TURBULENT

WAKE FLOWS

HINCHEY

PREAMBLE

At low speeds, fluid particles move along smooth paths: motion has a laminar or layered structure. At high speeds, particles have superimposed onto their basic streamwise observable motion a random walk or chaotic motion. Particles move as groups in small spinning bodies known as eddies. The flow pattern is said to be turbulent. A turbulent wake flow is one that contains some large eddies together with a lot of small ones. Such a flow could be found around the GBS on a stormy day. The large eddies generally stay roughly in one place. Fluid in them swirls around and around or recirculates in roughly closed orbits. The smaller eddies are associated with turbulence and are carried along by the local flow. The large eddies can usually be found inside wakes. Most of the smaller ones can be found near wake boundaries. They are generated in regions where velocity gradients are high like at the edges of wakes or in the boundary layers close to structures. They are dissipated in regions where gradients are low like in sheltered areas like corners. Turbulent wake flows are governed by the basic conservation laws. However, they are so complex that analytical solutions are impossible. One could develop computational fluid dynamics or CFD codes based on the conservation law equations. Unfortunately, the small eddies are so small that an extremely fine grid spacing and a very small time step would be needed to follow individual eddies in a flow. Small eddies are typically around 1mm in diameter. One would need a grid spacing smaller than 0.1mm to follow such eddies. CFD converts each governing equation into a set of algebraic equations or AEs: one AE for each PDE for each xyz grid point. Workable CFD is not possible because computers cannot handle the extremely large number of AEs generated. For example, a 100m x 100m x 100m volume of water near a structure like the GBS would need 10^6 x 10^6 x 10^6 or 10^{18} grid points if the grid spacing was 0.1mm. Also very many time steps would be needed to complete a simulation run. No computer currently exists that can handle so many grid points and so many time steps. The random motions of molecules in a gas diffuse momentum: they give gas its viscosity. Small eddies in a turbulent flow also diffuse momentum: they make fluid appear more viscous than it really is. This apparent increase in viscosity controls overall flow patterns and loads on structures. Models which account for this apparent increase in viscosity are known as eddy viscosity models. They can be obtained from the momentum equations by a complex time averaging process. The time averaging introduces the so called Reynolds Stresses into the momentum equations, and these are modelled using the eddy viscosity concept. Models have been developed which can estimate how eddy viscosity varies throughout a flow. Workable CFD is now possible because one can now use much larger grid spacing and time steps: it is no longer necessary to follow individual eddies around in a flow. When small eddies are accounted for in this way, they no longer show up in flow: they are suppressed by eddy viscosity. For the GBS case, a grid spacing around 1m would now be adequate. This means a 100m x 100m x 100m volume of water near the GBS would now need only 10^2 x 10^2 x 10^2 or 10^6 grid points.

CONSERVATION LAWS FOR HYDRODYNAMICS FLOWS

Hydrodynamics flows are often turbulent. Conservation of momentum considerations for such flows give:

$$\rho \left(\partial U/\partial t + U\partial U/\partial x + V\partial U/\partial y + W\partial U/\partial z \right) + A = - \partial P/\partial x$$

$$+ \left[\partial/\partial x \, (\mu \, \partial U/\partial x) + \partial/\partial y \, (\mu \, \partial U/\partial y) + \partial/\partial z \, (\mu \, \partial U/\partial z) \right]$$

$$\rho \left(\partial V/\partial t + U\partial V/\partial x + V\partial V/\partial y + W\partial V/\partial z \right) + B = - \partial P/\partial y$$

$$+ \left[\partial/\partial x \, (\mu \, \partial V/\partial x) + \partial/\partial y \, (\mu \, \partial V/\partial y) + \partial/\partial z \, (\mu \, \partial V/\partial z) \right]$$

$$\rho \left(\partial W/\partial t + U\partial W/\partial x + V\partial W/\partial y + W\partial W/\partial z \right) + C = - \partial P/\partial z - \rho g$$

$$+ \left[\partial/\partial x \, (\mu \, \partial W/\partial x) + \partial/\partial y \, (\mu \, \partial W/\partial y) + \partial/\partial z \, (\mu \, \partial W/\partial z) \right]$$

where U V W are respectively the velocity components in the x y z directions, P is pressure, ρ is the density of water and μ is its effective viscosity. The time averaging process introduces source like terms A B C into the momentum equations. Each is a complex function of velocity and viscosity gradients as indicated below:

$$A = \partial\mu/\partial y\ \partial V/\partial x - \partial\mu/\partial x\ \partial V/\partial y + \partial\mu/\partial z\ \partial W/\partial x - \partial\mu/\partial x\ \partial W/\partial z$$

$$B = \partial\mu/\partial x\ \partial U/\partial y - \partial\mu/\partial y\ \partial U/\partial x + \partial\mu/\partial z\ \partial W/\partial y - \partial\mu/\partial y\ \partial W/\partial z$$

$$C = \partial\mu/\partial y\ \partial V/\partial z - \partial\mu/\partial z\ \partial V/\partial y + \partial\mu/\partial x\ \partial U/\partial z - \partial\mu/\partial z\ \partial U/\partial x$$

Conservation of mass considerations give:

$$\partial P/\partial t + \rho\ c^2\ (\ \partial U/\partial x + \partial V/\partial y + \partial W/\partial z\ ) = 0$$

where c is the speed of sound in water. Although water is basically incompressible, CFD takes it to be compressible. Mass is used to adjust pressure at points in the grid when the divergence of the velocity vector is not zero.

A special function F known as the volume of fluid or VOF function is used to locate the water surface. For water, F is taken to be unity: for air, it is taken to be zero. Regions with F between unity and zero must contain the water surface. Material volume considerations give:

$$\partial F/\partial t + U\partial F/\partial x + V\partial F/\partial y + W\partial F/\partial z = 0 \quad .$$

TURBULENCE MODEL

Engineers are usually not interested in the details of the eddy motion. Instead they need models which account for the diffusive character of turbulence. One such model is the k-ε model, where k is the local intensity of turbulence and ε is its local dissipation rate. Its governing equations are:

$$\partial k/\partial t + U\partial k/\partial x + V\partial k/\partial y + W\partial k/\partial z = T_P - T_D$$

$$+ \ [\ \partial/\partial x\ (\mu/a\ \partial k/\partial x) + \partial/\partial y\ (\mu/a\ \partial k/\partial y) + \partial/\partial z\ (\mu/a\ \partial k/\partial z)\ ]$$

$$\partial \varepsilon/\partial t + U\partial \varepsilon/\partial x + V\partial \varepsilon/\partial y + W\partial \varepsilon/\partial z = D_P - D_D$$

$$+ \ [\ \partial/\partial x\ (\mu/b\ \partial \varepsilon/\partial x) + \partial/\partial y\ (\mu/b\ \partial \varepsilon/\partial y) + \partial/\partial z\ (\mu/b\ \partial \varepsilon/\partial z)\ ]$$

where

$$T_P = G\ \mu_t\ /\ \rho \qquad D_P = T_P\ C_1\ \varepsilon\ /\ k$$

$$T_D = C_D\ \varepsilon \qquad D_D = C_2\ \varepsilon^2\ /\ k$$

$$\mu_t = C_3\ k^2\ /\ \varepsilon \qquad \mu = \mu_t + \mu_l$$

where

$$G = 2 \left[(\partial U/\partial x)^2 + (\partial V/\partial y)^2 + (\partial W/\partial z)^2 \right] + \left[\partial U/\partial y + \partial V/\partial x \right]^2 + \left[\partial U/\partial z + \partial W/\partial x \right]^2 + \left[\partial W/\partial y + \partial V/\partial z \right]^2$$

where C_D=1.0 C_1=1.44 C_2=1.92 C_3=0.9 a=1.0 b=1.3 are constants based on data from geometrically simple experiments, μ_l is the laminar viscosity, μ_t is extra viscosity due to eddy motion and G is a production function. The k-ε equations account for the convection, diffusion, production and dissipation of turbulence. Special wall functions are used to simplify consideration of the sharp normal gradients in velocity and turbulence near walls.

COMPUTATIONAL FLUID DYNAMICS

For CFD, the flow field is discretized by a Cartesian or xyz system of grid lines. Small volumes or cells surround points where grid lines cross. Flow is not allowed in cells occupied by fixed bodies. Ways to handle moving bodies are still under development. Flow can enter or leave the region of interest through its boundaries. For hydrodynamics problems, an oscillating pressure over a patch of the water surface could be used to generate waves. An oscillating flow at a vertical wall could also be used for this. For CFD, each governing equation is put into the form:

$$\partial M/\partial t = N \quad .$$

At points within the CFD grid, each governing equation is integrated numerically across a time step to get:

$$M(t+\Delta t) = M(t) + \Delta t \; N(t)$$

where the various derivatives in N are discretized using finite difference approximations. The discretization gives algebraic equations for the scalars $P \; F \; k \; \varepsilon$ at points where grid lines cross and equations for the velocity components at staggered positions between the grid points. Central differences are used to discretize the viscous terms in the momentum and turbulence equations. To ensure numerical stability, a combination of central and upwind differences is used for the convective terms. Collocation or lumping is used for the T and D terms. To march the unknowns forward in time, the momentum equations are used to update $U \; V \; W$, the mass equation is used to update P and correct $U \; V \; W$, the VOF equation is used to update F and the location of the water surface and the turbulence equations are used to update $k \; \varepsilon$.

APPLICATIONS OF FLOW-3D CODE

FLOW-3D is a CFD software package for hydrodynamics and other flows <www.flow3d.com>. It can handle all sorts of complex phenomena such as wave breaking and phase changes such as vaporization and solidification. No other CFD package can handle these phenomena. A new feature known as the General Moving Object or GMO can simulate the complex motions of floating bodies in steep waves. The motions of the bodies can be prescribed or they can be coupled to the motion of the fluid. It allows for extremely complicated motions and flows. One can think of a GMO as a bubble in a flow where the pressure on the inside surface of the bubble is adjusted in such a way that its boundary matches the shape of a body. FLOW 3D uses a complex interpolation scheme to fit the body into the Cartesian grid. The sketch on the next page shows a FLOW-3D simulation of an oil rig in waves.