Fluid Solver Validation   February 23, 2018

Felix Newberry

This report describes the validation of a Fenics fluid solver. The code in question can be found in the Github repository under `/2_Cylinder_benchmark/cylinder_navier-stokes.py`.

1   Problem Setup

The fluid problem is solved with an Incremental Pressure Correction Scheme (IPCS). The fluid solver is validated with a 2D benchmark channel flow past a circular cylinder [1].

1.1   Geometry

The geometry of the problem is depicted in Figure 1.
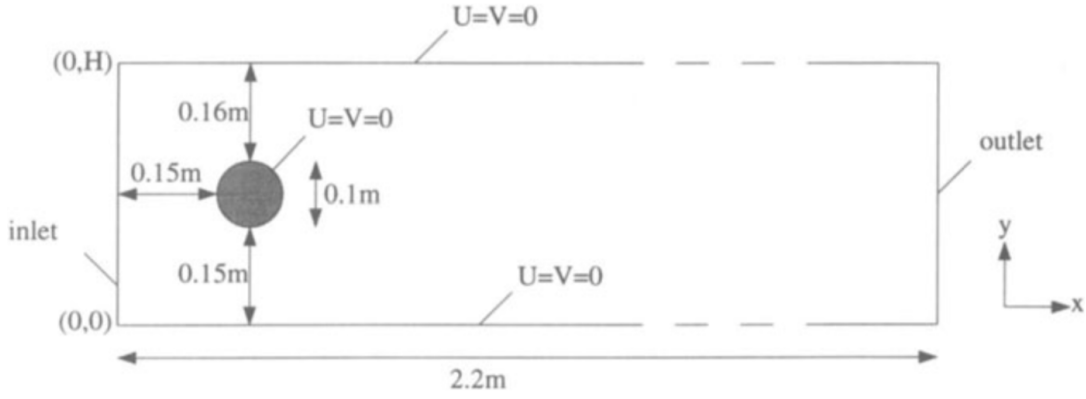

Figure 1: Fluid test case geometry [1]

A circular cylinder of diameter $D = 0.1m$ is placed slightly off center in a channel of height $H = 0.41m$. is the channel height and the

1.2   Boundary Conditions

The channel walls are no-slip condition while the outlet pressure is set to 0. The inflow is a parabolic velocity profile in the x direction defined as

$$U(0, y) = \frac{4U_m y(H - y)}{H^2}, V = 0 \tag{1}$$

Where $U_m = 0.3$ is the maximum velocity of the inlet and the mean velocity is defined as $\bar{U}$

1.3   Fluid Properties

The fluid considered is incompressible and Newtonian. The kinematic viscosity was defined as $\nu = 0.001m^2s^{-1}$ and the density as $\rho = 1.0 kgm^{-3}$.

1.4 Validation Metrics

The metrics for validation are the drag and lift coefficents c_D and c_L the change in pressure between the front and back of the cylinder. The drag and lift forces were calculated as

$$F_D = \int (\rho\nu\frac{\partial v_t}{\partial n}n_y - Pn_x)dS \tag{2}$$

$$F_L = -\int (\rho\nu\frac{\partial v_t}{\partial n}n_x - Pn_y)dS \tag{3}$$

Where S denotes the cylinder, n is the normal vector on S with x and y components n_x and n_y. v_t is the tangential velocity on S with tangent vector $t = (n_y - n_x)$ and P is the pressure.

The drag and lift coefficients are calculated

$$c_D = \frac{2F_D}{\rho\bar{U}^2 D} \tag{4}$$

$$c_L = \frac{2F_L}{\rho\bar{U}^2 D} \tag{5}$$

The change in pressure ΔP is calculated as $\Delta P(t) = P(0.15, 0.2, t) - P(0.25, 0.2, t)$.

1.5 Mesh

The Mesh used in this analysis was created in Fenics. A mesh parameter N defined both the number of facets on the cylinder and was the input resolution to the Fenics command `/generate_mesh(geometry,N)`. In addition, a refinement box was applied with vertices $(0.1, 0.1)$, and $(0.8, 0.3)$ in order to better capture the dynamics close to and in the wake of the cylinder. The respective meshes for $N = 32$ and $N = 256$ are displayed in Figure 2.
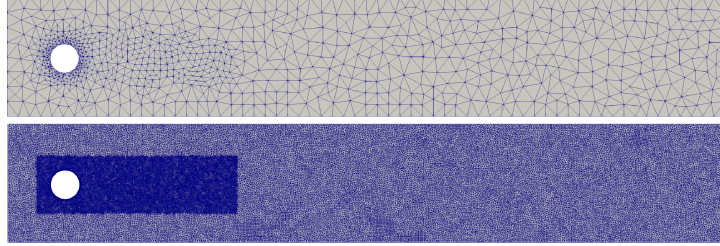

Figure 2: Course mesh, $N = 32$ and most refined mesh, $N = 256$

2 Validation

The mesh refinement results of C_D, C_L and P_{diff} are presented in Table 1. Figures 3, 4 and 5 illustrate mesh convergence to the benchmark solution for coefficients of drag and lift, and the pressure difference respectively. The red lines indicate the benchmark solution lower and upper bounds [1].

Table 1: Results of mesh independence study

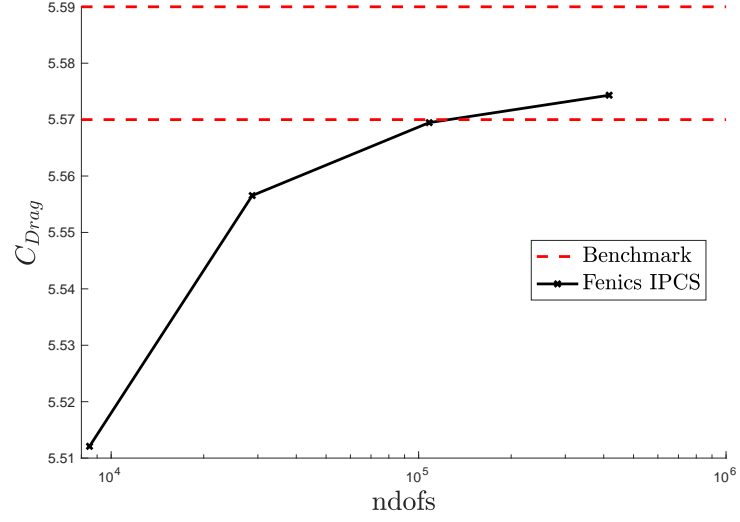| N	$ndof$	C_D	C_L	P_{diff}
32	8503	5.512	-0.01117	0.1142
64	28760	5.557	0.01018	0.1169
128	108217	5.569	0.01103	0.1173
256	416630	5.574	0.01094	0.1174

Figure 3: Drag coefficient convergence to benchmark values
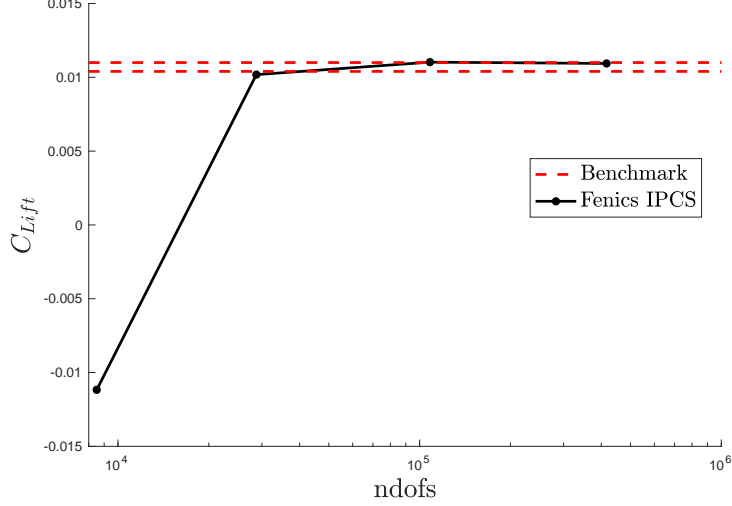
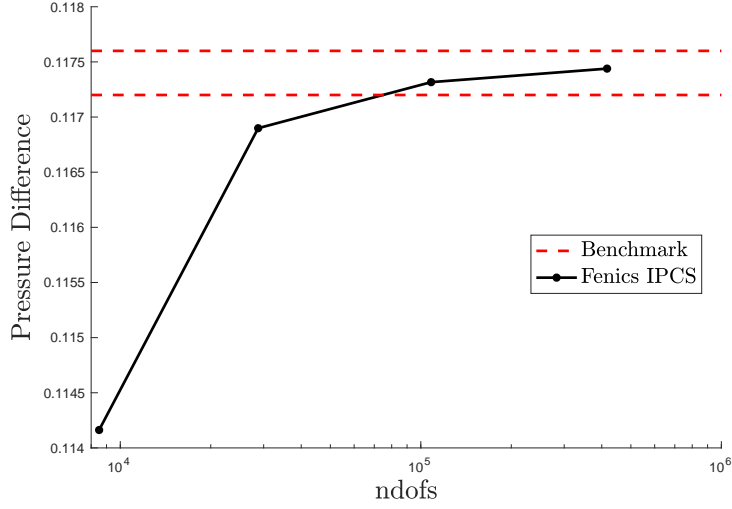Figure 4: Lift coefficient convergence to benchmark values

Figure 5: Pressure difference convergence to benchmark values

Figures 3, 4 and 5 all demonstrate convergence to the benchmark solution. Further mesh refinement is necessary to confirm mesh independence. The majority of the 17 methods presented in the benchmark solution to arrive at the bounds have a much higher grid refinement, 6 of them have an order of magnitude more more dofs.

3 FSI Application

I think these results are sufficient for application of this Fluid solver to a Fluid Structure Interaction problem. There remains some oscillatory behavior in the early development of the solution. Figure 6 depicts C_L for the initial 50 time steps on the course mesh of $N = 32$. Evidently the magnitude of the oscillations is reduced as the simulation progresses. A smaller time step causes a smaller magnitude in the oscillation.

This behavior could have important ramifications for fluid structure interaction, depending on the manner in which the oscillatory force on the structure interacts with the movement of the structure itself. If non-physical behavior does result, several solutions are available. The first is to use a smooth increase in velocity as the starting procedure. This is implemented for the non-steady solutions in the FSI

benchmark [2]. An alternative, though less elegant approach would be to run the fluid solver on its own for a set number of timesteps before introducing interaction.
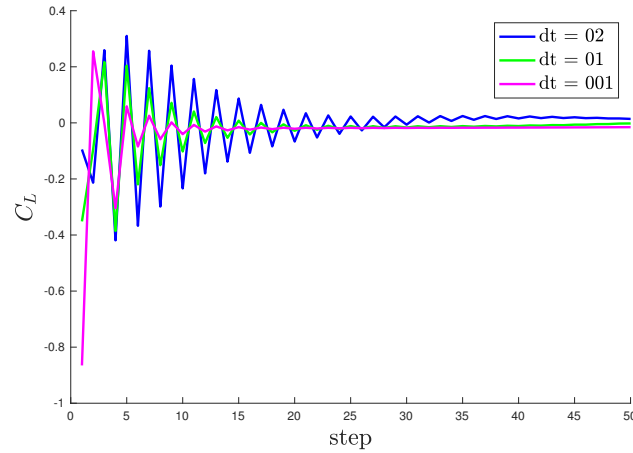

Figure 6: Oscillatory behavior in C_L with varying dt

References

[1] Michael Schäfer, Stefan Turek, Franz Durst, Egon Krause, and Rolf Rannacher. Benchmark computations of laminar flow around a cylinder. In *Flow simulation with high-performance computers II*, pages 547–566. Springer, 1996.

[2] Stefan Turek and Jaroslav Hron. Proposal for numerical benchmarking of fluid-structure interaction between an elastic object and laminar incompressible flow. *Lecture notes in computational science and engineering*, 53:371, 2006.