# Lid-driven cavity problem using projection method

Jin-Hyung Bae | Dept. of Mechanical Engineering , Turbulence Lab, Yonsei University

Undergraduate Research Programme, School of Mathematics and Computing (Computational Science and Engineering), Yonsei University

Abstract

- The lid-driven cavity problem has been used a test or validation case for incompressible Navier-Stokes equation solvers.

- Consider two-dimensional unsteady flow in a square domain with Dirichlet boundary conditions on all sides, with three stationary sides and one moving side.

- Based on the lid velocity U and cavity height (or width) H, Reynolds number can be defined as Re = UH/ν, where ν is kinematic viscosity.

- The velocity of the impulsively started lid is given by a step function $u_{lid} = U$ for t ≥ 0 and $u_{lid} = 0$ for t < 0.
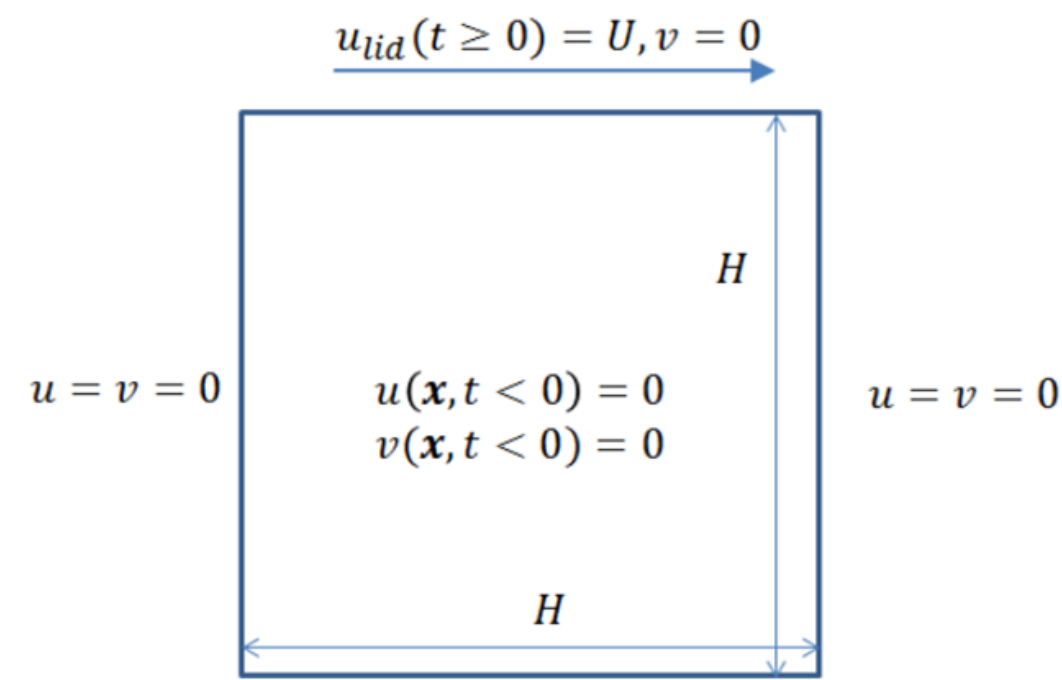
Key Concepts

• Staggered grid

- Numerical technique commonly used in fluid dynamics simulations
- Stagger the placement of velocity and pressure components within the grid, i.e., store velocities at the faces or edges of the grid cells, and pressure at the center of the cells.
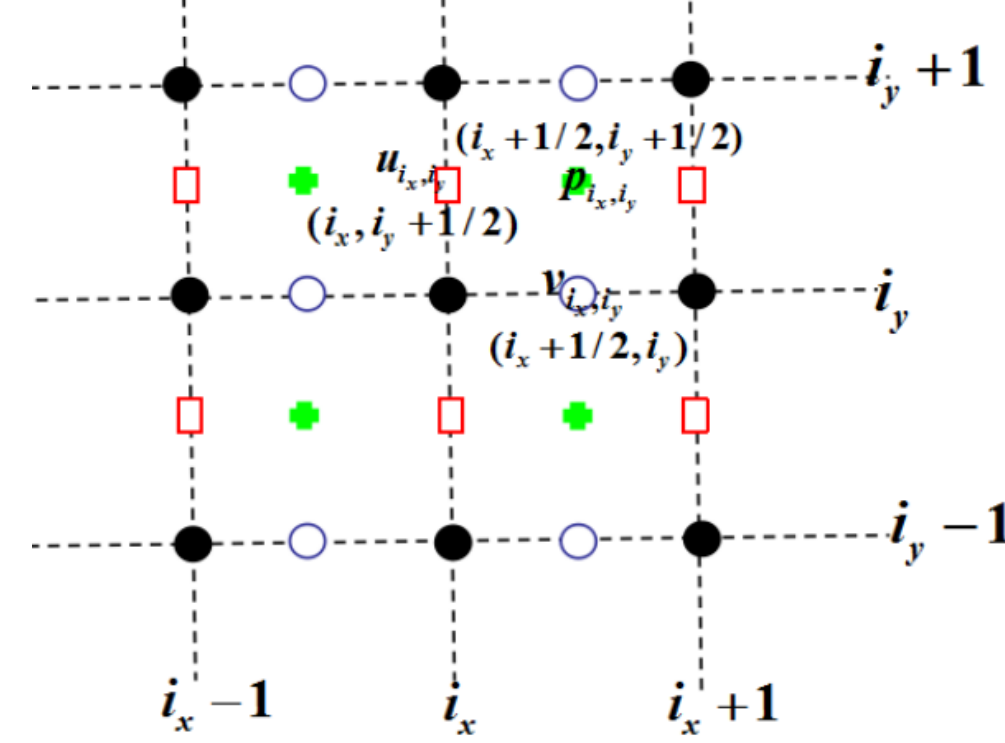
• Non-dimensionalized NSE & CE(Continuity Equation)

- Non-dimensionalized NSE(Navier-Stokes Equation) simplifies the analysis of different flow regimes and reduces computational complexity.

- CE(Continuity Equation) ensures mass is conserved, often simplified to the divergence-free condition for incompressible flows.

$$\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} = -\nabla p + \frac{1}{Re} \nabla^2 \mathbf{u},$$

$$\nabla \cdot \mathbf{u} = 0.$$

• Discretization

- The Navier-Stokes Equation and the Continuity Equation can be expressed using the discrete form based on a staggered grid, employing the second-order explicit Adams-Bashforth method for the convection term and the implicit Crank-Nicolson method for the viscous term.

$$\frac{u_i^{n+1} - u_i^n}{\triangle t} + \frac{1}{2}\left(3H_i^n - H_i^{n-1}\right) = \frac{1}{2Re}L\left(u_i^{n+1} + u_i^n\right) - Gp^{n+1},$$

$$Du_i^{n+1} = 0,$$

Numerical Procedures

1) Two-step Time-advancement Scheme
Intermediate velocity u^* and pseudo-pressure ϕ are introduced to decouple the velocity and pressure fields, simplifying the computation.

2) Intermediate Velocity Calculation
The intermediate velocity is a provisional velocity used as an intermediate step for calculating the actual velocity and pressure fields. The intermediate velocity is computed using discrete form of Navier-Stokes equation, and the velocity difference δu is defined to update the velocity, expressed as $u^* = u^n - \delta u$. The TDMA (Tridiagonal Matrix Algorithm) is applied twice in the 2D case, once in the x-direction and once in the y-direction, to solve for the intermediate velocity.

3) Pseudo-pressure Calculation
The pseudo-pressure ϕ^{n+1} is computed by solving a Poisson equation. This step is essential for enforcing the incompressibility constraint. Once ϕ^{n+1} is obtained, it is used to correct the intermediate velocity.

4) Velocity Correction
This step uses the pseudo-pressure ϕ to adjust the intermediate velocity u^*, ensuring that final velocity is divergence-free (i.e. incompressible). The corrected velocity is given by: $u^{n+1} = u^* - \Delta t \triangledown \phi^{n+1}$. This process ensures that the final velocity field satisfies the incompressibility condition by using the pseudo-pressure to adjust the intermediate velocity.

5) Pressure Correction
Pressure correction involves updating the actual pressure by establishing a relationship between the pseudo-pressure ϕ^{n+1} and the actual pressure p^{n+1}. The correction is made by modifying the discrete equations, and the pressure is updated accordingly.

6) Final Updated Velocity and Pressure
The final updated velocity u^{n+1} and pressure p^{n+1} are obtained after the correction steps. This process is repeated at each time step to predict the flow over time.
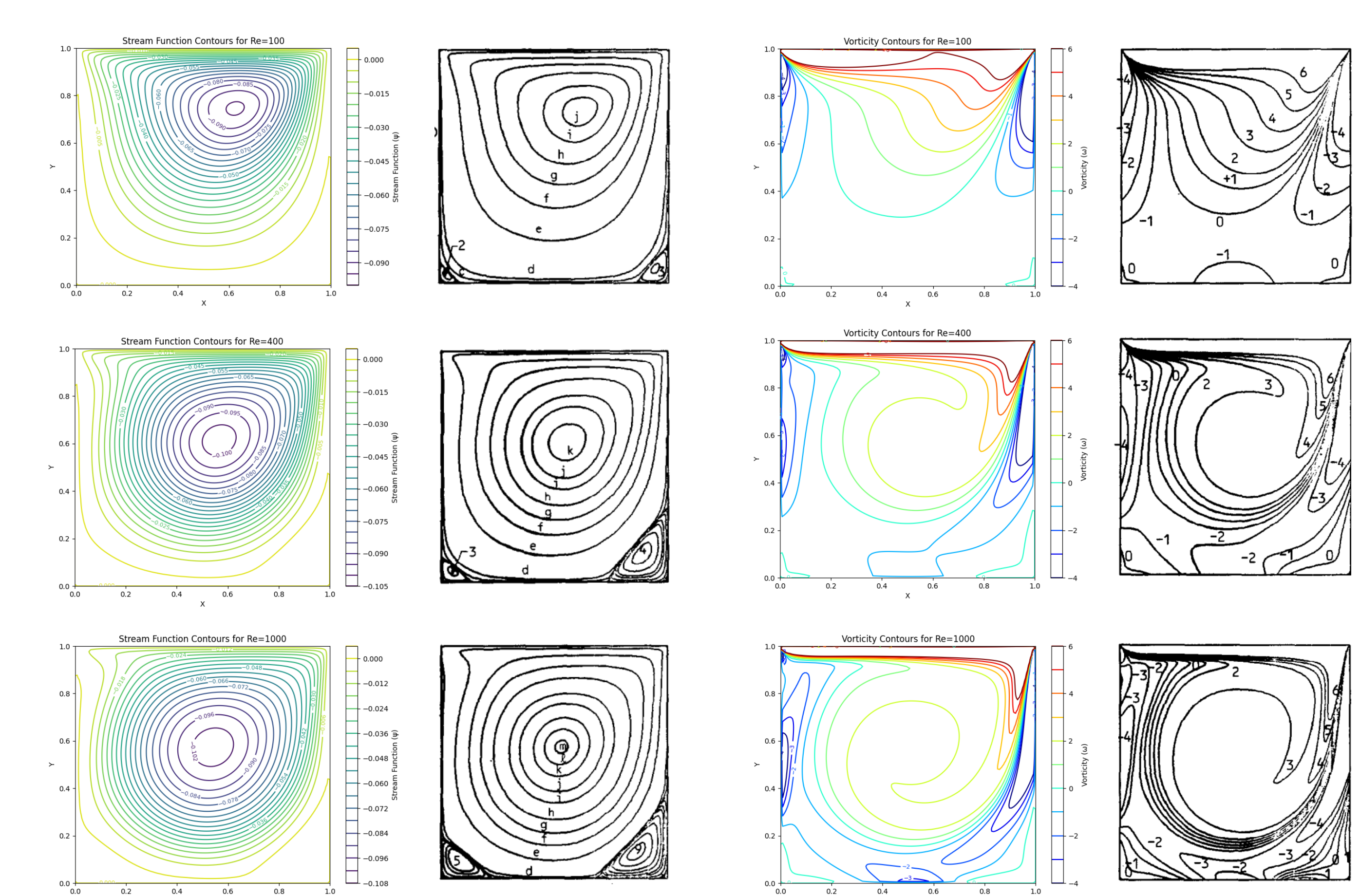
Results

- **Steady-state centerline velocity (u–velocity along vertical centerline)**

Re = 100   Re = 400   Re = 1000
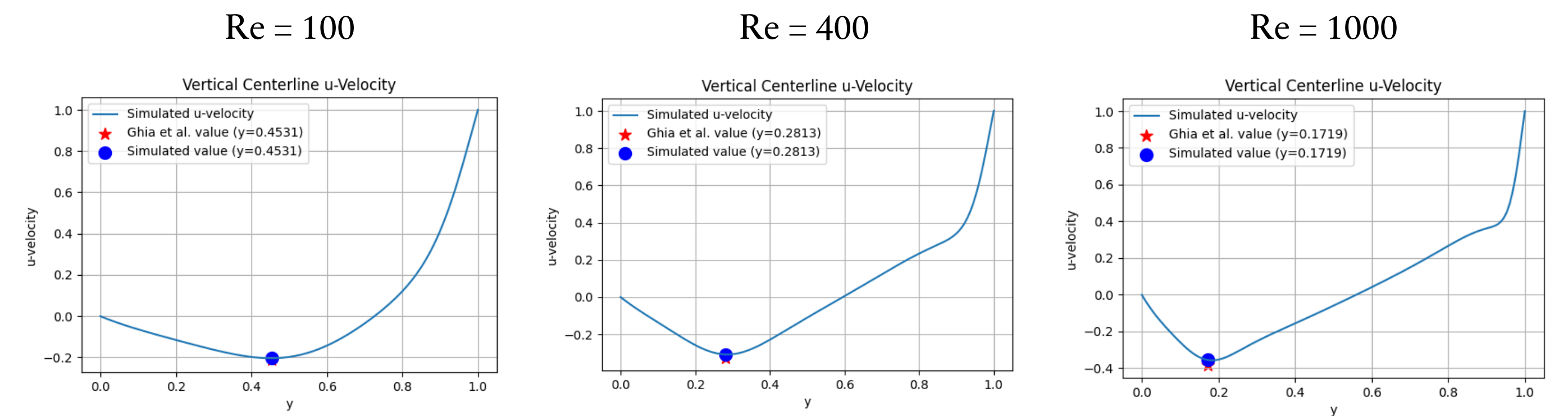
- **Steady-state centerline velocity (v–velocity along vertical centerline)**
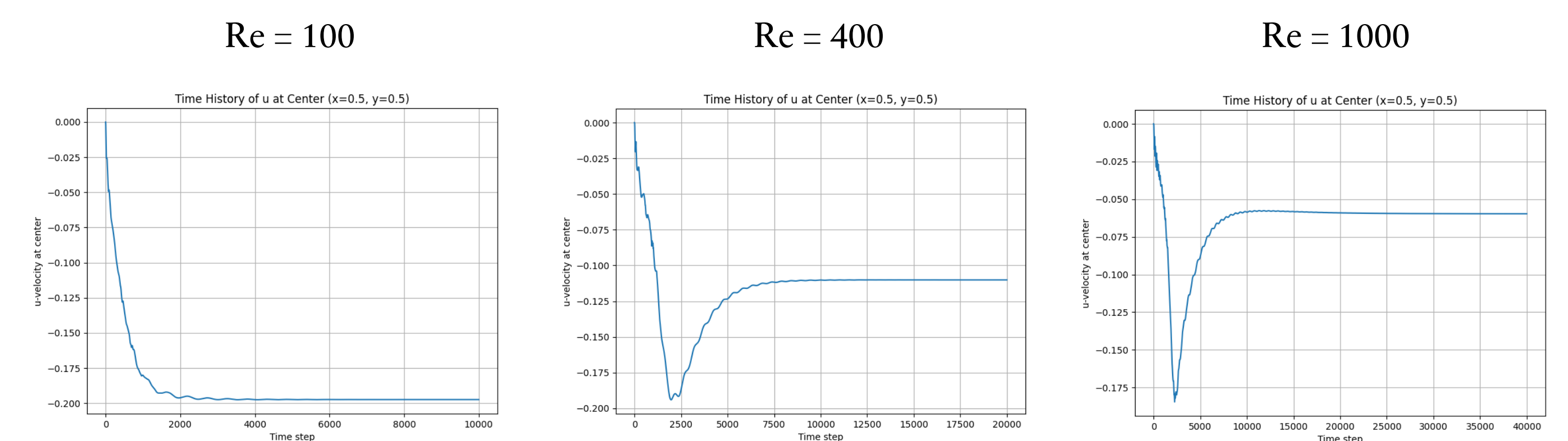
Re = 100   Re = 400   Re = 1000
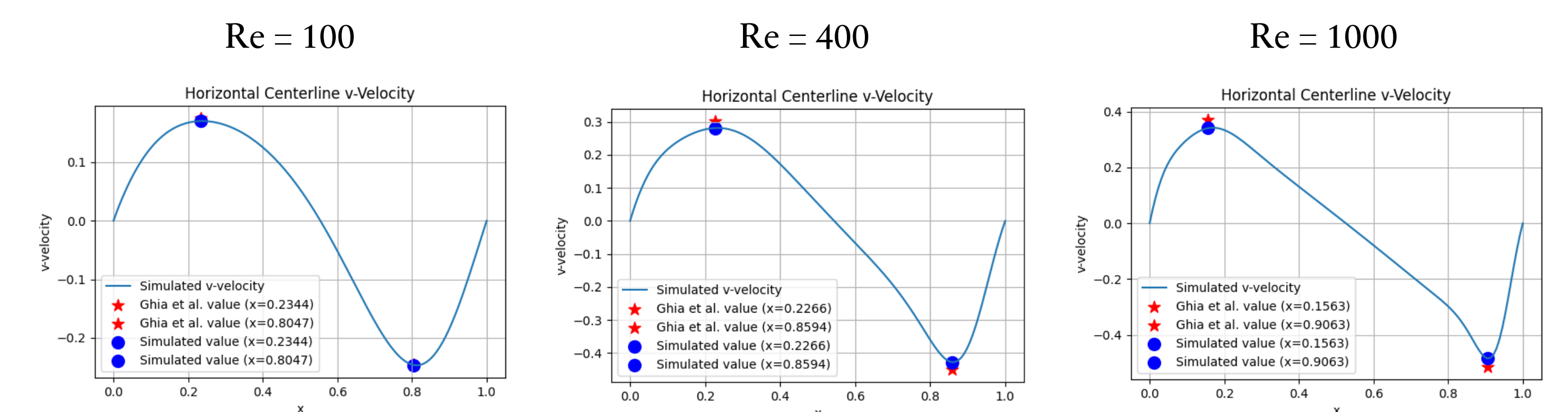
- **Steady-state streamline profile**   • **Steady-state vorticity profile**

- **Unsteady solutions (the time history of u–velocity at the center)**

Re = 100   Re = 400   Re = 1000

Discussion & Conclusions

- In this simulation, there is a slight difference when comparing the results between nx = 65 and nx = 129, whereas when comparing the results between nx = 129 and nx = 257, it was determined that the resolution was not increased further and nx, ny = 129 were judged as the optimal grid resolution.

- From the optimal grid resolution obtained based on the above, values were substituted into the CFL number equation to obtain the optimal time interval dt = 0.003.

- When Re = 100, the flow is dominated by the viscous force, so it shows a simple and predictable pattern. Only the primary vortex occurs in the form of stable laminar flow, and a small flow change appears only inside the boundary layer.

- When Re = 400, boundary layer separation may occur, and secondary vortexes may occur in addition to the primary vortex. It may be observed that the flow starts to become unstable, and the size of the vortex decreases and the number of vortex increases.

- When Re = 1000, the flow begins to approach turbulence, and the vortex is formed irregularly. The flow becomes more complex and shows an unstructured pattern. A rapid change in velocity occurs in the boundary layer, and a large number of small vortexes are formed inside the boundary layer.

- From the time history of unsteady solutions, it can be seen that the quasi-steady state is reached after t = 18s (nt = 6000) when Re = 100, t = 36s (nt = 12000) when Re = 400, and t = 60s (nt = 20000) when Re = 1000. As the number of Re increases, the required calculation becomes more complicated, and it approaches turbulence, resulting in a complex flow pattern, indicating that the required simulation time takes longer.