Conference Paper, Published Version

Böttner, Carl-Uwe; Anschau, Pascal; Shevchuk, Ivan

Analysis of the flow conditions between the bottoms of the ship and of the waterway

ANALYSIS OF THE FLOW CONDITIONS BETWEEN THE BOTTOMS OF THE SHIP AND OF THE WATERWAY

Carl-Uwe Böttner, Federal Waterways Engineering and Research Institute, Hamburg, Germany
Pascal Anschau, Schiffbau-Versuchsanstalt Potsdam, Germany
Ivan Shevchuk, Institute for Fluid Dynamics and Ship Theory, Hamburg University of Technology, Germany

SUMMARY

A physical model test was performed, to which a scaled model (1:40) was designed and constructed to carry two Laser Doppler Velocimetry sensors and provide optical access at different locations along the hull to detect the stream close to bottom of the hull and resolving the boundary layer flow at the hull. The measurement campaign was conducted in a shallow water towing tank and included a range of water depths and ship speeds such that the influence of each became analysable. The purpose of the experiment is to provide validation data to further corroborate a numerical approach and to gain deeper insight in the flow conditions in the gap flow underneath the vessel in very shallow water. In addition the extension of the effect of propulsion on the dynamic sinkage and trim in very shallow water was checked. Even though for one particular ship design and restricted to model scale 1:40, the results are regarded a valuable basis for further investigations on this phenomenon.

NOMENCLATURE

B	ship beam (m)
h	water depth (m)
L	length of the towing tank (m)
L_{PP}	length between perpendiculars (m)
T	draught of the ship (m)
V	speed (m/s)
$V_{Full\ Scale}$	speed through water, full scale (kn)
W	width of the towing tank
λ	model scale (-)
∇	displacement (m³)

1 INTRODUCTION

Through years of experimental tests in model scale to predict ship induced wave loads on bank protection in channels and waterways at Federal Waterways Engineering and Research Institute (BAW), a comprehensive collection of squat measurements had grown, allowing drawing some principal and systematic conclusions on the squat effect in shallow and restricted waters. One was the observation of a significant increase of the trim angle when water depth to draught ratio is decreased to less than h/T = 1.3. Change in the flow regime was suspected to be responsible for this effect.

Basically the squat phenomenon is a pure Bernoulli-effect and driven by local pressure conditions. Therefore it is tempting to apply potential flow approach to calculate the squat to be expected. The advantages to other computational fluid dynamics approaches are evident: at very low computational cost a similar useful result is obtained in an extremely short time. Modern Panel codes, solving potential flow equations for arbitrary shaped bodies like ship hulls, get the solutions in parts of a second up to some minutes CPU-Time.

Unfortunately for very shallow water conditions, which apply to water depth to draught ratios h/T < 1.3, the potential flow solutions for squat get inaccurate and show a big difference to the measurement data in towing tank (Böttner et al., 2011). One plausible explanation is that the basic assumption of potential flows, the negligibility of viscous effects in the flow is no longer valid. This appears self-evident, besides it is unclear why and how viscous effects appear on the scene. Considering the dimensions of the gap, a ship of a draught of 12 m still has more than 3 m under keel clearance at h/T < 1.3, it is not obvious how viscous boundary layers gain influence in the flow regime.

Initiated by these findings or possibly better said educated guesses, further research has been performed to validate the presumption and to identify the underlying mechanisms. Obviously, there are two aspects to be further investigated. First the possible influence of boundary layers wherefore the boundary layers and the viscous turbulence needs to be measured and calculated. And in continuation second whether this applies for full scale as well or is a phenomenon restricted to the conditions in model scale. The experiments reported here aim at the first aspect; the investigation of full scale flow conditions is reserved for numerical approaches.

To strengthen the assumption and gain deeper insight in the first aspect, computational fluid dynamics simulations with viscosity have been performed by solving unsteady Reynolds Averaged Navier-Stokes Equations (U-RANSE). Even though the model scale is 1:40, the pure dimensions are nevertheless challenging for numerical calculation, especially if the boundary layers are to be scrutinized. A quite elaborate investigation using the RANSE solver package OpenFOAM (Greenshield and Weller, 2019) was performed, which revealed boundary layers growing along the hull length stream downwards at both walls, the hull of the vessel as well as the floor of the towing tank and unify in the last third of the hull if speed

through water of the ship is high enough (Shevchuk et al., 2016).
The numerical results are strongly sensitive on the numerical and physical boundary conditions set. This situation is very often the impetus for validation through experimental data gained from according suitable experimental set-ups.

Therefore an experimental measurement campaign in the physical model was planned and launched. The aim was to determine the flow conditions in the close vicinity of the bottom of the hull and to resolve the boundary layer flow, if possible. The experimental set-up, the measurements and the results are introduced in the following chapters.

To answer the second aspect, the applicability of the results in the physical model to the natural conditions in scale 1:1, another numerical fluid dynamics investigation has been initiated. Obviously measurements at real ships sailing in the waterways and channels are practically very complicated if not impossible. But also in the virtual world of fluid dynamics simulations flow around a ship is one of the most demanding and challenging tasks, since the highest Reynolds-Numbers (range of Billions) of technical flows do occur here.

2 OPTICAL FLOW MEASUREMENTS IN TOWING TANK

2.1 EXPERIMENTAL SET-UP

Laser Doppler Velocimetry (LDV) is an optical flow velocity detection technique, where the Doppler-shift of reflected phase interference pattern is evaluated.

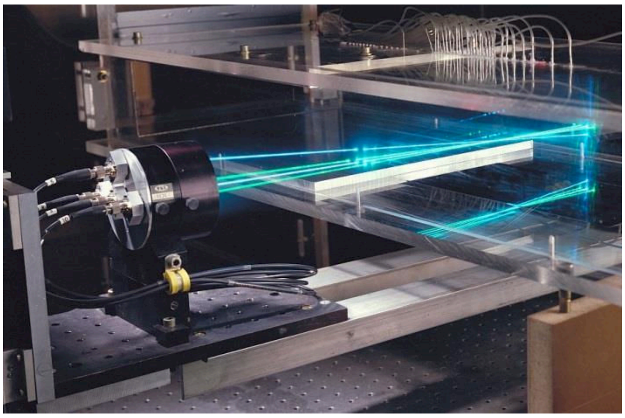

Figure 1. 2D-LDV where two different wave lengths are generated and oriented 90°, each of them is split and slightly shifted so that four beams are finally focussed by lenses and the back-scatter is analysed. (Foto: Wikipedia)

A coherent laser beam is split in two parts and the angle under which the two half beams are focussed in the control volume and permeate each other let them interfere and generate a set of straight fringes. The focal length of the lenses determines the size of the control volume. The interference pattern moves with a characteristic speed, which is subject to Doppler shift when reflected by a particle passing the control volume. The moving direction of the interference pattern gives the orientation of the detected speed component. For 2D-measurements a second beam in another, distinguishable wave length and oriented so, that the pattern moves rectangular to the first one is required, as in **Figure 1**, where a green and a blue coherent laser beam is focussed in the same control volume with a 90° shifted orientation.

Figure 2.A 1D and a 2D-LDV mounted on the model of a 14000 TEU Containership to detect the near wall flow conditions. (Foto: SVA-P)

A model of 14000 TEU Container ship (cf. Table 1) was designed and constructed especially to the needs of LDV measurements through the hull. To get insight in the flow conditions, 7 small windows were included (**Figure 3**) three along the longitudinal centre, two at the chine, and two in the aft section upstream of the propeller disc to get an idea of the propeller inflow and its changing depending on the propeller load. The model was fitted with propeller and rudder and the experiments have been performed in self-propulsion mode, i.e. model propulsion condition, no friction correction applied.

Table 1. Dimensions of the model ship

14000 TEU Container Carrier – M1628S001			
Length btw. Perpendiculars	L_{PP}	[m]	8.68
Beam	B	[m]	1.3
Draught	T	[m]	0.4
Displacement	∇	[m^3]	3.0127
Scale	λ	[–]	40

The towing tank tests were performed at Duisburg Towing Tank, which provides the required shallow water conditions. The tank dimensions at Duisburg are listed in Table 2. This facility also offers optical access from below by a window in the floor quite in the middle of the tank. This is used for wake measurements by particle imaging velocimetry, which was decided not to be used together with LDVs, to avoid any possible damages by pumping laser light of high intensity into the highly amplified photonic detectors.

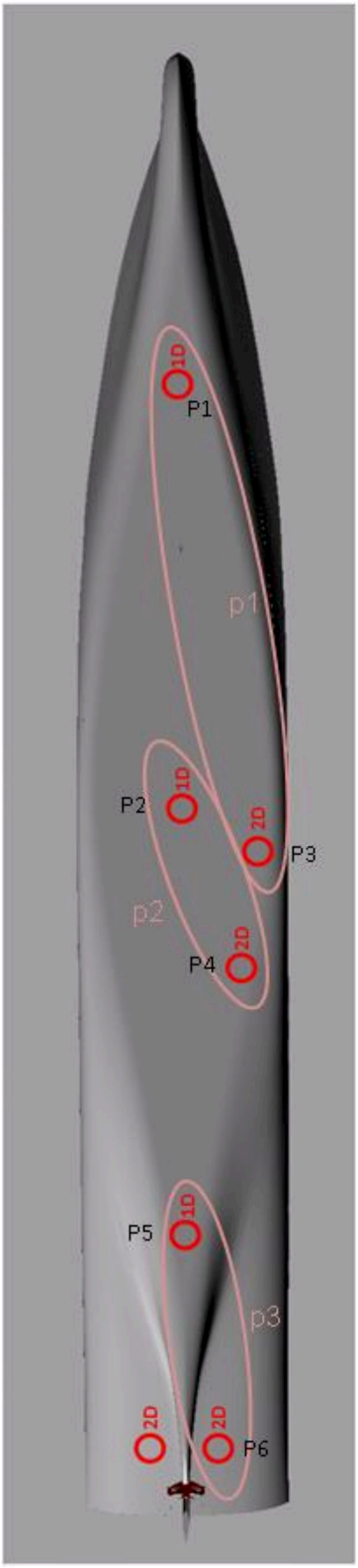

Figure 3. Positions of windows for LDV measurement

The mounting of the LDVs, visible partly in **Figure 2**, was designed such, that always two positions were occupied at the same time and two LDVs were in use. The LDVs had to be moved from one position to the next, therefore according weights could be shifted opposite to guarantee trim and ballast constant during the entire measurement campaign.

Table 2. Dimensions of the towing tank

Shallow Water Tank at DST, Duisburg, Germany			
Length	L	[m]	200
Width	W	[m]	10
Depth	h	[m]	0 - 1.2
Carriage speed, max	V_{max}	[m/s]	6.5

The LDVs were mounted on traverses fitted with stepping motors for precise positioning in vertical direction. Initial position was always at the outer edge of the window, such that the control volume just started to provide valid signals. From that position always when given minimum number of valid detections, the control volume was moved 5 mm further, away from hull towards the flow regime of the boundary layer.

2.2 EXPERIMENTS

Different speeds and different water depths were chosen (Table 3) in order to cover a significant range of conditions and to see possible transition regimes.

Table 3. Speeds and under keel clearances

Water depths

	h/T	Depth [m]	UKC [m]
h1	1.5	0.6	0.2
h2	1.25	0.5	0.1
h3	1.15	0.46	0.06

Speeds

Speed	Full Scale [kn]	Model [m/s]
V1	8	0.65
V2	10	0.81
V3	12	0.98
V4	14	1.14

Additionally the dynamic sinkage and trim were detected for comparison with results from other similar experiments. In shallow water experiments squat is one of the effects of interest.

There is a permanent discussion on the influence of the propeller and propeller load on dynamic sinkage and trim in shallow water conditions. Therefore some additional test runs for comparison were included, with fixed propeller (rotationally fixed) and with propeller dismantled (as in resistance towing test). Obviously the comparison is valid only for this particular ship design, but regarded helpful to get a more general insight in this phenomenon later when suitable data from other designs might eventually become available.

3 RESULTS

The main purpose of the experiments was to detect the flow conditions in the boundary layer at the bottom of the hull. This was initiated by the lack of data to validate the numerical findings presented in 2016 (Shevchuk et al., 2016).

3.1 BOUNDARY LAYER AMIDSHIPS CENTRE HULL

An interesting finding of the numerical investigation was a unification of the two boundary layers in the gap flow between the bottoms of the hull and the waterway. In the simulations this occurred basically in the centre and the aft third of the hull bottom plate. Therefore this experimental campaign was arranged to confirm the numerical findings. In the centre line of the hull there are the windows at position P1, P2 and P5, see **Figure 3**.

P1 is at the front part, as forward as possible to get optical access. P2 is positioned close to the main frame in the middle of the bottom plate and P5 is in the aft area, where the bottom plate is still plain enough to accommodate a window for optical flow measurements.

The experimental results in **Figure 4** to **Figure 7**indicate that the boundary layer could be resolved. The speed close to the hull is quite comparable for all of the four velocities. As well, an increase of the boundary layer flow speed with growing distance to the hull up to the ship speed can be observed in positions P2, P3 and P4 (**Figure 4** to **Figure 7**).

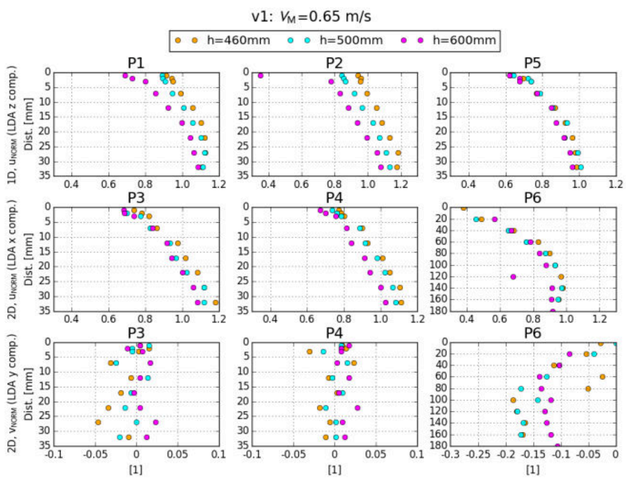

Figure 4. Boundary layer at the hull at different under keel clearances (h/T = 1.5; 1.25; 1.15) and positions (Figure 3), 8 kn in full scale.

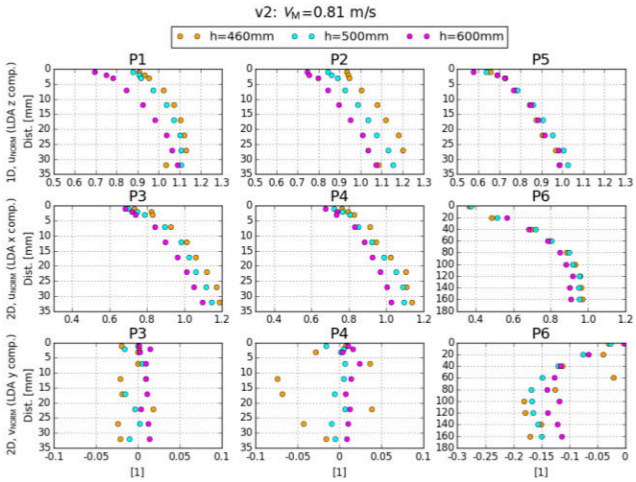

Figure 5. Boundary layer at the hull at different under keel clearances (h/T = 1.5; 1.25; 1.15) and positions (Figure 3), 10 kn in full scale.

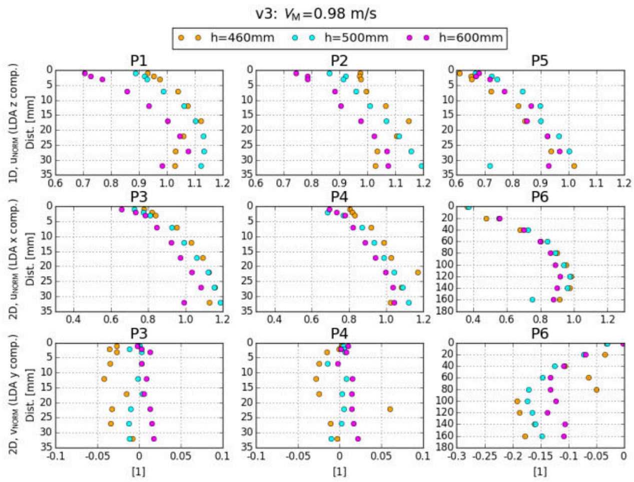

Figure 6. Boundary layer at the hull at different under keel clearances (h/T = 1.5; 1.25; 1.15) and positions (Figure 3), 12 kn in full scale.

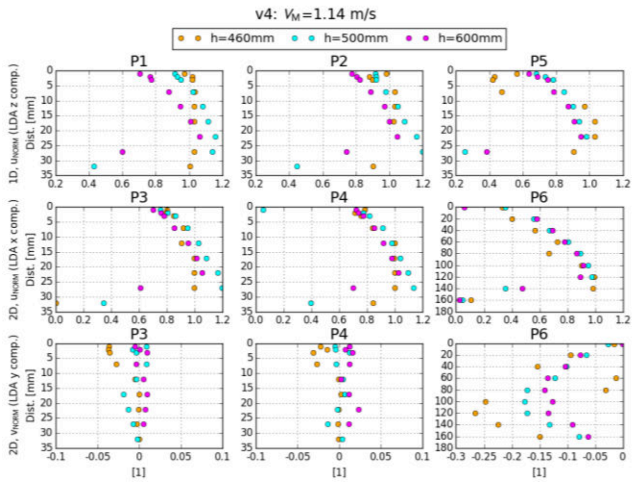

Figure 7. Boundary layer at the hull at different under keel clearances (h/T = 1.5; 1.25; 1.15) and positions (Figure 3), 14 kn in full scale.

3.2 BOUNDARY LAYER AT CHINE

Next to the central flow at the bottom of the hull, the flow conditions and its direction at the chine was of interest. Especially if there is some flow directed upwards around the chine and possibly some chine vortex generated. The windows at positions P3 and P4 (**Figure 3**) were designed as close to the chine as possible to still provide sufficient access for optical measurement techniques like LDV.

To gain insight in potentially twisted and or sheared flow conditions if upwards flow partially occurred, the 2D-LDV detectors were applied at these positions. In **Figure 4** to **Figure 7** there are for the particular positions P3, P4 and P6 two graphs each, showing component in direction of ship speed and traversal flow speed component.

There was no upstream effect detected, the orthogonal flow component detected is varying around the mean value 0 m/s (**Figure 4** to **Figure 7**) at P3 and P4 is for all of the 4 speeds and all of the under keel clearances investigated. The longitudinal component in the boundary layer is comparable to what has been observed at central positions P1 and P2 and is coherent with the overall figure suggested by the experimental results.

3.3 GROWTH OF THE BOUNDARY LAYER

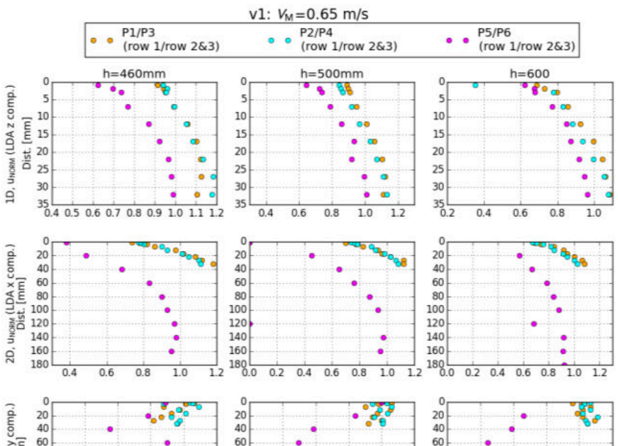

Figure 8. Boundary layer along the hull from bow (yellow) over midship (turquoise) to aft (pink) at different under keel clearances (h/T = 1.5; 1.25; 1.15) 8 kn in full scale.

The numerical findings are indicatively affirmed by the experimental results. In **Figure 8** to **Figure 11** the flow speeds are plotted for one speed and one water depth in one plot. The boundary layer increases for all speeds and all h/T-ratios at the aft region. According the assumed unification of the boundary layers, the boundary layer flow in P5 is almost constant over the range of velocities investigated (**Figure 8** to **Figure 11**). Additionally the width of the gap has an impact. As becomes obvious in each top row in **Figure 8** to **Figure 11**, where the 1D-LDV measurements along the centre are plotted, decreasing gap width induces increasing alternation of boundary layer at the aft ship. From the lowest width (water depth h = 460 mm) to the highest investigated (water depth h = 600 mm) the difference of the boundary layer from bow to aft decreases with increasing gap width or water depth.

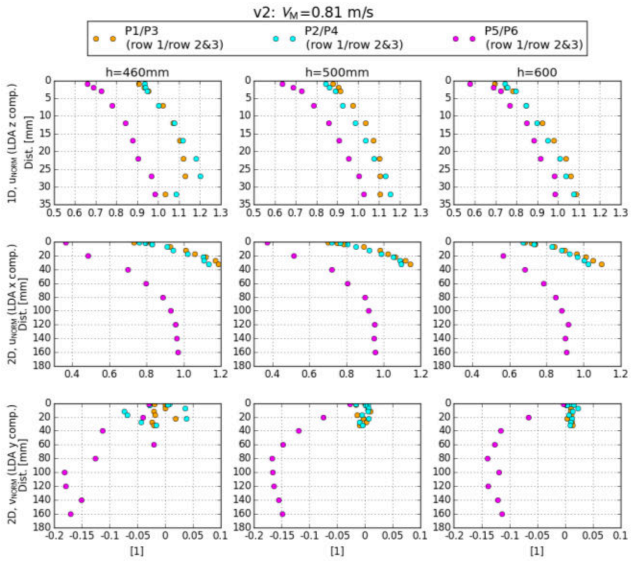

Figure 9. Boundary layer along the hull from bow (yellow) over midship (turquoise) to aft (pink) at different under keel clearances (h/T = 1.5; 1.25; 1.15) 10 kn in full scale.

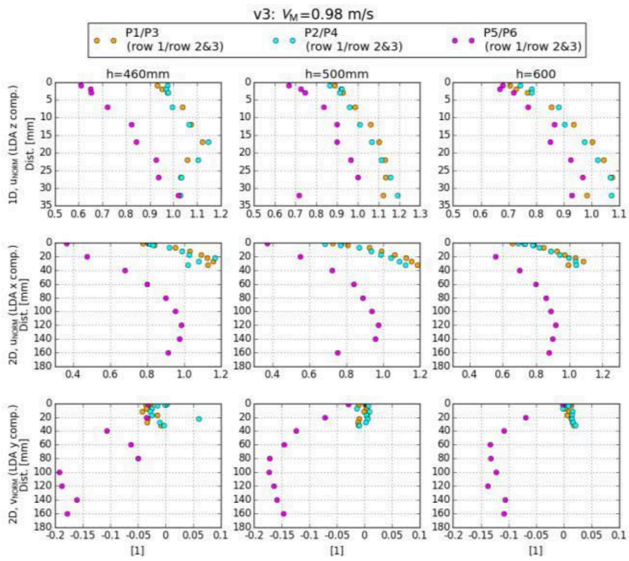

Figure 10. Boundary layer along the hull from bow (yellow) over midship (turquoise) to aft (pink) at different under keel clearances (h/T = 1.5; 1.25; 1.15) 12 kn in full scale.

In the second and third row, the results for the 2D-LDV measurements at Position P6 are plotted in pink color. At this position, which is located in the aft ship close to water line, bigger stepping was applied to detect variations in wake and inflow conditions to the propeller.

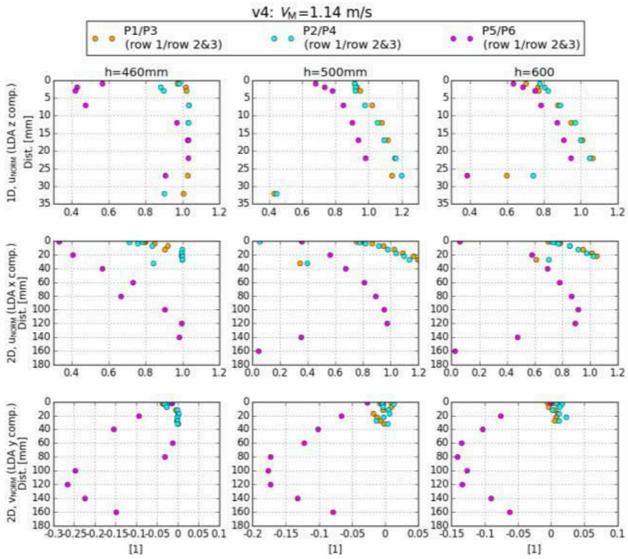

Figure 11. Boundary layer along the hull from bow (yellow) over midship (turquoise) to aft (pink) at different under keel clearances (h/T = 1.5; 1.25; 1.15) 14 kn in full scale.

3.4 INFLUENCE OF PROPELLER ON SQUAT

As already mentioned, there is an ongoing debate on the actual influence of the propeller induced pressure distribution on the dynamic sinkage and trim of the aft section. Especially in very shallow water, i.e. h/T < 1.3, where a considerable increase of the resistance is noticed and the propeller is opposed to increased load and torque, the propulsion induced pressure field is suspected of further reducing under keel clearance at the stern and changing trim angle. Regularly there is a significant change in trim observed in comparison to Squat in moderate shallow water conditions. Another well-known shallow water effect is an increase of resistance resulting in a higher loaded propeller working at higher torque. There is no doubt that this effect leads to a more prominent pressure field in the in- and outflow region of the propeller disc. From literature it is well known, that the pressure field induced by the propeller under load alters the flow conditions in the aft ship area. This is limited to the vicinity of the propeller's position and might be considered reaching no further upstream than up to 10 times propeller disc diameter (Vladimir Krasilnikov, 2014). The dominant interaction of the propeller's pressure field and the hull is by vibration induction when the tip pressure peaks pass by the aft ship above the propeller and induce noise and vibrations in the hull (Su et al., 2017; SVA Potsdam, 2015). What isn't that obvious is to reckon quantitatively the degree of propeller's influence on squat in very shallow water.

At the premises of BAW, there is no towing carriage in the test facility available since unnecessary for civil engineering waterway investigations. Therefore, all squat data at the basin of BAW are gained from physical model tests at the model self-propulsion condition solely. The physical model test campaign presented herein was performed at Duisburg Towing Tank, which offers a towing carriage and was regarded as a valuable opportunity checking the propellers influence to this particular ship at least, well knowing that this is additionally restricted to model scale.

Table 5. Squat with and without Propulsion

Speed = 0.98 m/s (12 kn in full scale) and h/T = 1.15			
Value	**Unit**	**Self-Propelled**	**No Propeller**
Sink_Bow	[mm]	25.03	27.7
Sink_Stern	[mm]	18.63	17.9
Trim	[°]	0.048	0.074

A comparison of squat with and without propulsion at similar draft, speed and water depth conditions is shown in Table 5. In case "Self-Propelled" the carriage follows the vessel which is hold captive in heading and yaw, but free to move in all of the remaining modes. "No Propeller" is realized by towing the model with the carriage at the given speed; the propeller has been dismantled and was replaced by a cap covering the hub.

The flow detections at position P6 were included to investigate potential change of propeller inflow regime due to shallow water conditions. Comparing **Figure 4** to **Figure 7** as well as in **Figure 8** to **Figure 11**, there was no influence of the under keel clearance on the flow observed in this position. Apparently there is none or only minor blocking effect on the wake by small under keel clearance. Removing the propeller has an effect and the propeller's contribution to the wake entirely on radial components becomes visible in **Figure 12**.

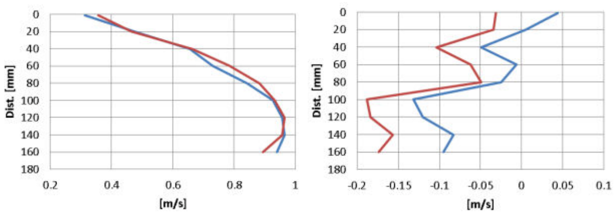

Figure 12. Flow conditions in position P6 in front of the propeller disc: with (red) and without (blue) Propeller mounted. Longitudinal (u-comp.) left and orthogonal (v-comp.) right, h/T = 1.15, V_M = 0.98 m/s

To investigate the alternation of the flow conditions by a either loaded or standing propeller, as it occurs while manoeuvring in harbours for instance, in another series some runs were repeated with the propeller fixed in rotation and the ship towed by the carriage. The blockage by the non-rotating propeller has a prominent effect on the flow. The stopped propeller generates a backwater upstream which results in reverse flow direction (**Figure 13**, left). Less surprisingly, radial component of flow in front of the propeller more or less vanishes for propellers stopped (**Figure 13**, right).

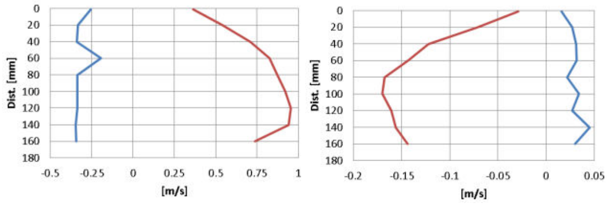

Figure 13. Flow conditions in position P6 in front of the propeller disc: rotating (red) and fixed (blue) Propeller mounted. Longitudinal (u-comp.) left and orthogonal (v-comp.) right, h/T = 1.25, V_M = 0.98 m/s

4 VALIDATION OF NUMERICAL RESULTS

Basically the purpose of the physical model test campaign was to provide data for validation of numerical fluid dynamics results and findings with another ship model and already presented at the 4th MASHCON (Shevchuk et al., 2016). A short citation of the publication shall be included here:

"[...] The most interesting results were obtained for the velocity distribution in the gap between the ship and the channel bottom.

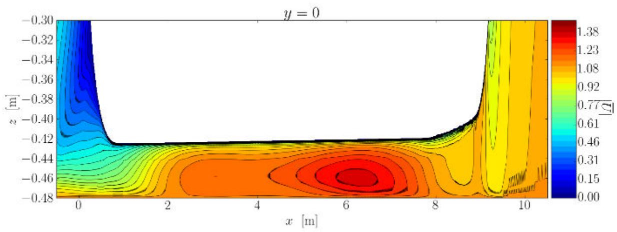

Figure 14: Velocity magnitude distribution at the middle line plane of PPM55 at h/T=1.15, U=1.13m/s

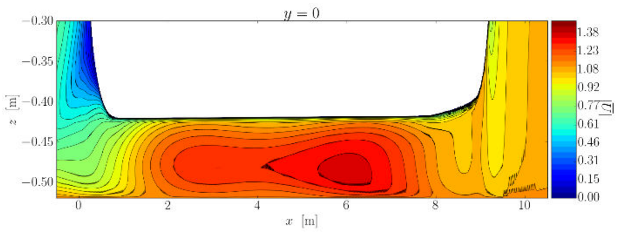

Figure 15: Velocity magnitude distribution at the middle line plane of PPM55 at h/T=1.3, U=1.13m/s

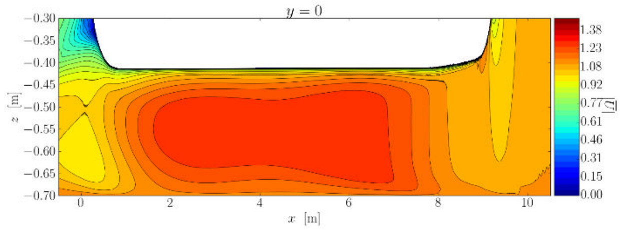

Figure 16: Velocity magnitude distribution at the middle line plane of PPM55 at h/T=1.75

In the Figure 14, Figure 15and Figure 16 one can see the distribution of the velocity magnitude at the middle line plane at different depths at the speed of 1.13 m/s. In the presented figures the following phenomenon can be observed. At h/T = 1.75 one can distinguish two separate boundary layers growing in the gap: one on the bottom and one on the ship hull. As h/T decreases, these two get united so that there is no region, for which it could be stated that the viscous effects are negligible there. [...]"

This particular numerical approach was applied to the ship model PPM52, which was built to accommodate Laser Doppler Velocimetry. The squat predictions and the flow condition close to the hull's wall and the tank's wall was the criteria for the numerical approach to prove validity. The Reynolds Averaged Navier-Stokes Equations were solved using OpenFOAM (Greenshield and Weller, 2019), the equations were solved with second order accuracy on a numerical grid of 23 to 25 Million control volumes, required to achieve non-dimensional wall distances smaller than one ($y^+ < 1$), allowing resolving the boundary layer completely without near wall modelling.

The agreement of the numerical and the experimental values is shown exemplary at a vessel's speed of 10 kn in full scale for different depth to draft ratios (h/T). The h/T ratios vary from 1.5 over 1.25 down to 1.15 (**Figure 17** to **Figure 19**). The RANSE simulation agrees well with the experimental findings discussed in 3.3. The boundary layer is extended in wall orthogonal direction with decreasing water depth and under keel clearance.

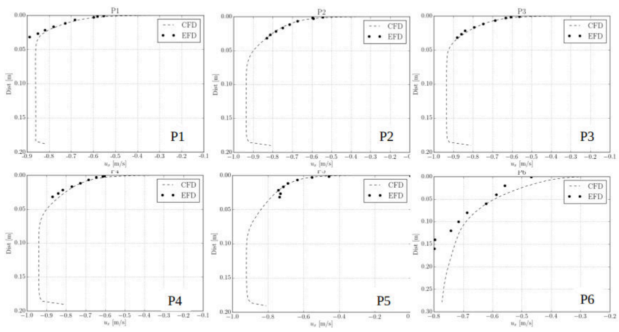

Figure 17. Comparison of numerical results and experimental data at six positions and vessel speed of 0.81 m/s (model scale) and depth to draft ratio (h/T) of 1.5

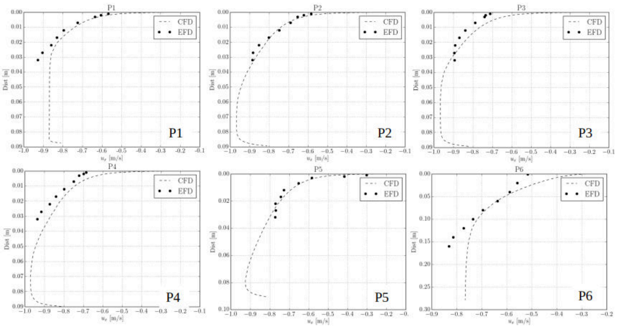

Figure 18. Comparison of numerical results and experimental data at six positions and vessel speed of 0.81 m/s (model scale) and depth to draft ratio (h/T) of 1.25

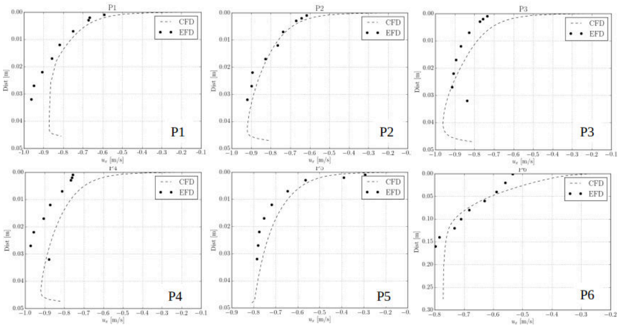

Figure 19. Comparison of numerical results and experimental data at six positions and vessel speed of 0.81 m/s (model scale) and depth to draft ratio (h/T) of 1.15

5 CONCLUSIONS

A model in scale 1:40 has been constructed for optical access to boundary layer flow conditions. Laser Doppler Velocimetry could be successfully applied and revealed the flow regime in the gap between the bottom of the hull and the towing tank floor. The two aims of the physical model measurements campaign were successfully reached: first to provide insight in the flow conditions close to the hull and second to provide data for evaluation and validation of numerical modelling approaches. Additionally the qualitative and quantitative contribution of the loaded propeller to the dynamic sinkage and trim in very shallow water was tested and approved.

The results are encouraging for further investigation of the phenomenon of squat in terms of scale effect. The propeller obviously gains impact on squat at very shallow water, but up o now, the evidence is in model scale only. Scaled model tests are well-known for higher wake numbers and more prominent wake than the same geometry at full scale. This is one of the challenges when predicting required propulsion and determining the proper self-propulsion point in full scale from propulsion tests in a towing tank in model scale. Accordingly, a less prominent impact of the propeller is expected in full scale. To which extend is subject of ongoing work.

6 REFERENCES

Böttner, C.-U., Heimann, J. and Uliczka, K., 2011. Numerical prediction of Squat of large Container Carriers on Waterways, in: Marine CFD 2011: 22-23 March, 2011, London, UK

Greenshield, C, and Weller, H., 2019. OpenFOAM and the OpenFOAM Foundation, available at: https://openfoam.org/.

Krasilnikov, V., 2014. Numerical Modeling of Ship-Propeller Interaction under Self-Propulsion Condition, in: Star Global Conference, Vienna, Austria.

Shevchuk, I., Böttner, C.-U. and Kornev, N., 2016. Numerical analysis of the flow in the gap between the ship hull and the fairway bottom in extremely shallow water, in: Proceedings of the 4th International Conference on Ship Manoeuvring in Shallow and Confined Water with Special Focus on Ship Bottom Interaction. Bundesanstalt für Wasserbau, Hamburg, Germany, pp. 37–42. https://doi.org/10.18451/978-3-939230-38-0.

Su, Y., Kim, S., Du, W., et al., 2017. Prediction of the Propeller-induced Hull Pressure Fluctuation via a Potential-based Method: Study of the Rudder Effect and the Effect from Different Wake Alignment Methods, in: Fifth International Symposium on Marine Propulsion: SMP'17.

SVA Potsdam, 2015. Pressure Fluctuation Measurement. available at: https://www.sva-potsdam.de/en/pressure-fluctuation-measurement/.

7 AUTHORS BIOGRAPHY

Carl-Uwe Böttner currently works as a research engineer with the Federal Waterways Engineering and Research Institute (BAW) in Hamburg on the fields of ship dynamics and ship handling simulation. He is responsible for consulting the authorities in terms of ship handling simulation.

Pascal Anschau currently works as a senior research engineer with Potsdam Towing Tank (SVA Potsdam). He is mainly responsible for computational fluid dynamics simulations and analysis of ship hydrodynamics. He studied Naval Architecture at Technical University of Berlin and holds a Diploma degree.

Ivan Shevchuk Works at TU Hamburg at the Institute for Fluid Dynamics and Ship Theory as a research engineer on the field of computational fluid dynamics. He holds a diploma and a PhD degree on Naval Architecture.