FLUENT - Forced Convection

Author: Rajesh Bhaskaran & Yong Sheng Khoo, Cornell University

Problem Specification
1. Pre-Analysis & Start-Up
2. Geometry
3. Mesh
4. Physics Setup
5. Numerical Solution
6. Numerical Results
7. Verification & Validation
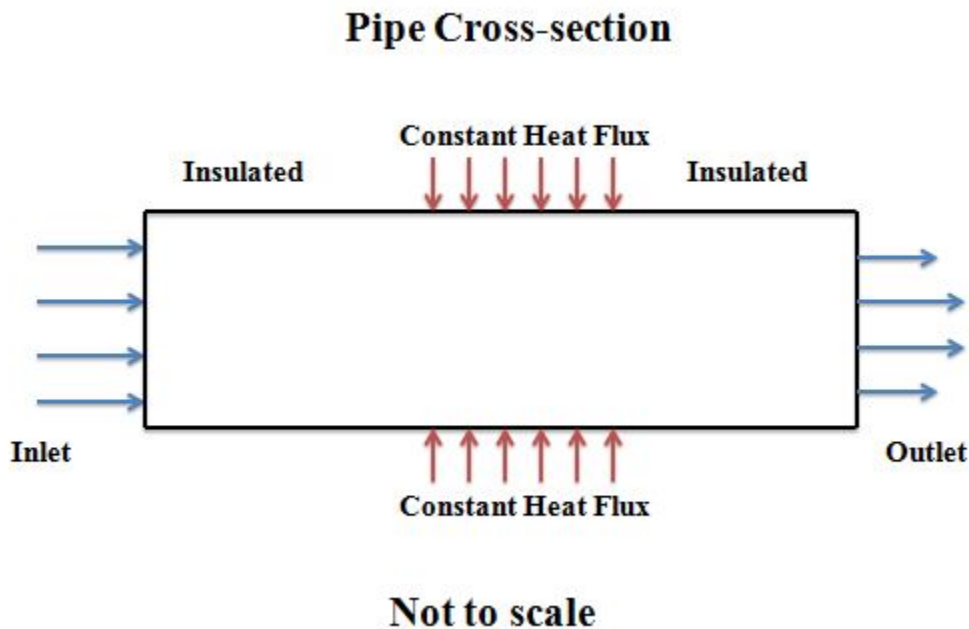Exercises
Comments

Turbulent Forced Convection

Created using ANSYS 14.0

⚠ This tutorial has videos. If you are in a computer lab, make sure to have head phones.

Problem Specification

This tutorial shows you how to simulate forced convection in a pipe using ANSYS FLUENT. The simulation corresponds to the forced convection experiment in MAE 4272 at Cornell University. 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. We'll use FLUENT to solve the relevant boundary-value problem and obtain the velocity, temperature, pressure and density distribution in the pipe. Inputs necessary for the simulation, such as the velocity at the pipe inlet and heat flux added at the wall, are obtained from one particular experimental run. Results from the simulation will be compared with corresponding experimental values.

Handout: Powerpoint slides used in the videos

Simulation Inputs Obtained from Experiment

The following inputs are necessary to specify the domain, boundary conditions and material properties for the Boundary Value Problem (BVP) that we'll solve using FLUENT.

Pipe Geometry:
Circular cross-section
Pipe radius = 2.94e-2 m
Pipe length = 6.045 m

Material Properties:
Coeff. of viscosity = 1.787e-5 kg/(m s)
Cp = 1005 J/(kg K)
Thermal conductivity = 0.0266 W/(m K)
Molecular weight = 28.97 g/mole

Inlet:
• u = 30.06 m/s
• v = 0 m/s
• T = 298.15 K

Outlet:
• Pressure = 97225.9 Pa

Wall:
• Heating between x = 1.83 m and x = 4.27 m
• Wall heat flux = 5210.85 W/m2
• Wall roughness: 0 (assume smooth)
• Wall thickness: 0 (assume negligible)

Ambient conditions:
• Ambient pressure = 98338.2 Pa

Go to Step 1: Pre-Analysis & Start-Up