International Journal of Engineering & Technology, 7 (4.36) (2018) 121-124

International Journal of Engineering & Technology

Website: www.sciencepubco.com/index.php/IJET

Research paper

Application of Panel Method to Calculate Ship Resistance

Tran Ngoc Tu, Nguyen Manh Chien

Shipbuilding Faculty, Vietnam Maritime University, Haiphong City, Vietnam
**Corresponding author: tutn.dt@vimaru.edu.vn*

Abstract

In this study, the ship resistance in calm water was calculated using panel method under potential flow theory. This is a CFD (Computational Fluid Dynamic) approach with the assumption that the fluid is non-viscosity. Therefore, only wave resistance is calculated by solving nonlinear free surface problems. The frictional resistance is predicted by ITTC formula. The calculation is performed with well-known benchmark case – Kriso Container ship (KCS) – where the experiment result of ship resistance is available to compare and validate the calculation.

Keywords: *Ship resistance, panel method, potential flow, CFD*

1. Introduction

The fast development of computation resources is making Computation Fluid Dynamic (CFD) becoming a powerful tool for ship designers in solving the problems related to hydrodynamics. One of the basic problems is ship resistance calculation. The traditional approach is using some well-known empirical method such as Holtrop Menen, Series 60, HollenBach [3]. The advantages of this method is fast and does not require much input data. However, the lack of accuracy is a problem, especially for new and unconventional ship [1]. Besides, by solving the fluid equations numerically, the CFD method is a direct method to calculate ship resistance. Depending on the assumption to simplify the fluid equations, there are some CFD approaches are available. Figure 1 shows some CFD approaches in terms of computational time and level of accuracy [7].

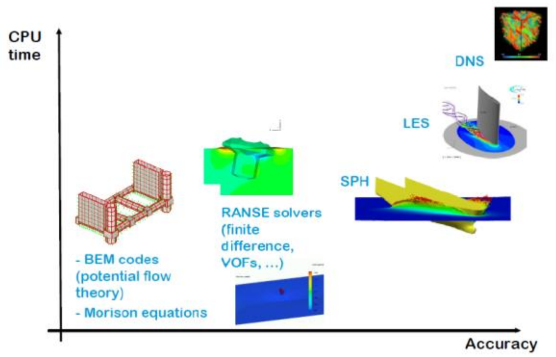

Figure 1. Some CFD approaches with CPU time and accuracy

In practice, there are only BEM (boundary element method) codes, RANSE solver and LES using for ship hydrodynamics problem. This study presents the theoretical background and application of the BEM codes: panel method under potential flow theory for ship resistance calculation. The advantage of this method is that the computational resources are significantly less than the RANSE and LES method. Many authors have presented the result of ship resistance calculation using RANSE method [9] [10] [11] with good agreement result with experiments. However, the computational time is still a problem, especially in the case of performing many calculations for hull form optimization. That is why panel method is still used in ship resistance calculation.

To study the ship resistance calculation using panel method, in this paper, the authors use the well-known hull form KCS [2] that is public and there is experimental result available. So the result is validated with experiment.

2. Theoretical background of Potential Flow and Panel method

2.1. Basic assumption and equation of potential flow

The fluid is assumed to be non-viscosity and incompressible. So that the continuity equation becomes: $\nabla \cdot u = 0$, with u is velocity vector; and the vorticity $\omega = \nabla \times u = 0$. In this case, there is a function Φ such that:

$$u = \nabla\Phi \tag{1}$$

where: u - velocity vector; Φ - velocity potential function.

Using potential function above, the continuity equation becomes Laplace equation:

$$\nabla^2\Phi = 0 \tag{2}$$

The potential function can be solved by using known boundary condition. Once Φ is determined, the pressure p in the flow can also be found by solving Bernoulli equation:

$$\frac{1}{2}\rho\left|\nabla\Phi\right|^2 + p = const \tag{3}$$

2.2. Singularity distribution

Laplace equation is a linear differential equation. It means that any combination of elementary solutions can also be a new solution. A singularity is a mathematical object which generates an elementary flow. By combining some elementary flows, one can compute a realistic flow. Each singularity generates a potential flow. Some examples of singularity are sources, sinks, dipoles, vortices etc.

2.3. Panel method

The calculation domain (the free surface and hull in our ship resistance calculation case) is divided into N panel, then each panel is placed a singularity distribution (sinks, dipoles, vortices). At every panel, the velocity potential is solved and also the flow field u is solved by relationship: $u = \frac{\partial \Phi}{\partial n}$

2.4. Ship resistance problem under potential flow theory

The total resistance of the ship can be decomposed into viscous frictional resistance R_V and wave resistance R_W. In the wave resistance part, with the hypothesis of non-viscosity and irrational fluid, the wave breaking resistance is neglected. There is only wave field resistance R_{WP} can be calculated using potential flow theory [7].
For viscous frictional resistance R_V. The viscous frictional resistance R_V is calculated by ITTC formula.

$$R_V = (1+k)R_{F0} \tag{4}$$

Where: R_{F0}: frictional resistance of flat panel, can be calculated by ITTC 1978 formula
k: form factor
The advantage of this method is fast computation due to low number of unknowns and the calculation system is linear. The hull body and the free surface are meshed by panels. The result given by this method is wave field around the hull and the wave resistance by integral of pressure field. The result of this method is good for usual hulls.

2.5. Equation of the ship resistance problem under potential flow theory

Considering a hull having a constant forward speed in initially calm water. The set of equations is: $\Delta\Phi = 0$ - in fluid domain D – Laplace equation; $\left(\frac{\partial \Phi}{\partial n}\right)_C = \left(\overrightarrow{V_0}.\vec{n}\right)_C$ - Slip condition on ship hull (C); $\frac{\partial \eta}{\partial t} + \vec{v}.\overrightarrow{grad\eta} = \frac{\partial \Phi}{\partial z}|_{z=\eta}$ - Kinematic boundary condition at free surface; $\eta = -\frac{1}{g}\left(\frac{\vec{v}^2}{2} + \frac{\partial \Phi}{\partial t}\right)_{z=\eta}$ - Dynamic boundary condition at free surface; $\Phi \to 0$ - at infinite-radiation condition.
The free surface boundary conditions can be linearized by the formulation of Neumann-Kelvin or double model. By using the method of singularities of Rankine, the singularities (sources, dipoles…) are put on all boundaries: body and free surface (to take into account the free surface).

3. Introduction to FS - Flow - a panel code developed by DNV-GL

FS-Flow is a general purpose Rankine source panel code, which solves the boundary value problem of potential theory. A selection of meshing types, singularity distributions and boundary conditions can be specified, including the nonlinear free surface boundary condition. The code further features internal mesh generation algorithms based on IGES or point data geometry input.
In the software the free surface conditions may be applied either linearly or nonlinearly. Nonlinear methods have been found superior to linear methods with respect to predicting the wave amplitudes, the phases and the wave resistance. For a numerical treatment the nonlinear free surface conditions are linearized around a known base solution. Since the free surface conditions are to be applied at the a priori unknown free surface an iterative solution scheme is used. In the iteration scheme the flow quantities, the free surface position and the dynamic floating position of the vessel are updated in an alternating manner. After each iteration step the geometry of the free surface is modified (adapted) and the sinkage and trim of the vessel is adjusted. Once the forces and moments are in balance and all boundary conditions are fulfilled the calculations are considered to be converged (convergence criteria). The dynamic position of floating bodies may be computed by equalizing sinking, heeling and pitching moments.
The wave resistance can be computed by both, hull pressure integration and by a sophisticated wave pattern analysis technique. The code automatically re-meshes the hull and a body fitted free surface mesh.
Viscous resistance is considered in terms of a friction line in combination with the wavy wetted hull surface and a form factor determined from integration of the local velocity perturbations.
This code is in-house code, developed by DNV-GL and often use internally within DNV-GL [6].

4. Case study - resistance calculation of KCS

4.1. Test case

KCS is well known test case to validate CFD calculation. KCS (Kriso container ship) is a modern container ship with bulb bow and stern. The basic dimension of the shape of KCS is shown in the Table 1 and Figure 2 below [2]. The calculation is performed with model scale and the experiment result is available by NMRI to compare the result.

Table 1. Basic dimension of KCS hull (in model scale)

	Symbol	Unit	Full Scale	Model
Length between perpendiculars	L_{PP}	[m]	230.0	7.2786
Length of waterline	L_{WL}	[m]	232.5	7.3586
Breadth at waterline	B	[m]	32.2	1.0190
Depth	D	[m]	19.0	0.5696
Draught	T	[m]	10.8	0.3418
Displacement volume	∇	[m^3]	52030	1.6497
Area of wetted surface	S	[m^2]	9530	9.4984
Block coefficient	C_B	[-]	0.6508	0.6508
Midship section coefficient	C_M	[-]	0.9849	0.9849

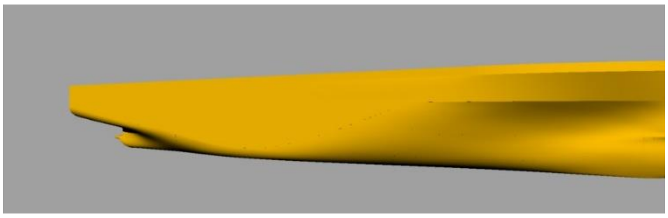

Figure 2. 3D view of KCS hull

4.2. Calculation of form factor and frictional resistance of flat panel

The form factor is calculated using empirical methods by some well-knowns authors. For example, in this case, the authors choose the empirical formula by Holtrop-Mennen [3]. The frictional resistance coefficient is calculated using ITTC 1978 formula [4].

4.3. Calculation of total resistance coefficient

The 3d KCS hull is imported to the software to generate the panels for hull and free surface. The total number of panels for free surface and hull are 3465 panels and 1114 panels respectively (only considering a half of ship and free surface). Figure 3 shows the panel distribution of the free surface and around hull. The form factor is inputted into the software and the method to calculate frictional resistance is also selected inside the software. The output of the problem is total resistance coefficient C_T and free surface elevation.

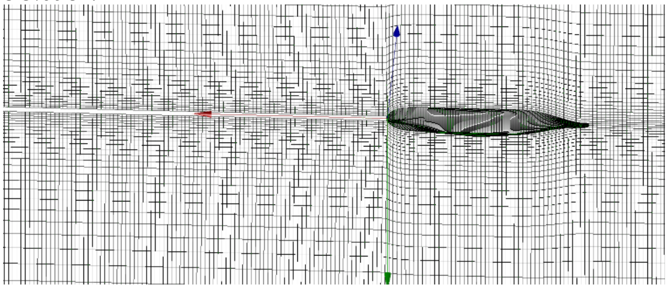

Figure 3. Panel distribution of KCS hull and free surface

5.Result and discussion

The ship resistance is calculated with speed v = 2.196 m/s; and the result is compared with towing tank result provided by NMRI (Japan National Maritime Research Institute. The result of total resistance coefficient is shown in Table 2 below:

Table 2. Result of ship resistance calculation using panel method

v (m/s)	Fr	k	R_T(N)	$C_T.10^3$ (panel method)	$C_T.10^3$ (Experiment) [2]	Difference, %
0.915	0.108	0.26011	15.07	3.748	3.796	-1.264%
1.281	0.151	0.26011	29.39	3.701	3.641	1.648%
1.647	0.194	0.26011	46.65	3.536	3.475	1.755%
1.921	0.22	0.26011	61.91	3.429	3.467	-1.096%
2.196	0.26	0.260	84.01	3.534	3.711	-4.770%

The result of calculation shows the good estimation of panel method, the differences with experiment is less than 5%. The panel method is also able to show the wave cut and wave pattern.

Actually, the value of form factor inputted into the program influences much to the result of viscous resistance. In this study, the authors select the best form factor (based on Holtrop – menen empirical method), to get the result close to experiments. However, we should be aware of the limitation of the panel method. Highly viscous effects such as a recirculation zone at the stern cannot be modeled correctly. Practice shows that this method has not had success in treating ships with deep transom sterns, or hull with significant flare or with overhang at the stern. Both the above characteristics are very common in today's fleet of commercial ship. Another limitation is the fact that the panel methods do not account for viscosity. In this program, "viscous resistance is considered in terms of a friction line in combination with the wavy wetted hull surface and a form factor determined from integration of the local velocity perturbations" [6]. So the accuracy of the viscous resistance depends on form factor, which is estimated by empirical method.

Practically, panel method often uses to do the hull form optimization due to short computational time. (for example: the bow part, where the assumptions of potential flow do apply to a good degree of accuracy). So many hull variations are calculated to get the best hull in term of resistance. The hull form is modelled by parametric method so at the end the Pareto frontier might be given for consideration and selection. The RANSE method is applied at the end of optimization process to validate the result.

The table 3 below shows the result of RANSE solver ISIS (in Fine Marine software) provided by Numeca [8]. The result is just less than 1% differences compare to experiment result

Table 3. Result of ship resistance calculation using RANSE solver ISIS [8]

Total drag [N]	C_T x 10^3 (CFD)	C_T x 10^3 (EFD)	Error (%)
81.8	3.70	3.71	-0.4

The RANSE computation does not give us the direct result of wave resistance to compare with panel method. However, we can try to extract the wave resistance by the relationship with viscous pressure resistance and frictional resistance. In this case: wave resistance Rw = pressure resistance Rp - viscous pressure resistance Rvp. Besides, viscous pressure resistance Rvp = viscous resistance Rv - frictional resistance Rf.

So Rw = Rp - Rvp = Rp - (Rv - Rf) or Cw = Cp - (Cv - Cf). Then we have Cw from RANSE calculation is Cw x 10^3 = 0.342.

The wave coefficient calculated by panel method is Cw x 10^3 is 0.328. The two results are similar.

The wave pattern calculated by RANSE and panel method are shown in Figure 5 and 6

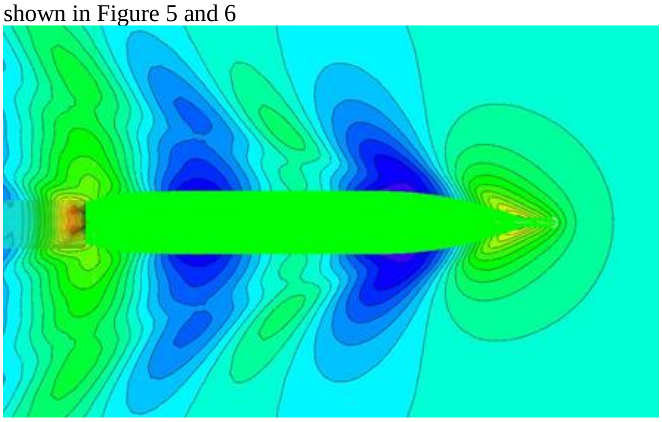

Figure 5. Wave pattern calculated by panel method

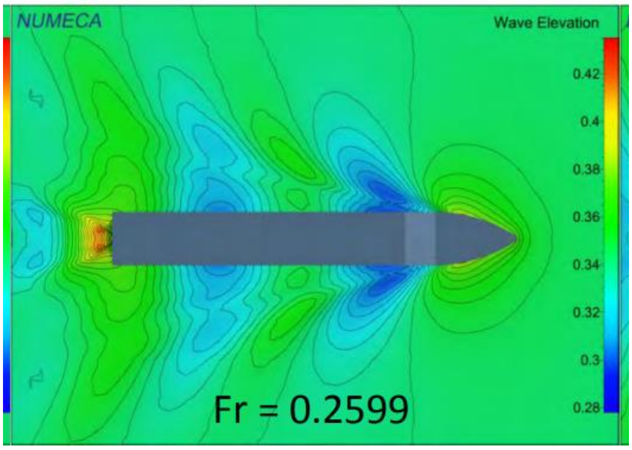

Figure 6. Wave pattern calculated by RANSE method

6.Conclusion and further study

The paper has presented the theoretical background and application of panel method to estimate the ship resistance. The calculation is performed for well-known test case and show the good result in comparison with experiment. The most advantage of panel method is fast computation so that it can be used to perform hull form optimization. In this case, many designs will be evaluated to select the best one in term of resistance. After that, further validation uses viscous flow solver such as RANSE (Reynold Averaged Navier Stoke Equation) or model test can be applied. This is becoming standard approach in practice in many ship hydrodynamics office.

However, the users should be aware of the limitation of panel method under potential flow theory as discuss above, for example, hull with deep transom, or flow separation at stern

Further development of this study can be using panel method and parametric hull form modeling to perform hull form optimization.

References

[1]. Tran Ngoc Tu, Nguyen Manh Chien, Nguyen Thi Hai Ha, Nguyen Thi Thu Quynh. Systematize the empirical methods to predict ship resistance in preliminary design stage. Proccedings of national conference on mechanical and transportation engineering Volume 2 (2017), P. 384-389.

[2].NMRI, KCS Test case, 2008, http://www.simman2008.dk/KCS/container.html

[3]. MOLLAND, ANTHONY F., 2011. Ship resistance and propulsion: practical estimation of ship propulsive power.

[4]. Holtrop J, Mennen GGJ. An approximate power prediction method, July 1982.

[5]. ITTC Conference, 1978

[6].DNVGL, Hydrodynamics Advisory Services, https://www.dnvgl.com/services/ship-hydrodynamic-performance-analysis-and-advice-4962

[7]. D.C.Kring. Time domain ship motions by a three-dimensional Rankine panel method. PhD Thesis, Massachusetts Institute of Technology, Department of Ocean Engineering, May 1994.

[8]. Ludwig Kerner, 2014, Benchmark Report KCS resistance calculation, Numeca International

[9] G. K. Saha and M. A. Miazee, “Numerical and Experimental Study of Resistance, Sinkage and Trim of a Container Ship,” Procedia Eng., vol. 194, pp. 67–73, 2017

[10] Y. M. Ahmed, O. Yaakob, M. F. A. Rashid, and A. H. Elbatran, “Determining Ship Resistance Using Computational Fluid Dynamics (CFD),” J. Transp. Syst. Eng., vol. 1, pp. 20–25, 2015

[11] Y. M. Ahmed, “Numerical simulation for the free surface flow around a complex ship hull form at different Froude numbers,” Alexandria Eng. J., vol. 50, no. 3, pp. 229–235, 2011