JOURNAL OF MARITIME RESEARCH

Vol XI. No. III (2014) pp 43–49

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

Effect of Structural Deformation on Performance of Marine Propeller

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

ARTICLE INFO

Article history:
Received 20 August 2014;
in revised form 20 september 2014;
accepted 30 October 2014.

Keywords:
Propeller RANS,
Fluid-Structure-Interaction,
Deformation, FVM, FEM

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 Reynold's Averaged Navier Stokes (RANS) equation was solved for steady, incompressible, turbulent flow through the propeller. Numerical solution was obtained using Finite Volume Method (FVM) 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 (Edward V. Lewis et. al., 1988), (Ghosh J. P. et. al., 2004). 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 H. N., 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

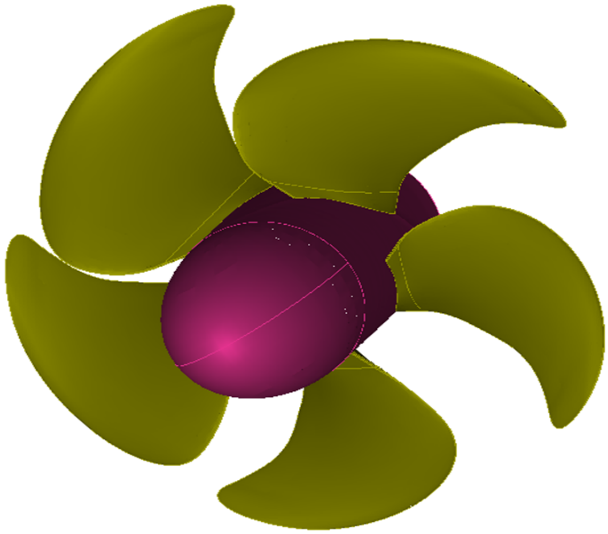
Computation of viscous flow through propeller was demonstrated in 22^{nd} ITTC conference in Grenoble, France in (Chung K. N. et. al., 1998); (Sanchez Caja A., 1998). In the last decade, author has carried out CFD analysis of contra-rotating propeller (Das H. N. et. al., 2002), hull-propeller interaction (Banerjee N. et. al., 2007) and study of propeller noise (Krishna Kumar G. V. et. al., 2008). Many studies on static analysis of propeller blades are available in literature. Stress analysis for isotropic material by Sudhakar M. (2010) and study for composite propeller by Seetharama Rao Y. et. al. (2012) are few 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 Lab. Defence Research & Development Organization. (NSTL), DRDO, Visakhapatnam, India
[2]Scientist. Additional Director. CFD Division. E-mail address: das.hn@nstl.drdo.in.
[3]Researcher, NSTL, DRDO, Visakhapatnam, India
[4]Professor, Applied Mechanics Department, Indian Institute of Technology (IIT), Delhi, India .
*Corresponding author. E-mail address: das.hn@nstl.drdo.in.

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^o 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. Size of grid and extent of domain was decided from prior experience of CFD studies for other propellers. So a separate grid and domain independence study is not done here.

4.2. Grid for Structural Analysis

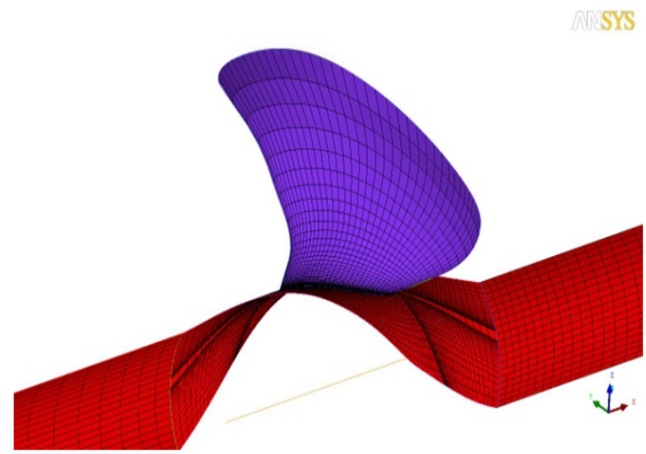
The grid from only the blade surface was imported to ANSYS mechanical APDL software. A view of imported mesh is shown in Fig 4. Total 361 elements (around 400 Nodes) were used over the blade.

5. Settings up the Problem

5.1. Flow Solution

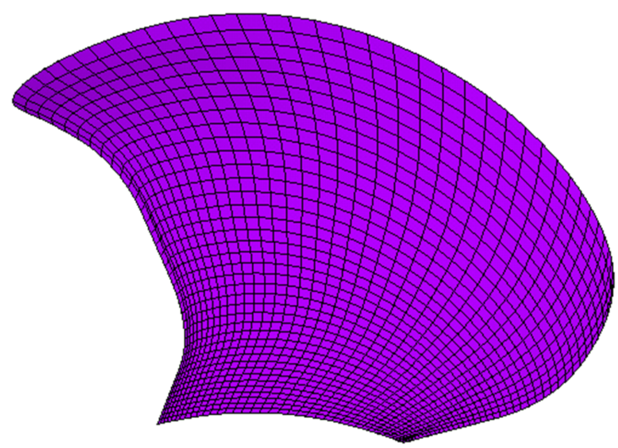
The problem was solved using the segregated solver of ANSYS Fluent 12.0. In brief the code uses a finite volume method (FVM) for discretization of the flow domain. The Reynolds Time Averaged Navier-Stokes (RANS) Equations were framed for each control volume in the discretised form. For the present solution, standard scheme is used for pressure and a SIMPLE

Figure 2: Surface Grid over Propeller.



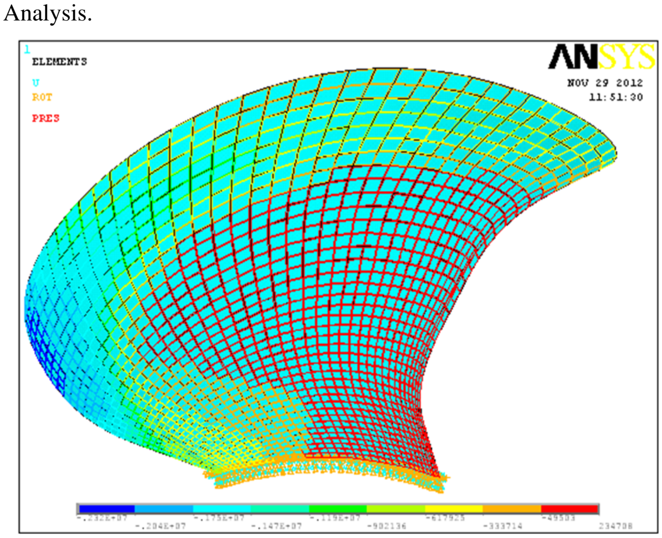
Source: Authors

Figure 3: Grid over Surface of Blade for Structural Analysis.



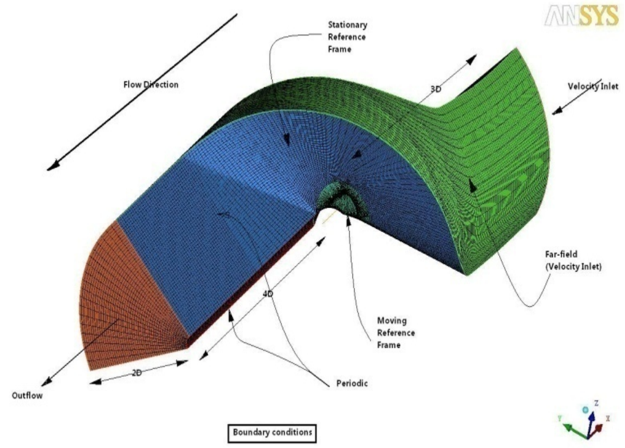
Source: Authors

Figure 4: Boundary Conditions with Applied Pressure for Structural Analysis.



Source: Authors

Figure 5: Extent of Domain and Boundary Conditions for Flow Analysis



Source: Authors

(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 Documentation (2011). 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-C 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

$$K = 10^{-4} U_\infty^2, \quad \epsilon = \frac{C_\mu \rho K^2}{5\mu}$$

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. 5):

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

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 eight 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 1011 N/m2 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 4 and 5.

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. Results

Analysis is carried out for the hydrodynamic performance of the propeller. Open water characteristics i.e., thrust (K_t) and torque coefficients (K_q) as well as efficiency (η) were computed at different advance ratios (J), defined as

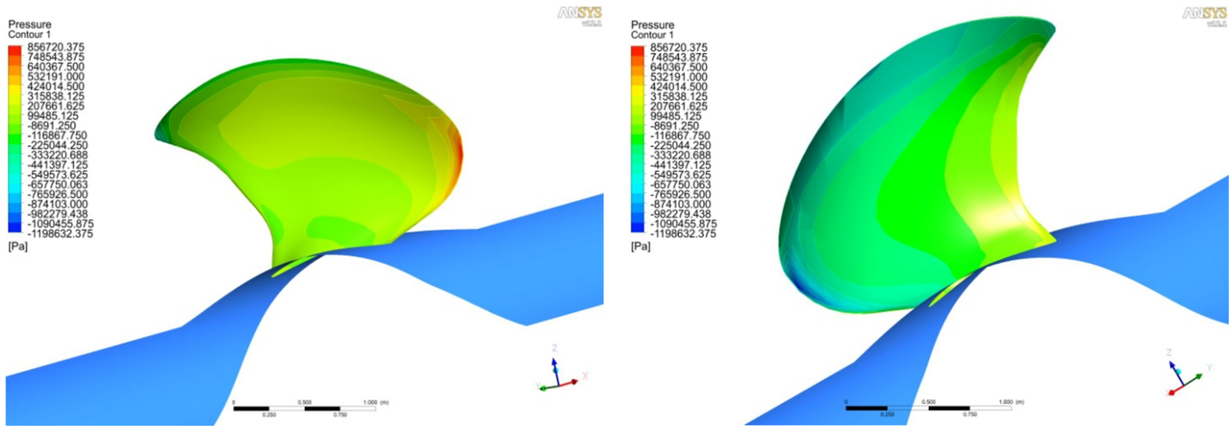
$$K_T = \frac{T}{\rho n^2 D^4}, \quad K_Q = \frac{Q}{\rho n^2 D^5} \tag{1}$$
$$\eta = \frac{J}{2\pi}\frac{K_T}{K_Q}, \quad J = \frac{U_\infty}{nD}$$

According to the convention, thrust and torque are expressed as non-dimensional quantities which remain same under similar operating condition.

The propeller was analysed under a constant linear velocity of inflow (U_∞). Its rpm was varied to obtain different values of the advance ratio. Analysis was done for five advance ratios, ranging between 0.6 and 1.3. Pressure distribution over the propeller blade for two different conditions (J = 0.6 and 1.2) are plotted in Fig 6 and 7. Velocity vectors around the blade for these two conditions are shown in Fig. 8 and 9. The open water characteristics of the propeller are plotted in Fig 14.

Von-Mises stress over propeller blade for its two different operating conditions is shown at Fig 10 and 11. The deformed

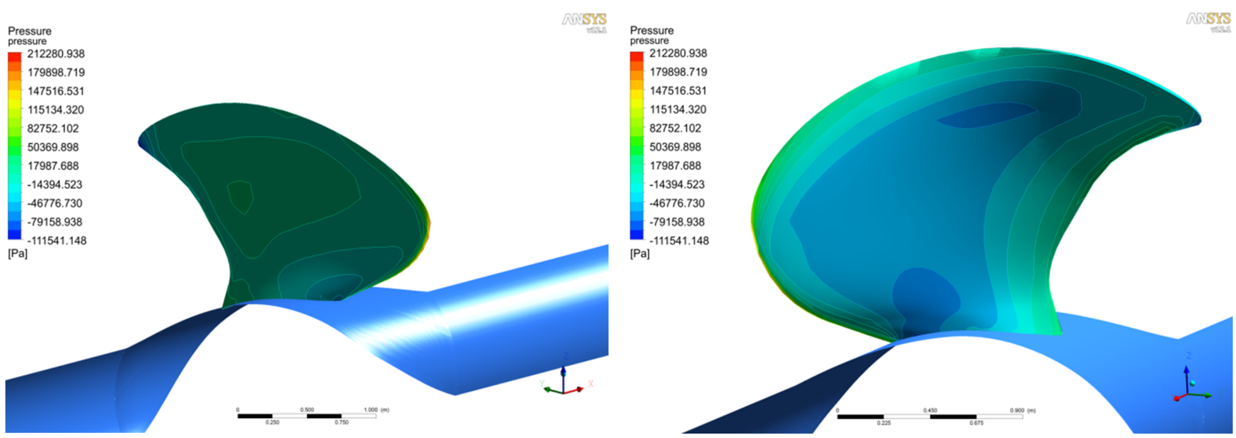
Figure 6: Pressure Distribution over Face & Back J=0.6



(a) Face (b) Back

Source: Authors

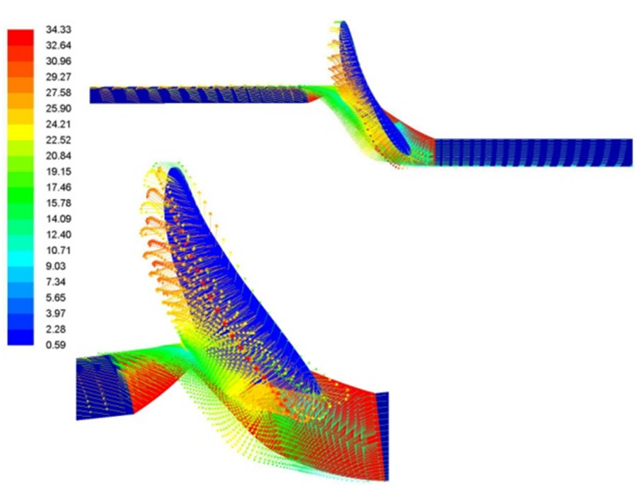
Figure 7: Pressure Distribution over Face & Back J=1.2



(a) Face (b) Back

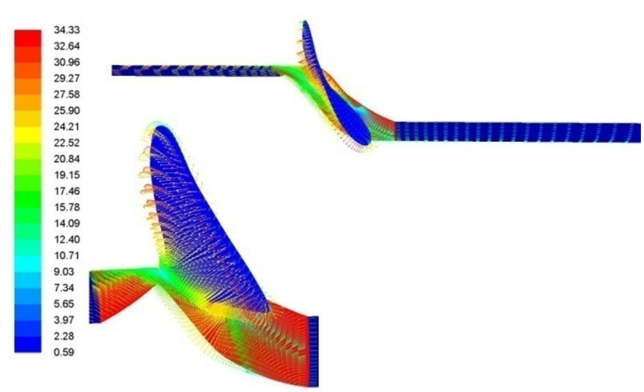
Source: Authors

Figure 8: Velocity vector, J = 0.6



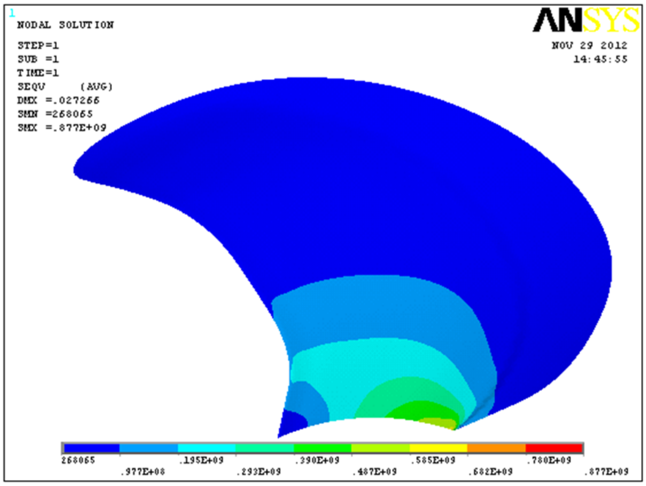
Source: Authors

Figure 9: Velocity vector, J = 1.2



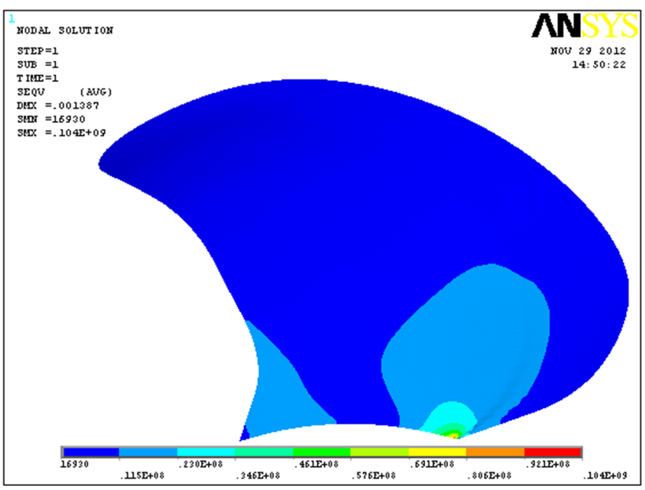
Source: Authors

Figure 10: Von Mises Stress(N/m2) over Propeller Blade, J=0.6



Source: Authors

Figure 11: Von Mises Stress (N/m2) over Propeller Blade J=1.2



Source: Authors

shape of the blade is shown in Fig.12 and 13. The maximum deformation is observed as 0.006428D. This deformation is corresponding to an Advance Ratio of 0.6.

The open water characteristics for original and deformed propeller geometry are shown in Fig 14. Experimental results were available for a scaled down propeller model (NSTL Internal Report 2010); so CFD results could be compared with observations from experiment (see table 1).

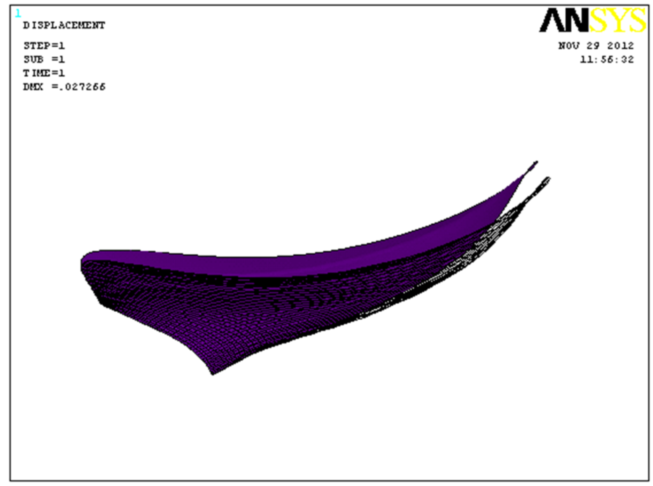
7. Conclusions

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. 14) giving good validation of this study.

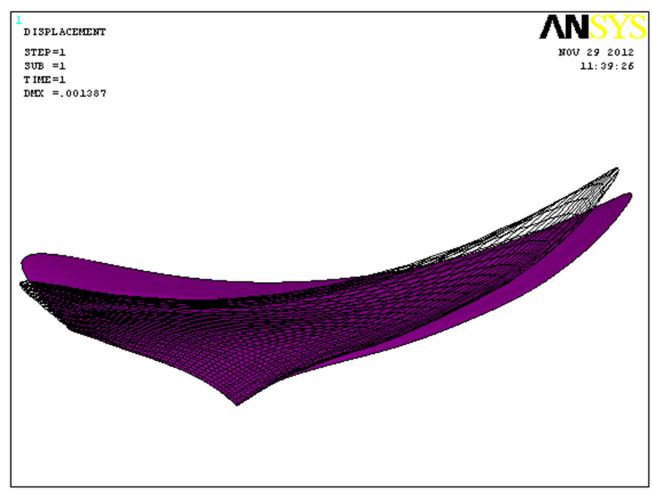
Deformation of this metallic propeller is found to be small and hence the hydrodynamic performance of propeller remains

Figure 12: Deformed Shape at J=0.6



Source: Authors

Figure 13: Deformed Shape at J=1.2



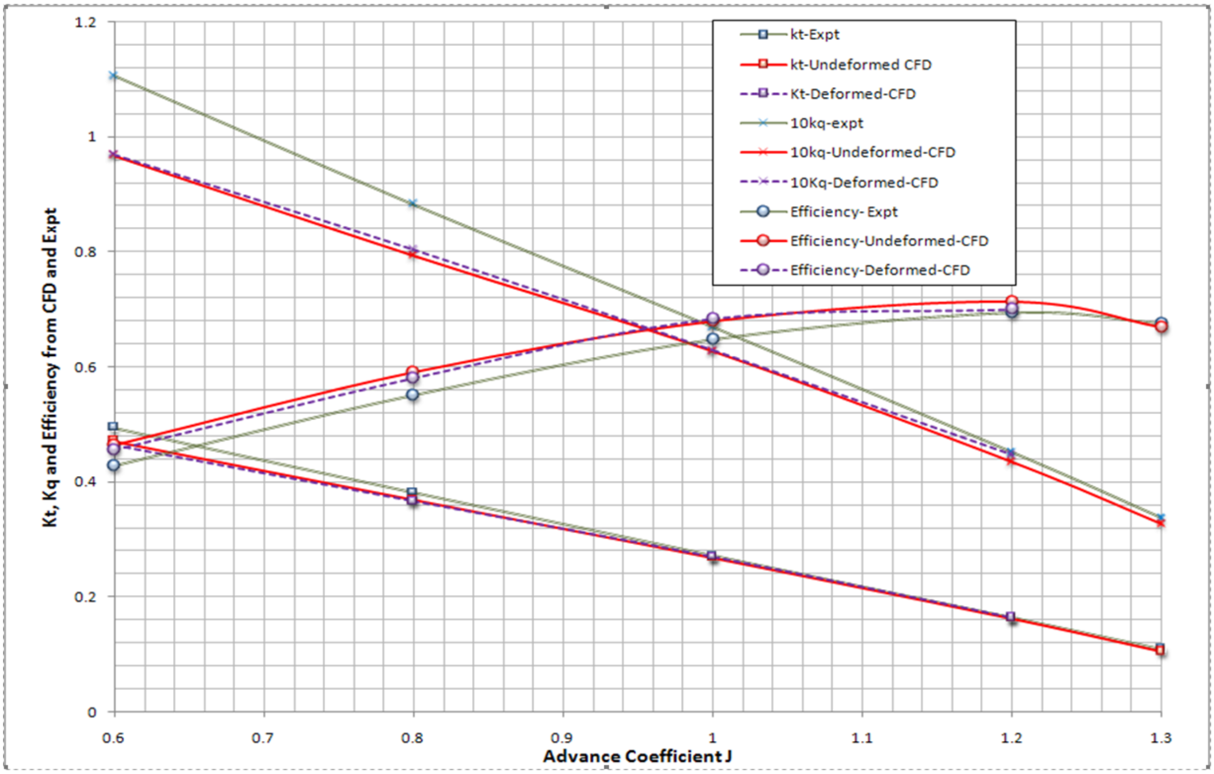
Source: Authors

Table 1: Open Water Characteristics for Propeller: Before and after Deformation

	Before Deformation		Deformed		% ge Difference	
J	kt	kq	kt	kq	kt	kq
0.6	0.471	0.097	0.463	0.097	1.71	-0.16
0.8	0.368	0.079	0.366	0.081	0.54	-1.24
1	0.268	0.063	0.269	0.063	-0.66	-0.19
1.2	0.162	0.043	0.164	0.044	-0.90	-2.86
1.3	0.105	0.033				

Source: Authors

Figure 14: Open Water Characteristics for Deformed & Undeformed Shape



Source: Authors

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.

8. 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.

References

ANSYS FLUENT®12.0 Documentation, 2011

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

Chung K. N., Fedric Stern and Min K. S., Steady Viscous Flow Field Around Propeller P4119, Propeller RANS/ Panel Method Workshop, 22nd ITTC Conference in Grenoble, France, 1998

Das H.N. and Jayakumar P., "Computational Prediction and Experimental Validation of the Characteristics of a Contra-Rotating Propeller", NRB seminar on Marine Hydrodynamics, Feb 2002

Das H. N. "CFD Analysis for Cavitation of a Marine Propeller" 8th Symposium on High Speed Marine Vehicles, HSMV 2008, Naples, Italy, 22-23 May 2008

Edward V. Lewis, Principles of Naval Architecture Volume II, Published by The Society of Naval Architects and Marine Engineers, Jersey City, NJ, 1988

Ghosh J. P. and Gokarn R. P., Basic Ship Propulsion, Allied Publishers Pvt Ltd., 2004

Krishna Kumar G.V., Saji V. F., Das H. N. and Panigrahi P. K. "Acoustic Characterization of a Benchmark Marine Propeller Using CFD" National Symposium on Acoustics (NSA-2008), NSTL, Visakhapatnam, 22 - 24 Dec 2008.

NSTL Internal Report on "Hydrodynamic Model Tests For New Design Frigate (Open Water, Self Propulsion & 3d Wake Survey Tests)"; Report Number NSTL/HR/HSTT/221/2 November 2010

Sanchez Caja A., P 4119 RANS Calculations at VTT, 22nd ITTC Conference in Grenoble, France, 1998

Seetharama Rao Y., Mallikarjuna Rao K. and Sridhar Reddy B., "Stress Analysis of Composite Propeller by Using Finite Element Analysis", International Journal of Engineering Science and Technology (IJEST), Vol. 4 No.08 August 2012

Sudhakar M., "Static & Dynamic Analysis of Propeller Blade" M Tech Thesis submitted to Andhra University, 2010.

Appendix. 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_∞ Free-stream Velocity

η Efficiency

μ Viscosity

ρ Density of Water