FORCED CONVECTION SIMULATION FOR HT2 EXPERIMENT
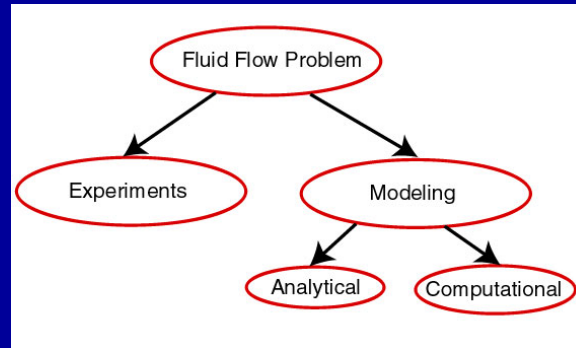
MAE 4272

Fall 2012

Mechanical & Aerospace Engineering

Cornell University

Experiment vs. Simulation


- Computational Fluid Dynamics (CFD): Computational modeling of fluid flow
  - Also called "Computer Simulation" or just "Simulation".
- CFD software in HT2: ANSYS FLUENT™
  - Used to obtain an **approximate** solution to the governing equations.

ANSYS FLUENT™ Software

- One among many general-purpose CFD solvers used in industry.

- Can solve the Navier-Stokes and Euler (inviscid) equations approximately.

- A wide range of physics can be included
  - Turbulence, heating, etc.

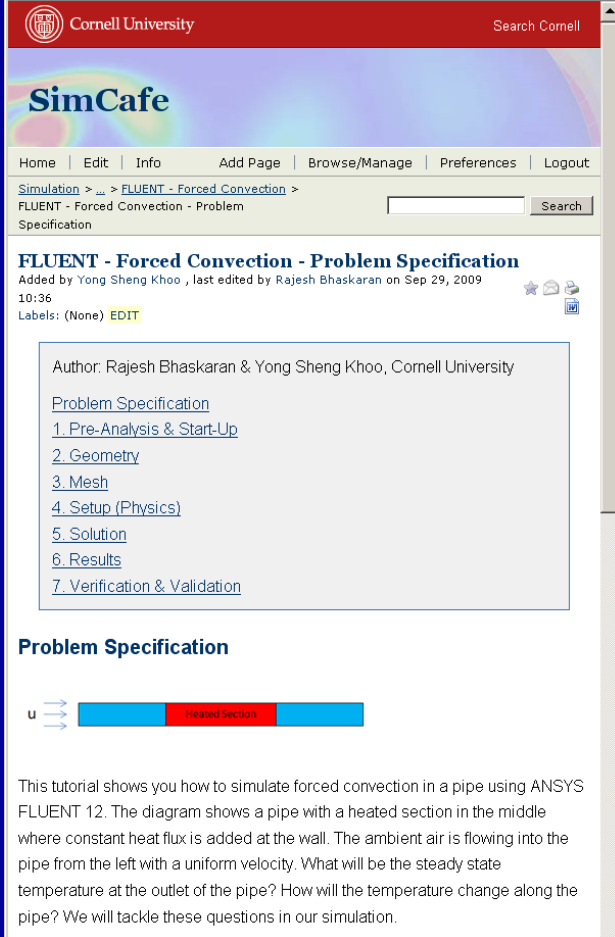- No endorsement of ANSYS FLUENT implied.

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"
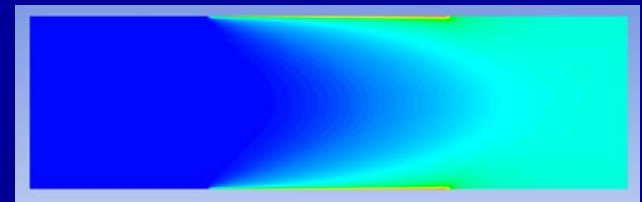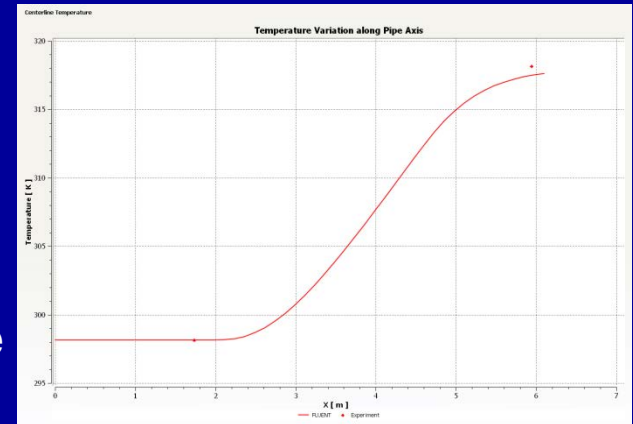
CFD Simulation

- 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).

Computer Labs with FLUENT

- CIT labs
  - B7 Upson
  - 318 Phillips
  - ACCEL lab in Carpenter Hall
  - See lab manual for details

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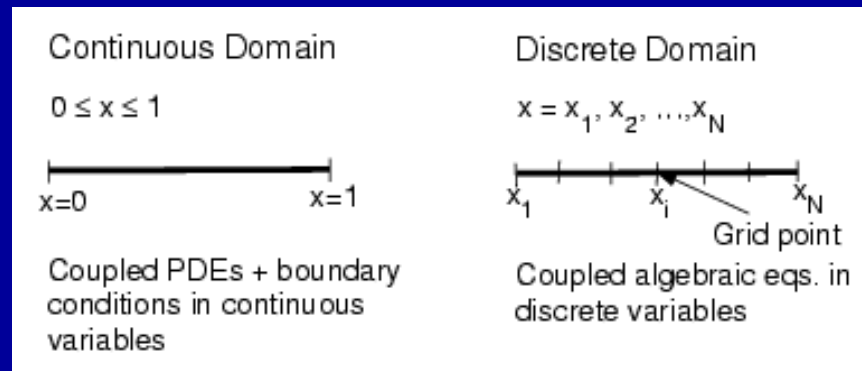
- Eg.: Continuous Domain: $p = p(x)$, $0<x<1$
  Discrete Domain: $p_i = p(x_i)$, $i=1,2,…,N$
- Truncation error introduced. Can be reduced by refining the mesh.
- Mesh refinement study required to assess the level of truncation error.

Exercise
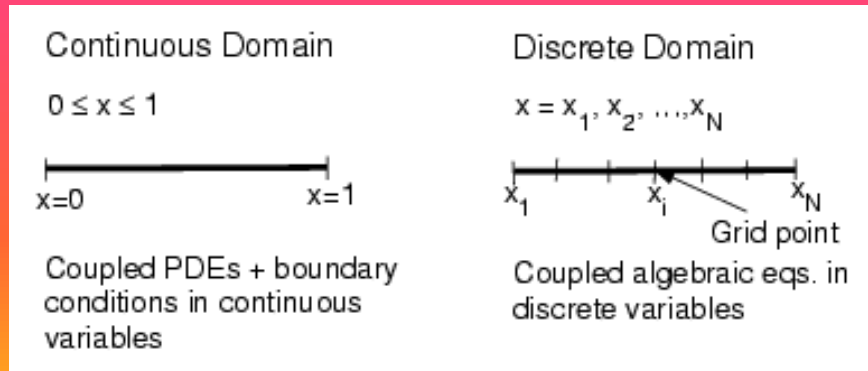
- Discretize and solve the following equation,

$$\frac{dm}{dt} = Am$$

- For A=0.01/day; t between 0 and 100 days. Start with a step size of 20 days; m=10 lbs at t=0.
- Before solving, what is your expectation for m(t)?

Cat food guidelines: the exponential cat
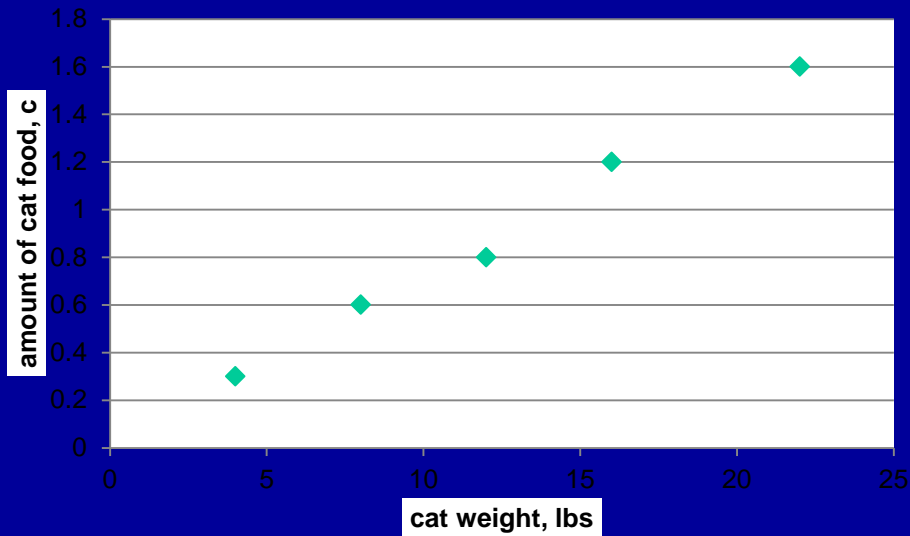
Weight of cat, lbs	Recommended amount of cat food, c
4	1/3-1/4
8	½-2/3
12	¾-1
16	1-1 1/3
22	1 1/3 -2

(Scatter plot: amount of cat food, c (y-axis, 0 to 1.8) vs. cat weight, lbs (x-axis, 0 to 25). Data points approximately at (4, 0.3), (8, 0.6), (12, 0.8), (16, 1.2), (22, 1.6).)

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

BOUNDARY CONDITIONS

GOVERNING EQUATIONS

DOMAIN

Cornell University
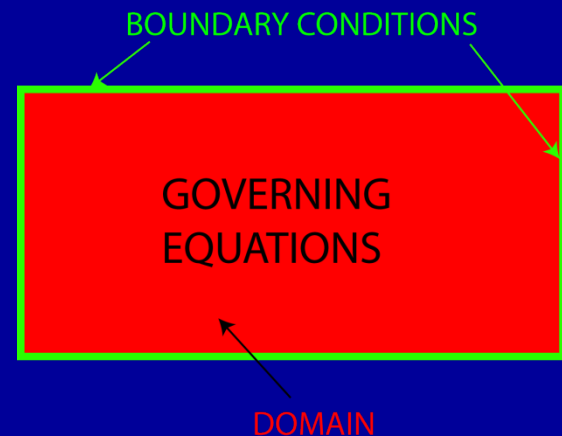
Domain

- 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.

- Solve axisymmetric form of the governing equations.
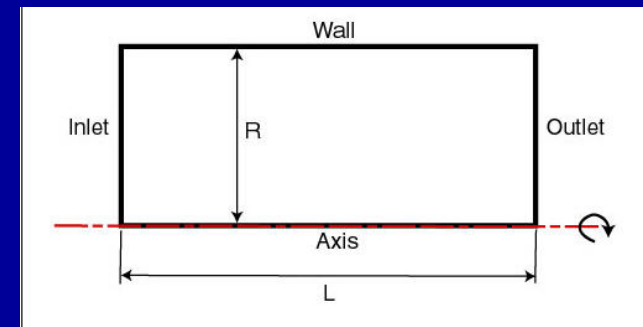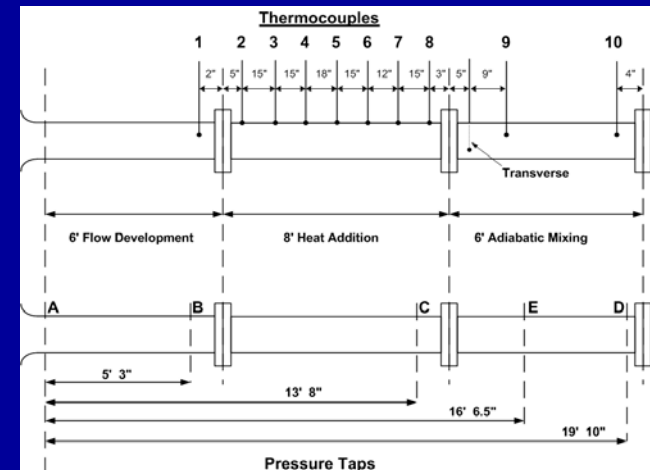
Short Exercise: Pre-Analysis Step

- *Prior to* performing a CFD simulation, you should make back-of-the-envelope estimates.
  - Use these to check CFD results
  - Helps avoid "garbage in, garbage out"
- *Sketch* the following
  - Gas T vs r at start, middle, and end of heated section; middle and end of mixing section
  - Mixed mean gas T vs axial position over the whole reactor length
  - Wall temperature in axial direction
  - Centerline pressure vs. axial position; cold and hot flows
  - Axial velocity vs. r at inlet, middle, and end of flow development section

Governing Equations

- Coupled non-linear partial differential equations

- No. of unknowns=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
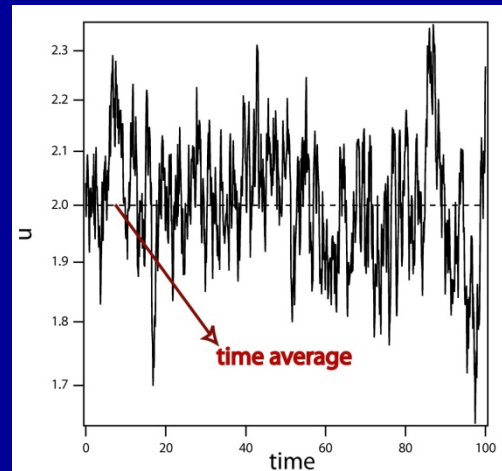
Turbulence

- Cannot usually resolve rapid fluctuations in turbulent flow
- We solve only for averaged quantities: $u = \overline{u} + u'$
- 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:

$$-\rho\overline{u'v'} = \mu_t\left(\frac{\partial\overline{u}}{\partial y} + \frac{\partial\overline{v}}{\partial x}\right)$$

$\mu_t(x,y)$: "Turbulent viscosity"

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

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

k-ε Tubulence 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 Tubulence 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 $\rho = \dfrac{P_{ref}}{RT}$

- 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

- No. of unknowns = 7

- No. of equations = 7
  - 6 coupled non-linear partial differential equations
 - 1 algebraic equation

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

Governing Equations: 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
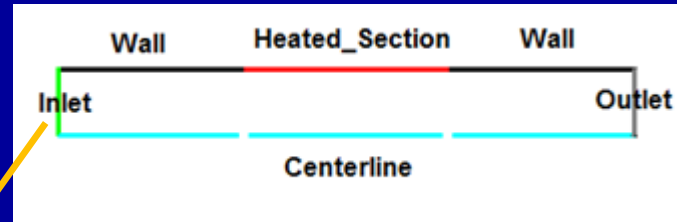
Boundary Conditions

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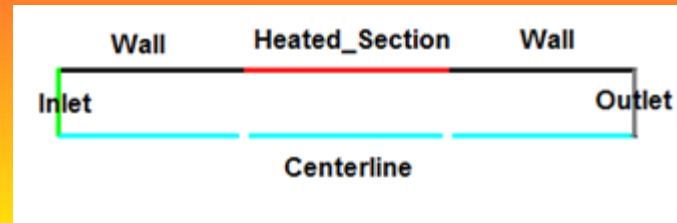
- FLUENT provides a variety of boundary types: "velocity inlet", "pressure outlet" etc.
- For each labeled boundary, you have to pick the appropriate boundary type and then input the settings (velocity, pressure etc). for that boundary type.

Boundary Conditions at Inlet

- Use "velocity inlet" boundary type.
- Assume uniform flow in axial direction at inlet.
  - Measured pressure drop across nozzle ➜ Mass flow rate ➜ Average inlet velocity
- Temperature is measured value at inlet.
- Inlet k and ε are plausible *estimates* based on turbulence intensity and boundary layer thickness.
  - Solution is not sensitive to these since most of the turbulence is generated in the boundary layers.

Boundary Conditions at Outlet
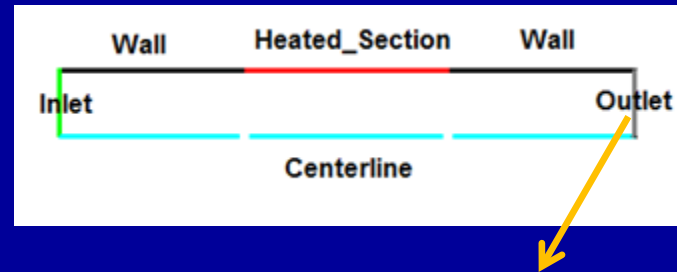
- Use "pressure outlet" boundary type.
- Need to input measured gauge pressure (baseline is "operating pressure").
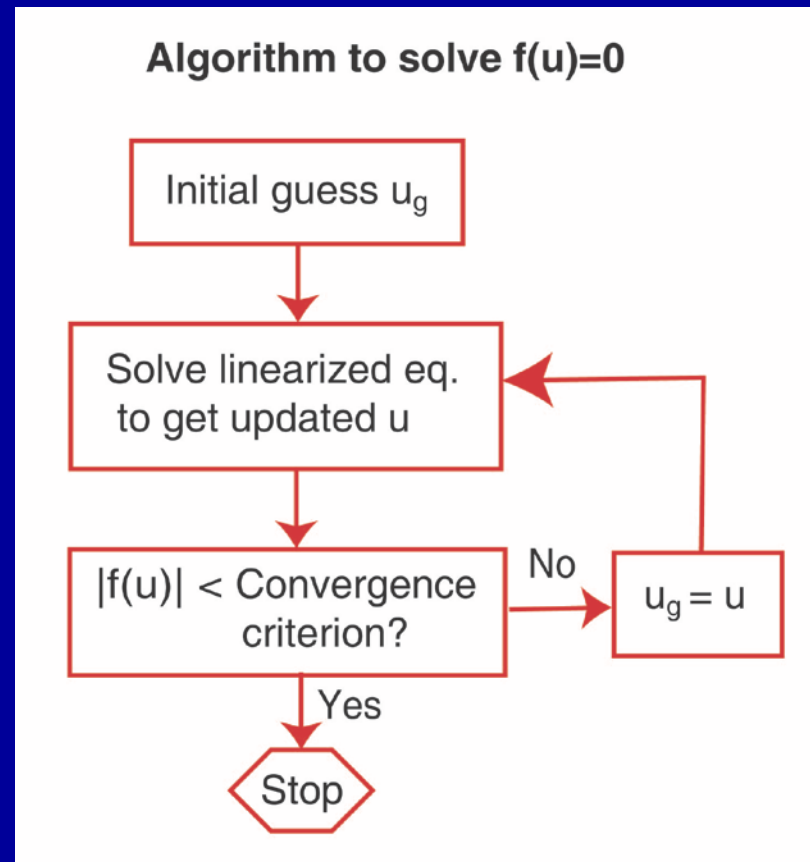
Boundary Conditions for Heated Section
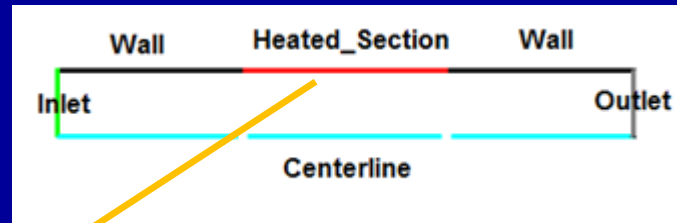
- Use "wall" boundary type.
- FLUENT imposes no-slip condition for velocity.
- Specify measured constant heat flux.
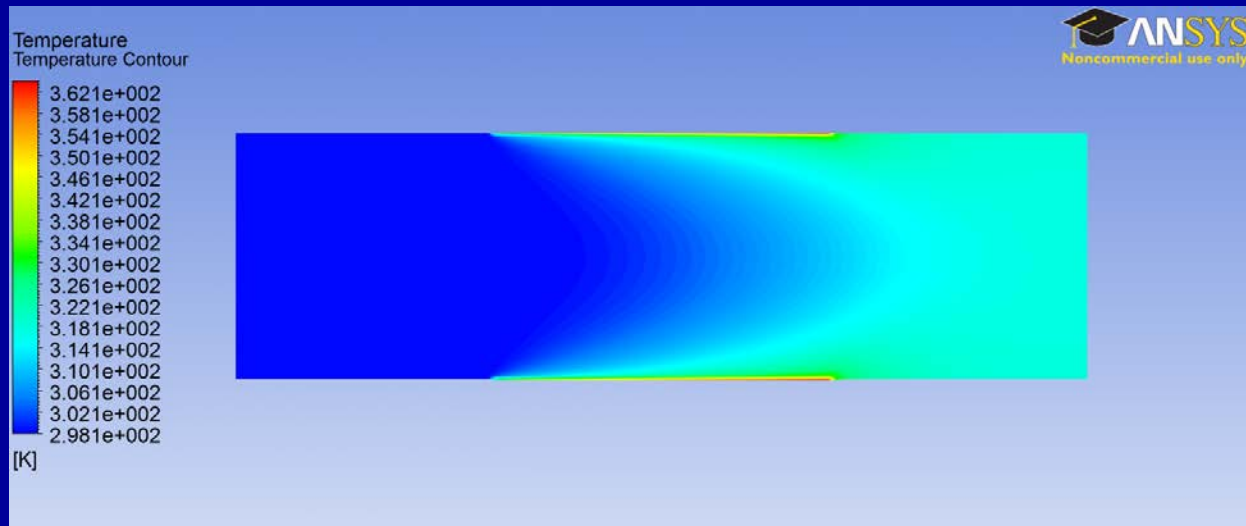- We'll neglect heat conduction within pipe wall

Solution

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

Algorithm to solve f(u)=0

Initial guess u_g

Solve linearized eq. to get updated u

$|f(u)| <$ Convergence criterion?

No → $u_g = u$

Yes

Stop

Results

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

Results

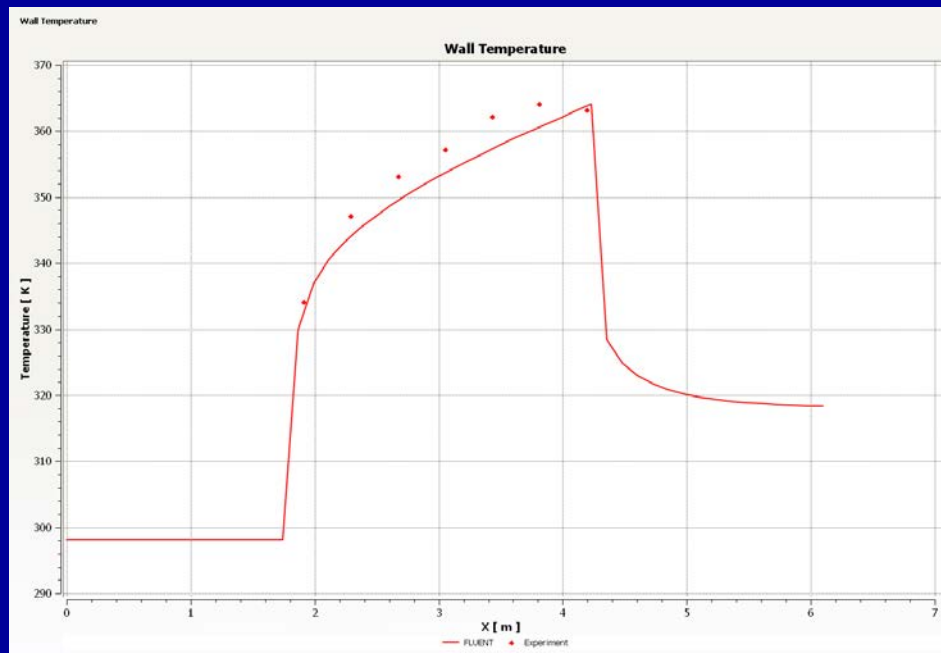- Velocity vectors in the first section: Do you see any flow development? Squint hard!
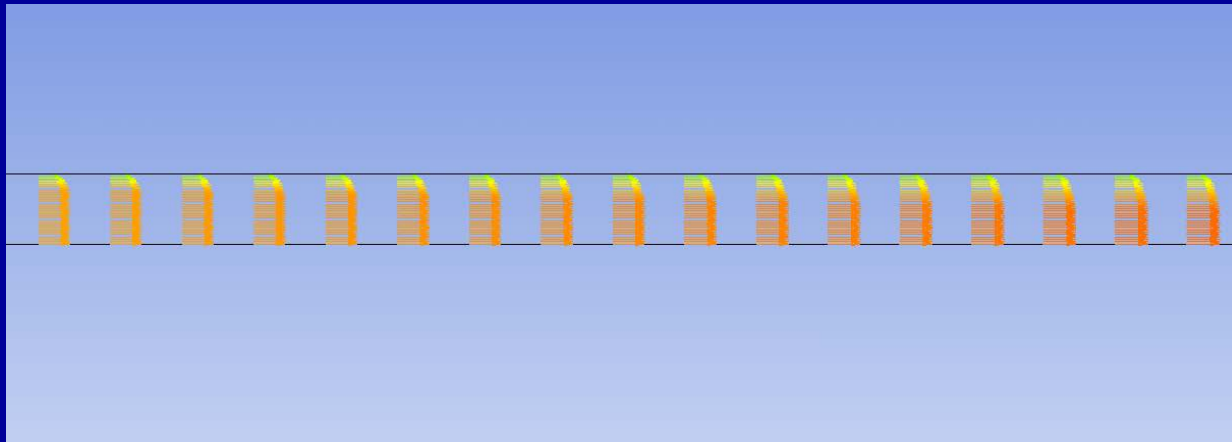
Results

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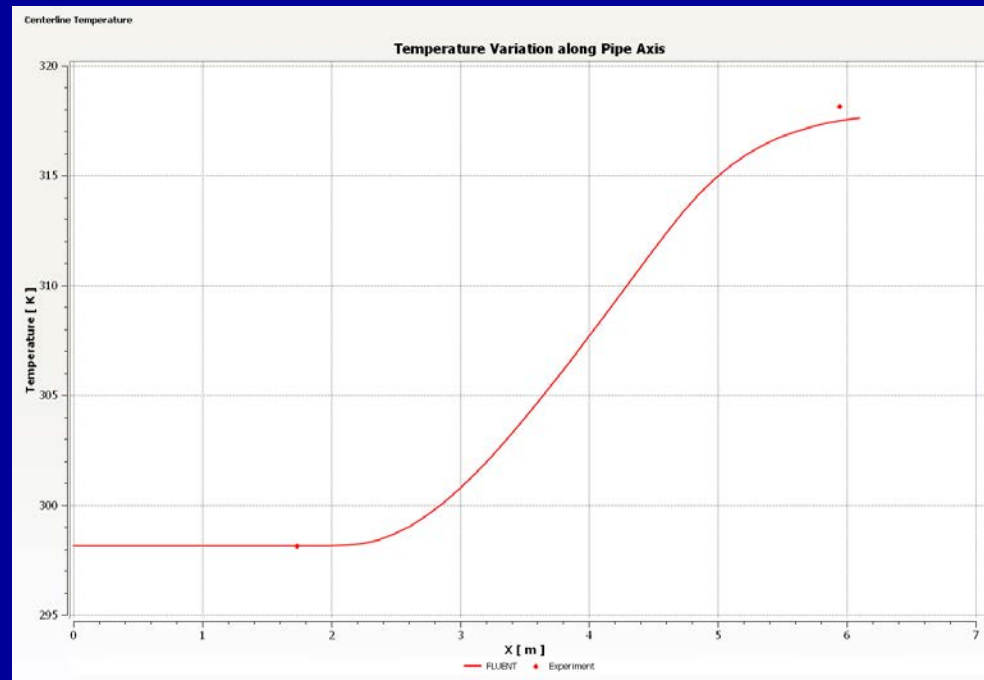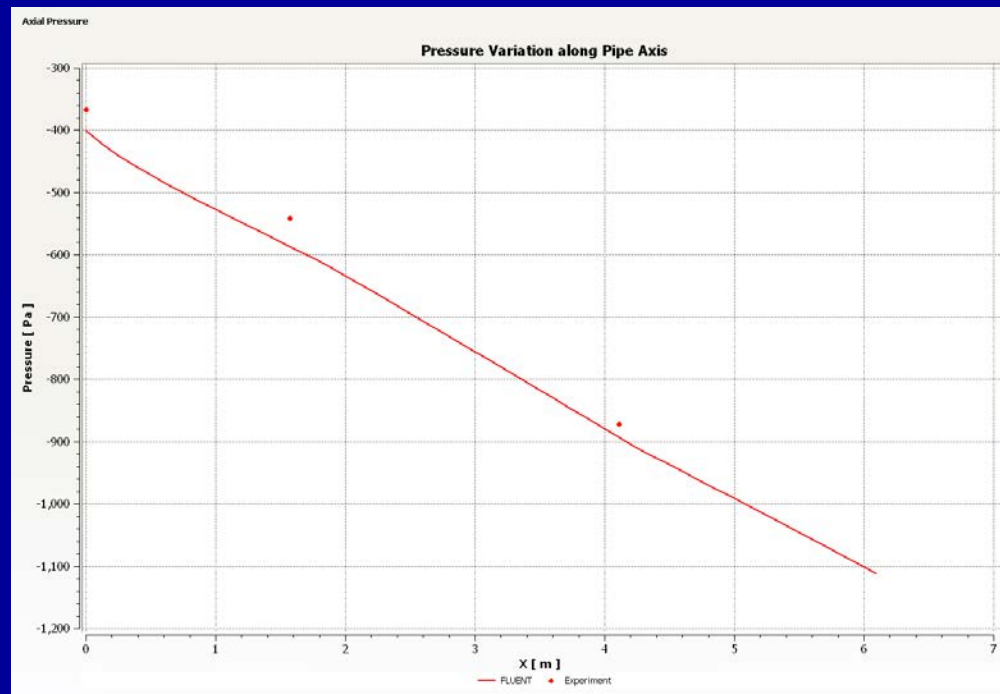
Results

- Wall temperature variation.
 - How does this match with your sketch?

Results

- Pressure variation along pipe axis.
 - How does this match with your sketch?

Results: Nu and f from FLUENT

- To calculate Nu and f from T and P, use same procedure as the experiment.

- The necessary T and P values can be obtained from the relevant FLUENT plots.

- *Export* button in post-processor exports plot data into an Excel file.

- Mean *Tw* should be calculated between the same locations as expt. (thermocouple locations 2-8).
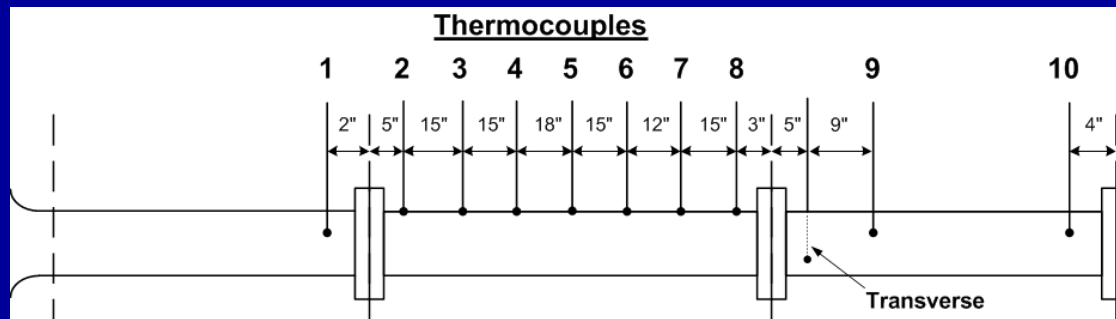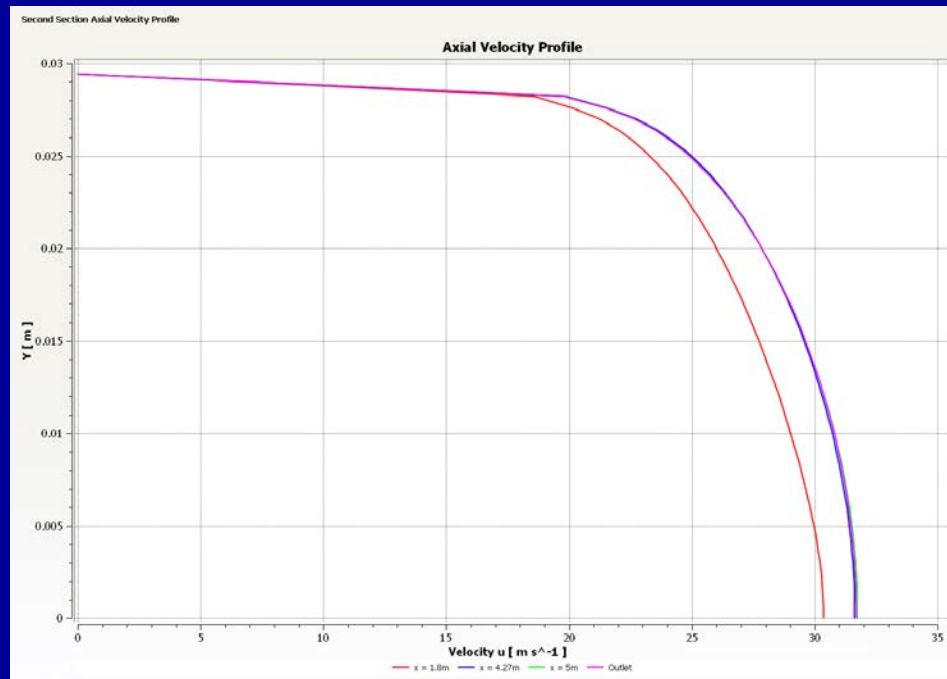
Results

- Axial velocity profiles in first section.
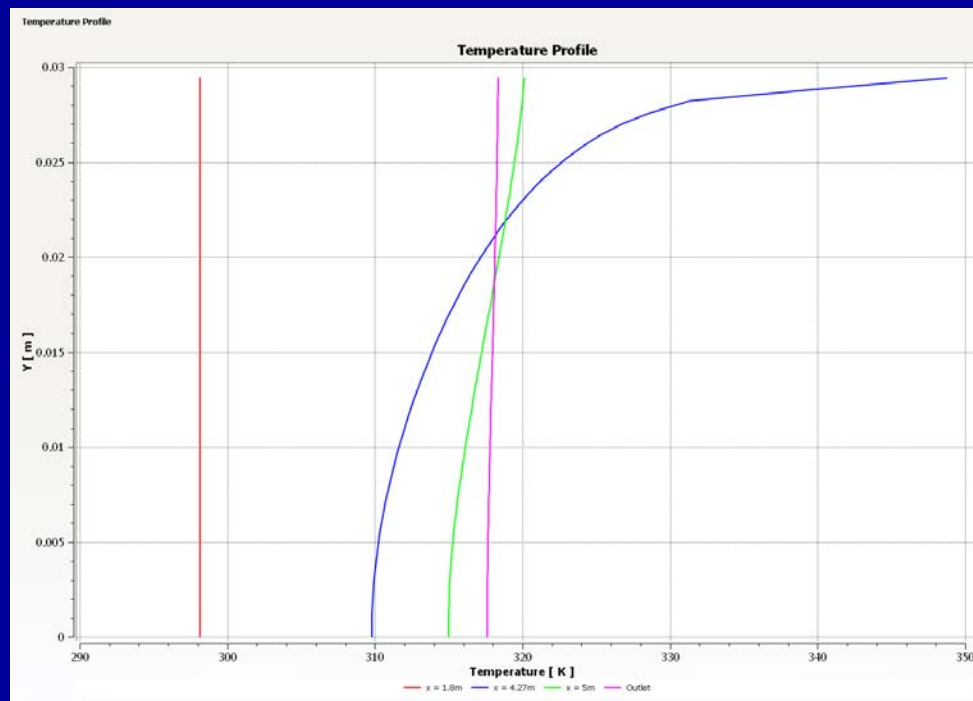 - Is the flow fully developed as it enters the heated section?
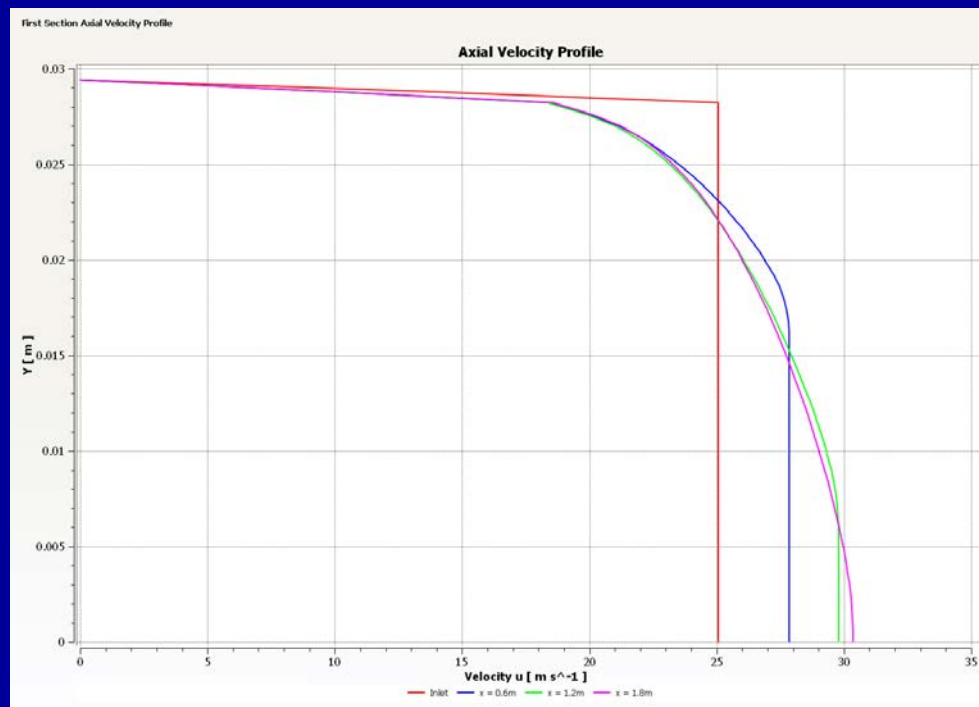
Results

- Axial velocity profiles in heated and mixing sections.

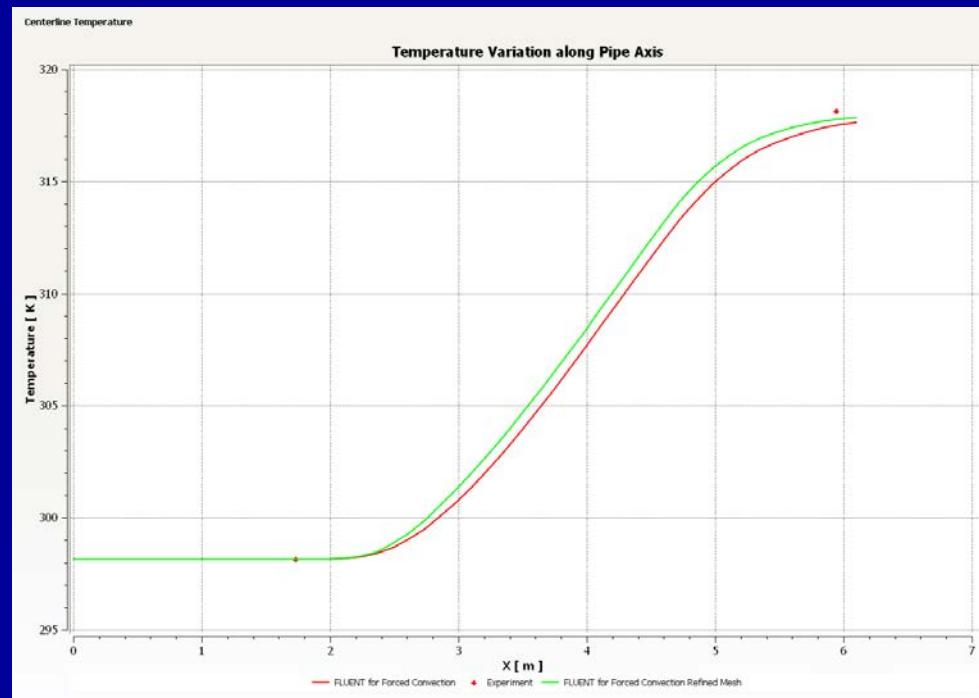 – Does flow accelerate or deccelerate with heating?

Results

- Temperature profiles at various locations.
  - How does temperature vary 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.

4. Include a summary of your FLUENT settings as an appendix in report. In FLUENT, select

   – Report > Input summary