ITTC Quality System Manual

Recommended Procedures and Guidelines

Guideline

Guideline on the CFD-based Determination of Wind Resistance Coefficients

7.5 Process Control

7.5-03 CFD

7.5-03-02 Resistance and Flow

7.5-03-02-05 Guideline on the CFD-based Determination of Wind Resistance Coefficients

Updated / Edited by	Approved
Specialist Committee on Ships in Operation at Sea of the 29th ITTC	29th ITTC 2021
Date: 06/2020	Date: 06/2021

Table of Contents

1. INTRODUCTION 3
2. PARAMETERS AND SYMBOL 3
3. RELEVANT ITTC GUIDELINES 3
4. COMPUTATIONAL DOMAIN 3
   4.1 General Setup 3
   4.2 Discretization and Level-of-Detail 4
   4.3 Computational Grid 4
   4.4 Turbulence Modelling 4
   4.5 Boundary Layer Treatment 4
5. BOUNDARY CONDITIONS 5
6. CALCULATION PROCEDURE 5
7. EVALUATION PROCEDURE 5
8. QUALITY ASSURANCE 6
   8.1 Verification and Validation 6
9. REFERENCES AND BIBLIOGRAPHY 6

Guideline on the CFD-based Determination of Wind Resistance Coefficients

1. INTRODUCTION

The present guideline describes recommended practice in computations of aerodynamic forces acting on above the water part of the ships. It is premised to use of the complementary document to F.2 method of Appendix F in 7.5-04-01-01.1, 2017.

The guideline provides general recommendations concerning the computational model setup and execution of the computations. It also gives the unified definitions of non-dimensional force coefficients and the coordinate system for aerodynamic forces.

2. PARAMETERS AND SYMBOL

A_{VX} Transverse projected (frontal windage) area [m^2]
A_{VY} Lateral projected (Side windage) area [m^2]
C Normal component of an aerodynamic force [N]
C_{DAX} Longitudinal aerodynamic force coefficient
C_{DAY} Transverse aerodynamic force coefficient
D Tangential component of an aerodynamic force [N]
q_A dynamic pressure [Pa]
V_{AA} Wind (air) speed [m/s]
V_{A1} Height Average wind speed used for C_{DAX} [m/s]
V_{A2} Height average wind speed used for C_{DAY} [m/s]
R_{AAX} Longitudinal aerodynamic force
R_{AAY} Transverse aerodynamic force
β Angle of attack [°]
ρ_A Air density [kg/m^3]
H_{BR} Height of the top of the bridge [m]
H_L Ship's mean height in lateral plane[m]

3. RELEVANT ITTC GUIDELINES

For the application of CFD technology in maritime applications, the International Towing Tank Conference has defined best practices in several guidelines. These fundamental proceedings do also apply to the assessment of wind resistance coefficients and thus shall be taken into account accordingly. Reference is made to:

7.5-03-01-01 Uncertainty Analysis in CFD, Verification and Validation Methodology and Procedures, 2017
7.5-03-01-02 Uncertainty Analysis in CFD, Guidelines for RANS Codes, 2017
7.5-03-01-03 CFD User's Guide, 1999
7.5-03-01-04 CFD Verification, 1999
--*-* Quality Assurance in Ship CFD Application, 2020

4. COMPUTATIONAL DOMAIN

4.1 General Setup

Computational domain created for the purpose of determination of wind resistance coefficients covers only the selected volume of air surrounding the above water part of the ship, i.e. the ship geometry is cut at the waterline.

A series of calculations with different sizes of domain should be used to confirm the size of a computational domain that assures necessary and sufficient accuracy to estimate the wind forces.

The computations can be carried out at model scale and/or at full scale. In the case of validation of numerical results vs. wind tunnel data, the domain size should also correspond to the wind tunnel measurement section size.

4.2 Discretization and Level-of-Detail

In case of validation of numerical results, the CAD model of the ship used in the computations should mimic as accurately as possible the actual geometry of the wind tunnel model. In general, the adopted Level-of-Detail of the CAD model used for the computations of wind resistance coefficients should be based on realistic assessment of the computational resources and available computational time [3,4]. The recommended rule is that the generated computational mesh should assure smooth transition from smallest cells on smallest geometry details to the cells in the flow around the ship. The level of detail should then be as high as possible within affordable mesh size with taking into account the above rule. This means that e.g. covering the details with mesh makes little sense if their viscous wake cannot be modelled properly.

4.3 Computational Grid

As the wind resistance coefficients are usually evaluated for a range of inflow angles, a method of varying the direction of the flow relative to the hull must be selected. This can be realized both by rotating the ship geometry in the domain with fixed boundary conditions and single mean flow direction as well as by varying the velocity direction on the boundaries of the domain while leaving the hull location in the domain unchanged [1]. The selection of more convenient method depends on the solver features, e.g. the mesh type and possibility of solution-based mesh refinement. In both cases, it should be assured that the mean flow is not significantly affected by numerical diffusion when it is not parallel to mesh lines, and that the viscous wake of the ship superstructure is modelled equally accurate for all angles [2]. For the different flow directions the domain should be big enough to prevent larger influence of the boundaries at certain angles.

In case the geometry of the above water part is complex, an unstructured grid is usually used, but a structured grid can also be used because structured/unstructured grids have a minor effect on the calculated wind forces as long as a sufficient grid density is ensured.

Particular attention should be paid to proper discretization of the regions of the wake of ship superstructure elements. A verification of the solution in respect of mesh quality is recommended after the first run to assure that the characteristic flow structures do not get strongly affected by an insufficient mesh density. The wake modelling should be also equally accurate for all wind directions.

In the same manner, with regard to the number of elements, it is required that the results in series of calculations with different numbers of elements converge.

4.4 Turbulence Modelling

For engineering purposes, the RANS (Reynolds-Averaged Navier-Stokes) flow model is generally used with an appropriate turbulence model. The turbulence model shall be able to give accurate predictions in ship aerodynamics and the higher order models should be applied e.g.: k-epsilon or k-omega model and so on.

4.5 Boundary Layer Treatment

Due to the low contribution of shear forces to the total air resistance of the ship, full resolving of the boundary layer down to the laminar sub-layer region is not necessary. The use of wall functions and mesh resolution (wall y+)

corresponding to their requirements are applicable instead [5].

5. BOUNDARY CONDITIONS

Regardless of the adopted method of varying the direction of the flow relative to the hull, the recommended setup of boundary conditions is given below [3].

At the inlet, i.e. the upstream boundary, the prescribed velocity distribution and the uniform velocity distribution can be used as a specified wind profile.

At the outlet and top side constant pressure should be applied. No-slip wall for the waterplane is recommended especially in the case when a uniform inflow is adopted.

When validation of numerical results vs. wind tunnel data is carried out, non-slip boundary condition should be applied to a bottom boundary of the flow domain in order to mimic the shear flow due to the floor of the chamber. If the model is not exposed to local shear flow around the top or sidewalls of the chamber, then free-slip or symmetry conditions can be applied to top or side boundaries in order to save computational efforts.

6. CALCULATION PROCEDURE

In general, time resolved computations are recommended for the evaluation of wind forces due to usually blunt shape of the vessel's superstructure, resulting in unsteady, separated flow. However, both of steady and unsteady computation is applicable for estimating wind resistance. If it is necessary to capture flow separation, unsteady computation is preferable.

The values of considered incident angles should cover the range 0~180° for fully symmetrical geometries and in 0~360° for unsymmetrical ones.

The reference velocity value applied at the inlet should be selected so that the resulting Reynolds number corresponds to fully turbulent flow regime. This value should be also close to the wind speed values expected to be used in final force predictions.

The basic criterion of terminating the computations is the convergence of forces. In case of strongly unsteady flows, the duration of the run should be long enough to assure reliable evaluation of the mean value.

7. EVALUATION PROCEDURE

The output of CFD computations should be reported in compliance with the below presented coordinate system and with given definitions of the force coefficients. Fig. 1 below shows the coordinate system and sign convention for forces and wind angle to be used in reporting the results.

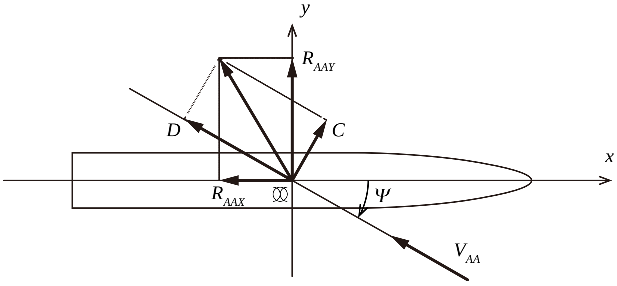

Fig. 1 Coordinate system

The force coefficients should be calculated using the aerodynamic forces expressed in the coordinate system presented above according to the following formulae:

$$C_{DAX}(\Psi) = \frac{R_{AAX}(\Psi)}{q_A \cdot A_{VX}} \quad \text{(E-1)}$$

$$C_{DAY}(\Psi) = \frac{R_{AAY}(\Psi)}{q_A \cdot A_{VY}} \quad \text{(E-2)}$$

where:

$$q_A = \frac{1}{2} \cdot \rho_A \cdot V_{AA}^2 \quad \text{(E-3)}$$

The angle of attack Ψ is 0° when the air velocity is directed to the stern. For reference, wind speed V_{AA} is recommended to use the height average wind velocities [6, 7] V_{A1} and V_{A2} for the calculation of $C_{DAX}(\Psi)$ and $C_{DAY}(\Psi)$ respectively.V_{A1} and V_{A2} are the average speed obtained from the calculated velocity profile at the origin of the coordinate system under the condition where the ship model is removed from the computation domain as shown in Fig. 2, which is obtained by dividing the rectangular area drawn with dashed lines in Fig. 2 by representative heights H_{BR} and H_L. H_{BR} is the height of the top of the navigation bridge from the sea surface and H_L is the average height which is derived from dividing the side projected area A_{VY} of a ship by the ship length (L_{OA}) as shown in the Fig. 3 and Fig. 4.

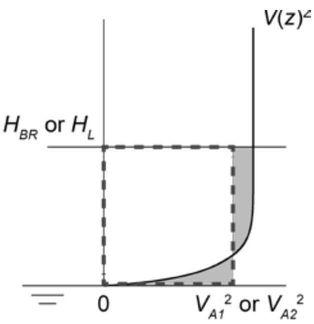

Fig. 2 Conceptual figure of height average wind velocities V_{A1} and V_{A2} analysed with integration

$$V_{AA} = \begin{cases} V_{A1} \text{ for } C_{DAX} \\ V_{A2} \text{ for } C_{DAY} \end{cases} \quad \text{(E-4)}$$

$$V_{A1}{}^2 = \frac{1}{H_{BR}} \int_0^{H_{BR}} V(z)^2 dz \quad \text{(E-5)}$$

$$V_{A2}{}^2 = \frac{1}{H_L} \int_0^{H_L} V(z)^2 dz \quad \text{(E-6)}$$

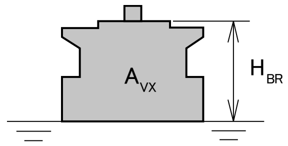

Fig. 3 Conceptual figure of H_{BR}

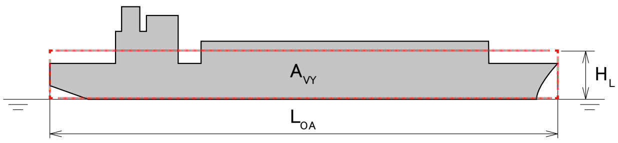

Fig. 4 Conceptual figure of H_L

8. QUALITY ASSURANCE

8.1 Verification and Validation

The first step in validation of CFD based wind forces coefficients was reported in 29th SOS Committee report. It presents the comparison between wind tunnel tests and a CFD case study with the level of confidence and margin of differences for two ships' models.

9. REFERENCES AND BIBLIOGRAPHY

[1] Wnęk, A. D., & Guedes Soares, C. (2015). CFD assessment of the wind loads on an LNG carrier and floating platform models. Ocean Engineering, 97, 30–36. doi:10.1016/j.oceaneng.2015.01.004

[2] Versteeg, H.,Malalasekera,W.,2007.An Introduction to Computational Fluid Dynamics: The Finite Volume Method. Pearson Education Ltd., Harlow, England.

[3] Aage, C., Hvid, S. L., Hughes, P. H. and Leer-Andersen, M. (1997). Wind loads on ships and offshore structures estimated by

CFD. In 8th Int. Conf. on Behaviour of Offshore Structures (BOSS), Delft

[4] Werner Blendermann, Katrin Hellwig & Eberhard Schuckert (2011) Wind Loads on a Passenger/Car Ferry by CFD Computations and Wind Tunnel Tests, Ship Technology Research, 58:2, 82-88

[5] Launder, B.E. and Spalding, D.B. 1974. “The Numerical Computation of Turbulent Flows”, Computer Methods in Applied Mechanics and Engineering, 3, pp. 269-289.

[6] Kume, K., Ohba, H., Orihara, H. and Mizokami, S.: Wind Velocity Profile and Representative Wind Velocity for Wind Resistance Measurement of Ship Models, Journal of JASNAOE, Vol.30, pp.1-13, 2019

[7] Kume, K., Bielicki, S., Kobayashi, H. and Ohba, H.: Validation of dimensionless method using height average wind velocity for wind forces, Journal of JASNAOE, Vol.31, 2020

[8] Wide, J.J., Schrijvers, P., Witz, J.: CFD Benchmark Study from the Wind Load Joint Industry Project, Offshore Technology Conference, OTC-28832-MS, 2018

[9] Yoo, D., Jong-Chun, P., Schrijvers, P., Koop, A.: Predicting wind loads on single vessels and in side by side offloading configuration for FPSO and shuttle, Proceedings of the Twenty-eighth (2018) International Ocean and Polar Engineering Conference, Sapporo, Japan, June 10-15, 2018