JOURNAL OF MARITIME RESEARCH

Vol. X. No. 3 (2013), pp. 47 - 50

ISSN: 1697-4040, www.jmr.unican.es

Effect of Structural Deformation on Performance of Marine Propeller

H.N. Das[1,2,*], P. Veerabhadra Rao[1,3], C. Suryanarayana[1,4] and S. Kapuria[5]

ARTICLE INFO

Article history:
Received 15 July 2013;
in revised form 30 July 2013;
accepted 16 October 2013

Keywords:
Marine propeller, structural deformation, hydrodynamic.

ABSTRACT

Propeller geometry is very crucial for its performance and a little deviation in shape can cause changes in its hydrodynamic performance. Hydrodynamic loading causes deformation to the propeller blades, which leads to change in shape. Effect of this change of shape on hydrodynamic performance of a propeller is being studied in the present paper. A five bladed bronze propeller is chosen for the analysis. Its open water efficiency was estimated for original and deformed shape. Pressure based RANS equation was solved for steady, incompressible, turbulent flow through the propeller. Numerical solution was obtained using Finite Volume Method within Ansys Fluent software. FEM based solver of ANSYS Mechanical APDL was used to make the structural calculations. Fluid-structure interaction was incorporated in an iterative manner. The study however shows very little change in its hydrodynamic performance due to the deformation of propeller blades.

1. Introduction

Geometry of propeller is very crucial for its performance. A little deviation in its geometry may largely influence the performance of a propeller. A previous study reveals that some deviation in geometry of a propeller during fitting into a ship caused variation in its performance from its original design (Das, 2008).

This raised curiosity about performance of any propeller when it is deformed under hydrodynamic loading. The present study concentrates on open water performance of a five bladed propeller. CFD analysis was carried out for undeformed geometry of the propeller to obtain hydrodynamic pressure. This pressure was then applied to the propeller to estimate its deformations. A FEM code ANSYS Mechanical APDL was used for this. A further CFD analysis was carried out with this deformed shape to get the hydrodynamic performance of the deformed propeller. This process was repeated for few times to arrive at hydrodynamic load and a compatible deformed shape of the propeller.

2. Literature review

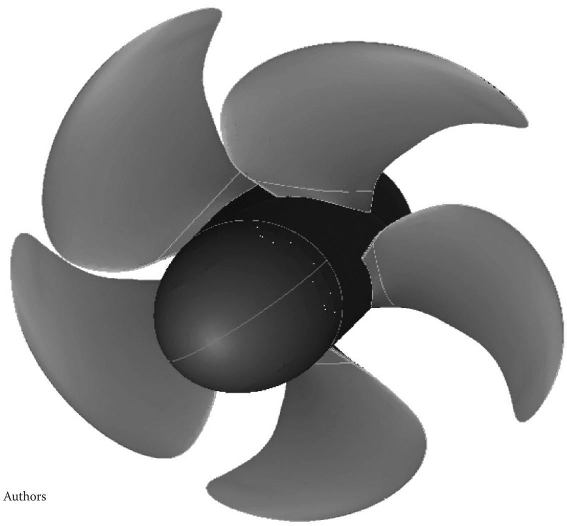
Computation of viscous flow through propeller was demonstrated in 22nd ITTC conference in Grenoble, France in 1998 (Chung et al., 1998; Sanchez Caja, 1998). In the last decade, Das et al. have carried out CFD analysis of contra-rotating propeller (Das and Jayakumar, 2002), hull-propeller interaction (Banerjee et al., 2007) and study of propeller noise (Krishna et al., 2008). Many studies on static analysis of propeller blades are available in literature. Stress analysis for isotropic material by Sudhakar (2010) and study for composite propeller by Seetharama et. al. (2012) are some examples.

3. Geometry of the propeller

A five bladed propeller is considered for the present study (Fig. 1). Considering its diameter to be as D, other geometrical parameters are expressed. The hub diameter is 0.313D. Pitch ratio (p/D) of its blades at radial section of 0.7D is 1.547. The propeller was modelled using Catia V5® software.

[1] Naval Science & Technological Laboratory (NSTL), Defence RD Organisation, Vigyan Nagar, 530027, Visakhapatnam, India. Tel +918912586373, Fax +918912559464.
[2] 2Additional Director, Email: das.hn@nstl.drdo.in.
[3] 3Researcher, Email: veerabhadrarao.peta@gmail.com.
[4] 4Head Propulsion Division, Email: suryanarayana.ch@nstl.drdo.in, Tel +918912586165.
[5] 5Professor, Indian Institute of Technology Delhi, Hauz Khas 11016, New Delhi, India. Email: kapuria@am.iitd.ac.in, Tel +911126591218, Fax +911126581119.
[*] Corresponding author.

Figure 1. Solid Model of Propeller.



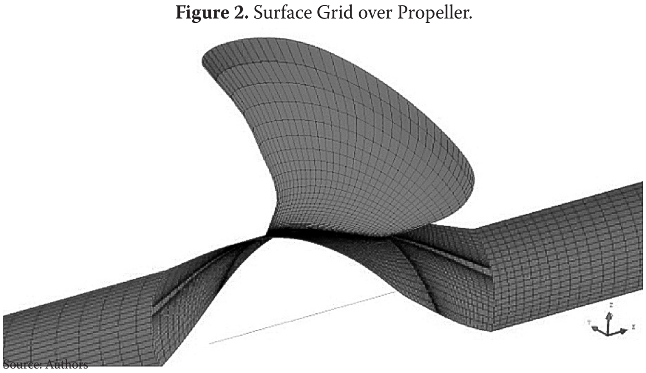
Source: Authors

4. Grid generation

4.1. Grid for Fluid Study

The surface model of propeller was imported from Catia to ANSYS ICEM CFD 12.0. A suitable domain size was considered around the propeller to simulate ambient condition. A sector of a circular cylindrical domain of diameter ~4D and length of ~7D was used for flow solution. The sector of 72° was so chosen that only one blade is modelled in the domain. Periodic repetition of this sector simulates the whole problem. A multi-block structured grid was generated for the full domain using ICEM CFD Hexa module. The grid thus generated was exported from ICEMCFD to ANSYS Fluent 12.0 solver. Extent of domain and grid over the blade is shown in Figs. 2 and 3. A grid with total 0.268 million cells were employed to descritise the flow field.

Figure 2. Surface Grid over Propeller.

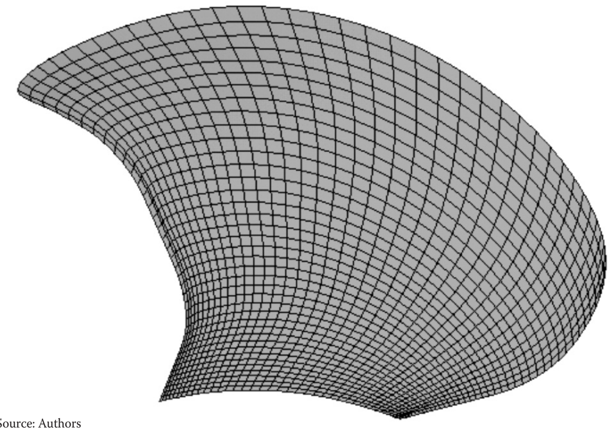


Source: Authors

4.2 Grid for Structural Analysis

The grid from only the blade surface was imported to ANSYS mechanical APDL software. A view of imported mesh is shown in Fig 5. Total 361 elements (around 400 Nodes) were used over the blade.

Figure 3. Grid over Surface of Blade for Structural Analysis



Source: Authors

5. Settings up the problem

5..1 Flow Solution

The problem was solved using the segregated solver of ANSYS Fluent 12.0. In brief the code uses a finite volume method for discretization of the flow domain. The Reynolds Time Averaged Navier-Stokes (RANS) Equations were framed for each control volume in the discretised form. For the present solution, STANDARD scheme is used for pressure and a SIMPLE (Strongly Implicit Pressure Link Equations) procedure is used for linking pressure field to the continuity equation. The detailed formulation of numerical process is given in (ANSYS FLUENT). The computations were carried out on an Eight Core Dell Precision T7500 Workstation (64bit Xeon E5640 Processor @2.67 GHz, 4GB RAM, 64 Bit Windows XP OS). The flow is treated as incompressible and fully turbulent. Standard K-Є model has been used for modelling turbulence. The near wall turbulence was modelled using standard wall functions and the free stream turbulence has been prescribed as follows

$$K = 10^{-4} * U_{\infty}^{2}$$

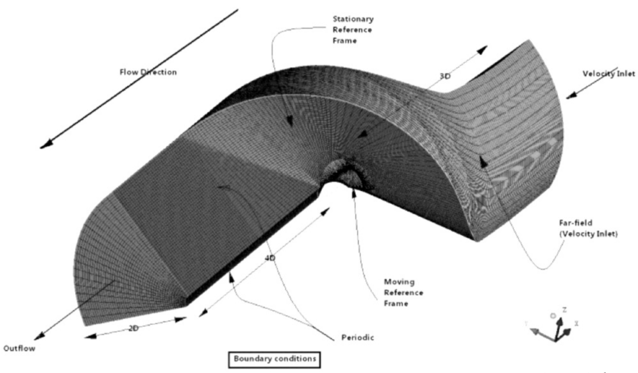
$$\varepsilon = \frac{C_{\mu}\ \rho\ K^{2}}{5\ \mu}$$

The continuum was chosen as fluid and the properties of water were assigned to it. A moving reference frame is assigned to fluid with different rotational velocities to simulate appropriate advance ratio. The wall forming the propeller blade and hub were assigned a relative rotational velocity of zero with respect to adjacent cell zone. A constant uniform velocity was prescribed at inlet. At outlet outflow boundary condition was set. The farfield boundary was taken as inviscid wall.

The following boundary conditions are used in this analysis [Fig. 4]:

(i) Velocity Inlet
(ii) Outflow
(iii) Moving Wall
(iv) Inviscid Wall
(v) Periodic

Figure 4. Extent of Domain and Boundary Conditions for Flow Analysis



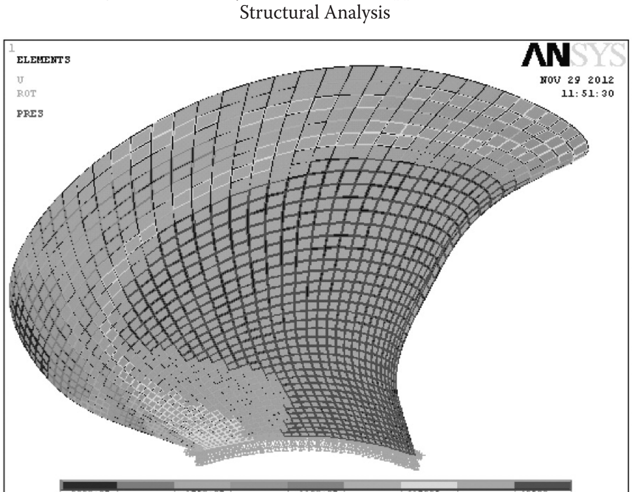
Source: Authors

5.2. Deformation Study

The deformation of the propeller blade was estimated using ANSYS Mechanical APDL 12.0 software. The solver used Finite Element Method (FEM) for descritisation. For structural analysis, only one surface of the blade was modelled. The pressure, estimated from flow solution, was applied to this blade surface. Fluent's output of pressure distribution over two surfaces of blade, face and back, was written to a file. A program picked up the pressure values from this file and put to the nearest node points over the single surface of the blade, to be used in Mechanical APDL software. An four nodded shell elements i.e., SHELL 181, available with ANSYS solver were chosen for the analysis. Propeller blade was considered as cantilever. The root of the blade was considered as fixed, restraining all degrees of freedoms there.

The blade was made of Aluminium Nickel Bronze, which has Young's Modulus 10^{11} N/m^2 and Poisson's Ratio of 0.34. A constant thickness of 0.1 m was applied for the blade. This makes the volume of the blade approximately same to the actual blade. Mesh and boundary condition for FE solver is shown in Fig 3 and 5.

Figure 5. Boundary Conditions with Applied Pressure for Structural Analysis



Source: Authors

5.3. Fluid-Structure Interaction

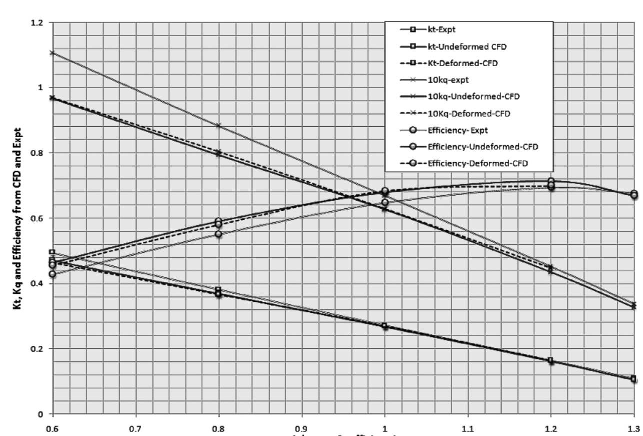
The deformed shape of the propeller blade under each operating condition was transferred to ICEM-CFD software. After developing the actual blade around this deformed surface, mesh was again generated. This mesh was exported to Fluent and corresponding operational conditions in terms of propeller rpm and linear velocity was assigned in the solver. The hydrodynamic results obtained from flow solution represent the behaviour of the deformed propeller. A new pressure distribution now develops over the blade due to the change in geometry. The new load is again exported to ANSYS APDL software for deformation analysis. The original blade geometry is considered for this. The process is repeated iteratively till the time when pressure distribution does not change any further between two successive iterations.

6. Conclusions

The present study indicates that capability of computational methods to solve complex engineering problem like fluid-structure interaction for a propeller-flow.

CFD results agreed well with experimental observations (Fig. 6) giving good validation of this study.

Figure 6. Open Water Characteristics for Deformed & Undeformed Shape.



Source: Authors

Deformation of this metallic propeller is found to be small and hence the hydrodynamic performance of propeller remains almost the same before and after deformation.

Study shows that a bronze propeller is rigid enough to hold its shape under operational conditions, so that its hydrodynamic performance is not affected due to structural deformations.

7. Future works

A composite propeller is expected to deform more than metallic one. The present propeller with composite material may be analysed to ascertain that. A detailed fluid-structure-interaction study will be carried out for this.

Nomenclature

D	Diameter of Propeller
J	Advance Ratio
K_t	Coefficients of thrust
K_q	Coefficients of torque
n	Revolution per second for propeller
p	Pitch
Q	Torque of Propeller
T	Thrust of Propeller
$U\infty$	Free-stream Velocity
η	Efficiency
μ	Viscosity
ρ	Density of Water

References

ANSYS FLUENT® 12.0 Documentation.

Banerjee, C.N., Das, H.N. and Srisudha, B. (2007): Computational Analysis And Experimental Validation of Hull Propulsor Interaction For An Autonomous Underwater Vehicle (AUV), *Proceedings of the Seventh Asian CFD Conference*, 26-30 November, Bangalore, India.

Chung, K.N.; Stem, F. And Min, K.S. (1998): Steady Viscous Flow Field Around Propeller P4119, Propeller RANS/ Panel Method Workshop, *22nd ITTC Conference,* Grenoble, France.

Das, H.N. (2008): CFD Analysis for Cavitation of a Marine Propeller, *Proceedings of the 8th Symposium on High Speed Marine Vehicles,* 22-23 May, Naples, Italy.

Das, H.N. and Jayakumar, L.C.P. (2002): Computational Prediction and Experimental Validation of the Characteristics of a Contra-Rotating Propeller, *Proceedings of the NRB seminar on Marine Hydrodynamics,* 1-2 February, Visakhapatnam, India.

Ghose, J.P. and Gokarm, R.P. (2004): *Basic Ship Propulsion.* Kharagpur: Allied Publishers Pvt Ltd.

Krishna, G.V.; Saji, V.F.; Das, H.N. and Panigrahi, P.K. (2008): Acoustic Characterization of a Benchmark Marine Propeller Using CFD, *Proceedings of the National Symposuim on Acoustics (NSA-2008),* 22-24 December, Visakhapatnam, India.

Lewis, E.V. (1998): *Principles of Naval Architecture Volume II*. New Jersey, NJ: The Society of Naval Architects and Marine Engineers.

NSTL (2010): *NSTL Internal Report on Hydrodynamic Model Tests For New Design Frigate (Open Water, Self Propulsion & 3d Wake Survey Tests).* NSTL.

Sanchez Caja, A. (1998): P 4119 RANS Calculations at VTT, *22nd ITTC Conference,* Grenoble, France.

Seetharama Rao, Y.; Mallikarjuna Rao, K. and Sridhar Reddy, B. (2012): Stress Analysis of Composite Propeller by Using Finite Element Analysis, International Journal of Engineering Science and Technology (IJEST) 4(8).

Sudhakar, M. (2010): *Static & Dynamiz Analysis of Propeller Blade*. Thesis (PhD). Andhra University.