Table of Contents

Edited	Approved
Propulsion Committee of 27th ITTC	27th ITTC 2014
Date : 02/2014	Date : 09/2014

Cavitation-Induced Pressure Fluctuations: Numerical Prediction Methods

1. PURPOSE OF PROCEDURE

The purpose of this procedure is to ensure accurate, consistent and reliable full-scale predictions of cavitation-induced pressure fluctuations.

The primary background document for this procedure is the report of the 23rd ITTC Specialist Committee on Cavitation Induced Pressure Fluctuations. References are given for typical methods. However, cited references do not supersede other, similar methods not cited.

2. NUMERICAL PREDICTION OF CAVITATION-INDUCED PRESSURE FLUCTUATIONS

This section is written to provide guidance to naval architects in shipyards, owners and consultancies, including model basins, on how to use available methods.

Methods for calculation of cavitation-induced pressure generally fall into two categories: one that is based on empiricism, relying heavily on model test results, and one that is based on solving the flow problems by first principles. The two types of method will be treated separately here.

For both types of method, sometimes a code user cannot access or revise a code. This limits useful application, since theoretical or numerical inadequacies may not be apparent to a user, or may not be addressable via his own modifications to the code.

2.1 Empirical Methods

Empirical methods are based on analysis of measured data, typically model test results. The analyses are usually statistical (e.g. Holden et al., 1980), but methods using neural networks (Koushan et al., 2000) are also used. For a successful analysis, a large number of tests should be included. However, the many parameters describing ship and propeller geometries and cavitation test conditions, as well as the constant development of ships and propellers, make it difficult to collect a sufficient amount of data. With these reservations in mind, empirical methods should be used in the early stage of design, particularly for a relatively traditional ship and propulsion arrangement.

Generally, the user of an empirical method should make sure that the ship and propeller under consideration are covered by the cases upon which the method was built. For this purpose it is most helpful if thorough documentation is available, including correlation with both model- and full-scale measurements.

The wake distribution is very difficult to predict with sufficient accuracy at the early stage of design. It can be done on a statistical basis for the type of ships under consideration, or by using a simplified description of the ship hull form. Usually the mean wake and the wake peak are needed.

The propeller geometry is usually described with a few overall parameters, including diameter, number of blades, pitch, and blade area ratio, skew and thickness. Tip unloading can be indicated by a reduction in pitch. It is important that the basis of the method comprises the specific characteristics of the propeller in question.

Prediction of details of cavitation such as type (sheet, bubble, tip vortex etc.) and extent are not necessarily included in empirical methods. This generality of course requires that the cavitation performance of the propeller under investigation is similar to that of the propellers that are the basis of the method. If the cavitation is represented for instance by a simple formula for sheet cavitation, it should be evaluated whether this type is representative for the cavitation of the propeller being investigated.

Predictions of pressure are usually done at a few points or maybe at a single, representative point on the hull surface. Usually only pressures at blade frequency are predicted, but pressures at twice blade frequency may be calculated. The accuracy of those pressures is rather limited. If forces are required, pressure fluctuations should be predicted at several points, with phases.

2.2 Numerical Methods

Ideally it ought to be possible to make a complete numerical calculation of cavitation-induced hull pressures given the hull, propeller geometry and operating conditions. However, most methods rely to some extent on model test results, in particular the onset flow to the propeller, i.e. the ship wake field, and the loading condition. Moreover, for the most part only sheet cavitation can be predicted with reliability, whereas it is somewhat more difficult, if at all possible, at the present stage of development to treat the other types of cavitation (22nd ITTC Specialist Committee on Computational Methods for Propeller Cavitation, 1999).

2.2.1 Ship Wake Field

The ship wake used as onset flow to the propeller can be computed directly, for instance by RANS methods (Larsson et al., 2000). The most common procedure, however, is to use results of wake surveys from model experiments. Many organizations scale those data to full scale and to effective wake. On the assumption that all calculations deal with the full-scale flow, both corrections should in principle be applied.

The loading condition J_T-K_T or J_Q-K_Q can also be defined on the basis of calculation only for full scale, but generally results of propulsion tests are used. Those results should be corrected to full scale, along with the wake distributions, to ensure the best description of the full-scale case.

Calculation of the effective wake, taking into account the interaction between the inflow vorticity and the propeller can be done by coupling a propeller panel method with an Euler or RANS solver (Choi, 2000; Choi & Kinnas, 2000a, 2000b, 2001; Rijpkema *et al*., 2013; Krasilnikov, 2013; Sánchez-Caja *et al*., 2014).

2.3 Calculations for Non - Cavitating Propeller

Calculation of the flow over a propeller in the non - cavitating condition can be considered as the first step in the total computation. This calculation should generally be done by lifting-surface (vortex-lattice) or boundary-element (panel) methods that best and most reliably describe the flow (22nd ITTC, 1999). This approach applies in particular to high-skew propellers. RANS methods for propellers in the flow abaft a ship hull have been developed but still need more validation. Those methods are still impractical for routine use.

2.3.1 Cavitation Prediction

The prediction of cavitation serves dual purposes. First, it provides the basis for assessment of the detailed cavitation performance of a propeller with a view to modification if in a design situation. Second, it gives the cavity geometry, including its history, i.e. the time variation of the cavity volume. This variation is necessary for most hull surface-pressure calculation methods.

This committee has found it impossible to recommend one particular method for predicting cavitation. The general recommendation is to use the most-advanced and most-complete procedure available. The user should be aware of the limitations of the method used, both in the theory and found in comparisons with experiments. In the following paragraphs, recommendations, where possible, are given for prediction of the various types of cavitation.

Sheet cavitation is the most common and easiest type of cavitation to deal with theoretically. Most numerical procedures address this type. Some methods use 2-D cavitating profile techniques along with lifting-surface and boundary-element procedures for non - cavitating propellers. The most-advanced methods treat cavitation as an integral part of the procedure, with the non - cavitating analysis as a first step of an iterative procedure. Methods that can address partial as well as supercavitation should be used (Kinnas & Fine, 1994). Face (sheet) cavitation should be included, in particular if off-design conditions or controllable-pitch propellers are treated.

Tip vortex cavitation is an important type, in particular for high-skew propellers, for which the effects of blade sheet cavitation have been much reduced and where tip vortex cavitation plays an important role in fluctuating pressures. Only a few methods have been presented that address this type of cavitation (Szantyr, 2000).

For cloud, bubble, root and hub vortex cavitation, only a few methods have been published and are in use. For bubble cavitation, a method (Szantyr, 2000) for assessment relies on the dynamics of a test nucleus in the pressure field on the blade. For cloud cavitation there appears to be no reliable means of prediction.

RANS and two-phase flow methods are promising for the problem of a propeller in an inhomogeneous inflow with unsteady cavitation, though at present, its simulation accuracy (for sheet cavitation) still depends on many aspects, such as grid density, discretization scheme, turbulence model, and cavitation model, etc. (Salvatore, et al. 2009).

2.4 Hull Pressure Calculation

As stated by the 22nd ITTC Specialist Committee on Cavitation Induced Pressure Fluctuations (22nd ITTC, 1999 pp. 555), “the key for the accurate prediction of unsteady hull pressures is accurate prediction of time variation of cavity volume.” Assuming that this has been achieved in the earlier step of the calculation, the pressure in an unbounded fluid can be computed by the unsteady Bernoulli equation. Alternatively the acoustic wave propagation equation can be used (e.g. Bloor & Kinns, 2000), but this is hardly worthwhile for points close to the propeller. It is more difficult to include the effects of the hull and the free surface. Many organizations use solid hull boundary factors. Such factors should take into account the shape of the hull and the position of the points where the pressure is calculated. A more accurate, but also more complicated, approach is to include the actual hull shape. Here an additional boundary-value problem must be solved with no water penetrating the ship surface (Neumann condition) and usually a high-frequency condition on the free surface. For example, by regarding the unsteady sheet cavity as a modification to blade surface shape, the boundary-value problem for the hull with such a propeller having time-varying geometry can be solved by the surface panel method (Kanemaru and Ando, 2011). However, it remains unknown that, even if the time variation of cavity volume is accurately simulated, how much error would be introduced by solving the hull-propeller problem with a potential flow method.

2.4.1 Presentation of Results

Results should be presented in terms of pressure amplitudes, phases, and time series up to typically third blade rate. It is hardly realistic that higher-order amplitudes can be predicted with sufficient reliability. Results of cavitation calculations should also be presented, stating which cavity model was used and which types of cavitation were treated.

3. PARAMETERS

3.1 Parameter to be Taken into Account

The main parameters that need to be considered during pressure fluctuation computations are presented below. If we include the parameters "worthwhile to have," the computational accuracy could be improved.

General Information:

- Type of ship
- Engine power and RPM
- Propeller main particulars (diameter, blade number)
- Shaft immersion
- Tip clearance
- Ship main particulars (worthwhile to have)
- Propeller design conditions (worthwhile to have)

Propeller Operating Conditions:

- Onset flow axial velocity (ship wake) distribution
- Propeller RPM, thrust or torque
- Onset flow tangential and radial velocity distribution (worthwhile to have)
- Stern wave height (worthwhile to have)

Propeller Geometry:

- Detailed geometry (radial distributions of pitch, chord, skew, rake, and thickness; chordwise thickness and camber shapes)

Hull Geometry:

- Drawing of stern shape including arrangement of appendages to construct either calculation grid or hull boundary factors
- Offsets of ship stern and appendages (worthwhile to have).

4. VALIDATION

4.1 Uncertainty Analysis

The trend of increasing reliance on numerical predictions in the shipbuilding community motivates a better understanding of the uncertainty of these predictions. A rigorous Verification and Validation (V&V) procedure has been proposed for CFD simulations (Stern, et al., 2001; 22nd ITTC 1999, pp. 213-218 (Uncertainty Analysis for CFD); ITTC Quality Manual Section 4.9-04-01-0l). However, there is no universally accepted V&V procedure for CFD.

While uncertainty assessment is well established in experimental fluid dynamics (EFD), it is still controversial in CFD (Larsson et al., 2000). One should not lose sight of the traditional comparisons of experimental data and computations and base everything on the V&V results (Ebert and Gorski, 2001).

4.2 Benchmark Tests

The following selected ITTC reports presented experimental results of fluctuating pressures on hull. Since detailed data on neither propeller nor ship were presented, it is still not possible to do any comparative calculations based on the information in these reports alone.

(1) Comparative Noise Measurements with the Sydney Express Propeller Model (16th 1981, vol. 1 pp.447-453)

(2) Comparison of Hull Pressure Amplitudes for Sydney Express Propeller (17th 1984, vol. 1, pp.248-252)

(3) Comparison of Propeller-Induced Hull Pressure Measurements for the "SYDNEY EXPRESS" Propeller Models (18th 1987, vol. 1, pp.209-210)

(4) Propeller-Induced Hull Pressures (19th 1990, vol. 1, pp.182-187)

(5) Further Measurement of Pressure Fluctuation on 'SYDNEY EXPRESS' Propeller (19th 1990, vol. 1, pp.213-219)

(6) Comparative Measurements on German Tanker "St. Michaelis" and the "Sydney Express" (20th 1993, vol. 1, pp.230-231)

(7) Comparative Measurement of Pressure Fluctuation on the "St Michaelis" (20th 1993, vol. 1, pp.236-240)

(8) Measurements of Hull Pressure Fluctuation (21st 1996, pp.65-69)

(9) Measurement of Hull Pressure Fluctuation, Round Robin Tests (22nd 1999, vol. 2, pp.547-585)

5. REFERENCES

Bloor, C. and Kinns, R. (2000), "Development of acoustic boundary element models for the prediction of fluctuating hull forces due to propeller," Proc. NCT'50 Int. Conf. on Propeller Cavitation, Newcastle, U.K. pp 247 - 262.

Choi, J.K. (2000), "Vortical Inflow - Propeller Interaction Using an Unsteady Three-Dimensional Euler Solver," Ph.D. Thesis, Dep't of Civil Engineering, The University of Texas at Austin.

Choi, J.K. & Kinnas, S. (2000a), "Non-axisymmetric Effective Wake Prediction by Using an Unsteady Three Dimensional Euler Solver," Proc. Propellers/Shafting 2000 Symposium, Virginia Beach, USA.

Choi, J.K. & Kinnas, S. (2000b), "An unsteady 3-D Euler solver coupled with a cavitating propeller analysis method," Proc. 23rd Symp. on Naval Hydrodynamics, Val de Reuil, France, pp. 156-176.

Choi, J.K. and Kinnas, S.A. (2001), "Prediction of non-axisymmetric effective wake by a three-dimensional Euler solver," J. of Ship Research, Vol. 45, No. 1, pp. 13-33.

Ebert, M.P. and Gorski, J. (2001), "A Verification and Validation Procedure for Computational Fluid Dynamics Solutions," NSWC, Carderock Division Report No. 50-TR-2001/0006.

Holden, K.O., Fagerjord, O. and Frostad, R. (1980), "Early design-stage approach to reducing hull surface forces due to propeller cavitation", Trans. SNAME, vol. 88, pp. 403-442.

Kanemaru, T. Ando, J. (2011), "Numerical Analysis of Pressure Fluctuation on Ship

Stern Induced by Cavitating Propeller Using a Simple Surface Panel Method ‘SQCM’,” Second International Symposium on Marine Propulsors (SMP’11), Hamburg, Germany.

Kinnas, S. and Fine, N. (1994), “A Nonlinear Boundary Element Method for the Analysis of Unsteady Propeller Sheet Cavitation” Proc 19th Symp. on Naval Hydrodynamics, pp. 717-737.

Koushan, K., Halstensen, S.O. and Sandtorv, L.G. (2000), “Systematic investigation of blade design influence on cavitation performance and on induced pressure pulses,” Proc. NCT’50 Int. Conf. on Propeller Cavitation, Newcastle, U.K. 175 - 191.

Krasilnikov V. I. (2013), “Self-Propulsion RANS Computations with a Single-Screw Container Ship”, Third International Symposium on Marine Propulsors (SMP’13), Launceston, Australia.

Larsson, L., Stern, F., and Bertram, V., (eds), (2000), Göteborg 2000 Workshop on Numerical Ship Hydrodynamics, Göteborg, Sweden.

Rijpkema D., Starke B., Bosschers J. (2013), “Numerical Simulation of Propeller-Hull Interaction and Determination of the Effective Wake Field Using a Hybrid RANS-BEM Approach”, Third International Symposium on Marine Propulsors (SMP’13), Launceston, Australia.

Salvatore, F., Streckwall, H., and van Terwisga, T. (2009), “Propeller cavitation modelling by CFD - Results from the VIRTUE 2008 Rome workshop”, Proc., First Intl. Symp. on Marine Propulsors (SMP’09), Trondheim, Norway.

Sánchez-Caja A., Martio J., Saisto I., Siikonen T. (2014), “On the Enhancement of Coupling Potential Flow Models to RANS Solvers for the Prediction of Propeller Effective Wakes”, Journal of Marine Science and Technology, published online in February 2014.

Stern, F., Wilson, R.V., Coleman, H., and Paterson, E. (2001), “Comprehensive Approach to Verification and Validation of CFD Simulations – Part 1: Methodology and Procedures,” ASME J. of Fluids Eng.

Szantyr, J.A. (2000), “Analytical methods for prediction of propeller cavitation and its consequences,” Proc. NCT’50 Int. Conf. on Propeller Cavitation, Newcastle, U.K. pp. 37 - 56.