ME-474 Numerical Flow Simulation

Exercise: boundary conditions (BCs)

Fall 2021

Use the 2D meshes provided on Moodle to simulate the steady incompressible flows described below. The Reynolds number is $Re = 40$ in all cases. The diameter of the object is $D = 1$ cm. Use a fluid of density $\rho = 1$ kg/m^3 and dynamic viscosity $\mu = 10^{-3}$ kg/(m.s). Note that the mesh is rather coarse (about 4000 elements), and that for external flows the size of the domain is rather small (13D x 3D).

1. Flow past an infinite solid cylinder (no-slip wall), with a uniform incoming flow (streamwise velocity U_{in}), and symmetric (or zero-shear slip wall) BC on the upper $y = cst$ boundary.

2. Same as (1) but with a no-slip wall moving at $U = U_{in}$ on the upper boundary.

3. Same as (1) but in a channel with no-slip walls, with a fully developed parabolic velocity profile at the inlet (define Re with the maximum velocity). You can use either an "expression" (defined directly in Fluent) or a "profile" (text file generated with the Matlab code provided on Moodle).

4. Flow past a solid sphere (no-slip wall), with a uniform incoming flow (streamwise velocity U_{in}), and symmetric BC on the outer $r = cst$ boundary. Consider 2 cases: (i) fixed sphere, (ii) sphere rotating around $\mathbf{e}_z$ with an angular velocity such that the maximum azimuthal velocity on the sphere wall is the same as the inlet velocity: $\max(U_\theta) = U_{in}$.

5. Same flow as (1) but past a homogeneous porous cylinder of viscous resistance 2×10^5 m^{-2} and zero inertial resistance.

6. Same as (1) but with periodic BCs at the inlet/outlet (i.e. fully developed flow with streamwise periodicity $L_x = 13$ cm).

7. Flow past an infinite solid cylinder (no-slip wall) driven by a constant tangential wall shear of magnitude $\mu|\partial u/\partial y| = 0.5$ Pa on the upper boundary and oriented such that the wall velocity is in the positive x direction.

8. Same as (1) with a 1st-order spatial discretization for the convective term of the momentum equation. Compare the drag coefficient of the solutions obtained with 1st and 2nd-order discretizations. (You can compute aerodynamic force in Results / Reports / Forces. To compute coefficients, you must define suitable reference values used to normalize forces: density, velocity, and area in 3D or length in 2D. This can be accessed via Physics / Solver / Reference values in the ribbon, or Setup / Reference values in the tree.)