Simulation of Vortices in Two-Dimensional Lid-Driven Cavity Flows

An interesting physical situation arises when one considers a sealed cavity with shear imposed at one or more boundaries. One or more large eddies occupies the major volume of the cavity, while small (Moffat) vortices form in sharp corners. Visualize a boundary moving like a conveyor belt, without changing the volume or shape of the cavity. In this exercise, you will consider the effects of varying two parameters: **Reynolds number** (of the characteristic shape[1]) and **grid resolution** (number of control volumes).

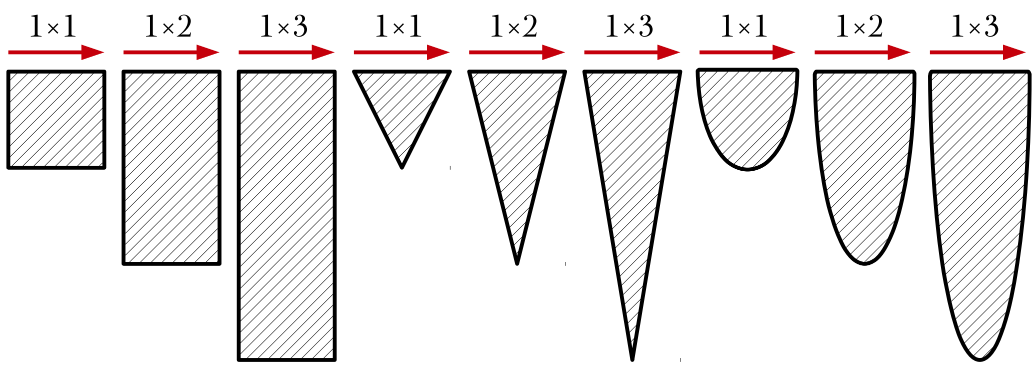
You will be assigned a base shape and a parameter to vary over that shape. Select your shape from rectangle, triangle, or half-oval. The parameters to be varied will be *either*

> **Reynolds number (100, 400, 1000) on fixed grid of ~6400 control volumes per unit area**

or

> **the number of control volumes per unit area (~400, ~1600, ~6400 per unit area) at fixed Re=1000**.

For each shape, you will consider aspect ratios varying from 1:1 to 1:3, as depicted below. You should report on nine simulations in total:three aspect ratios of one shape times three parameter settings in either Re or mesh resolution.



You will document the simulations in a 10–12 page report (with figures) containing sections:

a) Problem description (shape, grid, etc.)
b) Details of the simulation (settings and capabilities of Fluent)
c) Numerical parameters (laminar flow, grid size, boundary velocity, initialization, etc.)
d) Computational times (iterations and CPU time to complete)
e) Observations of numerical behavior (residual convergence rate, mesh behavior, etc.)
f) Discussion of the flow physics (vortex number, size, and position)—include thresholds of qualitative changes in behavior
g) Verification and validation (to extent possible; compare to experimental results)

1.Take the characteristic length and velocity to be the *depth of the cavity* and the *velocity of the shearing boundary*, respectively, in defining the Reynolds number Re. As the problem is determined by Re, you have full discretion in selecting L, u, fluid composition, etc. Document and justify these choices in your report.

Include the following plots in your report, with data from each case you study:

a) Convergence of mass, momentum residuals
b) Streamlines
c) Line plots of velocities along principal vertical axis
d) Contours of pressure and vorticity

The report should be formatted with 1.5-line spacing, 1-inch margins on all sides, and set in 10.5–12-point serif font. All figures and tables (if any) should be numbered and have labels and captions. Cheng (2006) exemplified how to document numerical simulations for this type of problem. Submit an electronic copy to the TA by 5:00 p.m. on the due date.

References

Biswas, S, Kalita J C (2016) Moffatt vortices in the lid-driven cavity flow. arXiv:1601.05186 [physics.comp-ph].

Cheng, M, Hung, K C (2006) Vortex structure of steady flow in a rectangular cavity. *Computers & Fluids 35*:10, 1046–62. doi:10.1016/j.compfluid.2005.08.006.

Moffat, H K (1964) Viscous and resistive eddies near a sharp corner. *J Fluid Mech 18*:1, 1–18.doi:10.1017/S0022112064000015.

Taneda S (1979) Visualization of separating Stokes flows. *J Phys Soc Jpn 46*, 1935–1942. doi:10.1143/JPSJ.46.1935.