Mandatory Assignment 1

Jørgen D. Tyvand

October 1, 2015

1)

2)

For both the LES and RANS models, we start from the Navier-Stokes equations for incompressible flow:

$$\nabla \cdot (\rho \mathbf{u}) = 0$$

$$\frac{\partial(\rho u)}{\partial t} + \nabla \cdot (\rho u \mathbf{u}) = -\frac{\partial p}{\partial x} + \mu \nabla^2 u$$

$$\frac{\partial(\rho v)}{\partial t} + \nabla \cdot (\rho v \mathbf{u}) = -\frac{\partial p}{\partial y} + \mu \nabla^2 v$$

Lengthy derivations of the LES and RANS equations will not be given, but a short explanation of each will give the general process.

For Large Eddy Simulation (LES), we use spatial filtering to separate varying sizes of eddies. A cutoff width Δ is introduced, for which information about eddies smaller than the given width will be ignored/destroyed. A spatial filtering using a filter finction $G(\mathbf{x}, \mathbf{x}', \Delta)$ is introduced, giving in the following form (3.84 in the book):

$$\overline{\phi}(\mathbf{x}, t) \equiv \int_{-\infty}^{\infty} \int_{-\infty}^{\infty} \int_{-\infty}^{\infty} G(\mathbf{x}, \mathbf{x}', \Delta)\phi(\mathbf{x}', t) \mathrm{d}x_1' \mathrm{d}x_2' \mathrm{d}x_3'$$

where $\overline{\phi}(\mathbf{x}, t)$ and $\phi(\mathbf{x}, t)$ are the filtered and unfiltered functions respectively. The filter function $G(\mathbf{x}, \mathbf{x}', \Delta)$ can be given in several ways, but the one used in finite volume implementations is the top-hat/box filter function

$$G(\mathbf{x}, \mathbf{x}', \Delta) = \begin{cases} \frac{1}{\Delta^3} & |\mathbf{x} - \mathbf{x}'| \leq \Delta/2 \\ 0 & |\mathbf{x} - \mathbf{x}'| > \Delta/2 \end{cases}$$

Using this filtering on the Navier-Stokes equations, we get the LES momentum equations (the intermediate step from 3.88a-c to 3.89a-c in the book for rewriting the term $\nabla \cdot (\rho \overline{\phi \mathbf{u}})$ is not shown):

$$\frac{\partial(\rho \overline{u})}{\partial t} + \nabla \cdot (\rho \overline{u}\, \overline{\mathbf{u}}) = -\frac{\partial \overline{p}}{\partial x} + \mu \nabla^2 \overline{u} - (\nabla \cdot (\rho \overline{u \mathbf{u}}) - \nabla \cdot (\rho \overline{u}\, \overline{\mathbf{u}}))$$

$$\frac{\partial(\rho \overline{v})}{\partial t} + \nabla \cdot (\rho \overline{v}\, \overline{\mathbf{u}}) = -\frac{\partial \overline{p}}{\partial y} + \mu \nabla^2 \overline{v} - (\nabla \cdot (\rho \overline{v \mathbf{u}}) - \nabla \cdot (\rho \overline{v}\, \overline{\mathbf{u}}))$$

The boundary conditions for the LES problem is as following:

Uniform velocity of 10 m/s in the x-direction at the inlet
Zero velocity gradient at the walls and outlet
Zero pressure gradient at the walls and inlet
A uniform value of 0 for the pressure at the outlet

For the simulations i have used 2 different mesh refinements, one with 10x20 cells for the inlet and outlet boxes and 100x20 for the center boxes, as well as a doubled mesh of 20x40 and 200x40 boxes. The case files were originally copied from the PitzDaily case for incompressible flow with PISO, and edited from there. For the convection term i have tested different upwind schemes, and landed on filteredLinear. An upwind scheme will include more information from upwind cells, and therefore (hopefully) give a more correct and stable calculation. I have tested the following 4 schemes for the coarsest mesh:

Non-upwind linear (as found in the unaltered PitzDaily files)
upwind
linearUpwind
filteredLinear

The following four movie files shows a simulation of these four schemes for 0.5 seconds on the 100 mesh:

The filteredLinear scheme has also been used for the 200 mesh, as well as a even finer mesh of 250x50 central boxes. As we can see from the following videos, the solutions are mot mesh independent.

I also found that the solver crashed if i used a too fine mesh with a too coarse time step. In the final calculations i have used timesteps of $1.0 * 10^{-5}$, but with a timestep of $1.0 * 10^{-4}$ the Courant number grew to between 1.5 and 2, at which the calculations crashed. This implies grid sensitivity with respect to grid size and time step size.

3)

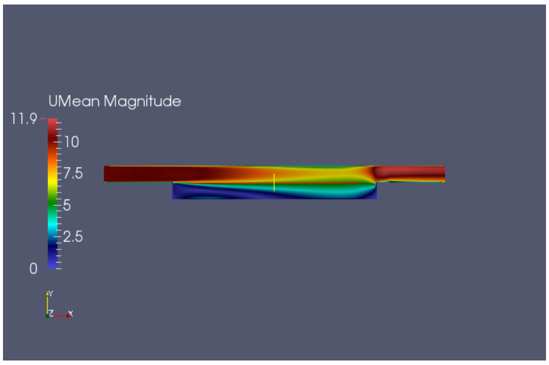
For the RANS models i have chosen to use the simpleFoam solver for the $k-\epsilon$ and $k-\omega$ models. The general RANS equations are as follows:

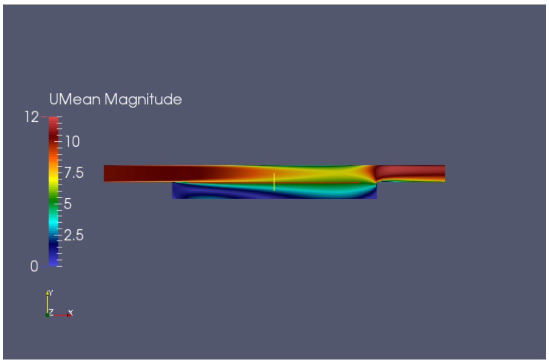
Here the overline marks the mean terms, and the marked terms are the fluctuation terms.

For the $k-\epsilon$ model, I have used the value for k that was used in the PitzDaily case. This is because the inlet velocity is the same, and i have assumed that a similar turbulence intensity is appropriate. Analyzing the value for ϵ in the PitzDaily case, i found that they have used a value of 0.1 times the inlet opening for the turbulent length scale. I have adjusted my value for ϵ using the same ratio.

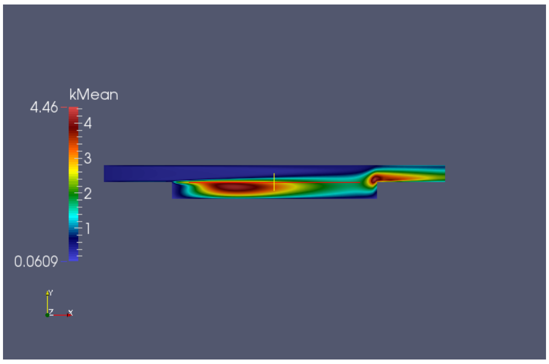
There are two additional equations to be solved for the $k-\epsilon$ model:
Running the $k-epsilon$ case, i have found the following converged solutions for the flow, using the 100 and 200 meshes respectively:

The following figures show the mean velocity for the two mesh sizes:





And finally we have the mean kinematic energy:
We see that there is virtually no difference between the two cases, so there is little or no



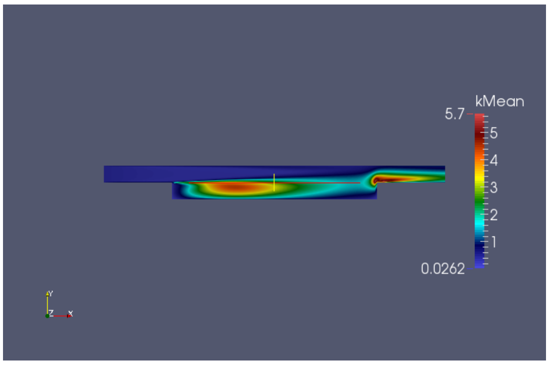
mesh sensitivity using simpleFoam for this RANS problem.

For the $k - \omega$ model, i have used the fact that $\omega = \frac{\epsilon}{k}$, and used the values from the $k - \epsilon$ problem to calculate ω

4)

5)

Since turbulence is a three-dimensional phenomenon, I would assume that some critical information could be lost using LES as a 2D model. One example could be an eddy with



primarily extension in the z-direction, that might have a width below the cutoff value in the x- or y-directions. Thus the impact of this eddy on the mean flow, which might be significant, could potentially be lost by using only a 2D approach.