CFD ANALYSIS FOR OFFSHORE SYSTEMS: VALIDATION AND APPLICATIONS

Daniel F. C. e Silva*, Paulo R. Pagot†

PETROBRAS
Av. Horácio Macedo 950, Prédio 32
Cidade Universitária – Ilha do Fundão
Rio de Janeiro – RJ
CEP: 21941-915
* e-mail: danielfc@petrobras.com.br
† e-mail: pagot@petrobras.com.br

ABSTRACT

The Scientific Methods group in Petrobras Research Center works on development and application of multidisciplinary simulation tools for several engineering problems related mainly to offshore systems. Recently, there were many different cases where Computational Fluid Dynamics was successfully employed. This work presents a collection of cases where CFD simulations were validated against experimental data and directly used to guide solutions for practical problems. The case #01 consists on calculating the current loads on a FPSO and investigating the influence of appendices such as bilge keels and rudders on the near flow field. In a very similar way, case #02 extends this procedure to wind loads determination. Case #03 presents the calculation of hydrodynamic forces on a torpedo anchor during its installation. The simulation results coupled with a simplified dynamic model allows evaluating the directional stability to different torpedo models. A whole FPSO topside geometry is modeled on case #04, which investigates the flow pattern near the FPSO Helideck. The simulation velocity and turbulence profiles were compared to wind tunnel measurements. These cases, which will be briefly presented, show how CFD tools can be suitable for many practical problems solutions. All simulations here presented were performed using ANSYS CFX®.

Case #01 – Determination of current loads on offshore units

In order to predict the dynamic behavior of floating units as well as calculate the mooring lines loads, it is necessary to determine the loads promoted by the sea water flow around the unit hull. Since a low Froude number regime can be considered, only the vessel underwater portion need to be modeled. All current generated forces can be evaluated using a steady-state simulation for each heading angle[1]. Several headings were automatically evaluated using ModeFrontier®. Figure 1 presents the transversal force coefficients, compared to towing tank experiments[2]. This validation case encourages the use of this procedure for other vessels.

Figure 1 – Current transversal force coefficients compared to IPT (Instituto de Pesquisas tecnológicas) towing tank experiments. For the 90 and 270 headings, the streamlines in a transversal plane are presented.
Case #02 – Determination of wind loads on offshore units

The wind loads also need to be evaluated for offshore units for the same reasons described on the previous case. Therefore, a very similar procedure was developed to calculate the forces and moments induced by the airflow around offshore units\(^1\). Using a domain starting from the mean surface level (equivalent to the wind tunnel floor) and considering only the platform geometry above water, again a steady state simulation was conducted for each individual heading angle condition. Figure 2 shows a comparison between the simulation and the experiments from wind tunnel measurements\(^3\).

Case #03 – Torpedo anchor directional stability during the installation process

The geometry of offshore anchors is usually determined from a geotechnical perspective. This case presents the proposal of simple modifications on a torpedo anchor to improve its hydrodynamic performance concerning its directional stability. Additionally, the study of a real case torpedo deformation is presented. Although the directional stability investigation requires a dynamic analysis, the CFD simulations were performed on a steady state regime to show for which inclinations the anchor tends to recover the vertical position. The trajectory was investigated through a simplified dynamic model and compared with field measurements\(^4, 5\). On the Figure 3, pressure contours and flow streamlines on the region near the torpedo deformed fin can be seen.

Case #04 – Investigation of the airflow around an offshore helideck

The offshore helideck wind flow is usually subject to the platform interference. The helideck airspace velocity and turbulence fields are important issues to promote safe helicopter take-off and landing operations. This case brings some CFD results of a helideck wind flow 3D-field defined by the local conditions and constrained by the FPSO structure. A discussion about the chosen CFD boundary conditions is also presented. These CFD results are compared with the wind tunnel model-scale velocity and turbulence measurements\(^6, 7\). The atmospheric boundary layer wind tunnel measurements were performed with use of two different techniques: Particle Image Velocimetry (PIV) and Constant Temperature Anemometry (CTA). The British standard CAP437\(^8\) is examined and suggestions are made herein. Figure 4 shows a contour of vertical velocity near the Helideck airspace.
References


