ISSN 2278 – 0211 (Online)

Structural Analysis of a Contra Rotating Propeller by using Finite Element Method (FEM)

Ch. Suryanarayana
Naval Science and Technological Laboratory, India
M. Nageswara Rao
Naval Science and Technological Laboratory, India
P. Nooka Raju
Naval Science and Technological Laboratory, India

Abstract:
In order to assess the static and dynamic response of an AFT propeller of Contra rotating propeller (CRP) for entire range of its operation, an analysis was carried out by FEM (Finite Element Method) and validated by analytical approach. From the Static analysis it is found that the stress and deflection are well within safe limits. The elastic deformation are so small,that they won't affect hydrodynamic load distribution.. The dynamic analyses of CRP indicate that it is safe in its entire range of operation as far as phenomenon of Resonance is concerned. Also it was observed that maximum harmonic response of the propeller on the application of dynamic loading is far lesser than its failure limit within specified operating range.

Keywords: *Contra-Rotating Propeller; Dynamic response; Finite element method, Harmonic Response, Shape functions, Stress, Deflection, Unsymmetrical bending, Taylor's method*

1. Introduction

The Propeller is a vital component which is essential for the safe operation of a ship at sea. It is important to ensure adequate strength of ship propellers to endure the forces that act upon them. On the other hand, providing excessive strength would result in heavier propellers withthicker blades than the required, leading to reduction in propeller efficiency. As a matter of fact, this necessitates a methodto calculate the forces acting on a propeller and the resulting stresses, so that the propeller has just the necessary strength for safe operation in service.

The forces that are exerted on a propeller blade arise from the thrust and torque of the propeller and the centrifugal force on each blade caused by its revolution around its axis. Due to implicit intricacy in shape of propeller blades, the accurate calculations of the stresses resulting from these forces is extremely difficult. In fact while it is quite possible to determine the thrust and torque of a propeller with reasonable accuracy for a ship moving ahead at a steady speed in calm water, but it is difficult to figure out the loading on a propeller when a ship oscillates violently in a seaway and the propeller emerges out of water and then plunges sharply into it sporadically. In this current paper, the steady hydrodynamic loading conditions are considered for the static and dynamic analysis of the aft propeller of CRP.

There is no direct exact solution method that is available for determining the stress of the propeller blades owing to the following reasons

i. The plane of application of loads that resulted from the thrust and torque are neither along the centriodal plane nor parallel to it
ii. It consists of a complex three dimensional aerofoil sections for which the precise location of the shear centre is a difficult process.

However there are certain approaches which give an idea about the strength of the propeller which are as follows

i. Taylor's method
ii. Theory of un symmetrical bending
iii. Finite element method (FEM)

1.1. Taylor's Method

A first approach to the strength problem was made by Taylor, who considered a propeller blade as a cantilever rigid fixed at the boss. The stresses are evaluated following theory of simple bending using sections of the blade by a cylinder, which have straight faces and curved backs. The greatest tensile strength was calculated to occur at the trail edge and the greatest compressive stress at the center of the back. This method being the simplest of all is still widely used for simple and conventional propeller geometries, with narrow blades

1.2. Limitations

a. This method is not well applicable when used for Propellers with wide blades and width comparable to length.
b. This method will result in relatively 30 % variation. 1.2.THEORY OF UNSYMMETRICAL BENDING

This theory is based on bending of a beam where the plane of loads acting on the beam is neither lies on the principle centroidal axes nor parallel to it. Neutral axis is determined by using principle axes of moment of inertia as a reference axis.. With this neutral axis, the loads along X and Y axis acting at shear center will be transformed on to the principle axis and bending stresses from these loads will be calculated based on beam theory.
In order to apply this theory the following assumptions are made Assumptions in theoretical calculations

a) Uniform cross section is assumed throughout the blade.
b) Stress due to Poisson's effect is not considered in the calculation.
c) The transverse shear stresses produced due to the loads applied are neglected.
d) Torsional shear stresses produced are neglected
e) The moment produced due to the clearance between CP and CG is not considered. Limitations

The result obtain from this method may not be exact owing to the above assumptions.

1.3. Finite Element Method (Fem)

Finite element analysis [1-2] is a numerical procedure for analyzing and solving wide range of complex engineering problems (may be structural, heat conduction, flow field…) which are complicated to be solved satisfactorily by any of the available classical analytical methods. In this method of analysis a complex region defining a continuum is discretized into simple geometric shapes called Finite elements. The material properties and the governing relationships are considered over these elements and expressed in terms of unknown values at element corners. As assembly process, duly considering the loading and constraints, results in a set of equations. Solutions of these equations give the approximate behavior of continuum.
The finite element method overcomes the difficulty of the variational methods because it provides a systematic procedure for the derivation of the approximation functions. The method is endowed with two basic features, which account for its superiority over other competing methods. First, a geometrically complex domain of the problem is represented as a collection of geometrically simple sub domains, called finite elements. Second, over each finite element the approximation functions are derived using the basic idea that any continuous function can be represented by a linear combination of algebraic polynomials. The approximation functions are derived using concepts from interpolation theory and are therefore called interpolated functions (Shape Functions). Thus the finite element method can be interpreted as a piece-wise application of the variational methods (e.g. Ritz and weighted – residual methods), in which the approximation of functions are algebraic polynomials and the undetermined parameters represent the values of the solution at a finite number of pre-selected points, called nodes, on the boundary and in the interior of the element. From interpolation theory one finds that the order (or degree) of the interpolation function depends on the number nodes in the elements.

1.4. Advantages

i. Both Static and Dynamic Analysis can be done with reasonable accuracy by proper selection of element type and size.
ii. Stress contours over the entire propeller section can be obtained.
iii. The Processing time is very less.
iv. Unsteady loading conditions can also be applied and the behavior of the blade can be determined.
v. Among all this is the only method best suitable for complex Hydrodynamic geometries like propeller.

It is due to these advantages and unique characteristics of FEM; it is selected /adopted for the structural analysis of the aft blade of CRP.

2. Literature Survey

A literature survey was conducted to appraise current status of research in the field of material selection using different methods, theoretical analysis of stresses, deflection and identify great areas requiring focused attention specifically relevant to the project topic.
The spectrum of papers collected could be broadly categorized into theoretical study on propellers and material selection, and FEM approach. Many investigators discussed in relation with the strength of the propeller blade.
The strength requirement of propellers dictate that not only should the blades be sufficiently robust to withstand long periods of arduous service without suffering failure or permanent distortion , but also that the elastic deflection under load should not alter the geometrical shape to such an extent as to modify the designed distribution of loading.
Taylor [3] a first approach to the strength problem was made by Taylor, who considered a propeller blade as acantilever rigid fixed at the boss. The stresses are evaluated following theory of simple bending using sections of the blade by a cylinder, which

have straightfaces and curved backs. The greatest tensile strength was calculated to occur at the trail edge and the greatest compressive stress at the center of the back. This method being the simplest of all is still widely used for simple and conventional propeller geometries, with narrow blades. But the method is suspect when used for propellers with wide blades and width comparable to length.

J.E.Conolly [4] combined theory with experimental work for wide blades. He has taken a three bladed twelve inch diameter propeller made of manganese bronze for the experiments. From the experiments he found that the stresses are greatest at the middle line and fall to low values at the edges. Model experiments using simulated loads indicated that elastic deflection of propeller blade should not cause significant change of geometrical shape and but might the responsible for cavitation at the leading edge of thin blades.

Terjesontvedt [5] studied the application of finite element methods for frequency response and improve to the frozen type of hydrodynamic loading. The thin shell element and the triangular type and the super parametric shell element are used in the finite element model it presents the realistic and dynamic stresses in marine propeller blades. Stresses and deformations calculated for ordinary geometry andhighly moved propellers are compared with experimental results.

Chang suplee [6] etal investigated the main sources of propeller blade failure and was resolve the problem system statically. An FEM analysis is carried out to determine the blade strength in boll and full condition and range of safety factor for the propeller under study is determined.

3. Theoretical Computation Of Bending Stress Experienced By The Crp Blade

As FEM is based on a numerical technique and the results are not exact and there is a need to validate these results with known analytical solutions. Although there is no exact solution is possible for this type of intricate hydrodynamic geometry with few assumptions the following theoretical analysis was done to compare the FEA results and for validation

By applying un-symmetrical bending theory, the resultant forces due to thrust and the torque acting on each sections of the blade are shown in Figure 1.

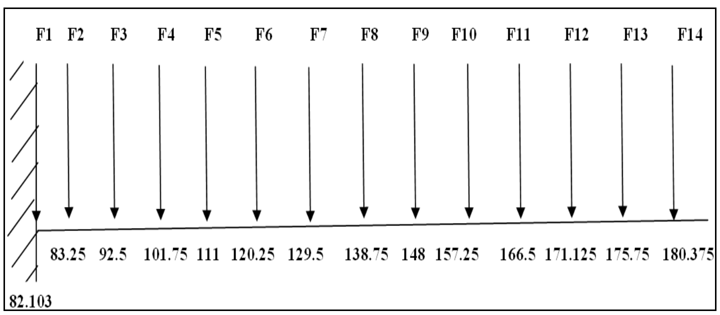

Figure 1

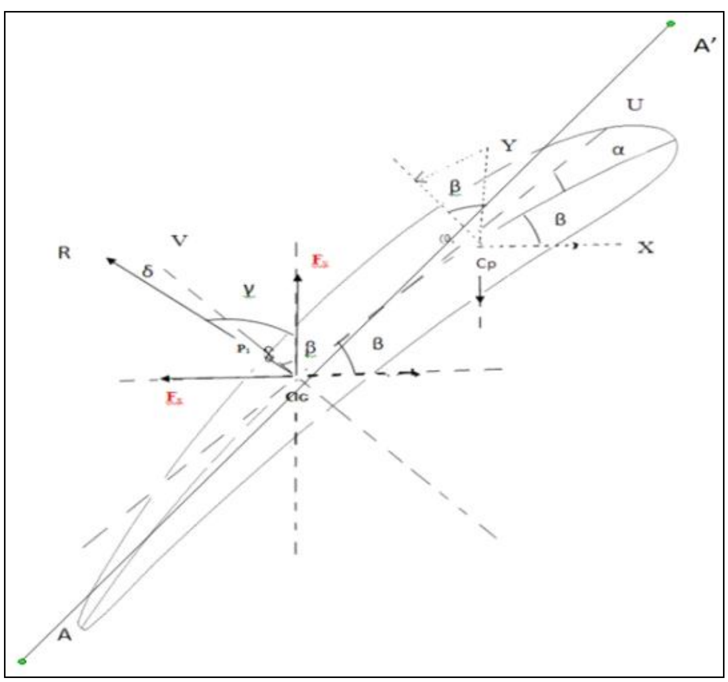

Figure 2

Sections	F_x (N)	F_y (N)	Resultant (F)	γ
82.103	2.989	1.981	3.585	33.534
83.25	38.051	25.578	45.848	33.909
92.5	55.387	41.368	69.130	36.755
101.75	67.766	55.675	87.703	39.405
111	75.707	67.854	101.664	41.868
120.25	79.622	77.309	110.979	44.155
129.5	80.246	83.91	116.104	46.278
138.75	77.364	86.674	116.179	48.248
148	71.38	85.301	111.226	50.077
157.25	60.592	76.934	97.929	51.776
166.5	24.288	32.653	40.695	53.357
171.125	19.159	26.473	32.678	54.106
175.75	12.691	18.01	22.032	54.829
180.375	4.535	6.606	8.012	55.530

Table 1: Load Data

ROOT SECTION DETAILS

P1 = (-6.76054,-1.71156, 0) at P1 , u = 0, v = 6.9 mm I_u = 7172.4 mm4

I_v = 134871.26 mm4 C.G = (-4.28, 4.92, 55.00) β = 43.5^0

The above section details are obtained from modeling software **UNIGRAPHICS** owing to the abstruse geometrical shape which makes the moment of inertia calculation more difficult and cumbersome.

Iu

Tan α =

tan β **α = 2.8^0**

Iv

Since the angle is very negligible we can consider principle axis as natural axis.

σz=

Muv +

Iu

Mvu

at P1

Iv

Assuming neutral axis is same for all applied loads (since the angle varies from 0 to 0.30) Now **β = 43.5^0**

For

F1,δ1 = (γ – β) = (39.535310 -38.20) = 1.335308^0 similarly

F2, δ2 = 7.840699^0

F3, δ3 = 8.502151^0

F4, δ4 = 8.039741^0

F5, δ5 = 7.275664^0

F6, δ6 = 5.861053^0

Similarly they are calculated for the rest of the sections. At point P1

σz=

Muv +

Iu

Mvu

=

Iv

Muv Iu

[Since u = 0 at P1]

Mu = M Cos δ = (FR1 X 2.5 Cos1.3353080 + (FR2 X 7.5 X Cos 7.840690) + (FR3 X 12.5 X Cos 8.502151) + (FR4 X 17.5 X Cos 8.0397410) + (FR5 X 22.5 Cos 7.2756640) + (FR6 X 30 X Cos(5.8610530) + (FR7 X 40 X Cos 3.67736 0) + 210.29 + 507.05+898.50+1310.85+1761.23+5002+7134 = 43622.5 N – mm

It is evident from the Figure 2 Maximum stress at point P1 where v = 6.9 and u = 0

$$\sigma_z = \frac{43622.5}{7172.4} \times 6.9 = 45.56\ \text{MPa}$$

Conclusion:
The bending stress experienced by the blade through theoretical calculation = 45.56 MPa

4. Modeling of Contrarotating Propeller Blade

The type of modeling that is required in order to model the contra rotating propeller is surface modeling. Surface modeling has been developing rapidly due to the shortcomings and inconveniences of wireframe modeling. To create a surface model, the user begins by constructing wireframe entities and then connecting themappropriately with the proper surface entities. In the current analysis the modeling platform selected for developing the geometrical model of contra rotating propeller blade is CATIA. The work area dedicated for carrying out the surface operations in CATIA is known as generative shape design.

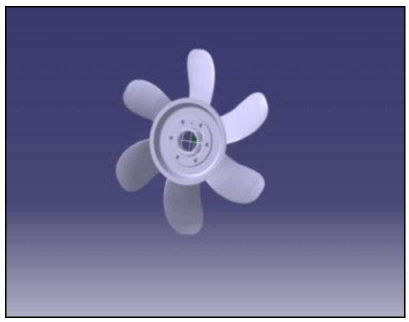

Figure 3: Final blade model with hub

Type	Screw propeller
Orientation	Left handed
Diameter	0.37 m
No of Blades	6
Thrust	4118 N
Torque	510 Nm
Ad/Ao	0.37

Table 2: Specifications of contra rotating propeller

5. Discrietization of the Contra Rotating Blade

For carrying out the task of meshing the CRP blade, HYPERMESH software is selected as a platform as it is highly robust, with higher versatility and competent of discretizing more complex geometries like propeller blades. Based on topology of meshing, it is categorized broadly into two types

i. Tetra meshing
ii. Hexa meshing

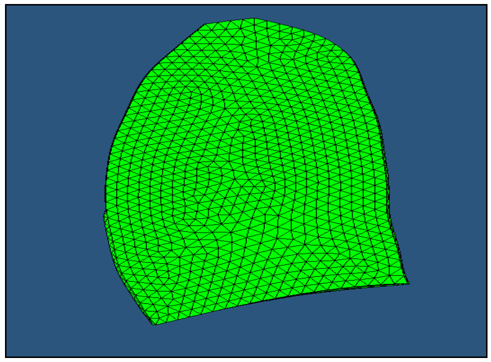

Figure 4: Tetra meshing of CRP blade

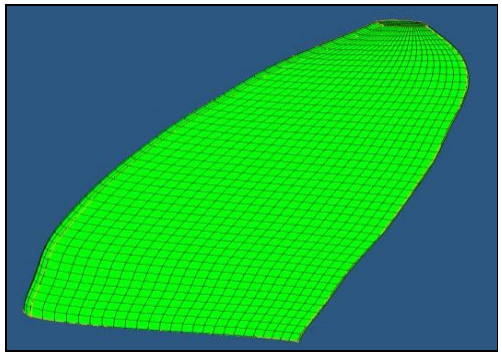

Figure 5: Hexa meshing of CRP blade

5.1. Elements Used in Theanalysis the Elements [Considered In The Analysis Are

1. SOLID 45
2. SOLID 95

5.1.1. SOLID 45

i. SOLID45 is used for the 3-D modeling of solid structures.
ii. It is also called eight noded brick element.
iii. The element is defined by eight nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions.
iv. The element has plasticity, creep, swelling, stress stiffening, large deflection, and large strain capabilities
v. It is a first order element

5.1.2. SOLID 95

i. SOLID95 is a higher order version of the 3-D 8-node solid element SOLID45.
ii. The element is defined by 20 nodes having three degrees of freedom per node: translations in the nodal x, y, and z directions. The element may have any spatial orientation.
iii. SOLID95 has plasticity, creep, stress stiffening, large deflection, and large strain capabilities
iv. Apart from the eight nodes of SOLID45, it also consists of twelve mid nodes, totally making twenty nodes.
v. It is a second order element**.**

Advantages over SOLID45

i. It shows linear strain behavior unlike solid 45 which exhibits constant strain behavior.
ii. It can tolerate irregular and intricate shapes likes **propeller blades** without as much loss of accuracy.
iii. SOLID95 elements have compatible displacement shapes and are well suited to model curved boundaries.

6. The Element Stiffness Matrix

Displacement approximation

$U(x,t) = \sum_b N_b(x) U_b(t) = N(x)U(t)$

Where N_b are element shape function, U_b (t) are time dependant nodal displacements and the sum ranges over the number of nodes associated with an element

$U(\xi,t) = \sum_b N_b(\xi) U_b(t) = N(\xi)U(t) \quad X(\xi) = \sum_b N_b(\xi) X_b = N(\xi)X$

Where x represents nodal coordinate parameters and are the parametric coordinates for each element. An approximation for the virtual displacements is given by δu (ξ) δU(ξ) = ∑Na (ξ)au=N(ξ) u

U (ξ, t) U(ξ, t) = ∑b Nb (ξ)Ub(t) = N (ξ)U(t) X (ξ) = ∑b Nb (ξ)Xb= N (ξ)X

Derivative

δNa = δxj δNa

δξ

δξ δxj

δNa δξ

δNa

= J

δ x

Where

δNa =

δξ

δNa δξ δNa δξ δNa

δξ

δNa

; =

δx

δNa δξ δNa δξ δNa

δξ

; J=

δx δx δx

δξ δξ δξ

δx δx δx δξ δξ δξ

Iδx δx δx I

δξ δξ δξ

In which J is the jacobian transformation between X and ξ. Using the above results the function derivatives are given by

δNa = J-1 δNa

δx δξ

Strain – displacement equations

ξ= δu ∑b(δNb)Ub= ∑b Bb Ub= BU

In a general three dimensional problem the strain matrix at each node of defined by

b

B^T =

Nb1x1 0 0

0 Nb1x2 0

0 0 Nb1x3

Nb1x2 0 Nb1x3

Nb1x1 Nb1x3 0

0 Nb1x2 Nb1x2

The element stiffness matrix is

1 T 1 T

K= I B EBdv =I B EBt |f|dξ

-1 -1

7. Mesh Convergence

One of the most overlooked issues that affect accuracy, namely; mesh convergence. This refers to the smallness of the elements required in a model to ensure that the results of an analysis are not affected by changing the size of the mesh

7.1 Steps Involved in Mesh Convergence Test

i. The node list with coordinate is checked for the required location to the nearest proximity.
ii. Followed by determining the equivalent stress at that node(required location)
iii. This process is repeated for next much finer element sizes
iv. At a particular element size, there stress value will get converged.

In the current work mesh convergence tests are carried out for both tetra and hexa meshed model of CRP blade considering Solid 45and Solid 95 as the element types

The whole idea about the iterations with variety of element sizes and element types in the above test is

1. To achieve the assurance of the accuracy of the results from various levels of solutions or in other words to internally validate in terms of different types of meshing.
2. Followed by the **optimization** of the mesh quality in terms of accuracy, solution processing time and the memory allocation.

8. Optimization of FEM Model

As mentioned in the previous chapter, the optimization of the FEM Model is conducted based on three judging criteria which are

i.Processing time
ii.Memory allocation
iii.Accuracy

It is evident from the table showing various iterations that the element **SOLID95 with element size three of hexa mesh type (3hexa 95)**, is the optimized value as it predominately satisfies the deciding factors when compared with others. It also depicts that that stress value or the % of accuracy obtained from the optimized mesh **3hexa 95** is in concurrence with accuracy levels of tetra mesh iterations, as of which it is internally validated. So it is selected as the quality mesh to a maximum degree possible for carrying out the analysis of CRP blade.

Element type	**Element size**	**Number of nodes**	**Number of elements**	**Processing time (in seconds)**	**Memory allocation (in MB)**	**Accuracy (%)**
TETRAMESH						
SOLI D45	0.7	74312	33916	58	955	**106**
	0.6	112206	52128	137	1400	**105**
	0.5	163079	76258	229	1988	**104**
SOLI D 95	4	6051	3087	5	8	**114**
	3	12458	6760	9	17.5	**109.8**
	2	39364	23395	93	59.62	**108.5**
HEXAMESH						
SOLI D 45	4	1606	1104	3	2	**106**
	3	3960	2940	4	6.312	**95.8**
	2	11648	9486	10	19.812	**89.10**
SOLI D 95	4	5956	1104	2	4.5	**78.00**
	3	**14758**	**2940**	**5**	**11**	**104**
	2	44340	9486	29	37	**100.4**

Table 3: Optimization

9. Analysis

9.1. Static Analysis

Static analysis is concerned with the behavior of elastic continuum under prescribed boundary conditions and statically applied loads.. Under water vehicle with CRP blade is chosen for Finite Element analysis. The deformations and stresses are calculated for isotropic material. The single blade without hub is taken for blade analysis 3D solid element SOLID 95 with element size three (3) with hexa mesh which is the optimized FEM model for isotropic material has been considered The boundary conditions Ux =0, Uy=0, Uz=0 are applied i.e., deformation in all X,Y&Z directions are fixed at the root end of the blade. .The tangential forces produced due to the torque in positive X direction and thrust in positive Y direction are applied at different nodes which the CP (Centre of pressure). The CRP blade was considered as acantilever beam fixed at one end and free at other end. With the given hydrodynamic load data static stresses and the deflection of the blade are determined with satisfying equilibrium equations, compatibility state of deformation and boundary conditions on stress simultaneously. Both material and geometrical linearity is considered in the analysis. Linear stress analysis is performed assuming the deformations are very small.

9.1.1 Static Analysis Results

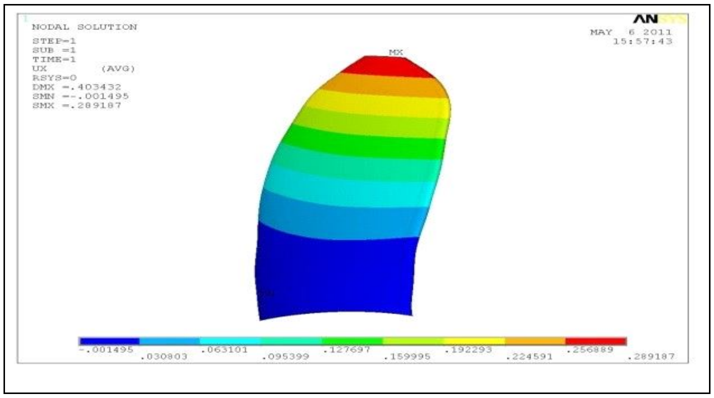

Figure 6: Maximum Deflection (X direction): 0.289 mm

9.1.1.1deflections

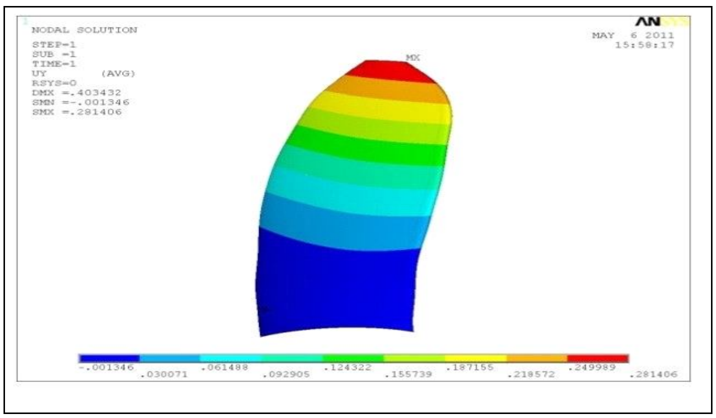

Figure 7: Maximum Deflection (Y direction) : 0.201 mm

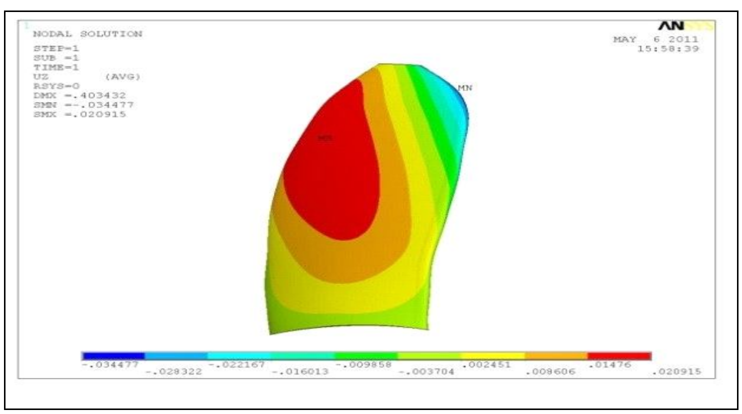

Figure 8: Maximum Deflection (Z direction) : 0.03438 mm

USUM	U X	UY	UZ
0.4034	0. 2874	0. 2795	0. 035

Table 4: Deflections

9.1.1.2 Stress

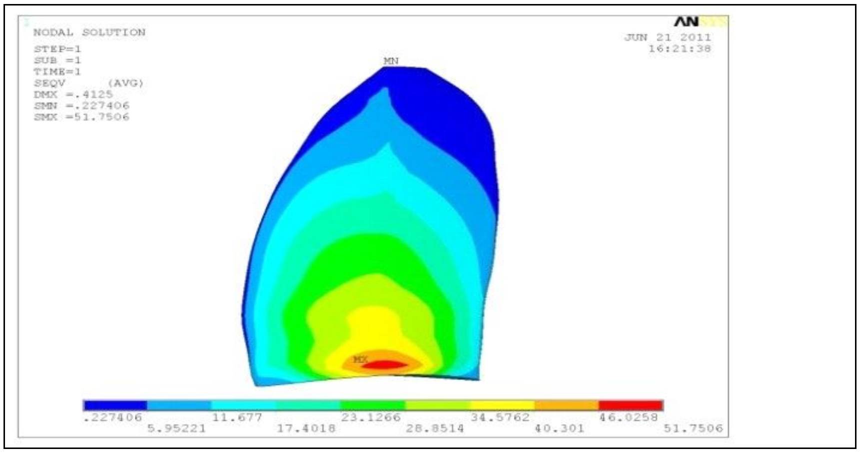

Figure 9: Stress

9.1.1.2.1 Maximum Bending Stress

The Distortion Energy theory (Von- Misses) is considered as an yield criteria the reason being propeller material is aluminum which is ductile in nature in the entire range of operation of the propeller as well as it depicts complete state of stress for the CRP blade. The maximum bending stress (Equivalent) that the propeller blade is subjected is at the root section of 51.75MPa

Unsymmetrical bending method is used for the theoretical analysis The root section geometry details like the principle moment of inertia I_{XX} and I_{YY} have been taken from modeling software UNIGRAPHICS. The bending stress experienced by the blade through theoretical calculation= 45.56 MPa

Stress Value		**% of Agreement**
Theoretical	By ANSYS	112
45 M Pa	51.75M Pa	

Table 5

Hence the ANSYS results are validated theoretically and found that they are accurate to a level of 112%.

The very reason for having a margin of 12% error with the theory and ANSYS results is due to the series of assumptions that are followed in the theoretical analysis owing to the high degree complexity of the geometry of the propeller blade for which exact analytical solution is very tedious and cumbersome process.

Assumptions in Theoretical Calculations

i. The root cross section is assumed all through the propeller blade (uniform cross section).
ii. Poisson's effect is not considered in the calculation.
iii. The shear stresses produced due to the loads applied are neglected.
iv. The moment produced due to the clearance between CP and CG is not considered.
v. The propeller is assumed to be subjected to simple bending.

The maximum stress that the propeller blade is subjected is at the root section of 51.75MPa.

9.2. Modal Analysis

Modal analysis is performed to determine the vibration characteristics i.e. The Natural Frequency and the mode shapes of the propeller. Block Lancoz mode extraction method is used for extracting the modes. The modal analysis of CRP blade is a large symmetric eigen value problem for which Block Lancoz method achieves higher convergence rate than the subspace method and it also uses sparse matrix solver.

The natural frequencies of the Contra rotating propeller obtained from modal analysis are 1. 1188.5 Hz (Figure16)

2. 3763.5 Hz(Figure 17)
3. 4428.9 Hz(Figure 18)
4. 5276.2 Hz(Figure 19)
5. 8381.0 Hz(Figure 20)

So, the fundamental frequency of propeller is 1188.5 Hz. The first five mode shapes are shown in fig. 10 to fig. 14

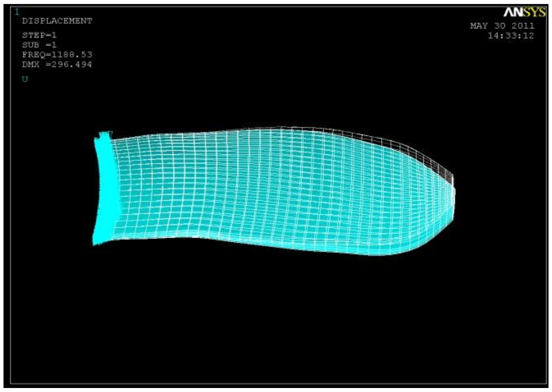

Figure 10: First mode

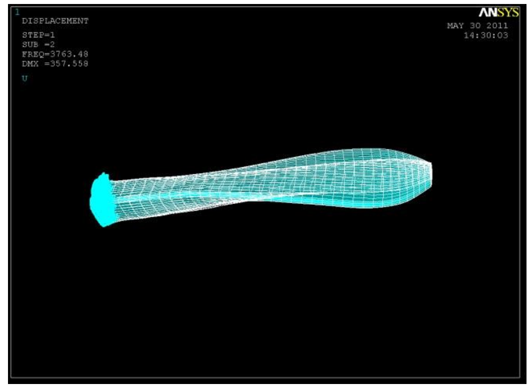

Figure 11

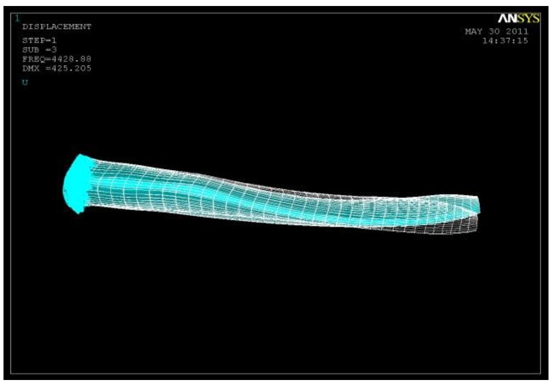

Figure 12: Third mode

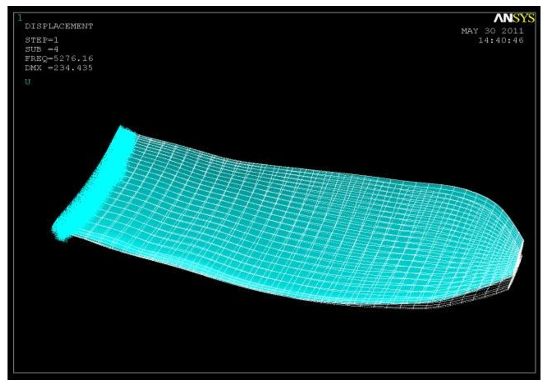

Figure 13: Fourth mode

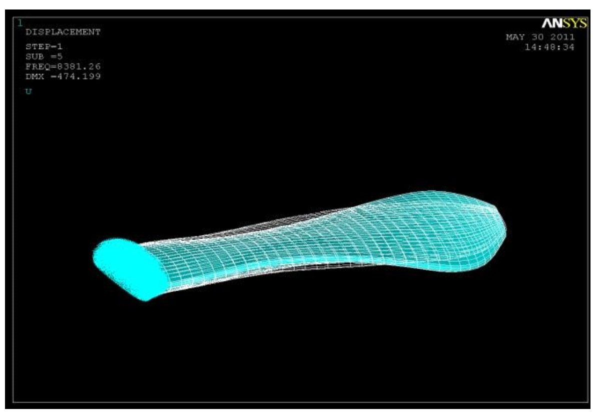

Figure 14: Fifth mode

9.2.1. Analytical Calculation of First Natural Frequency

$$f_1 = \frac{1.875^2}{2\pi}\sqrt{\frac{EI}{ml^4}}$$

Where **m** is mass per unit length
f_1= 1128.5 Hz
So the fundamental frequency of the CRP =1128.5 Hz
The fundamental frequency of propeller from ANSYS is 1188.5 Hz.

9.2.3. Validation of First Natural Frequency

So the modal analysis is theoretically validated and found that the fundamental frequency obtained from ANSYS is in concurrence with 105.32 % accuracy with the frequency reckoned theoretically.

9.3. Harmonic Analysis

A harmonic response analysis is performed to determine solution of time dependent equivalent of motion associated with CRP undergoing steady statevibration. All loads and displacements are assumed to varying sinesodially at the maximum frequency of contra rotating propeller. The full method is used for conducting harmonic analysis. It achieves higher convergence rate than the subspace method and it also uses sparse matrix solver. A constant damping ratio of 0.0405 is used in the analysis which is an experimentally established value for the aluminum alloy used for underwater applications.

In Harmonic analysis it is found that resonance occurs predominantly only at first natural frequency at 1188.5 Hz. The blade pass frequency of the propeller = 9240 Hz.

Angular velocity ω = 172.97 Hz

9.3.1. Harmonic Response Analysis Results the Response from the Harmonic Analysis Is Shown In Fig.

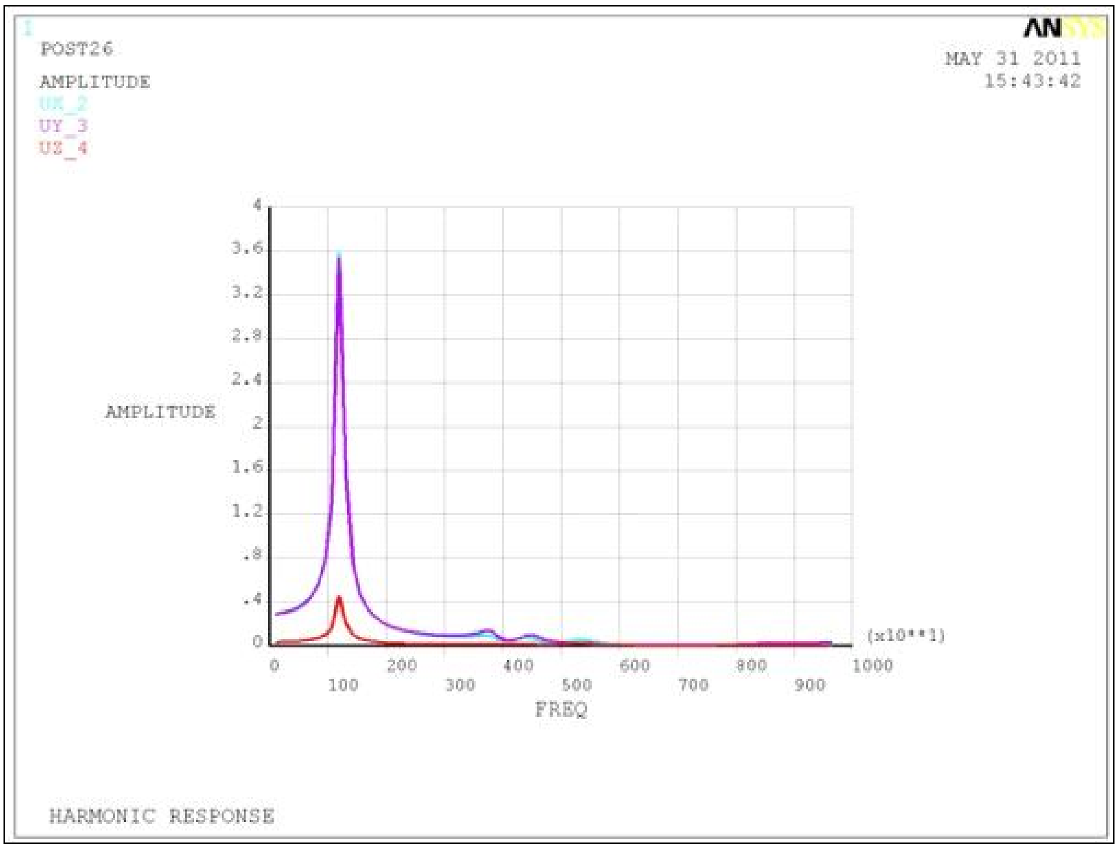

Figure 15: Frequency Response Function

Maximum deflection in X-direction: 3.6mm (at 1188.5Hz) Maximum deflection in Y-direction: 3.6mm (at 1188.5Hz) Maximum deflection in Z-direction: 0.45 mm (at 1188.5Hz)

9.3.2. Theoretical Results

At the condition of resonance $\omega = \omega_n$

So Magnification factor = $\frac{1}{2\zeta}$

$$\frac{1}{2\,(0.0405)} = 12.34$$

$$U_{X\ \text{at resonance}} = 3.56\ \text{mm}$$
$$U_{y\ \text{at resonance}} = 3.46\text{mm}$$
$$U_{z\ \text{at resonance}} = 0.43\ \text{mm}$$

So the harmonic response analysis results are theoretically validated and found that they are concurrent with each other.
Conclusion: So the harmonic response analysis results are theoretically validated and found that they are concurrent with each other.

Deflect ion	Static	At resonance by ANSYS	At resonance by theory	% of Agree ment
UX	0. 2874	3.6	3.56	101.1
Uy	0. 2795	3.5	3.46	101.1
Uz	0. 02096	0.45	0.43	104.2

10. Conclusions

It is found from the static analysis that the propeller blade is within safer limits and having a factor of safety of **7.5.**In dynamic analysis it is found that resonance occurs at first natural frequency in which propeller operates for a very less duration of time where the vibration propagation is very minimal and is far beyond blade pass frequency which indicates the design is safe with respect to dynamic response.Hence the Contra rotating propeller design is safe with respect to the hydrodynamic forces from both static as well as dynamic point of view.

11. Nomenclature

- A- A'-Neutral axis U-- Principal axis
- Cp -Centre of pressure
- CG -Centre of gravity
- X-Y -Coordinate axis
- P1 -Point
- Fx -Force applied in Xdirection Fy-Force applied in Y direction R-Resultant Force
- α -Angle between the Neutral axis and the Principal axis
- β -Angle made by the Neutral axis with the coordinate axis
- γ -Angle made by the Resultant with the coordinate axis
- δ -Angle made by the Resultant with thePrinciple axis
- N -Shape Functions
- J -Jacobian
- f -Frequency
- ω -Angular Velocity
- Iu - Principle Moment of Inertia in X direction Iy- Principle Moment of Inertia in Y direction σz- Bending stress
- K -Global stiffness matrix
- Q -Vector of nodal displacements
- F -Vector of nodal forces for complete structure.

11. References

i. Zincowich and Taylor, "Finite element method for solid and mechanics"
ii. Robert Cook, " TheBasis,Finite Element Analysis".
iii. Taylor,D.W, "The speed and power of ships", Washington,1933
iv. J.E. Conolly, "Strength of Propellers" Read in London at a Meeting of the Royal Institution of Naval architects on Dec.1 1960, pp 139-160.
v. TerjeSontvedt, "Propeller Blade Stress, Application of Finite Element Methods" Computers and Structures, Vol. 4, pp.193-204.
vi. Chang-Sup Lee, Yong-Jik Kim, Gun-Do Kim and In-SikNho. "Case Study on the Structural Failure of Marine Propeller Blades".
vii. ANSYS software Help Document.