A STUDY OF THE AIR FLOW OVER A SHIP FLIGHT DECK

Mauricio Guedes, mauricio532@hotmail.com, Brazilian Navy-CASNAV, Brazil
Carlos Breviglieri, carbrevi@gmail.com, Brazilian Air Force-ITA, Brazil
João Luiz Azevedo, joaoluiz.azevedo@gmail.com, Brazilian Air Force-IAE, Brazil
Edson Basso, edsonbss@gmail.com, Brazilian Air Force-IAE, Brazil
Luís Moraes, luis.fmoraes@hotmail.com, Brazilian Air Force-IAE, Brazil
Matsuo Chisaki, matsuomc@iae.cta.br, Brazilian Air Force - IAE, Brazil

Abstract

The present work presents a study of the air flow over a Brazilian Navy Inhaúma-class corvette flight deck, using Computational Fluid Dynamics (CFD) techniques, wind tunnel tests and *in situ* measurements. This study is part of a test campaign to determine ship-helicopter operational limits (SHOL) for this class of ships. The flight deck of the Inhaúma-class corvettes is one of the smallest decks in the world for flight operations with medium helicopters, such as Lynx. As launch and recovering of helicopters from ships are operations that can be hazardous, even for larger flight decks, tests are required in order to fully characterize the ship air wake over such decks. In the present work, both CFD calculations and wind tunnel tests are considered in order to minimize the risks of such operations.

1. INTRODUCTION

Helicopter operations are restricted by limitations established in the flight operation manuals of each helicopter. Typically, the operating limits involve aspects such as wind, altitude, or landing base slope, among others. However, basic helicopter flight limitations are usually determined in a land-based environment by the aircraft manufacturer and/or by the procuring activity. The more hazardous-prone ship operations require special procedures, which impose restrictions beyond those specified in flight manuals. These limitations are not supplied by the helicopter manufacturer, since they depend, to a large extent, on the specific class of ship involved and its environment. The test campaign for the definition of operational limits, for launch and recovery of helicopters on board of ships or off-shore oil platforms, requires a step-by-step approach in order to guarantee the helicopter safe operation. The final result of such test campaign is the launch/recover envelope of that helicopter in the particular ship/platform considered. The typical steps in this campaign include:

1. Study of the flow over the ship, and in particular over the ship flight deck, using CFD simulation and wind tunnel testing;
2. *In situ* measurements on the ship at sea in order to validate the simulations;
3. Measurements for the helicopter over land in order to determine operational limits for its ship-borne use;
4. Construction of a candidate flight envelope;
5. Complete trials at sea in order to validate the candidate flight envelope.

Recent progress in CFD provides the opportunity to compute the viscous, turbulent airflow over a surface combat ship and along the standard helicopter approach path in a time-accurate manner. Nevertheless, despite the improvement in CFD techniques, industry still does not accept that the first item of the test campaign, enumerated above, could be performed solely using CFD simulations. Therefore, wind tunnel tests are still required in order to fully close the initial simulation phase of the test campaign. Furthermore, a gradual substitution of wind tunnel tests by computational simulations also requires careful calibration and validation of the computational models. Therefore, in the present study, the computational results are compared to wind tunnel test data obtained at IAE/ALA TA-2 wind tunnel, located at Departamento de Ciência e Tecnologia Aeroespacial in São José dos Campos, Brazil. Beyond the obvious validation of the present simulations, it is also hoped that such comparisons will increase the confidence in the numerical model developed and, in the future, allow a decrease in the number of expensive wind tunnel test hours by replacing them with CFD simulations. The present study has used the CFD++ commercial CFD code (Metacomp Technologies, 2013) and the forthcoming figures present some representative results of the test campaign at the present time.

2. SIMULATIONS AND TESTS

For reference, the Inhaúma corvette can be seen in figure 1. She is 95.8 m long ship, it has a crew of 145 men and its maximum speed is 29 kts.

The CFD simulations and the wind tunnel tests were performed considering the position of the anemometers in the *in situ* tests. The figures 2 and 3 show the positions of the anemometers in the *in situ* tests and the planes of the laser sheets used in the wind tunnel tests.

3. NUMERICAL SIMULATION

The CFD simulation performed in this work uses the Reynolds Averaged Navier-Stokes equations (RANS) in a three dimensional unstructured framework. The finite volume method is used for discretization coupled with an implicit algorithm for time stepping. The realizable $k - \varrho$ model (Metacomp Technologies, 2013) is used for turbulence modeling. A preconditioned compressible formulation is adopted from the available commercial solver options. The initial condition sets a uniform flow at 10 m/s aligned with the ship. The ship surface boundary condition is prescribed as viscous wall and the sea surface as inviscid wall. The outer boundaries are defined as inflow and outflow boundary condition, from the characteristics relations of the Euler equations. The domain temperature is defined as 288^o K and a 1 atm value is used for pressure, during the initialization of the simulation. The typical CFD process involves three stages, namely the pre-processing, solution of the flow equations and postprocessing the results. The pre-processing step includes geometry creation or clean-up, domain definition and mesh generation. The solution step requires several input parameters, for instance, the definition of initial and boundary conditions, numerical schemes parameters and convergence criterion. The post-processing of the numerical solution enables the CFD analyst to visualize and query the variables in the flow field and geometry model. Each of these steps, for the present work, are detailed in the forthcoming subsections.

3.1. Geometry

The Inhaúma corvette geometry was provided by the Brazilian Navy as a STL file (Kai et al., 2003), shown in figure 4. However, the initial geometry of the ship contained a large number of detailed structures which were not relevant for the simulation context. A large amount of the total simulation time was devoted to geometry repair and cleaning. The irrelevant structures, such as ladders, internal geometries, radar supporting frames and so on were individually removed and the remaining holes into the ship surface were patched manually. Table 1 shows a breakdown of the required man hours, for this project, into categories. The geometry repair duties represent over 10% of the total simulation time, which translates to 12 man-hours. Moreover, the geometry and meshing steps are closely related, in the sense that the meshing requirements will drive geometry modifications. Figure 4(c) presents the curves and points required for proper mesh generation, which were not originally part of the geometry. If such entities were not present, the mesh elements would not respect sharp angles and could cause severe mesh problems, causing solver convergence problems or render the entire simulation unstable. The geometry and meshing efforts, despite highly laborious, are minimized when one requires many simulations of the same geometry. The analyst is able to study a range of flow conditions, ship alignments and free stream velocities with the same mesh. Such a remark is possible due to the nature of this wok, which involves no geometry and, consequently, mesh modifications from one flow condition to the other.

Table 1. Simulation time breakdown.

Task	Time	Share
Geometry	12 h	11 %
Mesh	70 h	66 %
Setup	0.5 h	0.5 %
Solver	16 h	15 %
Post-processing	8 h	7.5 %

3.2. Meshing

One of the first steps required for meshing is the definition of the domain limits and type of boundary conditions. Figure 5 shows the domain created for the ship simulation. It represents a first attempt for the ship spatial discretization. Further studies will investigate in more details the influence of the domain size and determine the optimum distances from the geometry of interest. However, for the prescribe conditions, as discussed in Section 2, the authors have found that the domain geometry is adequate. In other words, no reflection to the interior of the domain was observed. The domain is 800 m long, 200 m wide and 100 m tall, as the figure indicates. Figure 6 presents the surface mesh around the ship, from various angles. The larger geometric structures were preserved, mainly the ones that would bear an a prior effect on the flow field on the flight deck. The ship chimney, life-boat, radar dishes and guns are among such structures. It is important to note that, due to the simulation requirements, which involve flow analysis from different angles, with respect to the ship heading, no symmetry condition was applied to the geometry. The complete corvette geometry is meshed. One can observe, for instance, that the life-boat is installed only on the starboard side of the ship. Several iterations of the mesh generation process were performed, as Table 1 indicates, to achieve the optimal distribution and preserve the geometric feature representation of the mesh. The mesh has a total of 8,655,939 cells, composed mainly of triangles for the surface mesh, and tetrahedra and prismatic cells for the volumetric mesh

3.3. Simulation Setup

The setup of the simulation is a relatively quick procedure, compared to all of the other steps required for the present simulation, as shown in Table 1. The CFD++ solver (Metacomp Technologies, 2013) prescribes several default values for numerical method parameters, relaxation scheme coefficients and so forth. The initial and boundary conditions, as described in Section 2, are quickly set and the analyst has to define the equation set to use, as well as the turbulence model. The preconditioned compressible

RANS equations (Metacomp Technologies, 2013) are considered for this study, along with the realizable k − ǫ turbulence model (Metacomp Technologies, 2013). A CFL value of 150 is defined for this simulation. It is expected that the solution would converge to a steady state. However, with the possibility of unsteady vortex detachments, source term production and other variables, the solution might, for different configurations, be unsteady. However, for the particular setup of the present study, the residue histories of all equations indicate that a 4-order magnitude reduction is achieved. The turbulence model equations present a different behavior though, which might be associated with unsteady occurrences within the simulation. As Table 1 shows, a total of 16 hours of computer time was necessary to achieve this residue level. The solution is computed by a 8-core workstation in parallel.

3.4. Computational Results

The present section discusses the results observed for the particular setup of this simulation, that is, a free stream flow at 10 m/s aligned with the ship centerline. This configuration might be the worst case scenario, from a helicopter pilot perspective, due to the alignment of the flow with the ship. Large vortex structures are observed well over half of the flight deck. The results below are given in the SI unit system. Figure 7 shows velocity magnitude contours and vectors in planes A, B and C. A low-speed flow re-circulation is observed over the flight deck, which might be associated with the wake of the geometric structures upstream the flight-deck. It is relevant to note here that this particular distribution is associated with the solution at the final iteration of the simulation. Due to the unsteady nature of the ship structure wake propagation, although similar behavior is expected after convergence, a different visualization might occur if one looks at a different snapshot of the solution. Future studies and meshes might alleviate the viscous layer discretization requirements for faster CFD analyses.

4. WIND TUNNEL TESTS

Parallel to the numerical simulations, experiments were conducted at the IAE/ALA TA-2 wind tunnel, located at Departamento de Ciência e Tecnologia Aeroespacial in São José dos Campos. The tests were performed with a 1/65 scale model of the corvette. The first part of the new campaign for quantitative measurements took place in june 2012, and used Stereo-Particle Image Velocimetry (S-PIV) for the quantitative measurements. S-PIV is an optical method used to obtain instantaneous three components velocity flow field. Figure 8 shows the model inside the wind tunnel test section with the cameras of the S-PIV system outside. In detail the green square represents the image area.

The fluid is seeded with tracer particles and it is illuminated so that particles are visible. The motion of the seeding particles is used to calculate speed and direction (the velocity field) of the flow being studied. The TA-2 S-PIV apparatus consists of two digital cameras (2048x2048 px², $1\ \mathrm{px} = 0{,}068\ \mathrm{mm}$), a 200 mJ laser Nd-Yag optical arrangement to limit the physical region illuminated and a synchronizer to act as an external trigger for control of the camera and laser.

4.1. Test Conditions

No boundary layer was used in this first part of the campaign. Figure 3 shows the top view position of the laser sheets, and the figure 9 shows the dimensions of the laser sheet. The tests were done with the model aligned with the wind direction. The wind speed was set at 50 knots (25.72 m/s). The A, B and C laser sheets are located to match the positions of the anemometers in the *in situ* test. Each measurement had 200 flashes separated by 105 μs. The software DANTEC™ has been used to treat the images. After data analysis, it was found that it was necessary more flashes, as the standard deviation was too high, specially in the region with stronger perturbation caused by the superstructure.

4.2. Wind Tunnel Tests Results

Figure 10 shows the results in three vertical planes matching the nine points where the data has been collected using anemometers.

5. MEASUREMENTS CONDUCTED *IN SITU*

The measurements were conducted in 1993-1994 in a Brazilian Navy Inhaúma-class corvette. Two anemometers attached to a mast (see figure 11) at 5m and 8m on the deck and repositioned throughtout 9 fixed points (see figure 2b).

The tests were conduct for different wind intensities and directions. In these tests, the gyro of the ship has been recorded, and the sea state and wave direction, along with the ship's speed, in order to obtain the movement of the ship concerning these factors. Unfortunately, the limitations in ship availability prevented the measurements in all the situations programmed, limiting experimental data were gathered.

The data gathering took three minutes, more seven minutes to change position of the mast. Thus, the measurement in each point took 10 minutes leading a more than 90 minutes testing. In this meantime the wind speed changes. Efforts to maintain the relative wind speed constant were done by adjustments in the ship speed.

6. CFD SIMULATIONS, WIND TUNNEL AND *IN SITU* DATA COMPARISON

The comparison among time averaged CFD simulations, time averaged wind tunnel data and *in situ* data collected have been done with the wind aligned. (Actually, *in situ* data has been collected with relative wind direction between 355° and 5°). Wind tunnel data has been collected with a test section mean velocity of 50 knots whereas CFD simulations have been performed at 10 m/s = 19.4 knots. The *in situ* data is collected under a wide variety of wind conditions (20 to 30 knots though typically 23 to 28 knots). To ensure proper velocity magnitude comparisons among the different data sets, all data was scaled to have the same mean velocity magnitude in the measuring anemometers in the ship.

Figures 12 and 13 compare the velocity direction and magnitude at common positions that correspond to where data was collected using anemometers.

Reviewing figures 12 and 13 we notice there is reasonable coincidence comparing the *in situ* and wind tunnel data and the CFD simulation results. The flow directions, with some exceptions, are aligned within 5-15°. The wind direction data does differ significantly more than 15° at locations close to the hangar. We believe that these disparities arise because in these positions the air wake is in the recirculating region of the flight deck. Moreover, as mentioned before, the amount of flashes in the wind tunnel tests was small, and in the region more unstable this flaw is more notable.

The coincidence was greater at the points located 8 m above flight deck.

Comparing the average speed, the three methods were close to each other. The largest difference occurs in the regions that have more disturbed flow.

7. CONCLUDING REMARKS

The numerical simulation, the wind tunnel and *in situ* tests of air flow for the Inhaúma corvette, were performed.

Despite some different wind magnitude and direction at particular positions, the tests have shown reasonable coincidence between collected data *in situ* and wind tunnel compared to CFD flow simulations. The CFD was able to determine the regions where perturbation caused by the ship superstructure occurred. It could be used to concentrate ~~the~~ test efforts in wind tunnel and *in situ* for a smaller region, saving time and costs, and increasing the tests precision.

The CFD analysis presented here shows the viability of numerical launch/recover envelope determination for helicopters in the particular ship considered. The most expensive component of the work is related to geometry and mesh preparation, due to the complexity of the model. Experimental data is currently being processed in order to allow a deeper validation of the numerical simulations. Future studies will perform such comparisons and, eventually, improve the mesh discretization and numerical parameters tuning for convergence. It is hoped that such comparison will increase the confidence in the numerical model development and, in the future, allow a smaller number of expensive wind tunnel test hours by replacing them with CFD simulations.

In order to investigate the areas of disagreement, a new tunnel test campaign is scheduled this year and a more refined 3D model is being developed for new CFD simulations.

8. ACKNOWLEDGEMENTS

The authors gratefully acknowledge the partial support for this research provided by the Financiadora de Estudos e Projetos (FINEP), under the Research Grant nº 01-10-0642-00 Ref nº 2618/09 and by the Conselho Nacional de Desenvolvimento Científico e Tecnológico (CNPq), under the Research Grants No. 312064/2006-3 and No. 471592/2011-0. This work is also supported by Fundação Coordenação de Aperfeiçoamento de Pessoal de Nível Superior (CAPES) through a Ph.D. Scholarship for one of the authors. The authors are also indebted to the partial financial support received from Fundação de Amparo à Pesquisa do Estado de São Paulo (FAPESP), under Grant No. 2013/07375-0.

9. COPYRIGHT STATEMENT

10. REFERENCES

[1] Kai, C.C., Fai, L.K. and Chu-Sing, L., 2003. **Rapid Prototyping Principles and Applications**. World Scientific Pub Co Inc, 2nd edition.

[2] Metacomp Technologies, 2013. “CFD++”. URL http://www.metacomptech.com/. [Online; accessed 2013 June 10th].

[3] Marcus Borges, **Relatório da 1ª Campanha de Ensaios em Túnel de vento da Corveta Inhaúma**, Instituto de Aeronáutica e Espaço, 1991.

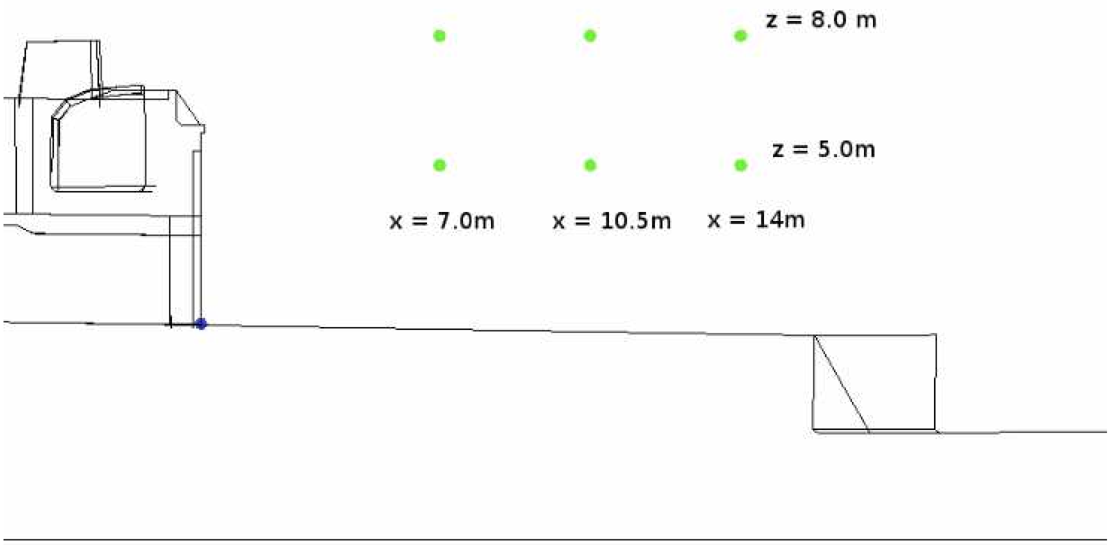
[4] Ray R. Snyder, Hyung Suk Kang, Cody J. Brownell, Luksa Luznik, David S. Miklosovic and John S. Burks, **USNA Ship Air Wake Program Overview**, 29th AIAA Applied Aerodynamics Conference, Honolulu, Hawai, 2011.

[5] Susan A. Polsky, David S. Miklosovic, **CFD Study of Bluff Body Wake from a Hangar**, 29th AIAA Applied Aerodynamics Conference, Honolulu, Hawai, 2011.

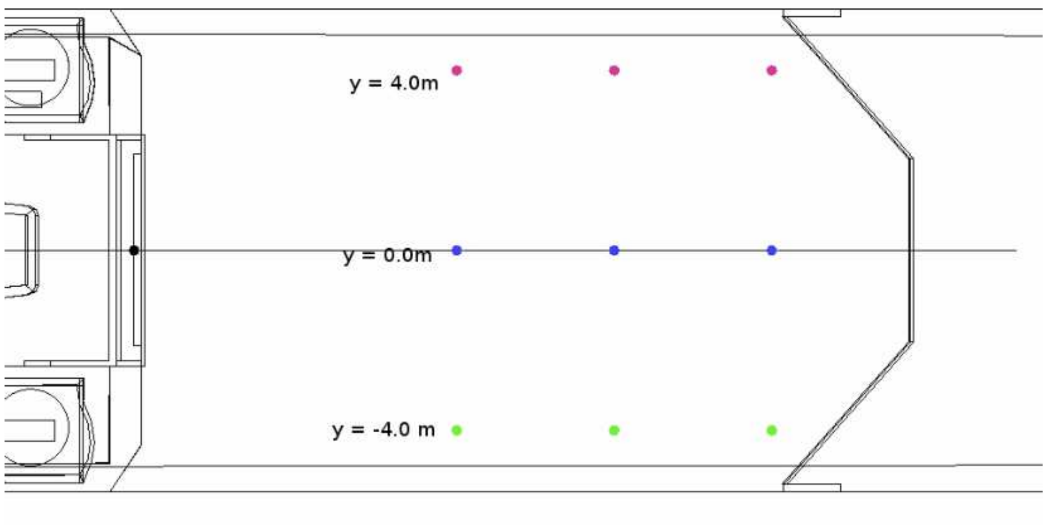
[6] Matsuo Chisaki, **Confiabilidade dos Coeficientes Aerodinâmicos Obtidos em Túnel de Vento do**

Instituto de Aeronáutica e Espaço (IAE), MSc thesis, Taubate University, São Paulo, 2010
[7] Luís Morais, Matsuo Chisaki, **Ensaios Bidimensionais em Túnel de Vento**, CTA / IAE / ASA-L, São José dos Campos, Brazil.
[8] GUEDES, M.; VIEIRA, M.; MARTINS, E.. **Brazilian Navy Air Wake Program**, AIAA Modeling and Simulation Technologies Conference, Minneapolis, Minnesota, 2012.

Figure 1 – Inhaúma-class corvette in operation



a) Side view



b) Top view

Figure 2 – Position of the anemometers in the *in situ* tests

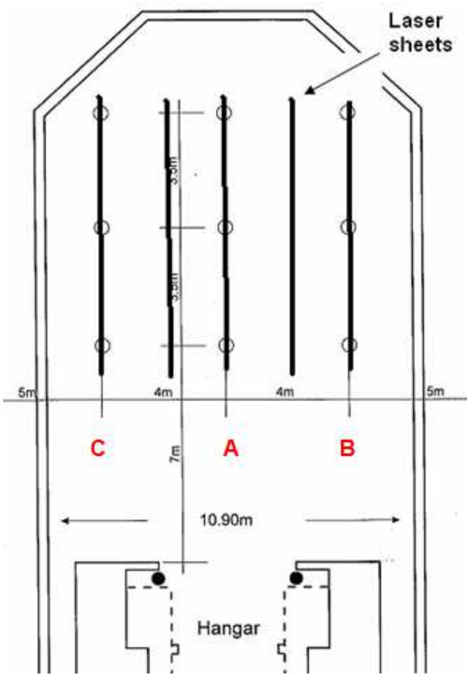
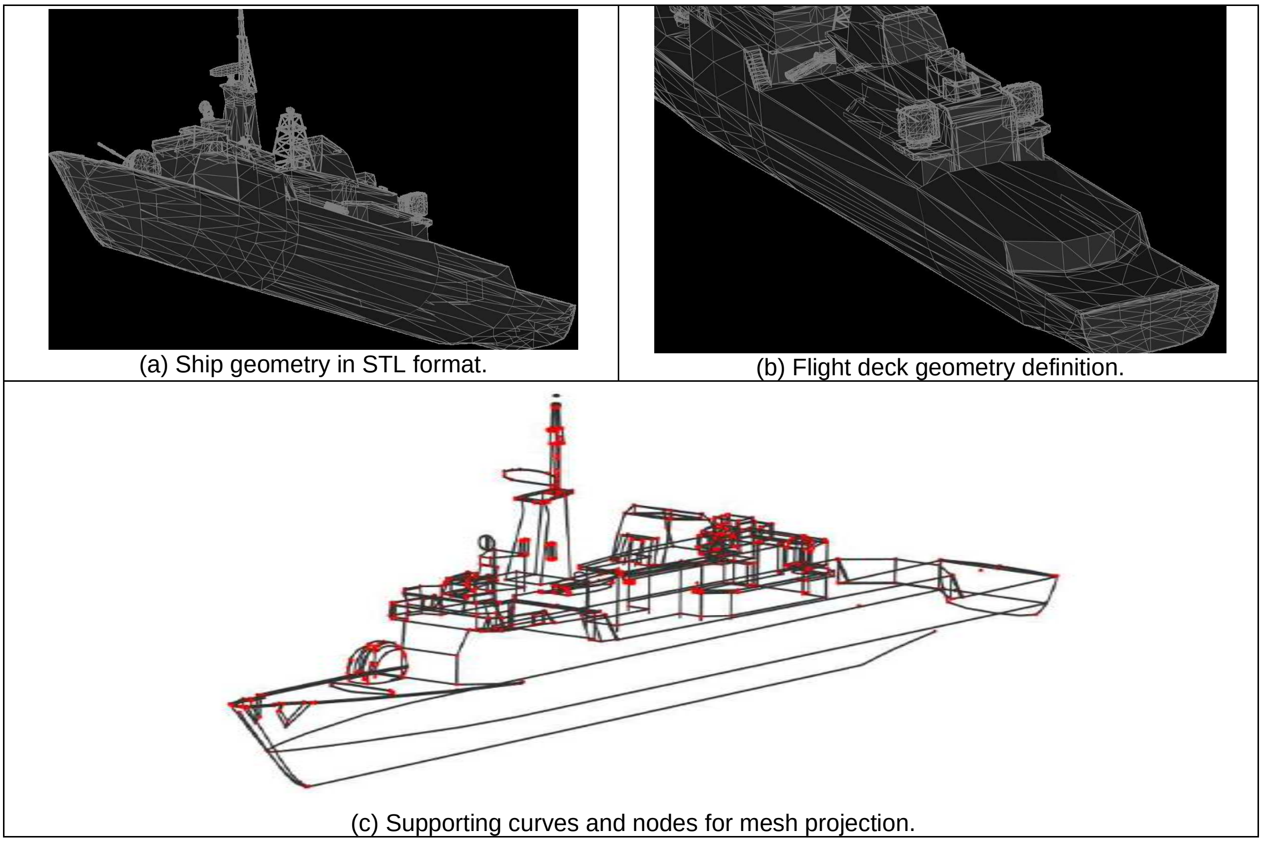


Figure 3 - Schematic representation of the laser sheets in the wind tunnel tests



(a) Ship geometry in STL format.

(b) Flight deck geometry definition.

(c) Supporting curves and nodes for mesh projection.

Figure 4. Computational geometry of the Inhaúma-class corvette.

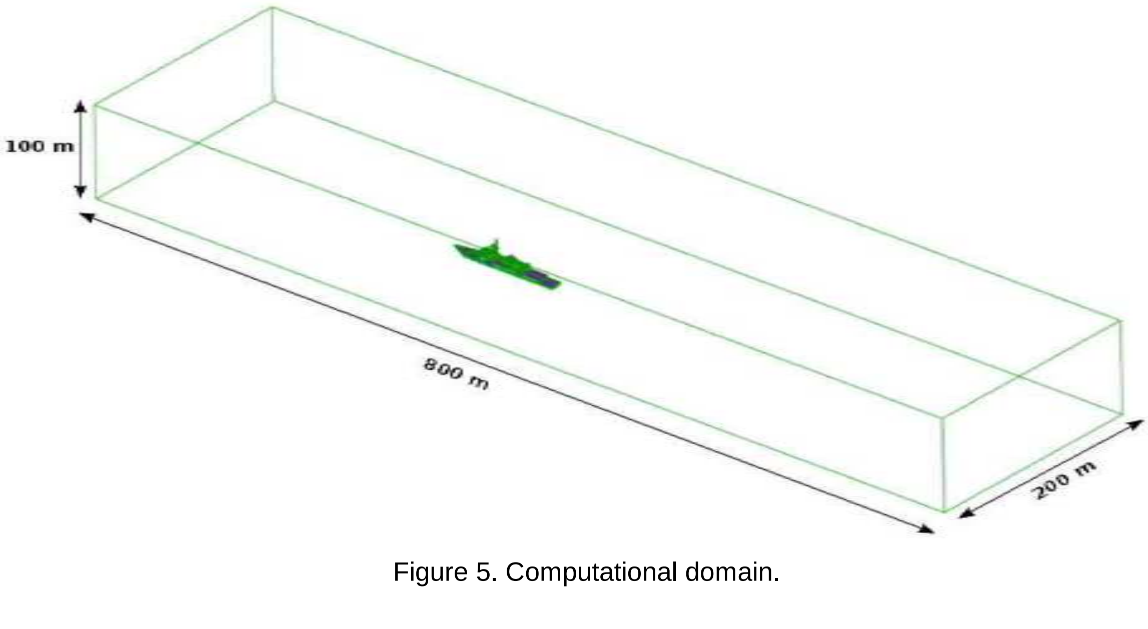


Figure 5. Computational domain.

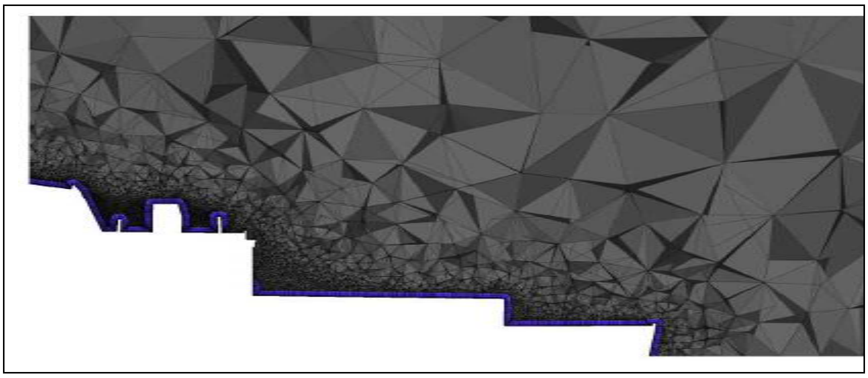
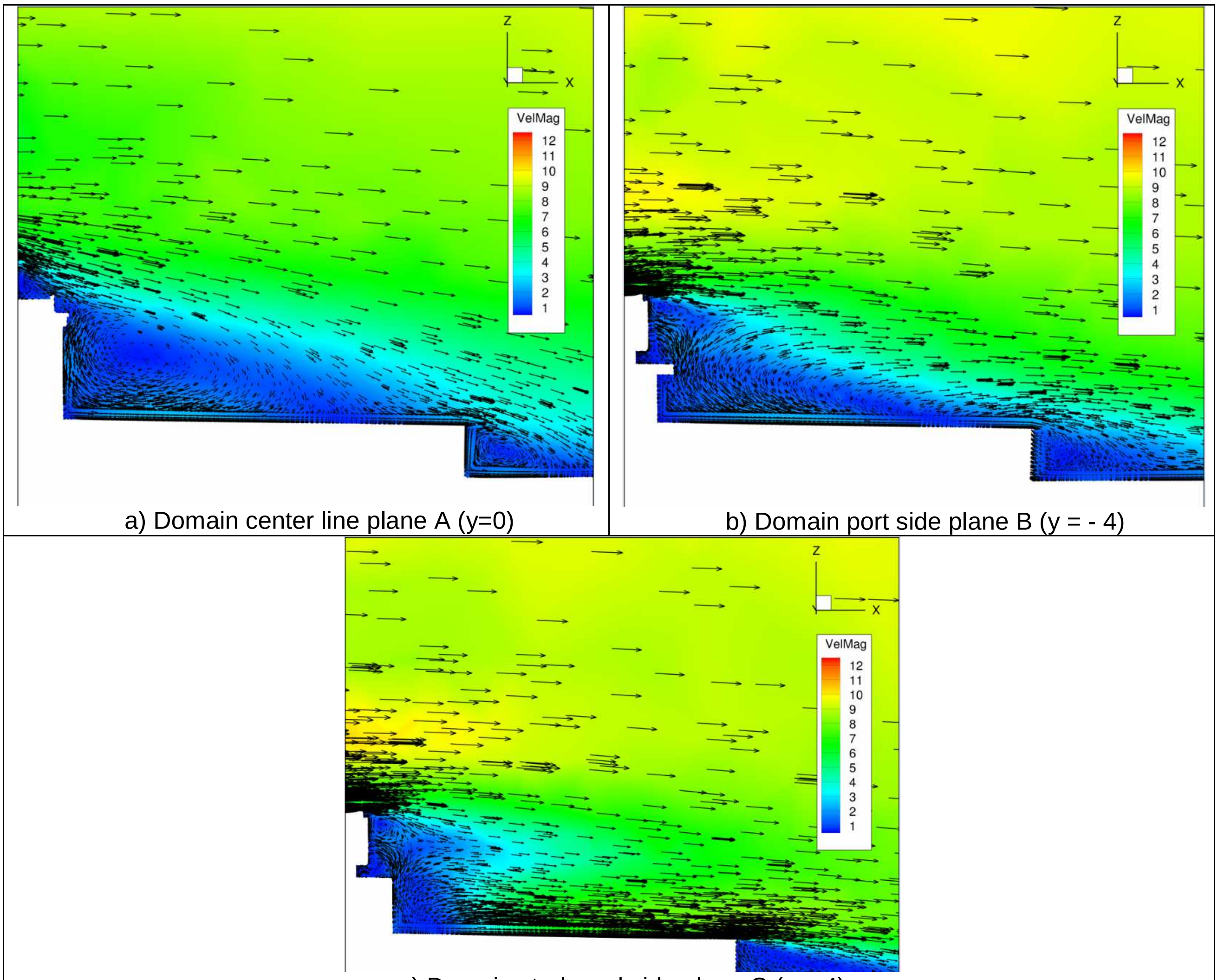


Figure 6. Volumetric mesh visualization on flight deck region.



a) Domain center line plane A (y=0)

b) Domain port side plane B (y = - 4)

c) Domain starboard side plane C (y = 4)

Figure 7. – CFD results: velocity magnitude [m/s] over the flight deck

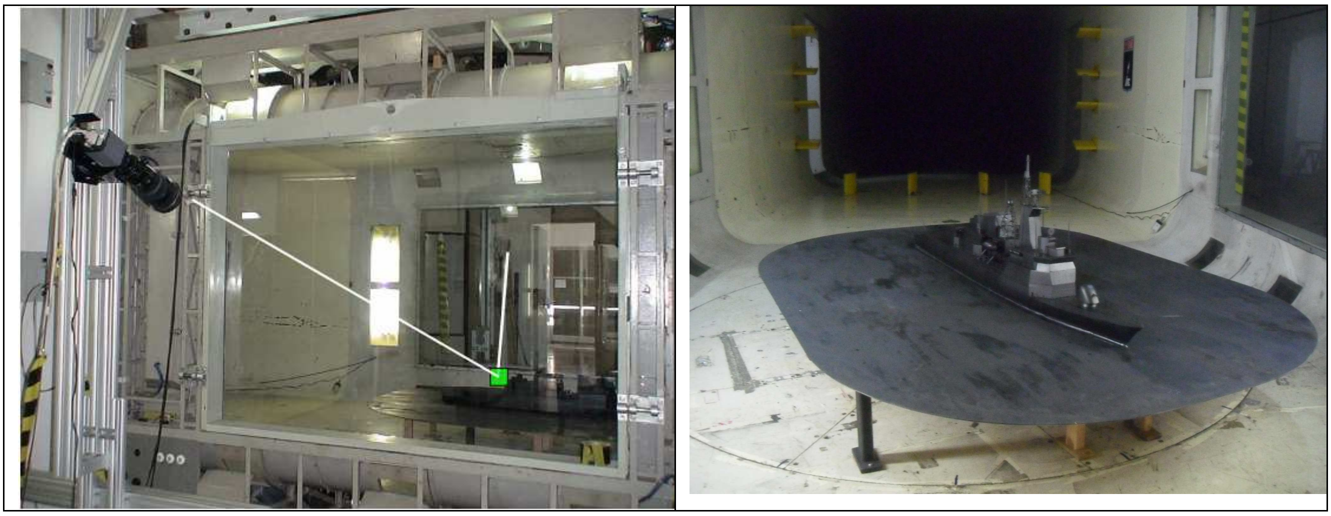


Figure 8 – 1/65 scale model in the IAE wind tunnel

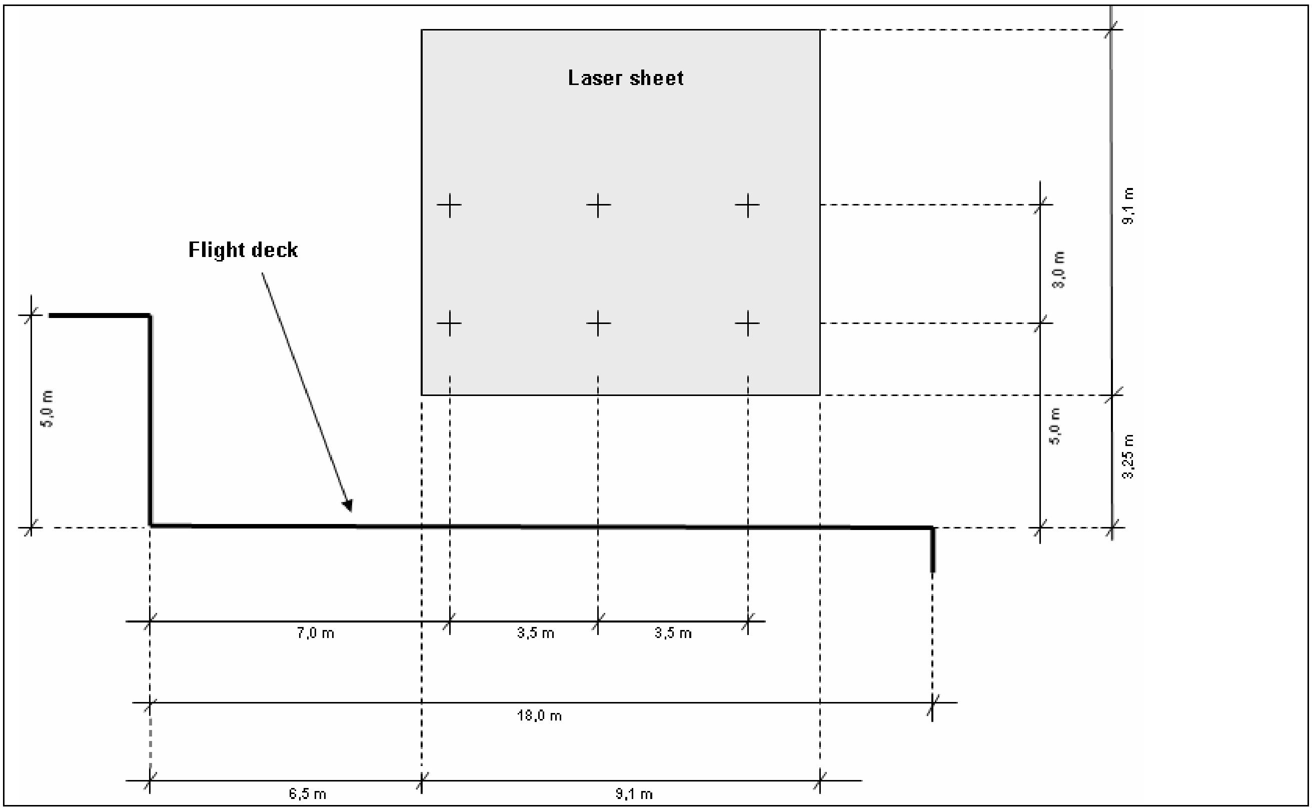
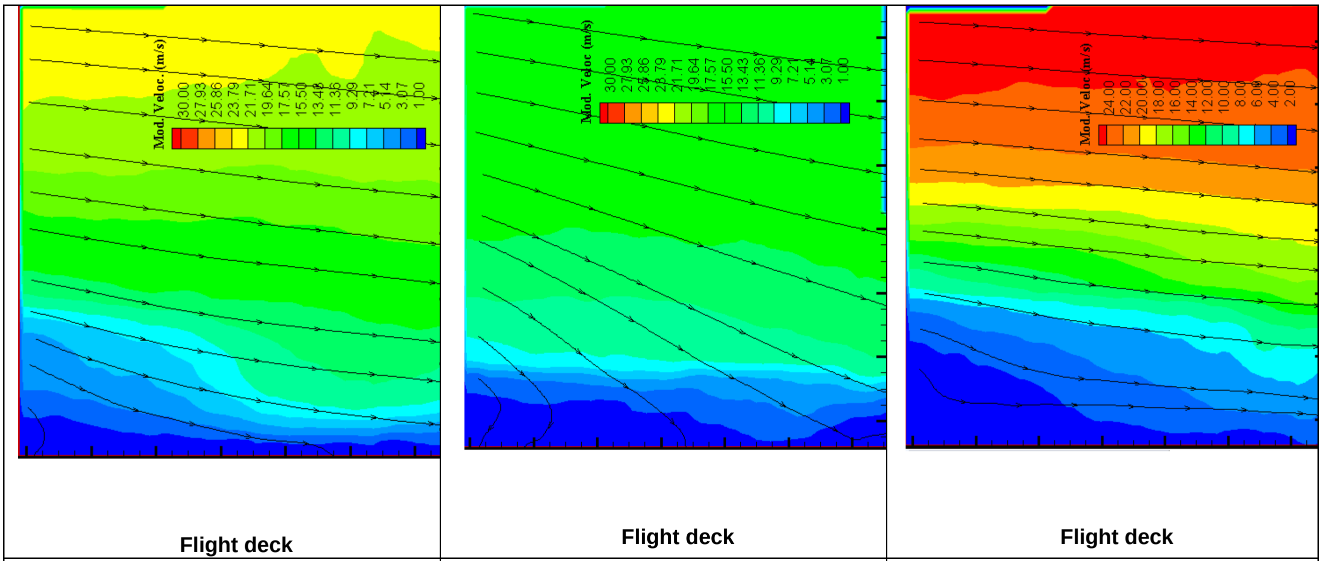


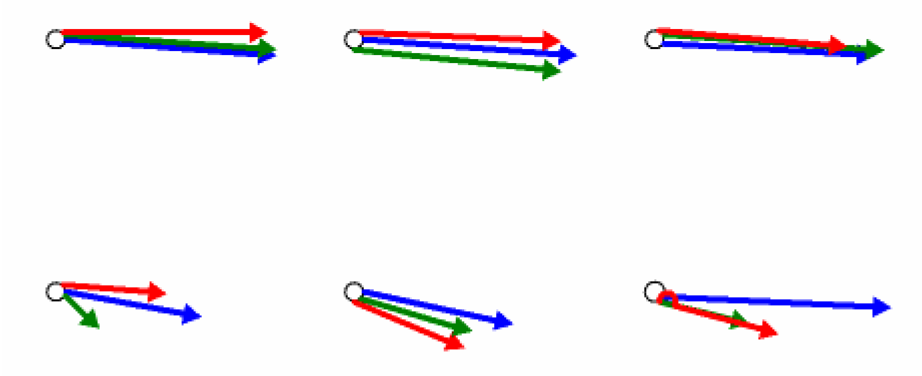
Figure 9 – Laser sheet position and dimension



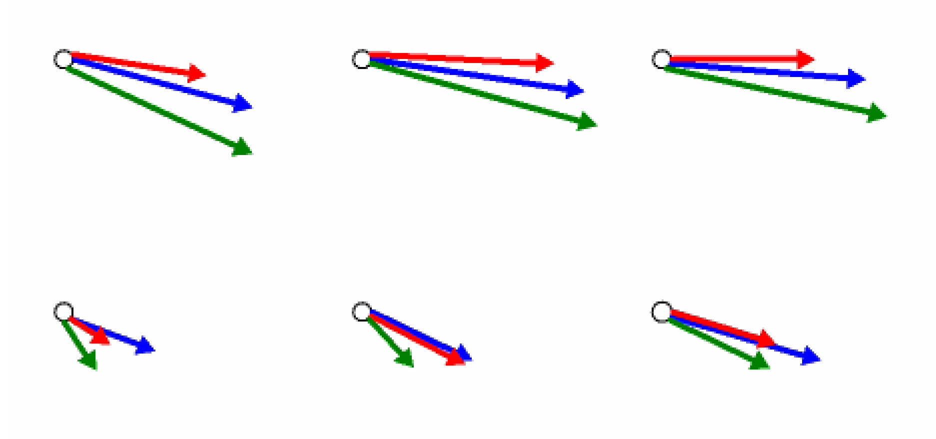
a) Domain starboard plane C (y=4)	b) Domain center line plane A (y=0)	c) Domain port side plane B (y=-4)

Figure 10 – Wind tunnel results: velocity magnitude [m/s] over the flight deck

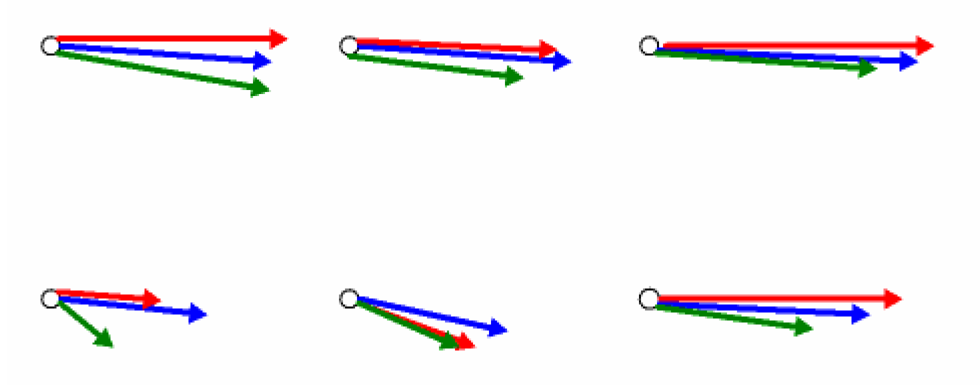
Figure 11 – Gill anemometers at the measurement mast



a) Domain starboard plane C (y=4)

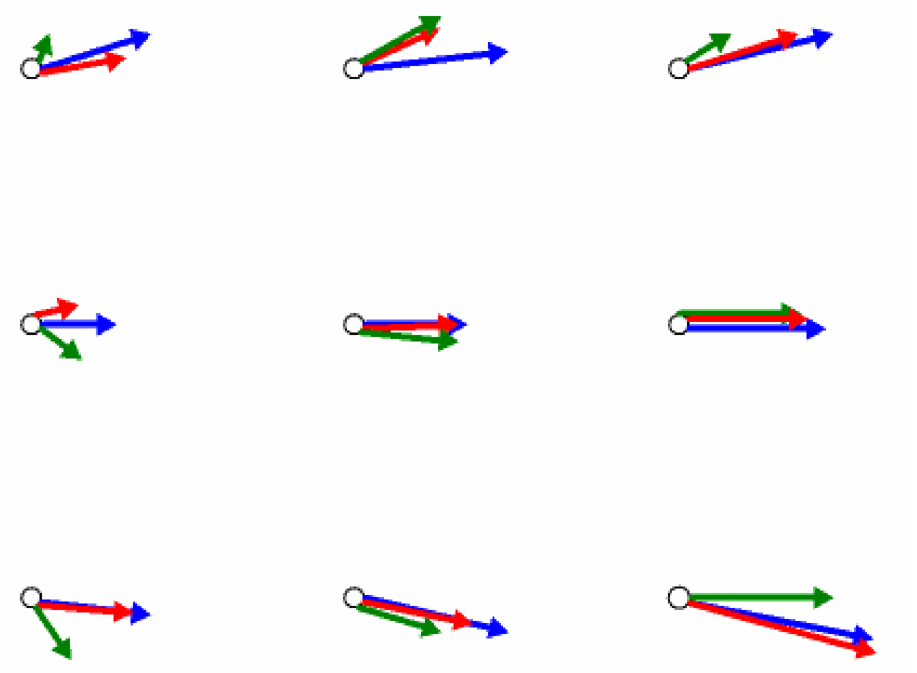


b) Domain center line plane (y=0)

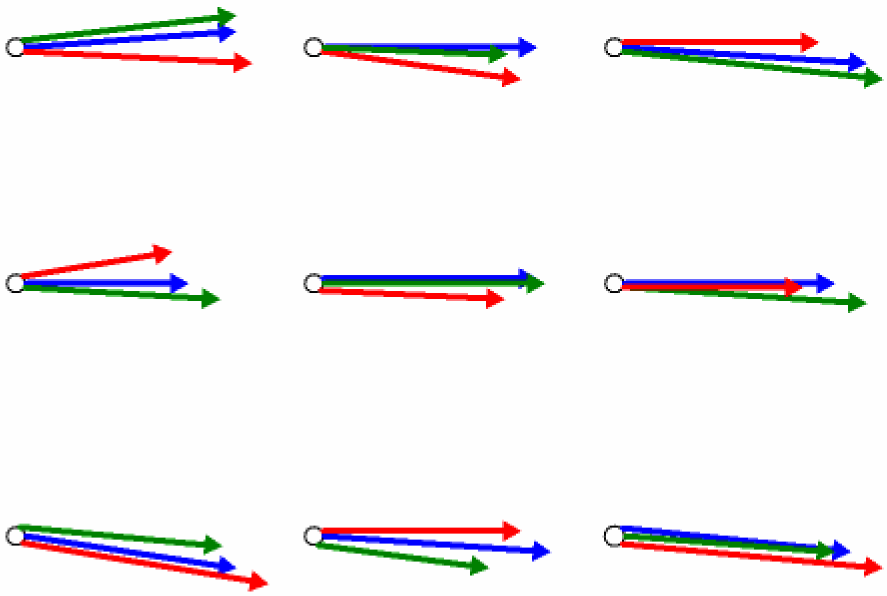


b) Domain port side plane B (y=-4)

Figure 12 - Vertical planes for a head wind (red arrows are *in situ* data, green arrows are wind tunnel data and blue arrows are CFD data)



a) Top view, 5m height over the flight deck



b) Top view, 8 m height over the flight deck

Figure 13 - Horizontal planes for a head wind (red arrows are *in situ* data, green arrows are wind tunnel data and blue arrows are CFD data)