Final Project

Final Project

Option 1: Use one of the proposed project

Option 2: Propose your own project

- Required components
  - verification - use an exact or manufactured solutions to determine the accuracy
  - modeling (one of the two)
    - use of a physical model - turbulence, heat transfer, rotation, etc.
    - use of a user-defined model - boundary conditions, material property, etc.
  - validation - compare the predictions to available experimental data

Examples - UDF

1) Vince Terrapon: wrote UDFs to solve a non-Newtonian fluid problem. The fluid is assumed to be a solution of water and long molecules (polymers). The constitutive model for the fluid is based on a conformation tensor (6 scalar PDEs) that describes at a macroscopic level the orientation of the molecules. The method requires the solution of 6 PDEs to obtain the effective viscosity. The application was the laminar flow in a channel.

2) David Cook: considered a chemical mechanism for methane-air flames. The problem requires the solution of several PDEs corresponding to the various species involved in the reactions. The application was a simple 1D flame

Examples - Physical Models

1) Andy Chan: Studied the unsteady flow in a Hard-Drive compartment. He first looked at a rotating disk problem (classical solution of the NS equations - verification) and then considered a realistic geometry

2) Chance Moreland: Studied the flow around an airfoil with a discrete surface roughness. He compared the predictions to experiments and to the analysis performed with a simplified approach (xfoil)

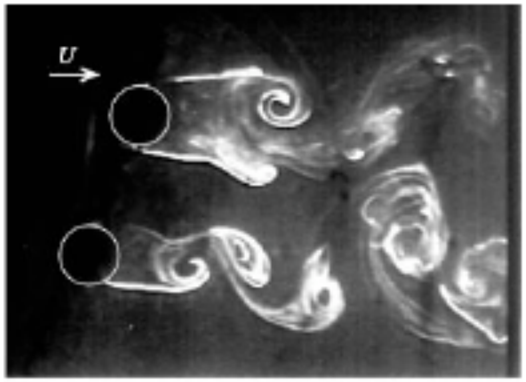
Final #1

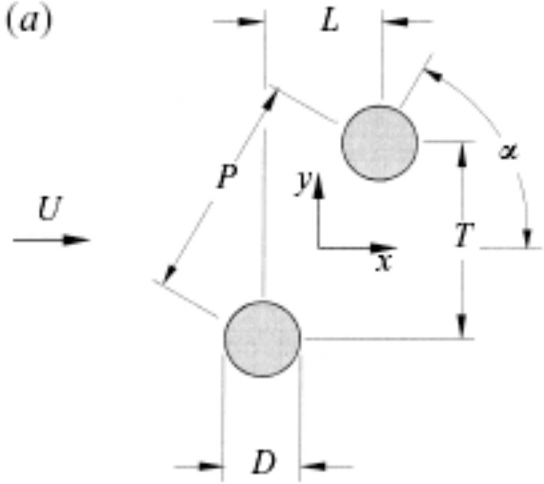
Objective: simulation of the unsteady flow around two-cylinders configuration

The flow around several two-cylinders configurations is investigated experimentally in the paper:

Sumner, Price and Paidoussis"Flow-pattern identification of two staggered circular cylinders in cross-flow"Journal Fluid Mechanics (2000) vol 411 pp 263-303

- *Extract the test conditions from the paper (select two configurations of your choice - not too similar)*
- *For the first case, verify the grid sensitivity and the turbulence modeling sensitivity (at least two grids and two turbulence models) by comparing qualitatively the flow structures to the experimental pictures and quantitatively the vortex shedding frequency computed to the experimental data reported*
- *For the second case, use the best combination of grid and turbulence model and compare with the experiments*

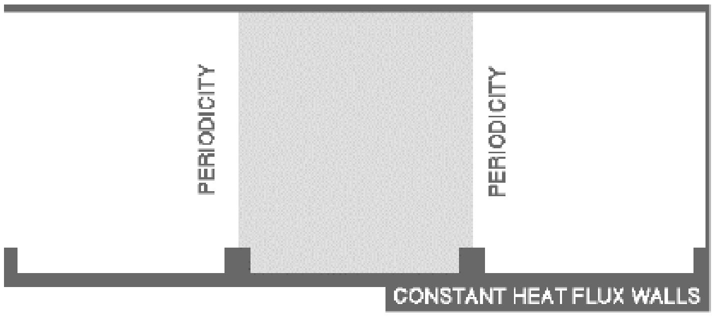




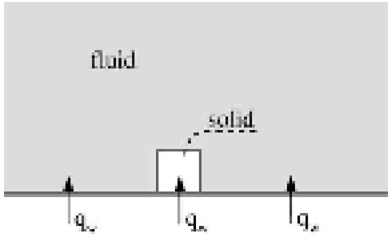
Final #2

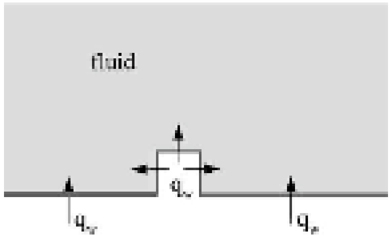
Objective: Heat Transfer in Ribbed Channels

Ribbed Channels are very common test cases for CFD.



- *Extract the test conditions from the ERCOFTAC case 7.2 description (see class Web Site)*
- *Compute the flow and the heat transfer and compare to the experiments available from the class Web Site*
- *Verify the grid sensitivity and the turbulence modeling sensitivity (two grids and two turbulence models)*
- *Investigate the effect of the shape of the rib (triangular, rectangular, semicircular, etc) (two shapes + square rib)*
- *Model the heat transfer within the rib (assume material properties of aluminum) and apply the constant heat flux condition at the base of the rib. Compare the two type of boundary conditions reported on the side.*





Final #3

Objective: Development of a Vorticity-Streamfunction formulation for solving the Navier-Stokes Equations

- *Derive the equations for the vorticity-streamfunction formulation*
- *Implement the model using UDF in Fluent for 2D problems*
    - *One equation for the vorticity (scalar advection/diffusion equation)*
    - *One equation for the streamfunction (Poisson equation)*
    - *Additional scalars to store the velocity*
    - *How to compute the pressure?*
    - *What are the boundary condition for the vorticity?*
- *Apply to a channel flow and a backstep in laminar regime*
    - *Channel - Re = 100 (based on H and centerline velocity)*
    - *Backstep - Re = 1000 (based on step heigh and centerline velocity)*
- *Compare the results to the standard solution computed using Fluent*
    - *Compare the vorticity field, velocity field and the pressure*