Rotorcraft/ship dynamic interface simulation developments at NLR

Jasper van der Vorst, Joop Gooden and Jaap van Muijden
National Aerospace Laboratory, NLR
Anthony Fokkerweg 2, 1059 CM Amsterdam, The Netherlands
Telephone: +31205113322, Fax: +31205113210
E-mail: vorst@nlr.nl

Project background

NLR is developing a simulation capacity in the framework of the ROSDIS project (ROtorcraft Ship Dynamic Interface Simulation) aimed at supporting the current method of determining Ship-Helicopter Operational Limits (SHOL) for the Royal Netherlands Navy (RNLN) in the future through the use of piloted simulation.
A dedicated wind tunnel test campaign was conducted on ship models to both generate data for direct use in a helicopter flight simulator and to validate ship airflow calculations using CFD techniques.

In the framework of this project attention has been paid to international cooperation: a project has been set up with ONERA in the field of ship air wake measuring and modeling. Also, results from the project have been shared in the RTO AVT-148 panel ("Launch and Recovery in a Ship Air Wake Environment ")

Wind tunnel test set-up

In a previous dynamic interface simulation project at NLR several deficiencies in available wind tunnel data have been identified. As a result, a follow-on project was designed to investigate the airflow around the deck in more detail. In order to connect with future flight trials and to connect with ongoing research at ONERA it was decided to select the Royal Netherlands Navy Landing Platform Dock-2 (LPD-2) ship and various Simplified Frigate Shape (SFS) wind tunnel models.

The main purpose of the wind tunnel test campaign conducted in the DNW-LST was to obtain time-average and turbulence (intensities and spectra) flow data mainly above the flight deck of the LPD2.
Due to the complicated flow field and the extended separations occurring behind the ship’s super structure, Particle Image Velocimetry (PIV) was selected as the most appropriate measurement technique, which can cover the whole flow field on the flight deck. However, although PIV is perfectly able to determine the instantaneous flow field, measuring turbulence quantities like spectra by means of PIV still is not straight-forward. The unsteady PIV data will therefore be

complimented with hot wire turbulence data obtained at certain positions for equal conditions in a previous test campaign.

The tests were performed in the Low Speed wind Tunnel (LST) of DNW. This is an atmospheric wind tunnel of the closed return type with a contraction ratio of 9. The maximum velocity in the (empty) test section is about 80 m/s. The tunnel is equipped with an interchangeable aeronautical test section and a fixed non-aeronautical test section.
The interchangeable test section is 3 m wide and 2.25 m high and has a length of 5.75 m. Downstream of this interchangeable aeronautical test section is the fixed non-aeronautical test section with a length of 3 m. Each test section is equipped with a turntable, which is flush with the floor of the test section. Although the tunnel can be equipped with a system to simulate the wind velocity distribution in a natural atmospheric boundary layer over the full height of the test section, this simulation will not be installed for the present test. Only the ordinary tunnel floor boundary layer, being around 0.10 m thick at the model position, will be present.
Two different ship models were used: a model of the LPD2 ship and a SFS model. The LPD2 ship model was of a 1:100 scale. Both models were mounted on the 1.8 m diameter, 2D-turntable in the floor of the aeronautical test section. The Simple Frigate Shape (SFS) was equipped with different bows (see Figure 1):

- the original TTCP SFS-shape, that has no bow but a blunt forward face (‘SFS’);
- the NLR bow, designed to represent a more normal ship shape and the flow patterns that are representative for that in a boundary layer flow (‘SFSN’). The bow shape is entirely analytic to allow for easy gridding for CFD applications;
- the pyramidal ONERA-nose with a ridge, designed for testing on a ground board without boundary layer (‘SFSO’), this configuration is tested to get data on the ONERA shape in a boundary layer flow.

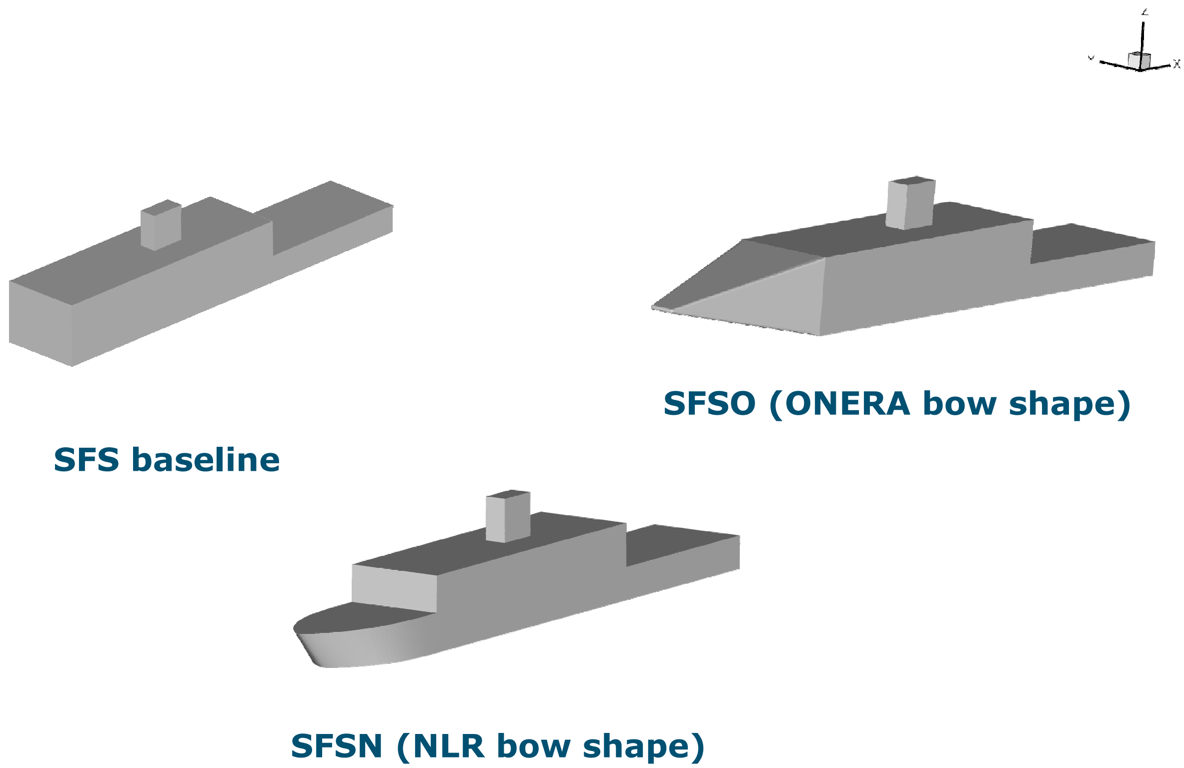

Figure 1: Simple Frigate Shape layouts

The DNW 3-component PIV system was used, allowing 3D velocity data to be acquired in horizontal planes at small steps in height above the helicopter deck. These measurements could be compared to full scale wind climate data, obtained at a number of spots on the deck at 3, 5 and 10m above the flight deck. The PIV cameras were mounted in the upper turn table, such that the field of view relative to the ship remains unchanged if β is changed. The physical dimensions of the field of view were around 0.4 x 0.4 m2, using an 85 mm focal length camera lens, to cover the full flight deck in one picture frame (see Figure 2). Also some data on the leeward and windward side of the upper structure have been obtained for LPD-2. The PIV measurements were performed at a fixed frame acquisition frequency of about 2 Hz. At each data point 1024 flow samples have been taken, to enable obtaining information on the turbulence intensities. Added to the measurement system was smoke, oil (Figure 4) and tufts (Figure 3) for flow visualization purposes to support the interpretation of the quantitative measurements taken.

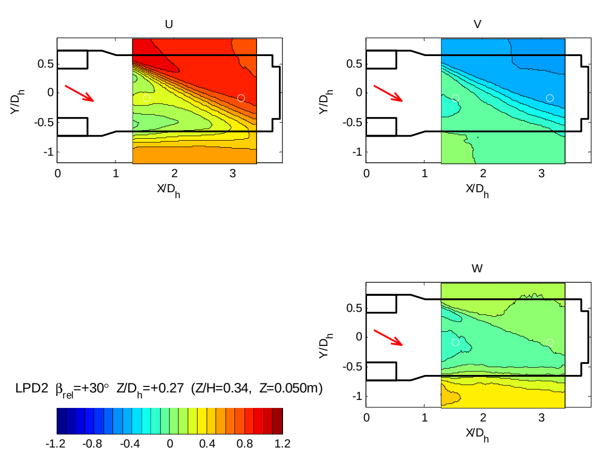

Figure 2: PIV results

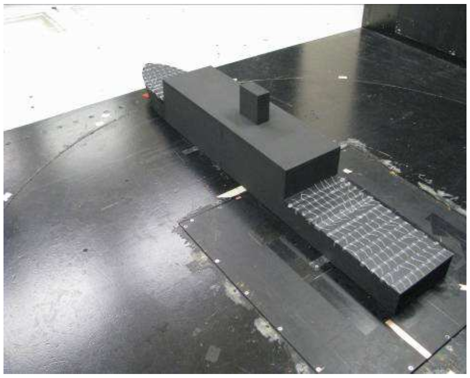

Figure 3: Tufts on wind tunnel model

Aerodynamically, a ship is a bluff body what makes the airflow insensitive to the Reynolds number. Therefore the wind tunnel speed can be chosen to suit the test optimally. A wind tunnel speed of 30 m/s has proven to be a good practical value for the subject purpose. The measurements in general were performed at β = 0°, +30°, +60°, +90°, and +150° green wind (i.e. wind coming from starboard). For one wind direction a red wind is chosen to obtain data on the windward side of the upper structure of LPD-2 (β = -30°).

Wind tunnel test results

The wind tunnel experiments were conducted during 2 test campaigns in DNW-LST, in October 2008 for 4 days and in January 2009 with a duration of 12 tunnel occupation days.

The bulk of the data processing and analysis of the PIV and hot wire data was done by NLR. Next, data analysis focused on the SFS frigate shapes. The analysis of the PIV data on these shapes has started mid 2009. Also the flow visualization data on the SFS have been analyzed.

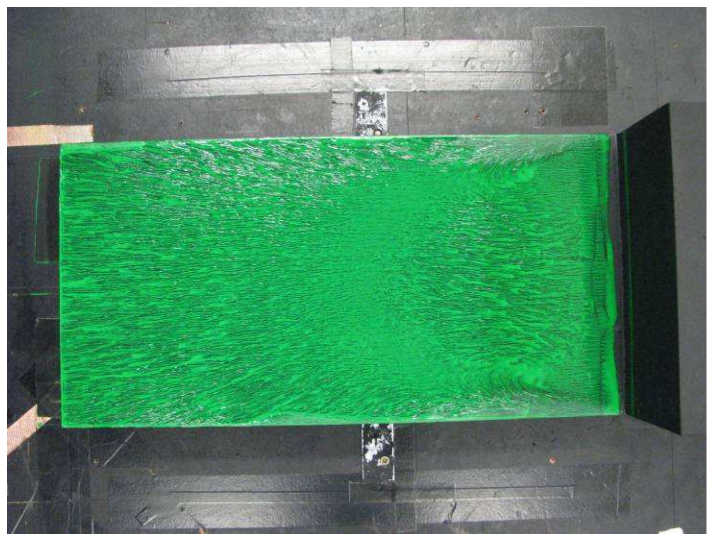

Figure 4: Oil pattern on SFS deck

As was found during earlier tests in other wind tunnels it appeared that the flow above the helicopter deck is sensitive to the wind direction for β's close to zero (-2° < β < +2°). It looks like some intermittent flow behavior exists causing the average pattern to quickly change from one side to the other side with varying β. For all SFS-configurations more or less asymmetrical flow behaviour is observed at β = 0°. The picture shows the surface oil pattern obtained on the basic SFS-configuration at β = 0° with the pattern on the deck biased to starboard. The bias is different for each nose shape. This is a complicating factor in comparing the experimental results with CFD at this wind direction.

In view of the existing flow instability around β = 0°, the SFS is less suited as a test case for small relative wind directions. This behaviour was not only observed in the present test, but has also been found by other investigators. Possibly this behaviour is caused by the fact that the location of the stagnation point on the front side of the model is not very well defined. Changing the bow shape did not seem to improve the flow stability. At wind directions larger than, say, 5° this problem seems to be less. As the SFS is more or less representative for normal ships, it must be remarked that this flow instability above the flight deck at small relative wind angles might also be present with normal ships.

Computational Fluid Dynamics approach

The purpose of CFD in rotorcraft/ship dynamic interface simulation is to enhance available experimental datasets beyond the measured range, and to have additional possibilities to study aspects such as the impact of rotor downwash on the overall flow field. The NLR in-house developed ENFLOW CFD system has been applied because it is suitable for complex geometries and can be run with different levels of fidelity. Current fluid modelling approaches in the ENFLOW CFD-system that are routinely applied comprise Euler, Reynolds-averaged Navier-Stokes (RANS) and a hybrid form of Large-Eddy Simulation (hybrid RANS-LES). Euler equations can be used for a "quick and dirty" look at interactions. Reynolds-averaged Navier-Stokes modelling can be used in steady and unsteady mode to resolve the main viscous flow characteristics over the ship, applying the k-ω turbulence model. Then to characterize more details of the flow, e.g. for specific side-slip conditions, hybrid RANS-LES may be used though computational times are significantly more demanding. A workable option for the application of hybrid RANS-LES has been obtained by using RANS near the model and the wind tunnel walls and to use LES in all other regions of the flow field.

In the framework of identifying a suitable computational approach for the flow field determining the helicopter/ship dynamic interface, initial calculations were conducted for the Simple Frigate Shapes (SFS) to be used in the comparisons with the wind tunnel tests. The SFS baseline is made up of rectangular blocks that approximate a frigate superstructure and flight deck. The SFSO adds the ONERA bow shape, a simple pyramid shape. The SFSN adds the NLR bow shape that looks like a frigate bow shape. Flow features and streamlines were plotted to compare steady Euler with RANS predictions for the various SFS- models in head winds. Flow features were studied using velocity and vorticity distributions. It has been concluded that RANS is the most suitable and physical relevant model to get a first impression of flow features around a ship. For further investigations of physical relevance of computed flow fields, RANS has been applied, for instance to adjust mesh sizes to achieve realistic boundary layer inflow for wind tunnel experiments and to achieve sufficient detail around the flight deck, and to study the impact of turbulence models on the flow solutions. Hybrid RANS-LES might be used for specific cases that require further attention, e.g. specific side-slip conditions showing a large impact on helicopter operations. Figure 5 shows streamlines (threedimensional as well as in the symmetry plane) obtained from steady RANS simulations for the simple frigate shape at zero side-slip angle.

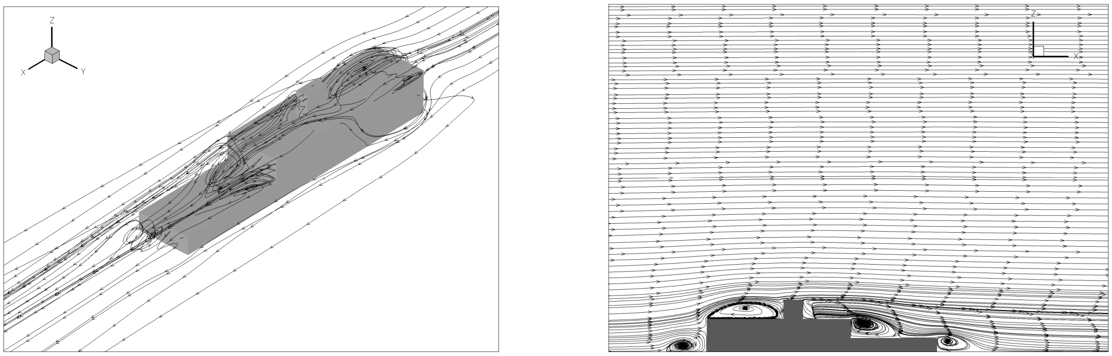

Figure 5: SFS streamlines.

During 2009-2010 the development of a realistic, workable computational approach for the determination of ship air wakes is being conducted, consisting of an assessment of the aforementioned computational modelling approaches against available experimental data. Experimental data are used to study the mesh resolutions required to resolve relevant levels of flow detail, and to obtain specific information regarding time-dependent behaviour of the flow which supports the choice for specific flow models. Experimental data are also used for the verification and validation of computational results.

In the end, CFD simulations will provide a full set of flow data around the ship that complements the available experimental data set. Usually, the experimental dataset focuses on the immediate vicinity of the flight deck. Within CFD-simulations, the whole flow field within the modelled domain is automatically captured. As a result, the look-up tables for fluid flow properties around ships for flight simulator applications can be enhanced, providing a full 360 degrees dataset at a range of vertical levels above the ship.

Computational Fluid Dynamics results

Currently, initial meshes and flow simulations have been obtained for simple frigate shapes using Euler, steady RANS, and unsteady RANS models. Also, RANS flow simulations have been obtained for the LPD-2. Comparisons with experimental data (specifically using oil flow patterns, later on using PIV-data) have started mid 2009 in order to assess the suitability of flow models and mesh resolutions. Indications have been obtained that the RANS flow model provides a more useful and relevant first impression of the flow field than the Euler model. Investigations are ongoing regarding the impact of the mesh resolution and turbulence models around the flight deck to resolve the main flow features. Special care has been taken to adjust the dimensions of the computational domain in order to obtain a comparable boundary layer thickness near the ship during simulations as is provided by the tunnel floor boundary layer in the experiments.

Furthermore, recent advances in the way hybrid RANS-LES should be applied to obtain physical relevant results, together with a significant increase in installed computer power, have paved the way to a more routinely application of this approach. Currently, hybrid RANS-LES simulations are ongoing to address the additional flow field information contained within this approach. The following figure shows an instantaneous view of three isosurfaces of vorticity around SFSN, obtained with the hybrid RANS-LES approach.

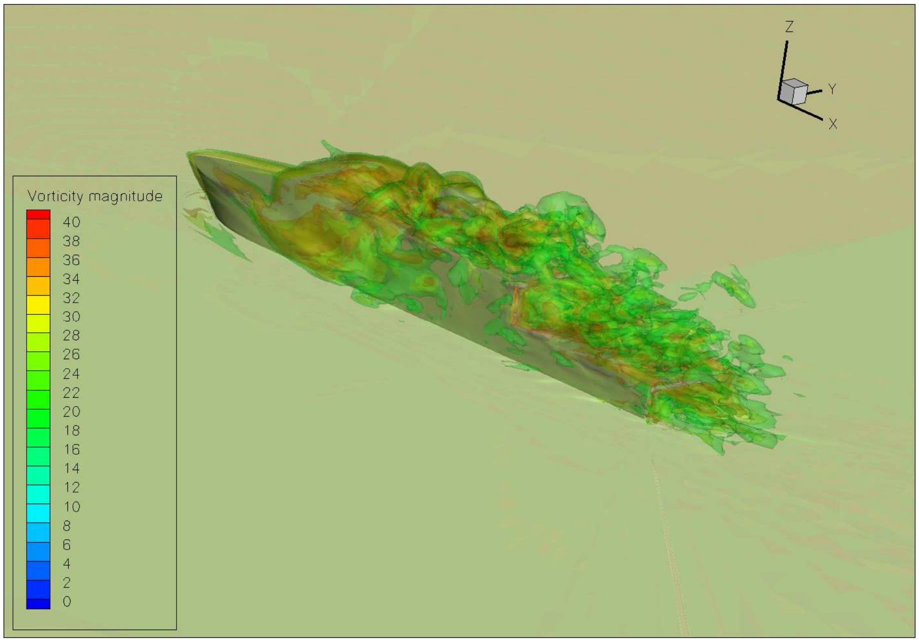

Figure 6: Isosurfaces of vorticity around SFSN with hybrid RANS-LES

Initial simulations have shown that RANS-based simulations give better and physically more relevant flow results than Euler-based simulations, provided that a suitable turbulence model is used. The computational effort required for RANS-based flow simulations for such configurations can be classified as moderate in comparison with simulations for aircraft. Further investigations are ongoing to establish if steady or unsteady simulations make a significant difference, and if hybrid RANS/LES simulations for flow fields with separated regions adds value to the outcome of the simulations.

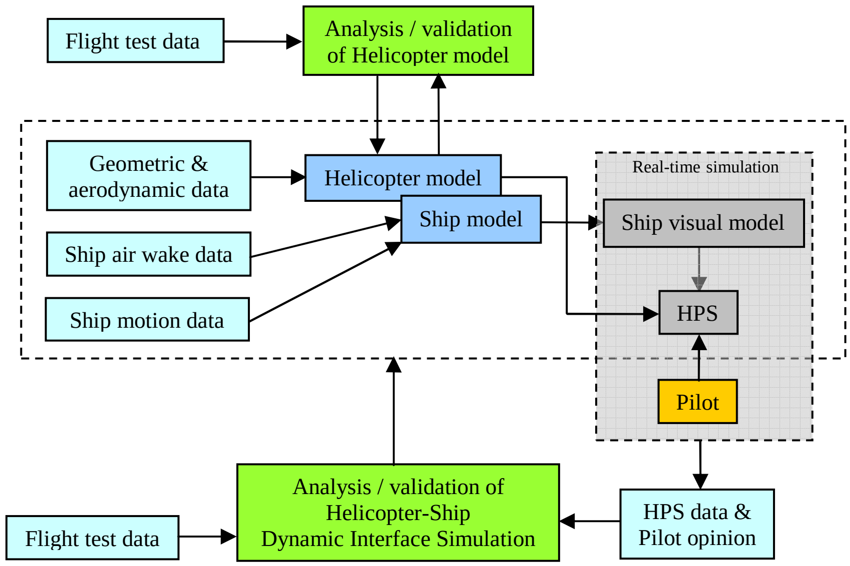

Figure 7: ROSDIS simulation approach

Pilot-in-the-loop simulations

At the end of the ROSDIS project piloted simulations will validate the airwake measurements and calculations described above. Figure 7 visualizes the individual sub-models and they way they are related. Focus in the ROSDIS project is on the ship airwake submodel.

During simulation sessions in a previous dynamic interface simulation project pilots have commented on a limited field of view, resulting in a lack of visual reference.
As a result NLR's research helicopter simulator 'Helicopter Pilot Station' (HPS) has undergone a major upgrade at the end of 2009 (see Figure 8). The visual projection system was upgraded to a field of view of 180° horizontally by 70° vertically (see Figure 9). At the same time the image generators and cockpit hardware were upgraded.

Figure 8: Overview of the upgraded Helicopter Pilot Station

The combined results of the wind tunnel testing and Computational Fluid Dynamics developments will be used in a piloted simulation session in NLR's Helicopter Pilot Station
Besides testing in NLR's simulator it is investigated if the results of the airwake research in the ROSDIS2 project can be incorporated into the NH90 Full Mission Flight Trainer that is being developed for the Royal Netherlands Navy.

Figure 9: View from NLR's new Helicopter Pilot Station.

Several other features will be incorporated into the Helicopter Pilot Station before the start of the piloted simulations, such as an automatic Flight Deck Officer, a detailed visual model of the applicable ship (LPD2 Johan de Witt, see Figure 10) and a Glide Path Indicator. Besides the piloted simulations, also a replay capability of flight test data will be demonstrated. Ship motion data from measurements with an Inertial Measurement Unit and GPS on board the LPD2 at sea have been Kalman filtered and will be used in the simulator.

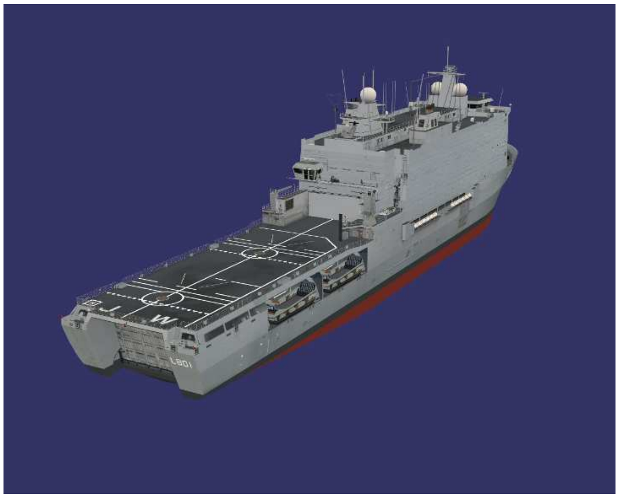

Figure 10: Detailed visual model of LPD2.

International cooperation

The development of the simulation environment is being conducted as part of a 5-year national program, that ran in parallel to AVT 148, allowing experience and data exchange with NATO partners. Furthermore a bi-national effort in being conducted with ONERA from France in the area of ship air wake modeling.

Future work

- During the LPD2/SFS measurements in DNW's LST 4TeraByte of PIV-data has been recorded. More attention will be paid to extracting information regarding unsteady flow.

- The LPD2 CFD calculations will be completed and validated against wind tunnel measurements. Time and budget permitting some initial calculations will be performed of a rotor in the ship's airwake.

- A simulation trial parallel to actual trials at sea will be performed to investigate the use of simulation in NLR's SHOL determination process.

The ROSDIS project has been running at NLR since 2003. In the latest stage of the project research has been conducted into the measurement of ship airwakes using PIV techniques and into the calculation of ship airwakes using Computational Fluid Dynamics. NLR has made a significant step forward in its ability to measure and predict the ship's airwake for use in rotorcraft/ship dynamic interface simulation.