CFD based investigation of potential power saving for different rudder types, positions and pre-swirl fins

Claus D. Simonsen, Casper Nielsen, Christian Klimt-Møllenbach, Christian Råe Holm and Anton Minchev
FORCE Technology

Introduction

FORCE Technology and Grontmij | Carl Bro in Denmark have been involved in a joint project under DCMT (Danish Centre for Maritime Technology). Based on promising CFD results obtained in [1], [2] and [3] plus indications that the rudder size or type and rudder leading edge-propeller distance could influence the propulsive power, the goal of this project was to investigate the fuel saving effect of different rudders and their longitudinal position behind the ship. Further, the intention was also to study pre-swirl stators located in front of the propeller. The investigation was conducted as a combination of CFD and towing tank testing to enable validation of the CFD computations. Focus was on a complete CFD model for hull, propeller and rudder, which can account for the mutual interaction between all three components when the flow field is calculated.

In the present work the propeller is the same, while three different rudder types (horn, spade and flap) were evaluated behind a 180m bulk carrier in order to study how the different rudders influence the propulsive performance for a given speed at the ships self-propulsion point. Further, for two of the rudders (horn and spade) a study of the longitudinal position was made. It should be noted that by introducing different rudder types with different sizes the lift characteristics of the rudders will be different. This can potentially influence the maneuverability of the ship, since the rudders produces different steering forces for a given rudder angle. The ship was originally fitted with a horn rudder, so the idea was to use a new spade and flap rudder that should produce same steering force to maintain the maneuverability. So to check the designs, an additional study was made, where the rudders were deflected to a specified rudder angle and the steering force was determined. This check was made both with CFD and in the towing tank. It should be noted, that the maneuvering aspect was not originally included in the project, but it was judged to be important. Therefore more focus was placed on this and less on the pre-swirl stators.

All experimental data was measured in FORCE Technology’s towing tank and all meshing and flow simulation were conducted with Star-CCM+ provided by CD-adapco.

Test case and CFD method

The study is carried out for the 180m Diamond 34 Bulk Carrier from Grontmij | Carl Bro, Figure 1. For the rudder position study, two rudders are considered. One is the original horn rudder (ORIG) shown in Figure 2. The other is a spade rudder based on a NACA20 section (NACA), Figure 3. Three rudder-propeller distances are considered. If C is the distance between leading edge of the original rudder and the propeller centre and D is the propeller diameter, the three positions are defined as: C=45.5%D (Base), C=37.1%D (Pos. 1) and C=28.6%D (Pos. 2). See Figure 2. For the NACA rudder the distances are C=44.1%D (Base), C=35.7%D (Pos. 1) and C=27.2%D (Pos. 2). The distances occur after moving the rudder 20mm and 40mm forward relative to base positions in model scale.

For the steering force study, three rudders are considered. Two of them are the ones described above, while the last one is a flap rudder (FLAP) from Becker Marine Systems. See Figure 4 and Figure 5. Each of the three rudders is located in the base position, i.e. with the rudder stock in the as built position. Concerning the rudder angle each rudder is investigated in ± 20 degrees.

Finally, the stator fin study is conducted for the ship with original rudder in base position. For this configuration a set of stator fins was designed and computations were made for a couple of fin angles. See Figure 6 and Figure 7.

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>L&lt;sub&gt;pp&lt;/sub&gt;</td>
<td>[m]</td>
</tr>
<tr>
<td>B</td>
<td>[m]</td>
</tr>
<tr>
<td>T</td>
<td>[m]</td>
</tr>
<tr>
<td>S</td>
<td>[m&lt;sup&gt;2&lt;/sup&gt;]</td>
</tr>
<tr>
<td>C&lt;sub&gt;B&lt;/sub&gt;</td>
<td>[-]</td>
</tr>
<tr>
<td>D&lt;sub&gt;P&lt;/sub&gt;</td>
<td>[m]</td>
</tr>
<tr>
<td>Z</td>
<td>[no. of blades]</td>
</tr>
<tr>
<td>P/D&lt;sub&gt;0.7&lt;/sub&gt;</td>
<td>[-]</td>
</tr>
</tbody>
</table>

Table 1: Hull and propeller data.

The computations are done in model scale using a scale of 1:23.73. The main particulars and propeller data are shown in Table 1. Finally, one model speed condition
was chosen for all configurations namely 1.479 m/s (14 knots full scale) corresponding to a Froude number of 0.173.

Figure 1: 180m bulk carrier.

Figure 2: Original rudder. Upper: (Base case, C=45.5%D). Middle: (C=37.1%D). Lower: (C=28.6%D)

Figure 3: NACA rudder. (Base case, C=45.5%D)

Figure 4: Flap rudder in base position.

Figure 5: Flap rudder in base position.

Figure 6: Stator fin configuration, port.

Figure 7: Stator fin configuration, starboard.

The computations are performed with the Reynolds Averaged Navier-Stokes (RANS) solver StarCCM+ from CD-adapco. The code solves the RANS and continuity equations on integral form on a unstructured mesh by means of the finite volume technique. Transient calculations are considered using a second order difference scheme. Spatial discretization is performed with second order schemes for both convective and viscous terms. The pressure and the
velocities are coupled by means of the SIMPLE method. Closure of the Reynolds stress problem is achieved by means of the isotropic blended $k-e/k-\omega$ SST turbulence model with an all $Y+$ wall treatment. For the propeller rotating behind the ship, a rigid body approach is applied, i.e. the propeller geometry is rotated throughout the solution. The free surface is modeled with the two phase volume of fluid technique (VOF). Further details about the code can be found in [4].

Influence of rudder-propeller distance on propulsive power

For simulation of the self-propelled ship, both hull and rudder are modeled with a trimmed mesh, which is a hexa-dominant polyhedral mesh. Compared to the meshes used in [1] and [2], the present grid is the same in the bow region, but has been refined in the stern region; see Figure 8 and Figure 9.

The propeller is modeled with typical polyhedral cells. This type of mesh is assumed to give a better resolution of the geometry due to the shape of the polyhedral cells, particularly in the blade edge regions. Further, refinement zones are applied along the edges in order to resolve the steep gradients in these areas.

Figure 8: Mesh in bow region.

Figure 9: Mesh in stern region.

Figure 10 shows the mesh on the propeller. In order to combine the hull and propeller meshes, a cylindrical region with the propeller is embedded in the hull mesh and the two are connected via an interface which is updated as the propeller rotates. The final combined mesh is shown in Figure 11. Finally, it should be noted that the near wall spacing of the grids on no-slip surfaces are in the range from $y+\approx 1$ to $y+\approx 30$.

In the present simulations, the self-propulsion point from the model test has been applied in the CFD simulation, i.e. the propeller revolutions from the model test is applied in the CFD model. Table 2 and Table 3 show the applied propeller settings.

Figure 10: Propeller Mesh.

Figure 11: Combined hull propeller mesh.

Focus in the present report is on the integral quantities, thrust, torque, power and rudder side force, the flow field quantities will only be discussed briefly to give an example on the output of simulations. In the figures below, the pressure distribution is shown for suction and pressure sides of the propeller in the configuration with the NACA rudder in the base position. The general flow features for the combined hull-propeller model are similar to the ones presented and discussed in [1] and [2]. A comparison between the features found in [1] and [2] and the present results shows that the NACA rudder has not changed the overall picture of the flow field. This means that it can still be seen how the propeller is working harder (higher negative dynamic pressure on suction side) in areas where the surrounding fluid has to be accelerated the most, namely where the propeller blade passes the uppermost low speed region around the 12 o'clock position, Figure 12. The closer one gets the hull surface, the slower the fluid velocity will be, due to the presence of the hull boundary layer and the “shadow” from the skeg.

On the pressure side, Figure 13, the same effect is noticed – higher pressure as the propeller blade rotates.
through the low velocity zone in the upper part of the wake. Finally, the load on the blade also reflects the cross flow direction in the wake, i.e. the load reflects whether the blade moves downwards or upwards through the wake.

![Figure 12: Starboard side, suction side. NACA rudder.](image)

![Figure 13: Port side, pressure side. NACA rudder.](image)

Table 2: Measured and computed data for original rudder.

<table>
<thead>
<tr>
<th>Rudder</th>
<th>C/D (%)</th>
<th>RPM</th>
<th>EFD T (N)</th>
<th>Q (Nm)</th>
<th>CFD T (N)</th>
<th>Q (Nm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>ORIG</td>
<td>45.5</td>
<td>565.4</td>
<td>42.78</td>
<td>1.333</td>
<td>45.63</td>
<td>1.348</td>
</tr>
<tr>
<td>ORIG</td>
<td>37.1</td>
<td>559.2</td>
<td>42.22</td>
<td>1.318</td>
<td>44.87</td>
<td>1.319</td>
</tr>
<tr>
<td>ORIG</td>
<td>28.6</td>
<td>559.3</td>
<td>42.97</td>
<td>1.331</td>
<td>45.89</td>
<td>1.339</td>
</tr>
</tbody>
</table>

Table 3: Measured and computed data for NACA rudder.

<table>
<thead>
<tr>
<th>Rudder</th>
<th>C/D (%)</th>
<th>RPM</th>
<th>EFD T (N)</th>
<th>Q (Nm)</th>
<th>CFD T (N)</th>
<th>Q (Nm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>NACA</td>
<td>44.1</td>
<td>563.4</td>
<td>41.66</td>
<td>1.330</td>
<td>44.80</td>
<td>1.327</td>
</tr>
<tr>
<td>NACA</td>
<td>35.7</td>
<td>560.7</td>
<td>41.75</td>
<td>1.323</td>
<td>44.75</td>
<td>1.320</td>
</tr>
<tr>
<td>NACA</td>
<td>27.2</td>
<td>557.3</td>
<td>41.95</td>
<td>1.319</td>
<td>44.85</td>
<td>1.314</td>
</tr>
</tbody>
</table>

Table 2 and Table 3 show the measured (EFD) and computed (CFD) thrust and torque values. For both rudders fair agreement between measurement and CFD is found. Thrust values are slightly over predicted, while the torque values compares quite well.

Figure 14 shows measured and computed model scale shaft power. The power is computed based on the propeller torque, Q and revolutions per second n as

\[ P = 2\pi n Q \]

From the figure it is seen that the CFD results show same trend as the measured data. On absolute level quite good agreement is also obtained, since the power is computed within 1% of the measured power, Figure 15.

![Figure 14: Computed and measured power for original and NACA rudders.](image)

![Figure 15: Deviation between measured and computed power for original and NACA rudders.](image)

From Figure 14 it is seen that the two rudders show different trends in the power as the rudder is moved closer to propeller. The power obtained with original rudder decreases with reduced distance, but there seems to be an optimum position at the middle position, Pos. 1, after which the power increases again.
Figure 16 shows the saving relative to base position for the original rudder. The model test indicates a maximum saving of 2.3% while CFD predicts 3.2%.

The power obtained with the NACA rudder also decreases with reduced distance, but in opposition to the original rudder there is no local minimum, so maximum saving occurs at position closest to the propeller, Pos. 2. Figure 17 shows the saving relative to base position for the NACA rudder. The model test indicates a maximum saving of 1.9% while CFD predicts 2.1%.

If the performance of the two rudders is compared, the NACA rudder seems to reduce the power, except for the middle position, where the original rudder gives the best results.

**Influence of the rudder type on propulsive power and steering force**

The idea is to investigate how the rudder type and indirectly the size influences the propulsive power at zero rudder angle and to check if the smaller rudders are able to deliver the required steering force, i.e. the steering force of the original rudder at a given rudder angle, which in this case is ±20 degrees.

Concerning the propulsive power at zero degree rudder angle the rudder position study indicates that the thinner NACA rudder requires less propulsive power than the original rudder. However, the position study is conducted with “ideal rudders”, which means that all fixed and movable parts of the rudders are combined to one surface, i.e. there are no gaps between parts. However, the rudder angle study is conducted with “real” rudder geometries at the base position, i.e. with rudders with movable parts and gaps between parts. Table 4 shows the self-propulsion data from the model test with movable rudders. It is seen that the differences in power between ORIG and NACA rudders that was observed in the position study now disappears so that the two rudders basically gives the same power. The same trend is observed when the FLAP rudder is tested. So, for rudders with movable parts it seems that the rudder type is less important for the propulsive performance compared to the distance between rudder and propeller, at least at the base position and for the considered rudder types. It should be noted, that this study was not repeated with CFD calculations.

<table>
<thead>
<tr>
<th>Rudder</th>
<th>C/D (%)</th>
<th>RPM</th>
<th>T (N)</th>
<th>Q (Nm)</th>
<th>P (W)</th>
</tr>
</thead>
<tbody>
<tr>
<td>ORIG</td>
<td>45.5</td>
<td>564.8</td>
<td>42.42</td>
<td>1.352</td>
<td>80.0</td>
</tr>
<tr>
<td>NACA</td>
<td>44.1</td>
<td>563.7</td>
<td>42.27</td>
<td>1.355</td>
<td>80.0</td>
</tr>
<tr>
<td>FLAP</td>
<td>38.4</td>
<td>565.6</td>
<td>42.38</td>
<td>1.348</td>
<td>79.9</td>
</tr>
</tbody>
</table>

Table 4: Measured data for movable, ORIG, NACA and FLAP rudders.

As shown above there was not much difference in propulsive power with the three different rudder types, but it is still interesting to check the steering force to get an idea about how different rudder types performs compared to each other. In this case both measurements and CFD computations are performed. The model test covered seven different rudder angles, namely -30, -20, -10, 0, 10, 20 and 30 degrees. Concerning the propeller revolutions, they were kept constant during the model test for all rudder angles for each rudder. The applied RPM’s were taken from the self-propulsion point at zero degrees rudder angle. Table 4 shows the propeller settings for each rudder.

Concerning the CFD simulations, the same CFD model was used as in the position study. Only difference is that the rudder geometry was changed to include the movable parts of the rudder. The simulations were conducted as replicates of the model test using the propeller revolution from Table 4. Only rudder angles of ±20 degrees were computed.

Figure 18: Computed and measured side force on rudder for positive rudder angles.

The results of the study are presented in Figure 18, which shows the steering force for positive rudder
angles. The measured data shows that up to 20 degrees rudder angle, the flap rudder seems to be most efficient giving a higher force than the original rudder. The NACA rudder on the other hand does not seem to be as efficient as the original rudder, since it for all rudder angles give smaller steering force. At 30 degrees angle, the flap rudder side force seems to drop significantly to a level below the two other rudders. The reason is most likely that the flap, which has a higher angle of attack than the rudder due to the flap angle may be stalling in this situation.

For negative rudder angle, the rudder performance is somewhat different as shown in Figure 19. For the NACA and ORIG rudders the behaviour is the same as for positive rudder angles, i.e. the ORIG rudder performs better than the NACA rudder. Concerning the flap rudder, it performs better than the other rudders for 10 degrees. At higher angles, problems were encountered and the measurement failed, so these points are not presented.

Finally, if the rudder side forces for ± rudder angles are compared, the sizes of the force for a given ± angle are different. The reason for this is to be found in the propeller swirl downstream of the propeller. The rudder is located right in the propeller race, where the flow is rotating. This will give different inflow fields to the rudder depending of which way it is deflected.

![Computed and measured side force on rudder for negative rudder angles.](image)

<table>
<thead>
<tr>
<th>Angle</th>
<th>Flap</th>
<th>NACA</th>
<th>ORIG</th>
</tr>
</thead>
<tbody>
<tr>
<td>deg.</td>
<td>diff % D</td>
<td>diff % D</td>
<td>diff % D</td>
</tr>
<tr>
<td>-20</td>
<td>-</td>
<td>2.4</td>
<td>4.7</td>
</tr>
<tr>
<td>20</td>
<td>-5.5</td>
<td>-5.7</td>
<td>-0.3</td>
</tr>
</tbody>
</table>

Table 5: Difference between computed and measured steering forces in % of measured value D.

Figure 18 and Figure 19 also shows the computed rudder side forces. For positive rudder angles, the computed forces follow the same trend as the measured data, i.e. the flap rudder gives higher force than the ORIG rudder and the NACA rudder gives lower force than the ORIG rudder. The difference in force between flap and ORIG rudders seems to be smaller in CFD than EFD. For negative rudder angles, CFD shows same trend as for positive rudder angles, i.e. side force largest for flap rudder and smallest from for NACA rudder. This is different from the measurement, which shows that the flap rudder produces the lowest force. Based on the comparison between measurement and computation in the Table 5 it generally seems that the computations agree fairly well with the measurement except for the condition with -20 degrees rudder angle with the flap rudder. It cannot be said for sure, but the reason for this behaviour could be a combination of the rotational direction of the flow in the propeller slipstream and the flap, which the CFD code does not capture or a problem in the measurement.

**Influence of pre-swirl fins on propulsive power**

The last study that is conducted within the project covers a study of the influence of pre-swirl stators in front of the propeller. This study is purely based on CFD, so no experimental data is available for validation of the obtained results.

The idea with a pre-swirl stator is to change the cross flow characteristics of the wake field and hereby change the propeller inflow and make the propeller work more efficiently. The transverse flow in the nominal propeller inflow field is typically dominated by flow coming from below as sketched in Figure 20. When the propeller rotates in the flow field, the blades will see a different flow field depending on which side it is located. With a right hand rotating the blade on port side will move in the same direction as the cross flow while it will move in opposite direction on starboard side. Consequently, the blade on starboard side will experience a large effective angle of attack than the blade on port side, which again will lead to higher blade load on starboard side compared to port side. Based on this behavior the idea with the pre-swirl fins located upstream of the propeller is to reduce the upcoming flow on port side by deflecting the flow in downward direction and increasing the upcoming flow on starboard side by deflecting the flow in upwards direction. Figure 6 and Figure 7 show the applied fin configuration, which was designed by FORCE. As seen from the figures three fins are mounted on port side, while one is mounted on starboard side. The reason is that reducing the strong upcoming cross flow requires more effort than reinforcing the flow on starboard side. Further, each fin adds drag to the resistance, so there is no reason to use too many fins.
When the fins were mounted on the hull they were initially angled based on the flow direction in the region where they operate in order not to introduce extreme angles of attack and separation on the fins.

These simulations are run to find the self-propulsion point using a relaxation force of 21.0N. To give an example of the flow field results Figure 21 and Figure 22 show the dynamic pressure distribution on the hull, rudder, propeller and pre-swirl fins. On port side it can be seen how the fins have low pressure on the upper side and high pressure on the lower side indicating that they are lifting and deflecting the water downwards toward the propeller. Opposite behavior is observed on starboard side. So the fins have the desired effect. It should be noted that the fins can be fine tuned by rotating them around their longitudinal axis. In the present work, computations have initially been made with “as designed” configuration, so without tuning.

Table 6: Computed self-propulsion data for ORIG with and without pre-swirl fins.

<table>
<thead>
<tr>
<th>Configuration</th>
<th>RPM (rpm)</th>
<th>Q (Nm)</th>
<th>P (W)</th>
</tr>
</thead>
<tbody>
<tr>
<td>ORIG w/o fin</td>
<td>559.5</td>
<td>1.315</td>
<td>77.02</td>
</tr>
<tr>
<td>ORIG w fin</td>
<td>550.4</td>
<td>1.313</td>
<td>75.66</td>
</tr>
</tbody>
</table>

Table 6 shows the result of the self-propulsion simulation with and without pre-swirl fins. It is found that the power is reduced with 1.8% after introducing the pre-swirl fins, so there seems to be a potential saving by using the fins. It should be noted that no structural considerations were made in connection with the design of the fins. Finally, it should also be noted the data for ORIG rudder without fins are not comparable to results in Table 2, since the relaxation force and the propeller position are different. However, this should not influence the power saving effect.

Concluding remarks

In the present work RANS CFD has been used to perform a numerical study in model scale for a bulk carrier equipped with different rudder-propeller configurations represented by different distances between rudder and propeller, different rudder types and a pre-swirl stator fin configuration. Except for the rudder region, the hull grids are basically similar for all simulations. The propeller grids have been generated in a systematic way, so refinement zones, topology and mesh type are the same for all simulations.

Focus in the present work is not on the flow field, since this was covered in previous work. However, a brief check of the flow field shows how the load on the propeller blade is influenced by the wake and evidently high pressure gradients are observed at both the leading and trailing edges. It can clearly be seen that the rotating propeller gives an asymmetric load distribution on the rudder. So the present model gives results in line with previous findings.
Concerning the study of the influence of rudder-propeller clearance, it is found that moving the rudder closer to the propeller will reduce the required power. Depending on rudder type, it seems possible to save around 2% power when moving the rudder forwards compared to the base position. In spite of a slightly over prediction, CFD seems to be able to capture the saving fairly well, indicating that the tool has a potential for evaluation of this type fuel saving problem.

Regarding the idea of using smaller and more efficient rudders in order to reduce the required power does not seem to have the desired effect. Based on measurements it seems that all three rudders give the same power at the considered rudder stock position. With respect to the rudder side force the NACA rudder generally seems to give less steering force compared to the original rudder, while the flap rudder performs better for rudder angles in the range from -10 to 20 degrees. Outside this range it performs worse. When simulating the ±20 degree rudder case for all rudders with CFD the side force is predicted within 6% of the measured values except for one angle with the flap rudder. Also the simulation ranks the side force in agreement with the measured trend.

Finally, with respect to applying pre-swirl fins as fuel saving device, a set of fins were designed by FORCE and evaluated by means of CFD based self-propulsion predictions. Similar simulations were made for the ship without the fins. A comparison of the required power obtained with and without fins, shows that the fins give a saving of 1.8%. The fins have not been fine tuned and no validation against experimental data has been made, but the results look promising.

Taking the complexity of the flow problem and the uncertainties of both simulation and measurement into account the results generally looks good and indicates the RANS CFD may be useful for evaluation of fuel saving devices.

Future activities

All the above simulations for the rudder position study and the rudder angle study are conducted with propeller setting taken from the model test. It would be interesting to investigate the results, if the numerical self-propulsion points were determined and used in the assessment of the propulsive power. Further, no model test results are available for comparison between computations and measurements for the pre-swirl stator fin study. It would be interesting make a more formal validation of these computations based on model test results.

Acknowledgements

This research is sponsored by The Danish Maritime Foundation through DCMT.

References

[1] Simonsen C. D. and Nielsen, C., K
“CFD based estimation of potential power saving for ships with fuel saving rudder-propeller device”
Final report for DCMT fuel saving project I December 2011, www.dcmt.dk

“Using CFD for Simulation of Ships with Different Fuel Saving Rudder-Propeller Devices”
Numerical Towing Tank Symposium, Duisburg, Germany, 10-12 October 2010
