FORCED CONVECTION SIMULATION FOR HT2 EXPERIMENT

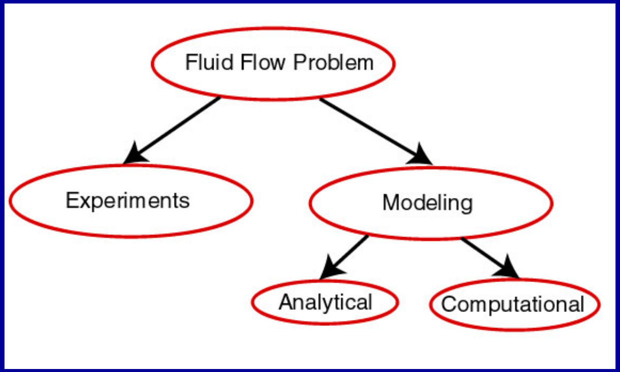
MAE 4272

Fall 2010

Mechanical & Aerospace Engineering

Cornell University

Experiment vs. Simulation



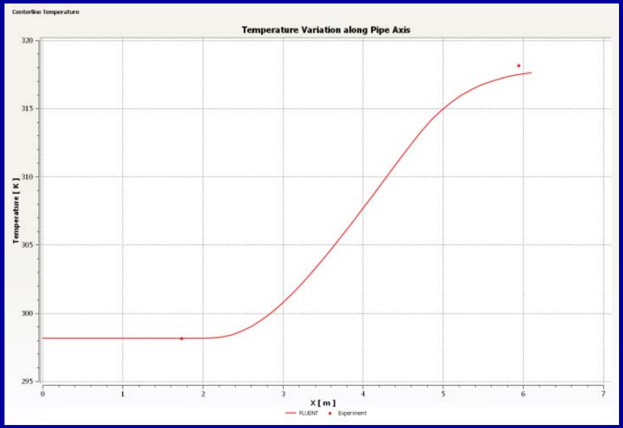
- Computational Fluid Dynamics (CFD): Computational modeling of fluid flow problems.
- Falls under the rubric of “Computer Simulation” or just “Simulation”.
- We’ll use FLUENT software for the HT2 simulation.
- FLUENT will be used to obtain an **approximate** solution to the governing equations.

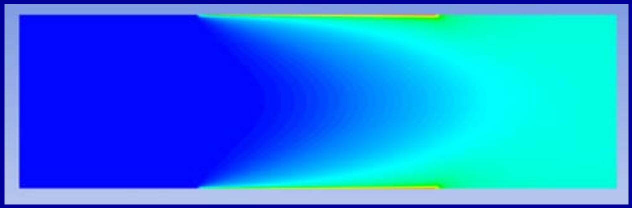
FLUENT Software

- 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, chemical reactions etc.
- No endorsement of FLUENT implied.
- Officially called *ANSYS FLUENT* ™.

Why Perform the CFD Simulation?

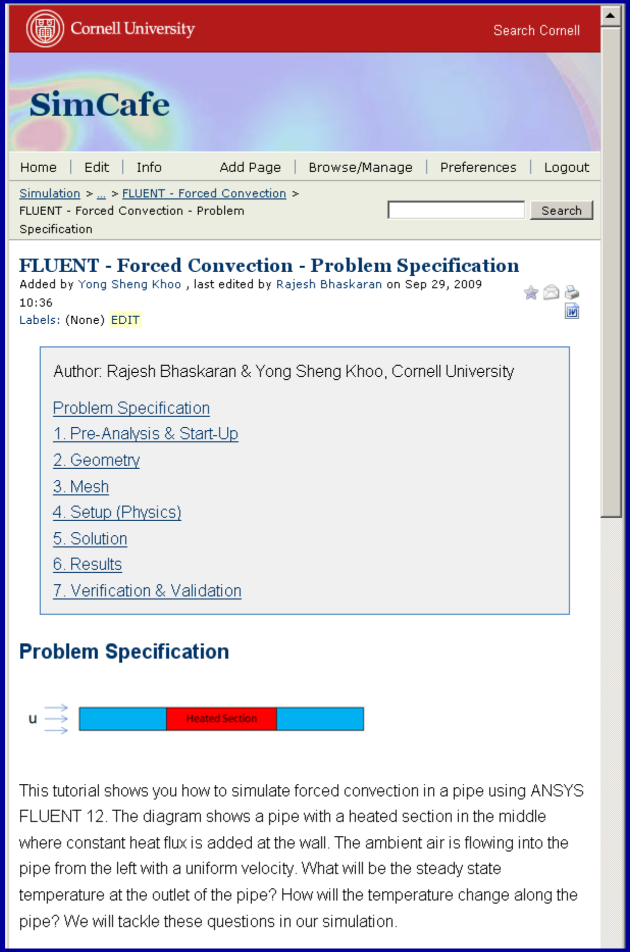
- We can
  - See how simulation can complement experiments.
  - Look “under the hood” using the simulation.
    - Get a better understanding of the flow than is possible from point measurements
    - Check assumptions made in post-processing experimental data
  - Get an overview of the CFD simulation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





CFD Simulation

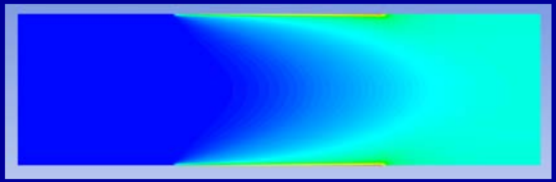
- A tutorial on how to apply FLUENT to simulate the forced convection experiment is available at: ***https://confluence.cornell.edu/display/simulation/forcedconvection***
- Run FLUENT and tutorial side-by-side.
- Skip geometry and mesh steps (mesh is provided).



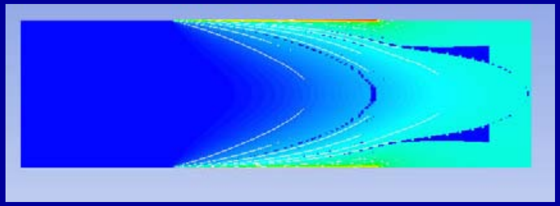
Computer Labs with FLUENT

- Use ACCEL lab in the Engineering Library.
  - See lab manual for details.
- CIT labs in B7 Upson and 318 Phillips also have FLUENT.
  - Video card incompatibility with FLUENT.
  - Temperature contours can look weird.

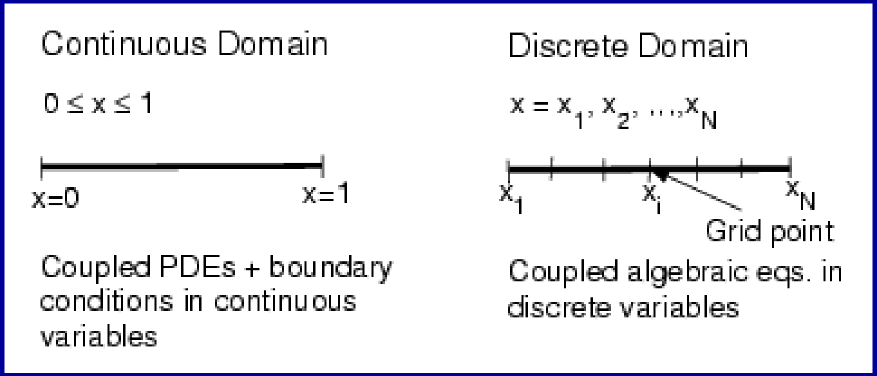
No Video Card Incompatibility
(ACCEL Lab)



Video Card Incompatibility
(CIT Labs)



Strategy of CFD

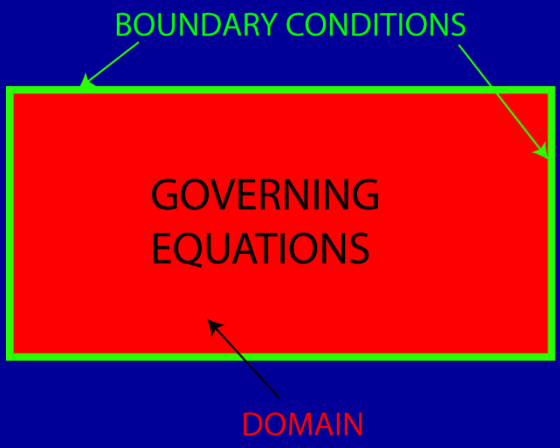


- Eg.: Continuous Domain: $p = p(x)$, $0<x<1$
  Discrete Domain: $p_i = p(x_i)$, $i=1,2,\ldots,N$
- Truncation error introduced. Can be reduced by refining the mesh.
- Mesh refinement study required to assess the level of truncation error.

CFD Simulation

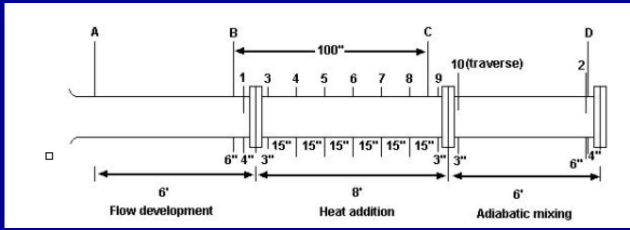
- We'll be using FLUENT to solve a boundary value problem.
- We need to specify the governing equations, boundary conditions and domain in FLUENT.
- FLUENT will obtain an approximate numerical solution to the BVP.

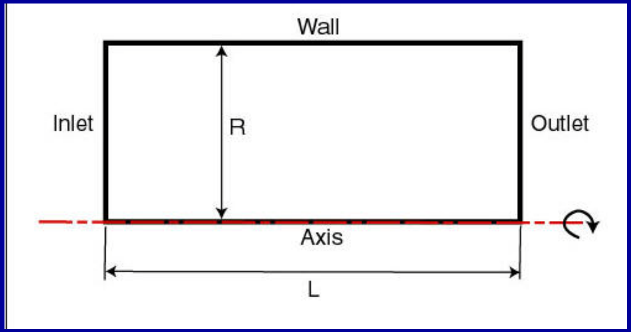
Boundary Value Problem



Domain

- Length of pipe included in the simulation: From A to D.
  - Note corrections to positions of pressure taps – see Blackboard and lab handout.
- Assume flow is axisymmetric. Hence, domain is rectangular.
- Rotate the rectangle 360º about the axis to get the full pipe geometry.
- Solve axisymmetric form of the governing equations.



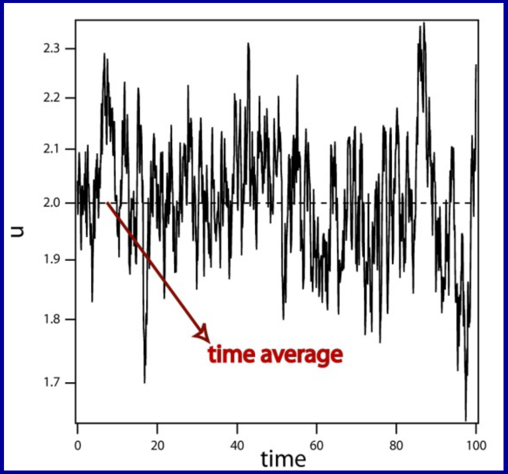


Governing Equations

- 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: $u' = u - \overline{u}$
- 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

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

- 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 $$\rho = \frac{P_{ref}}{RT}$$
- 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

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

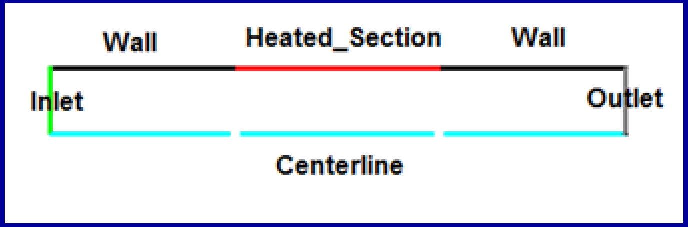
Equation	Dependent variables
Continuity eq. in cylindrical coordinates (Reynolds-averaged)	$\overline{u}$, $\overline{v}$, $\overline{\rho}$
Axial momentum eq. (Reynolds-averaged)	$\overline{u}$, $\overline{v}$, $\overline{p}$, $\overline{\rho}$, k, ε
Radial momentum eq. (Reynolds-averaged)	$\overline{u}$,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

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

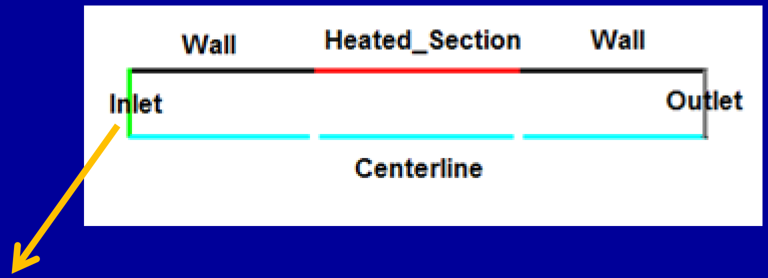
- Boundaries are labeled as follows.



- 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.

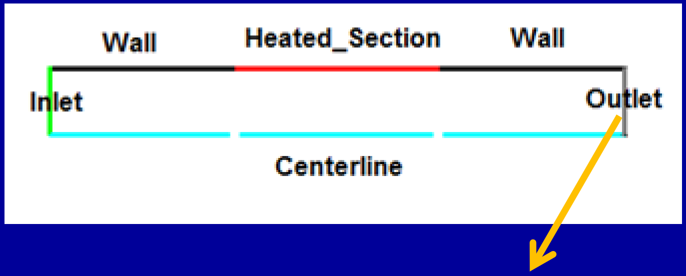
$\bar{\rho}$

Boundary Conditions at Inlet



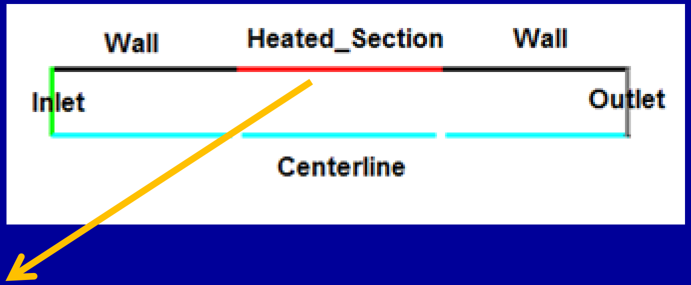
- Use “velocity inlet” boundary type.
- Assume uniform flow in axial direction at inlet.
  - Calculate velocity magnitude from measured mass flow rate.
- Temperature is measured value at inlet.
- Inlet values *k* and *ε* are wild guesses.
  - Solution is not sensitive to these since most of the turbulence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

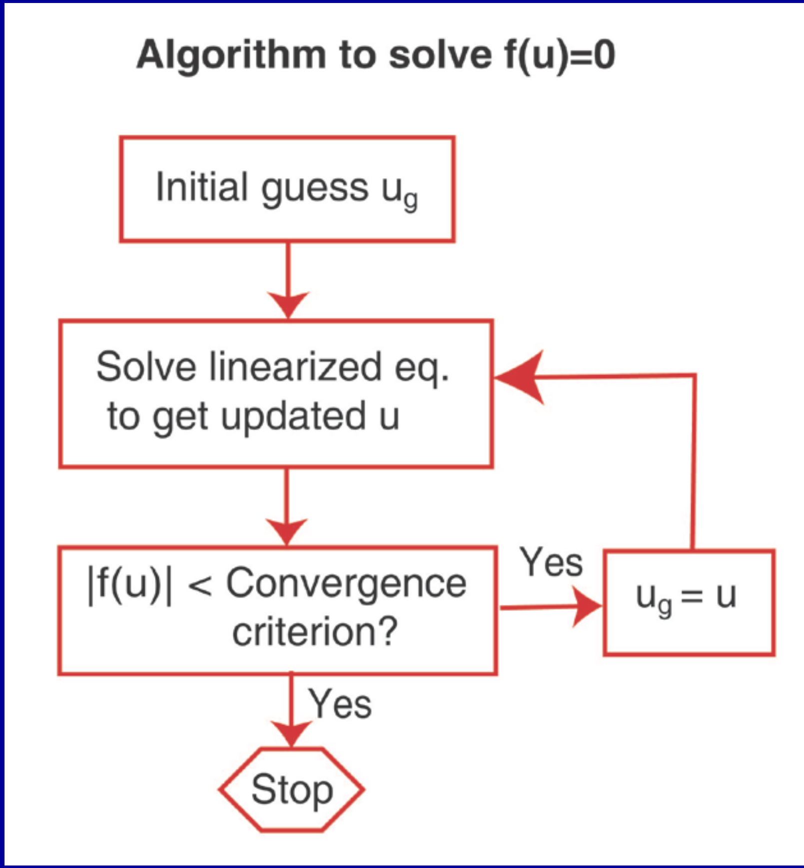
Boundary Conditions for Heated Section



- Use "wall" boundary type.
- FLUENT imposes no-slip condition for velocity.
- Specify measured constant heat flux.
- We'll neglect heat conduction within pipe wall

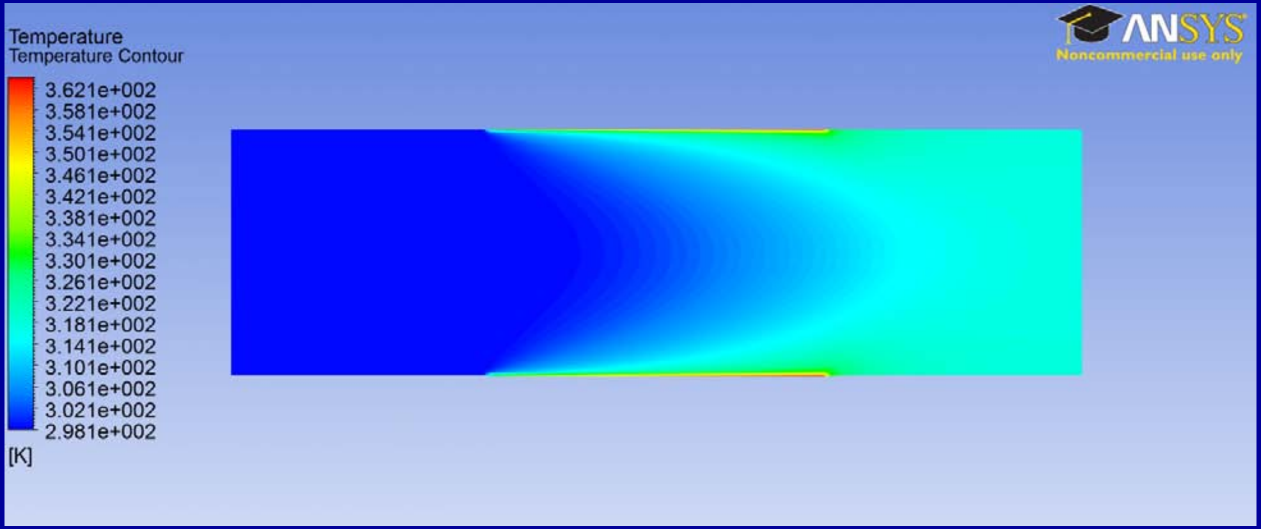
Solution

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



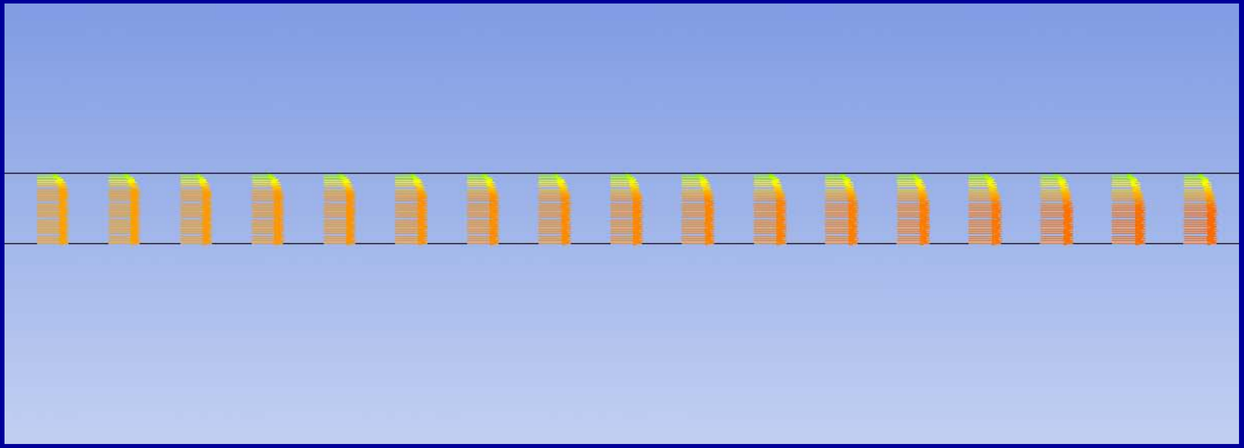
Results

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



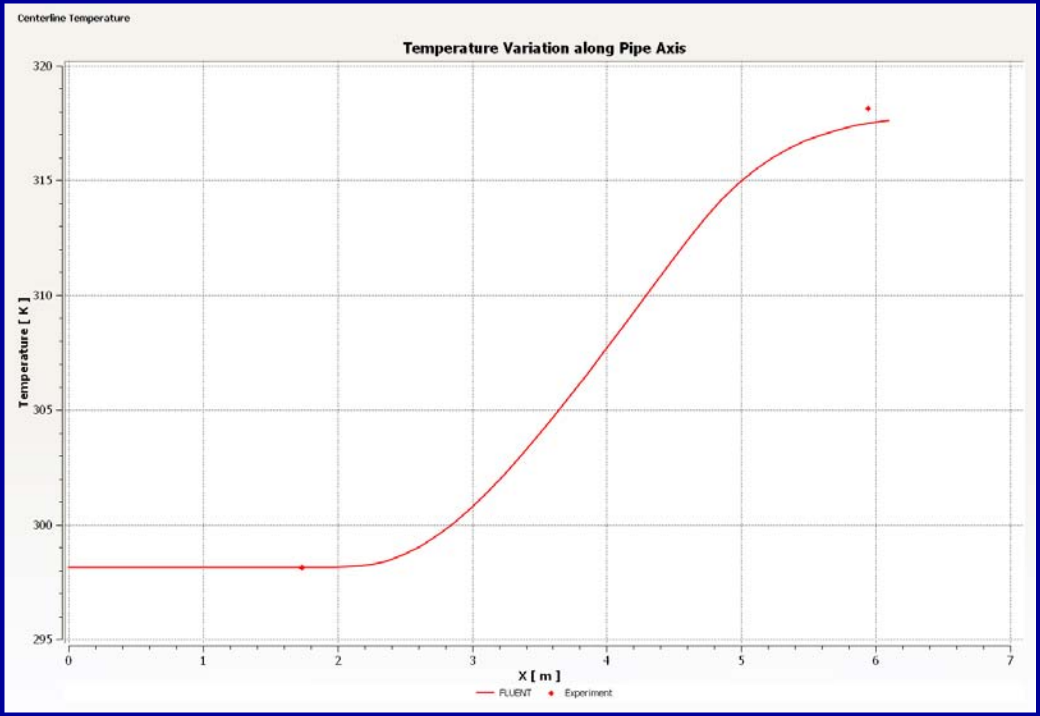
Results

- Velocity vectors in the first section showing flow development.



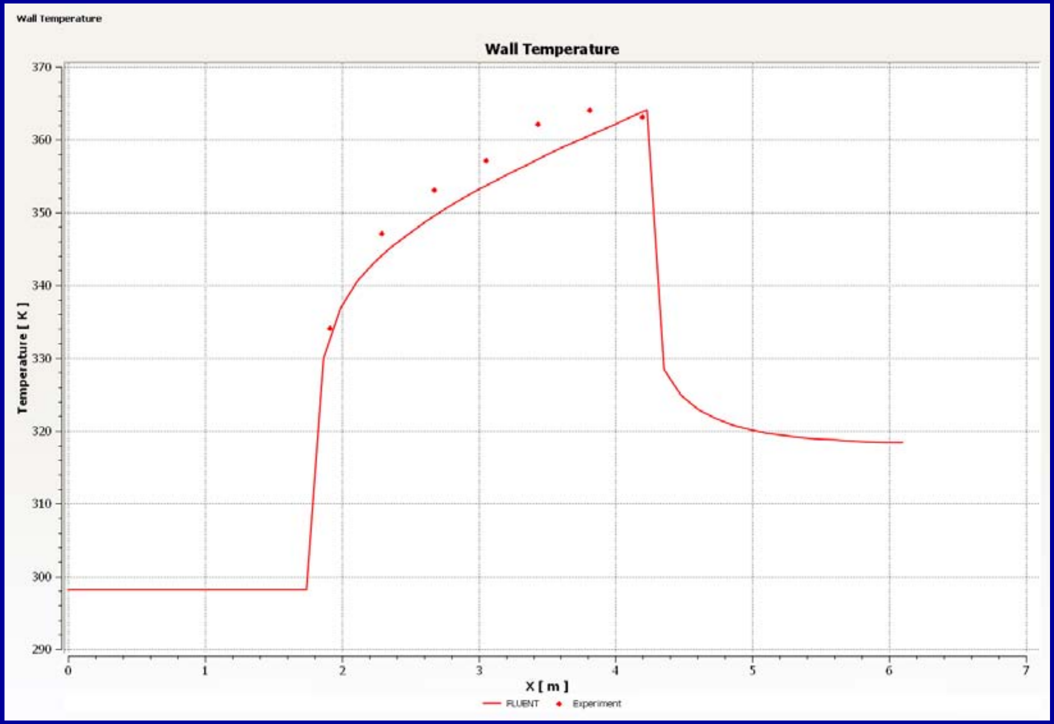
Results

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



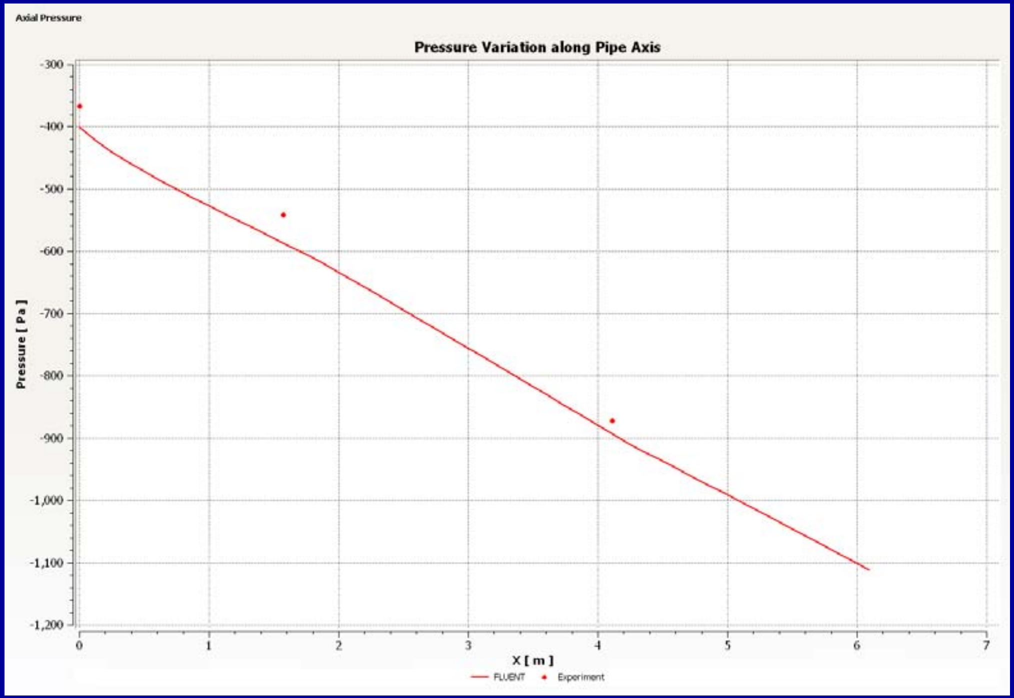
Results

- Wall temperature variation. Shows thermal entrance length effects.



Results

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

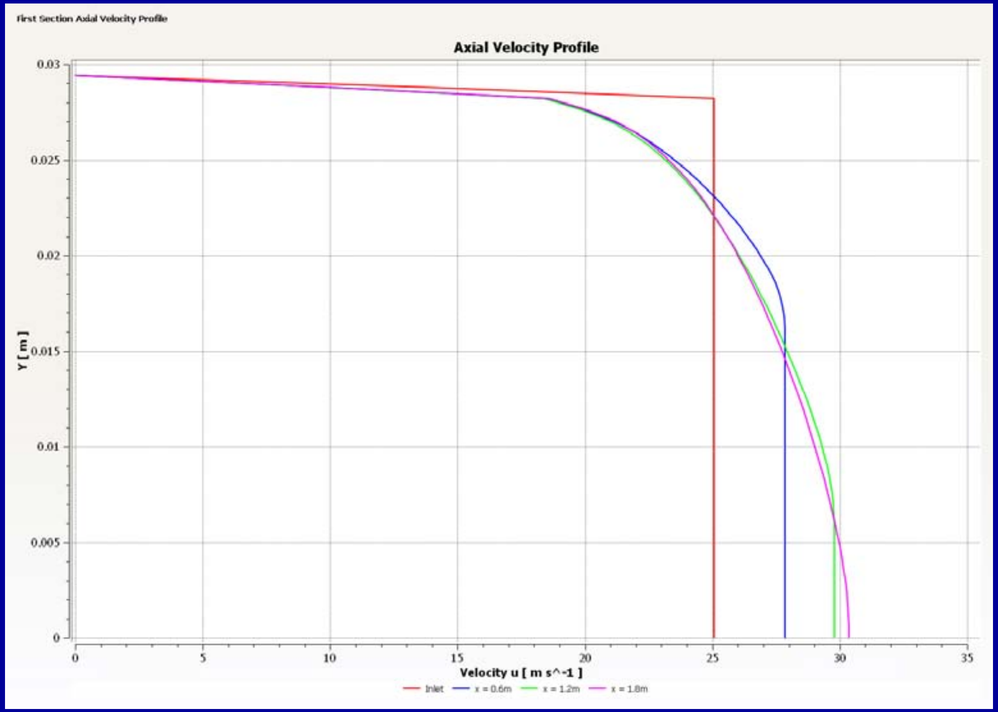


Results: Nu and f from FLUENT

- 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 (thermocouple locations 3 and 9) as the experiment.

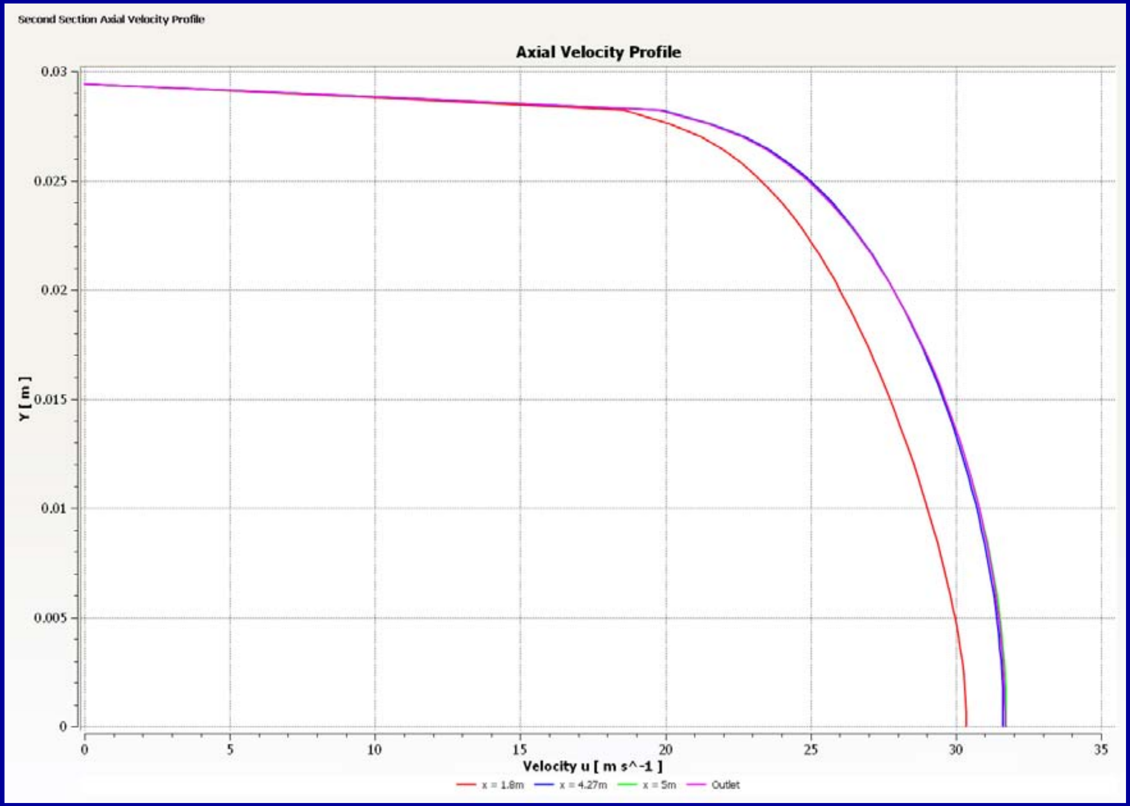
Results

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



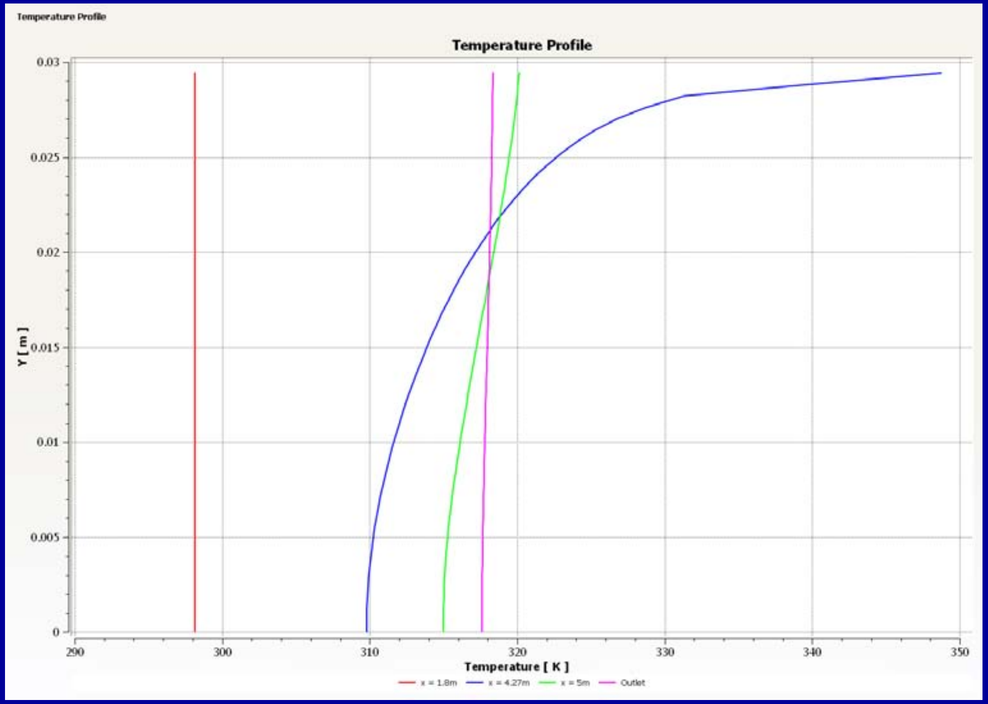
Results

- Axial velocity profiles in heated and mixing sections. How does the flow velocity change in heated section?



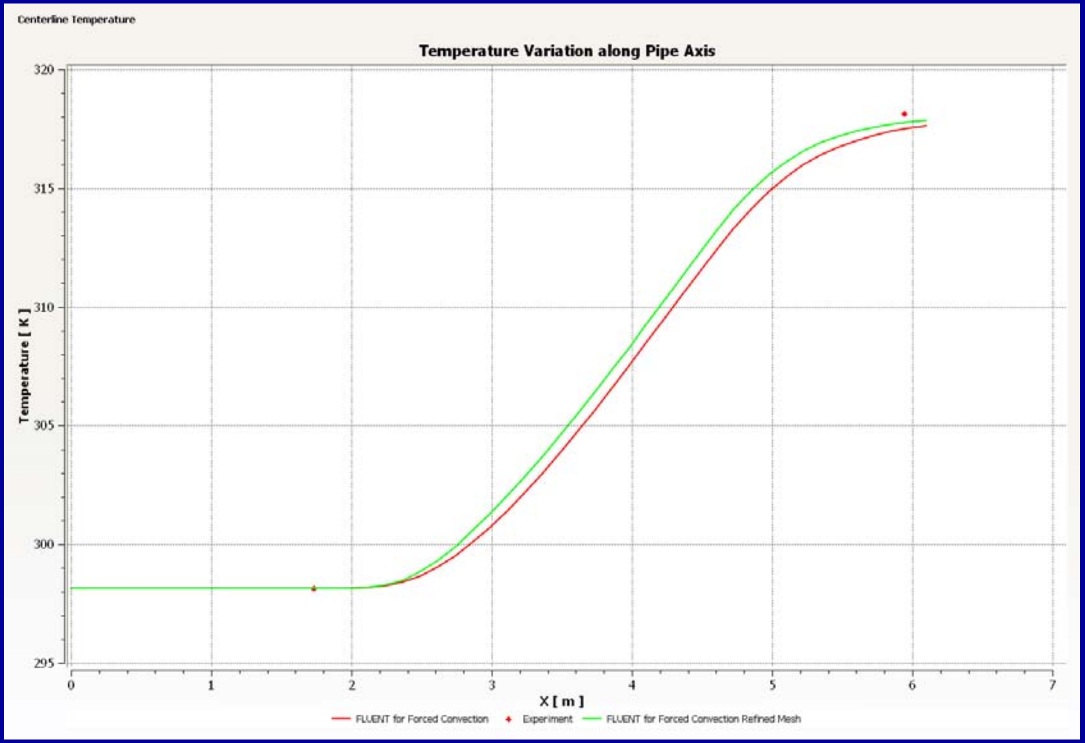
Results

- 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