Conference Paper, Published Version

Poojari, Deepti; Kar, A. R.

Effect of Lateral and Depth Restriction on Ship Behavior Using Computational Fluid Dynamics

Zur Verfügung gestellt in Kooperation mit/Provided in Cooperation with:
Flanders Hydraulics Research, Ghent University, Maritime Technology

Verfügbar unter/Available at: https://hdl.handle.net/20.500.11970/99845

Vorgeschlagene Zitierweise/Suggested citation:
Poojari, Deepti; Kar, A. R. (2016): Effect of Lateral and Depth Restriction on Ship Behavior Using Computational Fluid Dynamics. In: Uliczka, Klemens; Böttner, Carl-Uwe; Kastens, Marko; Eloot, Katrien; Delefortrie, Guillaume; Vantorre, Marc; Candries, Maxim; Lataire, Evert (Hg.): 4th MASHCON - International Conference on Ship Manoeuvring in Shallow and Confined Water with Special Focus on Ship Bottom Interaction. Karlsruhe: Bundesanstalt für Wasserbau. S. 9-15. https://dx.doi.org/10.18451/978-3-939230-38-0_2.

4th MASHCON, Hamburg - Uliczka et al. (eds) - © 2016 Bundesanstalt für Wasserbau ISBN 978-3-939230-38-0 (Online)
DOI: 10.18451/978-3-939230-38-0_2

EFFECT OF LATERAL AND DEPTH RESTRICTION ON SHIP BEHAVIOR USING COMPUTATIONAL FLUID DYNAMICS

D B Poojari and **A R Kar**, Indian Register of Shipping, India

SUMMARY

Ship to bank interaction is extremely important for navigational purposes. Restriction in waters is basically a solid boundary on the sides of the hull as in canals or at the bottom of the hull or both. This lateral and vertical restriction brings with it complications in the flow around the hull. These flow complications have a direct effect on the hydrodynamic forces and moments acting on the hull form and thus influence the ship motion. Information of the vessel‘s navigational characteristics under such conditions is essential and will prove useful. The conventional experimental methods (model testing) is time consuming and expensive and also it does not capture the actual picture of the flow around the hull. Computational fluid dynamics (CFD) on the other hand being a visualization tool provides a clear image of the flow in the domain making it more vivid although we still see some limitations and accuracy problems with it. The current work aims at predicting the behavior of a tanker ship when it moves through a location with both lateral and vertical restrictions. Two bank shapes namely rectangular and surface piercing bank with a slope and a fixed under keel clearance of h/T=1.5 were used to predict vessel behavior under such severe conditions. Squat prediction for these types of restrictions are also carried out which is another important aspect under such blockage. The simulations are carried out for two different starboard offset from the bank. RANS based CFD solver is used for force and moment predictions, the modeling and meshing carried out with a combination of ICEM and SHIPFLOW®.

NOMENCLATURE

L_{pp}	Length between perpendicular
Y'	Non-dimensionalised Sway Force
N'	Non-dimensionalised Yaw Moment
K'	Non-dimensionalised Roll Moment
h	depth
T	Draft
U	Forward speed
y_b	Clearance from the bank
R'	Non-dimensional Resistance
C_B	Block coefficient
C_T	Total Resistance coefficient
B	Breadth
$Y+$	Y plus value
w	Wake fraction
v_a	Inflow velocity at the propeller plane

1 INTRODUCTION

The green house gas emission control to mitigate the global warming is on the priority list of IMO. Although shipping contributes to the larger part of transport, the emissions from them are relatively much lower compared to that of other transport system. Most efficient, cost effective and environment friendly mode of transport is the maritime transport system. During the last few decades the shipping community has seen a phenomenal scale enlargement and speed increase but utilization of this shipping for transportation has not witnessed much growth. India is one of the biggest peninsulas in the world with a coastline spanning 7500km but the freight movement by coastal ships is only about 7%. There is an enormous potential to utilize, explore and enhance coastal and inland shipping which eventually will aid reduction in carbon emissions which is higher in other modes of transportation.

Increase in size and speed of ships showcases an impressive progress made in the field. This also imposes severe concerns on various aspects of its behavior, one such concern is when it operates in a restricted environment. The probability of collision, grounding etc increases which raises concerns about oil leakage, traffic curtailment, repair costs, loss due to out of service condition to name a few. Perceiving the ship behavior in confinements can save millions and bring a sense of safety. Transfer of ship into a shallow depth condition switches it to a sluggish or less reactive behavior compared to open sea. This can be credited to the restrictions that modify flow around the hull. In addition to this if a transverse restriction is bought in the complications will exaggerate.

Several RANS based force and moment prediction methods for numerical captive model tests are established. Some of them are [4], [6], [7], [8] and [9]. Investigations/Studies have been carried out in the past for analyzing the ship behavior in confinements. The study carried out by [5] and [10] brings out the effect of shallow waters and the bank effect with variable bank configuration on the ship also the ship to ship interaction is presented here. A numerical method to predict ship squat is presented and detailed in [2]. Simulations and experiments conducted by [3] details about the pressure, velocity and force moment changes observed in restricted conditions for an LNG carrier.

The use of numerical tool to analyze the flow behavior although not wholly reliable is a good source of information on various aspects of flow around a ship. Computational hydrodynamics is an evolving tool and is more or less helping designers resolve various issues in comparatively feasible time. The prime focus of the paper is to bring out the use of CFD in predicting the changes observed when a ship is put in a restricted region. The changes in forces, moments and the flow visuals are presented. The likely changes in the flow and the effects on propeller for a given depth are also presented. The effect of the changes in bank clearance on squat is also explored. Viscous solver from SHIPFLOW® tool is used for the RANS computations.

2 NUMERICAL COMPUTATION

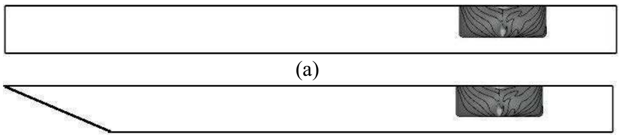
The simulations are carried out using a naval architect specific computational tool SHIPFLOW®. The tool is a hybrid steady state solver with both potential and viscous computation sources. The present study works on the global approach using the viscous solver XCHAP

2.1 DOMAINS

The numerical simulations were carried out for different cases. First the study was done for a vessel in shallow depth with freedom in the lateral direction. While another case for lateral depth restriction with two canal configuration was simulated. The domains chosen for the investigations are shown in Fig 1(a) and Fig 1(b).

2.1 (a) Canal 1

The canal is rectangular in shape with both the canal walls vertical. The second canal configuration is a surface piercing sloped wall with slope 1:4.

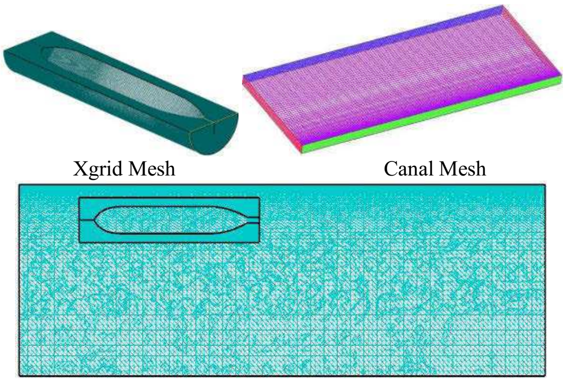

(a)

(b)

Figure 1. a) Vertical wall canal b) Sloped wall canal

2.1 (b) Mesh

The domain is discretized with overlapping grid. H_O type structured grid automatically generated using XGRID module around the hull is immersed in H_H type structured grid imported from ICEM meshing. This combination helps to achieve better Y+ values and thus resolve the viscous sub layer effectively and also the number of elements to resolve the physics is also reduced. The mesh in XGRID extends 0.1Lpp ahead of the ship and 0.1Lpp aft with a radial distance of 0.15Lpp and the canal grid extends 0.75Lpp ahead and 2Lpp aft of the ship. The total number of interpolation cells used is 0.3 million to get the simulation time in control and also to capture better results.


Xgrid Mesh

Canal Mesh

Overlapping Mesh

Figure 2. Overlapping structured mesh and component mesh

2.1 (c) Boundary conditions

The boundary conditions play a very crucial role in numerical computations, with the type of computations the boundary conditions are to be assigned to achieve the best possible reality aspects in the computations. In this case the conventional boundary conditions suffice to capture the details. The assigned boundary conditions are shown in the Fig 3.

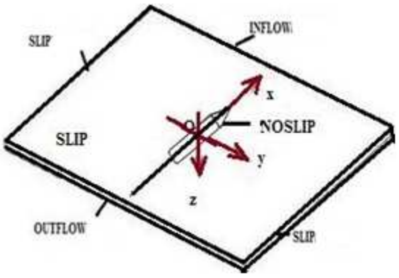


Figure 3. Boundary conditions

2.1 (d) Turbulence modeling:

The solver provides EASM, kω SST and kω BSL turbulence models for resolving the closure problem. The choice of an appropriate model confirms the accuracy of the computation. Although EASM works well with XCHAP, the kω SST was chosen which works better in capturing the desired physics in the type of flows that is dealt with here.

3 CASE STUDY

3.1 SHIP PARTICULARS

The investigation was carried out for famous benchmark ship KVLCC2 tanker model scale. The paper studies the lateral and vertical restriction problem for bare hull. The principal particulars of the full scale and model scale ship is given in the table.

Table 1. Full scale and model scale particulars

PARTICULARS	FULL SCALE	MODEL SCALE
Length (L_{pp})	320 m	4.97 m
Breadth (B)	58 m	0.90625 m
Draft (T)	20.8 m	0.325 m
Block Coefficient (C_B)	0.8098	0.8098
Vessel Speed (U)	7.6 m/s	0.984 m/s

3.2 SHALLOW WATER EFFECT

The study was carried out for shallow water condition predicts the forces and moments acting on the ship when moving through a harbor, port or canals. The simulations were carried out for a variable depth to draft ratio. The simulations reveals that the change in depth brings an evident transformation in the ship behavior as presented in [7]. The Fig 4 shows the resistance plot which clearly depicts the inverse relation of depth and resistance.

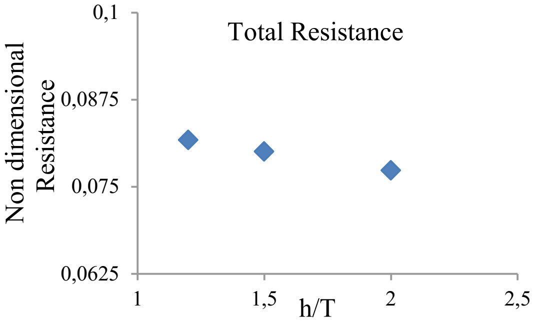


Figure 4. Resistance variation with h/T

3.3 BANK EFFECT

The ship bank interaction investigations were carried out for two variants of canals. The computations did not consider wave as the ship moved at a very low speed of 0.384m/s (6knots for full scale). The canal wall close to the ship remained vertical and the canal wall away from the ship was modified to a sloped wall. The effect of change in shape of the canal wall away from the ship was taken up to investigate possibilities of its influence on ship behavior. The depth of the canal was fixed at h/T-1.5. The two shapes of the canal used for investigation are shown in Fig 1.

3.4 RESULTS

The simulations were carried out for a Froude number of 0.055 and a Reynolds number of $1.926*10^6$.The results for a fixed water depth and two ship bank clearances are given in this section. The sway force (Y'), yaw moment (N'), rolls moment (K') and resistance measured in the numerical test are presented.

3.4 (a) Canal 1

The simulations investigate two ship bank clearances with y_b/L_{pp} of case (1) 0.237 and case (2) 0.264. An attempt to measure the forces and moments acting on the ship was made in the paper. The results show a drop in sway force, resistance and roll moment with increase in the clearance whereas the yaw moment increase with the increased clearance. This can be credited to the pressure distribution which changes with the distance from the canal wall. The pressure distribution on the hull shows in Fig 6 the unsymmetrical distribution on the ship bottom for case1 whereas the distribution seems more or less symmetrical for case 2.

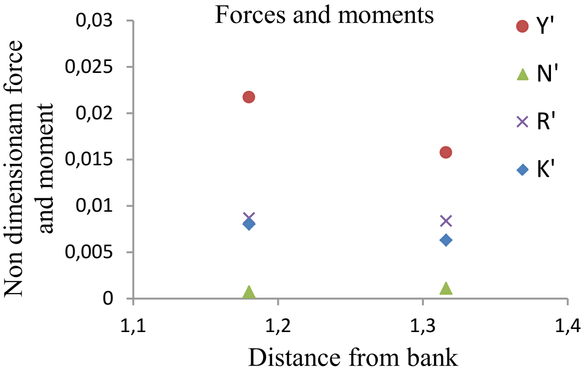


Figure 5. Force and moment variation with change in ship bank clearance

Figure 6 are depicts the pressure distributions on the hull when placed in canal 1with two different bank clearances The pressure distribution around the hull projects the suction effect caused due to the high velocity and low pressure in the region closer to the canal wall. The side of the hull closer to the bank wall shows drop in pressure as compared to the side away from the bank wall. It can also be visualized that as the bank clearance increases the severity of low pressure subsides. The effect of lateral blockage is seen not only on the sides but also notable on the hull bottom.

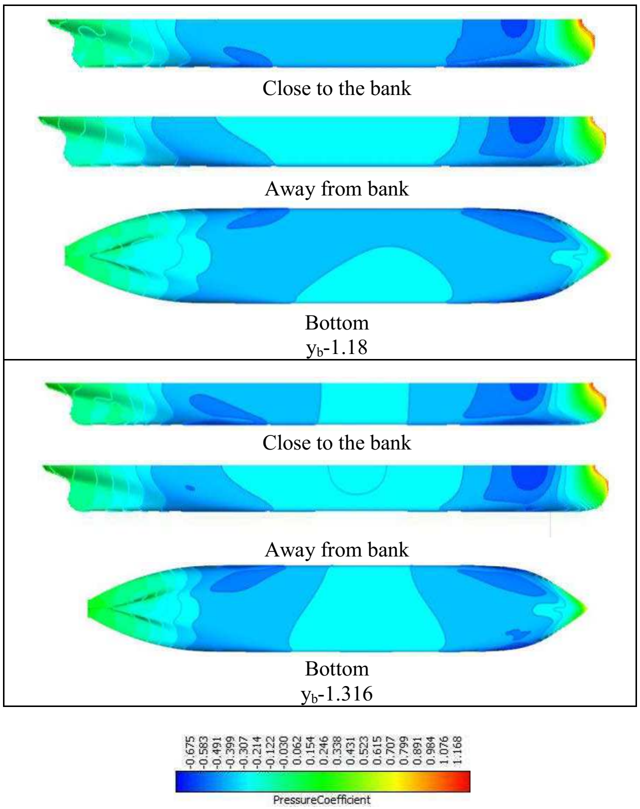


Figure 6. Pressure distribution on the hull

Another factor worth noting is the effect of the flow pattern change at the propeller location. The propeller performance is reflected by the pattern of the flow coming from the hull. The amount of wake in which the propeller works characterizes its performance. The flow reversals at the propeller plane can render the propeller producing zero thrust for half of its rotation. In order to avoid such objectionable situation assessment of the flow at the propeller plane is necessary. This will aid the understanding of the flow behavior and help make appropriate design changes so as to avoid vibrations, noise etc. caused by flow separations. The predicted flow pattern at the propeller plane is presented in Fig 7.

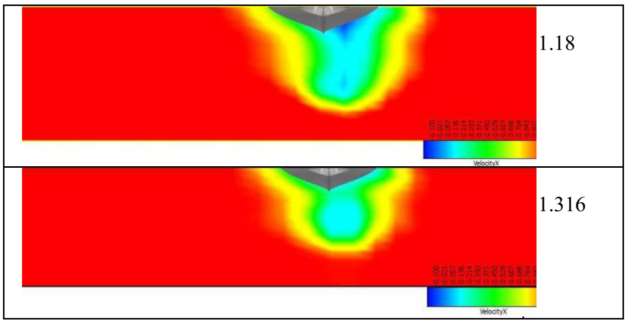


Figure 7. Axial velocity at the propeller plane

The wake distribution is unsymmetrical about the centerline. The flow separation is higher towards the region closer to the bank. The suction effect observed along the length of the hull also prominently affects the flow to the propeller. This will hinder the propeller performance and also affect the rudder effectiveness and make the ship stubborn to turn. Moving away from the canal wall relieves the severity of flow separation. It can be inferred that the closer the ship to the canal wall more probability of collision due to pressure drop and lowered propeller and rudder performance credited to the zero or negative axial flow at the propeller. So an optimal distance from the wall must be maintained to avoid any undesired outcome.

3.4 (b) Canal 2

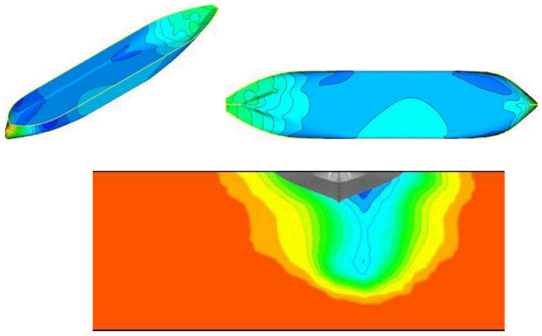
Similar tests were carried out for a ship placed in a canal with surface piercing wall with slope 1:4. Will this change of slope on the wall away from the ship bring about any change in the ship behavior? To get a clarification on this, a starboard clearance of 0.237 L_{pp} was investigated for h/T=1.5. The forces and moments calculated numerically for both canals are compared and presented in the table 2.

Table 2. Force and Moment comparison

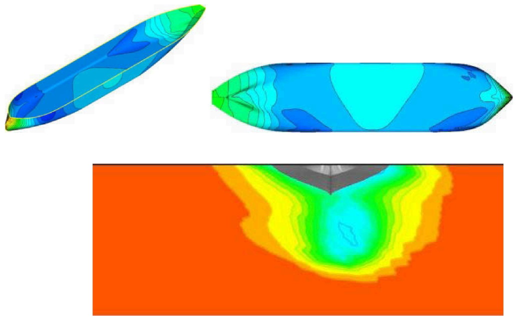
Yb=1.18	Non Dimensional values			
	Y	N	K	CT
Canal 1	2.17E-02	-7.04E-04	8.02E-03	8.67E-03
Canal 2	2.09E-02	-7.86E-04	7.94E-03	8.56E-03
Percentage difference	-3.47%	11.65%	-1.06%	-1.30%

It is evident from the results presented that the forces acting on the hull are more of a function of its distance from the wall closer to it. The table shows the yaw moment variation is about 11.6% and roll moment of about 1%, sway force variation is 3.5 % and the resistance variation of 1.3%. So a shape change of the wall away from the ship alters the sway force and yaw moment predominantly which plays an important role in maneuverability aspect of the ship. The pressure distribution shown in Fig 8 on the hull asserts the change observed. The unsymmetrical distribution seen about the midship section in canal 1 becomes more symmetrical when placed in canal 2 modifying the moments acting on the vessel.

The flow pattern is affected by the sloped wall. The sloped canal wall affects the hull bottom pressure and the axial velocity distribution at the propeller plane. The axial flow onto the propeller is better with the sloped canal than vertical canal wall. The intense drop in axial flow onto the propeller is eased by the sloped wall. The sloped canal wall although away from ship has not much effect on the magnitude of forces and some moments acting on the hull; it influences the overall flow characteristics around the ship.



Canal 1



Canal 2

Figure 8. Pressure distribution on the hull and Axial velocity at the propeller plane comparison for canal 1 and canal 2 for y_b-1.18

4 SQUAT

Ship passing a confined water pushes the water ahead of her, this water flows through the sides or the bottom of the ship. This flow pushes itself through the limited space available squeezing the streamlines together increasing the flow velocity. The water level around the hull reduces and this drops the vessel vertically down. This sinking is squat. Squat has existed forever but for the past forty years the increase in vessel size and speed has led to the reduction of under keel clearance to around 1.0 to 1.5 m alarming increased possibility of grounding in shallow and confined conditions.

The under keel pressure distribution at the ship centerline is compared for lateral restriction and open waters respectively.

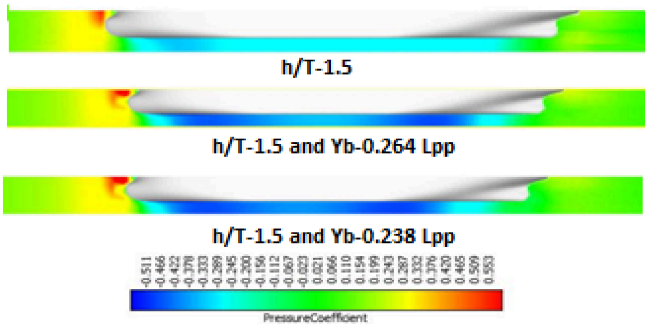


Figure 9. Pressure distribution below the ship

There is a remarkable drop in the pressure with the lateral restriction. The severity increases with the ship moving closer to the canal wall. The low pressure region confined to bow area with shallow depth spreads to 75% of the vessel length with both vertical and lateral restrictions.

Sinkage predictions were carried out for the canal 1 using the potential method for a fixed h/T=1.5 and ship bank clearance of 0.264 L_{pp} and 0.238 L_{pp} at different speeds. It is observed that the reduction of the bank clearance by 9.8 % has zero impact on sinkage. The speed is a deciding factor on the amount of sinkage the vessel undergoes. The higher the speed the higher is the squat shown in Fig 10.

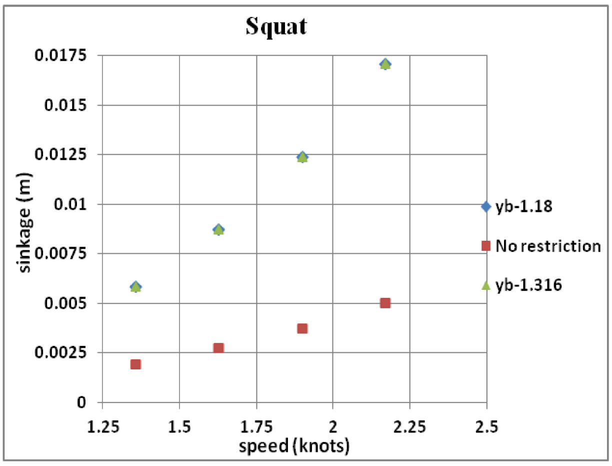


Figure 10. Squat predicted using potential method

5 SCALE EFFECT ON WAKE

The flow phenomenon is most sensible to the scale effect at the propeller. The computations were carried out for a scale of 1:64 which confines the output to the model scale; the scale effect will lead to wrong predictions at the full scale. The measurement of flow velocity observed at the propeller plane is a clear indication of the wake field observed at that plane for that scale. The wake fraction is measured as

$$w = 1 - \left(\frac{v_a}{U}\right)$$

The axial velocity plot at the propeller plane is an indication of the axial velocity into the propeller v_a and the probable wake. In order to investigate the scale effect the simulations were carried out for three different scales 1:110, 1:64 and full scale. Observation of the axial flow pattern indicates the variation in wake fraction with scale. The observed flow pattern at the propeller plane is shown in the Fig 11.

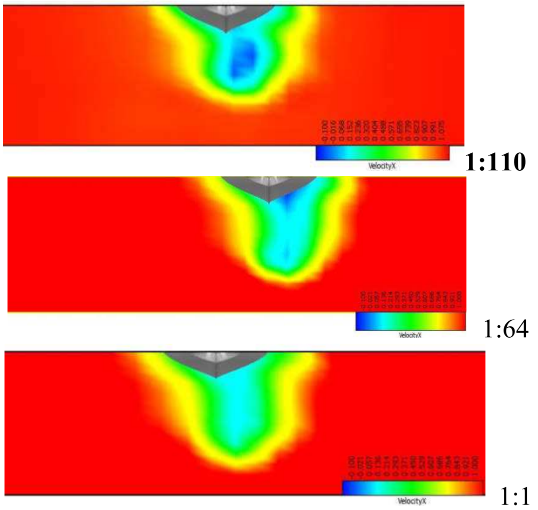


Figure 11. **Axial velocity comparison for different scale**

The figure 11 shows the changes in the axial flow pattern with scale change, this confirms reduction in the intensity of negative flow with increasing scale ratio. It can therefore be interpreted from the simulation results that the wake on full scale is less intense than the model and hence lower is the risk of vibrations or noise induced. The scale effect influences the scaled up results and must be accounted when scaling the results to full scale.

6 CONCLUSION

The investigation for lateral and vertical restrictions in the flow leads to the following conclusions.

1) The forces and moments shoot up with the addition of lateral restriction to the prevailing vertical blockage. This complicates the flow around the hull and renders the ship sluggish and stubborn making it difficult to maneuver.
2) The slope change of the bank wall away from the ship has a nominal effect on the forces acting on the hull but affects the yaw moment also the flow characteristics are modified.
3) The closer the ship to the canal wall higher the suction and chances of collision.
4) The axial flow to the propeller is also affected by the movement of the ship closer to the wall. The closer the ship to the wall the higher is the reversed flow observed, unsymmetrical wake at the propeller plane deteriorates its performance and reduces the thrust producing capacity at a given rpm.
5) Squat comparison for open water and lateral restriction shows indisputable increase with reduction in the ship bank clearance. Closer the ship to the canal wall higher is the squat.
6) Speed plays a vital role in measuring the sinkage of the ship. The change of ship bank clearance by 11% does not bring any effect on the sinkage and hence the squat.
7) Flow visuals provide a very close look into the actual pattern of the flow and guide the designers to accommodate all the design changes with utmost care and clarity.
8) The scale effect investigation leads to a conclusion that the wake is sensitive to scale effect and must be accounted for applying on full scale.

7 ACKNOWLEDGEMENTS

The authors deeply acknowledge and thank Björn Regnström (FLOWTECH) for his constant support and suggestions in resolving the issues faced. Also authors thank Michal Orych (FLOWTECH) for his valuable inputs. Authors also express gratitude to Sharad Dhavalikar (Indian register of shipping) for his frequent inputs and suggestions.

8 REFERENCES

1. Bertram, V. (2000) *Practical Ship Hydrodynamics*, Butterworth Publications.

2. Debaillon, B.; Lataire, E.; Vantorre, M. (2009). Comparison of bank effects on ship squat between experimental measurements and a numerical modeling system, *International Conference on Ship Manoeuvring in Shallow and Confined Water: Bank Effects*, Antwerp, Belgium: pp. 31-38.

3. Nakisa, M.; Maimun, A.; Sian, A.Y.; Ahmed, Y.M.; Priyanto, A.; Jaswar; Behrouzi, F. (2013). Three-dimensional numerical analysis of restricted water effects on the flow pattern around hull and propeller plane of LNG ship, *International journal of mechanics*, Vol. 7(3): pp. 234-241.

4. Poojari, D.; Janardhanan, S.; Kar, A.R. (2014). Maneuverability Assessment of a container ship using Steady RANS method **,** *Proceedings of the Twenty-fourth (2014) International Ocean and Polar Engineering Conference*, Busan, Korea**.**

5. Zou, L.; Larsson, L.; Delefortrie G.; Lataire E. (2011). CFD Prediction and Validation of Ship-Bank Interaction in a Canal. In: *Proceedings of the 2nd International Conference on Ship Manoeuvring in Shallow and Confined Water, Trondheim, Norway*.

6. Poojari, D.; Saj, A.V.; Janardhanan, S., Kar, A.R. (2014). Effect of Environmental Loads on the Maneuverability of a Tanker. *International Conference on Computational and Experimental Marine Hydrodynamics, MARHY* 2014, Chennai, India.

7. Janardhanan, S.; Krishnankutty, P. (2010). Estimation of sway-velocity based hydrodynamic derivatives in surface ship maneuvering. *International Journal of Ocean and Climate Systems,* Vol 1, No 3 and 4, pp 167-178.

8. Janardhanan, S.;Krishnankutty P. (2009) Prediction of Ship Maneuvering Hydrodynamic Coefficients Using Numerical Towing Tank Model Tests, *12th Numerical Towing Tank Symposium*, Cortona, Italy..

9. Poojari, D.; Saj, A.V.; Janardhanan, S.; Kar, A.R. (2015). Numerical Captive Model Tests and Trajectory Prediction for Ship Maneuverability in Shallow Water*, ICSOT INDIA: Coastal and Inland Shipping.*

10. Zou, L.; Larsson, L. (2013). Confined water effects on the viscous flow around a tanker with propeller and rudder. *International Shipbuilding Progress* 60(1): 309-343.

11. Broberg, L.; Regnstrom B.; Ostberg M. (2013). *SHIPFLOW Theoretical Manual*, FLOWTECH International AB., Gothenburg, Sweden,

12. Briggs, M.J.; Kopp, P.J.; Ankudinov, V.; Silver A.L. (2011). Ship Squat Comparison and Validation Using PIANC, Ankudinov and BNT Predictions. In: *Proceedings of the 2nd International Conference on Ship Manoeuvring in Shallow and Confined Water, Trondheim, Norway*.

13. Barass, B.; Derrett, D.R. (2006). *Ship Stability for Masters and Mates, Sixth Edition*

14. ITTC Recommended Procedures (2002), Manoeuvrability- Captive model test procedure," *Proceedings of 23rd ITTC.*

15. Pinkster, J.A.; Bhawsinka K. (2013). A real-time simulation technique for ship-ship and ship-port interactions. *The 28th International Workshop on Water Waves and Floating Bodies (IWWWFB 2013).*

9 AUTHORS' BIOGRAPHIES

Deepti B Poojari (Deepti.Poojari@irclass.org) currently holds the position of Surveyor (Hydrodynamics Group-Research & Innovation cell) at Indian Register of Shipping. She leads the manoeuvring research team and is involved in other consultancy work carried out in the division.

A R Kar (kar@irclass.org) holds the position of Senior Principal Surveyor at Indian Register of Shipping. He is the Head of the Dept for Hydrodynamics and Risk & Reliability Group-RIC division.