# Flow within Two Concentric Cylinders

Contributed by Prof. John Cimbala and Matthew Erdman, The Pennsylvania State University

Part 1: Analytical Solution

An incompressible Newtonian liquid is confined between two concentric circular cylinders of infinite length – a solid inner cylinder of radius R_i and a hollow, stationary outer cylinder of radius R_o. (See sketch; the z axis is out of the page.) The inner cylinder rotates at angular velocity Ω_i. The outer cylinder rotates at angular velocity Ω_o. The flow is steady, laminar, and two-dimensional in the r-θ plane. The flow is also *rotationally symmetric*, meaning that nothing is a function of coordinate θ (u_θ and p are functions of radius r only). The flow is also circular, meaning that velocity component $u_r = 0$ everywhere.

(**a**) Generate an exact expression for velocity component u_θ as a function of radius r and the other parameters in the problem. You may ignore gravity. Show all your work.

(**b**) Plot u_θ (m/s) as a function of radius r (m) for the particular case in which $R_i = 0.07$ m, $R_o = 0.12$ m, $\Omega_i = 100$ rpm, and $\Omega_o = -200$ rpm. For consistency, use a curve only (no symbols) on your plot.

Liquid: ρ, μ

Ω_o

R_o Ω_i

R_i

Rotating inner cylinder
Rotating outer cylinder
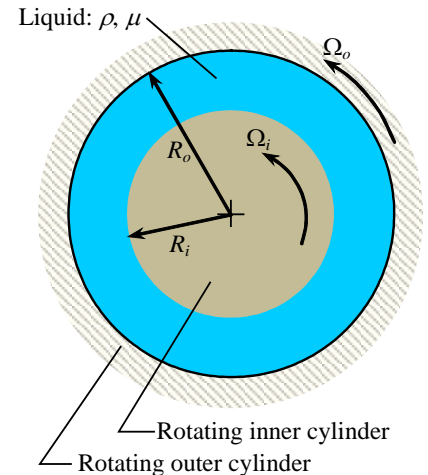
Part 2: Numerical Solution using ANSYS FLUENT

Repeat the above problem using ANSYS FLUENT. A *free* student version of ANSYS Workbench (including FLUENT) is available for students to download onto their own computers! You can get the free download at http://www.ansys.com/student. Make sure you follow the installation directions carefully. [Warning: Installation is more cumbersome than most software that you download because of the licensing, but it works.]

Watch the YouTube tutorial (https://www.youtube.com/watch?v=3DnLP9-UruA&feature=youtu.be) on how to use ANSYS Fluent to set up and solve a problem like this. Run the CFD code to convergence for this problem. Plot u_θ as a function of radius r on the *same plot* as the previous problem and compare. For consistency, use symbols only (no curve) for these data so that the CFD results are symbols and the analytical (exact) solution is a curve. If all goes well with both your analytical solution (previous problem) and your CFD solution, the two profiles should agree well. If not, you have probably made an error somewhere.