§19. Numerical Simulation on Laboratory Experiment of Magnetorotational Instability: Circulation in a Taylor-Couette Flow

Kageyama, A.,
Ji, H. (Princeton Plasma Physics Laboratory),
Goodman, J. (Princeton Univ.)

Magnetorotational instability (MRI) is now believed to be the most plausible instability that drives the anomalous angular momentum transport in magnetized accretion disks. Aiming the first laboratory demonstration of MRI, a liquid metal MHD experiment is undertaken at Princeton Plasma Physics Laboratory (PPPL). In this experiment, a liquid metal (gallium) will be confined in a cylindrical annulus region between two concentric cylinders. The two cylinders are rotated with different angular velocities, and the liquid metal in the vessel will rotate with some shear. When a sufficiently strong external magnetic field is applied to the liquid metal, MRI will take place, for the first time, in the laboratory[1)]. In order to produce a flow that is stable to the hydrodynamic (Taylor–Couette) mode but unstable to MRI, a prototype water experiment in the same geometry has been performed at PPPL. The initially proposed experimental setup consists of two concentric short cylinders. However, preliminary data indicate that the toroidal flow profile in this cylindrical, or rather a disk-like, vessel is strongly affected by the vessel's horizontal boundaries (the lid and the bottom). Since the profile control of the toroidal flow is essential for the MRI experiment, we performed 2-D numerical simulations of this Navier-Stokes system to understand the flow structure and the boundary effect in detail.

In this simulation, we suppose axisymmetry of the flow $\boldsymbol{v}$ ($\partial \boldsymbol{v}/\partial\varphi = 0$) in the cylindrical coordinate system (r,φ,z), and the incompressibility ($\nabla\cdot\boldsymbol{v} = 0$). We use the stream function–vorticity method, in which time developments of v_φ and ω_φ are followed, where $\boldsymbol{\omega}$ is the vorticity. The poloidal (r and z) components of the velocity are obtained by a stream function ψ as

$$v_r = \frac{1}{r}\frac{\partial\psi}{\partial z}, \qquad v_z = -\frac{1}{r}\frac{\partial\psi}{\partial r}.$$

The stream function ψ is calculated from ω_φ. The inner cylinder (radius R_1) is rotating at an angular velocity Ω_1, the boundary condition for v_φ at the inner cylindrical wall is given by

$$v_\varphi(R_1, z) = R_1\Omega_1.$$

The boundary condition at the outer cylinder ($r = R_2$) is given by

$$v_\varphi(R_2, z) = R_2\Omega_2.$$

Since, in the experiment, the cylinder's end walls (the lid and the bottom) are rotating with the same angular velocity as that of the outer cylinder, the boundary condition at the top ($z = +H/2$) and the bottom ($z = -H/2$) caps are given by

$$v_\varphi(r, \pm H/2)) = \frac{r}{R_2}\Omega_2.$$

We use the second order finite difference method in spatial derivatives and the fourth-order Runge-Kutta method for time advance. Typical mesh size is 100×100.

The geometry of this prototype experiment is as follows. The inner and the outer cylinder radii are $R_1 = 3.8$cm and $R_2 = 14.9$cm; The cylinders' height $H = 10$cm; The rotation rates of the cylinders are $\Omega_1 = 2000$rpm and $\Omega_2 = 150$rpm. The numerical simulation is performed with a wide range of Reynolds number Re ($= R_1^2\Omega_1/\nu$), from 10 to 3200.

It has been shown by the prototype water experiment that the fluid's rotation profile $v_\varphi(r)$ is very different from that of the so called circular Couette flow.

The numerical simulation has confirmed this result and the the simulated $v_\varphi(r)$ profile coincides well with the experimental data. Analyzing the simulation data in detail, it is shown that this deviation from the circular Couette flow is caused by the circulation flow in the poloidal (r,z) plane.

This circulation flow is induced by the force imbalance between the centrifugal force and the inward pressure gradient at the lid and the bottom boundaries. This is similar to the generation mechanism of the usual Ekman layer, but, in this case, the water in the vessel is rotating with nearly uniform angular momentum, rather than the uniform angular velocity as in the usual Ekman case.

One of the most remarkable feature of this circulation flow is the formation of a jet flow at the middle height ($z = 0$). The inward ($v_r < 0$) flow at the bottom (or the lid) boundaries clime up (or down) at the inner wall $r = R_1$ and they "collide" each other just at the middle, then the fluid go outward to form the jet. The structure of this circulation flow (with jet) is described in detail in our paper that will be submitted in near future.

References

1) Hantao Ji, Jeremy Goodman, and Akria Kageyama, Mon. Not. R. Astron. Soc. **325**, L1-L5 (2000)