

PLEASURE VESSEL VIBRATION AND NOISE FINITE ELEMENT ANALYSIS

¹ Macchiavello, Sergio ^{*}, ²Tonelli, Angelo

¹ D'Appolonia S.p.A., Italy, ²Rina Services S.p.A., Italy

KEYWORDS –

pleasure vessel, vibration analysis, resonance, structural components

ABSTRACT –

Resonances within pleasure vessels structures are cause of noise and vibrations conflicting with the high standard of acoustic comfort requested by this kind of ships.

D'Appolonia S.p.A. and Rina Services S.p.A. were involved in carrying out noise and vibration numerical analyses on a pleasure vessel design, aiming at identifying the natural frequencies and mode shapes of its structural components, to ensure that external excitation sources do not lead to resonances.

The analyzed pleasure vessel has a total length of 54.6 m and a maximum beam length of 10.2 m; it is composed by main, upper and sun decks. All structural components are made of aluminium.

The creation of the discretized model, starting from the 3D CAD model of the ship, was entirely developed using ANSA pre-processing software. In particular, each ship “block” was handled according to the following approach:

- repair and clean up of the model;
- extraction of middle surfaces and creations of connections between them;
- creation of shell and, where allowed, beam elements for meshing the structural parts;
- creation of .nas file, to be imported by Nastran solver.

Due to symmetry considerations, only the left half of the ship was modelled. The element length adopted was 200 mm for hull structures and 300 mm for super-structures.

The finite element (FE) model was produced in several .nas files, which were subsequently merged within the solver environment.

The analysis allowed identifying the natural frequencies of the structural components of the pleasure vessel: a comparison was made among them and the frequencies of the external excitation sources, in particular propeller shaft and propeller blades rotation frequencies. This comparison allowed identifying critical components and suggesting necessary modifications to the pleasure vessel design.

TECHNICAL PAPER -

1. INTRODUCTION

The objective of this study was to analyze the dynamic behaviour and to identify the overall vibration characteristics of the ship, so that components sensitive to induced vibration forces may be identified and assessed relative to acceptance criteria for vibration levels requested by technical specification.

In details, the abovementioned objective has been obtained through calculations of:

- natural mode of structural vibration,
- vibration levels distribution,

by means of a detailed finite element analysis of the overall ship. Both free and forced vibration analyses were performed.

The analyzed pleasure vessel has a total length of 54.6 m and a maximum beam length of 10.2 m; it is composed by main, upper and sun decks. All structural components are made of aluminium. Table 1 summarizes the most important characteristics concerning the vessel and the propulsion system.

Table 1: Ship and Propulsion System main parameters

Main Dimensions		
Length between perpendicular	50.5	m
Breadth	10.2	m
Depth to main deck	4,6	m
Draught, max	2,5	m
Draught, design	2.3	m
Main Propulsion Data		
Service speed	17.	kn
Engine power	2 x 1029	kW
Engine rotation velocity (MCR)	2100	rpm
Propeller diameter	1.7	m
Number of propeller blades	5	
Propeller rotation velocity (MCR)	475	rpm
Propeller rotation velocity (CSR)	400	rpm

The free vibration characteristics and the dynamic response of the mathematical model to the propeller and main excitation force were calculated by means of the finite element code MSC/NASTRAN release 2013.1.1.

2. METHODOLOGIES

Free Vibration Analysis

The first step in performing a vibration analysis is determining the natural frequencies and mode shapes of the structure with damping neglected. These results characterize the basic dynamic behaviour of the structure and they are an indication of how the structure will respond to dynamic loading. The natural frequencies of a structure are the frequencies at which the structure naturally tends to vibrate if it is subjected to a disturbance. The normal mode of vibration is the deformed shape of the structure at a specific natural frequency of vibration. Each mode shape is associated with a specific natural frequency. Natural frequencies and mode shapes are functions of the structural properties and boundary conditions. To obtain natural frequencies and mode shapes it necessary to solve an eigenvalue problem as described in equation below:

$$[K](\phi) = \omega^2 [M](\phi)$$

where

- [K] = symmetrical square stiffness matrix
- [M] = mass matrix
- (ϕ) = column mode shape matrix
- ω = circular natural frequency
- f = $\omega/(2\pi)$ natural frequency.

Forced Vibration Analysis

Forced vibration analysis is used to compute structural response to steady-state base oscillatory excitation. Since the steady-state oscillatory response occurs at the same frequency as the loading, it is possible to analyze the response in the frequency domain instead of time domain. The response may be shifted in time due to damping in the system. In frequency domain the shift in response is called a phase shift because the peak loading and peak response no longer occur at the same time. The important results obtained from the frequency response analysis usually include the displacements, velocities, and accelerations of grid points. The computed responses are complex numbers defined as

magnitude and phase (with respect to the applied force) or as real and imaginary components, which are vector components of the response in the real/imaginary plane. Applying to the model the resulting of the alternating propeller forces at different frequencies and solving the resulting dynamic problem represented by the following equations of motions obtain a frequency response:

$$[M](d^2u/dt^2)+[C](du/dt)+[K]u(t)=(F(t))$$

where:

[C]	= damping matrix
(d ² u/dt ²)	= column matrix of accelerations
(du/dt)	= column matrix of velocities
(u)	= column matrix of displacements
(F)	= column matrix of harmonic forces

A finite element model of the whole vessel was created to solve both the free and forced vibration equations.

Geometrical Model

The process of creation of the three-dimensional finite element model, starting from complete three-dimensional CAD model of the ship, was entirely developed taking advantage of ANSA pre – processor.

Some necessary modifications to the geometrical model were performed before creating the finite element model, in detail:

- removal of all the components having no structural function;
- removal of all the geometric details which don't have influence on the calculation;
- middle surfaces extraction of components that have been further modelled with shell elements;
- removal of components that have been further modelled with beam elements.

Three Dimensional Finite Element Model

A three-dimensional finite element model consisting of every structural component of the ship was created for carrying out the vibration analysis.

The structural components have been modelled using beam and shell elements. In detail, decks, bulkheads and shell plating were modelled using constant thickness membrane-bending shell elements NASTRAN's CQUAD4 and CTRIA3, respectively quadrilateral (4 nodes) and triangular (3 node) elements.

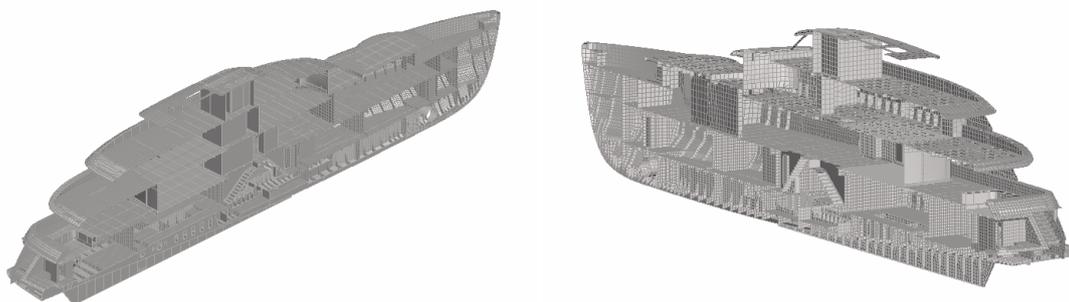


Figure 1 – Pleasure Vessel Finite Element Model: half shell model

Discrete girders, transverses, web frames, stiffeners and pillars were modelled with two-node beam elements, having axial, bi-directional shear, torsional and bi-directional bending stiffness and different constant properties according to the structure type. Concentrated mass elements for equipments with weight greater than 500kg and rigid links to connect

these equipments to the structures were used. The virtual mass of the water associated to the outside wetted shell has been calculated by the Nastran features using the formulation based upon the potential flow in an infinite volume of an incompressible fluid. So the virtual mass matrix for an incompressible fluid volume is treated in the same way as the mass matrix.

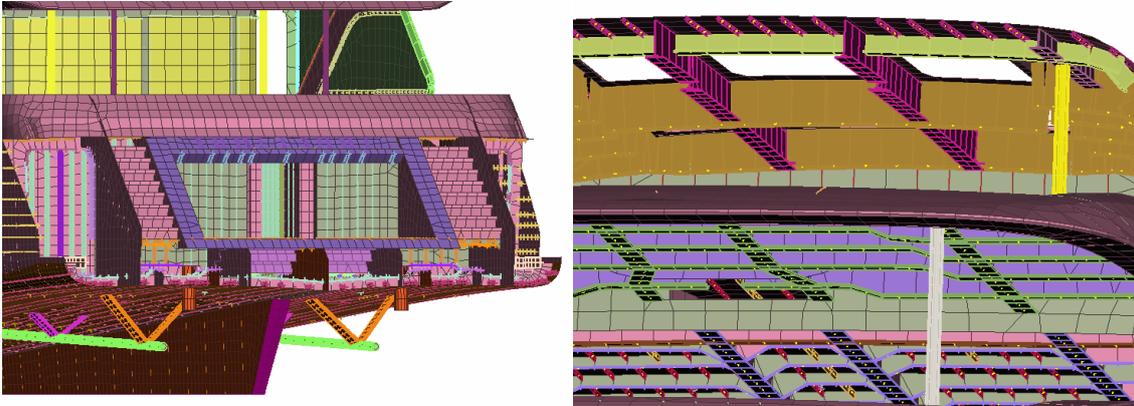


Figure 2 – Details of finite element model

The complete 3D model takes also into account lateral and torsional vibrations, even if the dominant excitation forces are approximately symmetric with respect to the centreline plane of the ship, lateral or torsional vibrations are not expected to be significant in this type of ship. The damping associated with ship hull vibration is generally considered as a combination of the following:

- structural damping,
- water friction
- pressure waves generation
- surface waves generation.

In the frequency response the structural damping with respect to the frequency values is expressed as a fraction of the critical one. Damping values are linearly interpolated or extrapolated directly from the values represented in Table 2.

Table 2: Damping values for critical frequencies

Frequency (Hz)	Damping ζ
0.1	0.05
100	0.10

Computational Grid

The finite element model represents the entire vessel, including hull and superstructure. Hull and related structures were modelled with elements having an average element length of 200 mm, while superstructures with 300 mm ones.

The fine-mesh model has been used to take into account the excitations up to higher frequency (almost 120 Hz). This was done so that the vibration response in the areas of concern could be determined with a better accuracy, while still providing a good representation of the stiffness characteristics of the entire vessel. The characteristics of the model are summarized below:

- total number of grid points: 119483
- total number of cbeam elements: 52174
- total number of cquad4 elements: 119787
- total number of ctria3 elements: 10400
- total number of rbe2 elements: 19

Boundary Conditions

No constraints were applied to the finite element model, so also the rigid motions have been calculated.

The boundary conditions considered for the vibration analysis are representative of the vessel advancing at constant speed forward in head sea. A representative load condition that envelops all loads distribution and close to the effective sea trial weight distribution has been selected.

The forced vibration analysis was performed as a frequency response analysis (amplitude versus frequency). The ship was excited, over the operating range, by:

- forces arising from misalignment of the propellers,
- propeller induced pressure pulses,
- engine crankshaft induced forces,
- bearing forces induced by propeller.

To obtain the induced vibration forces at different speeds (in the range 260 - 475 rpm), it is assumed that forces acting on the structure are scaled in function of the propeller rotation velocity. The misalignment of the propellers force has been applied on the shaft line in correspondence of the propeller centre. The integral of pressure pulses on the stern part are taken into account by a suitable equivalent pressure distribution applied on the selection of the wetted elements within a theoretical influence sphere based on the propeller size. The engine crankshaft induced forces have been applied to grid point representing the centre of gravity of the engine. The bearing forces have been applied in the centre of the propeller. The effect of the water was considered through the virtual mass applied to the wetted elements of the hull in sea trial draught.

Other excitation forces have been considered not relevant, so they are not presented in this report.

3. RESULTS

Free Vibration (Modal Analysis)

The mode shapes obtainable from the model developed are:

- three dimensional rigid body motion (first six natural modes with frequencies close to 0),
- longitudinal global ship hull vibration in correspondence of the reinforced frames (frequency range 2 -12 Hz),
- vibration of decks coupled with pillars, bulkheads substructures and transversal girder coupled with longitudinal girders (frequency range 5 - 15 Hz),
- vibrations of the stiffened panels between the transversal and longitudinal girders (frequency range 7 - 50 Hz).

For higher frequencies, several local natural modes are obtained; all the areas with significant vibration levels are discretized with adequate fine elements to compute the natural local girder modes. Vibration modes of ship structure have been calculated up to 120 Hz. The most characteristic global modes are summarized in Table 3.

Table 3: Natural frequencies at half load condition

Mode Shape	Natural Frequency [Hz]
First global vertical mode – 2 nodes bending	2.91
First global horizontal mode – 2 nodes bending	5.27
First torsional mode – 1 node	5.42
Second global vertical – 3 nodes bending	6.97

Patex 2013/64/815-Oct14 11:58:25
Defor: modale_1c08; All Mode 11; Freq = 3.895; Eigenvektor: Translational; (NON-LAYERED)

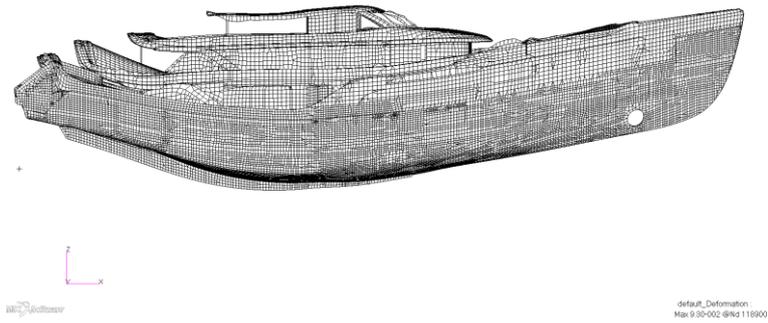


Figure 3 – First global vertical mode – 2 nodes bending at 2.91Hz

Patex 2013/64/815-Oct14 11:47:32
Defor: modale_1c08; All Mode 23; Freq = 5.428; Eigenvektor: Translational

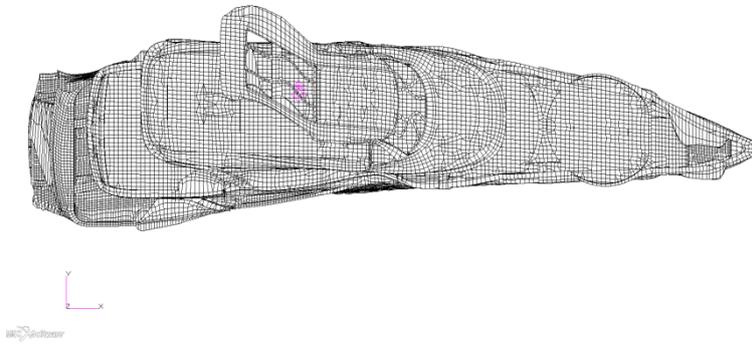


Figure 4 – First global horizontal mode – 2 nodes bending at 5.27Hz

Patex 2013/64/815-Oct14 14:08:52
Defor: modale_1c08; All Mode 26; Freq = 5.548; Eigenvektor: Translational; (NON-LAYERED)

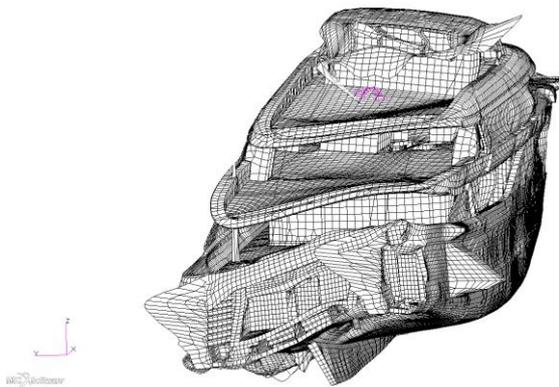


Figure 5 – First torsional mode – 1 node at 5.42Hz

Patent: 2013/04/015-001/141125/03
Definim: modale_scad01_A11 Mode-40; Freq = 7.9558; Eigenvektor; Translati0nal; (NON-LAYERED)
001

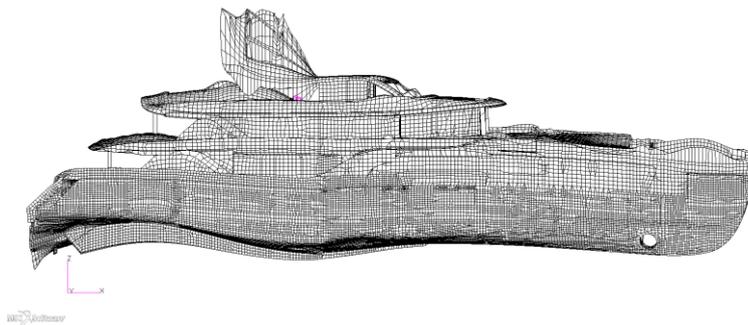


Figure 6 – Second global vertical mode – 3 nodes bending at 6.97Hz

Coupled global and local stiffened panel modes are in the frequency range comprised between 4 and 40 Hz.

Patent: 2013/04/015-001/141129/16
Definim: modale_scad01_A11 Mode-40; Freq = 7.8186; Eigenvektor; Translati0nal; (NON-LAYERED)

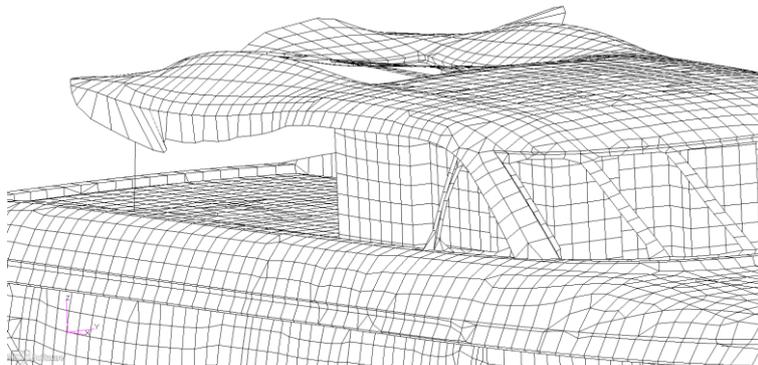


Figure 7 – Local modes of hard top – longitudinal bending at 7.84 Hz

Forced Vibration

To investigate the dynamic behaviour of the structure with different operating conditions and evaluate the resonance of the local structure with the excitation forces, graphs of the structural vibration velocity versus frequency are the most useful. The following pictures show the graphs of the resulting single components of vibration velocity amplitude for the range from the minimum propulsive power to 110% (120 to 500 rpm) for the excitation forces developed by the propellers. In general, as shown by the force vibration response results, the vibration velocity has peak values in points characterized by coincidence of the natural modes of the structure with the excitation forces.

In the following pictures, the curves show the global velocity at various frequencies for each node selected.

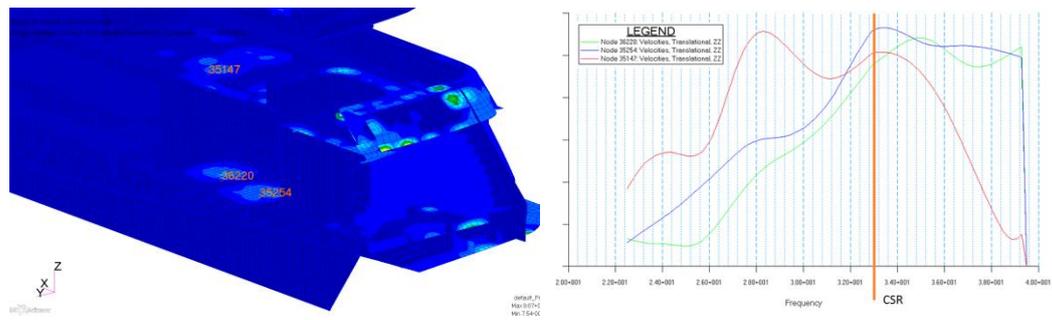


Figure 8 – Propeller pulse load condition: vertical component of vibration velocity distribution and vertical component of vibration velocity versus frequency graph for the selected nodes.

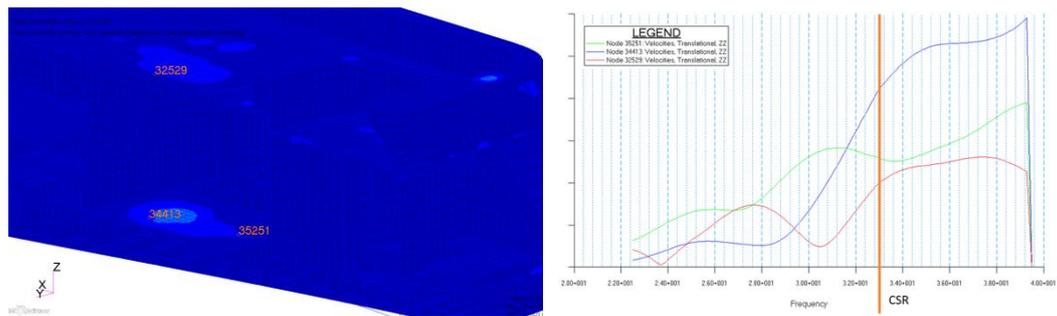


Figure 9 – Bearing force load condition: vertical component of vibration velocity distribution and vertical component of vibration velocity versus frequency graph for the selected nodes

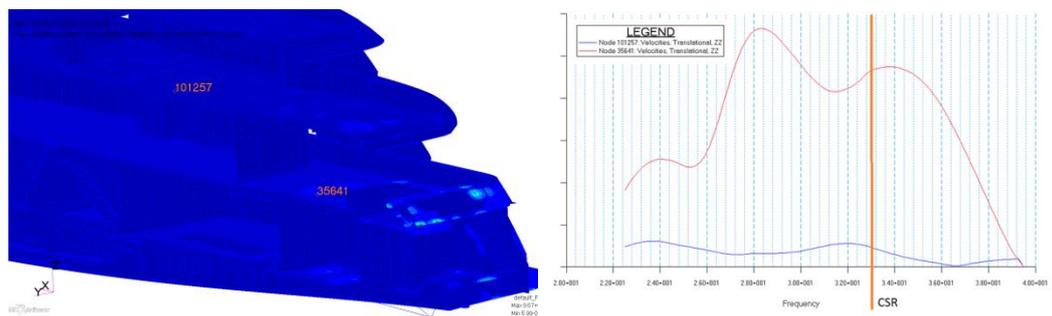


Figure 10 – Misalignment of the propellers load condition: vertical component of vibration velocity distribution and vertical component of vibration velocity versus frequency graph for the selected nodes

X. CONCLUSIONS

The free vibration analysis shows that the lowest natural frequencies are within the expected range. The first global bending deformation frequency, determined from calculation, is close to the first excitation frequency of the propeller shaft in normal cruising condition. Nevertheless the forced vibration analysis with the load conditions calculated and calibrated with past experience shows vibration levels in compliance with the RINA's COMFORT class criteria.

The calculated vibration levels are representative of the dynamic behaviour of the ship and have been useful to investigate the vibration distribution and to address structural modifications on critical areas.

In general, the vertical vibration velocity component is higher than the transversal and longitudinal ones due to the higher amplitude of vertical dynamic forces. In particular Figures 8, 9, and 10 show the distribution of the vibration velocity vertical component, while the forced response graphs give information about the expected vibration levels to compare with the limit values. Generally, all vibration velocity spectra have peak values at the resonant frequencies corresponding to the natural modes. The highest vibration levels are principally

originated by the propellers through the integral of the pressure pulses on the stern part, in resonance condition between the blade pass frequency and the local natural frequency of stiffened panels.

Due to the particular fine mesh size the FE model has been used also to develop noise calculation in the low frequency range to investigate the structural borne noise path and reduce the uncertainties in the noise levels obtained by means the Statistical Energy Analyses methodology.

In this case the structural model has been coupled with fluid element to calculate the noise levels inside the most important spaces of the vessel normally influenced by the structural borne noise generated by all relevant noise sources.

Finally the case study will be checked by the sea trial measurements in order to perform for the future more and reliable predictive analyses of noise and vibration.

REFERENCES

- (1) ANSA version 15.0.x User's Guide, BETA CAE Systems S.A., February 2014
 - (2) MSC Nastran 2014 Dynamic Analysis User's Guide, MSC Software, November 2014
 - (3) VA One documentation, ESI, <https://www.esi-group.com/software-services/virtual-performance/va-one>
 - (4) Rules for the classification of pleasure yachts, RINA, January 2013
 - (5) Rules for the classification of yachts designed for commercial use, RINA, January 2013
-