ISSN 2229-5518

CFD Simulation Comparison of Regular Wavy Viscous Flow Model with Different Reynolds Number around Cylinders

< Arindam Mandal>1, <Rajesh Mondal >2

1 Assistant Professor, Uttar Banga Krishi Viswavidyalaya,
W.B.-741235, India
Email:arindamubkv@gmail.com

2 Engineer, SembCorp Marine Ltd,
29, Tanjong Kling Road, Singapore
Email:rajeshmondl@gmail.com

Abstract: This paper focus on a simulation comparison of regular wavy flow with different Reynolds number (Re) around multiple cylinders. Fluid flow phenomenon is generally chaotic with different viscosity and small change of shape, surface roughness. This phenomenon can be simulated using two different Reynolds number on the CFD model incorporated with volume of fluid method (VOF) in Fluent Software. In this case, two different models, laminar and Reynolds Averaged Navier-Stokes (RANS) turbulence model are considered for the simulation and comparison. To realist the wave phenomenon in ocean, Stokes theory is used. Stokes theory is closer to approximate the regular wavy flow. To simulate the regular wavy flow, volume of fluid approach is employed. Volume of fluid is able to compute free surface flow along with wave breaking and impact. It is quite appropriate method to compute grid cells near the free surface which are partially filled with air and water according to state of art. The models (viscous laminar, k epsilon and Reynolds average turbulence model) are implemented with wavy flow around cylinder and compared the result of volume fraction and pressure profile in regular wave conditions.

KEY WORDS: FSI, VOF, Stokes Wave Theory, Open channel boundary conditions, Volume Fraction, Pressure

1. Introduction

In this paper, a CFD model has been used to perform the wavy flow simulation around multiple cylinders. Wavy flow is very chaotic and random at the time when it passes through the structural body. Practically, the supporting members of the structure near to the free surface of the water are subjected, more dominantly, to wave induced loads. Gradually, this loads leads to the fatigue of existing costal and ocean structures. It is important to take into account the fluid response on cylinders for structural assessment [4][5].

The CFD model is created with an appropriate dimension in global coordinate system. The diameter of the cylinder is 5m and centre to centre spacing is 20m. In ANSYS Workbench [1], design modeller is used as a pre-processing tool for creating the whole computational domain and mesh tool is used to generate volume mesh. In order to model the regular ocean wave, fifth order stokes's wave theory is used in ANSYS Fluent [1][3]. The free surface of the water phase is tracked by using the volume of fluid (VOF) technique in Fluent [1][2].

In ANSYS Fluent, open channel wave boundary conditions allow simulating the propagation in regular wave condition which is useful in the marine industry for analyzing wave kinematics and wave impact loads on moving bodies and offshore structures.

Assume the 5th order stokes's wave is an approximation of a real ocean wave. It is also assumed that structure is fixed along translation and rotational motion inside the fluid domain.

The models (viscous laminar, k epsilon and Reynolds average turbulence model) are used to simulate the wavy flow around cylinder and the result is compared for volume fraction and pressure.

2. GOVERNING EQUATION:

Mathematical Description of Volume of Fluid Method: According to references [1] and [2], the fluid domain is considered as a multiphase volume of fluid (VOF) model. The VOF can model two or more immiscible fluids by solving a single set of momentum equations and tracking the volume fraction of each of the fluids throughout the domain. The tracking of the interface(s) between the phases is accomplished by the solution of a continuity equation for the volume fraction of one (or more) of the phases. For the q^{th} phase, this equation has the following form:

$$\frac{1}{\rho_q}\left[\frac{\partial}{\partial t}(\alpha_q\rho_q)+\nabla.(\alpha_q\rho_q\vec{v}_q) = S_{\alpha_q}+\sum_{p=1}^{n}(\dot{m}_{pq}-\dot{m}_{qp})\right] \ldots (1)$$

Where, $\dot{m}_{pq}$ is the mass transfer from phase q to phase p and $\dot{m}_{pq}$ is the mass transfer from phase p to phase. By default, the source term on the right-hand side of (1), S_{α_q} is zero. The volume fraction equation is solved through implicit time discretization.

In the implicit formulation, the volume fraction equation is discretized in the following manner:

$$\frac{\alpha_q^{n+1}\rho_q^{n+1}-\alpha_q^{n}\rho_q^{n}}{\Delta t}V+\sum_f(\rho_q^{n+1}U_f^{n+1}\alpha_{q.f}^{n+1}) = \left[S_{\alpha_q}+\sum_{p=1}^{n}(\dot{m}_{pq}-\dot{m}_{qp})\right] \ldots . . (2)$$

where,

$n+1$ =index for current time step, n= index for previous time step, α_q^{n+1}=cell value of the volume fraction at time step $n+1$, α_q^{n}= cell value of volume fraction at time step n, $\alpha_{q.f}^{n+1}$= face value of the q^{th} volume fraction at time step $n+1$, U_f^{n+1}=volume flux through the face at time $n+1$, V= cell volume

Stokes's Wave Theories: The wave profile for a higher wave is given as, $\zeta(X,t) = A\cos\alpha + A^2k(b_{22}+A^2k^2b_{24})\cos2\alpha + A^3k^2(b_{33}+A^2k^2b_{35})\cos3\alpha + A^4k^3\cos4\alpha + A^5k^4\cos5\alpha$, where $\alpha = k_x x + k_y y - \omega_e t + \varepsilon$, x and y are the space coordinates in the $\hat{x}$ and $\hat{y}$ direction, respectively, ε is the phase difference, and t is the time.

The generalized expression for the associated velocity potential for shallow/intermediate waves is,

$$\Phi(X,t) = c\begin{bmatrix} A\,(a_{11}+A^2k^2a_{13}+A^4k^4b_{15})\cosh khsin\alpha + \\ A^2k(a_{22}+A^2k^2a_{24})\cosh 2khsin2\alpha + \\ A^3k^2(a_{33}+A^2k^2a_{35})\cosh 3khsin3\alpha + \\ A^4k^3\,(a_{44})\cosh 4khsin4\alpha + \\ A^5k^4\,(a_{55})\cosh 5khsin5\alpha + \end{bmatrix} \ldots (3)$$

The wave frequency ω is defined as follows for the shallow/ intermediate waves:

$$\omega = \sqrt{gk\,(1+A^2k^2c_3+A^4k^4c_5)\tan(kh)} \ldots (4)$$

where c is the wave speed, h is the liquid height, k is the wave number, a_{mn} , b_{mn} , c_{mn} are functions of wave length and liquid height for shallow/intermediate waves and constant values for short gravity waves and g is the gravity.

The velocity components for the incident wave boundary condition can be described in term of shallow/ intermediate waves.

The velocity components for shallow/ intermediate waves is given as,

$$u = \frac{\partial\Phi}{\partial x}\cos\theta,\ v = \frac{\partial\Phi}{\partial y}\cos\theta,\ u = \frac{\partial\Phi}{\partial z}\cos\theta \ldots (5)$$

Laminar Model: Laminar flow is governed by the unsteady Navier-Stokes equations. The laminar option does not apply a turbulence model to the simulation and is only appropriate if the flow is laminar. This typically applies at low Reynolds number flows. Energy transfer in the fluid is accomplished by molecular interaction (diffusion). In the case of high speed flows, the work of the viscous stresses can also contribute to the energy transfer.

Reynolds Averaged Navier-Stokes (RANS) Turbulence Model: RANS models solve additional transport equations for turbulence and introduce an eddy viscosity (also known as turbulent viscosity) to the simulation to mimic the effect of turbulence.

(i) k epsilon Model:

For kinetic energy k,

$$\frac{\partial(\rho k)}{\partial x}+\frac{\partial(\rho k u_i)}{\partial x_i} = \frac{\partial}{\partial x}\left[\frac{\mu_t}{\sigma_k}\frac{\partial k}{\partial x_i}\right]+2\mu_t E_{ij}E_{ij}-\rho\epsilon \ldots (6)$$

ISSN 2229-5518

For dissipation ϵ ,

$$\frac{\partial(\rho\epsilon)}{\partial x}+\frac{\partial(\rho\epsilon u_i)}{\partial x_i}=\frac{\partial}{\partial x}\left[\frac{\mu_t}{\sigma_\epsilon}\frac{\partial\epsilon}{\partial x_i}\right]+C_{1\epsilon}\frac{\epsilon}{\mathrm{k}}2\mu_t E_{ij}E_{ij}-C_{2\epsilon}\rho\frac{\epsilon^2}{\mathrm{k}}\ldots\ldots(7)$$

where,

u_i represents velocity component in corresponding direction

E_{ij} represents component of rate deformation

μ_t represents eddy viscosity

(ii) Reynolds- Average Model: It is a time average equations of motion for fluid flow.

$$\rho\bar{u}_j\frac{\partial\bar{u}_i}{\partial x_j}=\rho\bar{f}_j+\frac{\partial}{\partial x_j}\left[-\bar{p}\delta_{ij}+\mu\left(\frac{\partial\bar{u}_i}{\partial x_j}+\frac{\partial\bar{u}_i}{\partial x_j}\right)+\rho\overline{u_i'u_j'}\right]\ldots.(8)$$

where,

f_i is a vector representing external forces.

δ_{ij} is Kronecker delta

3. SIMULATION MODEL:

3.1 Geometry Model

The geometrical model and the coordinate point of interest have been shown in Figure1. The nine cylindrical structures are arranged in equal distance. The gap between the cylinders is 20m. A rectangular domain is created to conduct the simulation around the cylinders.

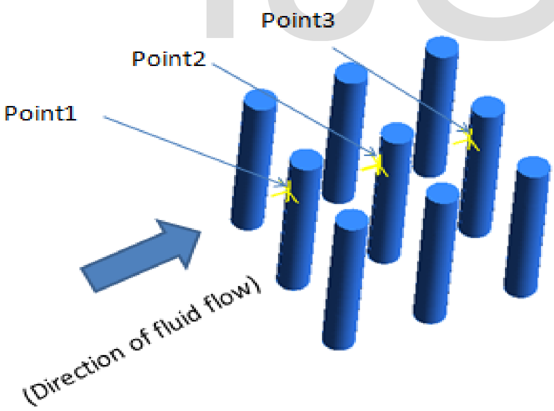


Figure 1, Cylinders, flow direction and points of interest

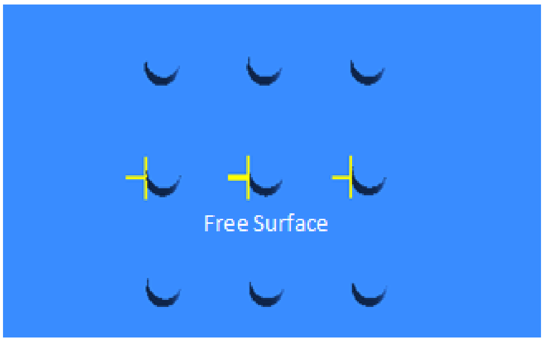


Figure 2, free surface and points of interest

The details of the simulation model are explained from Table1 to Table 4.

Table1: Main particular of cylinder structures

Cylinder length	65m
Cylinder diameter	10m
Under water length	50

Table 2: Environmental condition and material properties setting:

Open channel wave boundary condition		Material properties of air, water and structure	
Wave length	40m	Water density	1025 m/s
Wave height	2m	Air density	1.225 m/s
Number of wave	2	Dynamic viscosity of water (a)	1.007E-3 kg/ms
Wave heading angle	0	Dynamic viscosity of water (b)	1 kg/ms
Water velocity	4 m/s	Dynamic viscosity of air	1.789E-5 kg/ms
Water depth	80m	Density of steel	7850 kg/m^3
Free surface height	11m	Poisson's ration	0.3
Turbulence Intensity	5	Gravity	9.81 m/s^2
Turbulence Viscosity Ration	10	Number of Phase	2

ISSN 2229-5518

3.2 Mesh generation:

The computational mesh of the structural member is created with the help of ANSYS Fluent meshing tool. The volume mesh with tetrahedral cells is generated. The tetrahedral cell belongs to the unstructured mesh. The contact region is refined with smaller size mesh to successfully share the information between fluid and structure.

Table3: Mesh data

Fluid Mesh		Structural Mesh	
Number of total nodes	115150	Number of total nodes	12948
Number of total elements	108192	Number of total elements	37822

Table4: Solver set up

Fluent solver set up	
Pressure-velocity coupling	PISO
Spatial discretization	
Gradient	Least Squares Cell Bases
Pressure	PRESTO!
Momentum	Second order upwind
Volume fraction	Compressive
Transient Formulation	First Order Implicit

4. Solution Method:

Pressure-based solver: The pressure –based solver is applied to solve incompressible flow field.

If the only unknowns in a given equation are assumed to be for a single variable, then the equation set can be solved without regard to the solution of other variables.

In the discretization, second –order upwind interpolation scheme for the convection term because it use large stencils for 2nd order accuracy and it is essential with tri/tet mesh or when flow is not aligned with grid.

Gradients of solution variables are required in order to evaluate diffusive fluxes, velocity derivatives, and for higher-order discretization schemes. The gradients of solution variables at cell centers can be determined Least-Squares Cell-Based.

Interpolation schemes for calculating cell-face pressures when using the segregated solver in Fluent. PRESTO is applied as it is suitable for highly swirling flows, flows involving steep pressure gradients (porous media, fan model, etc.), or in strongly curved domains.

The algorithm, Pressure-Implicit with Splitting of Operators (PISO) chooses for unsteady flow problems. It is suitable to control the meshes containing cells with higher than average skewness.

Implicit pressure-based scheme (PISO):

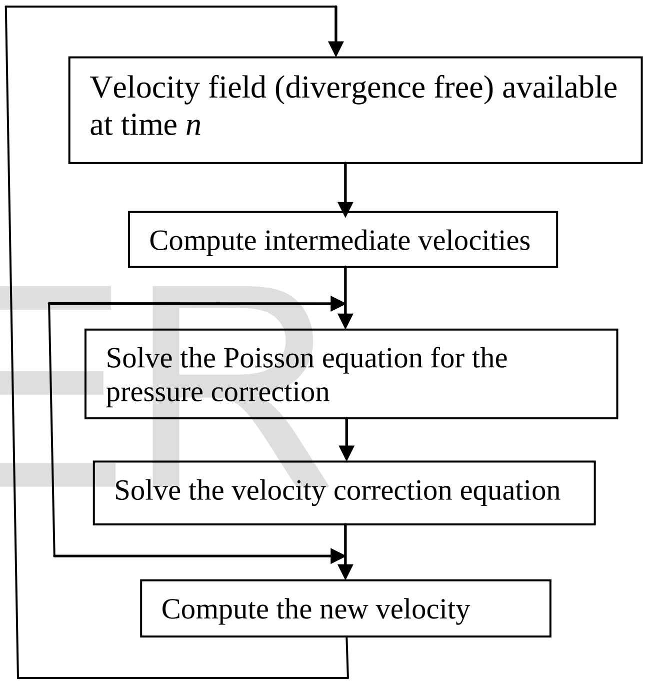
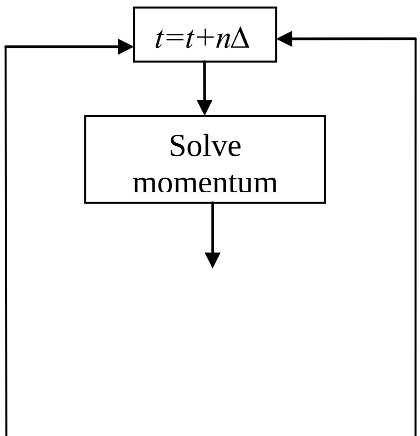


Figure 3, Flow Chart for PISO Solver

Iterative Time Advancement:



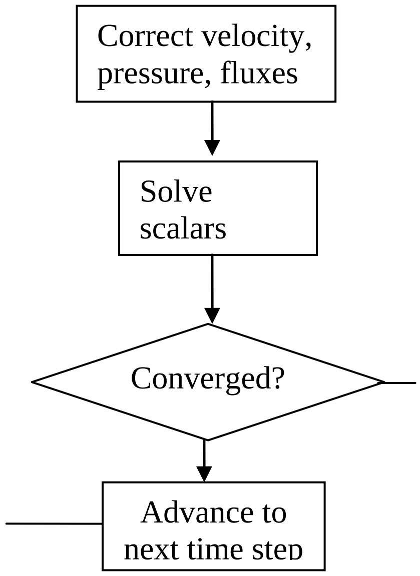


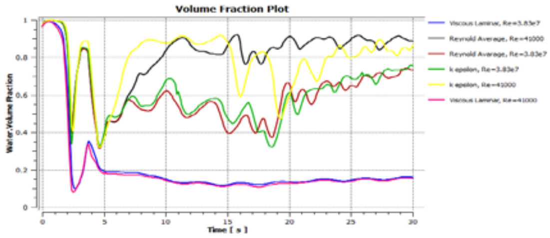
Figure 4, Iterative Time Advancement

RESULT AND DISSCUSSION:

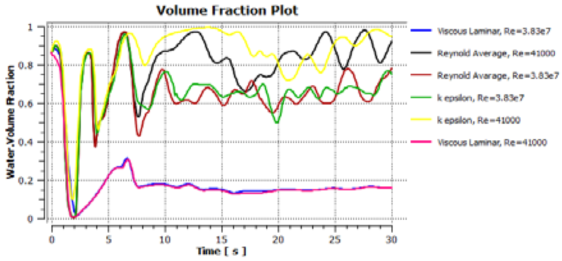
The total simulation time is 0 to 30s with the step size 0.1s. Graphical representation of volume fraction and corresponding pressure values at coordinate point1 (-34.9276, -0.0620768, 0.260382) point2 (-5.05432, 0.23094, 0.279969), point3 (24.9994, 0.000110158, 0.000376544) has been plotted in Figure 2 and Figure 3 respectively. The comparisons of all the models have been discussed as follows.

The volume fraction in viscous laminar model is not much changed in the points of interests with the change of Reynolds Number and the increment of time but a big change is found in RANS turbulent model (Reynolds Average model and k epsilon model) with the change of Reynolds Number. On the other hand, there is a large difference in volume fraction between laminar and turbulent model with the increment of time and Reynolds numbers. In the viscous laminar model, initially the volume fraction has been fluctuated much but with inclement of time it is showing a linear movement. In the RANS turbulent model (Reynolds Average model and k epsilon model), initially the volume fraction has been fluctuated but with inclement of time it is a keeping on the fluctuation. It is clear from the simulation that it makes instability in the model due to turbulence with viscous effect. The corresponding pressure plot has been shown in the figure 4.

At Point1,



At Point2,



At Point3,

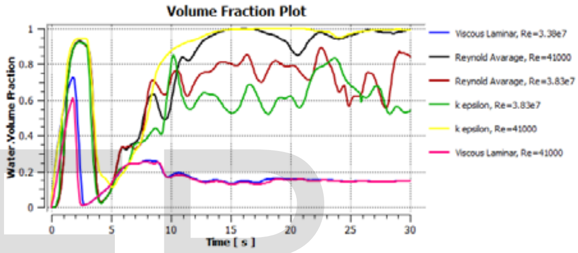
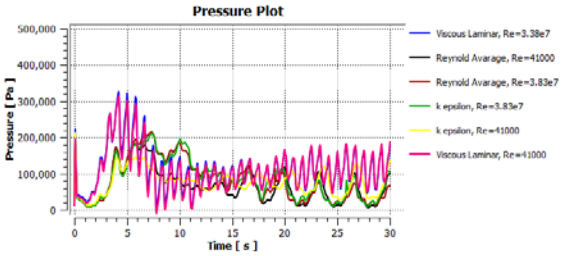


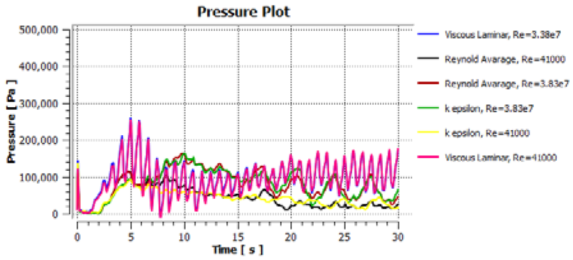
Figure 3: Volume fraction plot

The maximum pressure values are found at the simulation time 4.9s in the laminar model and at 7s in the RANS turbulent model. There is no much difference at laminar model in fluctuation of pressure with the increment of time and Reynolds Number but little difference in turbulence model for the low and high Reynolds Number. The pressure values in RANS turbulent model (Reynolds Average model and k epsilon model) are quite difference from the laminar model. It is observe that pressure values have reduced gradually towards point3 because of the disturbance of the fluid velocity by the point1 and point2's cylinder for both the laminar and RANS turbulence model.

At Point1,



At Point2,



At Point3,

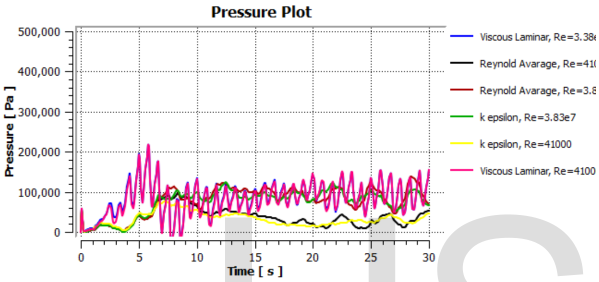


Figure 4: Pressure Plot

CONCLUSION:

The paper presents a CFD analysis using volume of fluid with the finite volume techniques. The simulation is conducted for 30s with larger time step due to limited computational resource. The result is quite good for cylinder from the structural point of view. This result may not be appropriated because of larger time step and coarse size mesh. But the result indicates that the procedure is consistent about the prediction of the laminar and turbulent model.

REFERENCES:

[1]ANSYS 16 User's Guide 2015
[2]FLUENT 6.3 User's Guide, September 2006
[3]Rammohan Subramania Raja, Thesis: Couple fluid structure interaction analysis on a cylinder exposed to ocean wave loading, department of applied mechanics, Chalmers University of Technology, Sweden 2012.
[4]https://en.wikipedia.org/wiki/Reynolds_number
[5] Tansley, Claire E.; Marshall, David P. (2001). "Flow past a Cylinder on a Plane, with Application to Gulf Stream Separation and the Antarctic Circumpolar Current" *(PDF)*. Journal of Physical Oceanography. 31 (11):3274–3283.