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

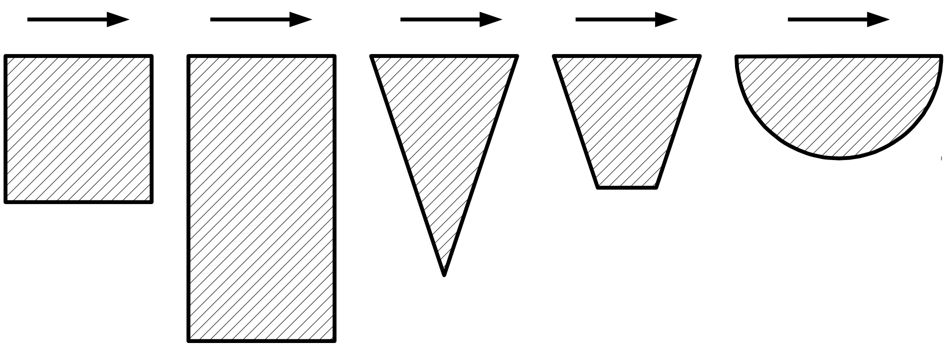
Moffat vortices form in corners due to the shear imposed on one of the boundaries (*i.e.*, one boundary moves like a conveyor belt, without changing the volume or shape of the cavity). These vortices form in driven cavity flows of square, rectangular, triangular, trapezoidal, and semi-circular shape. The effects of varying two parameters will be studied in this exercise: **Reynolds number** (of the characteristic shape[1]) and **grid resolution** (number of control volumes).

You will select a shape and a parameter to vary. The possible shapes are square, rectangle, triangle, trapezoid, or semicircle. The parameters to be varied will be *either*

> **Reynolds number (100, 400, 1000) at a fixed grid of 80×80 for a quadrilateral (square, rectangle, trapezoid) or around 6400 control volumes for an unstructured grid (triangle, semicircle)**

or

> **the number of control volumes (~2000, ~5000, ~8000 for an unstructured grid or 20×20, 40×40, 80×80 for a quadrilateral) at fixed Re=1000**.



You will document your simulations in a 10 page report (with figures) containing the 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 the following plots in your report, with data from each case your team will study:

a) Convergence of mass, momentum residuals
b) Streamlines
c) Line plots of velocities along principal directions (based on the geometry)
d) Contours of pressure

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, the fluid composition, etc., as long as you document and justify these choices.

The report should be formatted with 1.5 line spacing, 1 inch margins on all sides, and set in 11 point serif font. All figures and tables (if any) should be numbered and have labels and captions. Saha (2008) exemplified how to document numerical simulations for this type of problem (although your report will not be as technical). Submit a hard copy on the due date by 5:00 p.m. to the drop box located in 2270 DCL.

In addition, you should submit a streamline plot of your maximum resolution, maximum Reynold number simulation to the teaching assistant so that we can jointly review the results of this project in class.