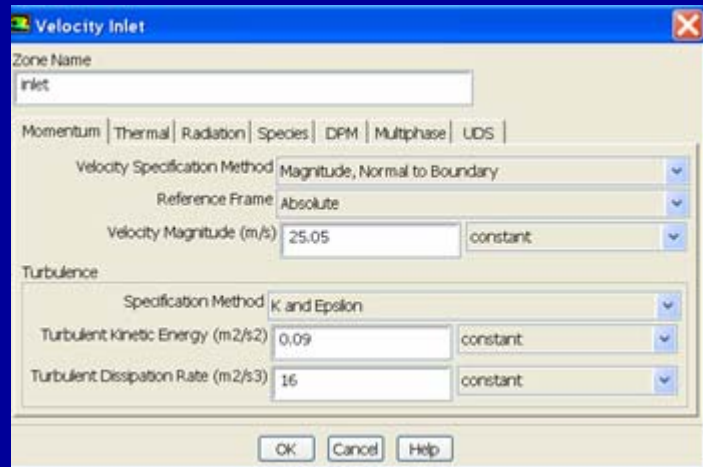
Boundary Conditions

- Boundaries are labeled as follows.



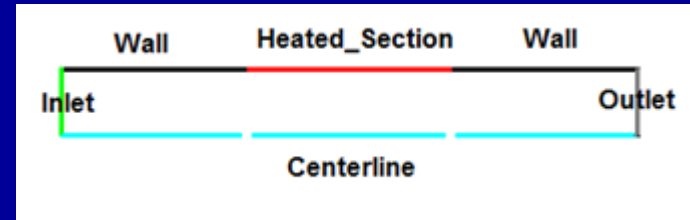
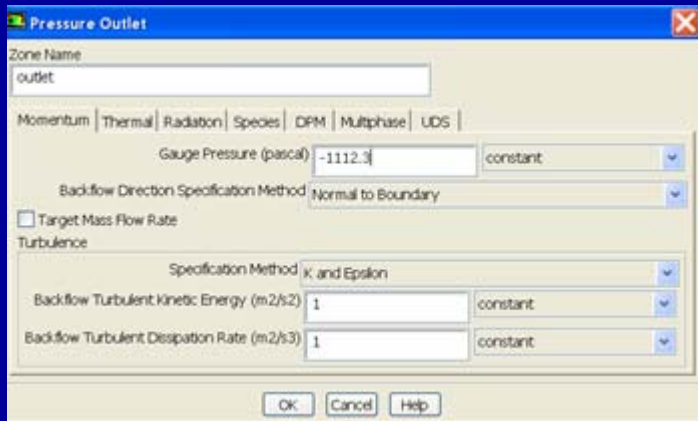
- FLUENT provides a variety of boundary types: “velocity inlet”, “pressure outlet” etc.
- For each labeled boundary, you have to pick the appropriate boundary type and then input the settings (velocity, pressure etc). for that boundary type.

Boundary Conditions at Inlet



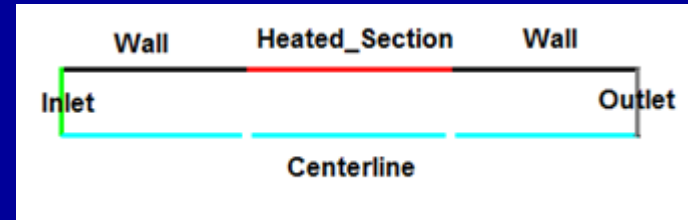
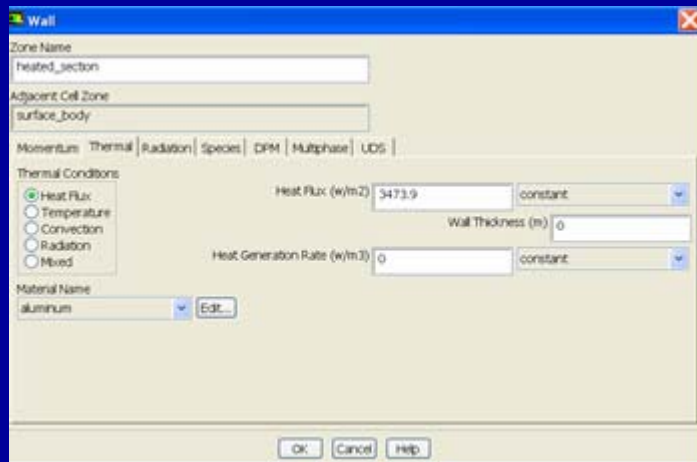
- Use “velocity inlet” boundary type.
- Assume uniform flow in axial direction at inlet.
- Calculate velocity magnitude from measured mass flow rate.
- Inlet values k and ε are wild guesses. Solution is not sensitive to these since most of the turbulence is generated in the boundary layers.

Boundary Conditions at Outlet



- Use “pressure outlet” boundary type.
- Need to input measured gauge pressure (baseline is “operating pressure”).
- Unfortunately, this pressure value is anomalous. Use a linear extrapolation of the pressure trend.

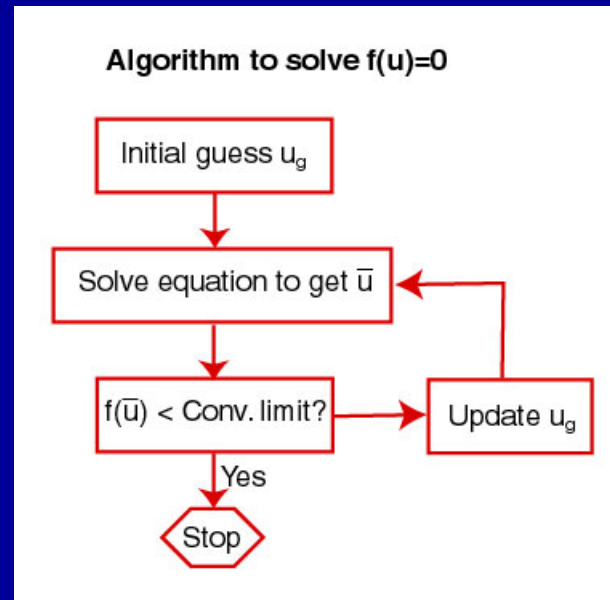
Boundary Conditions for Heated Section



- Use “wall” boundary type.
- Input measured constant heat flux.
- FLUENT imposes no-slip condition for velocity.
- We’ll neglect heat conduction within pipe wall

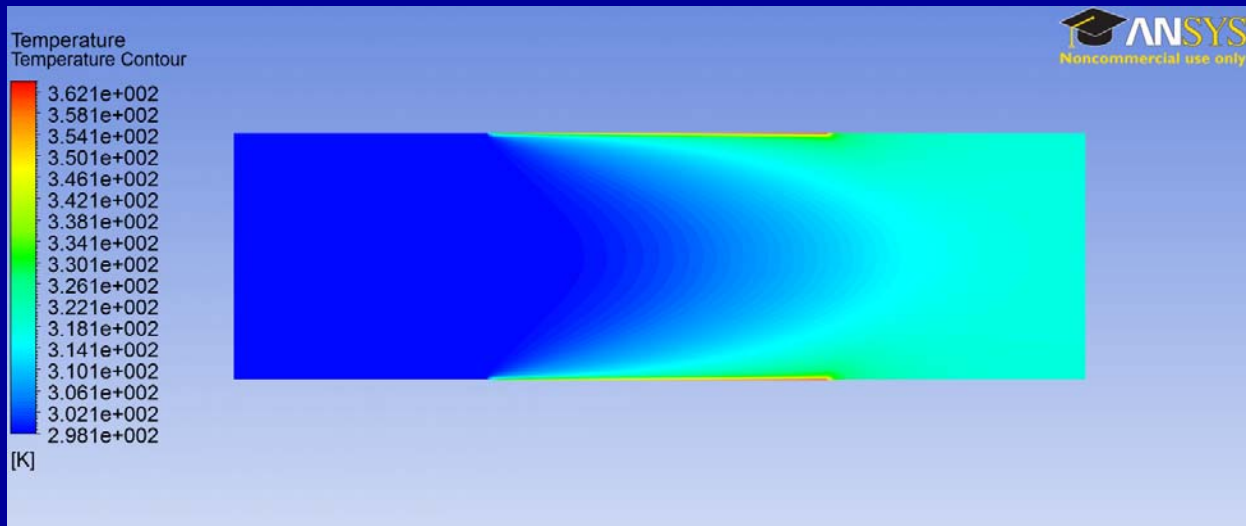
Solution

- Since governing equations are nonlinear, iterations are required to solve the equations.



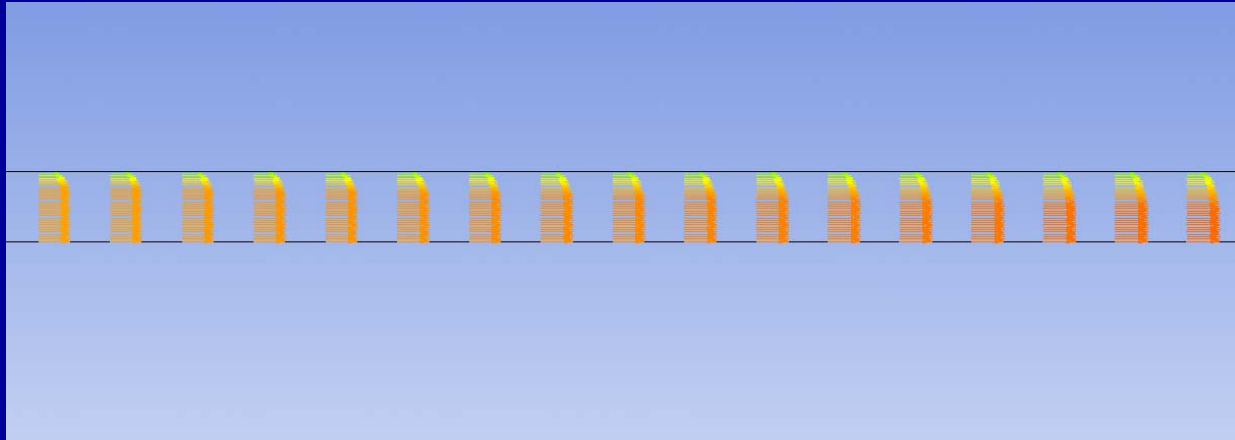
Results

- Temperature contours: Is the flow well-mixed at the end of the adiabatic mixing section?



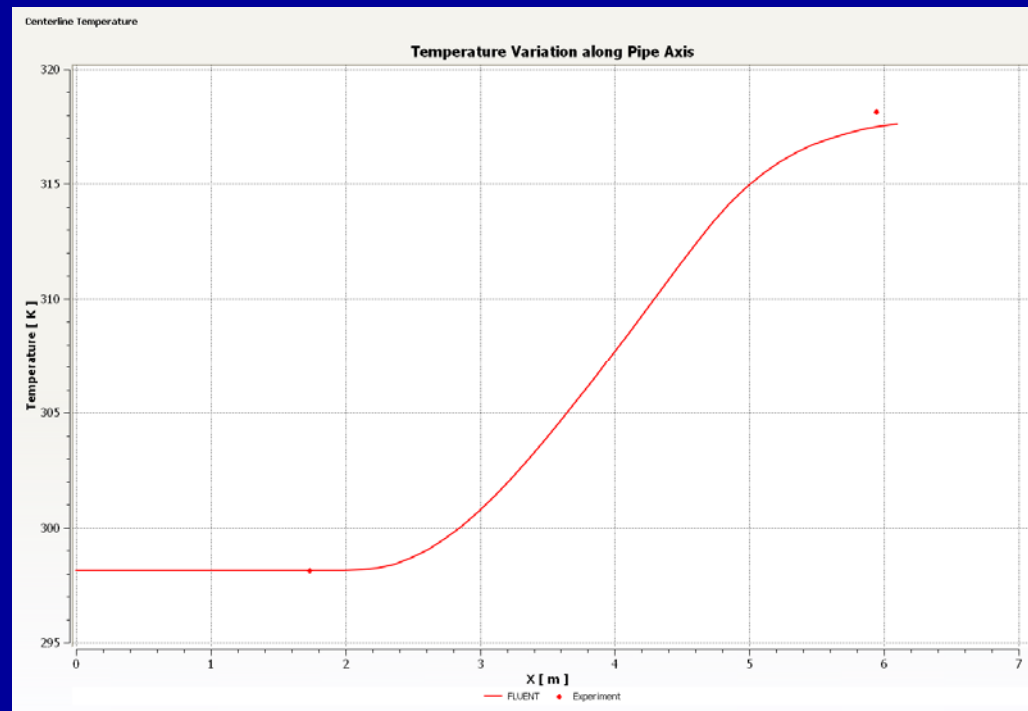
Results

- Velocity vectors in the first section showing flow development.



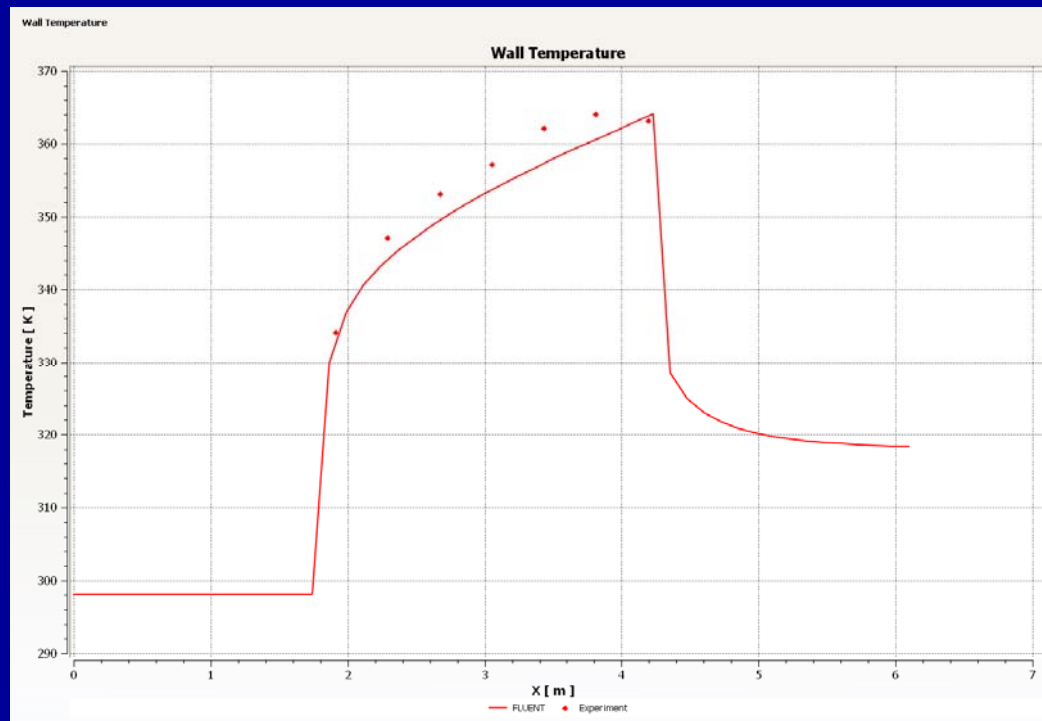
Results

- Temperature variation along pipe axis. Symbols represent experimental values.



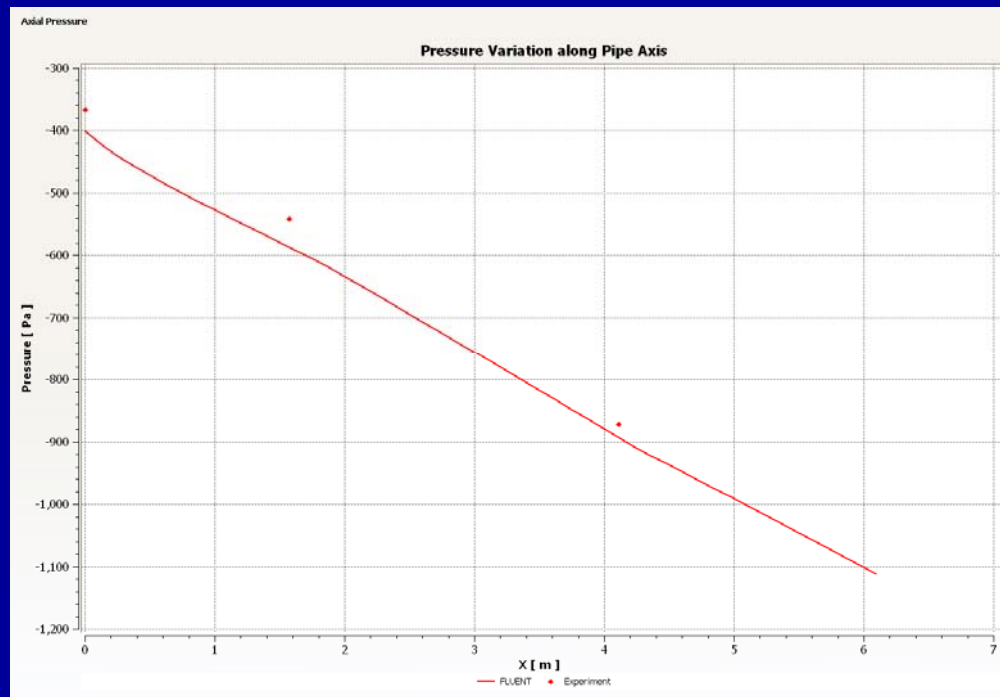
Results

- Wall temperature variation. Shows thermal entrance length effects.



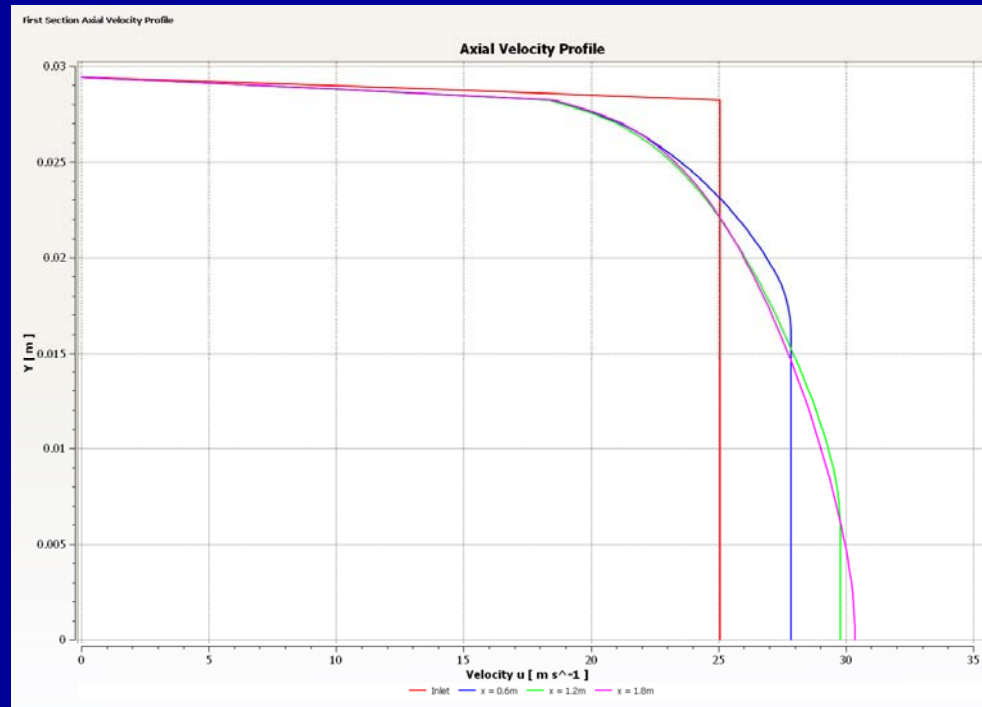
Results

- Pressure variation along pipe axis. Symbols represent experimental values.



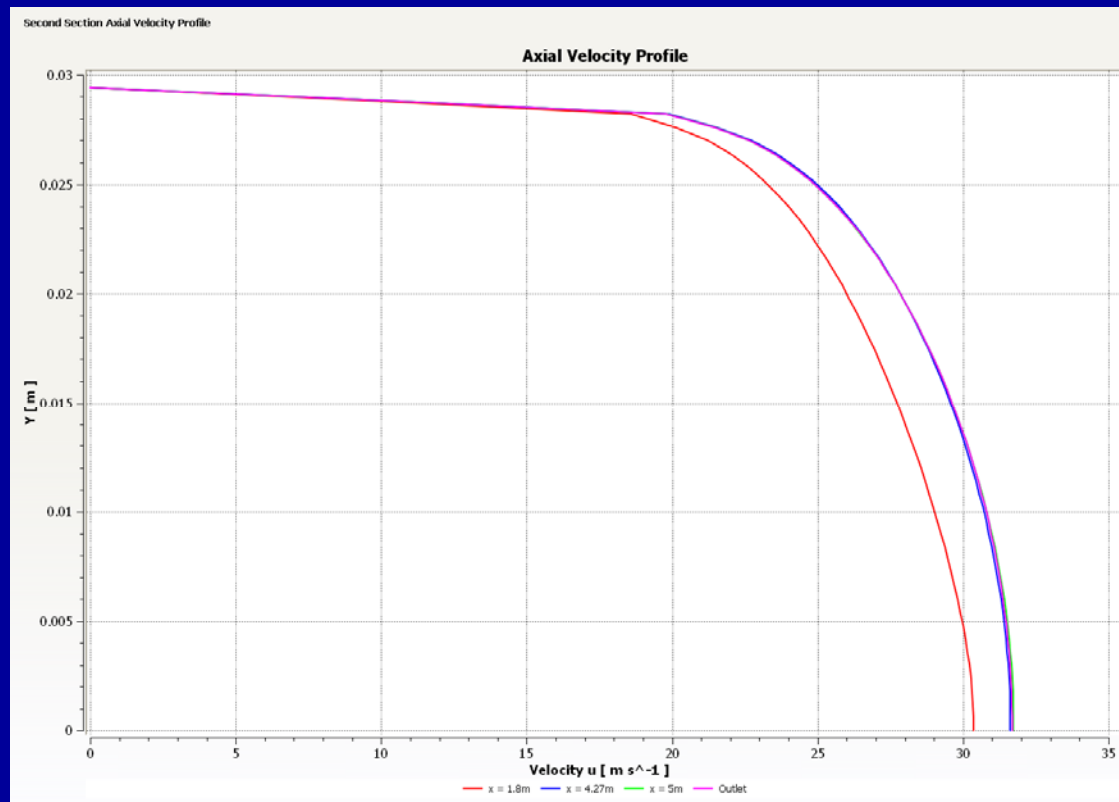
Results

- Axial velocity profiles in the flow development section.
- Plot shows flow is nearly fully developed as it enter the heated section.



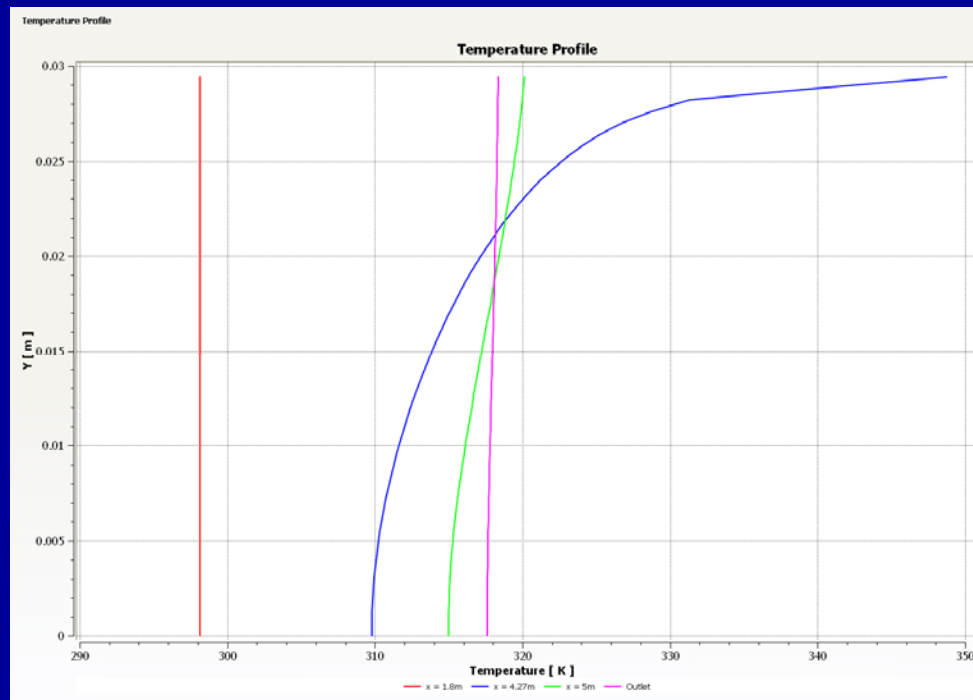
Results

- Axial velocity profiles in heated and mixing sections. Flow accelerates due to heating as expected.



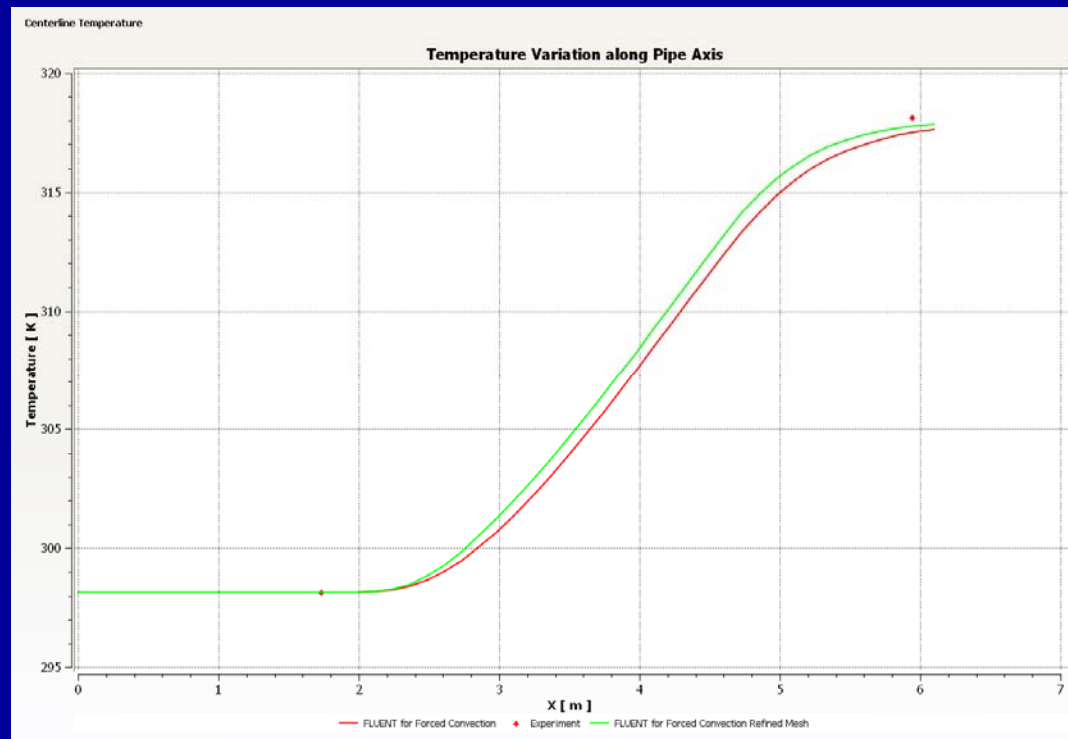
Results

- Temperature profiles at various locations.
- Plot shows temperature is nearly uniform at the outlet (end of the mixing section).



Verification

- Re-do solution on a refined mesh to check effect of mesh on solution.



CFD Tasks for HT2

1. Go through the online tutorial to learn how to apply FLUENT to simulate the HT2 experiment.
 - Skip *Step 2: Geometry* and *Step 3: Mesh*
 - Download the mesh using the link provided in Step 1
2. Repeat the simulation for your particular experimental conditions.
3. Compare your simulation results with your experimental results. Understand and comment on agreements and discrepancies.

See lab manual for computer labs with ANSYS FLUENT 12.

Contact for HT2 Simulation

- Dr. Rajesh Bhaskaran
- E-mail: bhaskaran@cornell.edu
- Fall 2009 Office hours (held in Swanson lab, 163 Rhodes Hall):
 - Wed. 4-5 pm
 - Fri. 2:30-3:30 pm