Rodolfo Ostilla Monico
Physics of Fluids Group, U. Twente, Netherlands

Highly turbulent Taylor-Couette flow: Direct Numerical Simulations

Summary:
Taylor-Couette (TC) flow is the flow in a fluid layer between two coaxial and independently rotating cylinders. TC has long been used as paradigmatic system in fluid mechanics, as it is a closed system with a high amount of symmetries but yet shows very rich phenomena. High Reynolds number TC flow remained relatively unexplored until the last five years, when renewed interest in the system caused a spike in experimental and numerical activity.
In this work, we have numerically simulated TC flow from the laminar, purely azimuthal flow up to Reynolds numbers of the order 10^5. To do this, a highly parallel computing code was developed, with a multiple-resolutions strategy to efficiently deal with the effect of scalars in the flow.
The behaviour of highly turbulent TC flow, and how it transitions to an “ultimate” regime, which is expected to be similar to that found in astro- and geo-physical flows is studied. The nature of turbulence in astrophysical accretion disks, which can be modeled using TC flow is also explored in this context. Second, the existence of an optimal transport, i.e. a maxima in the torque required to rotate the cylinders for a given shear rate as a function of the ratio between the rotation rates of both cylinders was also studied. This phenomena was earlier observed in experiments, but remained unexplained. Simulations have allowed us to unravel the physics behind this process in more detail.

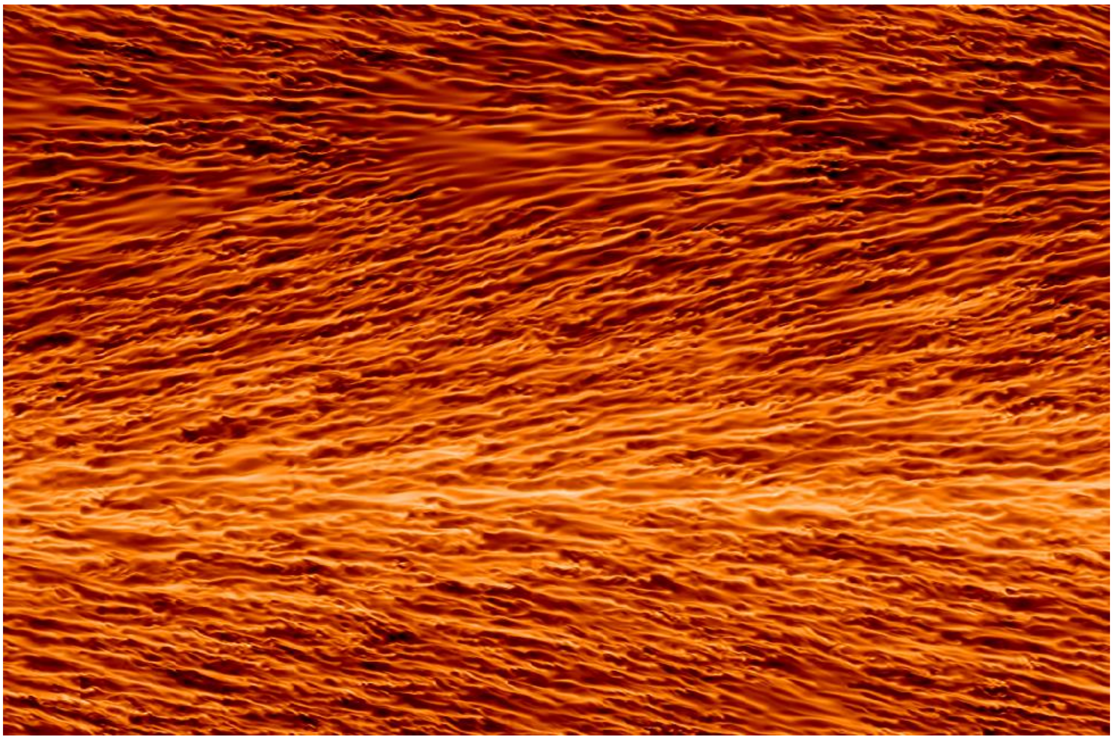

Image Caption:
Visualization of the turbulent structures in a two-dimensional cut through a simulation of Taylor-Couette flow, with a Reynolds number of 10^5 . Over a billion degrees of freedom were used in this simulation, adding up to a total of consumption of 1.2 million CPU hours for the whole simulation.