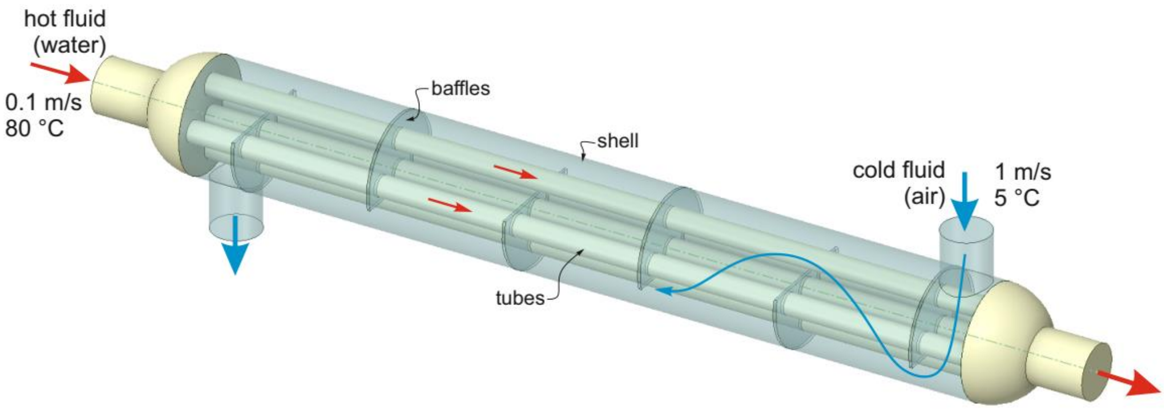
Introduction to Computational Fluid Dynamics
Coursework Instructions

Simulation of a Counter Flow Heat Exchanger

You will perform a steady-state simulation of the counter flow heat exchanger illustrated in the figure below. Details of the geometry and the boundary conditions are provided in the appendix (on the next page).

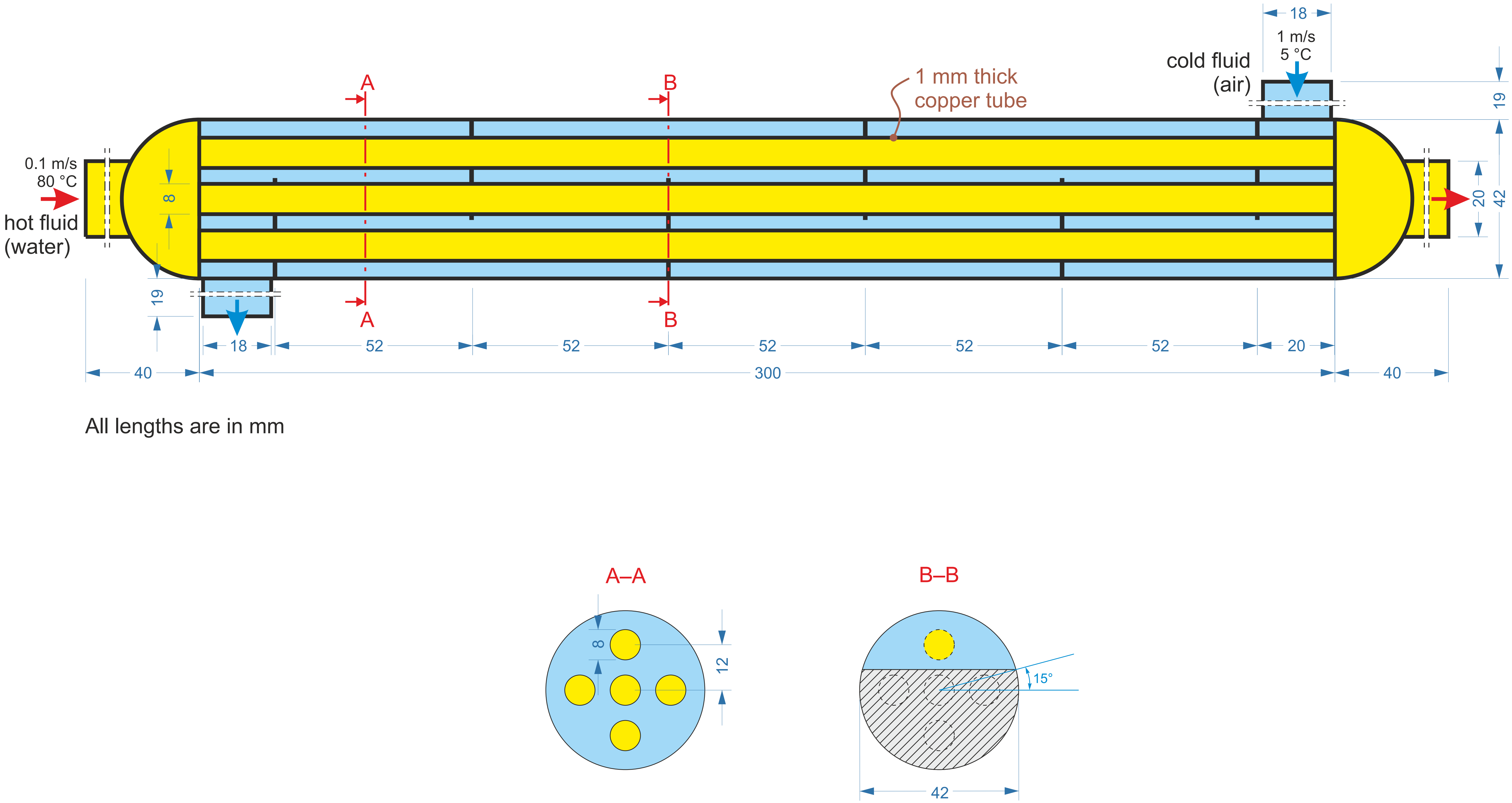


A step-by-step tutorial video on how to create the model geometry, generate the computational mesh and perform a simulation using ANSYS Fluent is available from the following link:

[Link to tutorial video]

After creating the model geometry, generate a computational mesh using ANSYS Fluent with the default mesh settings. Perform a steady-state flow and heat transfer simulation using the default *k*-ω SST turbulence model and determine: (1) the pressure drop between the inlet and the outlet for the cold fluid (air); (2) the temperature of the cold fluid at the outlet. Repeat the simulation using a laminar flow model. Check how much the values of the pressure drop and the change in temperature of the air vary for the two simulations.

Summarise your findings in a report. Start your report by briefly introducing the problem, explaining what equations were solved and what boundary conditions were applied. Use appropriate figures to present the computational mesh and the resulting fluid flow and temperature distributions. Discuss your results.



All lengths are in mm