

PAPER • OPEN ACCESS

## Research on Influence of Trim on a Container Ship's Resistance performance

To cite this article: Xianjiao Gao *et al* 2019 *J. Phys.: Conf. Ser.* **1300** 012105

View the [article online](#) for updates and enhancements.



**IOP | ebooks™**

Bringing together innovative digital publishing with leading authors from the global scientific community.

Start exploring the collection—download the first chapter of every title for free.

# Research on Influence of Trim on a Container Ship's Resistance performance

Xianjiao Gao<sup>1,2,\*</sup>, Kangning Sun<sup>3</sup>, Shengzhe Shi<sup>1,2</sup>, Bin Wu<sup>1,2</sup>, Zibin Zuo<sup>1,2</sup>

<sup>1</sup>AVIC Special Vehicle research Institute, Hubei Jingmen, China

<sup>2</sup>Key Aviation Scientific and Technological Laboratory of High Speed Hydrodynamic, Hubei Jingmen, China

<sup>3</sup>Chinese Flight Test Establishment, Shanxi xian, China

\*Corresponding author e-mail: 370481a02w9.cdb@sina.cn

**Abstract.** Faced with the double pressure of rising oil price and limitation of greenhouse gas emissions, many ship owners began to seek measures to minimize ship's resistance under specific conditions. Trim optimization has gained more and more attention in recent years for its flexibility and effectiveness in energy saving and emission reduction. The purpose of this paper is to perform trim optimization on a container ship. First, commercial CFD code of the ANSYS FLUENT was applied to calculate the target ship's total resistance. Then, in order to validate the effectiveness of CFD method, experimental result of ship model test was referred and it indicated that the numerical method was a reliable tool in prediction of the container ship's hydrodynamic performance. Finally, resistance corresponding to various trim conditions and speeds of three typical drafts were investigated, and it showed that trim did have impact on resistance. Based on the attained result, optimum trim value for actual navigation was suggested.

## 1. Introduction

Reduction of greenhouse gas emission has always been the focus of scientific research of environmental protection for many years, and shipping industry is one of the stakeholders in this issue. It is estimated that three percent of global carbon dioxide are caused by ships because of burning fuels [1]. Faced with double pressure of ever rising fuel prices as well as limitation of CO<sub>2</sub> emission from the International Maritime Organization (IMO), many ship industries take a lot of strategies to cut down fuel costs and reduce energy consumption, such as hull lines optimization, modification of bulbous bow, installation of energy-saving appendages, and trim optimization.

Among these measures, trim optimization is adopted by many ship owners for its advantages in reducing ship's resistance without changing structure of a ship or installing any equipment [2]. As we all know, a ship's resistance is closely related to its wetted surface area and underwater hull form, and different trim conditions would cause changes of a ship's streamline. Therefore, it is reasonable and feasible to reduce a ship's resistance by merely adjusting its trim value.

Owing to the improvement of computational power and parallel processing, computational fluid dynamics is now becoming more and more popular in simulation of complex flow. This paper chooses a 4250-TEU Container ship as research target, and optimization of searching for its minimum resistance



trim value by employing CFD is performed. First, numerical method is used to obtain the total resistance of target ship on even keel condition. Then the numerical results are compared with towing tank tests to validate its reliability. Finally, influence of trim on resistance performance is analyzed and some practical advices were given for actual navigation.

## 2. Numerical calculation

### 2.1. Ship model

A 4250-TEU container ship is selected as research target, the main reason for using this type hull is that it experiences many loading conditions with combination of different trims and drafts in actual operation, so it is of great possibility to obtain the minimum resistance condition by performing trim optimization. Main parameters of this model are presented in Table 1.

**Table 1.** Main characteristics of the container ship

Parameters	Dimensions(full scale)
Length overall(m)	261.1
Length between perpendiculars(m)	247.1
Molded breadth(m)	32.2
Molded depth(m)	19.3
Design speed(kn)	24
Design draft(m)	11
Displacement(m <sup>3</sup> )	55488
Wetted surface area(m <sup>2</sup> )	10040
Block coefficient	0.613

The three-dimensional ship model is generated in the generative shape design module of CATIA. Cross-sectional lines are selected as basic framework of hull surface. Because curvature of the ship's fore part and after part varies sharply, waterlines and buttock lines are also used as additional restrictions to ensure fairing of part. Figure 1 shows the three-dimensional model of container ship.



**Figure 1.** Three-dimensional ship model

### 2.2. governing equations

As computational fluid dynamics is based on finite volume method, fluid region is usually discretized into a finite set of control volumes, and numerical method is applied to solve relevant equations of each volume to get solution of the entire flow field [3]. In this paper, the three-dimensional, incompressible, pressure-based solver is used to calculate total resistance of ship model. Continuity and Navier-Stokes equations are the governing equations, which are defined by:

$$\frac{\partial u_i}{\partial x_i} = 0 \quad (i=1, 2), \quad (1)$$

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = f_i - \frac{1}{\rho} \frac{\partial p}{\partial x_i} + \nu \frac{\partial^2 u_i}{\partial x_j \partial x_j} \quad (i=1, 2), \quad (2)$$

Where  $u_i$ ,  $p$  and  $\rho$  are three components of velocity in three directions, pressure, fluid density and body force, respectively. And  $f_i$  is the body force. As the above equations are not closed, time averaged method is usually introduced to simplify equations. So, formula (2) can be expressed by

$$\frac{\partial \bar{u}_i}{\partial t} + \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial}{\partial x_j} (\overline{u_i u_j}) = -\frac{1}{\rho} \frac{\partial \bar{p}}{\partial x_i} + \nu \frac{\partial^2 \bar{u}_i}{\partial x_j \partial x_j} + f_i \quad (3)$$

Where  $\bar{u}_i$ ,  $\bar{p}$  represent the time-averaged velocity and pressure,  $\overline{u_i u_j}$  is the Reynolds stress, which is given by:

$$-\overline{\rho u_i u_j} = \mu_t \left( \frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j} \right) - \frac{2}{3} \rho \kappa \delta_{ij} \quad (4)$$

$$\kappa = \frac{1}{2} \overline{u_i u_i} \quad (5)$$

$$\delta_{ij} = \begin{cases} 0, & i \neq j \\ 1, & i = j \end{cases} \quad (6)$$

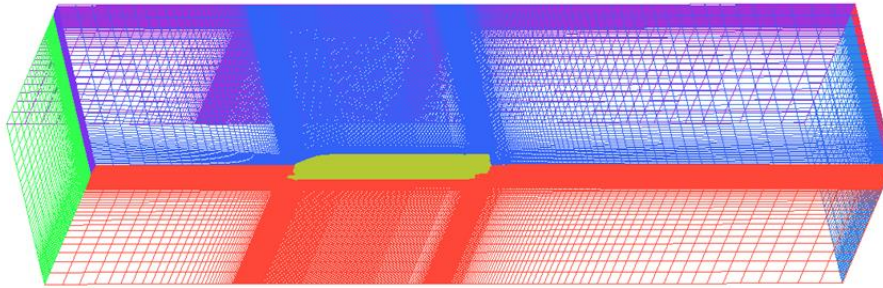
Where  $\mu_t$  is eddy viscosity coefficient,  $\kappa$  is turbulent kinetic energy,  $\delta$  is Kronecker function. The turbulence model RNG  $k-\varepsilon$  is used to simulate flow around ship hull, and it is a modified form of the standard  $k-\varepsilon$  that can improve simulation accuracy of strong strain flow, whirling flow as well as low Reynolds flow[4].

### 2.3. grid generation and boundary condition

Grid generation and numerical calculation were carried out using commercial code of ICEM and FLUENT, respectively. Calculation domain is a free-surface model and is discrete by H-O grid topology, three-dimension of the grid system is 258\*70\*90. The computational domain consist of seven parts: inlet plane, outlet plane, side plane, symmetry plane, top surface, bottom surface and hull surface. As flow around the ship hull changes fiercely, it is essential to ensure quality of boundary layer[5-6], especially first layer grid adjacent to ship hull surface. In this paper, the first grid space was determined by the following formula:

$$y^+ = 0.172 \left( \frac{y}{L} \right) \text{Re}^{0.9} \quad (7)$$

Where  $y$ ,  $L$ ,  $\text{Re}$  represent the first layer grid space, length between perpendiculars of ship, Reynolds number, respectively.  $y^+$  is the dimensionless wall distance of  $y$ . In order to capture flow characteristics of near wall surface, wall function method is used for the solution of boundary layer equation as well as improving computational efficiency. Generally speaking, this method is only effective when  $y^+$  is limited between 30 and 300, in this paper, we assign 40 to  $y^+$  based on some relevant experience. Figure 2 shows grid generation result of the calculation domain.



**Figure 2.** Gird generation of calculation domain

As mentioned before, calculation domain consists of seven surfaces, and boundary conditions for each surface are defined as follows: the inlet plane and top plane are set as velocity inlet, the outlet surface is set as pressure outlet, the symmetry plane is defined as symmetry, and no-slip wall conditions is applied to the ship hull, side and bottom surfaces. Table 2 illustrates the detail mathematical form of boundary conditions.

**Table 2.** Mathematical form of boundary conditions

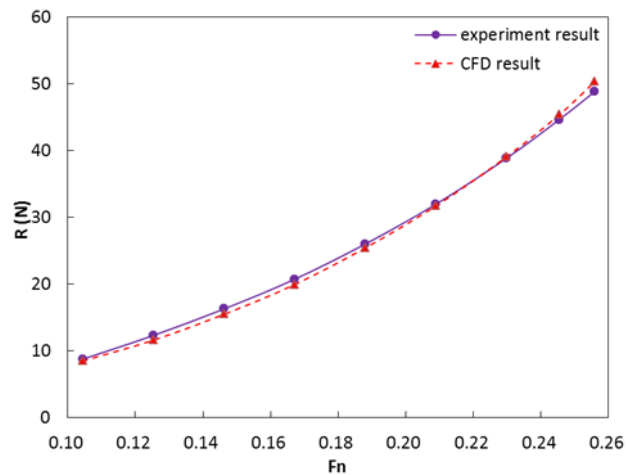
Description	$U$	$V$	$W$	$P$
Inlet	$U = V_m$	$V = 0$	$W = 0$	0
Outlet	$\frac{\partial U}{\partial n} = 0$	$\frac{\partial V}{\partial n} = 0$	$\frac{\partial W}{\partial n} = 0$	$P = P_{ref}$
No-slip Wall	0	0	0	-
Symmetry	$\frac{\partial U}{\partial n} = 0$	0	$\frac{\partial W}{\partial n} = 0$	$\frac{\partial P}{\partial n} = 0$
Inlet	$U = V_m$	$V = 0$	$W = 0$	0

### 3. Verification of resistance calculation under even keel condition

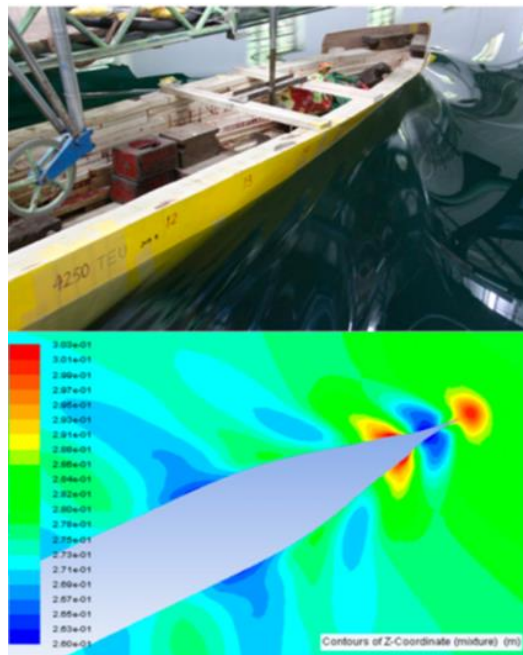
Development of computational fluid dynamics (CFD) is largely dependent on experiment and theoretical hydrodynamics [7]. Although it has become an important tool in scientific research and engineering analysis, reliability of numerical calculation needs to be verified by experiments. Scale of calculation model is set to 40, which is same as ship model test. Table 3 and figure 3 show comparison results of ship resistance (at design draft), figure 4 shows comparison of wave patterns at  $Fn=0.167$ .

**Table 3.** Resistance result of numerical calculation and experiment

$Fn$	Experiment Result(N)	CFD result (N)	Relative error (%)
0.104	8.76	8.485	-3.14
0.125	12.312	11.602	-5.76
0.146	16.285	15.482	-4.93
0.167	20.719	19.921	-3.85
0.188	25.997	25.419	-2.22
0.209	31.981	31.708	-0.85
0.230	38.818	39.032	0.55
0.246	44.636	45.349	1.60
0.256	48.834	50.316	3.03



**Figure 3.** Resistance comparison between calculation results and experiment results

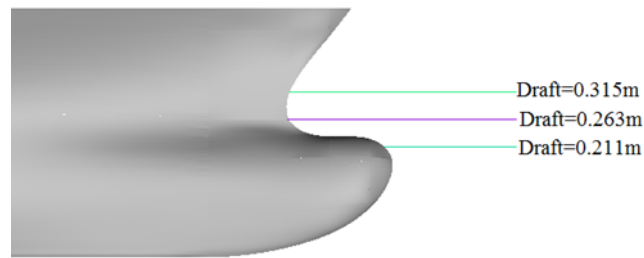


**Figure 4.** Comparison of wave patterns between calculation and experiment at  $Fn=0.167$ .

It can be seen from table 3 and figure 3 that the calculation results are in good agreement with experiment data, and the relative error is within 5.8%. Figure 4 shows wave distribution pattern at the speed of  $Fn=0.167$ , it can be seen that the simulated wave pattern is very similar with experiment result, and the CFD method has reliable accuracy in resistance performance prediction.

#### 4. Analysis of resistance performance at different trims and drafts

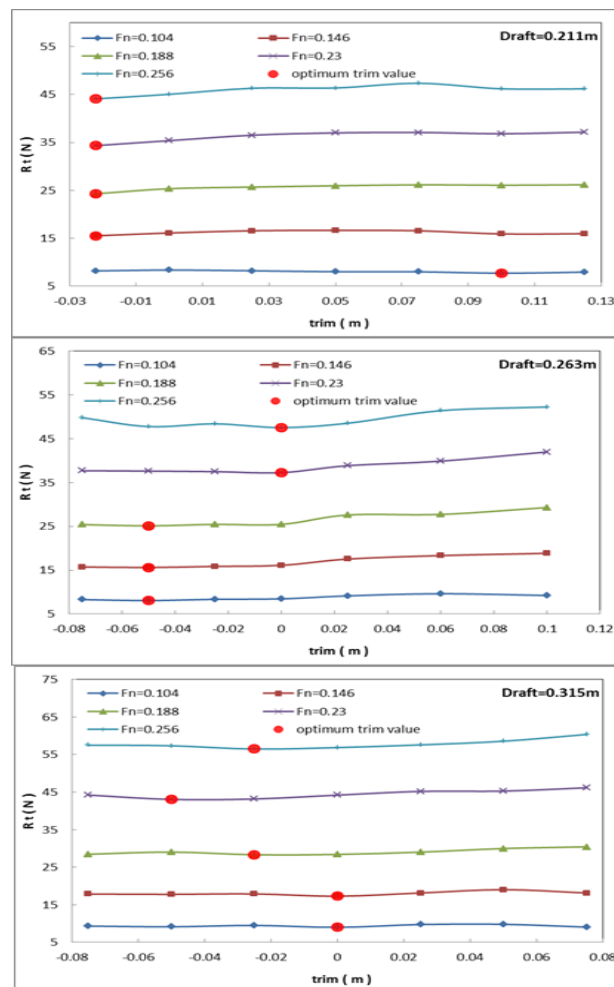
As we all know, a bulbous bow's underwater shape has an important influence on hydrodynamic performance, especially waving making resistance, so we choose three typical drafts (shown in Figure 5) that has different immersion depth of bulbous bow to study the influence of trim on resistance. The validated numerical method is used to calculate resistance at different trim conditions. Value of trim means difference between fore draft and aft draft, and trim by the bow is negative, trim by the stern is positive.



**Figure 5.** Typical drafts condition for calculation.

**Table 4.** Calculation results of ship model resistance.

Draft(m)	Fn	0.104	0.146	0.188	0.230	0.256
0.211	Trim(m)			Rt(N)		
	-0.022	8.17	15.51	24.29	34.34	44.12
	0.000	8.35	16.09	25.34	35.38	45.06
	0.025	8.2	16.55	25.68	36.52	46.32
	0.050	8.00	16.65	25.96	37	46.38
	0.075	7.98	16.55	26.15	37.09	47.36
	0.100	7.69	15.89	26.07	36.82	46.21
0.263	0.125	7.92	15.94	26.16	37.15	46.2
	-0.075	8.29	15.74	25.4	37.7	49.81
	-0.050	8.07	15.62	25.12	37.61	47.77
	-0.025	8.35	15.86	25.45	37.49	48.4
	0	8.46	16.11	25.46	37.26	47.54
	0.025	9.14	17.54	27.6	38.85	48.54
	0.06	9.63	18.33	27.73	39.89	51.4
0.315	0.1	9.23	18.85	29.3	41.98	52.24
	-0.075	9.33	17.86	28.45	44.21	57.5
	-0.050	9.14	17.78	29.03	43.05	57.33
	-0.025	9.51	17.89	28.32	43.17	56.46
	0	9	17.29	28.41	44.2	56.85
	0.025	9.75	18.15	29.02	45.19	57.57
	0.05	9.8	19.03	29.98	45.27	58.55



**Figure 6.** Comparison of resistance at different drafts and trims

**Table 5.** Three Scheme comparing.

	Fn	0.104	0.146	0.188	0.230	0.256
Draft=0.211m	Optimum trim value(m)	0.100	-0.022	-0.022	-0.022	-0.022
	Relative error(%)	-7.904	-3.605	-4.144	-2.940	-2.086
Draft=0.263m	Optimum trim value(m)	-0.050	-0.050	-0.050	0.000	0.000
	Relative error(%)	-4.610	-3.042	-1.335	0.000	0.000
Draft=0.315m	Optimum trim value(m)	0.000	-0.025	-0.050	-0.025	0.000
	Relative error(%)	0.000	-0.317	-2.602	-0.686	0.000

Table 4 and figure 6 shows resistance calculation results and comparison curves, ‘red dot’ means optimum trim value among all trim conditions. It can be seen obviously that trim does have an important effect on the ship’s total resistance. When the ship model has a draft of 0.211m (a condition that the bulbous bow is partly immerged), trim by the bow is profitable on the whole (except the speed point of  $Fn=0.104$ ), total resistance in the even keel condition is larger than that in the bow trim condition and smaller than that in the stern trim condition, optimum trim value is  $-0.022m$  when Froude number is larger than 0.104. However, when the ship operates near design draft, trim by the bow is favorable only at low speed range, at high speeds, even keel condition is the best choice since either bow trim or stern trim could cause increase of total resistance. When the ship model draft reaches to 0.315m, minimum



resistance occurs to even keel condition at low Froude number, when the velocity increases, bow trim is more favorable than even keel as well as stern trim condition.

Table 5 gives optimum trim value and its resistance reduction rate relative to even keel condition. It shows that trim can achieve 7.9% reduction of total resistance under specific condition, and the influence of trim on resistance is more significant for shallow draft than deep draft, so it is applicable to perform trim optimization for the purpose of energy saving.

In conclusion, trim can make a difference on the ship's resistance, but the optimum trim value varies with drafts and velocities. When the ship has a shallow draft, optimum trim value belong to the bow trim scope(except at the velocity of  $Fn=0.104$ ); trim by the bow is also effective at low Froude number near design draft, but optimum trim value changes to even keel condition at high Froude number; when the ship has a deep draught, the tendency becomes contrary to design draft, so trim optimization analysis should be performed for different conditions.

## 5. Conclusion

Ship trim optimization has become an important energy-saving strategy in recent years for its flexibility and simplicity in actual operation compared with traditional hull form retrofit. This paper selects a container ship as research target. First, CFD simulation technique is applied to calculation of model ship resistance at design draft in even keel condition. Then, in order to verify validity of numerical method, comparison between CFD results and experimental data is made, and it confirms that the numerical results are well matched with model test results, including both resistance value and wave pattern. Finally, we choose three typical drafts to analyze the influence of trim on total resistance at different speeds, and it shows a general rule for the container ship that trim by the bow and even keel condition are more efficient in reduction of oil consumption than stern trim is, so it is suggested that trim by the stern should be avoided during actual voyage, besides the optimum trim value is also suggested.

This paper performed trim optimization for a container ship with a combination of numerical simulation and experimental method, and it confirms that trim can be used as an effective and flexible measure in energy conservation and emission reduction, which provide a new direction for green ships.

## References

- [1] V. Eyring, H. W. Kohler, A. Lauer, et al. Emissions from international shipping: 2. Impact of future technologies on scenarios until 2050[J]. *Journal of Geophysical Research*, 2005, 110: 1-18.
- [2] Jisun Lee, Seonoh Yoo, Sangkyu Choi, et al. Development and Application of Trim Optimization and Parametric Study Using an Evaluation System (SoLuTion) Based on the RANS for Improvement of EEOI[C]// *Proceedings of the ASME 2014 33rd International Conference on Ocean, Offshore and Arctic Engineering*. San Francisco, California, USA, 2014.
- [3] Jacob A. Freeman, Christopher J. Roy. Verification and validation of Reynolds-averaged Navier–Stokes turbulence models for external flow[J]. *Aerospace Science and Technology*, 2014, 32: 84-93.
- [4] A. Celic, E. H. Hirschel. Comparison of eddy-viscosity turbulence models in flows with adverse pressure gradient[J]. *AIAA J*, 2006, 44(10): 2156–2169.
- [5] Boger DA, Dreyer JJ. Prediction of hydrodynamic forces and moments for underwater vehicles using overset grids[C]// *44th AIAA Aerospace Sciences Meeting*. Reno, Nevada, 2006.
- [6] Z.-R. Zhang, H. Liu, S.-P. Zhu, and F. Zhao. Application of CFD in ship engineering design practice and ship hydrodynamics[J]. *Journal of Hydrodynamics*, 2006, 18(3): 315-322.
- [7] Simonsen, CD and Stern, Fred. Verification and validation of RANS maneuvering simulation of Esso Osaka effects of drift and rudder angle on forces and moments[J]. *Computers & Fluids*, 2003, 32: 1325-1356.