

## Laminar Flow over a Backward-Facing Step

### Homework 7

**Handed out:** November 12

**Due in:** November 21

For this homework you are to solve for the steady laminar flow over a backward facing step. The flow is characterized by the Reynolds number. Many different definitions of the Reynolds number are possible for a backward-facing step. We will use the Reynolds number defined by the average inlet velocity  $\bar{U}$ , and the total height of the channel  $H$ .

$$Re = \frac{\bar{U}H}{\nu}$$

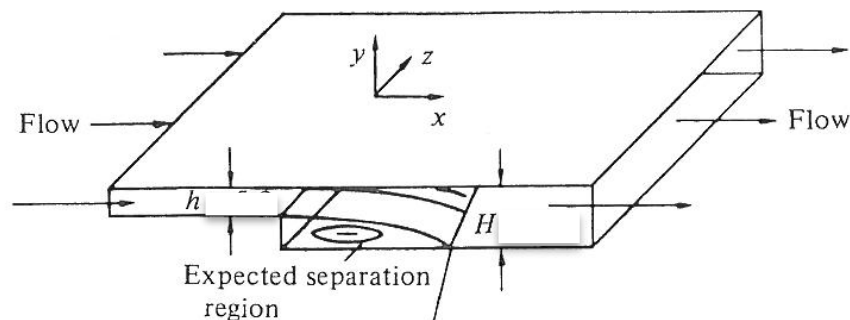


FIGURE 1. Taken from Armaly *et al.* (1983)

You will compare your results to previously-published results. The paper Gartling (1990) presents velocity profiles at two locations downstream of the step for a single Reynolds number. The geometry is such that the step height is half of the total channel height. The Reynolds number is 800. The upstream boundary condition is such that the velocity profile is parabolic.

**Step 1: Grid Generation:** Generate a two-dimensional grid for this flow using `blockMesh`. The upstream position of the domain extent should be far enough away from the step so that the velocity profile near the step is parabolic. Choose a cell size that corresponds to a mesh B from the paper by Gartling ( $\Delta x \sim 0.1$ ). You should always start with a coarse mesh. Get everything set up and working, and then it is easy to refine the mesh and monitor the sensitivity of the results with respect to the mesh spacing. You will build three meshes that should be systematically refined. You may use mesh stretching, or it can be uniform everywhere. The utility `refineMesh` will uniformly refine the grid.

**Step 2: Case Setup:** You must set up the initial and boundary conditions. The inlet velocity profile and viscosity coefficient should be chosen to produce the desired Reynolds number. If you select the average velocity to be 1 then it is easy to plot dimensionless results. Select the total time that you wish to simulate, and the frequency at which you wish to write the results. A good rule-of-thumb for the minimum length of time that is required can be taken from the average velocity and the total length of the domain, *i.e.*  $T_{\min} = L_{\text{total}}/\bar{U}$ . Ensure that you simulate for long enough to have converged statistics. You also have to choose a time-step size. Since the maximum velocity can be more than twice the average, I would estimate the largest time step that you can take using

$2\bar{U}$ , the minimum cell size, and the desired maximum Courant number. My recommendation is try  $\Delta t_{\max} = \sigma \Delta x_{\min} / 2\bar{U}$ , where desired Courant number can be in the range  $0.25 < \sigma < 1.0$ . Finally, choose appropriate discretization schemes. If your mesh is perfectly orthogonal, then you only need to be concerned with the convective discretization.

**Step 3: Execute the Simulation:** You can use the `foamJob` script to run the `icoFoam` solver and direct the output to a log file:

```
foamJob icoFoam
```

**Step 4: Examine, Plot, and Discuss the Results:** Focus on the horizontal and vertical components of the velocity vector at the station  $x/H = 7$ . Use figures and text to discuss the following aspects of your simulation:

- (1) (1pt) Length of runtime necessary for a steady solution to be reached. I suggest plotting the results for a single grid at a series of flow times.
- (2) (2pt) Grid dependence. Plot your solution on three grids. Use the same discretization settings at the same instant in time.
- (3) (2pt) Discretization settings. Plot your solution for at least two different spatial discretization schemes.
- (4) (3pt) Compare with the results of Gartling. Here plot both components of velocity from your best results. Also, comment on the difference in upstream boundary conditions. Perhaps you could plot your results at the step and compare with his profile.
- (5) (2pt) Generate a colorful picture that shows the flow field. You could use contours of velocity, streamlines, or velocity vectors. You do not need to show the whole domain, show the interesting part of the solution. Label this figure so that it is clear what you are showing.