WP 1 | Deliverable D.6.1

Application of CFD tools to the development of a novel propulsion concept

Author(s): Antonio Sánchez-Caja
Partner Code: VTT
Document Type: Deliverable
Document status: Public

Revision | Status | Date
--- | --- | ---
1 | Draft | 1-06-2013
2 | Draft | 14-06-2013
3 | Final | 30-06-2013

This document can only be reproduced in full or in part by quoting the authors and the source: "TRIPOD Project, FP7-EC Grant Agreement Number: 265809".
Application of CFD tools to the development of a novel propulsion concept

Summary

This deliverable summarizes how CFD tools have been used during the project in order to facilitate the design of the novel propulsion system. The computations were made for the Gudrun Maersk ship. They included both the retrofit and new building scenario.

As a result of a feasibility study, the pod unit was decided to be non-rotatable and with flaps for steering, which would reduce the costs of the pod installation and the energy consumption as there will be no need of machinery for rotating the pod. Therefore straight flow conditions are the main concern of the present study.

For the study of propeller-pod housing interaction, RANS code FINFLO has been adapted to include several actuator disk models rotating in arbitrary directions, which makes it easy to simulate the effect of the CRP propellers in the surrounding flow. A numerical approach relying on correction factors has been developed to make accurate the estimation of effective wakes also in the events where the tangential induced velocities are relevant as it is the case for CRP units. The approach aims to cancel the numerical error due to the coupling of a potential flow method with a RANS solver. The model takes into account the accelerations induced by the propeller in the flow via body forces. The propeller loading is calculated in an interactive way from the propeller geometry using a lifting line off-design model with an optional pitch-reduction feature for the tip vortices shed from the blades.

The computational approach allows splitting the complex flow due to the interaction of the main propeller, pod propeller and pod housing in basic problems where an effective inflow is obtained for each propeller. The effective inflow obtained in this way can be used for propeller analysis and design.
Contents

1 Introduction 4
2 Numerical Method 5
3 Pod Housing Analysis 5
4 Effective Wake 8
5 CLT Optimization 11
6 Conclusions 14
1 INTRODUCTION

This deliverable describes how CFD tools have been used during the project in order to facilitate the design of the novel propulsion system. The development of design for the different components of the propulsion system developed within TRIPOD (pod housing optimization, CLT propeller optimization, CRP optimization) benefits from the use of such advanced numerical tools.

The investigation was made for an existing container ship of 351,081 meter length between perpendiculars, driven by a six bladed conventional propeller, with a displacement of 120000 tons. Two scenarios were studied a retrofit and new building case. Two configurations were analyzed in the former case:

- a CLT propeller replacing the original propeller and
- a pod propeller working behind the original propeller as the rear propeller of a CRP unit. The original propeller is working somewhat unloaded (80% of the total power) relative to the existing single propeller configuration. For the pod propeller two alternatives are analyzed: a conventional and a CLT propeller.

In the new building scenario the hull is modified at the stern and CRP units of either conventional or CLT propellers are analyzed as alternative propulsion systems. The fore propeller is connected to the main engine and the aft propeller is installed on the pod as it was the case in the second configuration of the retrofit scenario.

Concerning the pod housing, as a result of a feasibility study at the beginning of the project, the pod unit was decided to be non-rotatable and with flaps for steering, which would reduce the costs of the pod installation and the energy consumption as there will be no need of machinery for rotating the pod. The first CFD computations dealt with the optimization of the pod housing.

Generally, the success of a propeller design depends on the accurate estimation of the effective wake, especially for full-form ships where large differences are expected between the nominal and the effective wakes. Furthermore, an accurate estimation of not only the axial but also the tangential effective wake may be critical in some particular applications such as CRP propellers. For CRP propellers the effective wake at the plane of each propeller depends not only on the shape of hull form but also on the effect of the other propeller and possible supporting appendages. This effect is especially important when the axial and tangential wakes are estimated at the location of the aft propeller, which is subject to the slipstream of the fore one. Therefore, effort was devoted to develop a method for improving the accuracy in the estimation of the effective wakes by CFD tools.

In a second stage, the propellers have been analyzed subject to the calculated effective wakes. Here the full geometry of the propeller has been modeled and analyzed in a quasi-steady or unsteady analysis using RANS and lifting surface methods. This analysis gives representative values of pressure magnitudes and distributions on the propeller blades.

Additional computational effort was made to optimize the CLT propellers. Systematic variations of the endplate geometry were analyzed in order to assess the impact of different shapes on propeller performance.
2 NUMERICAL METHOD

The FINFLO code was used for the assessment of propeller performance. The flow simulation in FINFLO is based on the solution of the RANS equations by a finite volume method. Unsteady flows can be approximated using mixing plane and quasi-steady methods. In the mixing-plane approach the flow quantities for both the rotating and non-rotating blocks are circumferentially averaged on both sides of the common interface and then transferred to the ghost cells as boundary values. In the quasi-steady approach the rotating and non-rotating blocks are connected without any averaging process. The simplified approaches may give sensible results when the interaction between the rotating and stationary domains is weak.

Alternatively, the use of actuator disk models allows the transformation of the unsteady problem resulting from the action of one or several propellers into an equivalent steady state one. This avoids the convection problems of the simplified approaches. Additionally, as there is no need to build the grid for the propeller blades the computational effort per geometry can be reduced, which results in a larger number of alternatives to be tested in a given span of time. This approach was selected for effective wake calculations combined with the quasi-steady approach for the treatment of the single propellers.

Unsteady vortex-lattice lifting surface methods were also used for prediction of cavitation extent of the different configurations analyzed within the project. They used as input data the effective wakes calculated with the actuator disk models. They are cheap in CPU time consumption.

Then, a possible way of handling the design problem of CRP propeller in a complex ship hull wake is to make a RANS computation using actuator disks for the representation of the propellers for the flow around the pod housing and ship hull. From this computation axial and tangential effective wakes are obtained. Next, each propeller can be analyzed independently subject to the calculated effective wakes. In this last analysis, either RANS or lifting surface methods can be used to evaluate the designs from the standpoint of efficiency and cavitation.

3 POD HOUSING ANALYSIS

RANS code FINFLO is used to simulate the flow around the pod housing. The propeller has been modeled by potential-flow actuator-disk theory. The actuator disk is coupled with the RANS solver and the flow solution is sought in an iteratively way. The estimated propeller loads are transferred from the potential to the viscous solver. Conversely, the estimated effective wake is transferred in the reverse direction to the propeller potential flow solver. The propeller loads are expressed in terms of body forces. Figure 1 shows details of the Rudderpod geometry, the location of the actuator disks and the grid shape on the surfaces of the pod housing.

The drag of the pod housing has been studied for different pod housing geometries. Figure 2 shows pressure distributions on the housing for different profiles on the strut and fin. A representative on-coming flow to the pod housing was simulated by means of two actuator disks models rotating in opposite directions, i.e. in CRP mode. The propeller loading was calculated in an interactive way from the propeller geometry using a lifting line off-design model.
Figure 1. Details of the Rudderpod showing the location of the actuator disks and the grid shape on the surfaces of the pod housing.

The CFD computations were considered accurate enough to range different types of strut profiles from the standpoint of drag. The ABB profile presented less drag when used in the strut and fin of the pod housing than other conventional NACA profiles. The ABB profile presented 14 percent less drag resistance than the other ones.

Table 3.1. Non-dimensional drag for the strut (S), fin (F) and pod (P)

<table>
<thead>
<tr>
<th></th>
<th>CX-STRUT</th>
<th>CX-FIN</th>
<th>CX-POD</th>
<th>CX-TOTAL</th>
</tr>
</thead>
<tbody>
<tr>
<td>ABB</td>
<td>0.00313</td>
<td>-0.00016</td>
<td>0.00973</td>
<td>0.01202</td>
</tr>
<tr>
<td>NACA 66</td>
<td>0.00480</td>
<td>0.00049</td>
<td>0.00911</td>
<td>0.01378</td>
</tr>
<tr>
<td>NACA 4 digits</td>
<td>0.00492</td>
<td>0.00028</td>
<td>0.00932</td>
<td>0.01389</td>
</tr>
</tbody>
</table>
Figure 2. Pressure distributions on the starboard side of Rudderpod showing the location of the actuator disks
4 EFFECTIVE WAKE

One of the key issues for an optimum propeller design is the correct estimation of the effective wake resulting from propeller-hull interaction, which in some cases may differ considerably from the nominal one. Notionally, the propeller flow at the location of the propeller consists of three components: the flow as altered by the ship hull in the absence of the propeller (nominal wake), the propeller induced velocities (induced wake) and the interaction between the two preceding flows (interaction wake). The interaction wake is the result of changes not only in boundary layer thickness (or more generally in spatial distribution of vorticity) but also in factors like wave patterns, which may be altered by the propeller suction and may in turn modify the velocity field at the propeller plane, etc. The effective wake is defined as the sum of the nominal wake and the interaction wake.

Within the CFD framework, the estimation of the effective wake at the propeller plane is usually made by combining a RANS solver for modeling the turbulent flow around the ship hull with a potential flow method for simulating the propeller action. Typically, the propeller induced velocities resulting from the potential flow solver are expressed in an actuator disk form before subtracting them from the total velocities obtained in the RANS computation. In this way the effective wake is obtained.

Additionally, potential flow methods for propeller design include simplifications concerning the shape of the propeller wake (e.g. lightly or moderately loaded wake models in propeller lifting line or actuator disk theory, inclusion or not of the hub effect, etc.). These simplifications would affect the accuracy of the effective wake predictions when such methods are used coupled to RANS solvers. In other words, the potential flow theory for a given propeller load yields induced velocities that are not exactly equal to those yielded by using a RANS solver with equivalent body forces for the same propeller load. It would then be desirable that such methods when coupled to viscous RANS solvers do not introduce errors in the estimation of effective wakes and consequently of propeller loads.

In this project, a method has been developed for the estimation of the effective wakes with an accuracy which is not much affected by the simplifications introduced in the basic potential flow propeller model. The method is based on a correction factor approach that cancels the numerical error due to the coupling of the potential flow method for the representation of the propeller with the RANS solver for the representation of the bulk flow. The work has a specially focus on the prediction of tangential effective wakes, which is important for the design of CRP propellers. For the study of propeller-pod housing interaction, the RANS solver has been adapted to include several actuator disk models rotating in arbitrary directions, which makes it easy to simulate the effect of the CRP propellers in the surrounding flow.

Figure 9 shows that the axial effective wake at the rear propeller is significantly affected by the action of the fore one. Figure 10 shows that the effect of the aft propeller on the fore one concerning the tangential effective wake is negligible. On the other hand, the fore propeller induces a strong tangential wake which is non-axisymmetric and which is affected by the shadow of the strut in the pod housing. Figure 11 shows the total velocities at the location of the fore and aft-propellers. The pressures are also shown on the Rudderpod surfaces.

Figure 12 shows the total effective wake at the location of the fore and aft-propellers. The pressures are also shown on the Rudderpod surfaces.
Figure 9. Axial effective wake at the propeller disks for the new building scenario.

Figure 10. Tangential effective wake at the propeller disks for the new building scenario.
Figure 11. Total velocities at the propeller disks. The colors represent pressures on the pod surfaces and velocities on the propeller disks. New building scenario.

Figure 12. Effective wake for the total velocity at the propeller disks. The colors represent pressures on the pod surfaces and velocities on the propeller disks. New building scenario.
5 CLT OPTIMIZATION

RANS computations for the optimization of endplate propellers were conducted. Systematic variations of the endplate geometry were made in order to assess the impact of different shapes on propeller performance. Several types of modifications including variations in plate contraction angle, in plate swept and flap angle were studied. Efficiency improvement is the main concern of this study. A special procedure for the generation of the computational grids is implemented in order to minimize computational errors in the comparison of the alternative geometries. Comparisons were made at full scale. Some scale effects on fully turbulent flow were also quantified.

The CFD computations were considered accurate enough to range the performance of different endplate propellers. Model tests were not considered necessary for this study.

Variations in efficiency around 3 percent were found among the different endplate versions. For variations in plate contraction angle, shapes presenting a moderate load at the plate with absence of tip vortex at the outer endplate edge yield better efficiency. Endplate flap angles of large size affect slightly the efficiency but reduce significantly the thrust coefficient. Modifications of the endplate geometry involving a forward swept benefit efficiency and modifications with backward swept result in strong tip vortex shed at the outer edge of the endplate. Removing a moderate part of the trailing edge of the endplate produces a beneficial effect on efficiency.

Figure 3. Comparison of pressure distributions on the suction side of the blades for various endplate modifications.
Figure 4. Comparison of pressure distributions on the pressure side of the blades for various endplate modifications.

Figure 5 shows the influence of the grid size on the efficiency prediction for the geometry variations on the same propeller. The light bars represent computations on the coarse grid, and the dark bars, computations on the fine grid. Each number in the abscissa represents a variation of endplate shape.

Figure 5. Comparison of efficiency predictions for 12 endplate versions using the coarse (white) and fine (black) grids.

Available open water tests were used to validate CFD computations. Computations were made for a representative advance number at full scale; therefore comparisons with tests are made with full scale predictions. For example for propeller CLT3, the computed thrust and torque coefficients compare well with the full scale predictions from model tests made by CEHIPAR, as shown in the figure below. In the figure the suffix “m” means model scale, and
“s” ship values. The computed values are represented by isolated circles and the results from model tests by lines connecting experimental points. Comparable results are those with suffix “s”.

Calculations have been made also using propeller lifting surface code UPCA91, which is based on potential flow theory with a cavitation model. These calculations used the effective wake obtained in the RANS computations with actuator disks representing the CRP unit and they incorporate the effect of the hydrostatic pressure. The effective wakes included the interaction between the propellers. A comparison of model test and computational results is made in Figures 7 and 8.

Figure 6. Comparison of computed and model test results for propeller CLT3

Figure 7. CLT4 back cavitation. Full load condition.
6 CONCLUSIONS

This deliverable summarizes how CFD tools have been used during the project in order to facilitate the design of the novel propulsion system. The computations were made for the Gudrun Maersk ship. They included both the retrofit and new building scenario.

As a result of a feasibility study, the pod unit was decided to be non-rotatable and with flaps for steering, which would reduce the costs of the pod installation and the energy consumption as there will be no need of machinery for rotating the pod. Therefore straight flow conditions are the main concern of the present study.

For the study of propeller-pod housing interaction, RANS code FINFLO has been adapted to include several actuator disk models rotating in arbitrary directions, which makes it easy to simulate the effect of the CRP propellers in the surrounding flow. A numerical approach relying on correction factors has been developed to make accurate the estimation of effective wakes also in the events where the tangential induced velocities are relevant as it is the case for CRP units. The approach aims to cancel the numerical error due to the coupling of a potential flow method with a RANS solver. The model takes into account the accelerations induced by the propeller in the flow via body forces. The propeller loading is calculated in an interactive way from the propeller geometry using a lifting line off-design model with an optional pitch-reduction feature for the tip vortices shed from the blades.

Figure 8. CLT4 back cavitation when CLT2 works in front of it. Propeller unsteady lifting surface vortex-lattice calculation (UPCA91).
The computational approach allows splitting the complex flow due to the interaction of the main propeller, pod propeller and pod housing in basic problems where an effective inflow is obtained for each propeller. The effective inflow obtained in this way can be used for propeller analysis and design.