

ABSTRACT

Background: The Navier–Stokes equations describe the motion of fluids such as water, air, and blood. They are central in both mathematics and physics, and in three dimensions they are tied to one of the Clay Millennium Problems.

This project: This poster studies the two-dimensional incompressible case, where the theory is better understood. It combines analysis of weak solutions with finite volume simulations of lid-driven cavity flow.

GOVERNING EQUATIONS

For an incompressible fluid, a simplified form of the governing equations is

$$\partial_t \mathbf{u} + (\mathbf{u} \cdot \nabla) \mathbf{u} - \nu \Delta \mathbf{u} + \nabla p = 0, \quad \nabla \cdot \mathbf{u} = 0.$$

- ▶ \mathbf{u} is the velocity.
- ▶ p is the pressure.
- ▶ ν is the viscosity, which describes how strongly the fluid resists deformation.
- ▶ The Reynolds number is

$$Re = \frac{UL}{\nu},$$

where U is a typical speed and L is a typical length.

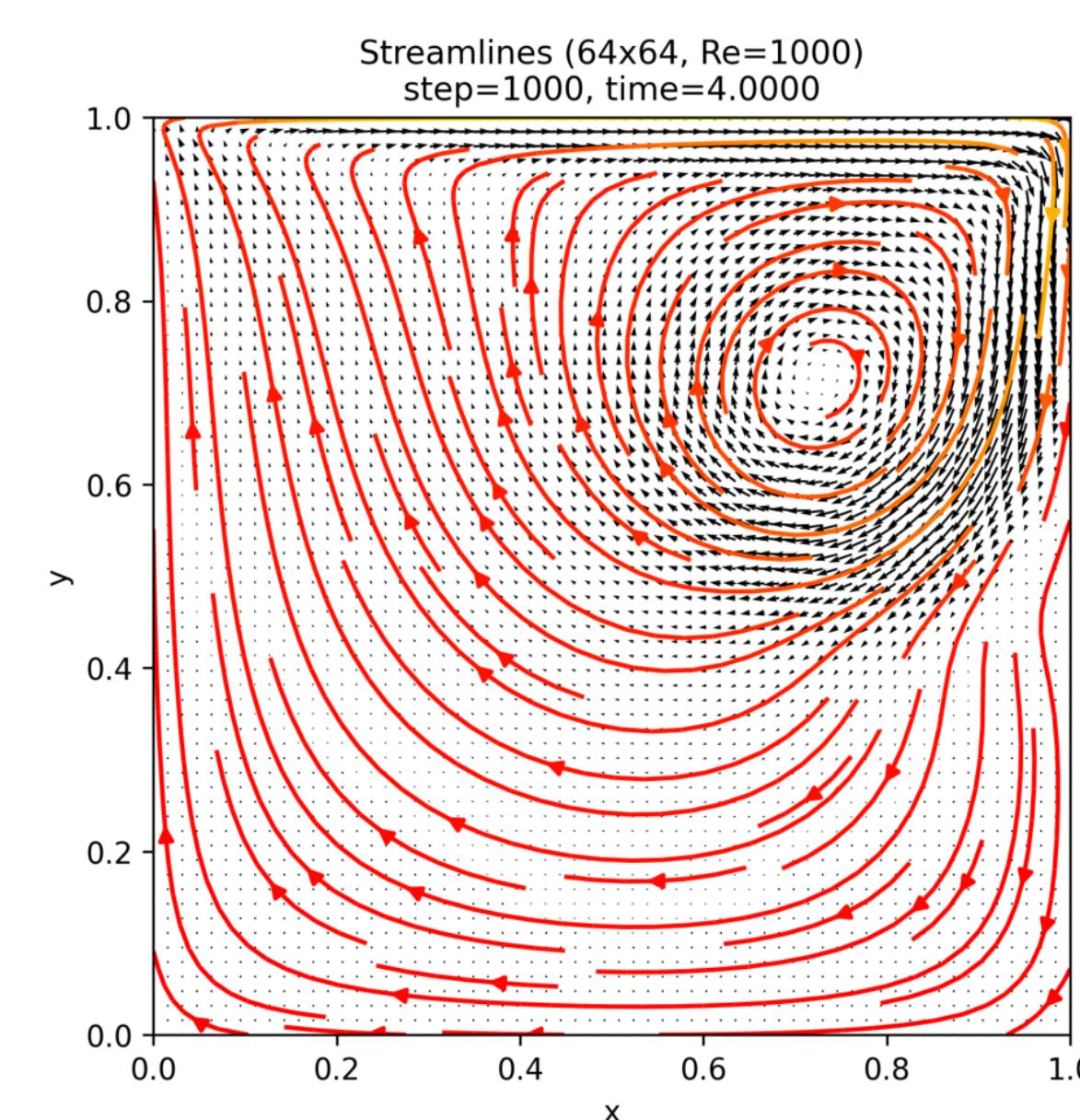
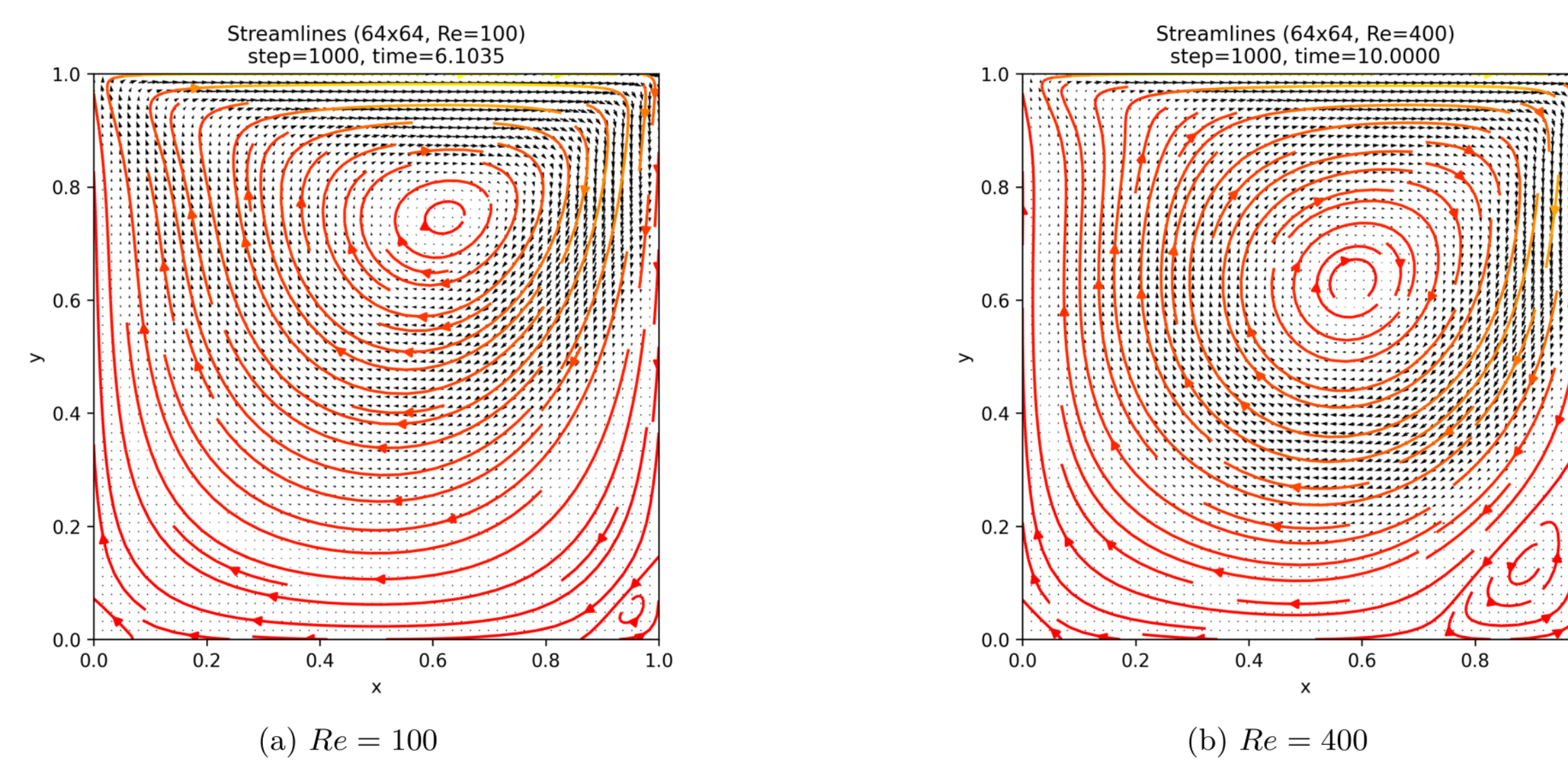
- ▶ A smaller Re means viscous effects are stronger relative to inertial effects, so the flow tends to be smoother.
- ▶ A larger Re means inertial effects are stronger relative to viscous effects, so the flow can develop sharper layers and more complex circulation.

MAIN ANALYSIS RESULT

The analysis develops the weak-solution framework for the steady two-dimensional incompressible Navier–Stokes equations.

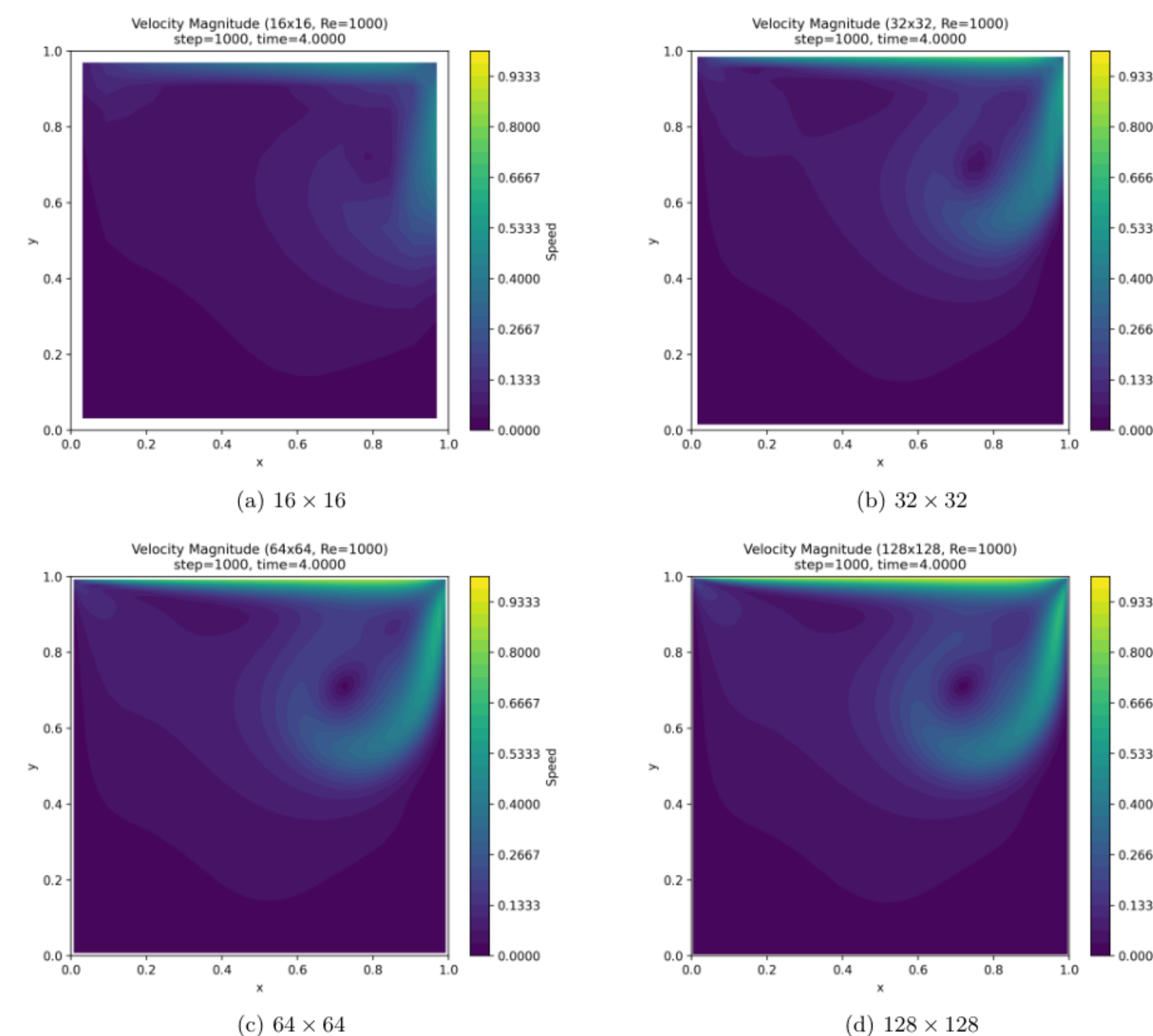
- ▶ **Existence:** a weak solution exists in two dimensions.
- ▶ **Uniqueness:** the solution is unique when the forcing is sufficiently small.

RESULT 1: REYNOLDS NUMBER



These streamline plots use the same 64×64 grid and compare $Re = 100, 400,$ and 1000 . This shows that larger Re leads to sharper and more structured flow.

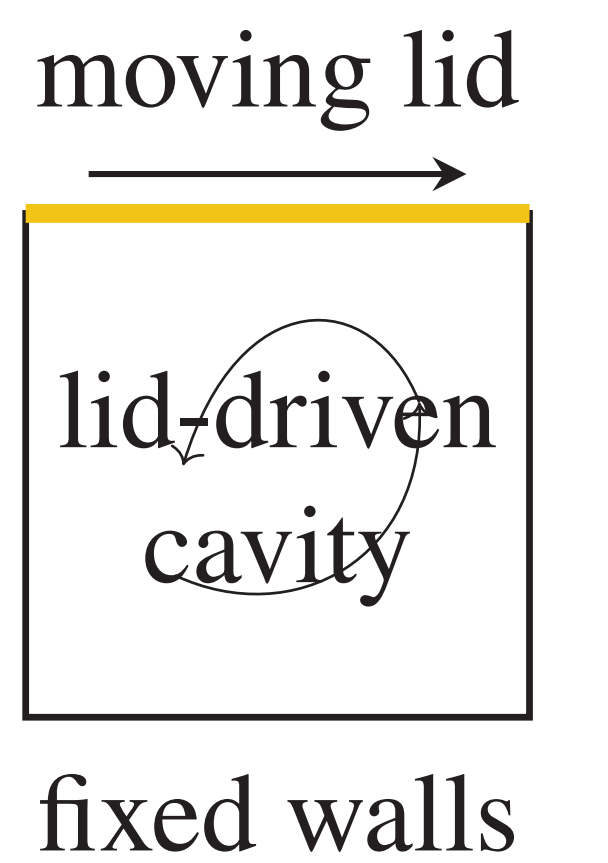
RESULT 2: MESH REFINEMENT



These velocity-magnitude plots fix $Re = 1000$ and compare meshes from 16×16 to 128×128 . As the mesh is refined, the structure becomes smoother and more clearly resolved.

NUMERICAL SETUP

To visualize the equations, I studied the *lid-driven cavity* problem. The fluid is contained in a square box. The top wall moves to the right, while the other three walls remain fixed. This motion drives a recirculating flow inside the cavity.



NUMERICAL METHOD

The computation uses a finite volume method on a staggered grid:

- ▶ pressure is stored at cell centers,
- ▶ horizontal velocity is stored on vertical faces,
- ▶ vertical velocity is stored on horizontal faces.

This arrangement improves pressure–velocity coupling and helps the corrected velocity remain nearly divergence free. It is also natural for incompressible flow because it respects conservation and the divergence constraint.

MAIN COMPUTATIONAL RESULT

The numerical study shows two main patterns:

- ▶ increasing the Reynolds number changes the flow structure;
- ▶ refining the mesh makes the computed flow clearer and better resolved.

Together, these results show how the numerical method captures both the physics of the model and the effect of computational resolution.

REFERENCES

- ▶ Galdi, Giovanni P. *An Introduction to the Mathematical Theory of the Navier–Stokes Equations: Steady-State Problems*. 2nd ed. Springer, 2011.
- ▶ Saad, Tony. *Professor Saad Explains*. YouTube channel, Episodes 11, 12, and 13, 2021.