Development of a novel power-saving device for full-form vessels

Dipl.-Ing. Friedrich Mewis

Abstract

With the advent of today's dramatically increased bunker prices, the reduction of fuel consumption has become a major topic. From a propulsion point of view, one possibility lies in the use of so-called power-saving devices; these are stationary devices positioned near the propeller that improve the overall propulsion efficiency.

This paper introduces a novel approach, the Pre-Swirl Duct (PSD). This power-saving device is developed as a wake equalising duct combined with an integrated pre-swirl fin system positioned ahead of the propeller. By pre-correcting the flow into the propeller, the device essentially reduces the rotational losses in the resulting propeller slipstream.

The PSD is suited to vessels with high block coefficient and speeds lower than 20 knots. This encompasses tankers and bulk carriers of every size, together with multi-purpose carriers and feeder type container vessels. The expected power reduction is in the range of 3 to 9%, depending on the propeller loading. A beneficial by-product of the PSD is a small improvement to the ship's yaw stability.

The PSD was developed in cooperation with Becker Marine Systems, Hamburg (BMS). BMS has registered this device under the trademark »Mewis Duct«.

Introduction

Power-saving devices 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.

Power-saving devices are designed to reduce flow losses around the working propeller. The main losses around a rotating propeller consist of:

- rotational losses in the propeller slipstream
- unequal ship wake inflow to the propeller with respect to the propeller rotation
- propeller hub and tip vortex losses

Power-saving devices that improve propulsion efficiency have been in use for over 100 years, for example Wagner (1929) reports on 25 year's 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). Devices for reducing the rotation losses include the SVA fin system, Mewis and Peters (1986), the Daewoo pre swirl fin system, Lee et al (1992) and the Hyundai Thrust Fin system which is fitted to the rudder, see Hyundai (2005). A well-known solution to reducing the losses in the propeller hub vortex is the PBCF (Propeller Boss Cap Fins), Ouchi et al (1990). The Kappel propeller utilises a special tip fin integrated into the propeller blades to reduce the tip vortex losses, see Andersen et al (1992).

It is clear that there are many existing power-saving devices on the market, each with extensive in-service and model testing experience. So it would appear to be 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 flow loss types. The Pre-Swirl Duct described in this paper is such a combination.

Design idea

The design goal of the PSD is to improve two components of the propeller flow:

- Equalising the propeller inflow by positioning a 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.
- Reducing rotational losses in the slipstream by integrating a pre-swirl fin system within the duct. The chord length of the fin profiles is smaller that the duct chord length, with the fins positioned towards the aft end of the duct.

An additional small improvement of the propulsion efficiency is obtained due to higher loads at the inner radii of the propeller.

Figures (1) and (2) show a general arrangement of the PSD.

The dimensions, location, fin and duct profiles, number of fins, circumferential

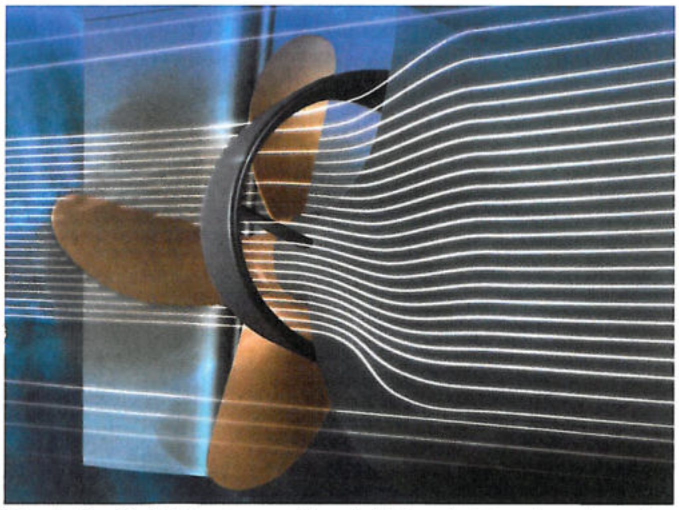

Fig. 1: Pre-Swirl Duct, propeller right-handed turning, view diagonal from ahead/starboard

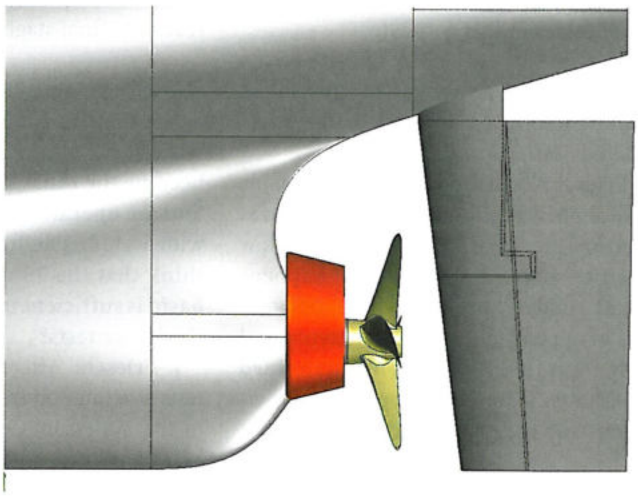

Fig. 2: Pre-Swirl Duct, right-handed propeller, side view from port

HANSA-Interview with Mr. Jürgen Friesch, Managing Director of HSVA

Dipl.-Ing. Jürgen Friesch, Managing Director of HSVA

HANSA: Today, HSVA is a service and consulting company for industrial customers worldwide. Which kind of service do you offer to your customers?

Jürgen Friesch: We are a central point for applied research in all fields related to maritime transport systems, ships and offshore structures. And this holds true for both, for open water and for icy waters. We offer advanced technologies and modern test facilities which are at the forefront in the world. Our engineers and scientists answer all questions and help to solve all problems related to ship and offshore hydrodynamics, to ship and propeller design questions and to tasks related to ice technologies. Customer orientated services have always, since 1913, been the solid foundation of our business activities.

HANSA: HSVA operates at the forefront of technology of hydrodynamics analysis. Can you give to our readers an overview about international and national research projects?

J. Friesch: We here at HSVA have a long track record of successfully run research and development projects, nationally funded but also under the umbrella of European frame work programs. More than ten percent of our annual turnover is spent on RaD activities. At the moment the focus of our research is on enhancing of our new numerical analysis tools, like the new RANS solver FRESCO. This is part of the work related to the development of the numerical towing tank. Another prime focus of our research is related to the holistic improvement of performance and efficiency of ships. A wide range of topics covering new hull forms, improved propulsor designs and questions on surface treatments are addresses in this context.

HANSA: Could you explain to our readers the most important features with respect to the cavitations phenomena?

J. Friesch: Cavitation is then unavoidable physical phenomenon that water vaporizes due to low pressures in areas where the flow velocities become high. This happens around rudders, propellers and different types of appendages. It results often – and nearly always if the design is not performed properly,- in material damages, in ship vibrations and in noise radiation.

HANSA: What would be the criteria whether to choose a three-, four- or e.g. six-blades propeller?

J. Friesch: The choice of the number of blades for a propeller is always a compromise between requirements concerning efficiency, cavitation and vibration excitation. The decision also needs to take into account the structural behavior of the ship, the data of the main engine and the whole shaft line arrangement.

HANSA: How far are – according to your experience – the figures of a computer simulation matching with the real ships' conditions?

J. Friesch: The results of simulations come more and more close to the real ship's condition, mainly for wave resistance, wake field predictions and propeller behaviour. Improvements are still necessary for manoeuvring and cavitation predictions with all related questions concerning vibrations, noise and erosion. Also the prediction of the resistance sometimes is not satisfactory. Results are still too much depending on the methodology of the grids used and the turbulence model. Also the experience of the user is strongly influencing the quality of the results.

HANSA: Do you still see an infinite development of the cavitations and modelling of vessels' hulls – or will there be a final stage one day?

J. Friesch: The development of both, ship hull forms and propellers, will never reach a final stage. The reason is, that the requirements continuously change, which means that we always have to look for new optimum solutions. Numerical and experimental tools are developing steadily and even faster and help us to come closer but never reach the final stage.

HANSA: HSVA started ice model testing in the 1930ies – could you let us know the stages of the development and what has become the today's standard? You are operating a 300 m long and 18 m wide water basin for normal tests. Do you think that the 78 m long and 10 m wide basin is sufficient to simulate a real scenario for ice tests?

J. Friesch: After performing ice model tests in winter on frozen lakes in natural ice in the 30ties, in 1958 Prof. Grim and Dr. Waas built one of the first model basins of the world and it was also during this time when HSVA's experts started to investigate how to scale the ice properties correctly. Basic tests were performed and soon revealed the need for a larger ice model basin which was build and put into operation in 1972. The basin became very busy and also the way how to generate model ice was further improved. In this basin all the investigations for the German research vessel »Polarstern« were performed and can be considered as one of the highlights of the fundamental research work on scale effects for ice model tests. But the tests also showed the limitations of a basin just 30m long and 6m wide. As a consequence in 1984 the third and large ice tank was opened, which is 78m long, 10m wide and 2.5m deep with an additional deep water section. This ice tank is one of the largest tanks in the world and now in successful operation since that time. Also the way how to build our model ice was continuously improved and calibrated via full scale measurements.

HANSA: Which type of bow shape do you favour after many years of testing and elaborating the optimum mould of ice strengthened hulls?

J. Friesch: As with ship geometries in open water, there is no single and unique bow shape. It's depending on the ice properties and on the main ice thickness you expect. To reach an excellent ice breaking behaviour it needs more than only a specific bow shape, it needs an over all optimised shape of the whole under water hull form.

HANSA: Let us now come to another item that has become very important these days. How far will it be feasible in you opinion to further improve vessels' hulls in order to maintain speed and simultaneously reduce fuel consumption?

J. Friesch: The most effective measures to save propulsion power should be taken during the design stage of a new building project:

- Carefull selection of the main dimensions, the service speed and the propulsion device. Make your new vessel as long and as slender as possible.
- Avoid strict hard point requirements, mainly in the engine room area
- Cooperate with an independent model basin and let the model basin optimise your design
- Form an effective design team of ship yard, ship owner, model basin and supplier of propeller and rudder.

If the ship is already in operation, maintain hull surface and propeller as clean as possible, operate the ship in optimum trim conditions and optimise both routes and service speed.

Thank you for this interview. □

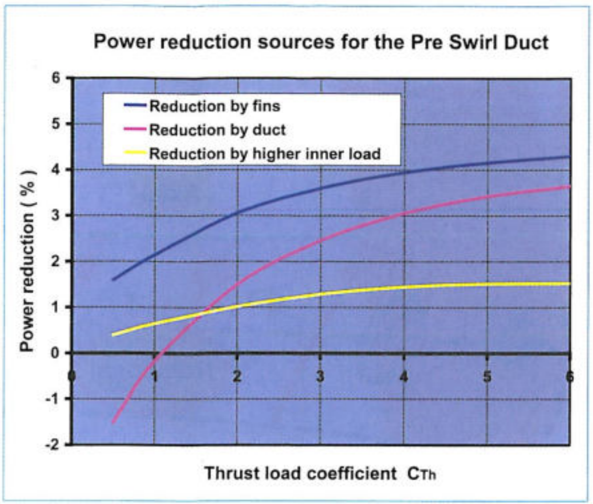

Fig. 4: Sources of power reduction of the PSD

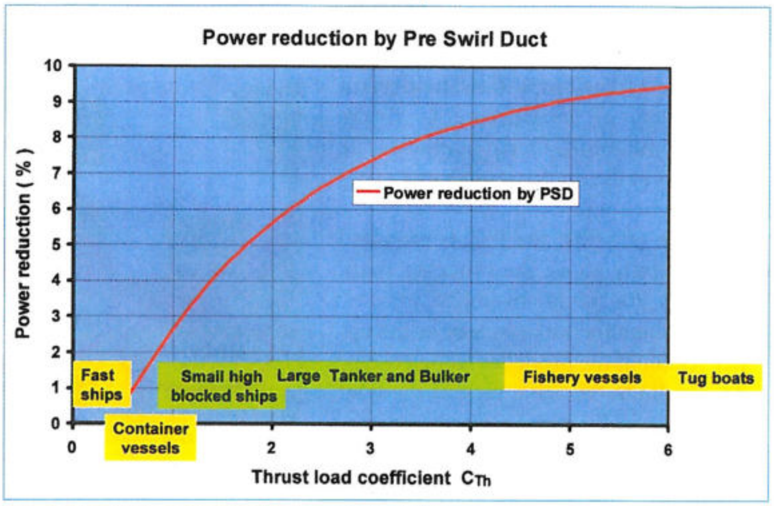

Fig. 5: Possible power reduction by the PSD

The best possibilities for improvement occur where C_{Th} is high and the ship speed relatively low, see figures (4) and (5).

The PSD is therefore well-suited for application to smaller container vessels, small vessels with high block coefficient, such as multi-purpose carriers, as well as bulk carriers and tankers of all sizes. Fishery vessels and tugs are typically equipped with Kort nozzles or Voith Schneider Propellers, so the installation of a PSD is generally precluded from these types. For all fast vessels including large container vessels the PSD is not appropriate at the present stage of development.

Optimisation by Computational Fluid Dynamics (CFD) methods

CFD calculations are valuable in assisting the optimisation process of ship lines and propulsion devices. Whereas estimation of absolute power is still a difficult task for current CFD techniques given the available computer power, the calculation of changes of power due to changes in configuration is well within the capabilities of today's CFD technology. This approach was therefore used for initial optimisation of the PSD.

The flow around the aft body of a ship and its propulsion devices can be numerically estimated using Reynolds-Averaged Navier Stokes (RANS) methods, which take into account the viscosity effects of the location and individual pitch alignments were determined and optimised based knowledge of the ship's flow field via a combination of measurements (wake field), computational fluid dynamics, scale model tests of the duct assembly and experience.

It can be seen that the PSD is mounted offset vertically above the propeller shaft. The reasons for this are twofold:

- To distribute the wake equally in the upper region of the propeller plane
- To distribute the wake race of the duct through a wider range towards the propeller

In addition, the duct eliminates tip vortices from the fins, acting in a similar fashion to end-plates or wing fences.

The rotational asymmetrical arrangement of the rudder fins is due to the shape of the ship wake. The upward turning blades work mainly with the ship wake flow whereas the downward turning blades work against it. Therefore to achieve the desired pre-swirl flow correction more fins are required on the upward turning side of the propeller to redirect the flow against the direction of the propeller blades. It should be noted that the bilge vortices from the hull make the design especially complicated and have to be taken into consideration.

The optimisation of the whole device is therefore a complex non-linear task, involving sub-optimisation of various key features.

Power saving – sources around the rotating propeller

The aft hullform, rotating propeller and rudder interact with each other to influence the overall propulsion performance. In particular, the propeller influences flow both ahead and astern of it, for example see the wake field images with and without the running propeller in figures (7) and (8).

Losses around running propeller

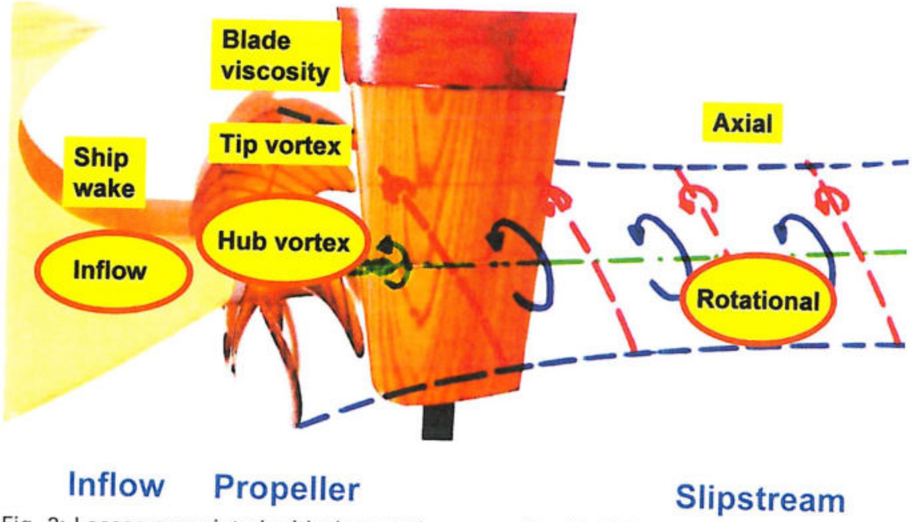

Fig. 3: Losses associated with the rotating propeller. Red denotes losses reduced by the PSD

There are three areas of influential losses around the rotating propeller: the inflow, the propeller itself and the resulting slipstream (propeller race), see figure (3). The following list gives an overview of improvable elements of the propeller flow and several possibilities for improving the propulsion efficiency, in other words improving the power saving.

1 Inflow

- Ship's wake: can be improved with better ship lines
- Asymmetrical inflow: can be improved by pre ducts, such as WED, SILD, PSD
- Pre rotation: can be improved by SVA Fin System, Pre Swirl Fins, PSD

2 Propeller

- Blade friction losses: can be improved with smaller blades, lower roughness
- Tip vortices: can be improved by integrated fences at the blade tips – the Kappel propeller
- Hub vortex: can be reduced by PBCF, PSD

3 Slipstream

- Rudder: can reduce the rotational losses, asymmetrical (twisted) rudder
- Rudder fins: can reduce rotational losses

The PSD reduces rotational losses in the slipstream by generating a pre-rotation of the flow in front of the propeller.

In summary, the new PSD reduces losses at the inflow to the propeller by equalising the inflow via the duct, reducing slipstream losses through the use of pre swirl fins and reducing hob vortex losses by increasing the propeller load at the inner radii.

The achievable power reductions depend on the ship speed and the propeller thrust coefficient C_{Th}:

$$C_{Th} = \frac{T}{\frac{1}{2}\rho \cdot V_A^2 \cdot D^2 \cdot \pi/4},$$

where ρ is the water density, V_A the advance velocity, D the propeller diameter and T the propeller thrust.

working fluid. Large Eddy Simulation (LES) and Direct Numerical Simulation (DNS) offer exciting opportunities for the future, but their demanding mesh requirements and resulting calculation times currently preclude their use for fast, efficient optimisation. Nevertheless, given a suitably refined mesh and careful turbulence modelling RANS calculations give an excellent insight into the flow properties.

Like all CFD methods, the results are many and diverse, so it is a formidable task to manage the vast quantity of data that is produced. Successful use of CFD therefore requires understanding and experience in use of meshing, the calculation process, extracting and recognising the important results as well as their interpretation.

Following some preliminary CFD calculation iterations, the following results were selected as a basis for judgement at each design step:

- Computed wake, with and without a rotating propeller, evaluated at various planes and operating conditions
- Duct flow and associated forces
- Fin flow and associated forces
- Additional resistance of components
- Differences in delivered power

By means of example figures (6) to (10) show the computed wake fields for five different operating conditions for one PSD design variant for a 300,000 DWT VLCC. Figure (6) corresponds to the computed wake field at the propeller plane for the naked hull without propeller, rudder or PSD. Figures (7) to (10) show the wake at a plane positioned in front of the propeller and just behind the duct for various configurations. Of particular note are the differences in wake between the cases with and without the rotating propeller, see figures (7) and (8).

Model test results

Self propulsion tests are used to determine the required power and achievable ship speed, and are suitable for the validation of the CFD results. The optimisation of the individual fin pitch settings is particularly quick and easy; however optimisation of other elements of the duct is more difficult and expensive, due to the requirement that each design iteration would require an additional duct model to be manufactured and fitted.

During 2008 an initial series of self propulsion tests with the PSD were carried out at HSVA for an active project of an 118,000 DWT Bulk Carrier. The propeller of this vessel is working at C_{Th}-values of about 2.3.

Figure (11) shows the installed PSD on the ship model. Initial optimisation of the duct and fin alignment angles and section shapes was performed by CFD analysis. This configuration was used as a basis for the model test programme.

The model test results are broadly in line with the CFD predictions, in terms of both power saving and optimal fin angles. The basis PSD as initially fitted resulted in a reduction of delivered power of 5.6 % at the ship design speed. The model-based fin optimisation process resulted in a further power saving of approximately 0.9 %. Thus

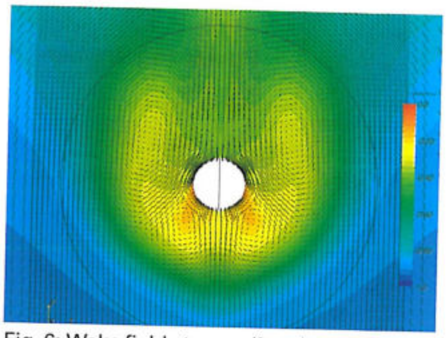

Fig. 6: Wake field at propeller plane, hull only

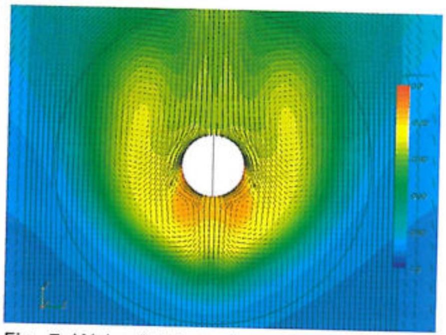

Fig. 7: Wake field at plane directly behind duct (in front or propeller), hull only

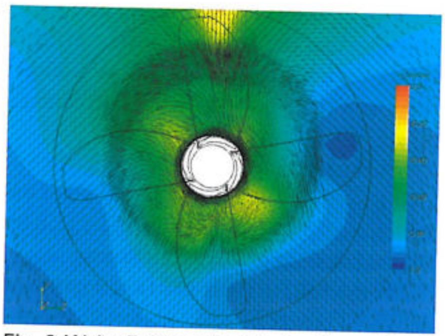

Fig. 8: Wake field behind duct with rudder & rotating propeller, no PSD

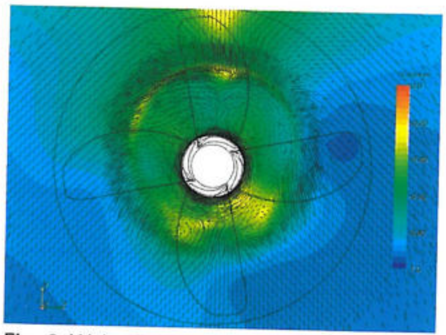

Fig. 9: Wake field behind duct with rudder, rotating propeller & PSD duct (without fins)

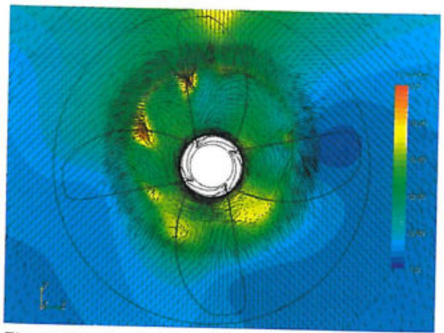

Fig. 10: Wake field behind duct with rudder, rotating propeller, full PSD duct with fins

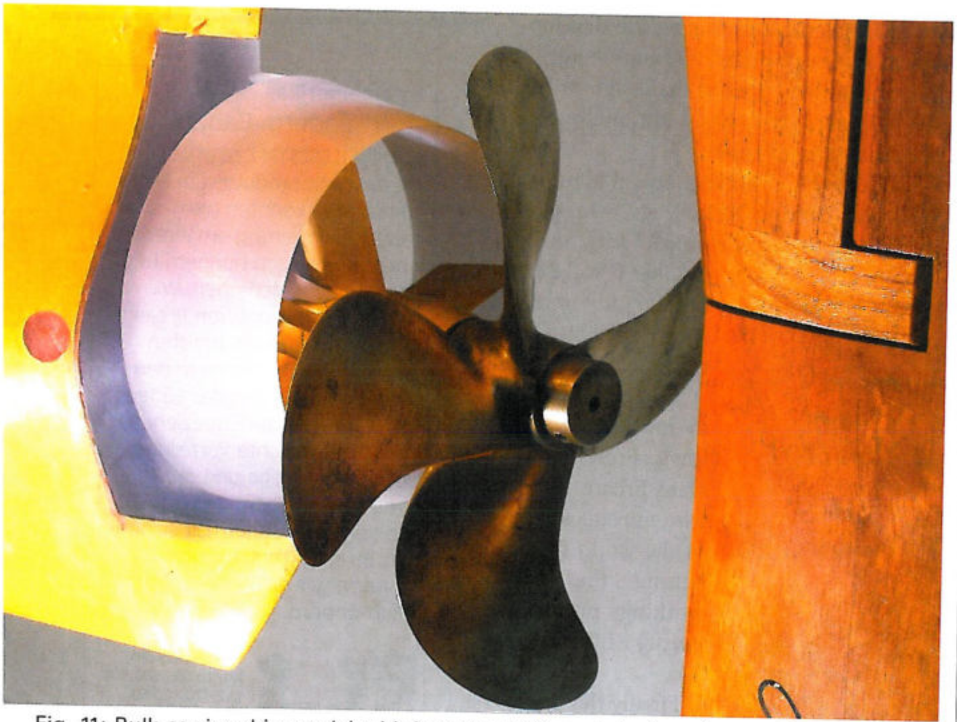

Fig. 11: Bulk carrier ship model with installed PSD model, view diagonal from the back/port

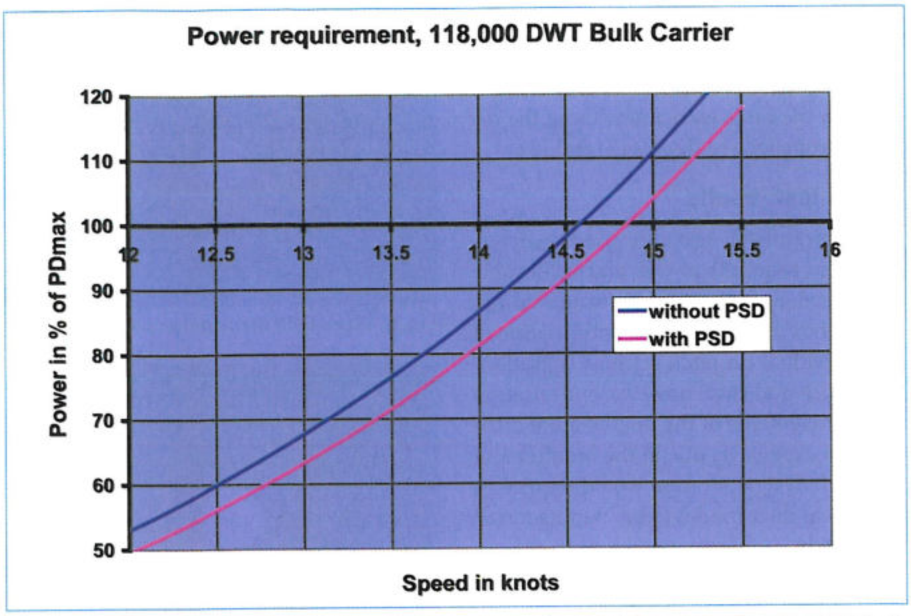

Fig. 12: lower reduction of about 6.5 % by PSD in self propulsion test, 118,000 DWT Bulk Carrier

an overall power saving of about 6.5 % at the design speed is achievable for this vessel; see also Figure (12). This figure also demonstrates that the degree of power saving is valid over a wide operational speed range; there is a tendency for higher percentage power reductions at lower speeds.

The increase in speed at the design delivered power is 0.27 knots as the result of installation of the PSD. The rpm reduction by PSD at constant power is 0.9 % only.

The duct alone resulted in a reduction of delivered power of 2.3 % at design speed.

Conclusions and future developments

The PSD is suited to ships whose propeller load C_{Th} is typically greater than 1.0 and speed less than 20 knots. Generally speaking this encompasses small container vessels, small vessels with high block coefficient, multi-purpose carriers, all tankers and bulk carriers. Optimisation of the PSD is required on a ship-by-ship basis. Modern CFD techniques are used to assist in this process, and the results have been validated against model tests.

Future planned developments of the PSD involve extending the design and optimisation process to include vessels faster than 20 knots.

Acknowledgements

Thanks from the author go to Mr Dirk Lehmann, Managing Director of the Hamburg-based Becker Marine Systems, who initiated the cooperation and has provided valuable support throughout.

Many thanks also to the CFD team at ibmv in Rostock, who have performed all of the CFD calculations during the development and design phase, and have contributed many ideas. Special thanks go to Mr. Steve Leonard, who has additionally corrected the present paper.

Thanks to Dr Zhao Ye, General Manager GreenSeas Marine Technology, who gave permission to use the ship model of the 118,000 DWT Bulk Carrier for reference model tests of the PSD.

Special thanks go to my former colleague Professor Dr. Hans-Erhard Peters, who introduced me in the world of the behind the ship working propeller twenty five years ago.

Finally thanks go to the staff at HSVA, who have carried out a complicated model test procedure and used new technical solutions for the modelling of the PSD to their customary high quality.

References

[1] Andersen, P., Kappel, J. J., Schwanecke, H., (1992), »On Development of Tip-Fin Propellers«, Jahrbuch der Schiffbautechnischen Gesellschaft, 86. Band, pp. 203 – 209, Berlin: Springer Verlag, 1992

[2] Lee, J. T., Kim, M. C., Suh, J. C., Kim, S. H. and Choi, J. K., (1992), »Development of a Preswirl Stator-propeller System for Improvement of Propulsion Efficiency: a Symmetric Stator Propulsion System«, Transaction of SNAK, Vol. 29, No. 4, Korea, Nov. 1992

[3] Mewis, F. Peters, H.-E., (1986), »Power Savings through a Novel Fin System«, 15. SMSSH Conference, Proceedings Vol. 1, pp 9.1–9.6, Varna, Bulgaria, 1986

[4] NN, (2005), »Hyundai Thrust Fin Improving Propulsion Efficiency, Flyer, Hyundai Maritime Research Institute, Ulsan, Korea, 2005

[5] Ouchi, K., Kawasaki, T., Tamashima, M., (1990), »Propeller Efficiency Enhanced by PBCF«, 4th International Syposium of Marine Engineering (ISME 90), Kobe, Japan, 1990

[6] Sasaki, N., Aono, T., (1997), »Energy Saving Device »SILD««, Journal of Shipbuilding Vol. 45, No. 135, Japan, Dec. 1997

[7] Schneekluth, H., (1986), »Wake equalising ducts«, The Naval Architect, London, UK, 1986

[8] Wagner, R., (1929), »Rückblick und Ausblick auf die Entwicklung des Contrapropellers«, Jahrbuch der Schiffbautechnischen Gesellschaft, 30. Band, pp. 195 – 256,, Berlin: Springer Verlag 1929

The Author:
Dipl.-Ing. Friedrich Mewis
Consultant, MSH,
Mewis Ship Hydrodynamics,
Dresden.
otto.fried@web.de