NE 336: Lab 4 Group 1

COMSOL

In this lab, we will use COMSOL Multiphysics to model and analyze a concentric tube heat exchanger under different operating conditions. The primary objectives are to understand the effects of flow configurations on heat transfer performance, analyze velocity profiles, calculate local heat transfer coefficients, and compare the results for co-current and counter-current flow configurations. This document outlines the problems you need to solve. If you need to review the steps of setting up simulations in COMSOL, please refer to the supplementary materials provided.

Lab Tasks

Please create *a Word document* to submit your answers to the questions in the lab. Once you are done, save it as a PDF and submit it along with the COMSOL reports for all simulations. More details are provided below.

Exercise: Modeling a Concentric Tube Heat Exchanger

You are tasked with modeling a concentric tube heat exchanger where hot and cold water flow without mixing. The inner tube carries hot water, and the cold water flows around the outer surface of the tube within a surrounding domain. There is no gap or annulus; the cold fluid directly contacts the outer surface of the tube wall.

You will analyze the heat transfer performance under two flow configurations:

1. **Co-current Flow**: Both fluids flow in the same direction.
2. **Counter-current Flow**: Fluids flow in opposite directions.

Problem Description

A concentric tube heat exchanger consists of a hot water tube enclosed within a cold water domain. The hot water flows inside the tube, and the cold water flows around the tube, directly contacting the tube wall. The heat exchanger is used to cool the hot water by transferring heat to the cold water.

Assumptions

- Steady-state operation.
- Laminar, incompressible, Newtonian flow.
- No heat loss to the surroundings (adiabatic/ insulated outer boundary of the cold water domain).
- Constant fluid properties.

Geometry

- **Inner Tube (Hot Water Flow):**
  - Inner diameter: $D_i = 0.1$ m
  - Wall thickness: thickness = 0.001 m
- **Cold Water Domain (Surrounding the Tube):**
  - The cold water flows around the outer surface of the tube.
  - Outer diameter of the cold water domain:

$$D_{\text{coldwater}} = 2 \times D_i + \text{thickness} = 2 \times 0.1\,\text{m} + 0.001\,\text{m} = 0.201\,\text{m}$$

- **Length of Heat Exchanger:** $L = 0.5$ m

Materials

No need to use the material library.

- **Tube Material:** Copper

$$k_{\text{tube}} = 385 \quad \text{W/(m} \cdot \text{K)}$$

- **Fluid:** Water (properties assumed constant with T):
  - Density, $\rho = 998$ kg/m^3
  - Dynamic viscosity, $\mu = 0.001$ Pa·s
  - Specific heat capacity, $c_p = 4182$ J/(kg·K)
  - Thermal conductivity, $k = 0.6$ W/(m·K)

Operating Conditions

- **Hot Water (Inner Tube):**
  - Inlet temperature: $T_{\text{hot,in}} = 80°\text{C}$
  - Inlet velocity: $u_{\text{hot,in}} = 0.02$ m/s (adjusted to ensure laminar flow)
- **Cold Water (Surrounding Domain):**
  - Inlet temperature: $T_{\text{cold,in}} = 20°\text{C}$
  - Inlet velocity: $u_{\text{cold,in}} = 0.01$ m/s (adjusted to ensure laminar flow)

Boundary Conditions

You need to consider which dimensionality is most appropriate for modeling the problem: a full 3D model or a 2D axisymmetric model. Analyze the geometry and flow conditions to make your decision and justify your choice in your report.

Boundary Conditions:

- **Hot Water Inlet (Inner Tube Entrance):**
  - Inlet velocity: $u_{\text{hot,in}}$
  - Temperature: $T_{\text{hot,in}}$
- **Cold Water Inlet (Domain Entrance):**
  - Inlet velocity: $u_{\text{cold,in}}$
  - Temperature: $T_{\text{cold,in}}$
- **Outlets:**
  - Pressure outlet for both fluids (zero gauge pressure)
  - **Outflow boundary condition for the energy equation**
- **Walls:**
  - No-slip condition at all fluid-solid interfaces . (Note, this also includes the outer bound of the cold water domain).
  - Thermal continuity at the interface between fluids and tube wall. (Done by default in the Heat transfer in fluids and solids physics, just ensure to define the tube as the solid).
  - Adiabatic (insulation) condition at the outer boundary of the cold water domain.
- **Symmetry or Centerline (if applicable):**
  - If using a 2D axisymmetric model, apply symmetry conditions at $r = 0$.
  - If using a 3D model, consider appropriate symmetry or boundary conditions as needed.

When meshing, use a **Normal** level of mesh refinement, ensuring that the mesh is sufficiently fine near the walls to capture boundary layer effects.

Reynolds Number Formula:

To verify that the flow is laminar, calculate the Reynolds number for both the inner tube and the cold water domain using:

$$\text{Re} = \frac{\rho u D_{\text{hyd}}}{\mu}$$

Where:

- u is the average fluid velocity (m/s)
- D_{hyd} is the hydraulic diameter (m)

Hydraulic Diameter:

$$D_{\text{hyd}} = \begin{cases} D_i & \text{for inner tube (hot water)} \\ D_{\text{hyd,cold}} = D_{\text{coldwater}} - (D_i + 2 \times \text{thickness}) & \text{for cold water domain} \end{cases}$$

Where:

- $D_{\text{coldwater}} = 2D_i + \text{thickness}$
- $D_{\text{tube}} = D_i + 2 \times \text{thickness}$

So:

$$D_{\text{hyd,cold}} = D_{\text{coldwater}} - D_{\text{tube}}$$

Solving the Flow and Heat Transfer Problem

Part 1: Co-Current and Counter-Current Flow Configurations

You will perform simulations for both co-current and counter-current flow configurations.

Tasks:

1. **Modeling Approach:**

   Decide whether to model the problem using a 3D model or a 2D axisymmetric model. Justify your choice.

2. **Geometry Creation:**

   Create the geometry of the heat exchanger with the given dimensions, ensuring accurate representation of the inner tube (with wall thickness) and the surrounding cold water domain.

3. **Meshing:**

   Generate an appropriate mesh that captures the flow and thermal gradients.

4. **Simulation Setup:**

   - Apply the boundary conditions as specified.
   - Set up the physics for laminar flow and heat transfer in fluids and solids.
   - Use constant fluid properties.

5. **Calculations:**

   - Calculate the Reynolds number for both flow passages to confirm laminar flow.
     - Inner tube Reynolds number:

$$\text{Re}_{\text{hot}} = \frac{\rho u_{\text{hot,in}} D_i}{\mu}$$

- Cold water domain Reynolds number:

$$\text{Re}_{\text{cold}} = \frac{\rho u_{\text{cold,in}} D_{\text{hyd,cold}}}{\mu}$$

- **Note:** Adjust inlet velocities if necessary to ensure laminar flow (Re < 2000).

6. **Simulations:**

   - Run simulations for both co-current and counter-current flow configurations.
   - Obtain steady-state solutions for velocity and temperature fields.

7. **Results:**

   - **Velocity Profiles:**
     - Plot the velocity profiles in both the inner tube and cold water domain at the middle of the exchanger ($z = L/2$).
   - **Temperature Distributions:**
     - Plot the temperature fields for both configurations.
     - Calculate the average outlet temperatures $T_{\text{hot,out}}$ and $T_{\text{cold,out}}$ for both fluids.
   - **(BONUS +3%) Local Heat Transfer Coefficients :**
     - Calculate the local convective heat transfer coefficient h_{hot} between the hot water and the tube wall at $z = L/2$.
     - Calculate the local convective heat transfer coefficient h_{cold} between the cold water and the tube wall at $z = L/2$.
     - Use the definition:

$$h = \frac{q''}{\Delta T}$$

       Where:
       * q'' is the local heat flux at the fluid-solid interface.
       * ΔT is the local temperature difference between the fluid and the adjacent tube wall.

8. **Comparison:**

   - Compare the results for co-current and counter-current flow configurations.
   - Discuss how the flow configuration affects the temperature profiles.

Part 2 (BONUS +7%): Effects of Flow Velocity and Temperature-Dependent Properties

In this bonus part, you will explore the effects of varying inlet velocities and temperature-dependent fluid properties on the performance of the heat exchanger.

Additional Considerations:

1. **Effect of Flow Velocity:**

   - Perform simulations with different inlet velocities (e.g., $u_{\text{hot,in}} = 0.01$ m/s and $u_{\text{hot,in}} = 0.03$ m/s) for the hot water, adjusting $u_{\text{cold,in}}$ accordingly to maintain laminar flow.

- Analyze how varying velocities affect velocity profiles, outlet temperatures, and local heat transfer coefficients.

2. **Temperature-Dependent Properties:**

   - Incorporate temperature-dependent properties for water (e.g., viscosity $\mu(T)$ and thermal conductivity $k(T)$).
   - Use appropriate correlations or data for $\mu(T)$ and $k(T)$ over the temperature range of interest.
   - Re-run the simulation for one set of inlet velocities (e.g., $u_{\text{hot,in}} = 0.02$ m/s and $u_{\text{cold,in}} = 0.01$ m/s).

3. **Analysis:**

   - Compare the results with the constant property simulations.
   - Discuss the impact of temperature-dependent properties on velocity profiles, outlet temperatures, and heat transfer coefficients.

What to Submit

1. A Word document (saved as PDF) containing:

   - Your decision on the dimensionality of the model (3D or 2D axisymmetric) and justification.
   - Calculations of Reynolds numbers.
   - Plots of velocity profiles in both the inner tube and cold water domain at $z = L/2$ for both flow configurations.
   - Plots of temperature distributions for both co-current and counter-current flow configurations.
   - Calculated average outlet temperatures for both fluids.
   - (BONUS) Calculated local heat transfer coefficients h_{hot} and h_{cold} at $z = L/2$.
   - Comparisons of the results for co-current and counter-current configurations.
   - Discussions on how flow configuration affects temperature profiles.
   - For the bonus part, include additional results, comparisons, and discussions as specified.

2. Brief COMSOL reports (as PDFs) for all simulations performed.

Bonus

For additional credit, explore the effects of varying inlet velocities and incorporating temperature-dependent fluid properties on the heat exchanger's performance. Submit a brief report on your findings along with the corresponding COMSOL reports.