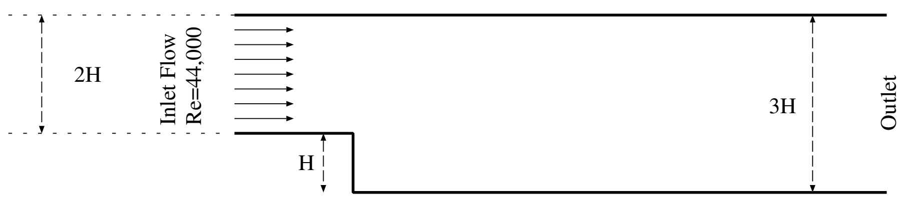
Tutorial 3: Backward Facing Step

Learning Outcomes:

Using differing RANS models in OpenFOAM

II.1 Backward Facing Step

A canonical case in the simulation of turbulent flow is the *backward facing step* problem. The domain is sketched below. In this problem, an incompressible fluid flows through a 2-d channel which starts being 2 units wide before abruptly increasing to be 3 units wide.



The reason for the interest in this problem is that the turbulent flow over the step produces a free shear layer downstream, with a recirculation region in the region behind the step. The case is easy to set up, and a great deal of experimental data is available to compare with (data for this exercise is taken from[1]).

You will attempt to simulate this problem using `simpleFoam` and compare the results from various turbulence models, specifically ;

- the standard $k-\epsilon$ model
- the RNG $k-\epsilon$ model
- the $k-\omega$ SST model
- a Reynolds Stress model; either `LRR` or `LaunderGibsonRSTM`

The `bfStep` directory contains a framework for you to use. The case will need modification; in particular the domain has an inlet of height H not 2H, and the fluid viscosity and/or inlet velocity will need to be adjusted to get the correct Reynolds number. The inlet values of k and ϵ will also need to be specified using the equations from Tutorial 1.

To run the $k-\omega$ model you will need to make some modifications to the case. In particular :

1. You need a `omega` field; this can be based on `epsilon`, so copy `0/epsilon` to `0/omega` and open it in an editor; change the dimensions (dimensions of omega are T^{-1}) and the inlet and internal values ($\omega = \epsilon/k$, so you can work these out yourself).

2. `simpleFoam` needs to know how to difference the `omega` equation. In particular you need to supply entries in `divSchemes` and `laplacianSchemes`. Again, these can be based on the entries for `epsilon`. Open `system/fvSchemes` in an editor, copy the line

   ```
   div(phi,epsilon) Gauss upwind;
   ```

   and change this to

   ```
   div(phi,omega) Gauss upwind;
   ```

   Do the same for the equivalent laplacianSchemes entry.

3. Finally `simpleFoam` needs to know how to solve the `omega` equation. This involves changing entries in `system/fvSolution` and provide entries for solvers and relaxationFactors. Again, the `epsilon` entries can be copied and changed.

For the Reynolds Stress model, an `R` field has to be specified. This can be constructed using the utility `simpleFoam -postProcess -func R` in the library. This creates a `volTensorField` representing the Reynolds Stress based on the turbulence parameters specified in the `k` and epsilon fields.

Your answer should be in the form of a short report which should cover the following points (minimum) :
[**Q.II.1**] What boundary conditions and flow properties (eg. H, viscosity) have you chosen to produce the correct Reynolds number?
[**Q.II.2**] What is the distribution of turbulent kinetic energy k and dissipation ϵ in the flow, as simulated by the standard $k - \epsilon$ model.
[**Q.II.3**] What is the length of the recirculation zone? This should be expressed as a multiple of the step height H. How does this vary between the different turbulence models?
[**Q.II.4**] You should compare velocity profiles between the different turbulence models and with experimental data.

References

[1] J. Kim, S. J. Kline, and J. P. Johnston. Investigation of a reattaching shear layer: Flow over a backward facing step. *J. Fluids Engng.*, 102:302, 1980.