Third International Symposium on Marine Propulsors
smp'13, Launceston, Tasmania, Australia, May 2013

The Becker Mewis Duct® - Challenges in Full-Scale Design and new Developments for Fast Ships

Thomas Guiard[1], Steven Leonard[1], Friedrich Mewis[2]

[1] ibmv Maritime Innovationsgesellschaft (IBMV), Rostock, Germany
[2]Mewis Ship Hydrodynamics (MSH), Dresden, Germany

ABSTRACT

The need for energy efficiency in ship design and operation has been continuously increasing over the last few years. Consequently there is a growing interest in energy-saving devices (ESDs) that aim to improve ship propulsive efficiency. The Becker Mewis Duct® (MD) is such an ESD. The main effect of the Becker Mewis Duct® is the reduction of energy losses of the propeller.

Since its launch in 2008, the Becker Mewis Duct® power saving device has experienced extraordinary success. To date over 100 have been delivered, with about 300 on order. Model tests for the Becker Mewis Duct® have shown an average power saving of 6.3%. The design of the Becker Mewis Duct® is largely based on CFD-methods with model tests remaining a core element of the overall process. The scaling of the model test results and the design of the Becker Mewis Duct® for full scale operation are sometimes subject to questions or discussion. A part of the paper will deal with the approach used in the design of the Becker Mewis Duct® to handle this problem. It will also describe special difficulties encountered when using CFD for the prediction of full-scale stern flows for slow full-block vessels. The intention is to raise questions and to increase the interest in this subject to enhance potential ways forward.

As a result of customer demand, during the last two years the MD concept has been extended for application to faster vessels with lower block coefficients, such as container ships and reefer vessels. This has resulted in a new product, called the Becker Twisted Fins® (BTF). Conceptually the BTF is similar to the MD, consisting of a radial series of pre-swirl fins encased by a heavily optimised pre-duct. In addition there is a series of radial outer pre-swirl fins fitted with winglet type end plates, and all fins are twisted for optimal pre-swirl generation.

Faster hullforms present special challenges for the development of power-saving devices. The paper will describe some of these challenges, the extensive use of computational fluid dynamics and present results of initial propulsion and cavitation model tests.

Keywords

Energy-saving devices, Becker Mewis Duct®, full-scale wake, roughness, Becker Twisted Fins®

1 INTRODUCTION

Energy Saving Devices (ESDs) are stationary flow-directing devices positioned near the propeller. These can be positioned either ahead of the propeller fixed to the ship's hull, or behind, fixed either to the rudder or the propeller itself.

Energy Saving Devices that improve propulsion efficiency have been in use for over 100 years, for example Wagner (1929) details 25 years of experience with the contra-rotating propeller principle.

Well-known devices for reducing the wake losses are the WED (Wake Equalising Duct), see (Schneekluth 1986) and the SILD (Sumitomo Integrated Lammeren Duct) as detailed in (Sasaki and Aono 1997). These devices are based on an idea of Van Lammeren (1949).

It is clear that there exist many energy-saving devices on the market, each with extensive in-service and model testing experience. It would therefore appear impossible to develop an absolutely new solution to the problem. However, by combining two or more components of already established principles new developments are possible. This approach offers even more possibilities by targeting a combination of different types of flow losses.

The Mewis Duct® described in this paper is such a combination, which is based on two fully independent working ESD-Principles:

- The Contra-Propeller-Principle, well known for more than 100 years, see Wagner (1929) and
- The Pre-Duct Principle first published in 1949 by Van Lammeren.

2 THE MEWIS DUCT®

The design goal of the Mewis Duct® in comparison with other ESD's is to improve two fully independent loss sources, namely:

- Losses in the ship's wake via the duct.
- Rotational losses in the slipstream via the fins.

The key advantage of the Mewis Duct® is to improve four components of the propeller flow:

- Equalisation of the propeller inflow by positioning the duct ahead of the propeller. The duct axis is positioned vertically above the propeller shaft axis, with the duct diameter smaller than the propeller diameter. The duct is stabilising the fin effect as well as producing thrust.
- Reduction of rotational losses in the slipstream by integrating a pre-swirl fin system within the duct. The chord length of the fin profiles is smaller than the duct chord length, with the fins positioned towards the aft end of the duct. The duct itself acts as a type of endplate to the fins, thus increasing their effectiveness.
- An additional small improvement of the propulsion efficiency is obtained from higher loads generated at the inner radii of the propeller which leads to a reduction of the propeller hub vortex losses; this effect increases with increasing hub to propeller diameter ratio.
- The all over all possible power reduction lies between 2% and 11%, in average over 64 projects 6.3%.
- Additional the installation of the MD lead to positive effects in propeller cavitation, yaw stability and rpm-stability in seaway.

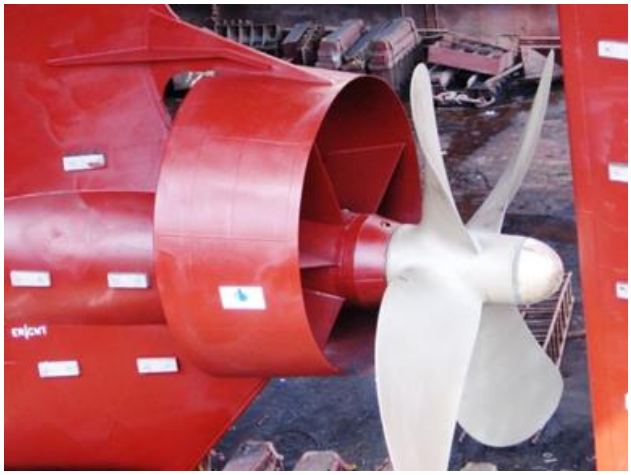

Figure 1 First Installed Full Scale Mewis Duct, STAR ISTIND, 54,000 tdw MPC, September 2009

The Mewis Duct has been on the market for 4 years and has developed into a very successful product, see Figure 2. Four key reasons are responsible for this success:

- The estimated gain is stable for different draughts and is independent of the ship's speed.
- The MD can be easily retrofitted since the resulting rpm-reduction tends to be in the region of just 1%.
- Becker Marine Systems guarantees the expected power reduction with the certification from model tests.
- The payback time is less than one year with today's oil prices.

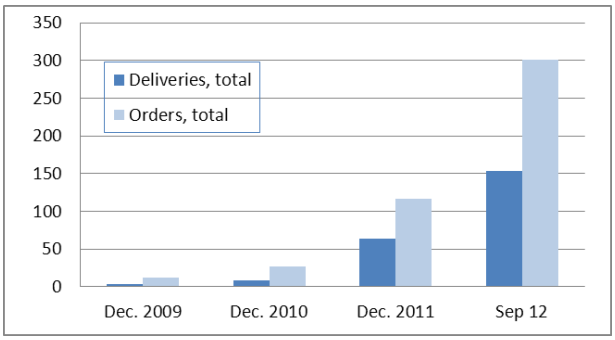

Figure 2 Mewis Duct®, Orders and Deliveries since 2009

Figure 3 shows the results of self-propulsion tests for 64 projects (as at September 2012) with and without MD fitted in 9 different towing tanks around the world plotted over the thrust loading coefficient (C_{Th}). The average power reduction is 6.3%; in design draught 5.7% and in ballast draught 7.3%.

The solid red line represents the theoretical calculated power reduction estimated in 2008 on basis of loss analysis before any model tests or CFD-calculations, but the real possibilities depend on more realistic conditions, such as the wake field of the ship, the propeller design and the quality of the Mewis Duct® design itself.

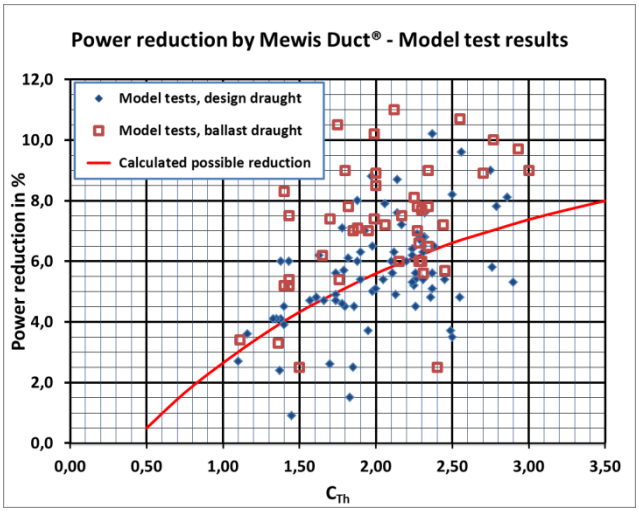

Figure 3 Power reduction by Mewis Duct®, model test results, average measured power reduction: 6.3%

3 DESIGN FOR FULL SCALE

In a typical Mewis Duct® design project a power saving guarantee is given which is based on model tests. One main reason for using model tests to check whether the guaranteed power saving is reached is the large scatter typically found in the results of standard sea trials. Possible ways to improve the accuracy by performing sea trials at precisely predefined conditions or by averaging over a large number of trials tend to result in significant additional effort and costs which are typically not accepted in a commercial context. Therefore model tests are a core element of each MD design project. Nevertheless model tests are subject to their own limitations. One of those is that results have to be scaled to full-scale values. In addition, for the optimised geometry of the tested device some scaling to the requirements of the full-scale flow may be necessary. The discussion about appropriate scaling procedures for ESDs, especially wake equalizing ducts, is probably as old as the devices themselves. In the following discussion the procedures used in the design of the Mewis Duct® to handle this problem are described and some insight is given into reasons for selecting the relevant procedures.

3.1 Procedures Used for the Full-Scale Design of the Becker Mewis Duct®

Largely based on experience, from the very beginning of the Mewis Duct® development it was assumed that the power savings observed on the full-scale ship would be very similar to those measured at model scale. To date, after a significant number of MD design projects, the experience made in sea trials and customer feedback from long-term monitoring confirms this assumption. Similar conclusions are also drawn in the work recently published by Dang et al (2011), based on model-scale experiments.

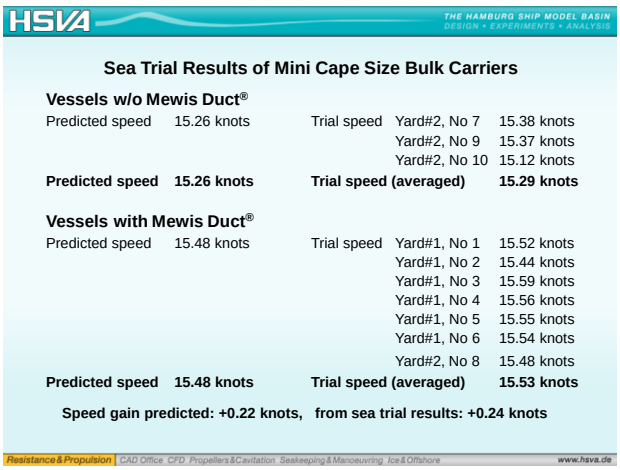

Figure 4 Sea Trial Results for 10 Sister Vessels with and Without Becker Mewis Duct®

With respect to full-scale measurements reference should be made to the sea trials for "AS Vincentia" as published for example in Mewis et al (2011) and to figure 4 which was kindly made available by HSVA. Both examples are selected because they are representing high quality data.

The results in figure 4 are showing shipyard trials for 10 sister vessels, where the possibility to average over a number of vessels, both with and without Becker Mewis Duct®, results in valuable data.

In the case of the "AS Vincentia" sea trials have been performed for the new ship without and with MD fitted within a few weeks. In both trials the sea area was the same, the weather conditions were good and the hull conditions were close to identical. Based on model tests a power saving of about 7.1% was predicted for the sea trial condition. The trial results showed a power saving of about 6.5%.

In both cases the power saving, or speed gain, predicted based on the model test correlates very well with the full-scale measurements.

With respect to the adjustment of the geometry itself to the full scale flow, the following procedure is used: After the fin pitch adjustment, which is typically performed during the model tests, the final fin setting selected is calculated at both full and model scale using CFD. In this way differences between the computed in-flow to the individual fins at the different scales can be relatively easily detected. The fin angles are then slightly adjusted according to the changes observed. This must be done with great care, keeping in mind the uncertainties associated with the CFD-based prediction of the ship's full scale wake.

3.2 Challenges in the CFD-Based Prediction of the Full-Scale Wake Field of Full Blocked Ships

The procedures chosen to handle the scaling issues in the Mewis Duct® design process might seem strange given today's powerful CFD methods. Nevertheless, especially for the main area of application, which are full-block ships such as bulk carriers and tankers, significant differences might be present between the real and the CFD-predicted full-scale wake field, as well as between different CFD-predicted full-scale wake fields computed with different modelling assumptions. It is likely that this is caused by the large influence of viscous effects including turbulence on the shape of the wake field for such ships. This is assumed to be the reason why they are especially sensitive to the modelling of turbulence and viscous effects.

Figures 9 to 14 show a selection of full-scale wake fields computed for one single ship at the same speed and draught. All of these wake fields are computed using good quality meshes and each of the viscous modelling approaches used might represent the "model of choice" for particular applications. This will be shown later when the individual wake fields are discussed while following the route from the easiest to the most feasible modelling approach.

3.2.1 Physics Models in Full-Scale CFD

Whenever a parameter is incorporated in the CFD method via a particular model it typically means that simplified equations are used which aim to represent the key effects of the physical parameter as well as possible for a

reasonable amount of effort. Such models typically have to be validated and adjusted based on experimental data. Due to the very limited availability of full-scale data for ships, such modelling approaches should be used with care when applied to the flow around full-scale vessels.

3.2.2 Wake Scaling as a Reference for Full-Scale Wake Calculations

The lack of empirical full-scale wake field data represents one of the main difficulties when judging how close a computed full-scale wake field might correlate to that of the real ship. One simple parameter that can help to get at least a rough reference based on the mean axial velocity to be expected for the full-scale wake field is the effective wake fraction. For every self-propulsion test the effective wake fraction is determined at model scale. Standard procedures are subsequently used to scale the effective wake fraction to full scale. The full-scale value is typically used by the propeller designer to adjust the propeller design to the in-flow speed it encounters on the real ship. It is important that this scaling is of a certain accuracy to avoid heavy- or light-running propellers. Therefore it can be expected that these standard procedures give relatively good results. Additionally one might also assume with reasonable accuracy that the same scaling procedures used for the effective wake fraction can also be used to scale the nominal wake fraction. In this way, independent to the model test or available empirical full-scale data, the nominal wake fraction at full scale can be easily approximated based on the model-scale wake, either calculated with CFD or measured in model tests. Although this procedure clearly contains a number of approximations, the resulting value can be an important reference point when judging the wide variations of full-scale wake fields that might result from CFD calculations for full-block ships. Equation (1), as given in Bertram (2000), is used to scale the effective wake fraction up to full scale.

$$w_s = w_m \times \frac{c_{Fs}}{c_{Fm}} + \ t + 0.04 \ \times \ 1 - \frac{c_{Fs}}{c_{Fm}} \qquad (1)$$

Where w_m is the wake fraction at model-scale, calculated with CFD, w_s is the wake fraction at full-scale, t is the thrust deduction, which is assumed to be about 0.2 and c_{Fs} and c_{Fm} are the friction coefficients for the full-scale ship and the model, according to ITTC 1957.

3.3 CFD-Model Setup

The ship used as a test case is a mid-size tanker with a length of about 180 m, a beam of 32 m and a draught of 11 m. Only one side of the ship is modelled using a symmetry plane at the mid-ship position. The free surface is approximated via a symmetry plane. The commercial CFD-Software STAR-CCM+ is used for all calculations. The flow simulations are performed by solving the Reynolds-averaged Navier-Stokes equations (RANSE) via the finite volume method. The standard mesh, used for all but two calculations, features an unstructured mesh with about 5.7 million cells for the half-ship model. The near-wall region is resolved using wall functions and 8 prismatic cell layers over most of the hull surface.

3.3.1 Modelling of Turbulence and Roughness

The turbulence models used are the *k-ω SST* model and the *RST* model. The *RST* model contains the more complex and detailed approach to model turbulence. In general it provides more accurate results. Especially for the prediction of ship wake fields at model scale the results are in general very good. Nevertheless it tends to cause instability in the solution process and converged solutions can be difficult to achieve. Therefore for most commercial applications there is no other option but to use one of the simpler turbulence models such as the *k-ω SST* model.

Additional to the effect of turbulence, the effect of hull roughness should be investigated. In the CFD software used the effect of surface roughness can be incorporated via a roughness model. This is effectively a modification to the wall-functions. A roughness value has to be specified which is defined as the equivalent sand-grain roughness height. According to the manual of STARCCM+ version 7.04.006 the model is based on the functions given in Cebeci et al (1977).

3.4 Full-Scale Wake Results for a 180 m Tanker with Different Viscous Modelling Approaches

In the following a short description is given for each of the presented wake fields, beginning with the simplest "standard-model" and increasing in complexity. The order is representing the steps made within ibmv when searching for the main reasons for the mismatch in the average axial flow speed between the extrapolation from model tests and the full scale CFD-calculations when using the "standard" settings. The captions contain the scale, the turbulence model and, for the full-scale wake fields, one additional option which contains either the mesh density variation or the CFD-setting for the hull roughness height.

3.4.1 Full-Scale Wake with "Standard" Settings

The first approach for doing full-scale wake calculations could be to use the *k-ω SST* turbulence model, which might be called standard for marine applications, in combination with a good quality mesh, such as described above under 3.3, with special attention on the near wall prism layer and reasonably good resolution. Figure 9 shows the resulting full-scale wake field calculated. The dramatic difference between the computed full-scale and the model-scale wake fields shown in figures 8 and 9 is obvious. Nevertheless, when checking the computed wake fraction in table 1 it becomes similarly obvious that the real full-scale wake is unlikely to correlate well with the computed one, since a fixed-pitch propeller, designed based on the effective wake fraction scaled from the model test measurements, would not give the correct ratio of ship speed to main engine rpm and power.

The first approach to look for possible improvements would be to investigate the effect of mesh quality on the results. Therefore one coarser and one finer mesh were generated. A refinement factor of 1.5 was used based on the average cell core distance of two adjacent cells. The

near-wall cell layer remained unchanged to avoid the influence of changed *y+* values which would lead to changes in the wall function representation. The resulting wake fields are shown in figures 5 and 6.

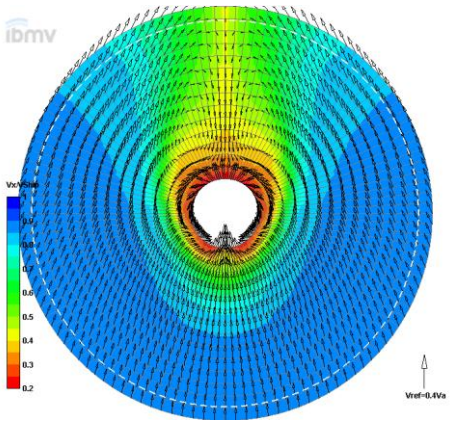

Figure 5 Full Scale, *k-ω SST*, Coarse Mesh

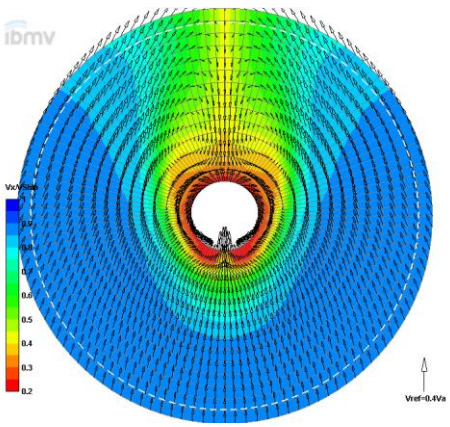

Figure 6 Full Scale, *k-ω SST*, Fine Mesh

The changes in the wake field due to the variation in mesh resolution are hardly visible. The computed nominal wake values indicate that the difference in the mean axial flow speed between the computed and the expected values is reduced with improved mesh quality. Nevertheless the improvement from the standard mesh to the finer version is just 1% of the total difference. Therefore it seems unlikely that this difference can be noticeably reduced by a further refinement of the volume mesh.

3.4.2 Full-Scale Wake with Roughness

The next step in searching for possible reasons for the high axial flow speed in the computed wake could be to consider roughness. As standard all walls in CFD are treated as hydraulically smooth. This is correct for all model-scale calculations. Nevertheless at full scale the standard paint roughness is already large enough that theoretically the surface should be treated as rough. In order to test the effect of roughness on the predicted full-scale wake field two calculations are made with the surface roughness set to 0.188 mm and 0.5 mm. A roughness value of 0.188 mm is chosen as a typical roughness value for in-service ships "(Lewis, E.V. (ed.) 1988)". The value of 0.5 mm is selected for two different reasons. One is to have some variation in the roughness value and to see, how different roughness values would influence the calculated wake field. The other is that real ships are suffering from many more surface imperfections than just the paint roughness. Some examples are welding lines, plate dents, paint defects and plate thickness differences. All these defects cause additional viscous losses. The idea is therefore to account for these additional viscous losses by simply increasing the specified roughness value in the CFD model. In this context the precise value of 0.5 mm is freely chosen with the intention to be able to see a noticeable change when compared to 0.188 mm.

Table 1 Nominal Wake Fraction Calculated with CFD and Difference to the Expected Value Based on Equation 1

	Nominal wake fraktion *w*	Difference in w_S (calculated with CFD) - w_S (scaled from model scale)
CFD Model Scale, *k- ω SST*	0,380	
CFD Model Scale, *RST*	**0,390**	
Estimated nominal wake for full scale based on *w(* calculated with model scale CFD, *RST turbulence model*) scaled according to Equation (1)	**0,316**	
CFD Full Scale, *k- ω SST*, Smooth	0,246	-22,2%
CFD Full Scale, *k- ω SST*, Coarse Mesh	0,242	-23,5%
CFD Full Scale, *k- ω SST*, Fine Mesh	0,246	-22,0%
CFD Full Scale, *k- ω SST*, Roughness = 0,188 mm	0,281	-11,2%
CFD Full Scale, *k- ω SST*, Roughness = 0,5 mm	0,318	0,5%
CFD Full Scale, *RST*, Smooth	0,257	-18,7%
CFD Full Scale, *RST*, Roughness = 0,188 mm	**0,313**	**-1,0%**
CFD Full Scale, *RST*, Roughness = 0,5 mm	0,358	13,3%

3.4.3 Full-Scale Wake with RST

Recently significant advances have been made within ibmv to achieve reliably converged full-scale wake solutions with the *RST* turbulence model. The resulting full-scale wake field is shown in figure 10. The differences observed between the *k-ω SST* and *RST* full-scale wakes are similar to those observed at model scale in figures 7 and 8. The bilge vortex is more strongly developed and located further away from the mid-ship plane. The axial velocity close to the vortex core is lower and the average axial velocity is closer to the expected value. The results seem very promising, although the difference in nominal wake fraction is only slightly reduced.

3.4.4 Full-Scale Wake with RST and Roughness

After obtaining promising results for the full-scale calculations with the *RST* turbulence model it is logical to combine the most realistic turbulence representation available with the most realistic hull surface representation. Therefore the *RST* turbulence model is combined with a rough surface representation as described above with a roughness height of 0.188 mm and 0.5 mm.

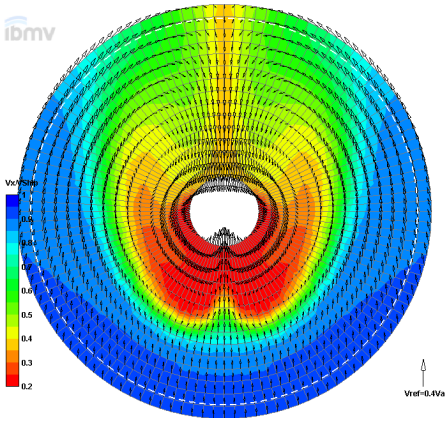

Figure 7 Model Scale, *k-ω SST*

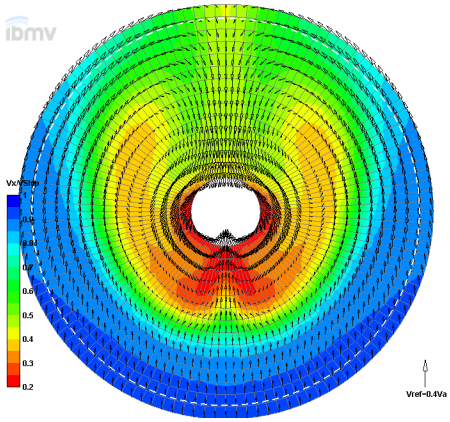

Figure 8 Model Scale, *RST*

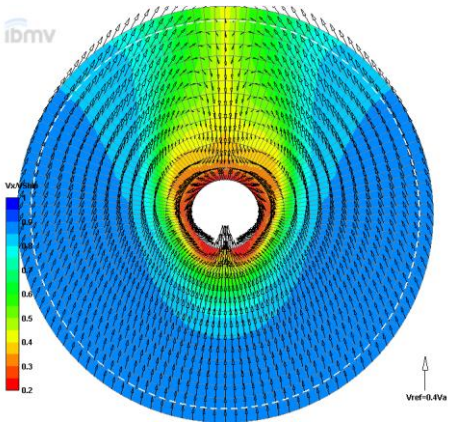

Figure 9 Full Scale, *k-ω SST*, Smooth

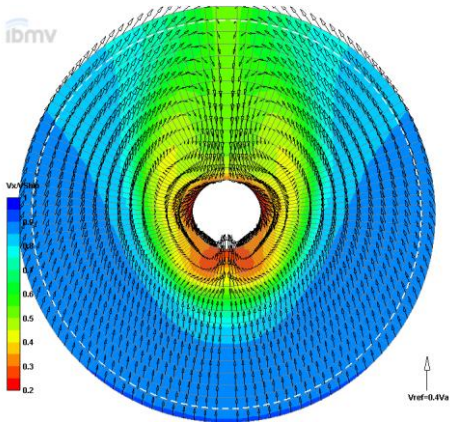

Figure 10 Full Scale, *RST*, Smooth

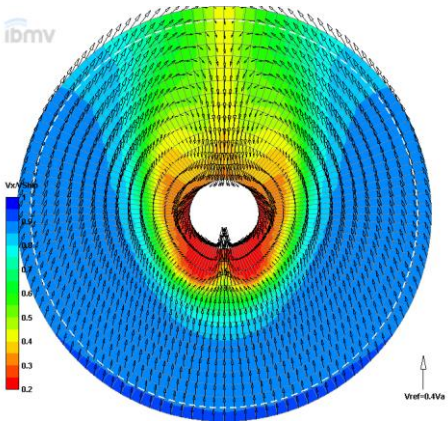

Figure 11 Full Scale, *k-ω SST*, Roughness = 0.188 mm

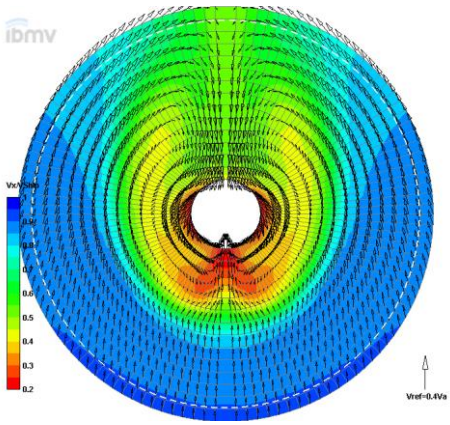

Figure 12 Full Scale, *RST*, Roughness = 0.188 mm

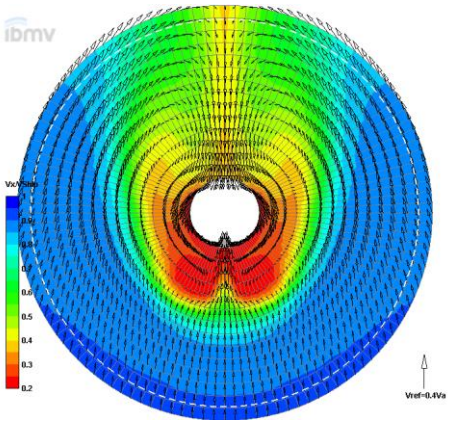

Figure 13 Full Scale, *k-ω SST*, Roughness = 0.5 mm

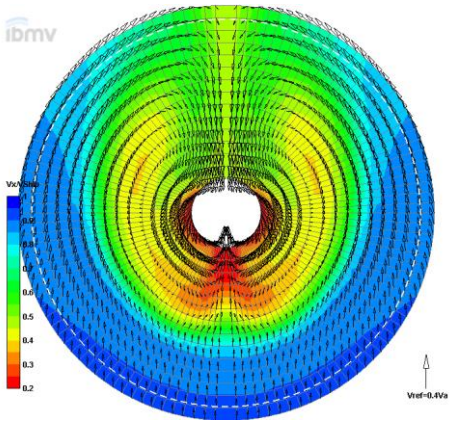

Figure 14 Full Scale, *RST*, Roughness = 0.5 mm

The results in figures 12 and 14 show significant changes to the wake fields compared with the smooth wall representation. With increasing roughness the results are becoming closer to the corresponding model-scale wake field computed with the *RST* model. As observed previously for the *k-ω SST* calculations, the axial flow speed decreases with increasing roughness. Interestingly the calculation with a roughness height of 0.188 mm, which is based on real ship data results in a nominal wake fraction which is basically identical with the expected value. In part this is a coincidence, since the remaining difference between both is much smaller than the error margins involved in the approximation of the expected value. Nevertheless, for this case it can be stated that without any additional adjustments or correlation factors the pure combination of the best turbulence model available with the most realistic value for the surface roughness of the ship hull removes the significant over-prediction of the axial velocity in the full-scale wake field.

3.5 Conclusions for the Full-Scale Design

Without having the possibility to validate the computed wake with experimental measurements in full scale it is difficult to judge which of the above discussed wake fields is the closest to the real ship wake. However, based on the modelling approaches used and the mean axial flow speed predicted, the *RST* turbulence model in combination with the surface roughness height of 0.188 mm is likely to give the most feasible results. It can be seen that the resulting full-scale wake field is much closer to the corresponding model-scale wake than the full-scale wake computed with "standard" settings. In this context the experience-based approach, in that the power saving observed at full scale is about the same as at model scale, seems much more feasible than when simply considering the wake predicted with "standard" settings. Additionally, the differences between the computed wake fields demonstrate why full-scale adjustments to the design of the Mewis Duct® are done with great care and are not purely based on the full-scale CFD-results.

Unfortunately, due to the instability of the *RST* model it cannot yet be reliably used for topologically more complex CFD simulations such as simulations with the Mewis Duct®, or any other power-saving device, propeller and rudder. These types of calculations are therefore limited to the use of simpler turbulence models. This has to be kept in mind when making design decisions based on the corresponding results. However the new results will be incorporated into the design process of the Mewis Duct® as much as possible and to date work within ibmv is progressing with the validation of the various modelling approaches by using the available full-scale measurements for the Ryuko Maru.

It should be explicitly mentioned, that the main reason for presenting these results and modelling approaches is not to present a "final" solution to the problem. Instead the intention is to highlight the problems encountered in the prediction of full-scale ship wake fields, to increase the interest in this field and to propose and enhance potential ways forward.

4 NEW DEVELOPMENTS

Since its launch the MD has had extraordinary success in both the new-build and retrofit markets. It is intended for application to slower-speed full-form vessels with propeller thrust loading coefficients generally higher than C_{Th}=1.0. This encompasses vessel types such as bulk carriers, tankers of all types and size and general purpose vessels. There therefore exists a very large potential market for a power saving device that is applicable to faster vessels with lower block coefficients and low propeller thrust loading factors ($C_{Th} < 1.0$). This encompasses primarily container vessels of all sizes, reefer ships and car carriers, both new-build and retrofit. To meet this demand a new product has been developed, the Becker Twisted Fins ®.

4.1 Design Challenges for High-Speed Vessels

Optimisation of the Mewis Duct® becomes increasingly difficult as the block coefficient of the vessel decreases:

- Such vessels have finer hull lines, which generally results in cleaner more uniform nominal wake fields. In particular the intensity of the bilge vortex is usually reduced. Figure 15 shows a comparison of the radial velocity component of the nominal wake for a typical container vessel with C_b=0.7 with that of a bulk carrier of C_b=0.84. The velocity vectors are also shown.

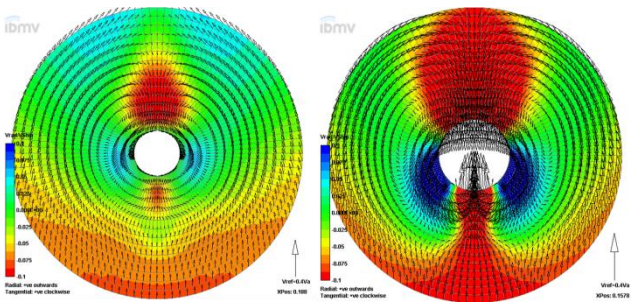

Figure 15 Radial Velocity Component Comparison for Container Ship (Left) and Bulk Carrier (Right)

- The pre-duct tends to produce drag in cleaner wake fields. Figure 16 shows the computed flow field on a plane inclined 45 degrees port side up through the propeller shaft for the same vessels.

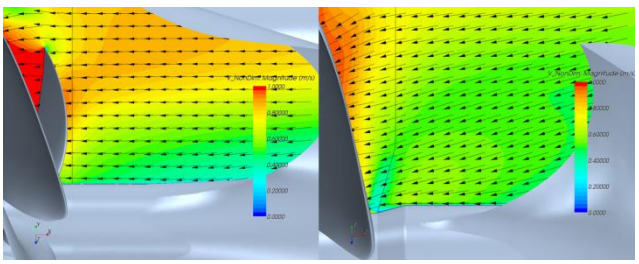

Figure 16 Effective Wake Flow Field Comparison for Container Ship (Left) and Bulk Carrier (Right)

Noticeable is that for the container vessel the velocity vectors are virtually longitudinally aligned. In order to generate thrust from a pre-duct a positive inflow angle is required. With a positive inflow angle, as is clearly present in the bulk carrier case on the right, the pre-duct section can be set to a positive pitch angle, thereby giving a small forward thrust component as the flow accelerates over the duct leading edge.

- Higher flow speeds can result in a higher risk of cavitation on the duct, especially in ballast condition. This may also have an adverse effect on the propeller.
- The duct and fins are subject to higher dynamic pressure load, which results in higher stress values and therefore more difficult structural design.
- Optimal alignment of the duct and fin sections becomes more critical since the whole system is more sensitive due to higher speeds involved.

4.2 Becker Twisted Fins®

The Becker Twisted Fins® is a logical development of the MD which incorporates alternative and new design features designed to overcome as much as possible the difficulties outlined in section 4.1. The BTF has now been under intensive development for over one year and has recently undergone successful model testing.

A typical optimised example of the BTF is shown in figure 17.

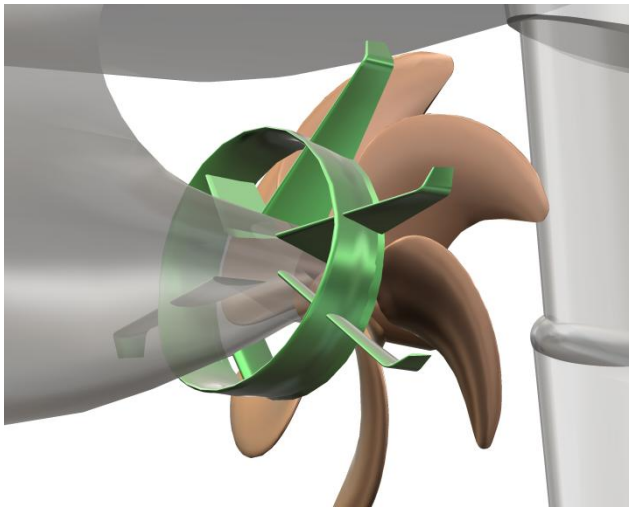

Figure 17 Becker Twisted Fins®

Since it is more difficult to generate thrust from the pre-duct an increased emphasis has been placed on the amount of generated pre-swirl. Therefore the fins have been extended beyond the pre-duct structure so that they form "inner" and "outer" pre-swirl fins.

All fins are tapered in the span-wise direction. A large chord at the fin root enables good structural integration to the hull, whereas a reduced tip chord minimises tip losses. Optional winglets or fin end plates can also be included.

All fins are twisted (in the sense of varying pitch) in the span-wise direction. This enables a more uniform hydrodynamic fin loading and controllable pre-swirl. For retrofit projects where no propeller modifications are permitted due care has to be taken to limit the degree of pre-swirl generated in order to minimise the reduction of shaft speed and consequent “heavy” propeller running.

The pre-duct is retained essentially as a structural support for the fins, however any duct drag penalties should be minimised. For low-block coefficient vessels the nominal wake above the propeller shaft tends to be predominantly axially aligned, which inevitably results in local drag. Therefore the pre-duct chord length above the propeller shaft is kept minimal, limited only by physical construction considerations. Below the propeller shaft the presence of the stern bulb usually results in an increased radial velocity component, so here the duct chord is extended in order to maximise any thrust that can be achieved. The duct pitch is also varied circumferentially to maximise efficiency.

4.2.1 First Implementation

The first project selected for development of the BTF was for retrofit to a series of ten 23-knot 7100 TEU container vessels from a German owner. The owner’s requirement was to have good power-saving performance for a reduced speed of 22 knots down to a slow-steaming condition of 17 knots in both design and ballast loading conditions. As is the case for Mewis Ducts®, all design and optimisation for the BTF was done using CFD with one set of self-propulsion model tests carried out at the end of the optimisation process. For this project over 50 design iterations were considered for CFD analysis. In addition to four sets of inner and outer pre-swirl fins two additional inner fins were incorporated to provide additional structural integrity to the ship hull.

Model tests were duly carried out at HSVA for the optimised duct design at a range of speeds and draughts. In these model tests a power saving of between 3.5% and 4.2% was measured over the required speed range. Subsequent cavitation tests were also successful, showing cavitation-free performance of the BTF for all of the required operating conditions.

The first BTF for this project is consequently due to be retrofitted at the end of 2012. Full-scale performance of the system will be carefully monitored and reported in due course.

5 Concluding Remarks

About 5 years after the first idea for the Becker Mewis Duct® the system is proving itself in service to be an efficient solution to reduce the required power for many vessels.

The Becker Mewis Duct® itself, as well as the design methods are subject to continuous development. One of the maybe most interesting topics is the scaling of the Mewis Duct® design and the model test results from

model to full scale. The paper is describing procedures used in the design of the Mewis Duct® and is highlighting difficulties encountered when trying to calculate scale effects with CFD.

Fast vessels, with speeds over 20 knots are typically not well suited for the application of the Mewis Duct®. The paper gives a small introduction into a new design concept developed to overcome this limitation the Becker Twisted Fins®. It also presents results of the first commercial application of the concept.

REFERENCES

Bertram, V. (2000). Practical Ship Hydrodynamics. Butterworth-Heinemann.

Cebeci, T., Bradshaw, P. (1977). Momentum Transfer in Boundary Layers, pp. 176-180, Hemisphere Publishing/McGraw-Hill.

Dang, J., Chen, H., Dong, G., Ploeg, A., Hallmann, R. & Mauro, F. (2011). 'An Exploratory Study on the Working Principles of Energy Saving Devices (ESDs)'. Proceedings of the Symposium on Green Ship Technology (Greenship'2011), Wuxi, China.

Lewis, E.V. (ed.) (1988). Principles of Naval Architecture Volume 2, p. 60, Second Revision, SNAME.

Mewis, F. (2009). 'A Novel Power-Saving Device for Full-Form Vessels', First International Symposium on Marine Propulsors, SMP'09, Trondheim, Norway.

Mewis, F., Guiard, T. (2011). 'Mewis Duct – New Developments, Solutions and Conclusions', Second International Symposium on Marine Propulsors, SMP'11, Hamburg, Germany.

Sasaki, N., Aono, T. (1997). 'Energy Saving Device "SILD"', Journal of Shipbuilding Vol. 45, No. 135, Japan.

Schneekluth, H. (1986). 'Wake Equalising Ducts', The Naval Architect, London, UK.

Van Lammeren, W.P.A. (1949). 'Enkele Constructies ter Verbetering van het Rendement van de Voorstuwing', Ship en Werf van 1 April 1949 No.7, Rotterdam, the Netherlands.

Wagner, R. (1929). 'Rückblick und Ausblick auf die Entwicklung des Contrapropellers', Jahrbuch der Schiffbautechnischen Gesellschaft, 30. Band, pp. 195 – 256, Berlin, Germany.