FORCED CONVECTION SIMULATION FOR HT2 EXPERIMENT
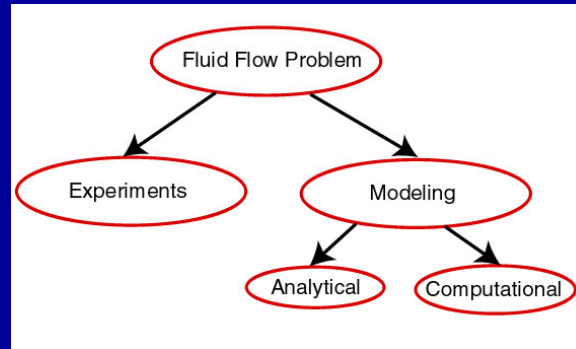
MAE 4272

Fall 2015

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.
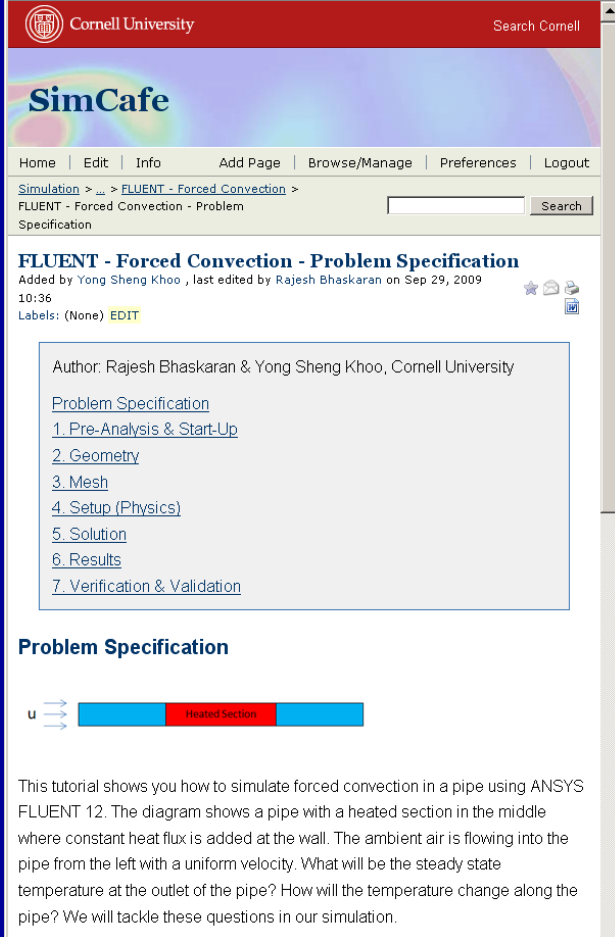   - "Garbage in, garbage out"
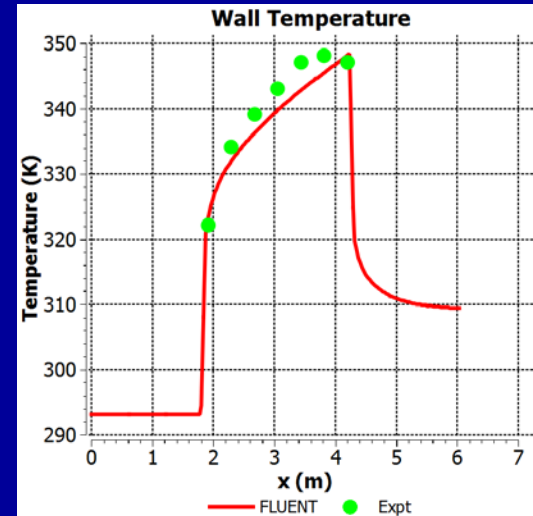
Temperature Distribution
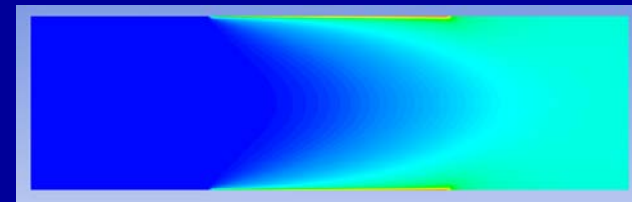
CFD Simulation

- A tutorial on how to apply FLUENT to simulate the forced convection experiment is available at: *https://confluence.cornell.edu/x/IxuhBg*

- Run FLUENT and tutorial side-by-side.

- Skip geometry and mesh steps (mesh is provided).

Accessing ANSYS Fluent

- Current version: 16.1
- Download student version for free at http://www.ansys.com/student
  - Has most modeling capabilities with node/cell limits: 32k for FEA and 512k for CFD
  - Mac users need Parallels Desktop (see https://confluence.cornell.edu/x/OY1nDg)
- CIT public labs (512k cell limit for CFD)
  - B7 Upson
  - 318 Phillips
  - ACCEL lab in Carpenter Hall

Strategy of CFD

Continuous Domain

$0 \le x \le 1$

x=0   x=1

Coupled PDEs + boundary conditions in continuous variables

Discrete Domain

$x = x_1, x_2, ..., x_N$

x_1 x_i x_N

Grid point

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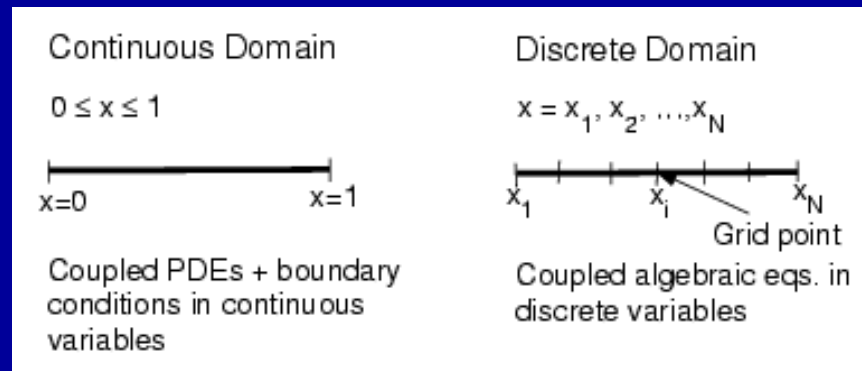
- Eg.: Continuous Domain: $p = p(x)$, $0<x<1$
  Discrete Domain: $p_i = p(x_i)$, $i=1,2,…,N$
- Truncation error introduced. Can be reduced by refining the mesh.
- Mesh refinement study required to assess the level of truncation error.

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
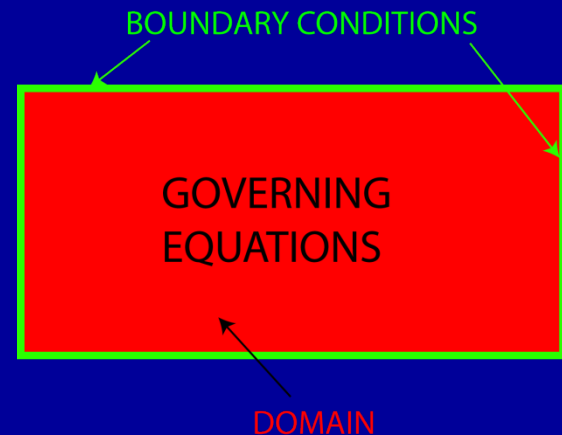
DOMAIN

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.
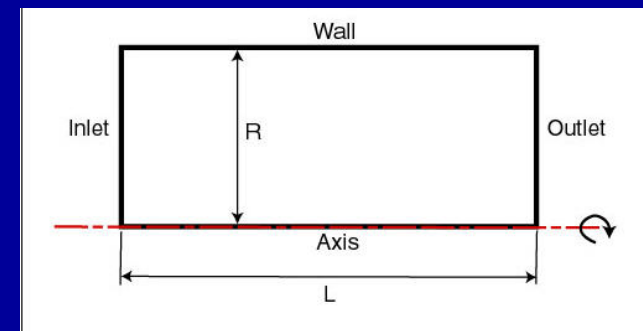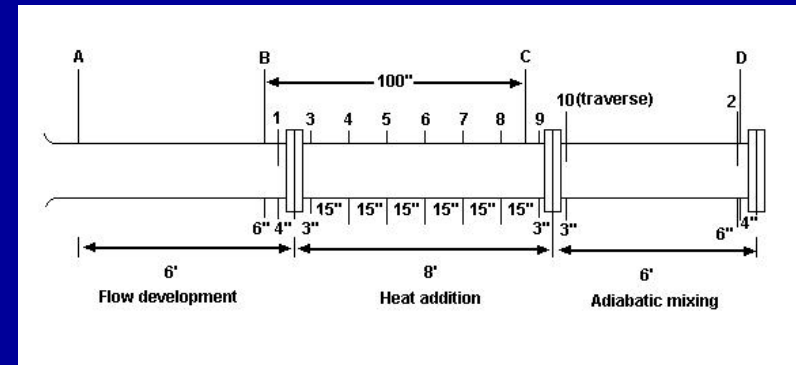
Short Exercise: Pre-Analysis Step

- *Prior to* performing a CFD simulation, you should make back-of-the-envelope estimates of expeted results.
  - Use these to check CFD results
  - Helps avoid "garbage in, garbage out"
- *Sketch* expected results for the following:
  - Centerline temperature variation in axial direction
  - Wall temperature variation in axial direction
  - Centerline pressure variation in axial direction
  - Axial velocity vs. r at inlet and end of flow development section (cold and hot flow)

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
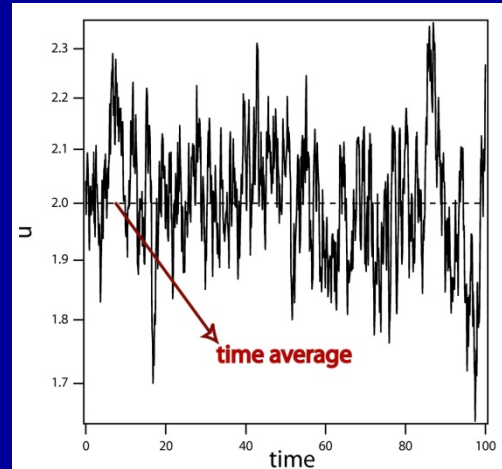
Turbulence

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

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

- 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 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	$\cancel{\bar{p}}$, $\bar{T}$, $\bar{\rho}$
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
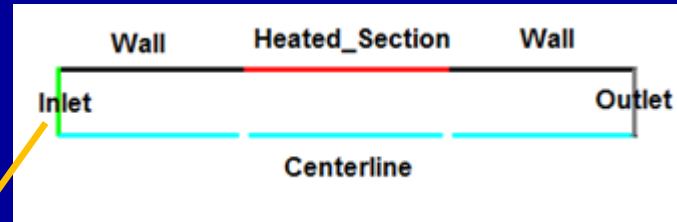
Boundary Conditions

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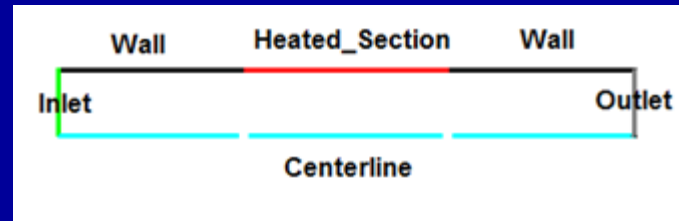
- FLUENT provides a variety of boundary types: "velocity inlet", "pressure outlet" etc.
- For each labeled boundary, you have to pick the appropriate boundary type and then input the settings (velocity, pressure etc). for that boundary type.

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*
  - Solution is not sensitive to these since most of the turbulence is generated in the boundary layers.

Boundary Conditions at Outlet
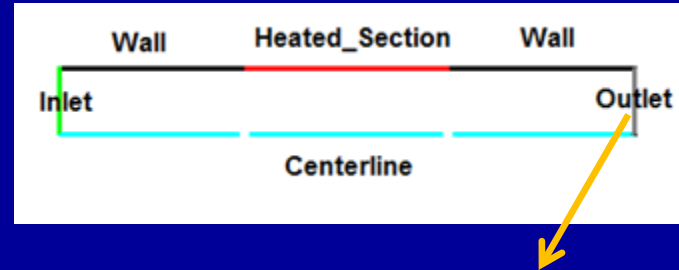
- Use "pressure outlet" boundary type.
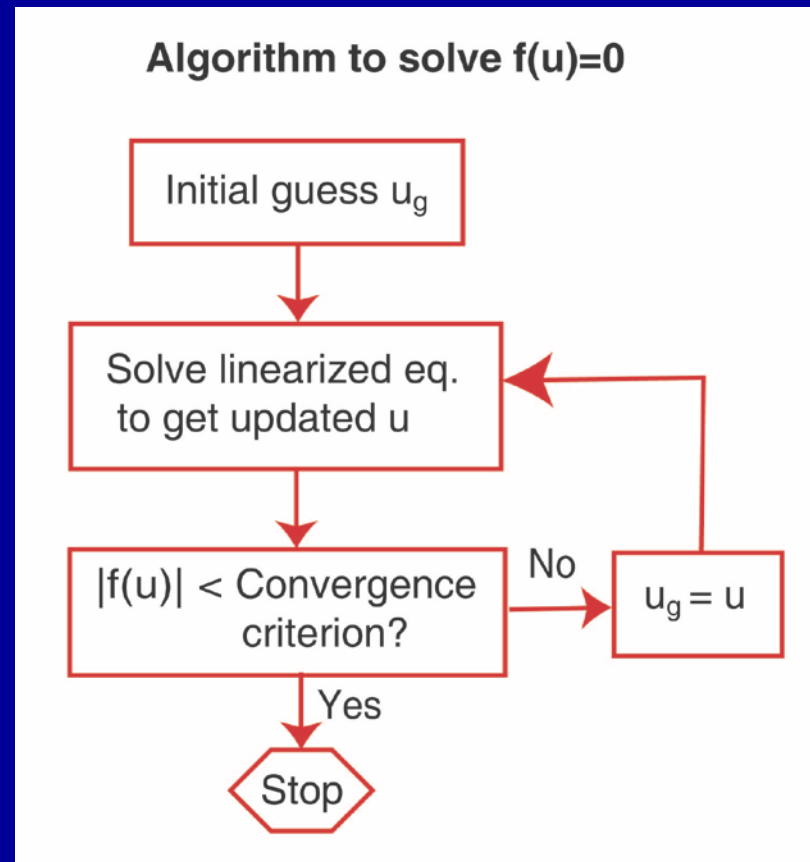- Need to input measured gauge pressure (baseline is "operating pressure").

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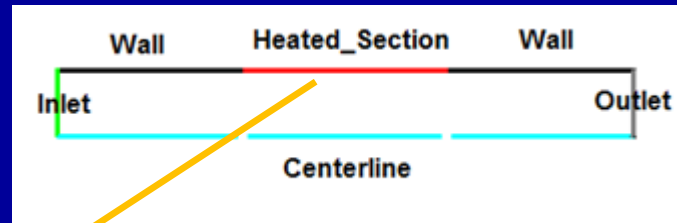
- Use "wall" boundary type.
- FLUENT imposes no-slip condition for velocity.
- Specify measured constant heat flux.
- We'll neglect heat conduction within pipe wall.

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?

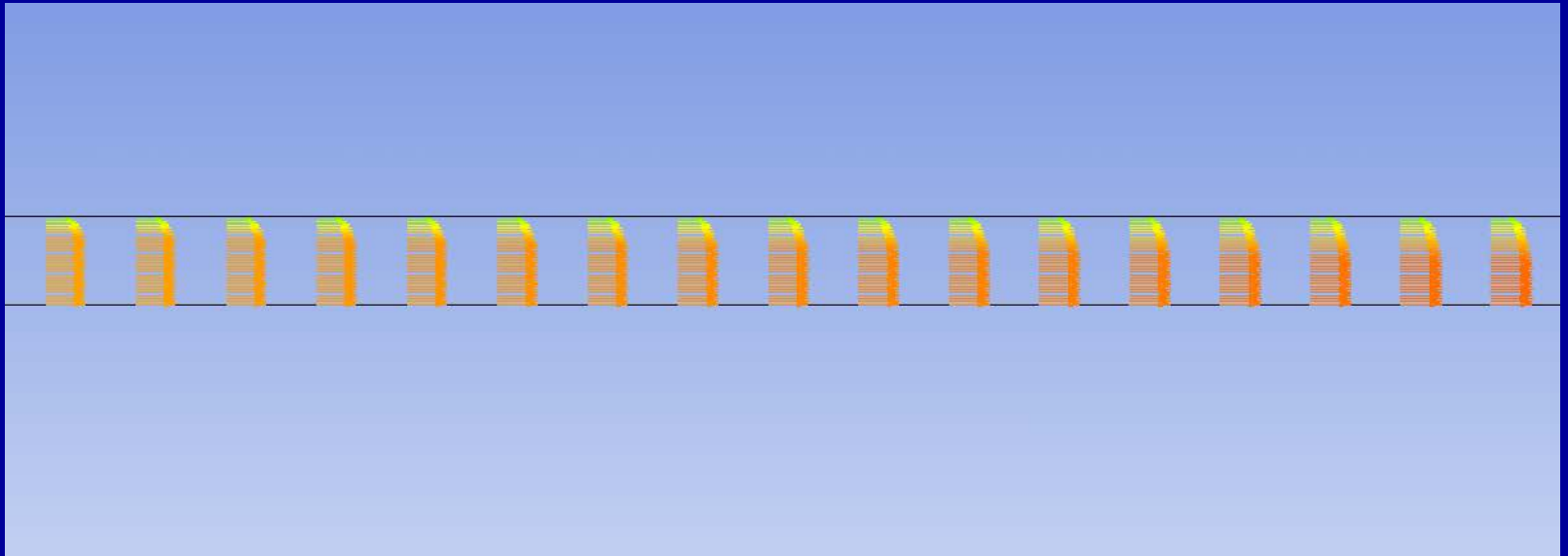No → $u_g = u$

Yes → Stop

Results

- Temperature contours:
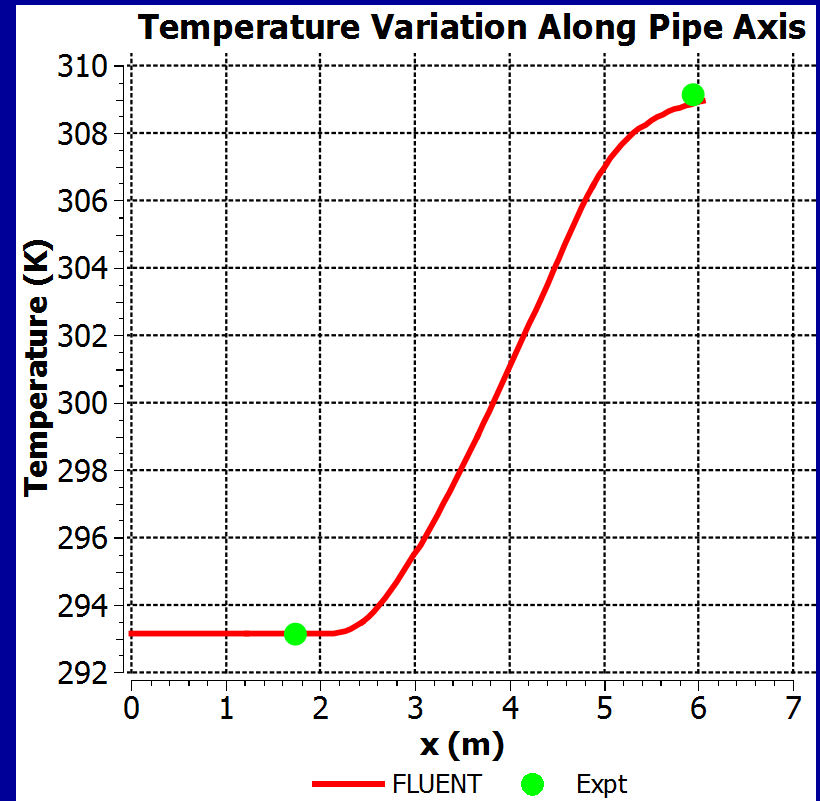 - Is the flow well-mixed at the end of the adiabatic mixing section as assumed in the experiment?
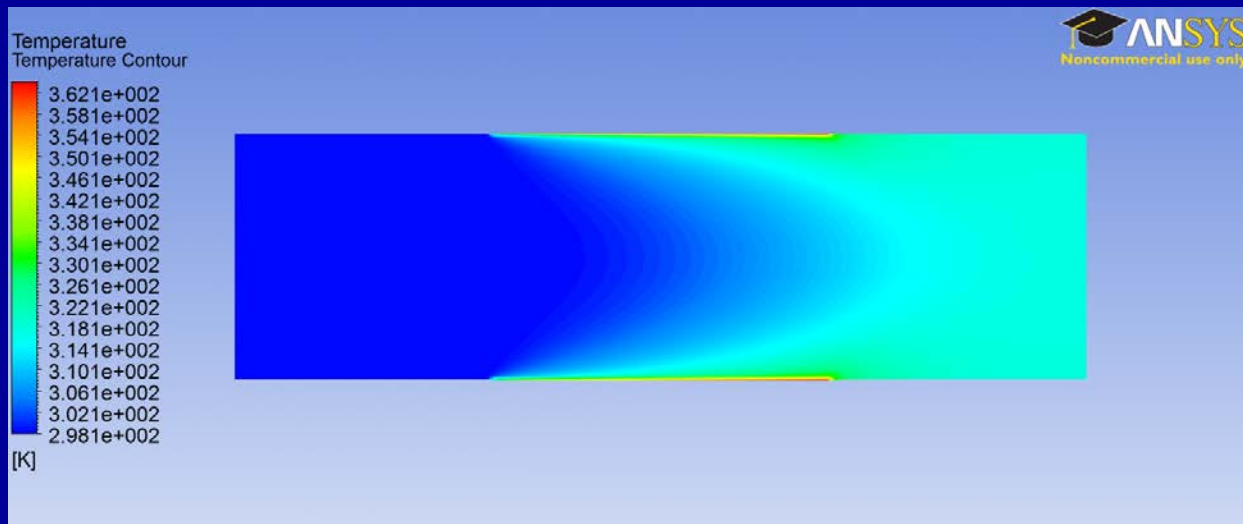
Results

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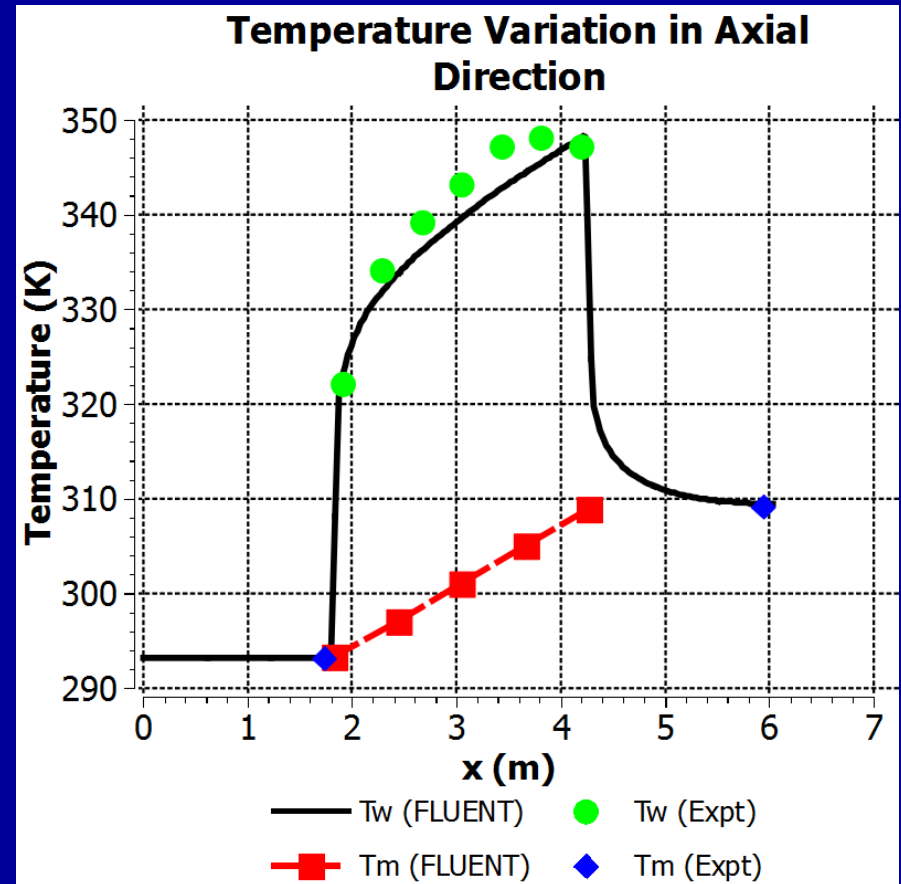 - Do you see any flow development? Squint hard!

Results

- Temperature variation along pipe axis.
  - Symbols represent experimental values.
  - How well does this match your sketch?

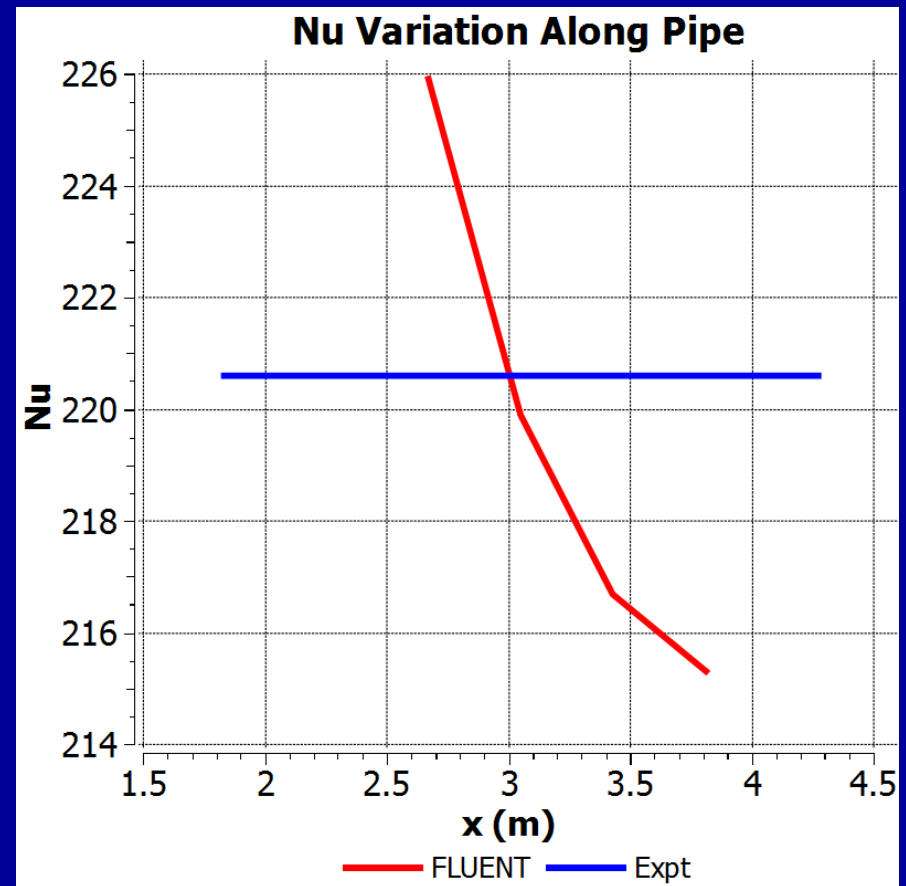Temperature Variation Along Pipe Axis

Results

- Axial variation of wall and mixed mean temperatures.
  - How well does the wall temperature variation match your sketch?

Results

- Axial variation of Nusselt number.

 - Average value of Nu from simulation is close to the experimental value

Nu Variation Along Pipe

Legend: FLUENT (red), Expt (blue)

Axes: x (m) from 1.5 to 4.5; Nu from 214 to 226

Results

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

Pressure Variation Along Pipe Axis — Pressure (Pa) vs x (m). FLUENT (red line) and Expt (green dots).
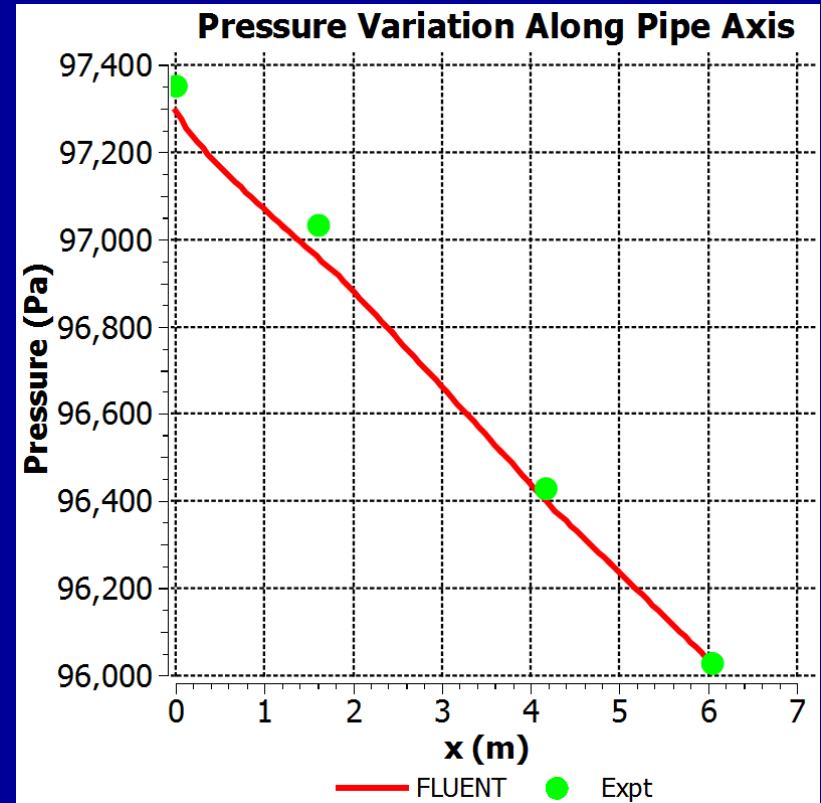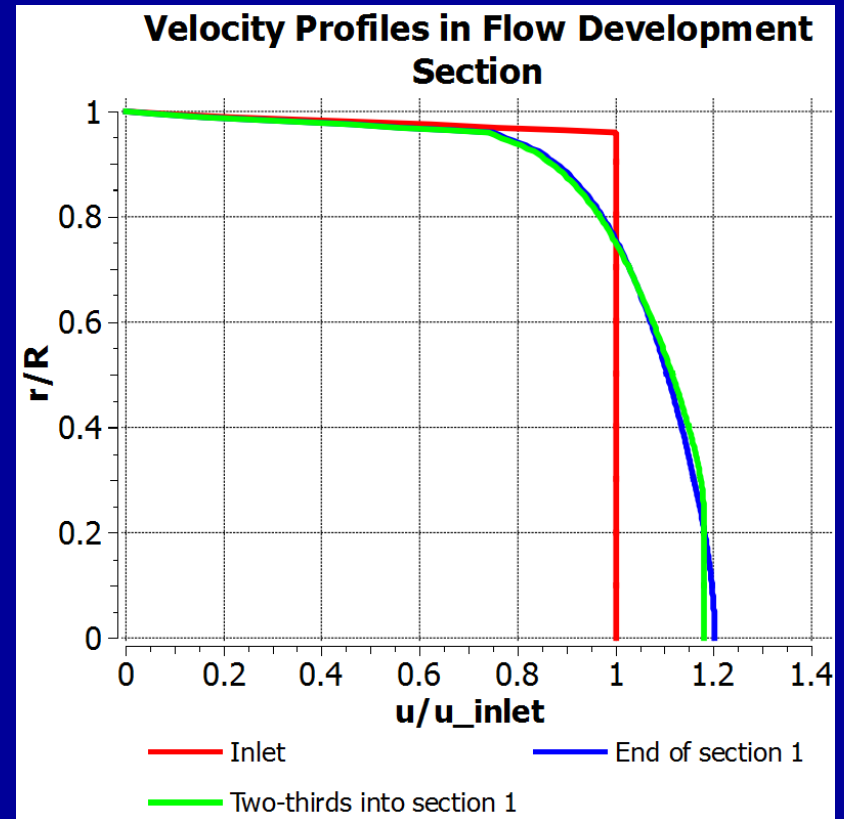
Results

- Axial variation of Fanning friction factor.
  - It is challenging to predict this accurately in turbulent flow simulations due to the steep velocity gradients at the wall
  - In our simulations, these steep gradients are not resolved but calculated using standard wall functions to save computation time


Cf Variation Along Pipe Wall
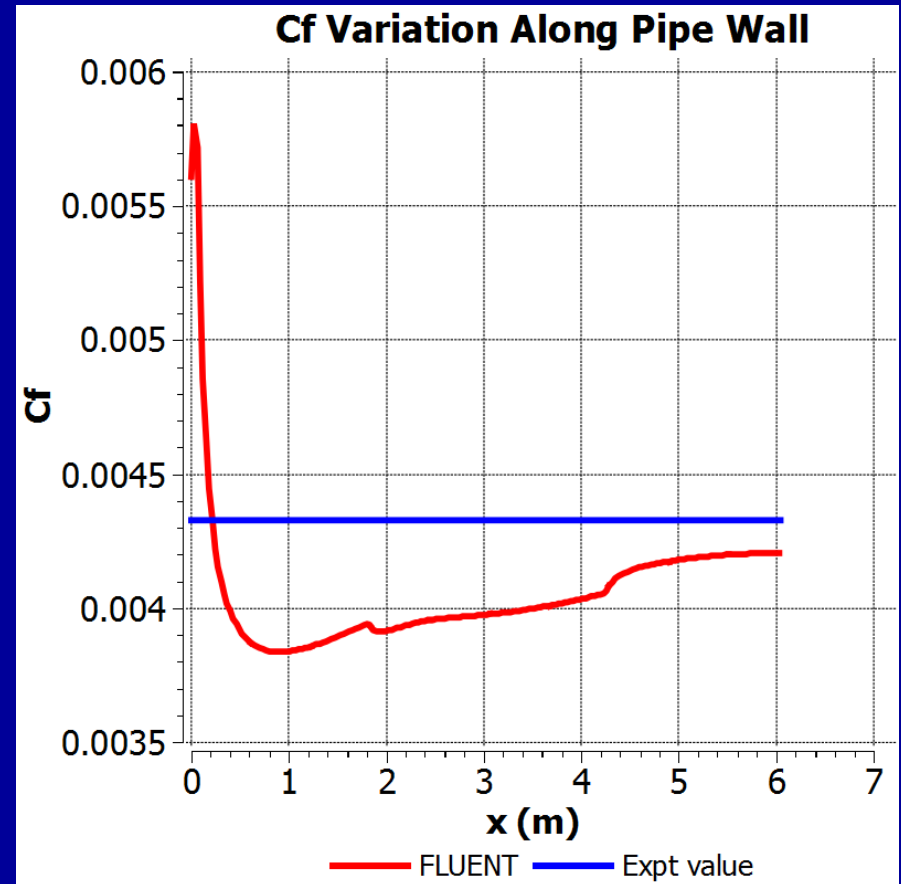
Results

- Axial velocity profiles in first section.
  - Is the flow fully developed as it enters the heated section?
  - Note steep gradient at the wall typical of turbulent flows.


Velocity Profiles in Flow Development Section

Results

- Non-dimensionalized axial velocity profiles in heated and mixing sections.

  – Does flow accelerate or deccelerate with heating? Why?

Velocity Profiles
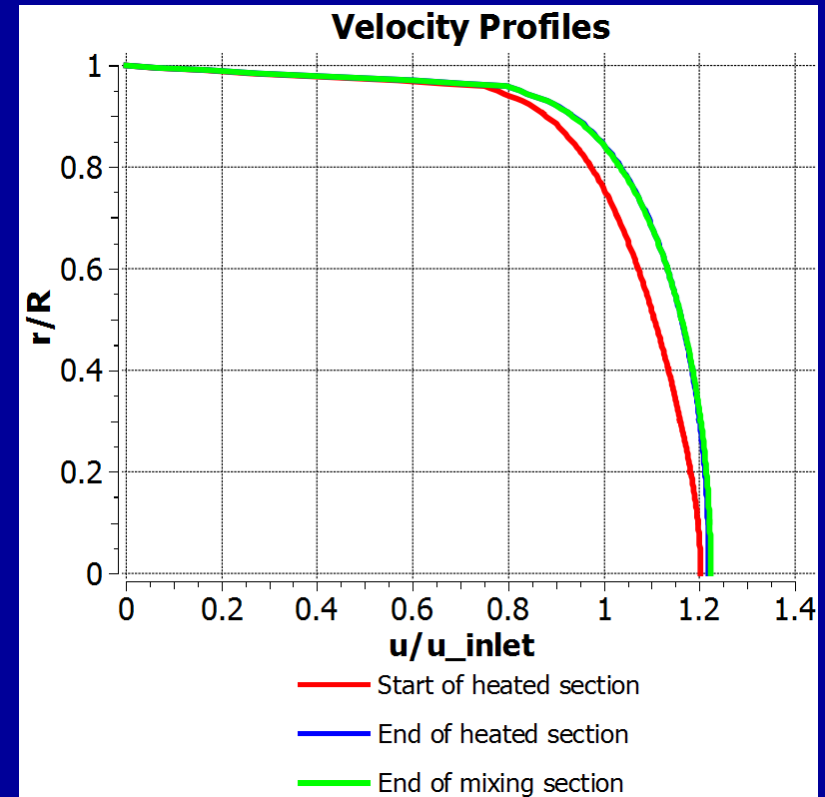
Results

- Temperature profiles at various locations.
  - How does temperature vary at the outlet (end of the mixing section)?

Temperature Profiles

Legend:
- Start of heated section (red)
- End of heated section (blue)
- End of mixing section (green)
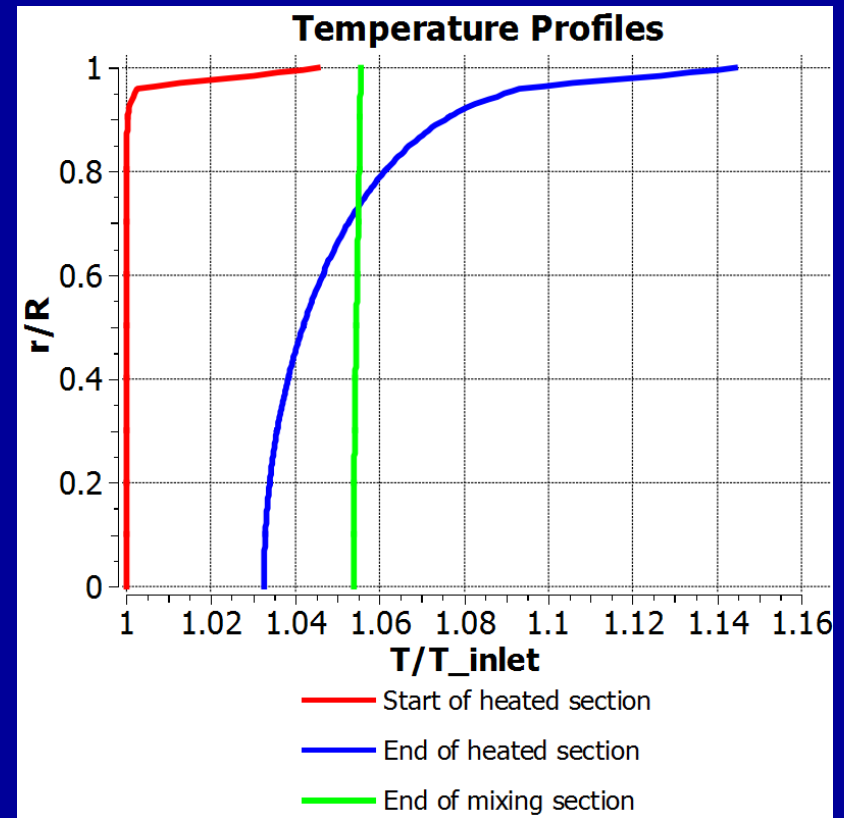
Verification

- Re-do solution on a refined mesh to check effect of mesh on solution.
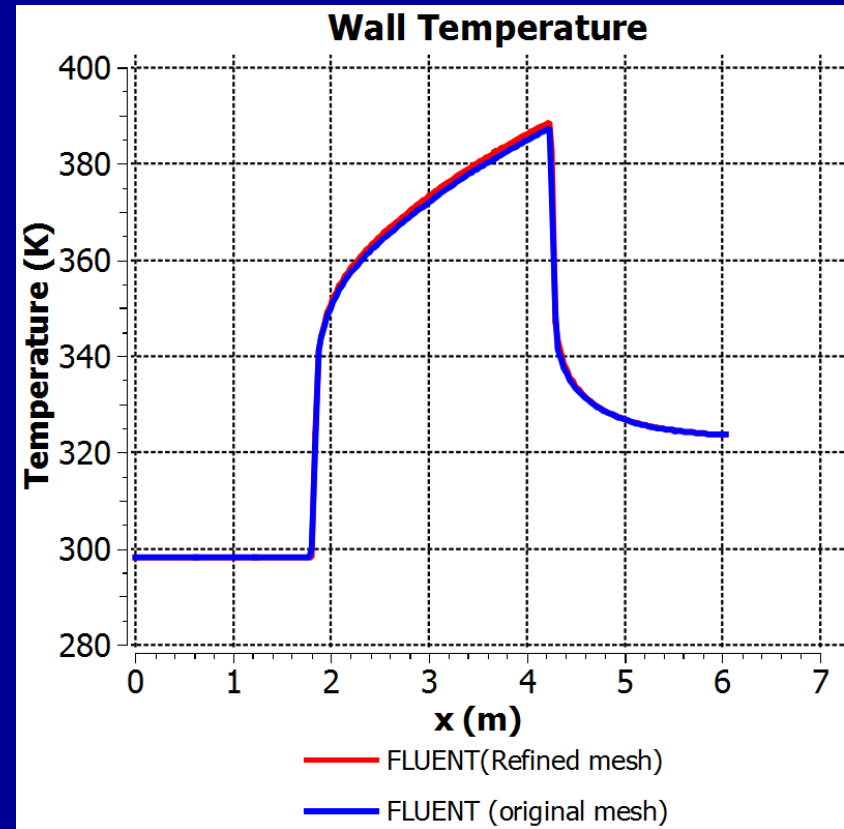
CFD Tasks for HT2

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