

WIND CHARACTERIZATION AROUND OFFSHORE PLATFORM FOR REAL-TIME HELICOPTER SIMULATOR

Filippo Colognato – Customer Training, Leonardo Helicopters
Stefano Scala – Software Engineer, Txt Next

Abstract

In the last years, flight safety has constantly improved. But accidents still occur, and ground proximity operations (e.g. take-off and landing, search and rescue, *et al.*) are among the most dangerous flight phases. With the development of offshore deep water fields, helicopter transportation and consequently helicopter deck becomes very important. To ensure helicopter safe operations, aerodynamic hazards around an offshore platform need to be identified and simulated during flight training simulations.

OpenFoam CFD features provides the full capability to evaluate air-to-ground interactions for the different boundary conditions (wind speed, wind direction), but also include onerous computational time not compatible with real-time simulation and requires handling with a huge quantity of data. To overcome the long-time calculation time, it has been decided to run the incompressible static RANS computation stand-alone and to create a database to store the wind velocities in grid points. To overcome the bulk of data to handle it has been decided to select a restricted area around the offshore platform of about 1km with a mesh refinement of about 5m and another one of about 150m with a mesh refinement of 1m, where velocity gradients are the highest. During real-time-simulation, it is extracted from the database only a set of CFD data using a query based on oil rig shape, wind direction, wind speed and relative helicopter position. To further reduce the computational cost only CFD results around helicopter are extracted and a new query will be performed every time that helicopter moves away of a predefined distance from current position. Wind velocities extracted have been evaluated in about 100 control points on helicopter, placed on the main rotor, tail rotor, fin, tabs and fuselage using a linear interpolation weighted on the distance of the 8 grid points nearest to each one. Unsteady effects are reproduced by passing band-limited white noise through appropriate forming filters, based on statistical indicators obtained by CFD computations. More than 1000 CFD computation on different oil rigs and ships were performed covering different sets of boundary conditions. A successfully integration within Full Flight Simulator real-time software was confirmed by very positive pilots' evaluation during all ground proximity maneuvers.

1. INTRODUCTION

Modern pilot training programs more and more rely on Flight Simulation Training Devices. The growing importance of similar devices can be explained with their cost effectiveness (if compared to real aircraft cost per flying hour) and with the intrinsic level of safety they guarantee. In fact, FSTDs allow training pilots for dangerous, safety-critical or mission-critical situations, like serious malfunctions (e.g. engine failure), emergency procedures (e.g. emergency landing, helicopter autorotation) or bad atmospheric conditions without any real hazards for the people involved.

In fact, critical meteorological conditions can threaten the flight safety and compromise the mission effectiveness, especially during complex procedures like ground-proximity or low-altitude operations. These are usual situations for helicopters, like in *Search and Rescue* (SAR) or Emergency Medical Services (EMS) missions.

Therefore, the correct modeling of atmospheric conditions in a Flight Simulator allows the trainee pilot to exercise for critical and complex maneuvers in a completely safe way.

A potential mission scenario, is represented by ground proximity operations near offshore platform or ships, where wind, gusts and other turbulent phenomena strictly influence complex maneuvers of the pilot.

2. WIND MODELING NEAR OFFSHORE PLATFORM

Wind, like every flow, is affected by the obstacles it finds along its course. Oil rig platform and ships can divert its path, and change the wind value and direction (i.e. *wind local vector*).

When a huge air mass encounters an offshore platform, to avoid the obstacle, it is forced to go around or pass above, thus generating ascending

and descending movements, often followed by turbulence.

To face the problem, an analysis of the phenomenon was performed, using numerical methods to simulate the flow behavior; the so called Computational Fluid Dynamics, based on the solution of Navier-Stokes equations, which describe the motion of fluids.

3. COMPUTATIONAL FLUID DYNAMICS APPROACH

Computational Fluid Dynamics methods have clearly demonstrated in several research papers (e.g. [1] or [2]) their capability to analyze the wind flow over terrain, solving the atmospheric boundary layer. But this capability comes with a price.

In fact, CFD numerical simulations usually require long computational time, not suitable for real-time simulation, like it is in a Flight Simulator. Moreover, the wider the domain of the problem, the more burdensome the calculation time and data quantity are, under the same spatial resolution.

Furthermore, such numerical analyses are function of the selected boundary conditions, and its significance is restricted to those conditions.

The approach to the problem has consequently reflected the limitations of computational fluid dynamics.

First, it has been defined the type of obstacle where the wind flow over terrain would be modeled. It is necessary to perform analysis of Oil rig platform and Landable Ships.



Figure 1 Oil Rig 1 Model

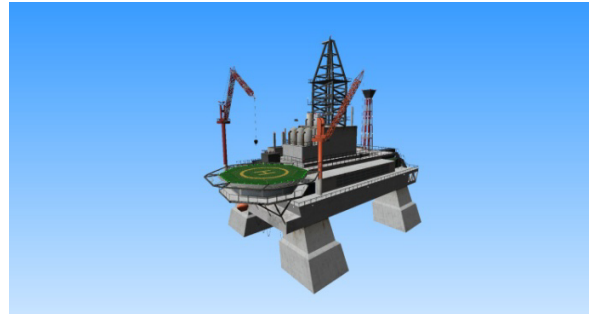


Figure 2 Oil Rig 2 Model

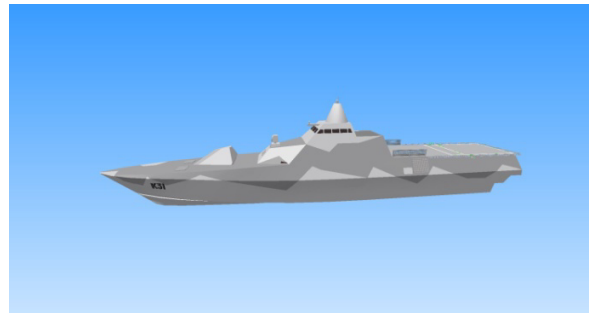


Figure 3 Landable Ship Model

Then, a set of boundary conditions was chosen, representative of all the possible cases. The boundary conditions are the values the numerical simulations have on the borders of the volume under analysis, and should be imposed to solve the numerical problem. These values correspond to the volume ingoing and outgoing wind; the ingoing wind represents the undisturbed flow, unperturbed by the ground orography. Therefore, the boundary conditions denote the direction and the velocity of the wind affecting the selected region.

Then, to determine a finite number of numerical analyses to be evaluated, the entire wind rose was represented with 36 directions (every 10 deg).

Accordingly, 5 intensities have been defined to summarize all the probable conditions:

- 5 – 15 – 25 – 35 – 45 kts

In conclusion, a series of at least 180 numerical simulations would be computed for each 3D model, which would be solved stand-alone. The results were then saved and used by the simulator flight model during run-time.

In this way, the flight model would be perturbed by the local wind velocity, and the trainee would

experience a difference in helicopter behavior, dynamics and handling.

3.1. Numerical Analysis with OpenFOAM

The numerical analysis and computation of the mountain local wind vector in the defined volume has been performed with OpenFOAM, an open source CFD software package, for Navier-Stokes equations solution. Several are the reasons for its choice. OpenFOAM has a large user base across most areas of engineering and science, from both commercial and academic organizations, and it has tools for meshing complex CAD geometries, and for pre- and post-processing. Moreover, its solver capabilities include, among the others, compressible and incompressible fluids, multiphase flows and buoyancy-driven flows, and its libraries allow to model a wide range of phenomena, from turbulence to chemistry [3].

Additionally, in scientific literature there are studies which explore the use of OpenFOAM to simulate wind stream above mountain terrain simulation (e.g. [4],[5] and [6]).

For the analysis of the specific case, the OpenFOAM simpleFoam solver was used, a steady-state solver for incompressible and turbulent flow. This solver already includes the equations for atmospheric boundary layer solution; an OpenFOAM tutorial illustrates its application to evaluate wind flow in areas interested by wind turbine installation [7].

In particular, steady-state Reynolds Averaged Navier-Stokes (RANS) equations [2], for *Newtonian* fluids, with SST $k-\epsilon$ turbulence model [8] was chosen.

Using OpenFOAM meshing tool *snappyHexMesh*, an unstructured mesh of approximately 1.3 million cells was created. This mesh was composed of two different domains: an external domain with a lower resolution of $200 \times 1150 \times 150$ meters and a refinement domain of $100 \times 150 \times 70$ meters, where the asymptotic wind was mainly modified by the presence of the obstacle.

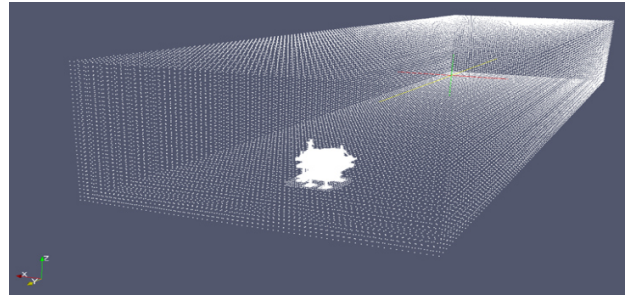


Figure 4 External domain $200 \times 1150 \times 150$ meters

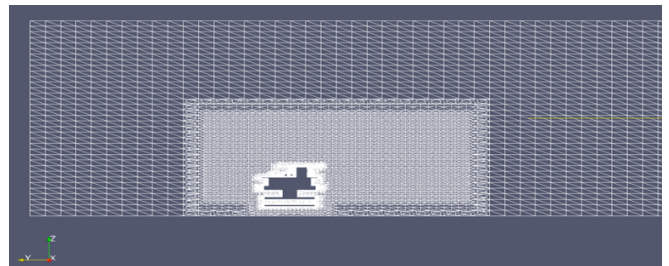


Figure 5 Refinement domain $100 \times 150 \times 70$ meters

3.2. Results

After about 360 hours of computer calculation 6 Gb of data were collected (2 hours and 35mb for each boundary condition).

Those data are composed by a couple of triplets, combining the three space coordinates with the 3-D vector of the local wind in that precise volume point.

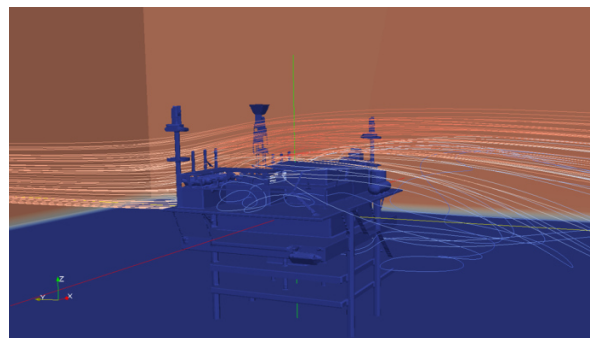


Figure 6 Stream tracer of the oil rig with direction 0 deg and 45 knots

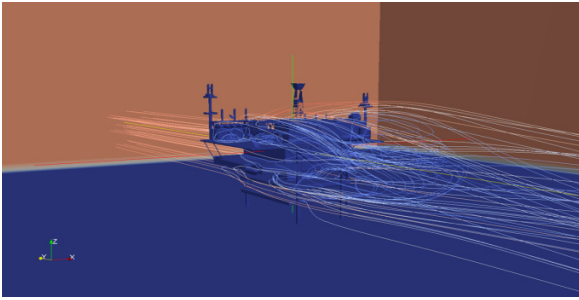


Figure 7 Stream tracer of the oil rig with direction 0 deg and 45 knots

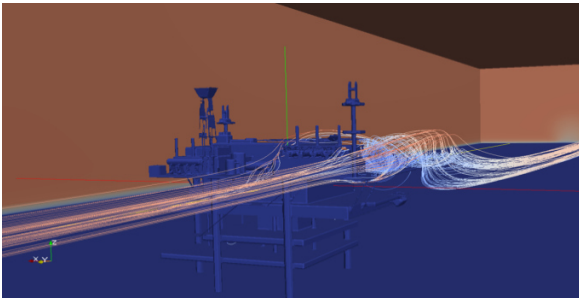


Figure 8 Stream tracer of a vortex generated on the oil rig with 0 deg and 45 knots



Figure 9 Stream tracer on the real oil rig platform

4. IMPLEMENTATION

The most important requirement during model implementation regards the capability to provide run-time the local wind vector to the flight model, with a frequency of 120 Hz (time-frame of 8.333 ms). The word local means “in 95 helicopter control points over aerodynamic surfaces”.

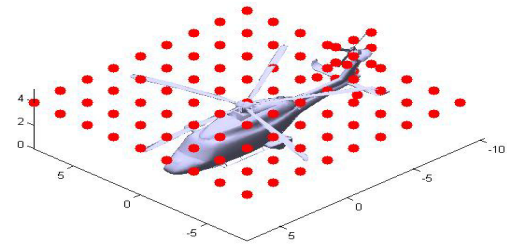


Figure 10 Helicopter control points - view 1

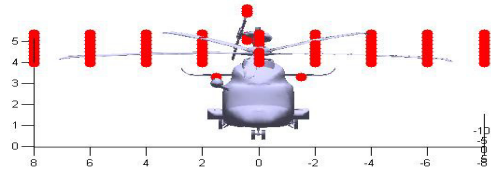


Figure 11 Helicopter control points - view 2

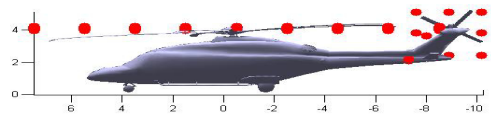


Figure 12 Helicopter control points - view 3

Knowing the helicopter relative position from the center of the oil rig and the entire CFD solution, for the selected wind speed/direction, run-time use of CFD results implies the extraction of a subsystem of points of the mesh, and the respective wind values from a database stored in the hard disk.

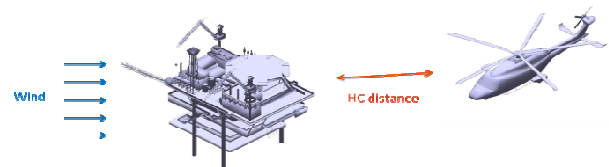


Figure 13 Oil rig and helicopter stl models

To store all the bulk of data produced by CFD analyses and to perform efficient query within, it was decided to extract the nearest points to the helicopter.

In order to satisfy the request of 8.333 ms time-frame it was developed an algorithm that optimize the time required for the query based on airspeed and attitude of the helicopter.

In fact, it is possible to extract from DB a wider portion of points (and respective wind data) in front of the helicopter and also depending on the airspeed (more speed means more points), to calculate the local wind in hc control points. In this way, it is no longer required to execute a query from DB every time-frame, but only when the helicopter is about to leave the portion obtained with the previous query.

A new query to the DB is executed if the helicopter is about to move away from the portion of points previously extracted.

Investigating the quality of the interpolation process that produces the local wind vector, it has been observed that the 8 points of the mesh closer to each helicopter control point are adequate to obtain a faithful representation of wind behavior.

During every integration step of real-time simulation, the main algorithm performs the following operations:

- it evaluates the actual distance between each helicopter control points and the points stored in memory (obtained by the previous query);
- it identifies 8 points closer to each control point, and it extracts the respective wind components;
- it calculates the local wind vector affecting the helicopter control point, and send it to the flight model; the local wind vector is calculated via interpolation among the wind vectors of the point selected and each point value is weighted according to its distance from helicopter control point.
- it verifies if the helicopter position is still inside the portion of points extracted.

The code has been completely developed in C++.

5. TURBULENCE

Another problem of this kind of approach is the necessity of use a wide discretization to lower the time requested by the standalone computation.

With such a wide discretization, it is difficult for CFD computation to reproduce wind micro-instabilities typical of flight in these conditions where there is strong gradient, where wind intensity and direction changes in a space smaller than the smallest space of the grid used for computation.

So, a wind component generated by a *Dryden wind turbulence model* was added, linking increasing turbulence intensities to statistical indicators of velocity components in all helicopter control points.

This turbulence model uses the Dryden spectral representation to add turbulence to the aerospace model by passing band-limited white noise through appropriate forming filters, according to the military reference MIL-F-8785C [9].

6. CONCLUSIONS

CFD methods have proven their validity even in such a wide and complex domain, and OpenFOAM has been a quite effortless (almost *Commercial off-the-shelf*) CFD tool.

OpenFOAM range of features provide us all the capabilities to obtain the best compromise between fidelity level and simulation real-time requirements. The results of CFD simulation appear consistent with Fluid Dynamics theory. Moreover, the pilots' first evaluation has been positive

Future developments may be focused on create mesh models and perform CFD computations of urban environments. The computational effort required will be higher and it will be necessary to find another compromise to reproduce flying in these areas.

7. REFERENCES

- [1] Leroy, Jerome. *Wind field simulations at Askervein hill*. VECTOR AS. 1999. Technical Report. Norway.
- [2] Stangroom, Paul. *CFD Modelling of Wind Flow Over Terrain*. s.l. : University of Nottingham, 2004.
- [3] [Online] <http://openfoam.com/features/>.
- [4] OpenFOAM Workshop Training Session, Wind Energy / Atmospheric Boundary Layer Tools and Tutorials. Churchfield, Matthew J. State College, PA, USA : The Pennsylvania State University, 2011. 6th OpenFOAM Workshop.
- [5] OpenFOAM simulations of atmospheric flow over complex terrain. J. Sumner, C. Masson, Y. Odemarky and M. Cehlinz. Goteborg : Chalmers University of Technology, 2010. 5th OpenFOAM Workshop.
- [6] Comparison of OpenFoam and EllipSys for modelling the wind ressources in complex terrain. P.E. Réthoré, B. Martinez, D. Cavar, F. Zahle, J. Berg, N.N. Sørensen, M. Kelly and A. Bechmann. Copenhagen : s.n., 2012. EWEA 2012, Europe's Premier Wind Energy Event.
- [7] Turbine siting example. [Online] http://www.openfoam.org/version2.1.0/numeric_s.php.
- [8] [Online] http://www.cfd-online.com/Wiki/SST_k-omega_model.
- [9] Defense, Department of. MilitarySpecification, Flying Qualities of Piloted Airplanes. 1980. MIL-F-8785C.