GCH8108E
HW5 - Steady state Navier-Stokes problems
February 27, 2025

This homework consists of solving the steady-state Navier-Stokes equations for different problems and will count towards 10% of your total grade for the semester. You need to hand in your code (in a single .zip file and without build files) and a LaTeX report describing the solution of each problem (use the template provided). **Submission deadline:** Thursday March 13th before the class (13h45).

We will use a straightforward approach. You should complete the set up of the geometry and boundary conditions of the three following problems, and the assembly of the matrices for both equations.

We have created a small parameter file where you can specify the problem you want to run (e.g., couette) and the viscosity, to avoid re-compiling the code every time. Feel free to add more parameters if needed. To run the program after compiling it, you need to run the code as follows: `./navier_stokes parameters.prm`. Remember to have the parameter file in the same path as the executable.

1 Weak form and Jacobian

Write the incompressible Navier-Stokes equations, its weak form and the final linear system solved at each nonlinear iteration.

2 Couette flow

The Couette flow problem refers to the flow of a viscous fluid in the space between two parallel plates, where one of them is moving relative to the other. We are going to use the Navier-Stokes solver to obtain the solution to this classical fluid dynamics problem.

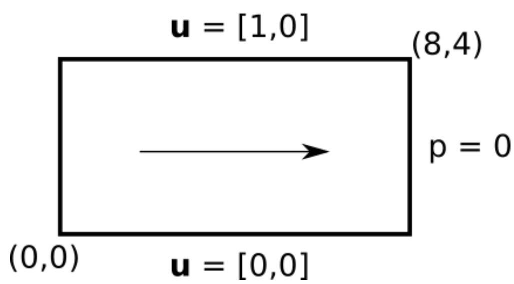

Figure 1: Schematic representing the Couette flow problem.

Taking into account the figure, create the geometry and assign the boundary conditions for the velocity and the pressure. Simulate the flow for different Reynolds number. How do you know if it converged? Do you get the expected values for the pressure and the velocity? Present the velocity and pressure profiles for two different cases: i) Re = 1 and ii) Re = 10.

3 Flow around the cylinder

This problem consists of a flow around a fixed cylinder with a constant upstream fluid velocity. We use the channel with a cylinder grid of `deal.ii` with the default parameters.

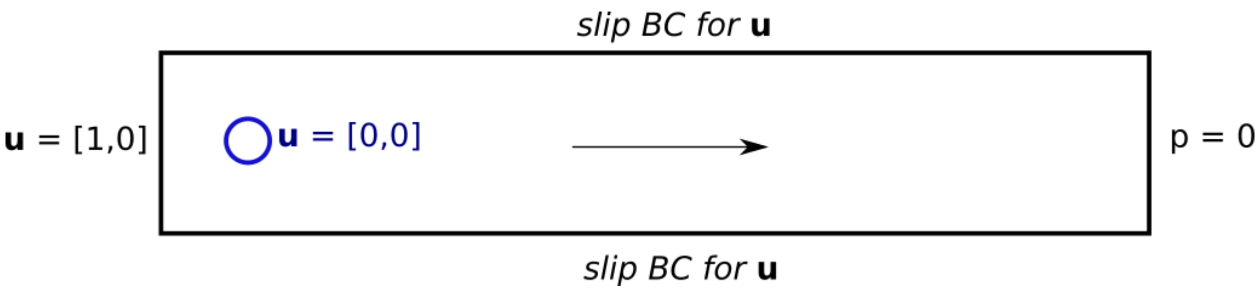

Figure 2: Schematic representing the flow around a cylinder problem.

Taking into account the figure, create the geometry and assign the boundary conditions for the velocity and the pressure. Simulate the flow at a Reynolds number of 1, 10 and 100. Present the velocity and pressure profiles of each case. How many cycles did it take for it to converge? Explain the observed physical phenomena and try to postprocess the velocity field in a creative fashion.

Bonus point (1): Calculate the force acting on the cylinder. The following code snippets should be able to help you in calculating things at faces of cells:

```
FEFaceValues<dim>  fe_face_values(fe, face_quadrature_formula,
                   update_values | update_quadrature_points |
                   update_normal_vectors | update_JxW_values);

for (const auto &cell : dof_handler.active_cell_iterators())
{
 if (cell->is_locally_owned())
 {
  for (unsigned int face_id = 0;
  face_id < GeometryInfo<dim>::faces_per_cell;
  face_id++)
  {
    if (cell->face(face_id)->at_boundary())
    {
      unsigned int boundary_id =
                 cell->face(face_id)->boundary_id();
      fe_face_values.reinit(cell, face_id);
    }
  }
 }
}
```

For this you need to create an adequate quadrature for the faces (of dimension `dim-1`).

4 Poiseuille flow

This flow has the same geometry as the Couette flow. The only difference, is that there is an inlet velocity prescribed and the walls are not moving.

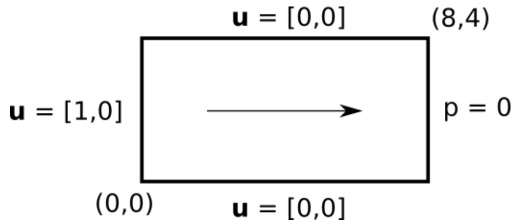

Figure 3: Schematic representing the Poiseuille flow problem.

Taking into account the figure, create the geometry and assign the boundary conditions for the velocity and the pressure. Simulate the flow at a Reynolds number of 1 and 10, and present the velocity and pressure profiles of each case. How many cycles until convergence was achieved?