[UCI]I804:24011-200000263904

2013년 8월
석사학위 논문

년 월 석박사학위논문

A comparative study between the result of theoretical calculation and model test for the performance confirmation of "Crown Duct"

"Crown Duct"의 성능검증을 위한 이론계산 및 모형시험의 결과비교 연구

조선대학교 대학원
선 박 해 양 공 학 과
조 은 비

성 명

A comparative study between the result of theoretical calculation and model test for the performance confirmation of "Crown Duct"

"Crown Duct"의 성능검증을 위한 이론계산 및 모형시험의 결과비교 연구

2013년 8월 23일

조선대학교 대학원

선 박 해 양 공 학 과

조 은 비

A comparative study between the result of theoretical calculation and model test for the performance confirmation of "Crown Duct"

"Crown Duct"의 성능검증을 위한 이론계산 및 모형시험의 결과비교 연구

지도교수 이 귀 주

이 논문을 석사학위신청 논문으로 제출함

2013년 4월

조선대학교 대학원

선 박 해 양 공 학 과

조 은 비

조은비의 석사학위논문을 인준함

위원장	조선대학교 교 수	박 제 웅 (인)
위 원	조선대학교 교 수	이 귀 주 (인)
위 원	President of FTC	TAMASHIMA Masahiro(인)

2013년 5 월

조선대학교 대학원

Contents

List of Tables
List of Figures
ABSTRACT

1. Introduction..................................1

2. Development of New Type Energy-Saving Device "Crown Duct"

2.1 Structural Configuration of Crown Duct..................3
2.2 Evaluation of strength of Crown Duct....................5
2.3 Evaluation of strutural strength and vibration......5

3. Model Test 13

3.1 Test facility......................................13
3.2 Hull model...14
3.3 Propeller model....................................14
3.4 Extrapolation by ITTC - 78 and Modified ITTC - 78..15
3.5 Result of model test...............................17

4. SHIPFLOW ANALYSIS............................19
4.1 XFLOW...20
4.2 XMESH...21
4.3 XPAN..21
4.4 XBOUND..22
4.5 XGRID...22
4.6 XCHAP...23

5. Theoretical calculation 23
5.1 Resistance Calculation Result......................25
5.2 Self Propulsion Calculation Result.................27

6. Conclusions..................................40

References

List of Tables

Table 3.1 Main particulars for model propeller..........................15
Table 3.2 At Full load condition.......................................17
Table 3.3 At Ballast condition...17
Table 5.1 Main Particular..24
Table 5.2 Comparison of resistance components..........................25
Table 5.3 Comparison of self propulsion components.....................27
Table 6.1 Comparison of CFD calculation results with the result of model test...40

List of Figures

Fig. 2.1 configuration of Crown Duct..4
Fig. 2.2 Structural illustration and symbols of Crown Duct.......................4
Fig. 2.3 Structural calculation for the case of symmetric load based on Casliglians theorem..7
Fig. 2.4 Crown Duct pressure on 9 o'clock blade..........................8
Fig. 2.5 Slamming model 9 o'clock of Crown Duct..........................9
Fig. 2.6 Structural model for fluttering vibration.......................11
Fig. 2.7 Rotational vibration and symbols................................12
Fig. 2.8 Vibration of 1.5 and 10.5 o'clock blades harmonized with the circumferential blade vibration.......................................13
Fig. 3.1 Towing Tank of SSPA...14
Fig. 3.2 Ship model with "Crown Duct" and propeller......................16
Fig. 3.3 Comparison of P_D...18
Fig. 4.1 Panel generation from an offset point distribution.............21
Fig. 4.2 Grid surrounding the aft half...................................22
Fig. 5.1 Design of Crown Duct..24
Fig. 5.2 Comparison of resistance components.............................26
Fig. 5.3 propulsion coefficients...27
Fig. 5.4 Wave pattern around the 50K Tanker..............................28
Fig. 5.5 Comparison of stream line 29
Fig. 5.6 Comparison of flow calculations.................................30
Fig. 5.7 Comparison of the stern flow....................................31
Fig. 5.8 Comparison of the direction of rotation speed...................32
Fig. 5.9 Comparison of tangential velocity...............................33
Fig. 5.10 Comparison of Transverse speed.................................34
Fig. 5.11 Comparison of Wake fraction....................................35
Fig. 5.12 Comparison of V_{tr}...36
Fig. 5.13 Comparison of Wake...37
Fig. 5.14 Comparison of Wake in $U_{x=1}, U_{x=3}$.......................38
Fig. 5.15 Comparison of Wake in $U_{x=4}, U_{x=5}$.......................39

ABSTRACT

“Crown Duct” 의 성능검증을 위한 이론계산 및 모형시험의 결과비교 연구

Jo, Eun bi

Advisor : Prof. Lee Kwi-Joo, Ph. D.

Department of Naval Architecture

& Ocean Engineering

Graduate School of Chosun University

본 논문에는 연료절감형 부가물인 “Crown Duct”의 성능검증을 위한 이론계산 및 모형시험의 결과를 비교하였다. SPP와 공동 개발한 “Crown Duct”를 부착하여 모형테스트와 CFD code “SHIPFLOW”를 이용하여 부가물 부착 시 성능 계산을 하였다.

“Crown Duct”는 부가물로 인한 저항을 최소화하기 위하여 Stator의 크기를 최소화하고 유동의 방향을 적절히 유도할 수 있도록 프로펠러 반경의 직경을 갖는 Semi-Duct를 혼합한 부가물이다. Stator는 유동계산결과를 참조하여 설계하였고, Semi-Duct는 NACA 섹션을 고려하여 설계하였다.

본 연구의 대상선형으로 50K tanker가 적용되었고, Duct의 Diffusive Angle이 다른 2가지 타입의 “Crown Duct”가 제작되어 스웨덴 SSPA의 예인수조에서 모형시험이 실시되었다.

1. Introduction

The more energy it consumes and energy saving technologies are essential for developments of green ships. Solution to this problem is, in the early stage, to design a hull form having not only the minimum resistance but also the optimum propulsive efficiency. However, the hull forms of general ships bear long term experience of operations which is not expected to be improved radically. Thus, it is considered more appropriate to attach energy saving devices rather than to improve the hull form directly. The effects of the appendages upon the wake distributions at a propeller plane are also investigated to clarify the effectiveness of them on the equalizations of the distributions, which are closely related to the ship propulsive efficiencies. It is found that appendage attached vertically to the ship stern can accelerate flow in the region, resulting equalization of the wake distribution. The results have been compared to other CFD and experimental results.

Traditionally, hull form designs have based on hulls already in service and known to perform well, with any changes to the design being investigated using expensive model towing tank tests. However, In recent years advances in computational fluid dynamics have made possible the analysis of new hull forms at a fraction of the cost of model tests, with good estimates of the hydrodynamic forces acting on the vessel being obtained (Van Oortmerssen 1990).

The use of computational techniques requires a numerical description of the hull shape. Various methods of defining the complex free-form shape of hulls for use in design optimization methods can be found in the literature(Lin et al 1963, Wyatt & Chang 1990, Larsson & Kim 1992, Lowe et al 1994).

This thesis placed between the result of theoretical calculation and model test for the performance confirmation of "Crown Duct" at its analytic center.

Vortices generated from the bilge at stern moves upward on both ship sides and moves downward near the center plane of the ship, leading to difference angle of attack of propeller blades on port and starboard side which may reduce the propeller efficiency. Against the stern flow mechanism, the stator fins generally plays a role as flow guiding device making a wake field near the upper part of the propeller plane uniform by both deflecting flow toward propeller and reducing the wake peak at propeller top position.

Conventional pre-swirl stator consists of several fins which have almost the same san length as the propeller radius and are fixed radially on the stern frame in front of the propeller. It is known that the wake factor (1-w) decreased considerably due to reflecting flow generated by stator fins in front of the propeller plane. But high stress fins concentrated at the standing point of side fin in heaving and pitching motion, and non-negligible added resistance due to this is the problem to be solved.

The method of concept and structural configuration of Crown Duct is described in Section 2. In Section 3 the method used to calculate the appendage attached a ship pressure distribution by SHIPFLIOW, Model Test is addressed in Section 4. Finally a comparative results of CFD calculation with model test in Section 5.

2. Development of New Type Energy-Saving Device "Crown Duct"

New type of complex stator having less added resistance and less stress on the stand point of side fins than conventional pre-swire stator, has been developed by combination of semi-duct with minimum blades. Small sized semi duct having diameter almost equal to 50% of propeller diameter is attached to the tip of the short horizontal fin which is designed to decrease the tip vortex and to generate the uniform flow to the propeller. this new energy-saving device consists of three fins on the stern frame and three plates on the duct.

In the following, I will try to show that structural configuration of Crown Duct.

2.1 Structural Configuration of Crown Duct

Fig 2.1 Shows the configuration of crown Duct. The structure is composed of one semi-duct and of five straight blades as two horizontal, one vertical and two inclined wings.

Fig 2.2 shows the frame configuration of Crown Duct of which blades are located at 3,9,12,1.5 and 10.5 o'clock in radial direction and with additional semi-duct.

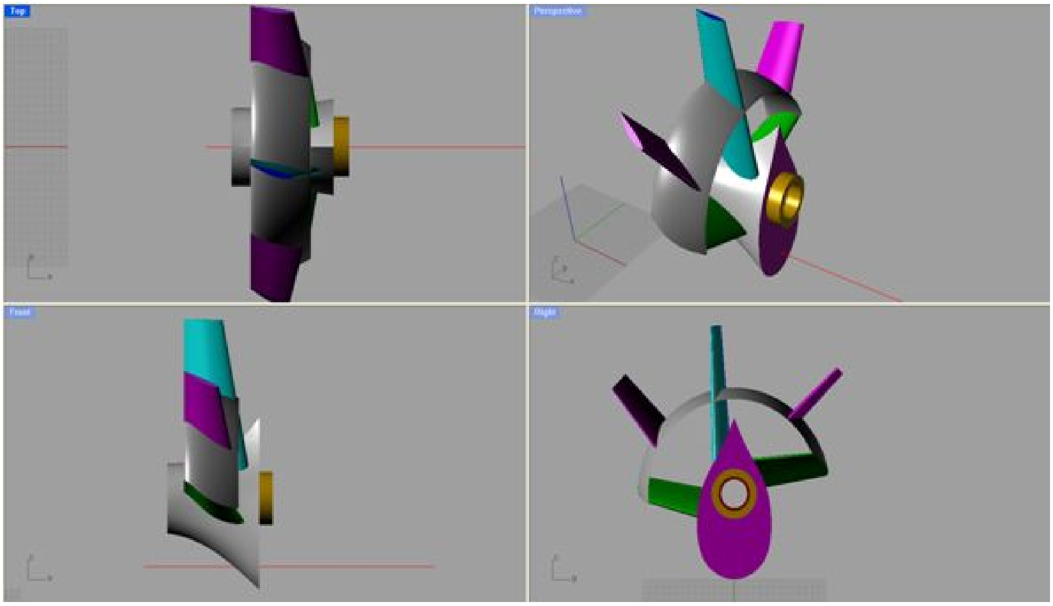


Fig. 2.1 configuration of Crown Duct

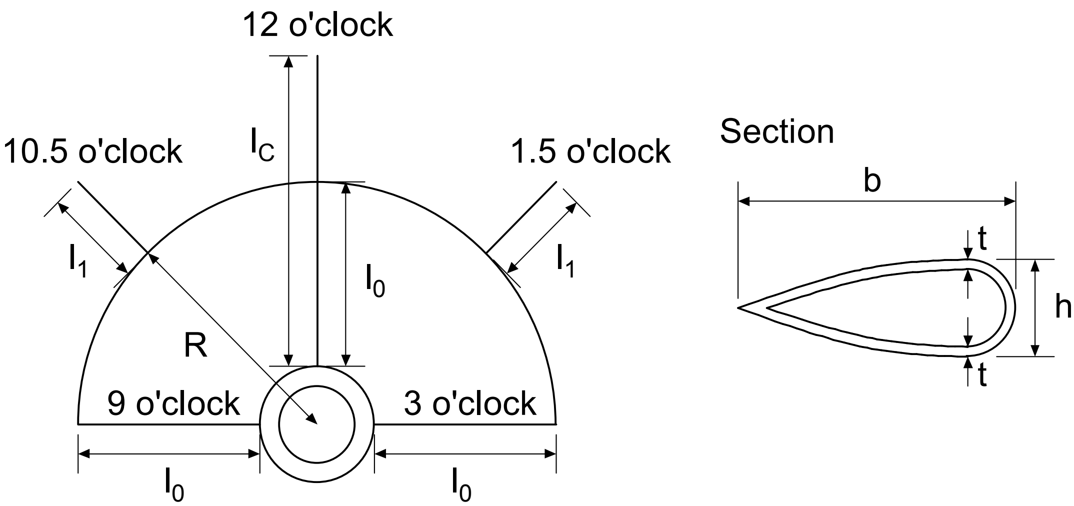


Fig. 2.2 Structural illustration and symbols of Crown Duct

2.3 Evaluation of strength of Crown Duct

Crown Duct is located at the boss of propeller. In case of damages of Crown Duct may bring serious damages of propeller. Normally structure calculation and analysis is carried out by numerical analysis. The evaluations of structure of Crown Duct are carried out through the fundamental theory of dynamics.

For the evaluation of forces and moments on structural elements, the following terms are taken into consideration

- Forces on hull in waves
- The propeller lift force
- The stress evaluation of structure based on the energy method
- The hydro dynamical pressure such as impact force is studied as a dynamic response of structure

Based upon the above mentioned cases a series of studies in strength and vibration has been performed. The numerical confirmation through FEM may be necessary in the stage of detail design of the structure.

2.4 Evaluation of strutural strength and vibration

- Hydro dynamical force and its distribution has been evaluated considering ship motion in waves
- Lift force on blades
- Impact pressure on blades

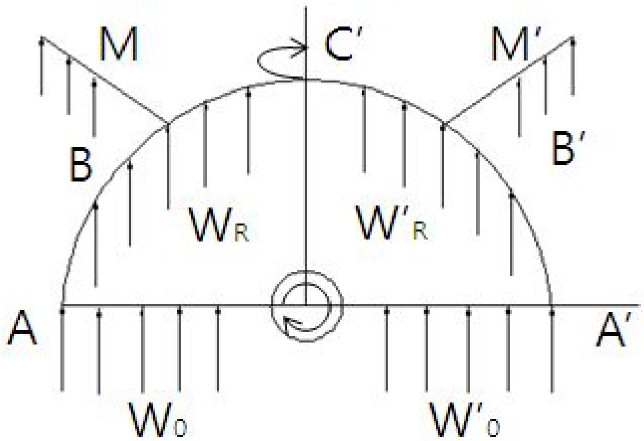
In the first stage of design, approximation method may apply to obtain the order of rigidities of structural elements. The approximated calculation method is introduced.

Confirmations of structural strength requirement has been done by energy method. The followings are examined theoretically.

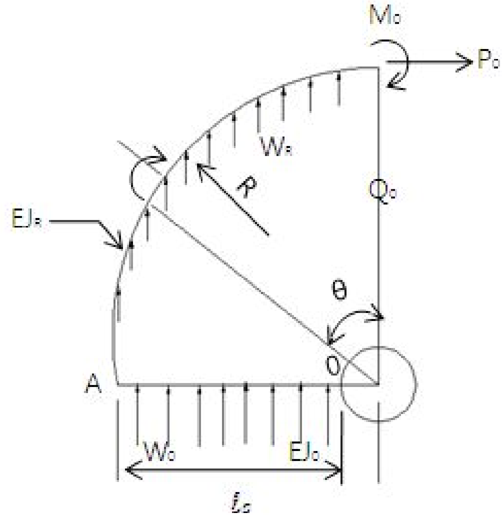
(1) Strength of subjected to symmetrical pressure

The pitching and heaving may bring symmetric pressure on blades of Crown Duct. For the above condition of motion, pressure calculation has been carried out based upon the Castigliano's theorem.

Distribution of pressure is shown in Fig 2.3



(a)Symmetrical distribution of load on Crown Duct



(b) Structural analysis model for the case of symmetric load

Fig 2.3 Structural calculation for the case of symmetric load based on Casliglians theorem.

The boundary forces, and moment is obtained by analysis. And using three boundary values, moment distribution in each blade can be obtained. Calculation can be carried out using the calculation sheet.

(2) Strength of structure in asymmetric pressure is shown in Fig 2.4. Due to rolling of the ship, the asymmetrical load will be induced in 3 and 9 o'clock blades. Most severe case of load for the case of asymmetrical distribution of the pressure on 9 o'clock blade is assumed as shown in Fig 2.4

In this condition of load, the asymmetric structure must be taken into consideration as shown in Fig 2.4 with the spring constant K at top part of duct. see calculation sheet.

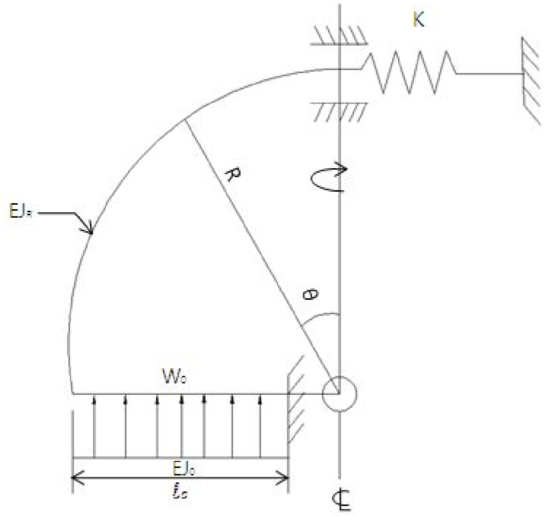
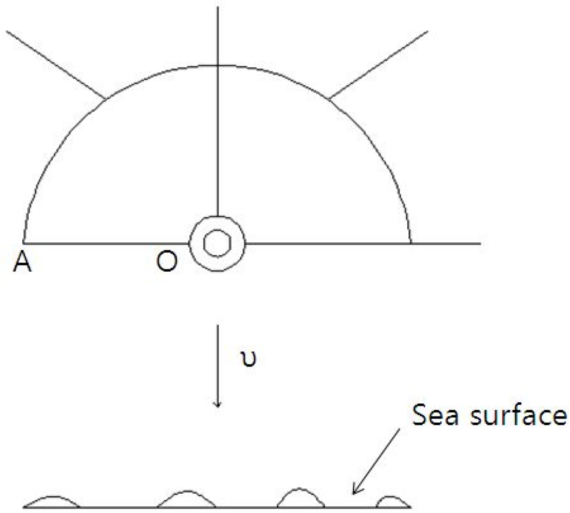


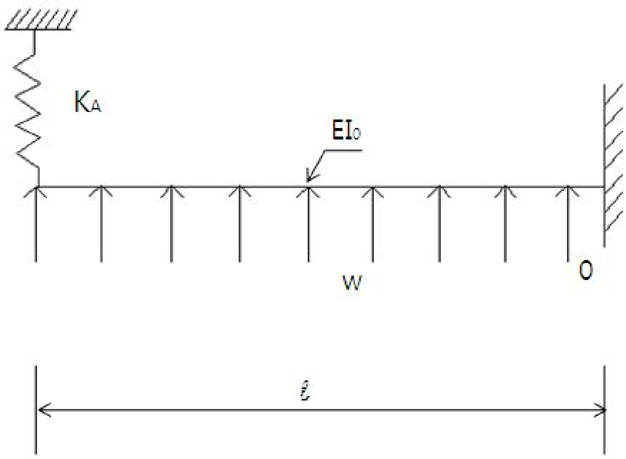
Fig 2.4 Crown Duct pressure on 9 o'clock blade

(3) Impulse structural response due to collision of blades to sea surface

A kind of stern slamming might be induced to the horizontal blade(3 and 9 o'clock). To obtain the response of structure against slamming load the dynamic response of blade must be calculated. The structural model is shown in Fig 2.5



(a) collision of duct to sea surface



(b) structural model

Fig 2.5 Slamming model 9 o'clock of Crown Duct

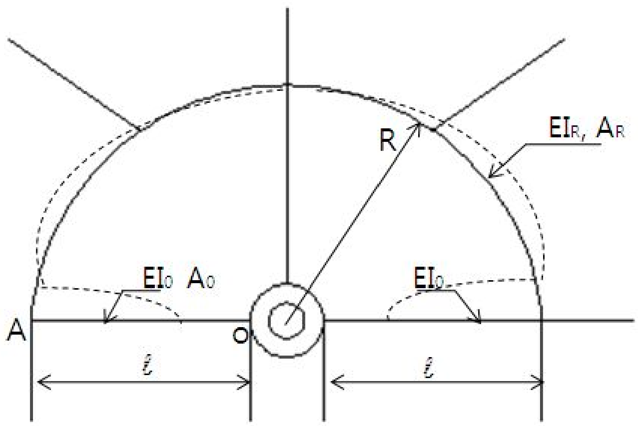
In order to do vibration free structural design of Crown Duct against propeller induced excitation force, following cases of calculation to be carried out.

- Fluttering vibration
- Rolling vibration
- Local vibration of 1.5 and 10.5 o'clock blade

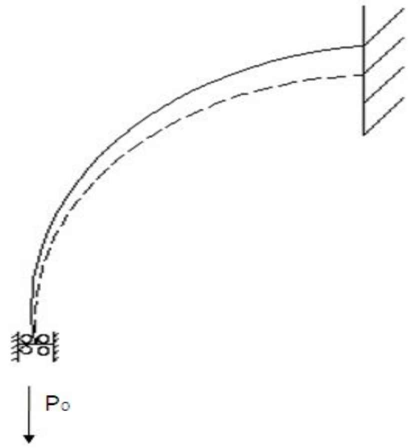
(1) Fluttering vibration

The vibration frequency calculation is carried out for the vibration mode as shown in Fig 2.6 (a).

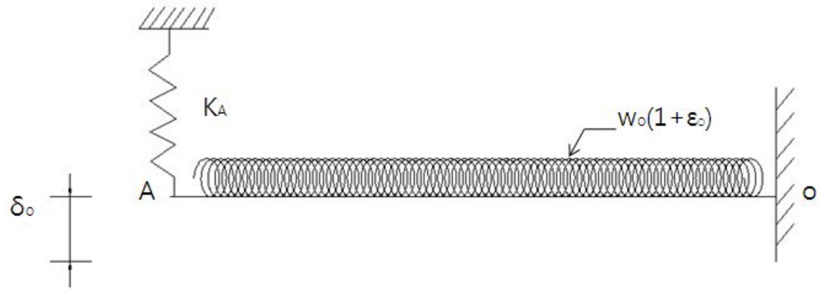
For the estimation of vibration in the mode of as shown in Fig 2.6 (a), the equivalent spring has been simplified as shown in fig 2.6 (b).



(a) Fluttering mode of vibration



(b) Model for estimation of vibration



(c) Structural model for estimation of vibration

Fig 2.6 Structural model for fluttering vibration

(2) Rolling vibration

As shown in fig 2.7, a rolling vibration may be induced in the structure, if no supporter is provided to the point C shown in the figure.

The natural frequency of rolling vibration shall be estimated. If the supporter is installed on the point e the vibration will be coincide with the fluttering vibration.

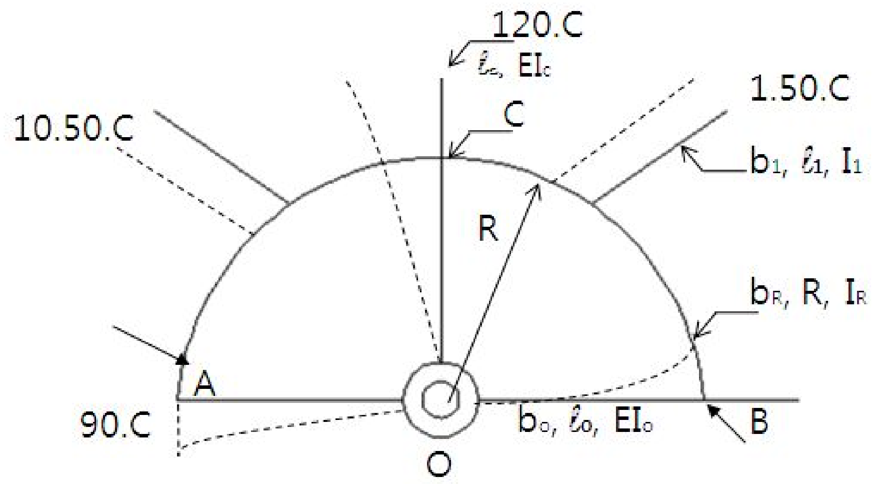


Fig 2.7 Rotational vibration and symbols

(3) Local vibration of 1.5 and 10.5 o'clock blade

The vibration of 1.5 and 10.5 o'clock blade will induce the deformation of duct as shown in Fig 2.8. The natural frequency of 1.5 and 10.5 o'clock blades are estimated in the condition as shown in the vibration mode of Fig 2.8

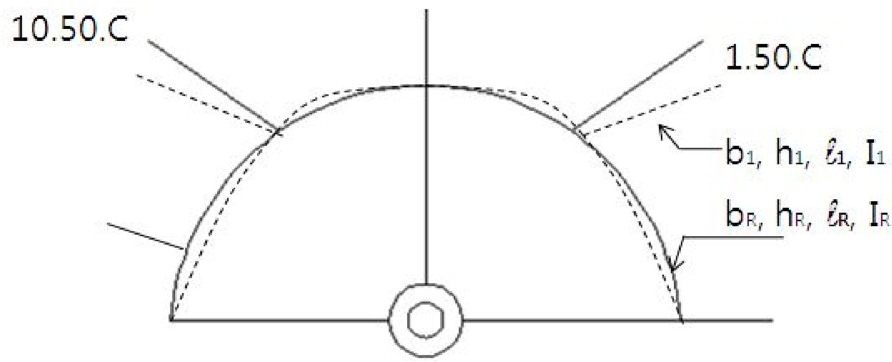


Fig 2.8 Vibration of 1.5 and 10.5 o'clock blades harmonized with the circumferential blade vibration

3. Model Test

By using a scale Model ship (1/25.417) of 50K tanker (Lpp = 174 m), we examine the effect of "Crown Duct". tests are carried out in the towing tank of SSPA.

3.1 Test facility

The towing tank has the following main particulars:

- Length 260 m
- Breadth 10 m
- water depth 5 m
- Maximum carriage speed V_{MAX}= 11 m/s

Fig. 3.1 Towing Tank of SSPA

3.2 Hull model

- Ship model was manufactured in divinycell, a foam plastic material.
- The model scale is 1:25.417
- The model has a turbulence trip wire at station 19.

3.3 Propeller model

The propeller model is fixed, right turning, four bladed propeller with dimensions according to Table 4.

Table 3.1 Main particulars for model propeller

Characteristic	Value
Diameter model scale	0.24 m
Diameter full scale	6.1 m
Pitch ratio P/D at r/R = 0.75	0.715
Blade area ratio A_D/A_o	0.53

3.4 Extrapolation by ITTC - 78 and Modified ITTC - 78

Without the Crown Duct mounted the predictions were made according to the 1978 ITTC extrapolation method. With the Crown Duct mounted a somewhat modified wake scaling was used.

The method has been discussed within the 1999 ITTC and tentatively accepted for evaluation of pre-swirl stator concepts. The wake scaling presumes that tests with the same propeller but without the stator have been performed as well. The difference between model effective wake with stator and the model wake without stator is considered as a potential wake created by the stator.

The hull potential wake and the frictional wake are scaled as for the model with out stator according to the normal ITTC - 78 method, to which the stator potential part is added. The amount 0.04 represents the potential wake created by the rudder at the location of the propeller.

Thus is in the modified ITTC 1978 extrapolation the full scale wake.

$$W_{Tsw} = (t_{wo} + 0.04) + (W_{Tmwo} - (t_{wo} + 0.04))*[(1+k)C_{Fs} + \triangle C_{Fs}/(1+k)C_{Fm}] + [W_{Tmw} - W_{Tmwo}]$$

where index

"W" stands for "with Crown Duct"

"wo" stands for "without Crown Duct"

"m" stands for "model"

"s" stands for "ship scale"

"T" stands for "thrust identity"

The form factor is based on the case without Crown Duct.

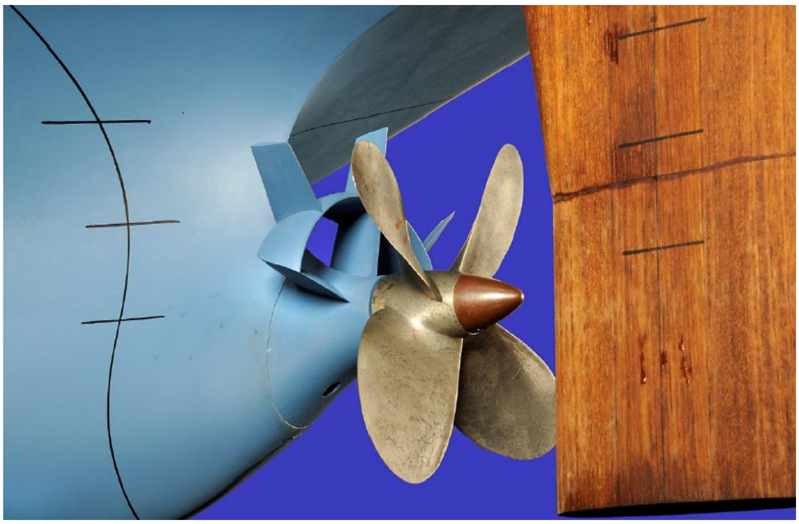


Fig. 3.2 Ship model with "Crown Duct" and propeller

3.5 Result of model test

Model test results at the Towing Tank show of 0.815% decrease of efficiency power compared to that a without Crown Duct. Maximum 3.4% decrease of delivered power at the design speed (15 knots) and 2.7% at Ballast condition.

Compared in Table 5 for full load condition and in Table 6 for ballast condition.

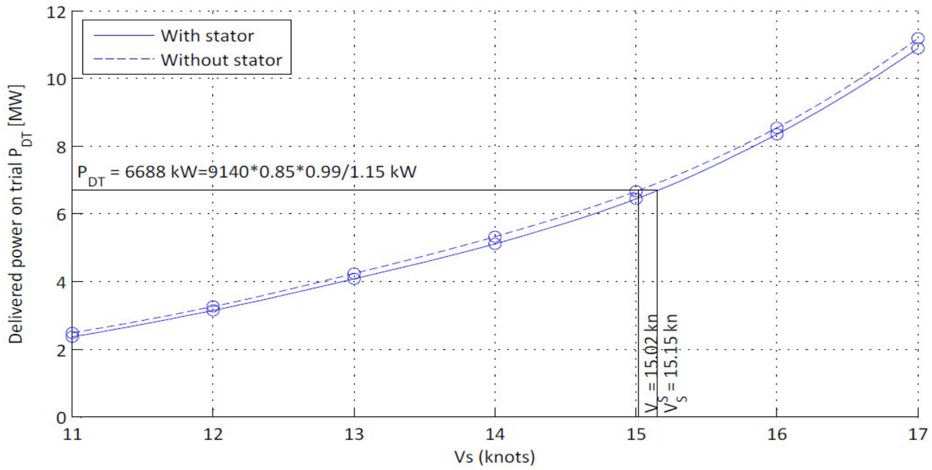
And in the Fig 4.3 shows, Horsepower is reduced in all speed range when attached the complex appendage.

Table 3.2 At Full load condition

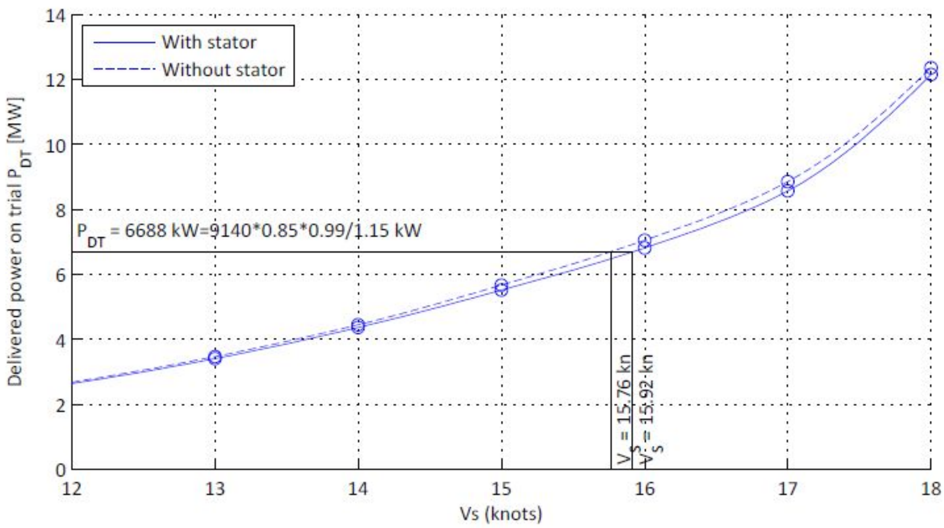
Condition	P_E(kW)	w	t	η_R	η_o	η_D	P_D(kW)
Bare hull	4783	0.342	0.205	1.013	0.557	0.689	6940
With CD	4744	0.451	0.252	1.022	0.501	0.707	6706
Diff.	-0.815%	+0.109	+0.047	+0.009	-0.056	+0.018	-3.37%

Table 3.3 At Ballast condition

Condition	P_E(kW)	w	t	η_R	η_o	η_D	P_D(kW)
Bare hull	4200	0.385	0.221	1.015	0.555	0.712	5899
With CD	4289	0.483	0.248	1.026	0.500	0.747	5742
Diff.	+2.12%	+0.098	+0.027	+0.011	-0.055	+0.035	-2.66%



(a) Comparison of P_D at full load condition



(b) Comparison of P_D at full load condition

Fig. 3.3 Comparison of P_D

4. SHIPFLOW ANALYSIS

Applications of computational fluid dynamics(CFD) to the maritime industry continue to grow as this advanced technology takes advantage of the increasing speed of computers. Numerical approaches have evolved to a level of accuracy which allows them to be used during the design process to predict ship resistance. Significant progress has been made in predicting flow characteristics around a given ship hull. Ship designers cause this information to improve a ship's design. However, not much effort has been dedicated to determining viscous drag, an important element in the development of a new design. The final checking and analysis of the bulb design is done in the CFD module SHIPFLOW. The wave making and frictional resistance as well as the flow r ound the hull for various bulb shapes have been calculated using SHIPFLOW. The flow around a body can be described mathematically as a function of fluid pressure and the three components of velocity. A set of governing equations of motions can be created, like the Navier-Stokes equations for turbulent flow, and solved in association with specific boundary condition. These equations are often complex to solve and rely on the use of Computational Fluid Dynamics(CFD). SHIPFLOW is a CFD tool specifically developed to solve marine related problems (SHIPFLOW, 1999).

In CFD analyses of marine vehicles, it is customary to use I, j and k to describe the grid dimensions, where I-direction is in the axial direction, j is normal to the body, and k is around the body's girth.

The following potential flow techniques are used in Zone 1 to predict pressures, velocities and streamlines. By assuming non-viscous(ideal) and irrotational flow the governing equations produced are the linear, partial differential Laplace equations based on mass continuity. The non-linear free-surface boundary conditions are linearised and solved by using an iterative process until satisfactory convergence is reached.

In Zone w the development of the boundary layer is investigated using momentum integral equations for the thin viscous layer along the hull. By ignoring cross flow in the boundary layer, which is created due to a pressure gradient in the vertical direction of the ship hull the results are ordinary differential equations which are solved by Runge-kutta techniques. The prediction cannot be used at the stern of a ship where a thick viscous region occurs due to convergence of the streamlines. Towards the stern of the vessel, Reynolds-averaged Navier-Stokes(RANS) equations along with mass continuity equations describe the flow in Zone 3. The solution of the complex Navier-Stokes equations requires a lot of computational time and is therefore restricted to the stern of the vessel only, where a denser panelization is created. The unsteadiness of the turbulent region is averaged out and instantaneous values of pressure and velocity are separated into a mean with fluctuations by the introduction of Reynolds stresses.

The programming is split into six modules and SHIPFLOW considers each module do no affect, for example, the second module. These six modules are listed below, in the order in which SHPFLOW assesses them

4.1 XFLOW

Defines the general physical properties of the surroundings, for examples the fluid, characteristics, initial ship position, ship speed, etc.

4.2 XMESH

Using the information from XFLOW, XMESH generates the panelization of the free surface and the vessel for use by the third module XPAN. The model can be viewed in the post processor.

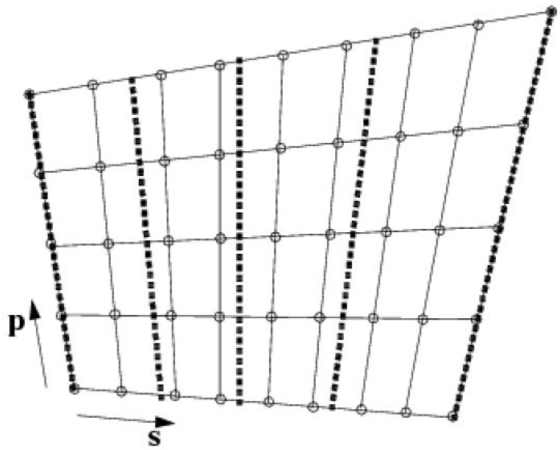


Fig. 4.1 Panel generation from an offset point distribution

4.3 XPAN

XPAN computes the potential flow around the model and free-surface, which are made up of quadrilateral panels each containing Rankine sources. XPAN can operate under linear or non-linear free-surface boundary conditions. Results obtained from XPAN are displyed by the post processor and listed in output files. The results include wave making coefficient(C_W), wave pattern, potential streamlines, pressure and velocity contours.

4.4 XBOUND

XBOUND is concerned with the thin turbulent boundary layer surrounding the hull. Using momentum integral equations SHIPFLOW provides the frictional resistance coefficient (C_f), boundary layer thickness, as well as other parameters associated with the boundary layer.

4.5 XGRID

XGRID generates the grid towards the stern of the vessel used to represent Zone 3 where the Navier-Stokes equations describe the fluid flow.

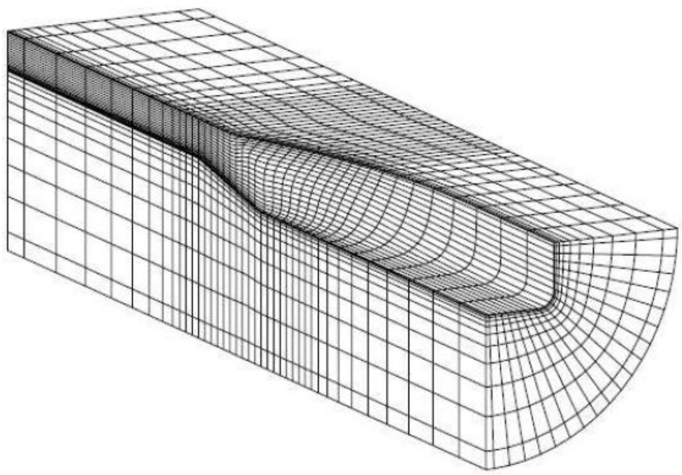


Fig. 4.2 Grid surrounding the aft half

4.6 XCHAP

The final module of SHIPFLOW solves the Reynolds-averaged Navier-stokes equations. XCHAP provides the viscous pressure resistance coefficient (Cvp) and therefore the total resistance C_t can be estimated. XCHAP can also be used to investigate the wake and values such as axial, radial and tangential velocities at carious planes towards the stern are obtained. The frictional, wave and total resistance coefficients as computed by SHIPFLOW, together with the total resistance as measured from the experiments and the Schoenherr and ITTC ship model correlation lines.

5. Theoretical Calculation

Although model test results are satisfactory, to minimize trial and errors and develop performance other ship with Crown Duct in the future. It is necessary to theoretical calculation by CFD CODE.

To begin with, I would like to examine design of "Crown Duct". In the preceding pages we have given an introduction to the concept "SHIPFLOW".

Computational Fluid Dynamics (CFD) is widely used in the ship design process. In particular during the initial design stage CFD has become an important tool. It enables the designer to evaluate a larger number of hull alternatives and thereby a better optimized and reliable design before the final validation. It is true that not only for new building but also for existing ships and retro fitting of ship energy saving devices. The tough competition on the shipbuilding market creates high demands on short lead times and competitive designs. This must be met by developments of effective CFD tools and integration with CAD(MEPD,2009).

CFD code makes cost down for the evaluation and prediction of performance of ship. Comparative theoretical calculation has been performed by CFD code of SHIPFLOW for design of Crown Duct configured as shown in Fig. 4.1

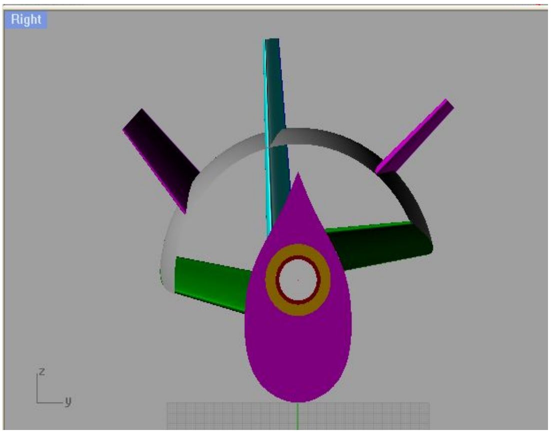


Fig. 5.1 Design of Crown Duct

The main Particulars of ship which has been simulated for the resistance and propulsion performance at design speed is shown Table 1.

Table 5.1 Main Particular

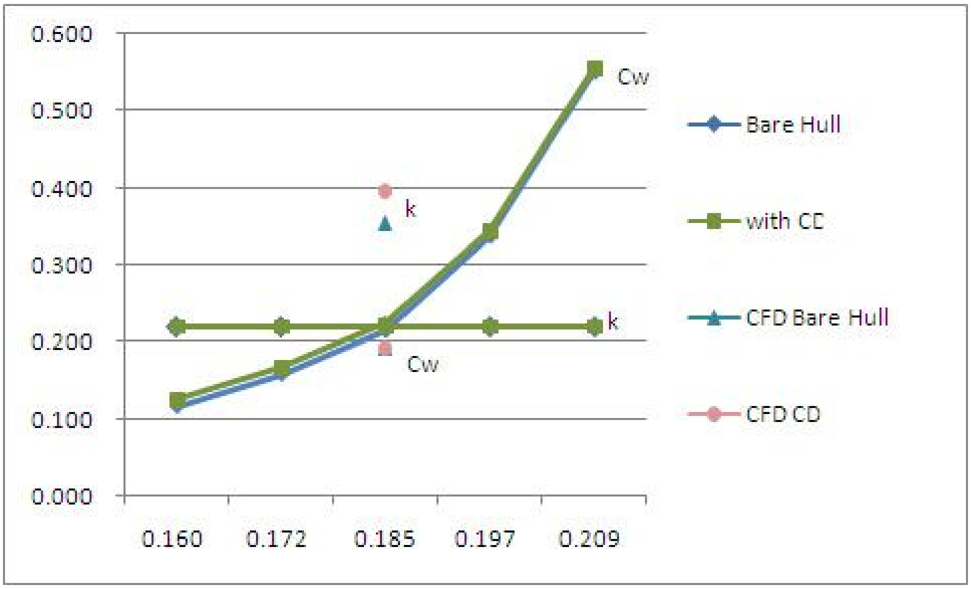
	Ship
Length between perpendiculars	174m
Length at load water line	178m
Breadth	32.2m
Design Draft	11m

5.1 Resistance Calculation Result

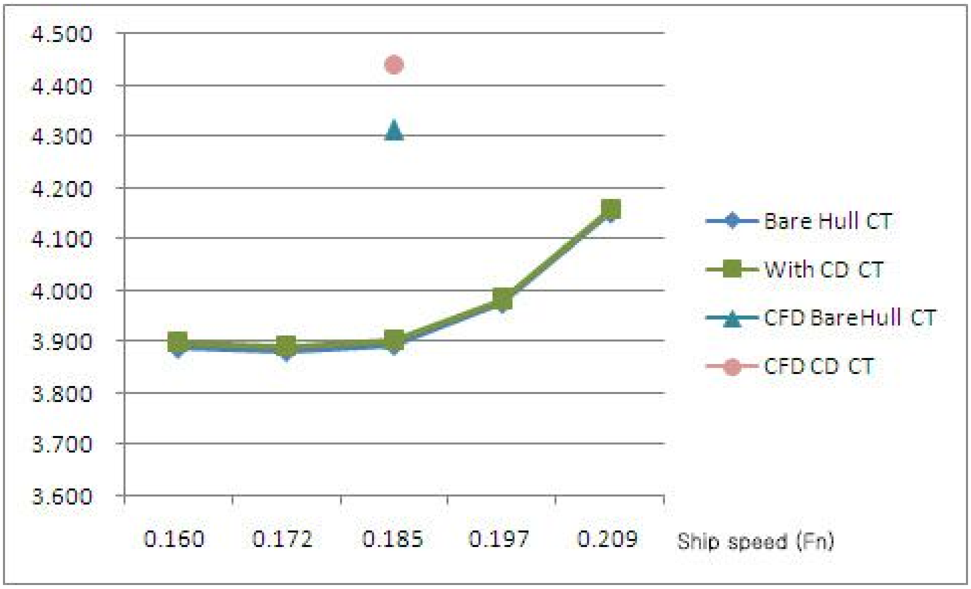
Resistance calculation results are compared in Table. 2 and in Fig 4.2

Table 5.2 Comparison of resistance components

	Bare hull	with CD
k	0.354	0.396
C_W	0.193	0.193
C_T	4.313	4.441



(a) Comparison of k, C_W



(b) comparison of C_T

Fig. 5.2 Comparison of resistance components

5.2 Self Propulsion Calculation Result

Increase of η_H and η_D compared to Bare hull. Self propulsion calculation results are summarized in Table.3

Table 5.3 Comparison of self propulsion components

	1-w	1-t	η_H	η_o	η_R	η_D
Bare hull	0.592	0.906	1.288	0.543	0.989	0.887
with CD	0.499	0.877	1.380	0.484	0.987	0.908

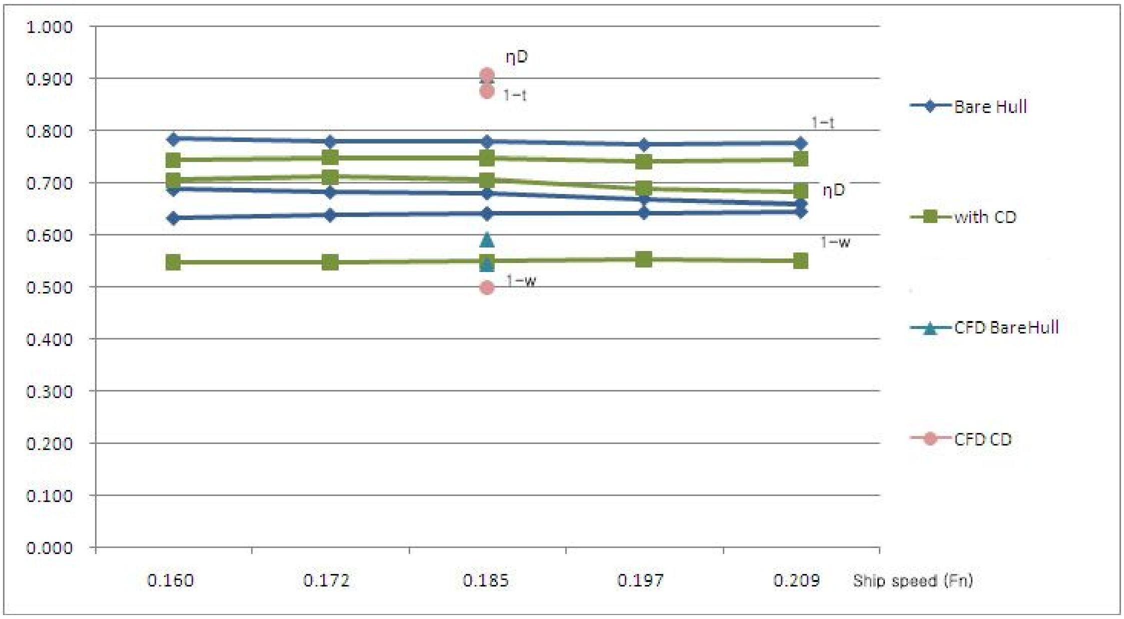
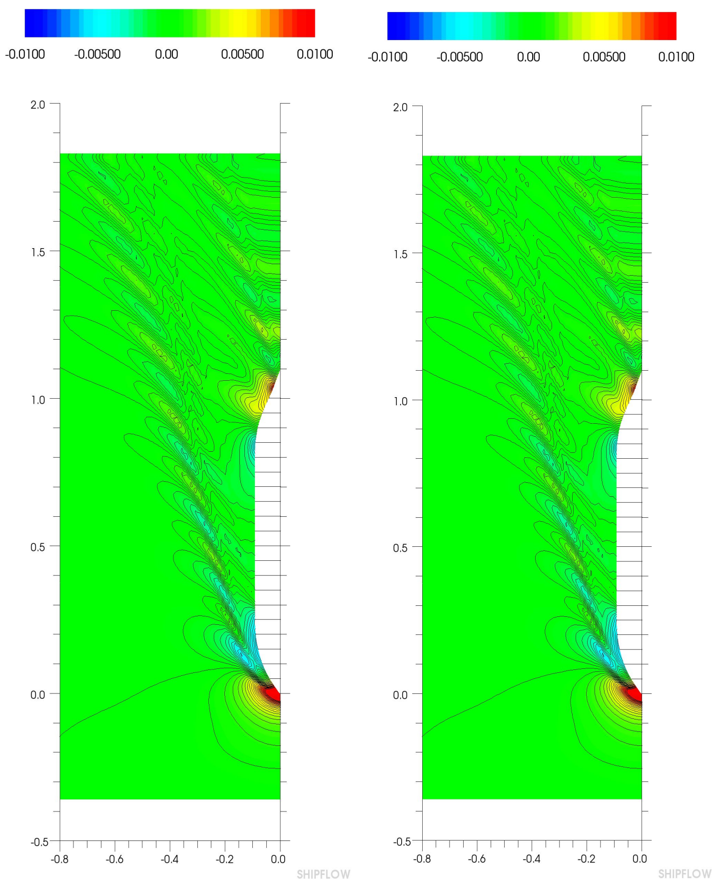


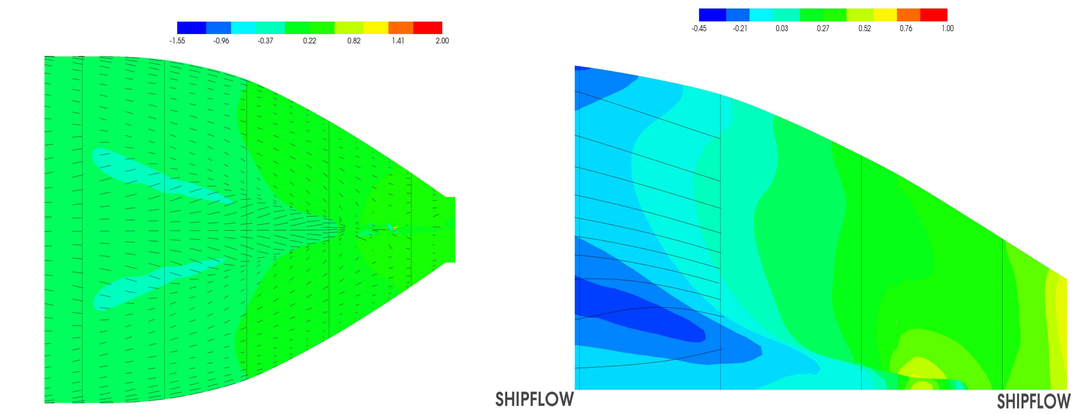
Fig. 5.3 propulsion coefficients



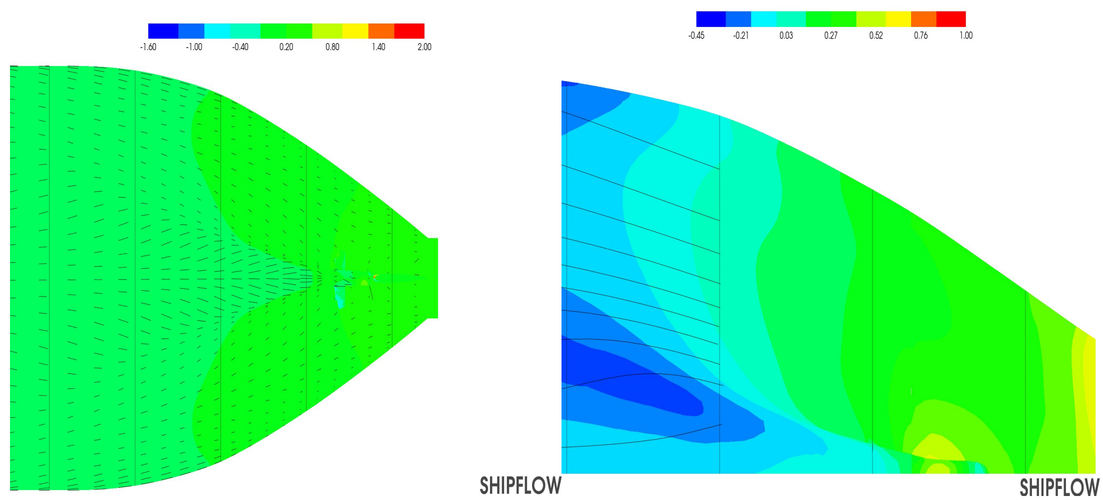
(a) Bare Hull (b)With Crown Duct

Fig. 5.4 Wave pattern around the 50K Tanker

Comparison with two different designs of Crown Duct for the wave profile along the hull is shown Fig. 5.4.

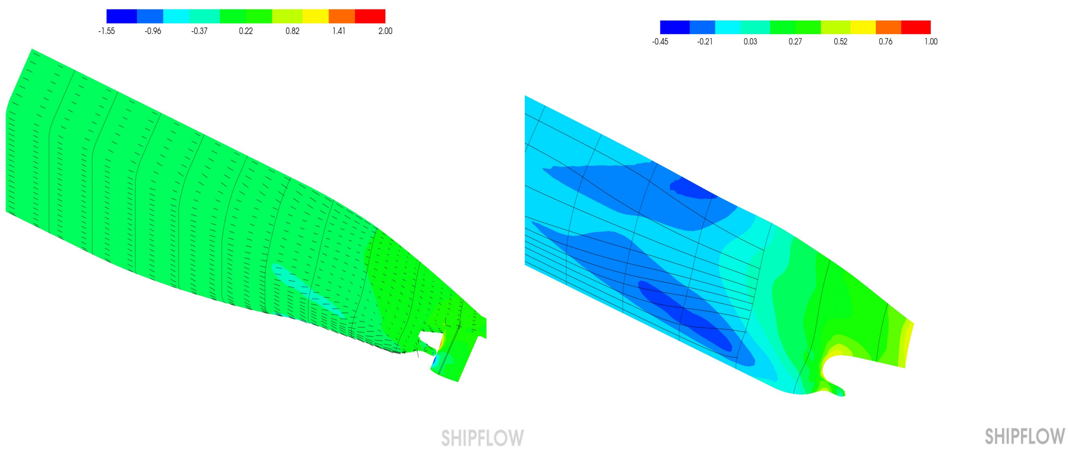


(a) Bare Hull

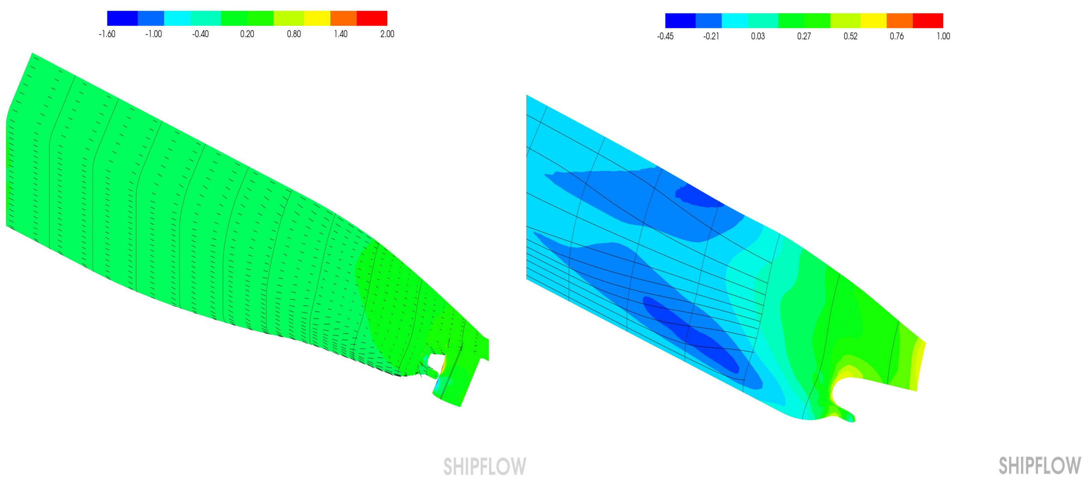


(b) With Crown Duct

Fig. 5.5 Comparison of stream line

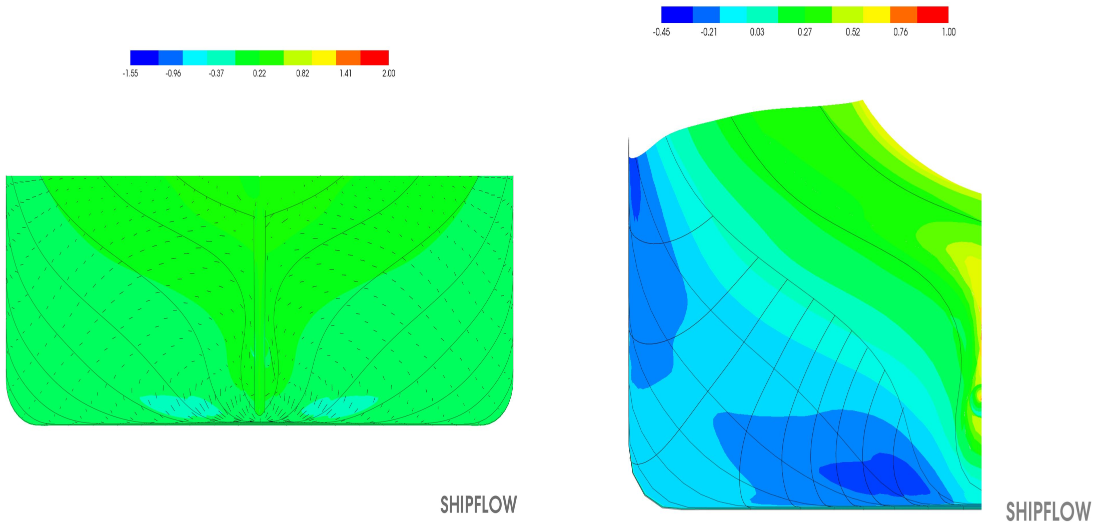


(a) Bare Hull

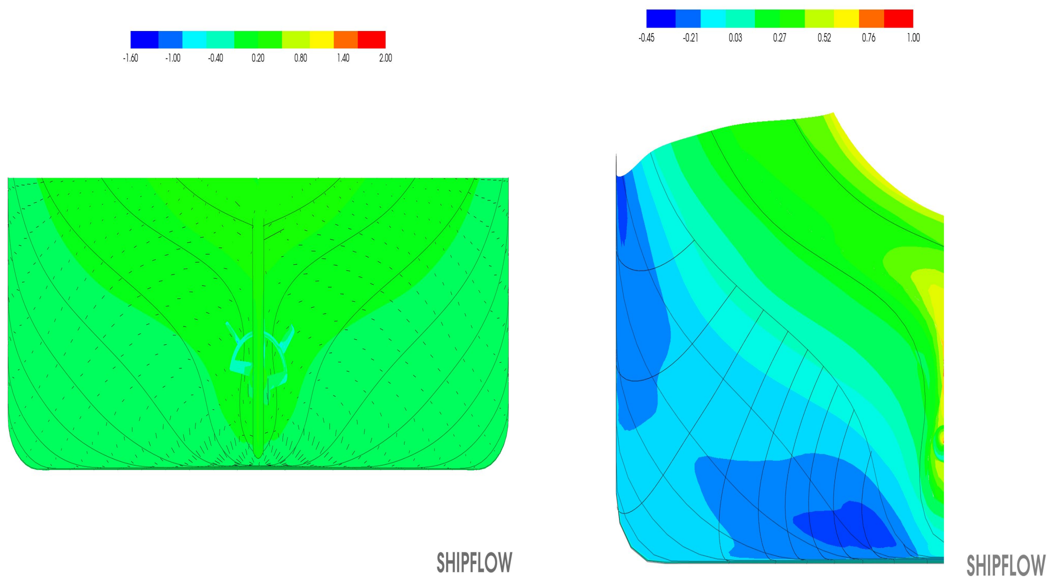


(b) With Crown Duct

Fig. 5.6 Comparison of flow calculations

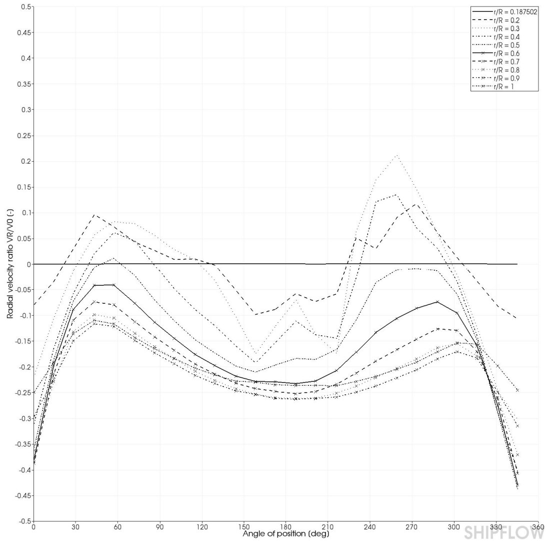


(a) Bare Hull

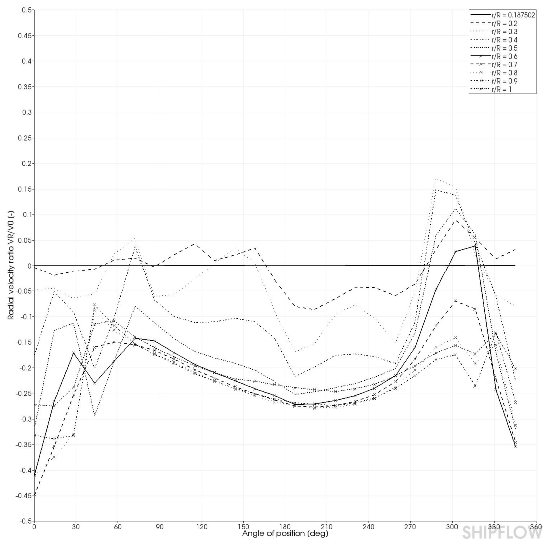


(b) With Crown Duct

FIg. 5.7 Comparison of the stern flow

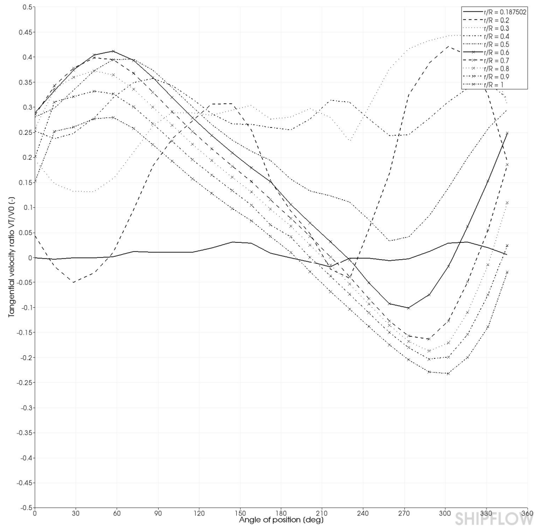


(a) Bare Hull

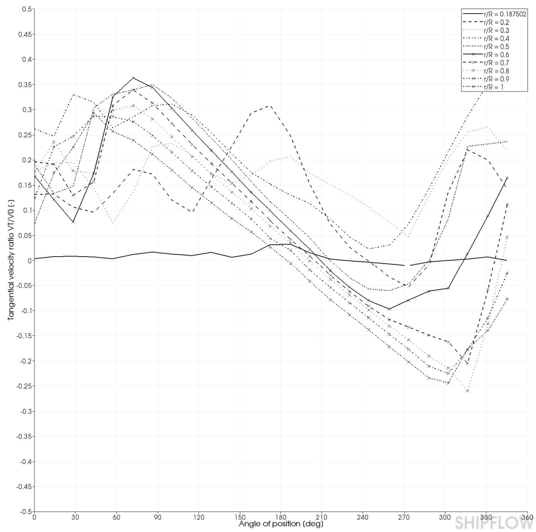


(b) With Crown Duct

Fig. 5.8 Comparison of the direction of rotation speed

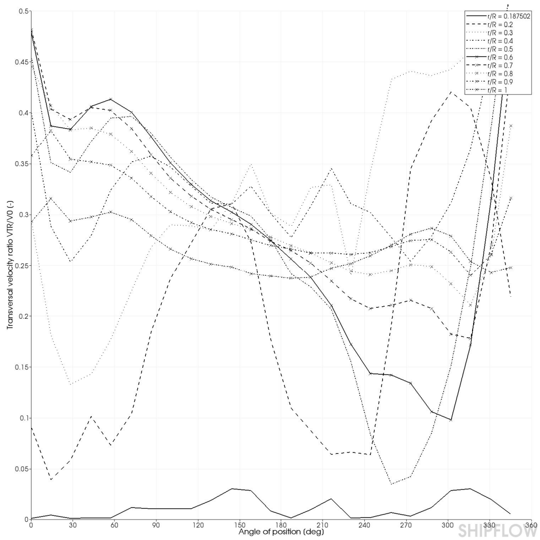


(a) Bare Hull

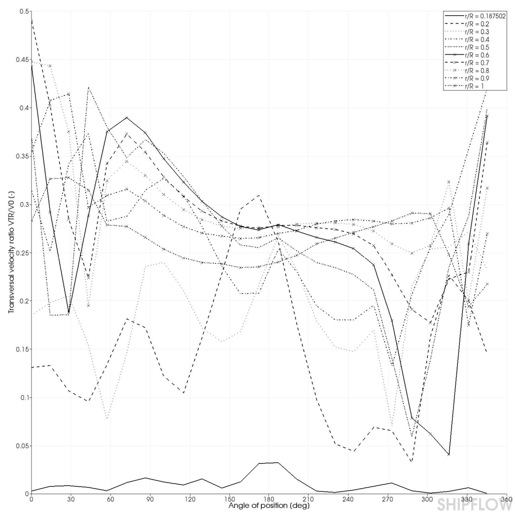


(b) With Crown Duct

FIg. 5.9 Comparison of tangential velocity

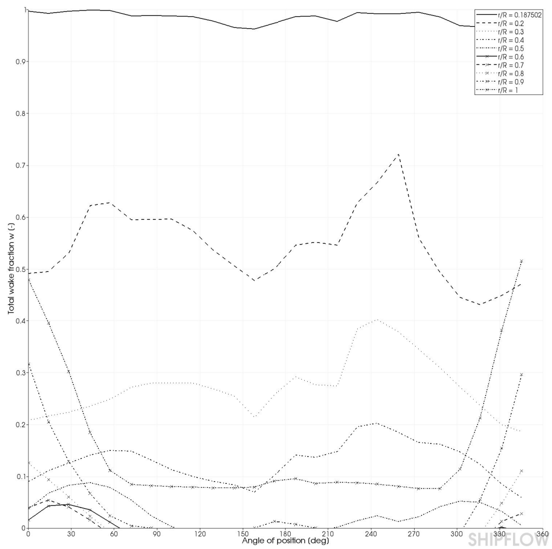


(a) Bare Hull

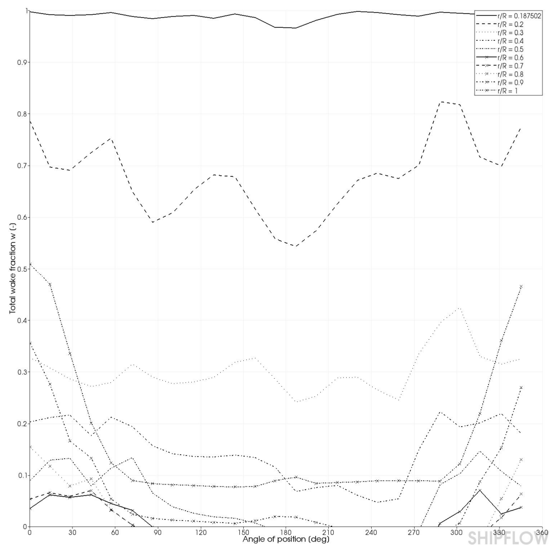


(b) With Crown Duct

Fig. 5.10 Comparison of Transverse speed

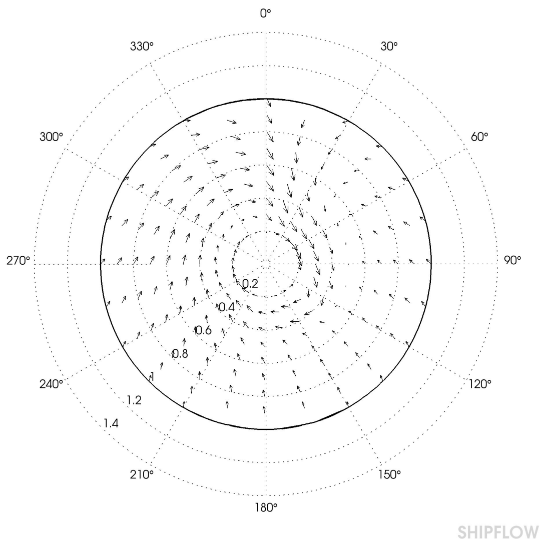


(a) Bare Hull

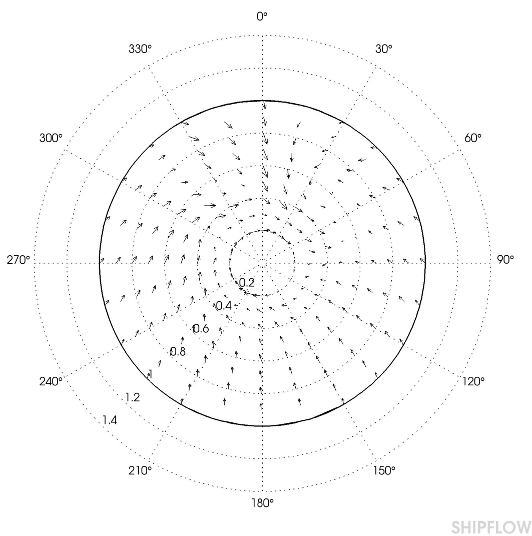


(b) With Crown Duct

Fig. 5.11 Comparison of Wake fraction

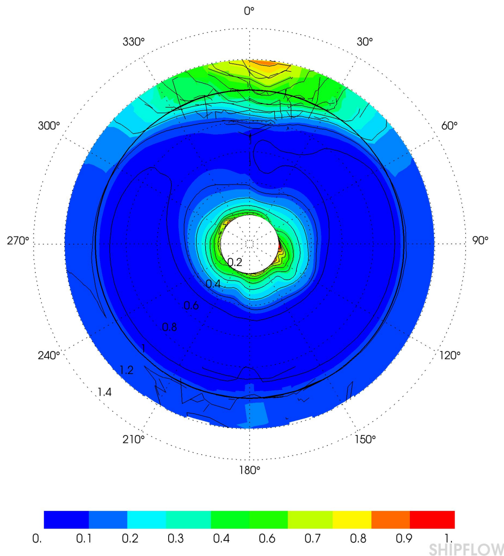


(a) Bare Hull

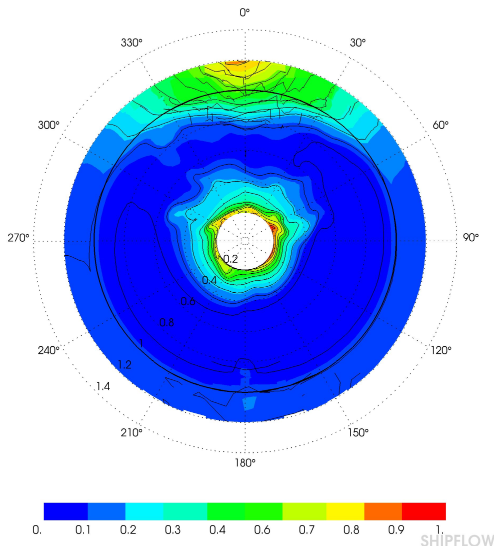


(b) With Crown Duct

Fig. 5.12 Comparison of V_{tr}

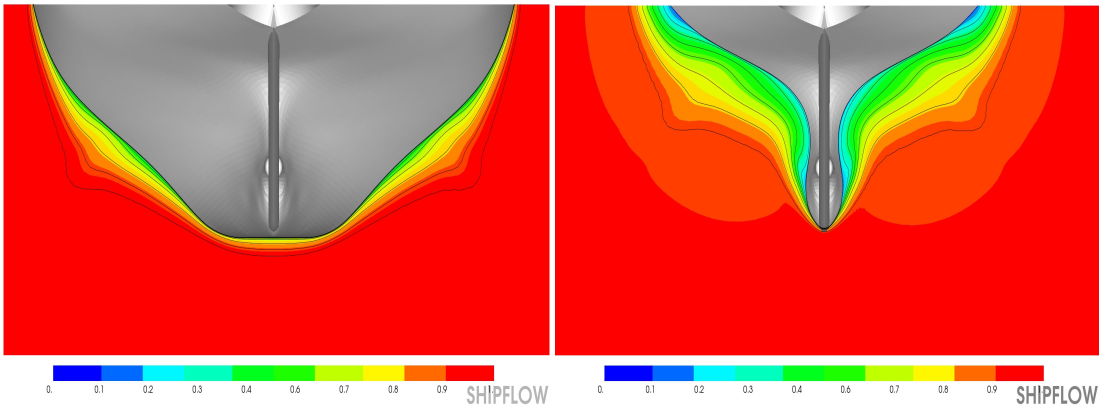


(a) Bare Hull

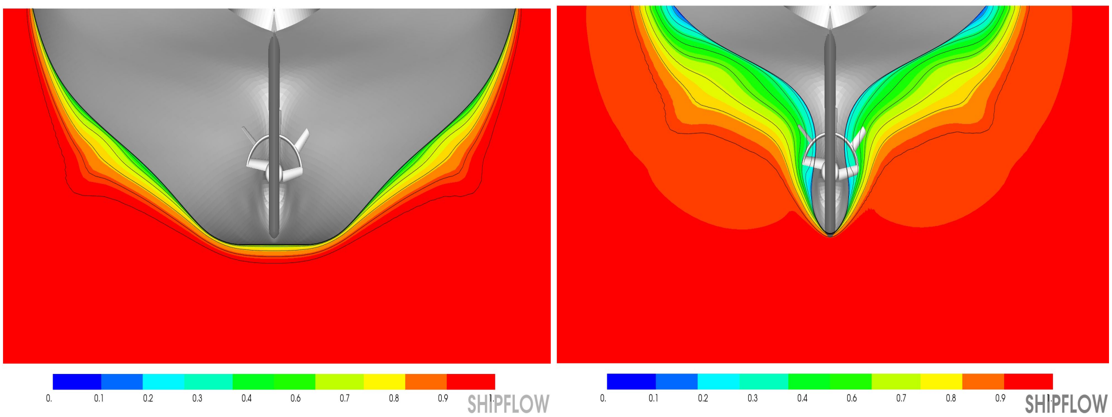


(b) With Crown Duct

Fig. 5.13 Comparison of Wake

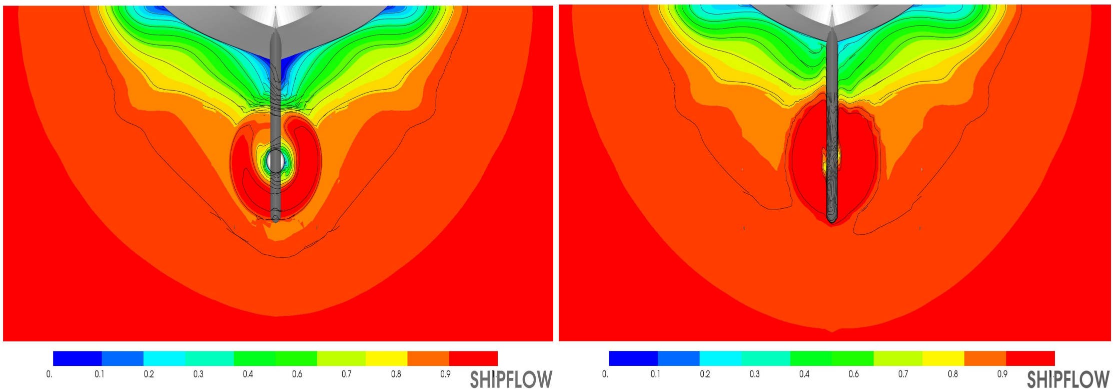


(a) Bare Hull

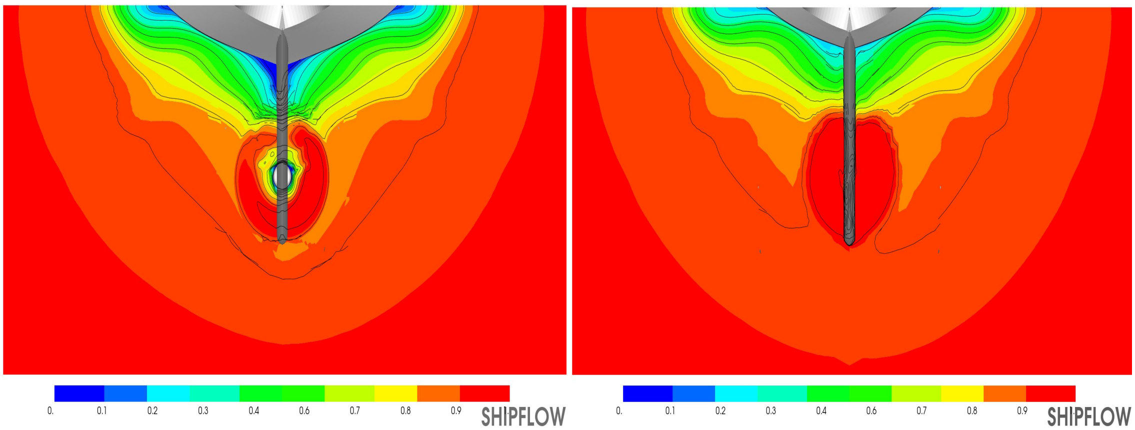


(b) With Crown Duct

Fig. 5.14 Comparison of Wake in $U_{x=1}, U_{x=3}$



(a) Bare Hull



(b) With Crown Duct

Fig. 5.15 Comparison of Wake in $U_{x=4}, U_{x=5}$

6. Conclusions

Diffusive angle of Duct has been designed to improve the thrust. The new concept of energy saving device "Crown Duct" shows efficiency gain 3.4% at full load condition and 2.7% at ballast condition in model test which was carried out at SSPA.

Comparative results of CFD calculation with model test ones are shown in Table 7. CFD Code "SHIPFLOW" is evaluated as applicable to the calculation of performance of ships with appendages.

Table 6.1 Comparison of CFD calculation results with the result of model test.

		C_W	1+k	P_E	w	t	η_R	η_o	η_D
Bare hull	CFD result	0.193	1.354	4557	0.408	0.094	0.989	0.539	0.887
	Model test result	0.228	1.220	4783	0.342	0.205	1.013	0.557	0.689
	Diff.	+0.035	-0.134	4.95%	-0.066	+0.111	+0.024	+0.018	-0.198
with CD	CFD result	0.193	1.396	4558	0.501	0.123	0.987	0.484	0.908
	Model test result	0.225	1.220	4744	0.451	0.252	1.022	0.501	0.707
	Diff.	+0.032	-0.176	4.08%	-0.050	+0.129	+0.035	+0.017	-0.201

References

1. Asaumi. T, Ikeda. T, Tamashima. M and Shinohara. H (2006)." Effectiveness of an Energy Saving Device 'Friend Fin' , Conference Proceedings ", The Japan Society of Naval Architecture and Ocean Engineering, Nov. Vol.2

2. Byoung-Nam Kim at al. (2009). "Flow Analysis around a High-speed Planig Hull Model", The Korea Society of Ocean Engineers, Vol.9

3. Hoshino T. at al. (2004). "Development of High-Performance Stator Fin by Using Advanced Panel Method", Mitsubishi Juko Giho Vol41 No.6

4. Kyoung-Hwa Kim at al. (2003). "Correlative Experimental Study Between The Results of Circulating Water Channel and Towing Tank Tests", The Korea Society of Ocean Engineers, Vol.6

5. Yong-Deok Kang at al. (2003). "A Study on the Design of a Biassed Asymmetric Pres-wirl Stator Propulsion System", The Korea Society of Ocean Engineers, Vol.

6. Kwi-Joo Lee at al. (2012). "A study on the Development of Energy-Saving Device "Crown Duct", The Korea Society of Ocean Engineers, Vol.

7. Keun Jae Kim, "SHIPFLOW calculations and resistance minimization" Dr thesis

8. L Broberg and M. Orych, 'An Efficient Numerical Technique to Simulate The Propeller Hull Interaction', FLOWTECH International AB, Sweden, 2012

9. Kyeong Hoon Cho, "A Study on the Hull Form Improvement by Inverse Design method", Master's thesis, 2008