ISSN 2176-5480

22nd International Congress of Mechanical Engineering (COBEM 2013)
November 3-7, 2013, Ribeirão Preto, SP, Brazil

NUMERICAL INVESTIGATION OF VORTEX SHEDDING ON MARINE RISERS

Julianna Vulcão da Silva
Naval Engineering Department - Federal University of Pará – Belém, CEP: 66075-110, Pará, Brazil.
vsjulianna@gmail.com

Danielle Regina da Silva Guerra
Mechanical Engineering Department – Federal University of Pará – Belém, CEP: 66075-110, Pará, Brazil.
daguerra@ufpa.br

Hamilton Pessoa Picanço
Naval Engineering Department - Federal University of Pará – Belém, CEP: 66075-110, Pará, Brazil.
hamilton.picanco@gmail.com

Abstract. *Marine risers are ocean cylindrical structures employed in the offshore industry in important sectors of the oil production chain and are subjected to severe environmental conditions. A recurring and prejudicial effect to these structures is the vortex shedding that directly influences their design parameters in addition to compromise the structural integrity and of the fixation systems due to the action of periodic forces that can cause fatigue failure. This paper presents the flow on marine risers by transient and two-dimensional numerical investigation performed with the commercial code FLUENT 14.0 for a range of high Reynolds numbers. The modeling of the problem considering the flow on a rigid and fixed cylinder with a characteristic diameter of steel catenary risers and drilling risers for which are delimited the dimensions of the computational domain and numerical mesh. The results show the pressure and velocity fields and quantitative data for forces and drag and lift coefficients that provide the understanding of the formation of vortices, aspects of oscillatory forces and frequency of vortex shedding, generation of alternate movements and support the study of the phenomenon vortex-induced vibration and interference of the flow on adjacent structures.*

Keywords: *numerical simulation, vortex shedding, marine risers, CFD, offshore structures.*

1. INTRODUCTION

Oil exploration in regions with rigid environmental conditions need to ensure the use of structures and materials that will withstand the natural characteristics of the environment in which they operate. Marine risers are ocean cylindrical structures that connect the seabed installation to the floating system and are subjected to adverse environmental conditions and act on important sectors of the supply chain of oil and gas, such as drilling, production and extraction. One of the environmental conditions responsible for the structural instability of these structures is the action of ocean currents that originate forces and periodic movements that influence their behavior detrimentally, inducing vibrations in the structure known as vortex induced vibrations (VIV) and causing the risk of fatigue failure.

The purpose of this study is to assess the characteristics of vortex shedding on ocean cylindrical structures through numerical simulation to predict hydrodynamic coefficients and forces. The work was performed through numerical commercial code FLUENT 14.0 and preprocessor GAMBIT 2.3.16.

Most flow problems related to vortex shedding on risers are in subcritical flow regime (300 <Re <3E+05) and cases of large scales in critical (3E+05<Re<3.5E+05) and supercritical regime (3.5E+05<Re<1.5E+06). The technique of computational fluid dynamics (CFD) is an alternative used to check the influence of these types of flows using the modeling of the risers as a rigid and fixed circular cylinders to better understand the action of the fluid on the structure, determine the necessary results inherent phenomenon and establishing a database for evaluation of design conditions.

The flow around an isolated cylinder is the theoretical basis for the study of the effects of vortex shedding on risers. Sumer and Fredsoe (2006) presented the main features of the hydrodynamics around ocean cylindrical structures. The authors discussed the behavior of the flow on the cylinder when subjected to constant current, the oscillatory flow, the regular and irregular waves, mathematical and numerical treatment of flow around cylinders and physical and mathematical modeling to evaluate vortex induced vibrations due the effects of wave and current.

Norberg (2003) presented a compilation of results for circular cylinder about the Strouhal number, r.m.s. lift coefficient, pressure distribution and r.m.s. pressure coefficient. The results for the r.m.s. lift coefficient were very important for the validation of the present work.

These works were important for comprehension the oscillatory behavior of the drag and lift coefficients and the Strouhal number, knowledge necessary for the development of the necessary steps to investigate the numerical study of the vortex shedding on risers.

2. NUMERICAL INVESTIGATION

In this paper, the study of vortex shedding on marine risers consisted in carrying out a numerical investigation of two-dimensional unsteady flow over circular cylinder for high Reynolds number in the subcritical flow regime using the turbulence model k-ω SST.

Initially the characteristic diameter was selected based on the dimensions of catenary and drilling risers and through it established the dimensions of the calculation domain, set up the mesh, the mesh convergence study for the Re=2.72E+05 and mesh quality evaluation. Then, the mesh was applied to selected cylinder configuration isolated and it simulations were performed with variation in Reynolds number and a range of speeds within a subcritical flow regime.

2.1 Mathematical modeling

The mathematical modeling of a physical problem is the representation of all important aspects of the system through mathematical models or analytics that govern their behavior.

The parameters which control the vortex shedding on risers are the Reynolds number and the Strouhal number determined by the Eq. (1) and (2). The Reynolds number, *Re*, is the dimensionless ratio that relates the inertial forces to viscous forces, defined by the expression:

$$R_e = \frac{\rho D U}{\mu} \tag{1}$$

Where ρ is the density of the fluid. D is the diameter. U is the free stream velocity and μ is the dynamic viscosity.

The Strouhal number, *St*, is the dimensionless that relates the frequency of vortex shedding with the characteristic parameters of the flow and body defined by the Eq. (2) for the fixed rigid circular cylinder in steady current:

$$S_t = f\frac{D}{U} \tag{2}$$

Where f is the frequency of vortex shedding. D is the diameter of the cylindrical structure and U is the free stream velocity.

The lift and drag coefficients, C_L and C_D, which are obtained at each time step during the calculate of transient simulation are determined by Eq. (3) and (4). These results of the pressure and viscous components on the surface of the riser, where the $F_D$and F_L are the drag and lift forces, respectively.

$$C_D = \frac{F_D}{\left(\frac{1}{2}\rho D U^2\right)} \tag{3}$$

$$C_L = \frac{F_L}{\left(\frac{1}{2}\rho D U^2\right)} \tag{4}$$

The numerical scheme employed in this study is based on the finite volume method, using a solver pressure based with double precision. The solution of the flow field, coefficients and forces acting on the risers were obtained by solving the Reynolds equations or Reynolds-averaged Navier-Stokes equations (RANS) that include the equation of continuity and momentum, represented by the Eq. (5), (6) and (7) for the two-dimensional case.

$$\frac{\partial \bar{u}}{\partial x} + \frac{\partial \bar{v}}{\partial y} = 0 \tag{5}$$

$$\rho\left(\frac{\partial \bar{u}}{\partial t} + \bar{u}\frac{\partial \bar{u}}{\partial x} + \bar{v}\frac{\partial \bar{u}}{\partial y}\right) = -\frac{\partial \bar{p}}{\partial x} + \frac{\partial}{\partial x}\left[2\mu\frac{\partial \bar{u}}{\partial x} - \overline{\rho u'u'}\right] + \frac{\partial}{\partial y}\left[\mu\left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right) - \overline{\rho u'v'}\right] \tag{6}$$

$$\rho\left(\frac{\partial \bar{v}}{\partial t} + \bar{u}\frac{\partial \bar{v}}{\partial x} + \bar{v}\frac{\partial \bar{v}}{\partial y}\right) = -\frac{\partial \bar{p}}{\partial y} + \frac{\partial}{\partial x}\left[\mu\left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right) - \overline{\rho u'v'}\right] + \frac{\partial}{\partial y}\left[2\mu\frac{\partial v}{\partial y} - \overline{\rho v'v'}\right] \tag{7}$$

ISSN 2176-5480

The terms $\bar{u}$ and $\bar{v}$ are the mean velocity components and $-\overline{\rho u'v'}$, $-\overline{\rho u'u'}$ and $\rho v'v'$ are known as Reynolds stresses and for the solution of the closure problem of turbulence, the equations are modeled by Boussinesq approximation, an algebraic model for the definition of the effective viscosity of the turbulent flow as laminar action of viscosity (molecular) and turbulent viscosity. Equation (8) presents the Boussinesq approximation compactly and indexical.

$$-\overline{\rho u_i' u_j'} + \frac{2}{3}\rho k \delta_{i,j} = \mu_t \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \tag{8}$$

Where μ_t is the turbulent viscosity, k the turbulent kinetic energy and $\delta_{i,j}$ is the Kronecker delta (for $\delta_{ij} = 1$ and 0 if $i \neq j$).

The turbulence model chosen for the solution of the closure problem was the k-ω SST that employs two equations based on the concept of turbulent viscosity and the direct formulation of the transport equations of the Reynolds stresses. The model k-ω SST, shear – stress transport, was developed by Menter (ANSYS FLUENT 12.0 Documentation) to effectively blend the robust and accurate formulation of the k-ω model in the region close to the wall to and the independence of free stream for far field of the model k-ε.

This model was applied to this study to perform the numerical investigation because it presents precision and reliability for flows with adverse pressure gradient, a factor present in flow over ocean cylindrical structures with vortex shedding.

The transport equations for the turbulence model k-ω SST according to ANSYS FLUENT 12.0 User's Guide, are:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left(\Gamma_k \frac{\partial k}{\partial x_j} \right) + G_k - Y_k + S_k \tag{9}$$

$$\frac{\partial}{\partial t}(\rho \omega) + \frac{\partial}{\partial x_i}(\rho \omega u_i) = \frac{\partial}{\partial x_j}\left(\Gamma_\omega \frac{\partial \omega}{\partial x_j} \right) + G_\omega - Y_\omega + D_\omega + S_\omega \tag{10}$$

In these equations, $\tilde{G}_k$ is the generation of turbulent kinetic energy due to mean velocity gradient; G_ω represents the generation of ω; Γ_k and Γ_ω represent the effective diffusivity of k and ω, respectively, Y_k and Y_ω represent the dissipation of k and ω due to turbulence; D_ω represents the cross-diffusion term, and S_k and S_ω sources terms are defined by the user.

The realization of the numerical investigation with the turbulence model k- ω SST requires approximations to the initial parameters in order to establish the conditions for entry of velocity that the model needs, such as the turbulence intensity I, the length of turbulence ℓ, energy turbulent kinetic k and dissipation rate specific ω. However, according to Mahbubar, *et al.* (2007), the turbulence intensity may be defined according to Eq. (11):

$$I = 0.16 R_e^{-1/8} \tag{11}$$

The length of turbulence is then defined according to Eq. (12), where D is the diameter of the cylindrical structure under study.

$$\ell = 0.007D \tag{12}$$

The turbulent kinetic energy k is obtained by the ratio between the average velocity and the free stream turbulence intensity, according to Eq. (13).

$$k = \frac{3}{2}(IU)^2 \tag{13}$$

And, the specific dissipation rate ω is obtained according to the Eq. (14), giving an empirical value of $C\mu=0,09$.

$$\omega = C_\mu^{-1/4} \frac{\sqrt{k}}{\ell} \tag{14}$$

2.2 Selection of the diameter and environmental conditions

ISSN 2176-5480

The selection of the characteristic diameter risers and the range of current velocities was based on data presented by Le Cunff, et al. (2002) in his theoretical research, numerical and experimental that evaluates the vortex induced vibration with constant current due to the phenomenon of vortex shedding using approximations based on the modal response solution of the structural equations in time along with the models of fluid and resolutions the Navier-Stokes equations for a model of simple equations.

The diameters used in the construction of risers vary according to the length of these structures and the depth in which they operate. These diameters may also be set according to the type of riser in question. For this paper, the selection of the diameter is determined from the dimensions of drilling and catenary risers. The diameter adopted is 0.533 m.

The environmental characteristics which determine the type of flow are very important for evaluation of the numerical investigation. For the present work, the flow is treated as an incompressible fluid, turbulent and viscous subjected to the action of steady current. The effects of gravity, evaluation of free surface and waves were not considered. The temperature and density were admitted constant. The density of sea water is considered equal to 1025 kg/m³ and the dynamic viscosity of 0.001003 kg/ms.

The sea current velocity reaches the long ocean cylindrical structures at various angles of incidence and intensity. In this work, the determination of the range of velocities was attributed according to depth (Le Cunff, et al., 2002). In an attempt to cover a range of speeds according to depths which the drilling and catenary risers are employed, this study was considered the values of current velocities ranging from 0.1 to 0.5 m/s.

2.3 Computational domain

In this step the dimensions of the system for the allocation of calculations are delimited. The size of the physical domain was based on the work of Zhu, *et al*. (2010) in their simulation of vortex induced vibration on arrays of cylinders, where the numerical simulation was performed for the case of isolated cylinder and 3x3 arrangement with Re=1.3E+05. To create the domain, meshing and assignment of boundary conditions was used preprocessor GAMBIT 2.3.16.

The geometry of the physical domain corresponds to a rectangle with proportional dimensions to the characteristic diameter where will be applied the mesh and boundary conditions. The region near the boundary of the cylinder has a specific geometric decomposition for the better adaptation mesh. Figure 1 shows the computational domain for the calculation of the study of vortex shedding on risers.

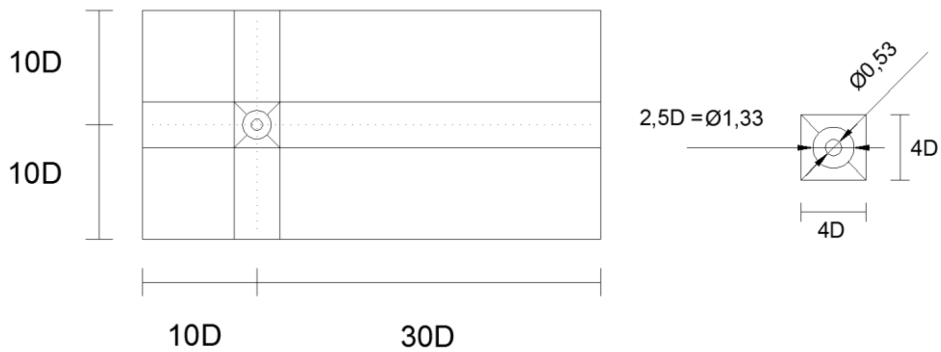

Figure 1. Computational domain for the numerical investigation

2.4 Mesh and mesh convergence study

The meshing was developed with refinements applied on the lines starting from the contour of the cylinder and progressively extending to the other adjacent faces. The distribution of the mesh on the lines is controlled by spacing parameters and gradation asymmetric accompanying the size of cells created with the refinement on the contour of the cylinder. The intention of applying refinements on the lines is to generate variations in the size of the cells and then control the increases of the size of the control volumes from specific regions, in this case, from the contour of the cylinder to capture the effects of the flow to high Reynolds numbers.

In the present work, seven numerical meshes were generated with variation of the number of nodes on the contour of the cylindrical structure. The meshes have only quadrilateral elements (QUAD) type mappable (MAP). A structured mesh where the grid is connected to a regular quadrilateral elements with the spacing of 0.1

The mesh convergence study was based on the values of the mean drag coefficient according to drag curve as a function of the Reynolds number given by Sumer and Fredsoe (2006). Table 1 presents the results obtained during the mesh convergence study for the current velocity of U=0.5 m/s.

ISSN 2176-5480

Table 1. Quantitative data of the mesh convergence study.

Mesh	Cells	Nodes	$\overline{C_D}$	$\overline{C_L}$
M1	860	928	0.77	0.03
M2	1072	1144	1.21	0.00
M3	1342	1428	1.00	0.17
M4	2442	2555	1.02	0.00
M5	3880	4020	1.01	0.00
M6	4557	4710	0.94	0.03
M7	14914	15188	0.45	0.00

The mesh selected for this study was M5. It was subsequently increased the number of nodes around the cylinder 50 to 56 for the better adaptation of the mesh in the area surrounding the cylinder. Figure 2 shows the curve of the mean drag coefficient with the result of the selected mesh for the study of vortex shedding and Fig. 3 shows the resulting mesh on the area near the cylinder.

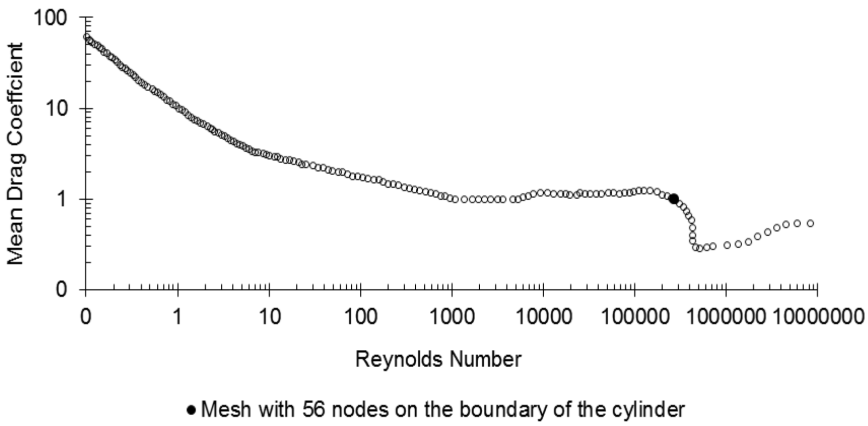

Figure 2. Resulting mean drag coefficient of the grid study convergence. Adapted from Sumer and Fredsoe (2006).

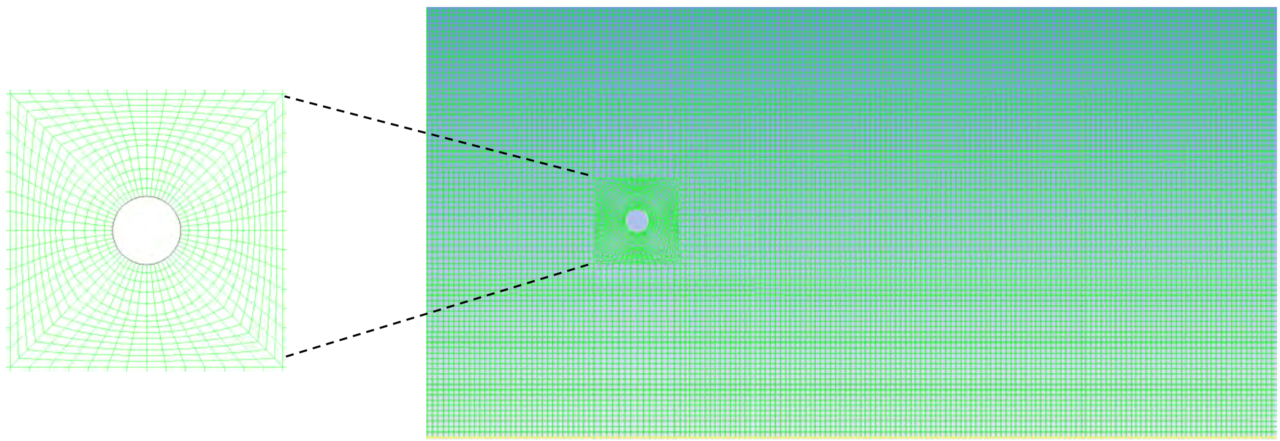

Figure 3. Numerical mesh applied to the cylinder

After the definition phase of the mesh through their numerical results, it was evaluated with respect to quality of discretization of the computational domain cells.

2.5 Mesh quality

The test of mesh quality was conducted by own preprocessor GAMBIT 2.3.16. and consists in checking the asymmetry of the mesh cells. The quality test was conducted on the equiangle skew (Q_{EAS}) or skewness that is the standardized measure of asymmetry of the cells, determined using Eq. (15).

$$Q_{EAS} = max\left\{\frac{\theta_{max}\text{-}\theta_{eq}}{180\text{-}\theta_{eq}}, \frac{\theta_{eq}\text{-}\theta_{min}}{\theta_{eq}}\right\} \tag{15}$$

When $Q_{EAS}=0$ means that the cells of the mesh do not have asymmetry and therefore are ideal cells. And $Q_{EAS}=1$ identifies the cells are completely distorted and may incur errors for the simulation. Figure 4 shows the quality of the mesh in the region of geometric decomposition according to the equiangle skew parameter for two-dimensional quadrilateral elements. Only this region was presented because it presents the most critical elements with asymmetry. The remaining cells of the computational domain feature great quality.

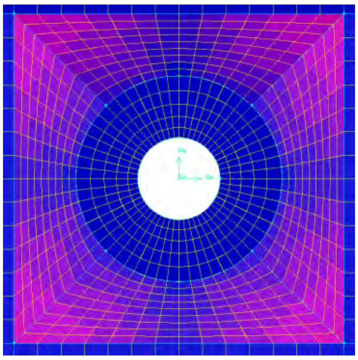

Figure 4. Mesh elements in accordance with the specified quality range

The highest values of Q_{EAS} for this region is 0.49 located at the vertices of the square surrounding the cylinder. The mesh used in this study provides excellent quality in the contour of the cylinder and regions outside the geometric decomposition. However, the mesh can be regarded as good quality to perform the numerical investigation. In order to check the sensitivity of the mesh for different flow conditions, simulations were performed for different Reynolds number (Re=2.72E +02, Re=2.72E +03 and Re=2.72E +04). The results for this variation are presented in the section of qualitative analysis.

2.6 Boundary conditions

The boundary conditions are additional restrictions required as components of mathematical modeling to assign the characteristic parameters for the flow and with them, to establish assumptions and constants that may facilitate the resolution of the problem. The boundary conditions used in this paper are presented in the Fig. 5 below:

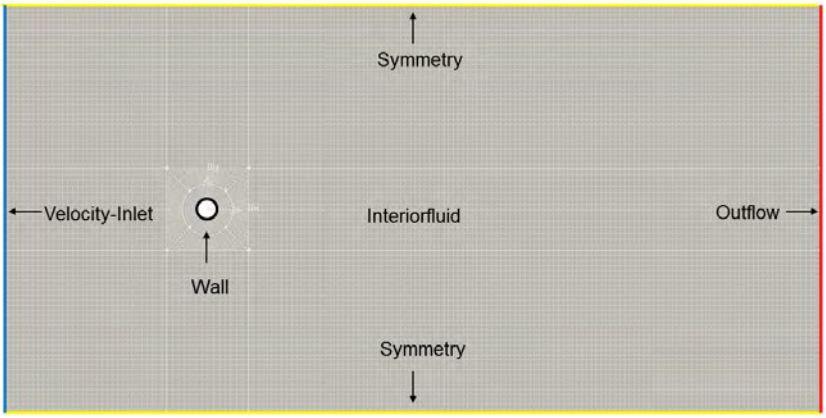

Figure 5. Boundary conditions on the computational domain

a) Velocity-inlet boundary was used for the range of current velocity between 0.1 and 0.5 m/s;
b) Outflow boundary was used for represent unknown variables in the exit;
c) Wall boundary was used in the riser contour with no slip condition;
d) Symmetry boundary was used on the upper and lower sides of computational zone parallel with the flow for indicate that in this case the shear stress is zero;
e) Fluid boundary represents all of cells in the computational zone where the calculations will performed.

ISSN 2176-5480

2.7 Simulation settings

For the solution of numerical investigation is important assignment of the parameters needed to solver. The model and controls of the solver used in this work were used to characterize the incompressible and unsteady flow. For unsteady simulation is necessary to determine the size of the time step calculation. In the case of vortex shedding on risers, the time step is based in the Strouhal number. For this work the theoretical Strouhal number for to establish the size of the time step is 0.2, constant value for the subcritical flow regime (Sumer and Fredsoe, 2006).

The numerical investigation was performed using an operational system 64 bits, processor Intel Core i5 M460 (2.53 GHz), 640 GB SATA, 6.0 GB RAM. Table 2 presents the settings of the numerical method applied in this study.

Table 2. Settings of numerical simulation on marine risers.

Simulation	2d-dp (two-dimensional with double-precision)
Fluid	Newtonian, Incompressible, Viscous, Turbulent
Density	ρ = 1025 kg/m³
Viscosity	μ = 0.001003 kg/m.s
Boundary Conditions	
Inlet	Velocity – Inlet
Outlet	Outflow
Cylinder	Wall
Upper	Symmetry
Under	Symmetry
Interiorfluid	Fluid
Solver Controls	
Solver	Segregated, Implicit, Unsteady
Unsteady Formulation	Second Order Implicit
Turbulence Model	$k - \omega$ SST
Pressure	Standard Discretization
Pressure-Velocity Coupling	PISO
Momentum	Segunda Ordem Upwind
Turbulent Kinetic Energy	Segunda Ordem Upwind
Specific Dissipation Rate	Segunda Ordem Upwind

3. RESULTS AND DISCUSSION

The operating conditions in which the risers are employed characterize flow regimes with high Reynolds numbers. However, the simulations were conducted for a range of operating current velocities, wherein the value of the current velocity U is between 0.1 to 0.5 m/s. The results of the vortex shedding on marine risers are composed for velocity and static pressure fields, hydrodynamics forces and coefficients and frequency of vortex shedding.

The evaluation of the coefficients of drag and lift through the time history obtained during the unsteady simulation for the range of current velocities exhibit oscillatory characteristics very similar. Therefore, this section presents the quantitative data corresponding to the range of current velocity, but only for the higher speed are presented graphically all the expected results.

3.1 Vortex shedding

The velocity and static pressure fields are presented as a function of the temporal variation calculation to allow visualization of the vortex shedding. Thus, were determined three steps of calculation and they were captured images and values for the variables in question.

Figure 6 below shows the contour of velocity magnitude and static pressure for U=0.5 m/s.

The results for the vortex shedding show through fields of velocity and static pressure of the flow patterns on cylinders in various regions, such as the stagnation region, favorable pressure gradient, the maximum speed on the top of the cylinder, adverse pressure gradient, separation point of flow, vortex shedding and wake. The region downstream of the cylinder introduces instability because it has low speed and low pressure.

Note that the pressure field is asymmetric and periodic, it varies with the process of vortex shedding for each instant of calculation considered.

The periodic behavior of the wake of vortices will reflect the oscillatory behavior of the drag and lift coefficients.

ISSN 2176-5480

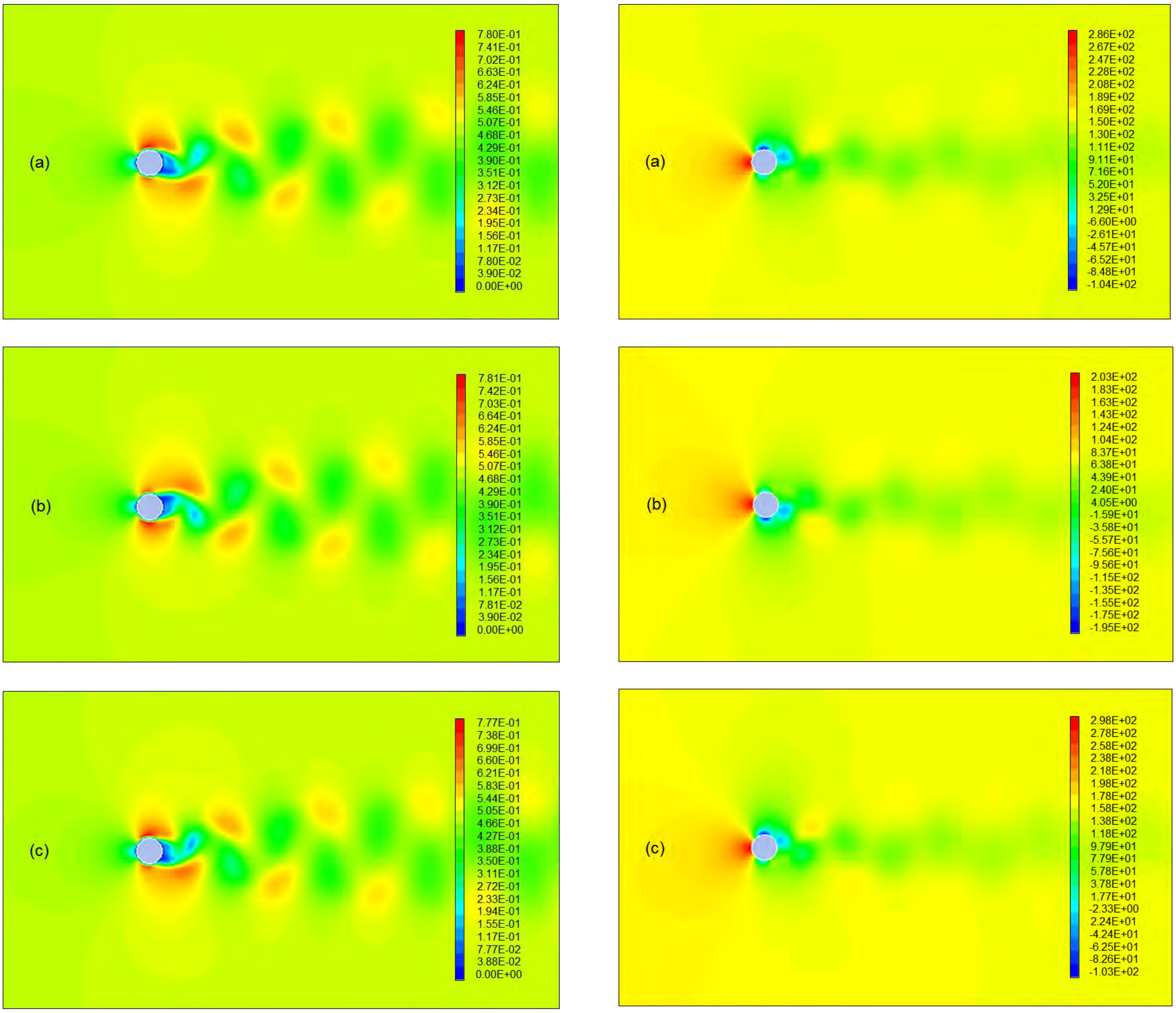

Figure 6. Contours of velocity magnitude (m/s) and static pressure (Pa). (a) Time=6.3900E+02. (b) Time=6.4113E+02. (c) Time=6.4326E+02

3.2 Hydrodynamics forces and coefficients

The forces and lift and drag coefficients are derived from the sum of the viscosity and pressure components acting on the cylinder surface. The unsteady simulation allowed the collection of time history of the coefficients that represent the behavior and magnitude with the progress of vortex shedding in accordance with the Reynolds number.

From the time history results were obtained for the drag and lift coefficients presented in Tab. 3 for the speed range. The drag and lift coefficients are presented in terms of mean values ($\overline{C_D}$); Statistical values r.m.s ($C_D{}'$ and $C_L{}'$); and maximum values ($C_{D_{MAX}}$ and $C_{L_{MAx}}$) representing the maximum amplitude of oscillation. The mean lift coefficient $\overline{C_L}$ was not submitted because it is equal to zero, in other words, the lift coefficient oscillate around a zero value.

Table 3. Drag and lift forces for the current velocity variation.

U [m/s]	Re	$\overline{C_D}$	$C_D{}'$	$C_L{}'$	$C_{D_{MAX}}$	$C_{L_{MAX}}$
0.1	5.45E+04	0.990	0.991	0.490	1.020	0.693
0.2	1.09E+05	0.987	0.987	0.493	1.017	0.697
0.3	1.63E+05	0.986	0.986	0.495	1.015	0.699
0.4	2.18E+05	0.985	0.985	0.496	1.014	0.701
0.5	2.72E+05	0.984	0.984	0.497	1.013	0.702

ISSN 2176-5480

The forces corresponding to the coefficients described are shown in Tab. 4.

The importance of knowledge of the forces acting on the cylinder surface during the process of vortex shedding is to establish the dynamic forces that oceanic cylindrical structures are subjected to determination of the design parameters. The forces varies exponentially with the velocity of flow, therefore, the higher stream velocity, U=0.5 m/s, corresponds to the highest levels of drag and lift forces.

Table 4. Drag and lift forces for the current velocity variation.

U m/s	Re	$\overline{F_D}$ [N]	F_D' [N]	F_L' [N]	$F_{D_{MAX}}$ [N]	$F_{L_{MAX}}$ [N]
0.1	5.45E+04	2.71	2.71	1.34	2.79	1.89
0.2	1.09E+05	10.79	10.79	5.38	11.11	7.62
0.3	1.63E+05	24.23	24.24	12.16	24.96	17.19
0.4	2.18E+05	43.03	43.04	21.67	44.32	30.64
0.5	2.72E+05	67.20	67.21	33.94	69.21	47.95

The dimensionless frequency of vortex shedding may be determined by time history of the lift coefficient using the algorithm of fast Fourier transform, FFT. The Strouhal numbers have values close to differences in decimal order and frequencies accompany proportionality with velocity *U*. The Strouhal number and vortex shedding frequencies for the range of current velocities are listed in Tab. 5.

Table 5. Strouhal number for the range of current velocity.

U m/s	Re	*f*	St
0.1	5.45E+04	0.04231	0.22552
0.2	1.09E+05	0.08462	0.22552
0.3	1.63E+05	0.12706	0.22574
0.4	2.18E+05	0.17024	0.22685
0.5	2.72E+05	0.21176	0.22574

3.3 Qualitative analysis

The study of the movement of fluids through of numerical simulation requires modeling tools and experimentation for the comprehension and validation of its causes and consequences. To ensure the reliability of the results. considering the modeling for the isolated cylinder. they were confronted with the literature. Regarding to mean drag coefficient. the results for the variation of Reynolds numbers and velocities were compared to Sumer and Fredsoe (2006) in Fig. 7. where are presented results from numerical and experimental solutions of the two-dimensional Navier-Stokes equations.

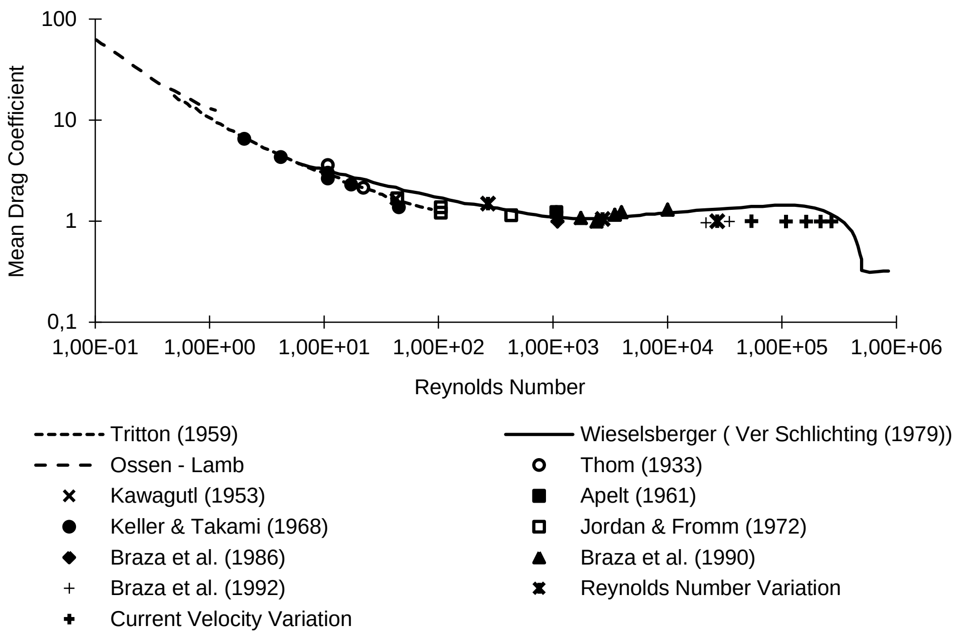

Figure 7. Mean drag coefficient for circular cylinder in steady current. Adapted from Sumer and Fredsoe (2006).

ISSN 2176-5480

In Fig. 7 the experimental results correspond to the data of Tritton (1959) and Wieselsberger (1979). The other results are numerical. The results of the r.m.s lift coefficients were compared with data of Norberg (2003) in Fig. 8 which is composed of experimental and numerical results.

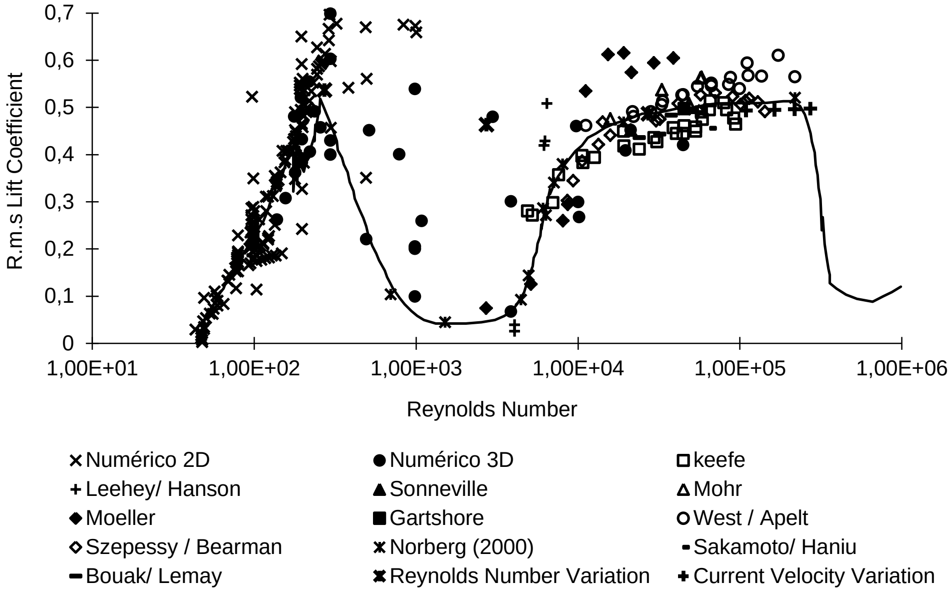

Figure 8. R.m.s lift coefficient versus Reynolds number. Adapted from Norberg (2003).

In his comprehensive study of hydrodynamics around cylindrical structures. Sumer and Fredsoe (2006) highlight comparative results for St from 2D solutions of the Navier-Stokes equations. However. these results vary for Re>1.0E+05. Therefore. the comparison of the results with respect *St* is only experimental results from Fig. 9.

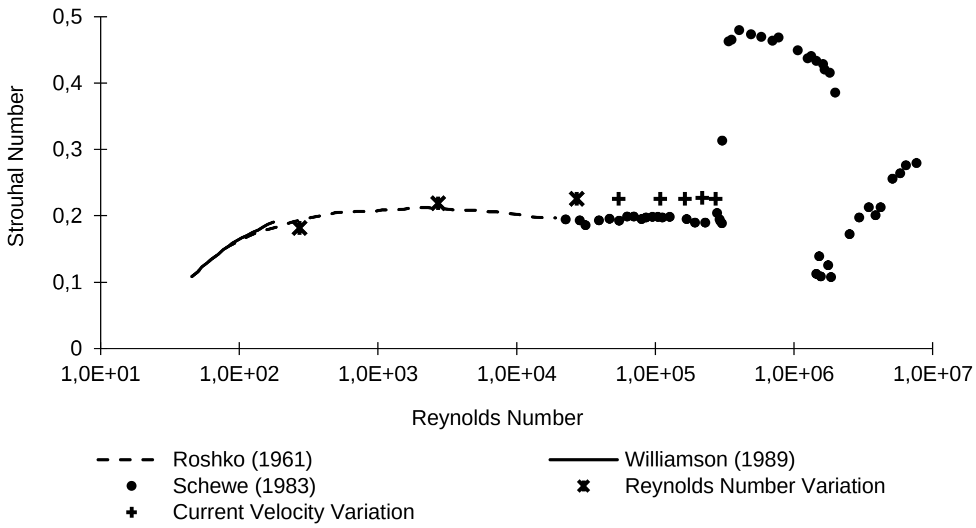

Figure 9. Strouhal number for smooth circular cylinder. Adapted from Sumer and Fredsoe (2006).

4. CONCLUSION

The main consequence of the ocean currents on marine risers is the vortex shedding process. The numerical investigation using the technique of computational fluid dynamics used the physical modeling of two-dimensional vortex shedding considering the flow over a rigid cylinder and fixed and constant current study was conducted to high Reynolds numbers in the subcritical regime. This approach represents the phenomenon under study. as it allows the analysis of the problem under realistic conditions and at low cost.

The results obtained in this study were able to reproduce the characteristics of vortex shedding as the oscillatory behavior and intensity of the drag and lift coefficients. the frequency of vortex shedding. the total forces and turbulent

ISSN 2176-5480

wake. Through the time history of the coefficients C_D and C_L generated during the calculation time of each transient simulation. it was possible to determine the oscillatory behavior of characteristic variables. such as the average. r.m.s. and maximum values and the frequency of vortex shedding.

The Reynolds number variation showed that for the range of Reynolds between Re=2.72E+02 and Re=2.72E+04 are in agreement with the experimental curve for the mean drag coefficient and with good representation of the Strouhal number. This is a good performance of the numerical mesh for different flow conditions.

In the current velocity variation between 0.1 – 0.5 m/s the oscillatory behavior of the coefficients proved very close with very similar results. With the analysis of contours for maximum velocity of flow was observed that the effects of free stream may be considered as inviscid flow and that they print the characteristics of the flow regime of the boundary layer. It was also noted the formation of vortices and non-uniform periodic wake.

At the end of this study. it is concluded that understanding the process of numerical simulation and solution technique of computational fluid dynamics enabled learning about physics and mathematics involved in the process of vortex shedding on ocean cylindrical structures. It can be argued then that the turbulence model k-ω SST used by the commercial code FLUENT 14.0 proven to be a tool capable of providing data of hydrodynamics forces and coefficients that can be employed in the development of projects. as it showed in good agreement with the literature and proved very efficient for the solution of the problem under study with a lightweight mesh refinements in the region close to the cylinder.

5. REFERENCES

ANSYS FLUENT, 2009. “FLUENT 12.0/12.1 Documentation”. 26 May 2013 <http://www.sharcnet.ca/Software/Fluent12/index.htm>

ANSYS FLUENT. “GAMBIT 2.4 Documentation”. 26 May 2013. <http://aerojet.engr.ucdavis.edu/gambithelp/>

Le Cunff, C., Biolley, F., Fontaine, E., Étienne, S., Facchinetti, M.L., 2002. “Vortex – Induced Vibrations of Risers: Theoretical, Numerical and Experimental Investigation”. Oil & Gas Science and Technology, No. 1, Vol.57, p. 59-69, Rev. IFP.

Mahbubar, R., Mashud, K., Abdul, A., 2007. “Numerical Investigation of Unsteady Flow past a Circular Cylinder Using 2-D Finite Volume Method”. Journal of Naval Architecture and Marine Engineering 4, p. 27-42.

Norberg, C., 2003. “Fluctuating Lift on a Circular Cylinder: Review and New Measurements”. Journal of Fluids and Structures 17, p. 57-96.

Sumer, B.M., Fredsoe, J., 2006. “Hydrodynamics around Cylindrical Structures”. World Scientific Publishing Co. Pte. Ltd, Advanced Series on Ocean Engineering, Vol. 26, Singapore, Rev ed.

Zhu, H., Lin, Y., Jia, Q., Yang, X., 2010. “Simulations of VIV on Arrayed Marine Risers”. In International Conference on Computer Design and Applications, ICCDA2010, Qinhuangdao, Hebei, China.

6. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.