**Transcript from Assignment 3**

* Wind drag coefficient
  + There is always a drag force from the moment velocity starts to increase past 0.001
  + There will be some drag/stress on the water surface
  + The values in the program is experimentally calibrated in the North Sea
    - Work well in open sea, coastal catchment
    - Might not work in marina, reservoir
* How high should we set the vertical eddy viscosity?
  + Depends on the model
    - For 1 or 2 eqn model, set 1e-6 (k-L, k-Epsilon)
    - Algebraic (zero order), k-L (first order), Constant, set 1e-3
    - As close to the actual condition as we want
* Keep top layer to be thin
  + For Deepwater, 0.1-0.5m for first layer
  + For Bed flow/sediment transport, 0.1-0.5m for bottom layer
  + One of the sigma layers calculates in the middle layer. (e.g. at 25m for a 50m deep lake)
* Why is there a problem for the code to balance the flow return?
  + There is no roughness on the wall (Free-still condition)
* Constant viscosity
  + No return flow, impact of viscosity
  + Everything has been destroyed
  + Play around with the return flow to see if we can get a return flow
  + Slope limiter function
* Depth (Constant)
  + Set-up problem
  + First 2 grid points, having issues
  + Constant can’t balance the equation, a lot of viscous stress problem
  + Constant viscosity is not suitable to solve for this problem
* Set 8x 10-4 for constant viscosity?
* First 2 grid cells have problem with the viscous dissipation
  + 4 cells out the code is doing quite well
  + If get the same vertical eddy viscosity, most slightly to get the same solution
* Correct way to stretch
  + Create a stretching ratio of 1.05 to 1.1, minimise artificial numerical scheme problem for higher order terms
* Enclosure file > can use the same from grid as the grid size is the same

**Is this solution correct? Refer to video**

* Wind dragging top layer, create gradient, somewhere in the middle, gradient of du/dz =0, not change in horizontal. There is where thermocline exist.
* Steady-state solution, wind didn’t drop off.
* Wind constant in one direction, expect to see a slight set-up
* Wind is pushing to the left, there is a return flow.
* There is a point where there is no velocity, due to return flow
* Wind is higher at the top, the return flow is more evenly distributed, only affected by bed friction
* Mass flux problem
* Why is the flow having a problem balancing a simpler return?
* No roughness > Free slip conditions; Wall is 1000m away from the next point
* Code does it solve full Naiver-Stokes equation? No. What does it actually solve? What kind of flow equation does it actually solve. Solve shallow water flow equation. We made assumptions to get to this. Characteristic Horizontal length >> Characteristic vertical length scale. Char horizontal velocity >> Characteristic velocity. Simply vertical momentum equation to a hydrostatic pressure assumption.
* Code is trying to cope, created artificial circulation
* Do it in 10 layers grid, it will be more distributed because grid cells are larger in the vertical
* So how do you help the code, use the second grid, stretch out the ends, the vertical flux coming down is smaller, because volume has increased. V will reduce because surface area going down has increased.
* Go with a lower wind, it will work for the second grid. (e.g. 0.5m/s)
* Realistic situation, start to stretch the grids, e.g. s1 grid
* Overall mixing is correct
* Try out what happens to viscosity
* Constant: 1e-6 > get a slightly different solution
* Evaluate what is happening in the middle of the grid
* Basically 4 tests, to evaluate what happens (ignore last 5 grid cells)
* Constant viscosity > no return flow coming up; impact of viscosity is that there is no viscosity. Everything has been destroyed. Play with value to see if can get a return flow.

k-Epsilon helped to stabilise the middle, but can’t deal with the sides.

Constant can’t balance the equation. Viscous stress problem. Constant viscosity is not the correct solution for this particular problem.

* Range of vertical eddy viscosity (0.1 to 1e-6); k-Epsilon at 0.1 also will have some problem
* Mass balance, first 2 grid cell, problems with viscosity dissipation
* 4 cells out, code doing quite well.
* If k-Epsilon is default, all models should get the same solution. Try out and see what comes closest. K-L can be quite similar. Algebraic depends on the values set. Constant not well liked for mixing, works well for steady.
* Why set 1e-6 for k-Epsilon, is to set a background value. Code will blow up if viscosity is at 0. Default value is 1e-8.
* Bigger dx as compared to the previous one, finite difference code, higher order terms, dx2 is very large, will have numerical diffusion problem. Trying to get the vertical out.
* Correct way to stretch, generate based on a stretching ratio of a 1.05 to 1.1. This will enable to minimize any artificial numerical problems.
* Increase size of grid by 5 times, Q=Av, A has increased by 5 times, v should decrease by 5 times. Error banding for k-Epsilon to go out to the edge. First 2 grid cells make them thin enough. Applicable for any shallow water.