Project 2 of
Vibration of Ships and Ocean Systems

Report of **Group 12,** composed by
Leonardo Lombardi (IST number **110188**),
Fabio Bentivogli (IST number **102376**), and
Davide Melozzi (IST number **102230**).

Course by Professors *Shan Wang* and *Sarat Chandra Mohapatra*
Lisbon, 2022

Table of Contents

analytical

1 Introduction

The goal of this study is to compute the natural frequencies and Eigen modes of a ship's flexural bending vibrations using the Mykelstad technique and Finite element modeling, which is proven by an empirical formula.

The project will cover course topics in order to calculate and estimate methods for solving a scientific problem, such as the vibrations of a ship, through the following steps:

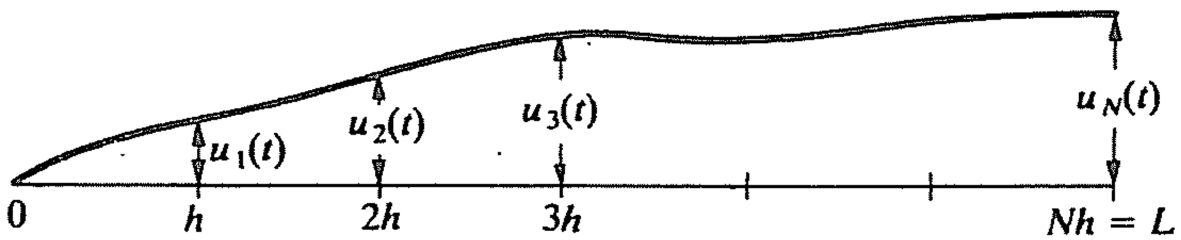
The commercial program ANSYS was used to do a numerical modal study of a ship beam, will be implemented the Mykelstad technique in computer code to find the natural frequencies and Eigen vectors associated with beam flexural vibrations and will be calculated the approximation of a ship's inherent frequencies.

2 Calculations based on Finite Element Method (ANSYS)

The Finite Element Method (FEM) is a numerical method that can be used for the accurate solution of complex mechanical and structural vibration problems. In this method, the actual structure is replaced by several pieces or elements, each of which is assumed to behave as a continuous structural member called a finite element. The elements are assumed to be interconnected at certain points known as joints or nodes. Since it is very difficult to find the exact solution (such as the displacements) of the original structure under the specified loads, a convenient approximate solution is assumed in each finite element.

The idea is that if the solutions of the various elements are selected properly, they can be made to converge to the exact solution of the total structure as the element size is reduced. During the solution process, the equilibrium of forces at the joints and the compatibility of displacements between the elements are satisfied so that the entire structure (assemblage of elements) is made to behave as a single entity.

The basic procedure of the finite element method, with application to simple vibration problems, is presented in this chapter. The element stiffness and mass matrices and force vectors are derived for a beam element, unstiffened and stiffened plate. After introducing the geometry and all the variables that describe each item, we put them into a computational software. This program is called ANSYS. Next, we studied and solved a modal analysis, identified the natural frequencies of each mode and displayed the behavior of each mode for a free vibration. The initial boundary conditions have been given for each problem; these conditions are necessary in order to establish the answer of the system through ANSYS.



The Finite Element Method, as practiced today, began as a method of structural analysis, being related to the direct stiffness method. This direct approach may be satisfactory for static problems, but encounters difficulties in handling dynamic problems, such as vibrations of continuous media. The idea of this chapter is too study the concepts and developments of free vibrations on different elements and put it in comparison with the analytical approach. In the last project we studied the vibrations of discrete systems, whereas this one is devoted to continuous systems. We shouldn't, however, consider that discrete and continuous systems represent different types of system exhibiting dissimilar dynamical characteristics. In reality the opposite is true, as discrete and continuous systems represent merely two mathematical models of identical physical systems. The basic difference between the two is that a discrete system has a finite number of degrees of freedom and continuous systems have an infinite number. Nevertheless, because discrete and continuous systems represent in general models of identical physical systems, they display similar dynamical behavior.

The equations of motion of continuous systems will be partial differential equations; these are identified for several continuous systems, including the transverse vibration of a beam, unstiffened and stiffened plate by considering the free-body diagram of an infinitesimally small element of the particular system and applying the Newton's second law of motion. The free-vibration solution of the system is found by assuming harmonic motion and applying the relevant boundary conditions. The solution gives an infinite number of natural frequencies and the corresponding mode shapes. The free-vibration displacement of the system is found as a linear superposition of the mode shapes, the constants involved being determined from the known initial conditions of the system.

2.1 Validation of the finite element method by considering a uniform beam and plates

In this chapter an uniform beam, unstiffened and stiffened plate free vibrations will be studied both analytically and with the FEM analysis on ANSYS. The formulas introduced have been implemented by introducing the elements and analyzing their initial boundary condition. These conditions occupy a peculiar and important variable when it comes to display the behavior of an element subject to modal analysis.

2.1.1 Consider a uniform beam and calculate the first three natural frequencies and Eigen functions analytically. The length of the beam and cross section are listed in Table A.

In the first paragraph the analytical formulas for the first three natural frequencies and the Eigenfunctions of an uniform beam must be found. In the table below we can see the variables related to our group that describe the geometry of the beam:

Uniform Beam

L [m]	**30**
B [m]	**1.2**
H [m]	**0.4**
E [N/m^2]	**2.00E+11**
ro [kg/m^3]	**7850**
Area [m^2]	**0.48**
I_xx [m^4]	**0.0064**
I_yy [m^4]	**0.0576**
I [m^4]	**0.064**
Volume [m^3]	**14.4**
Mass [kg]	**113040**

Table 1. Geometry and charateristics of the uniform beam

The free vibration solution can be found using the method of separation of variables as:

$$w(x, t) = W(x)T(t)$$

By expressing the equation of motion and rearranging the yields:

$$\frac{c^2}{W(x)}\frac{d^4W(x)}{dx^4} = -\frac{1}{T(t)}\frac{d^2T(t)}{dt^2} = a = \omega^2$$

where a = ω can be shown to be a positive constant. Therefore the equation can be rewritten as:

$$\frac{d^4W(x)}{dx^4} - \beta^4W(x) = 0\,, \quad \frac{d^2T(t)}{dt^2} + \omega^2T(t) = 0$$

where:

$$\beta^4 = \frac{\omega^2}{c^2} = \frac{\rho A\omega^2}{EI}$$

The solution of the first equation precedently introduced is assumed to of exponential form as:

$$W(x) = Ce^{sx}$$

where C and s are constants. After identifying the roots of this equation we can identify the new solution and it can be expressed:

$$W(x) = C_1 e^{\beta x} + C_2 e^{-\beta x} + C_3 e^{i\beta x} + C_4 e^{-i\beta x}$$

where C1, C2, C3, C4 are different constants in each case. The equation can finally be expressed more conveniently as:

$$W(x) = C_1(cos\beta x + cosh\beta x) + C_2(cos\beta x + cosh\beta x) + C_3(sin\beta x + sinh\beta x) + C_4(sin\beta x + sinh\beta x)$$

The natural frequencies of the beam can be determined as:

$$\omega = \beta^2 \sqrt{\frac{EI}{\rho A}} = (\beta l)^2 \sqrt{\frac{EI}{\rho A L^4}}$$

The function W(x) is known as the normal mode or the Eigenfunction of the beam and ω is called the natural frequency of vibration. As explained before, for any beam there will be an infinite number of normal modes with one natural frequency associated with each normal mode. The unknown constants C1 to C4 and the value of β can be determined from the known boundary conditions of the beam.

In our uniform beam the given boundary conditions are **clamped-clamped** which means that the beam is fixed at both ends. In this case, the transverse displacement and the slope of the displacement are zero. Hence, the boundary conditions are:

$$W(0) = 0,\ \frac{dW}{dx}(0) = 0$$

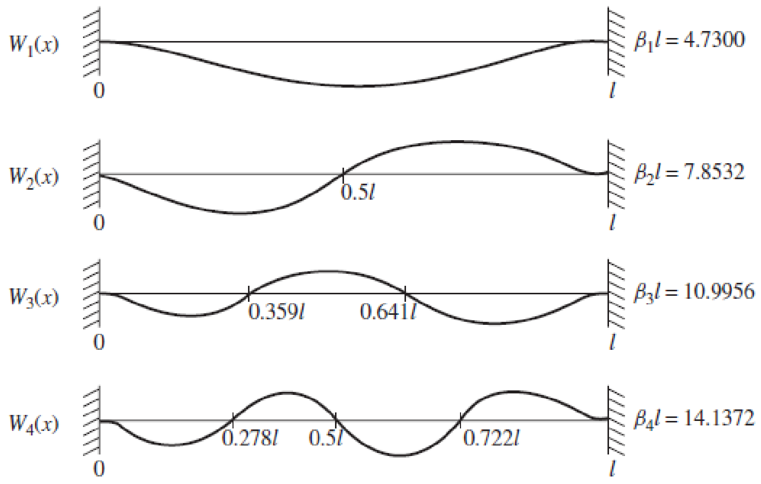
$$W(l) = 0;\ \frac{dW}{dx}(l) = 0$$

After applying the boundary conditions to the previous formula of the Eigenfunction, if $\beta_n l$ denotes the n-th root of the transcendental equation

$$cos\beta l * cos\beta l - 1 = 0$$

the corresponding mode shape can be obtained as:

$$W(x) = C_n * [(cos\ \beta_n x - cosh\ \beta x) - \frac{cos\beta_n l - cosh\beta_n l}{sin\beta_n l - sinh\beta_n l} * (sin\beta_n x - sinh\beta_n x)]$$



For each mode we identified values of βl and n frequencies for n modes where:

$$\omega_n = (\beta l)^2 * (EI/\rho A L^4)^{1/2},\ \beta_n l \simeq (2n + 1) * \pi/2$$

The tables below display the constant $(EI/\rho A L^4)^{1/2}$ and then the next tables the natural frequencies for each mode. In our case, it was requested to study only the first 3 modes of the free vibration:

Constants
0.6476003

1st mode		
βL (1st mode)	**[rad/s]**	**[Hz]**
beta_1	**omega_1**	**omega_1**
4.71239	**14.38101**	**2.28881**

Table 1. Natural frequencies for the first mode

2nd mode		
βL (2nd mode)	**[rad/s]**	**[Hz]**
beta_2	**omega_2**	**omega_2**

2nd mode		
7.853981634	39.94724	6.35780

Table 2. Natural frequencies for the second mode

3rd mode		
βL (3rd mode)	[rad/s]	[Hz]
beta_3	omega_3	omega_3
10.99557429	78.2965	12.46129

Table 3. Natural frequencies for the third mode

Next the three Eigenfunctions have been plotted. As expected the nodes related to each normal mode have the behavior expected. The amplitude of the function has been normalized ($C_n = 1$) and it was possible to clearly see the curves. The analytical method will be a reference point for the FEM analysis in order to understand the free vibration from both perspectives. As stated before, in a continuous system it is possible to identify an infinite number of normal modes but in our case we stopped to three of them.

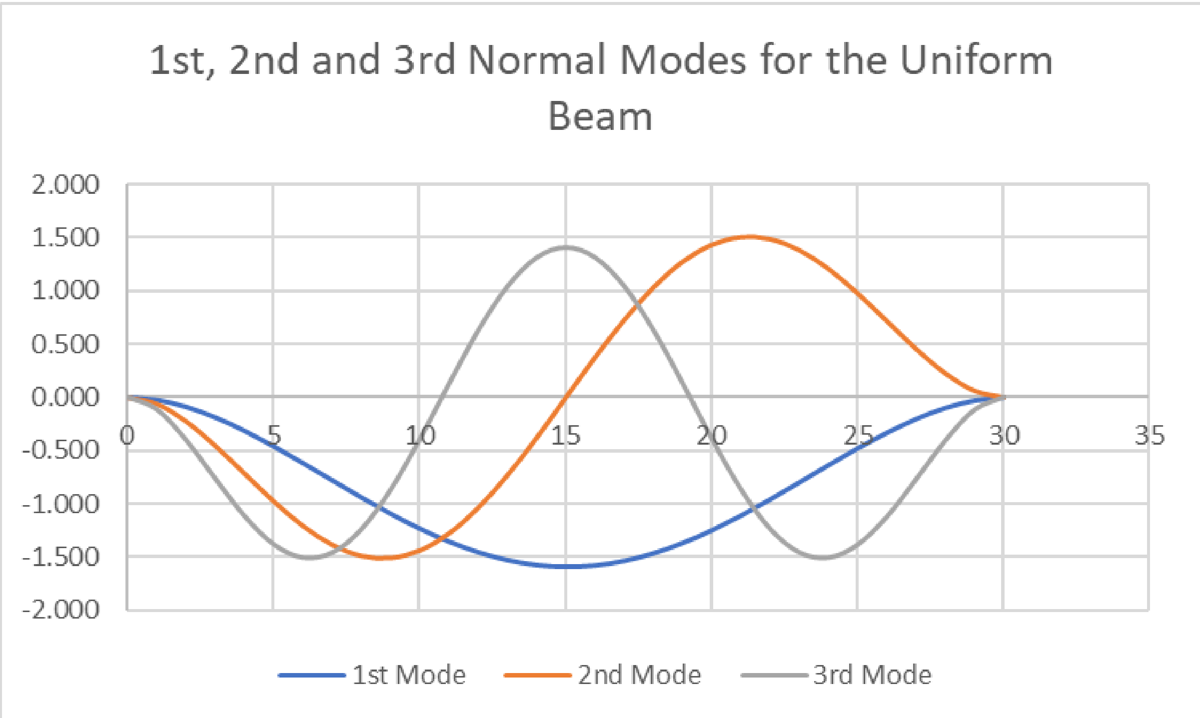


Figure 1. Behavior of the 1st, 2nd and 3rd mode

2.1.2 Solve the uniform beam problem of 1.1(a) with the FEM method using ANSYS.

Once the analytical analysis has been done, the FEM method has been implemented. For this project we used the APDL software from ANSYS and developed a code in order to introduce the same uniform beam characteristics that we used for the analytical analysis. The sequence used to produce the modal analysis with ANSYS will be here described:

- First, we introduce the ELEMENT TYPE which is BEAM3.
- The real constants have been introduced with the R-code input. The area, inertial moment on z-z and the thickness of the beam has then been introduced.
- The same goes for the MATERIAL PROPERTIES where the Young module, Poisson coefficient and density was added.
- Then for the geometry we used the MODELING paragraph where we created the KEYPOINTS (one at x = 0 and one at x = 30 m).
- A line was created and meshed to obtain a solid continuous system.
- The MODAL ANALYSIS was introduced and we extracted 3 natural frequencies for 3 different modes.
- The boundary conditions have been implemented to constrain all degrees of freedom on both ends of the beam.
- The solution in the SOLVE panel has been then found.
- After that, we read and plotted the results that will be displayed in the next figures.

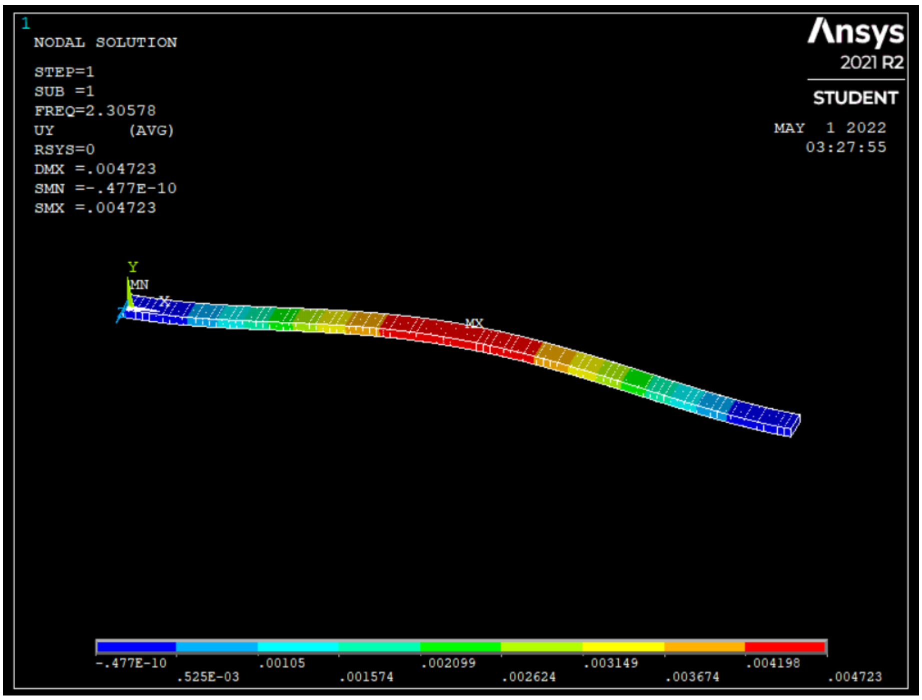


Figure 2. 1st normal mode for the uniform beam (ANSYS)

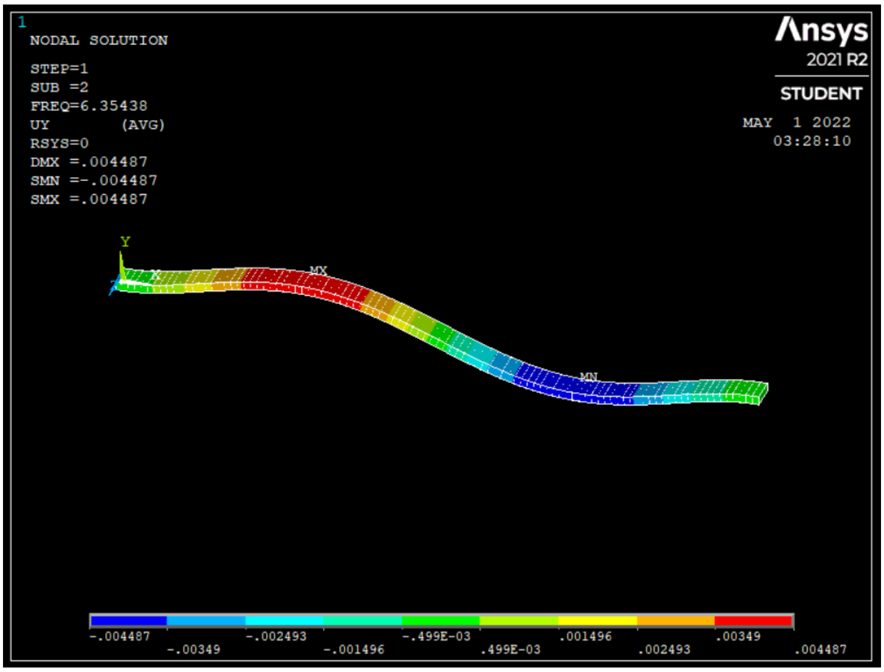


Figure 3. 1nd normal mode for the uniform beam (ANSYS)

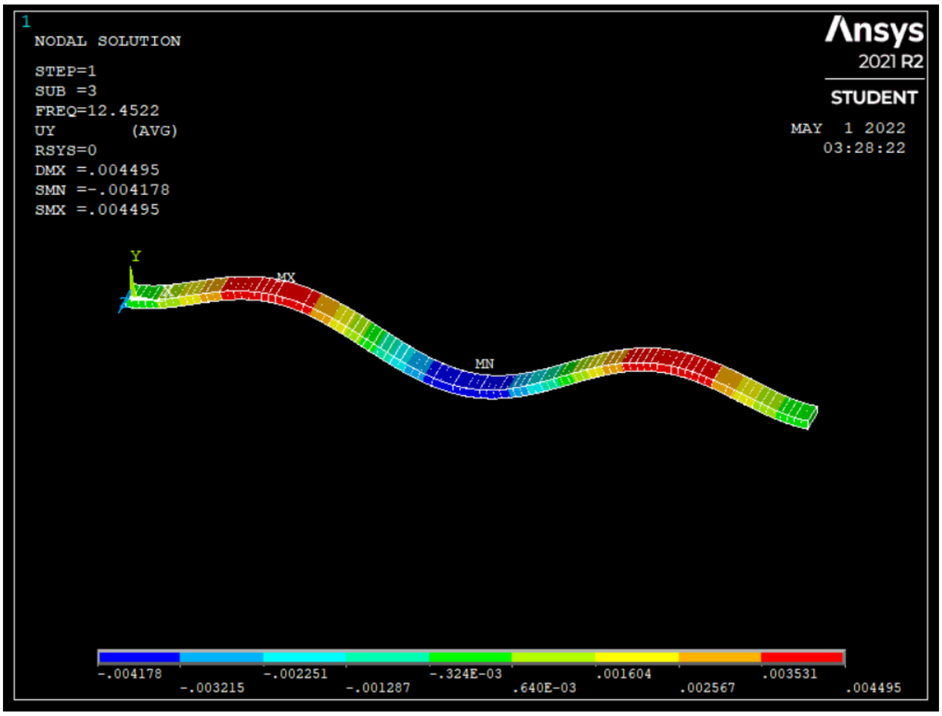


Figure 4. 3rd normal mode for the uniform beam (ANSYS)

After checking that our analysis has been correctly completed, we can finally analyze the discrepancy between the analytical and the FEM analysis.

	1st Mode	2nd Mode	3rd Mode
Analytical [Hz]	2.28881	6.35780	12.46129
FEM [Hz]	2.3058	6.3544	12.452
Discrepancy [%]	0.742381	0.053492	0.074549

Table 4. Discrepancy values between the analytical and the FEM analysis

As expected the discrepancy between the methods is very little. The FEM analysis was therefore done correctly and we can move on to the next paragraph to study a rectangular membrane.

2.1.3 Numerical modal analysis of an unstiffened plate using the software ANSYS. Determine the first three natural frequencies and natural modes of the vertical vibration. The length of the beam and cross section are listed in Table B

A plate is a solid body bounded by two surfaces. The distance between the two surfaces defines the thickness of the plate, which is assumed to be small compared to the lateral dimensions, such as the length and width in the case of a rectangular plate and the diameter in the case of a circular plate. A plate is usually considered to be thin when the ratio of its thickness to the smaller lateral dimension (such as width in the case of a rectangular plate and diameter in the case of a circular plate) is less than 1/20 .

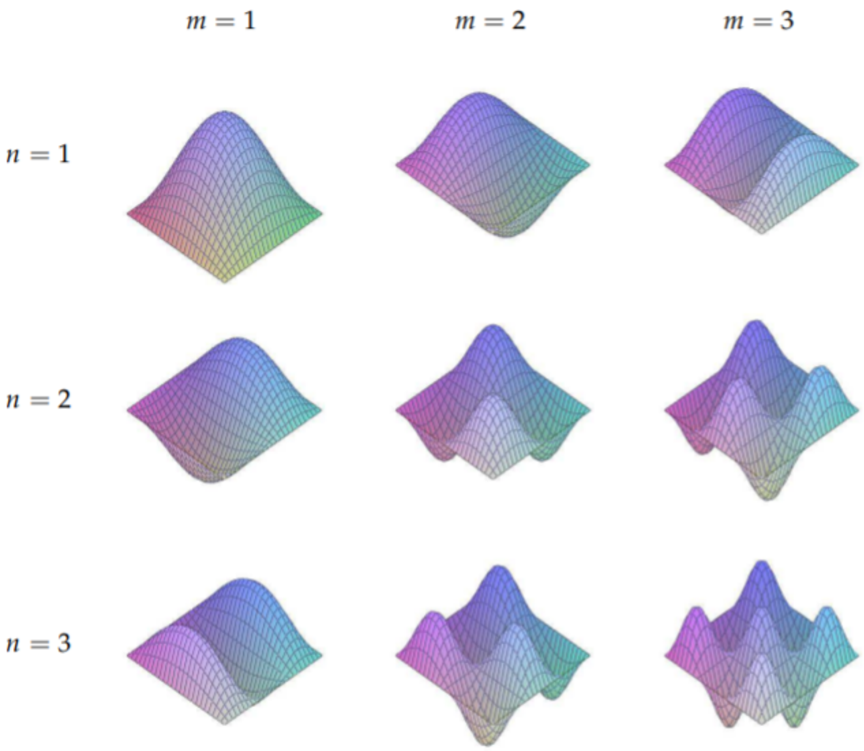


Figure 5. Representation of modes of a rectangular plate

The vibration of plates is important in the study of practical systems such as bridge decks, hydraulic structures, pressure vessel covers, pavements of highways and airport runways, ship decks, airplanes, missiles, and machine parts. The theory of elastic plates is an approximation of the three-dimensional elasticity theory to two dimensions, which permits a description of the deformation of every point in the plate in terms of only the deformation of the midplane of the plate.

Let's address the free vibration problem for a rectangular plate. Let the boundaries of the rectangular plate be defined by the lines x = 0, B and y = 0, L. To find the solution of the free vibration equation, with f = 0, we assume the solution to be of the type:

$$w(x, y, t) = W(x, y)T(t)$$

and obtain the following equations:

$$\frac{1}{T(t)}\frac{d^2T(t)}{dt^2} = -\omega^2, \quad -\frac{\beta_1^2}{W(x,y)}\nabla^4 W(x, y) = -\omega^2$$

where ω^2 is a constant and $\beta_1^2 = \frac{D}{\rho h}$

The second equation can be rewritten as:

$$\nabla^4 W(x, y) - \lambda^4 W(x, y) = 0$$

where:

$$\lambda^4 = \frac{\omega^2}{\beta_1^2} = \frac{\rho h \omega^2}{D}$$

We can express the previous equation as:

$$(\nabla^4 - \lambda^4)W(x, y) = (\nabla^2 + \lambda^2)(\nabla^2 - \lambda^2)W(x, y) = 0$$

which, by the theory of linear differential equations, the complete solution can be obtained by superposing the solutions to the previous equation of motion. In the end, we can obtain the final solution for the vibration of a rectangular plate which is:

$$W_n(x, y) = A_n sin(\alpha x)sin(\beta y) + A_2 sin(\alpha x)cos(\beta y) + A_3 cos(\alpha x)sin(\beta y) + A_4 cos(\alpha x)cos(\beta y)$$

where:

$$\lambda^2 = \alpha^2 + \beta^2$$

In our study case, the unstiffened plate we want to study is **simply supported** on all the sides. Therefore, the boundary conditions to be satisfied are:

$$w(x, y, t) = M_x(x, y, t) = 0 \quad for\ x = 0\ and\ a$$

$$w(x, y, t) = M_y(x, y, t) = 0 \quad for\ y = 0\ and\ b$$

These boundaries conditions can be expressed in terms of W, as:

$$W(0, y) = \frac{d^2W}{dx^2}(0, y) = W(a, y) = \frac{d^2W}{dx^2}(a, y) = 0$$

$$W(x, 0) = \frac{d^2W}{dy^2}(x, 0) = W(x, b) = \frac{d^2W}{dy^2}(x, b) = 0$$

When these boundary conditions are used, we find that all constants A_i, except A_1, are zero; in addition, we obtain two equations that α and β must satisfy:

$$sin(\alpha a) = 0$$

$$sin(\beta b) = 0$$

The frequency equations have solution given by:

$$\alpha_m a = m\pi, \quad m = 1, 2, \ldots$$

$$\beta_n b = n\pi, \quad n = 1, 2, \ldots$$

and the natural frequencies of the plates as:

$$\omega_{mn} = \lambda_{mn}^2 \left(\frac{D}{\rho h}\right)^{1/2} = \pi^2\left[\left(\frac{m}{a}\right)^2 + \left(\frac{n}{b}\right)^2\right]\left(\frac{D}{\rho h}\right)^{1/2}, \quad m, n = 1, 2, \ldots$$

where:

$$D = \frac{Eh^3}{12*(1-\nu^2)}$$

The Eigenfunctions $W_{mn}(x, y)$ corresponding to ω_{mn} can be expressed as

$$W_{mn}(x, y) = A_{1mn} sin\frac{m\pi x}{a} sin\frac{n\pi y}{b}, \quad m, n = 1, 2, \ldots$$

In the next table we want to introduce the variables and dimensions that characterize our plate:

Rectangular Plate	
L [m]	**30**
B [m]	**13**
H [m]	**0.011**
A [m]	**390**

E [N/m^2]	2.00E+11
rho [kg/m^3]	7850
I [m^4]	2.01391667

Table 5. Geometry and characteristics of the rectangular beam

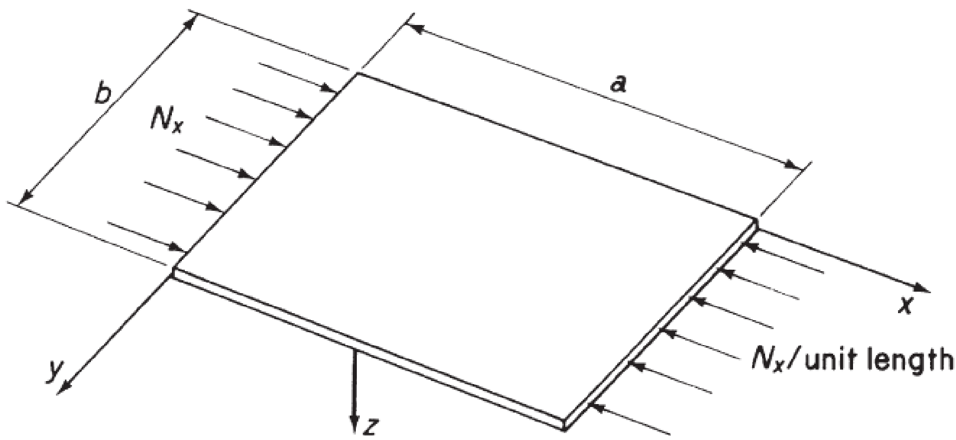


Figure 6. Coordinates and length sides of the beam

At this point, we simply used the formula to identify the natural frequencies. Since the plate is distributed on a 2-dimensional surface the set of frequencies will obtain the form of the next tables:

		m		
n	[rad/s]	1	2	3
	1	1.165493275	1.718257625	2.639532
	2	4.10920875	4.6619731	5.583247
	3	9.015401208	9.568165559	10.48944

Table 6 Set of natural frequencies in rad/s

	m			
n	**[Hz]**	1	2	3
	1	0.185494016	0.273469	0.420094
	2	0.654000885	0.741976	0.888601
	3	1.434845666	1.522821	1.669446

Table 7.. Set of natural frequencies in Hz.

These natural frequencies identify a set of modes that can be plotted and described in the graphs below:

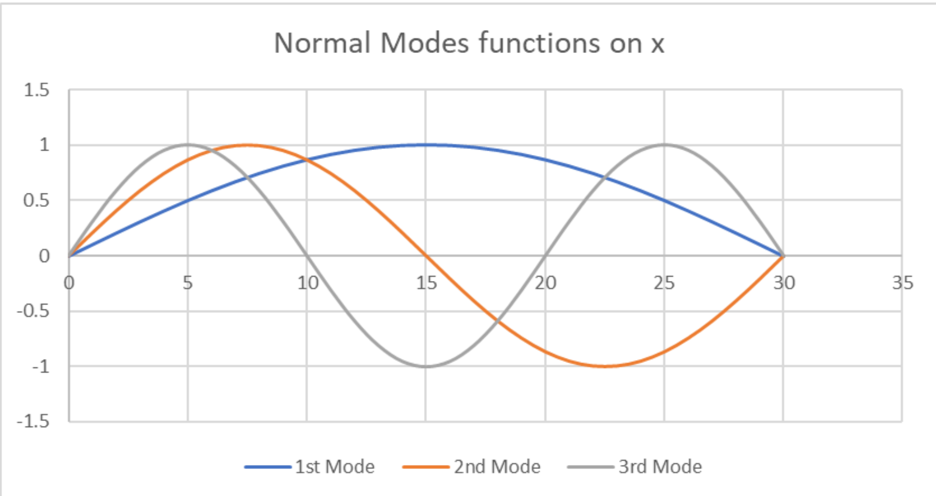
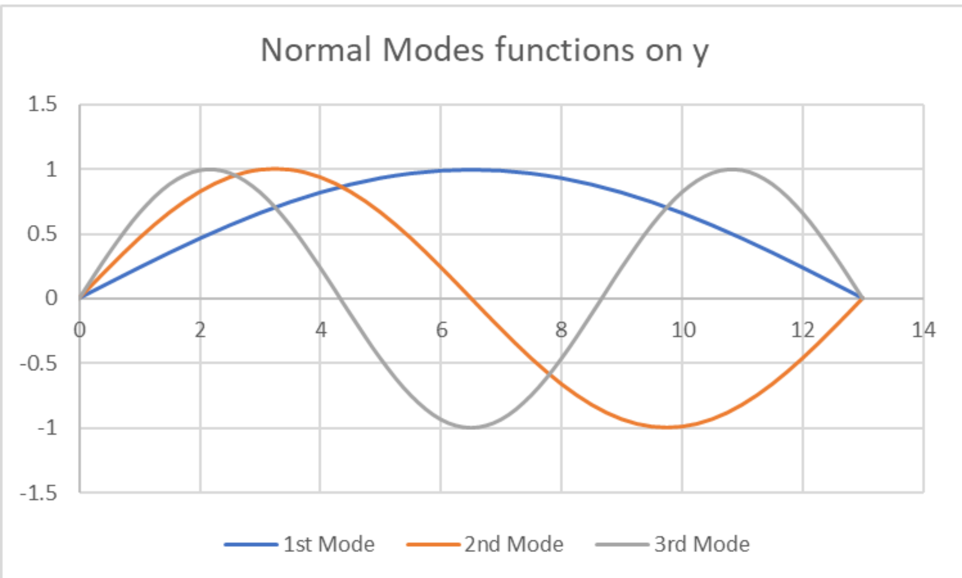


Figure 7 . Normal modes on x.



The next step is to compare these results with the FEM analysis. As before, we want to see the discrepancy between the values of the natural frequencies of the reference study (analytical solution) and the modal analysis in ANSYS.

2.1.4 Compare the numerical solutions from ANSYS with the available analytical values

Our next objective is to obtain the natural frequencies and modes with the FEM analysis and to compare them with the analytical solution. The discrepancy between the two will be studied and plotted in order to see if the FEM analysis was done properly.

The procedure is similar to what we've done in the previous paragraph and will be explained through some steps:

- First we introduce the ELEMENT TYPE which is SHELL63.
- The real constants are included with the R-command and we simply put R,1,0.011,0.011,0.011 where h = 0.011 m is the thickness of the rectangular plate.
- Then we modeled the plate by introducing the values of L and B.
- An area was created and meshed in order to obtain an element discretized in several infinitesimal elements in order to better reproduce the modal analysis.
- Since our plate is simply supported this basically means that all displacements on all axes for each boundary condition of the plate are the external lines.
- After this we introduce the MODAL ANALYSIS on the new analysis command. I decided to extract 10 natural frequencies which will correspond to 10 natural modes.
- Next the solution with the SOLVE command was found and before checking the final results obtained, the plate was properly placed in the 3D coordinates in order to better plot the behavior of the system in its free oscillation.
- After that, we ran the GENERAL POSTPROCESSOR.
- The READ RESULTS and PLOT RESULTS command was then used.
-

The table below shows the natural frequencies obtained through ANSYS:

ANSYS Analysis

1st mode	0.18548
2nd mode	0.7418
3rd mode	1.6685

Table 8. Natural frequencies obtained with ANSYS.

The first, second and third modes are, respectively, distributed as:

$$m, n_1 = (1, 1),\ m, n_2 = (2, 2),\ m, n_3 = (3, 3)$$

The figures below show the behaviour of the 3 modes identified:

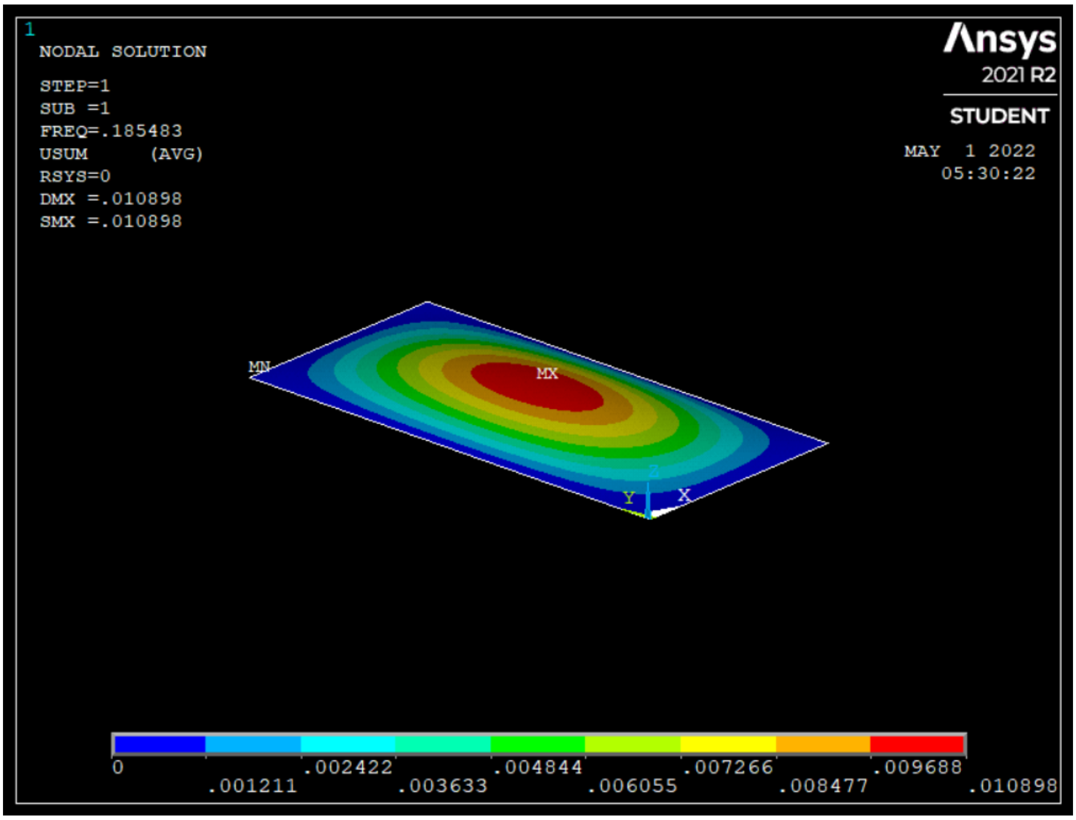


Figure 8 . Representation of the 1st mode on the ANSYS environment.

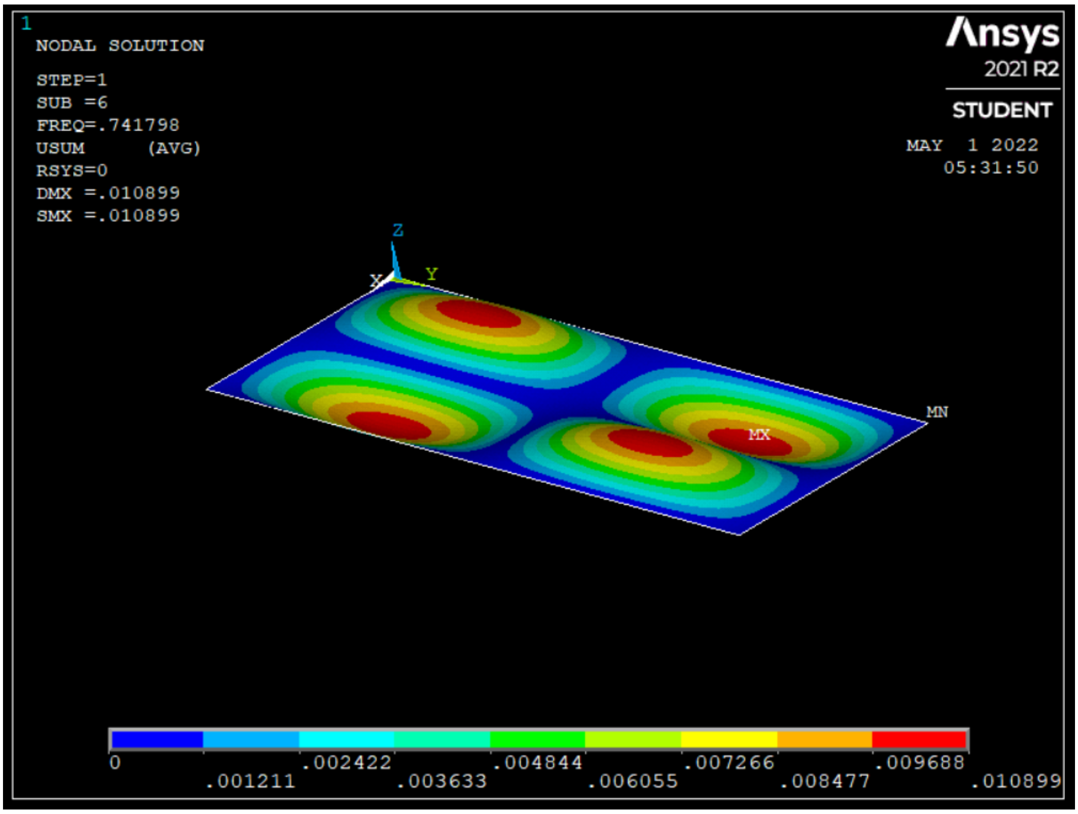


Figure 9 . 2nd mode on the ANSYS environment.

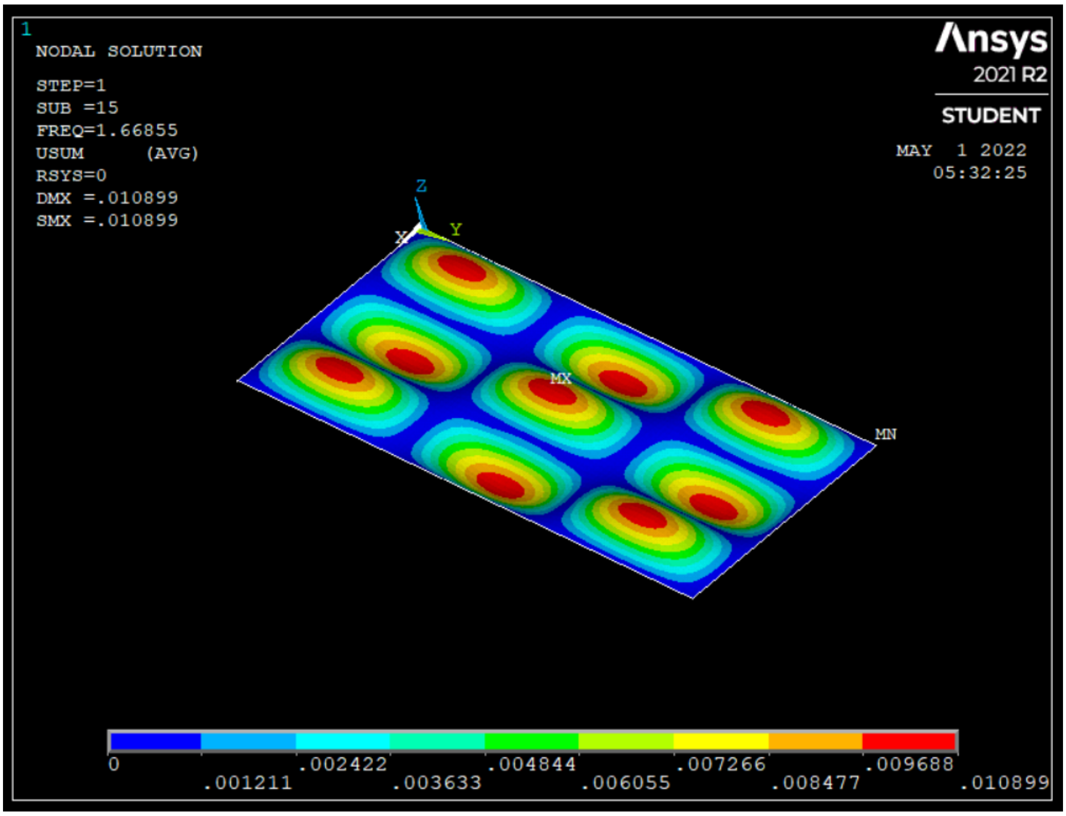


Figure 10. 3rd mode obtained on the ANSYS environment.

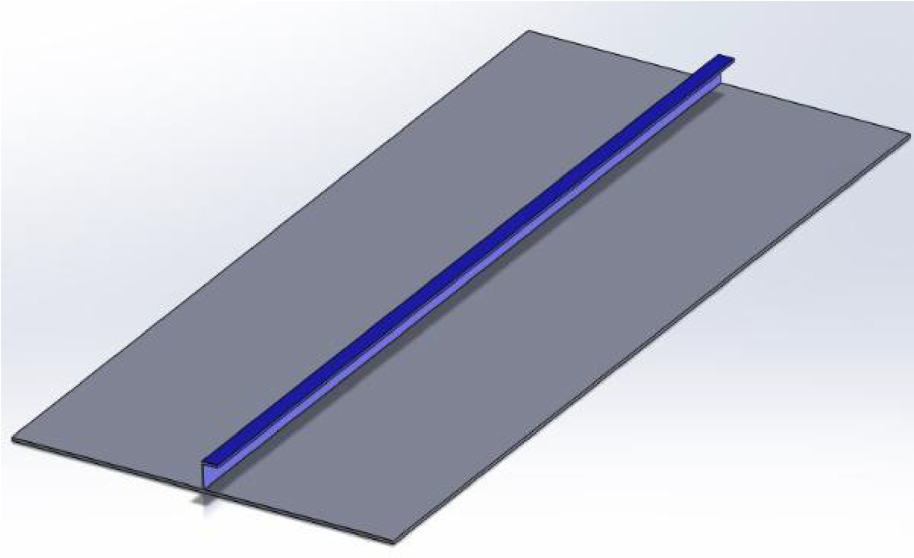
As before, we want to analyze the discrepancy between the numerical and the FEM solution. As we can observe from the table, both methods are very similar and converge to a very similar value of natural frequencies.

[%]	1st Mode	2nd Mode	3rd Mode
Analytical	0.185494	0.741976063	1.6694461
FEM	0.18548	0.7418	1.6685
Discrepancy	0.007556	0.023728997	0.0566741

Table 9. Discrepancy between analytical and FEM analysis.

2.2 Calculate the vertical flexural vibration of a stiffened panel using FEM method in ANSYS. Determine the first three natural frequencies and natural modes of the vertical flexural vibrations. The dimensions of the panel are listed in Table C.

Noise and vibration control is an increasingly important area in most fields of engineering. There are many vibrating parts in structures of ships, aircrafts and offshore platforms. The amplitude of their motions can be large due to the inherently low damping characteristics of these structures. Such noise is commonly eradicated by use of heavy viscoelastic damping materials which lead to increase in cost and weight. Vibration isolators between pieces of equipment and their supporting structures can be another solution. Clearly, isolating large structures can be difficult, expensive and in some cases, such as the wings of an aircraft, almost impossible. In recent years, much attention has been focused on active noise control of structures. However, their installation and maintenance can be expensive, so possible passive solutions would be preferable.



In the case of plates/shells, one common and cost effective approach in order to improve their NVH1 performance is to add stiffeners. Stiffened plates are lightweight, high-strength structural elements, commonly used in ships, aircrafts, submarines, offshore drilling rigs, pressure vessels, bridges, and roofing units [19, 21]. Most of these structures are required to operate in dynamic environments. Therefore, a thorough study of their dynamic behavior and characteristics is essential in order to develop a perfect strategy for modal vibration control [8]. The stiffeners enhance the rigidity of base structures by increasing their cross sectional second moment of inertia. The configuration of the stiffeners should be consistent with the natural modes likely to be excited by the service loads, so as to arrive at a design with a high strength-to-weight ratio [4]. In general, the stiffening of the structures is applied, because of two main reasons: Increasing load carrying capacity and preventing buckling, especially in the case of in-plane loading.

The equation of motion for free vibration of elastic bodies, with infinitesimal displacements is:

$$[\mathrm{M}]\{\ddot{d}\} + [\mathrm{K}]\{d\} = \{0\}, \tag{1}$$

where [M] and [K] are the overall mass and stiffness matrix, respectively. {*d*} is the displacement vector and dots denote derivatives with respect to time. Overall matrices in equation (1) are obtained by assembling matrices corresponding to each element, and applying appropriate boundary conditions. In this paper, the following hypotheses are made:

- The material of the plate and the stiffeners is isotropic, linear elastic and Hookian.
- In-plane displacements are neglected in order to reduce the computational time. If the plate edges are immovable in the plane, the

in-plane displacements will be much smaller than out-of-plane ones. Therefore, in such cases this can be a rational assumption.

- Stresses in the direction normal to the plate middle surface are negligible.
- Normal to the undeformed mid-plane remains straight and unstretched in length, but not necessarily normal to the deformed mid-plane. This assumption implies the consideration of shear deformation, but it also leads to the nonzero shear stresses at the free surface, because of constant shear stress through the plate thickness.
- Rotary inertia effect is included.
- The magnitude of transverse deflection (*w*) is small in comparison to the plate thickness (*h*).

Stiffness and mass matrices corresponding to each element are the summation of the plate stiffness and mass matrices, and the contributions of stiffeners to this element as

$$[K] = [K_p] + [K_s], \tag{2}$$

$$[M] = [M_p] + [M_s], \tag{3}$$

in which plate and stiffener are denoted by subscripts p and s.

Rectangular Stiffened Plate	
L [m]	30
B [m]	13
Thickness [m]	0.011
a [m]	15
b [m]	6.5
be [m]	0.3
hw [m]	0.3
tw [m]	0.011
Condit.	Simply-supp.

Table 10. Geometry and characteristics of the stiffened plate.

We want to perform our study case in the ANSYS software. For this reason a description of the steps will be thoroughly explained:

- First we introduce the ELEMENT TYPE which is BEAM3 for the stiffener.
- The next element to introduce is the rectangular plate which will be SHELL63..
- The real constants are included with the R-command and we simply put R,1,0.011,0.011,0.011 where h = 0.011 m is the thickness of the rectangular plate.
- Then we modeled the plate by introducing the values of L and B. We also found the KEYPOINTS in the center of the plate and at half of the base and long side. In total we have 9 keypoints.
- An area was created and meshed in order to obtain an element discretized in several infinitesimal elements in order to better reproduce the modal analysis.
- At this point, it's necessary to use the CROSS SECTION path to properly position the origin and the offset of the stiffeners in the right position of the plate.
- Since our plate is simply supported this basically means that all displacements on all axes for each boundary condition of the plate are the external lines.
- After this we introduce the MODAL ANALYSIS on the new analysis command. I decided to extract 10 natural frequencies which will correspond to 10 natural modes.
- Next the solution with the SOLVE command was found and before checking the final results obtained, the plate was properly placed in the 3D coordinates in order to better plot the behavior of the system in its free oscillation.
- After that, we ran the GENERAL POSTPROCESSOR.
- The READ RESULTS and PLOT RESULTS command was then used.

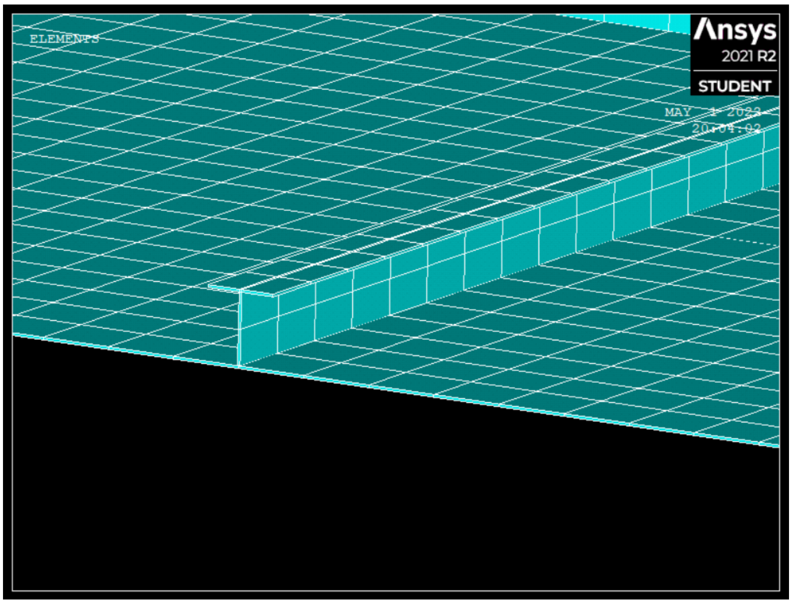


Figure. Stiffeners modeled and meshed through ANSYS.

These were the values in Hertz identified for each mode:

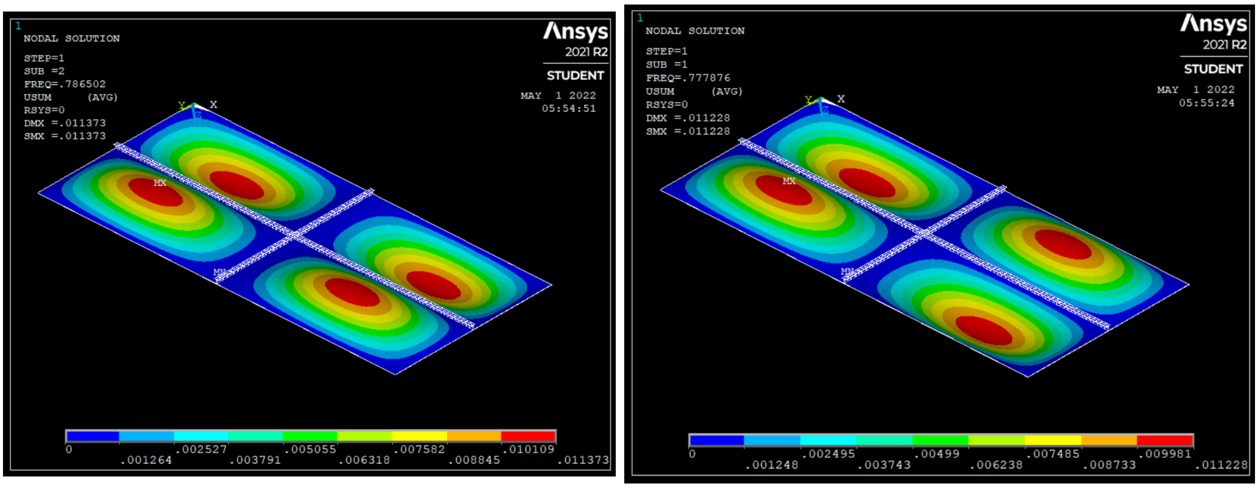


Figure 11. Comparison of the same frequencies corresponding to the 1st mode

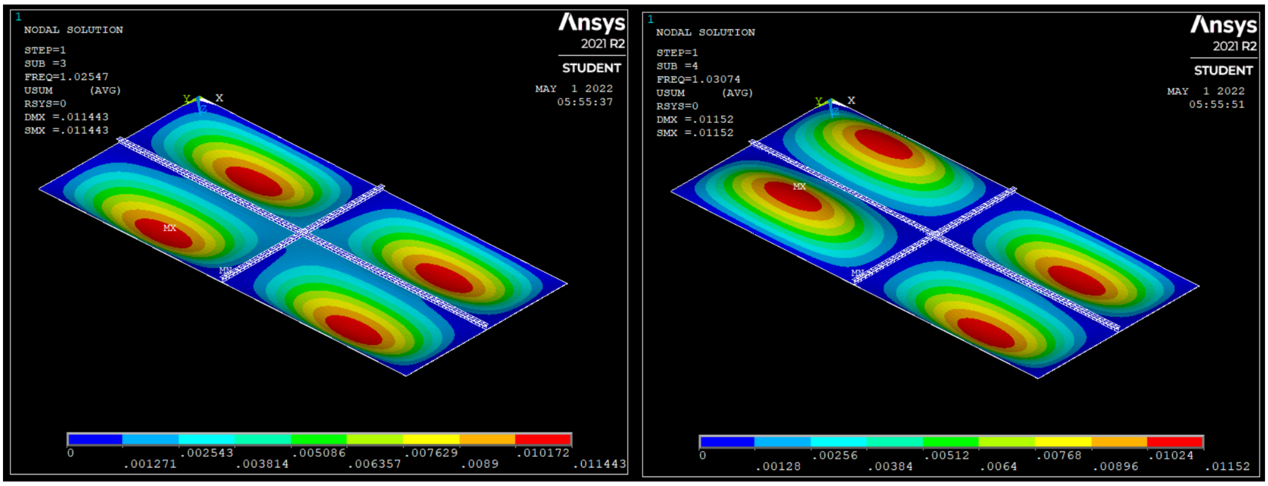
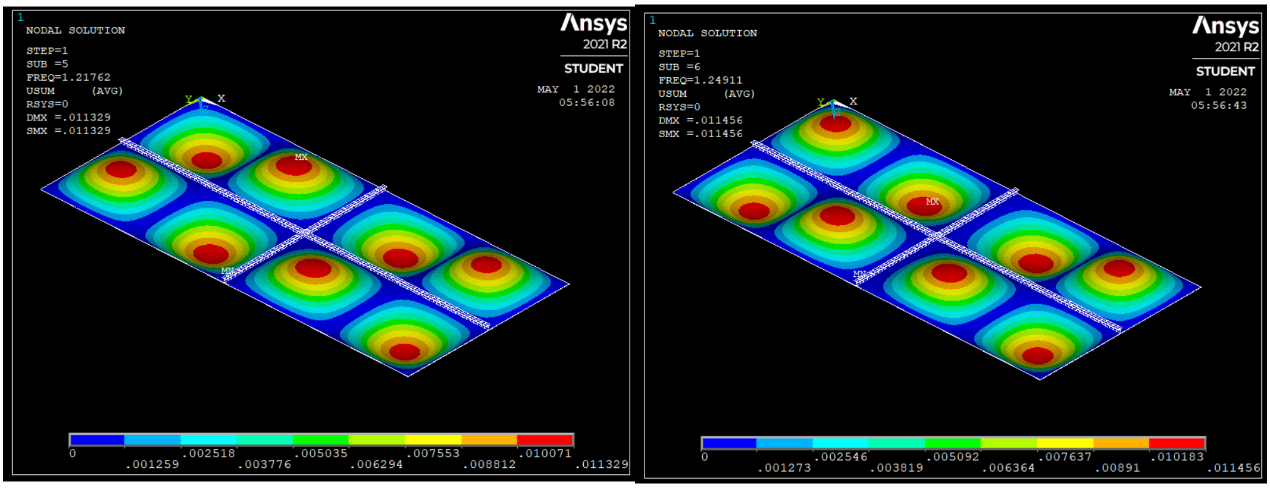


Figure 12. 2nd mode where the corresponding 2 frequencies are the same.

Figure 13. 3rd mode where the corresponding 2 frequencies are the same.



Through ANSYS we obtained a modal analysis that displayed pairs of frequencies with approximately the same values. It's safe to say that these frequencies represent the same modes.

ANSYS Analysis	
1st mode	**1st mode**
0.77788	0.7865
2nd mode	**2nd mode**
1.0255	1.0307
3rd mode	**3rd mode**
1.2176	1.2491

Table 11. Natural frequencies obtained with ANSYS.

In the next graph it's possible to see how the behaviour of the natural frequencies is tightly related to the number of stiffeners added. In our case the crossing of the stiffeners in the middle of both the long and short length of the plate create an expected pattern that may affect our study case in the possibility of studying more stiffeners.

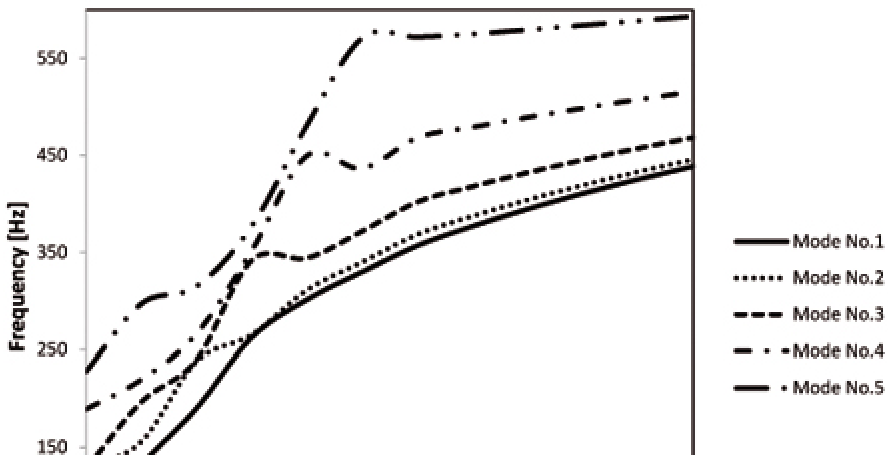


Figure 14. Correlation between the number of stiffeners and the natural frequencies.

The numerical analysis for free vibration analysis of thin rectangular solid plates is carried out using Finite Element Method under simply supported conditions with varying thickness. It is found that those natural frequency results are quite close to the reported literatures. As the thickness of the rectangular plate increases, the values are not so close to the analytical values.

2.3 Calculate the vertical flexural vibrations of a ship using FEM method implemented in ANSYS Calculate the first three natural frequencies and natural modes by using FEM method with considering the ship as a non-uniform beam

There are two types of vibration encountered on board ships. At certain rotations of the main engines, auxiliary machinery, the propeller, and the sea, the entire hull girder is put into a condition of vibration in the first kind. The movement of the hull in this situation may be readily observed by seeing down the length of the ship, and it can reach an amplitude of up to an inch at the bow and stern. This type of vibration is determined by the number of revolutions that occur in relation to the number of revolutions necessary to be utilized in long without loosening rivets. This type of vibration, which affects the entire structure, is known as synchronous or resonant vibration.

Estimating the natural frequencies for global ship vibrations is critical, especially for ships equipped with massive diesel engines. Large engines generate periodic forces and moments (1st and 2nd order excitations) that are incompatible with the ship's inherent frequencies.After the ship is built, it is impossible to remedy any possible resonance issues.

The motion of a particle or a thing that causes it to be moved from its stable equilibrium is defined as vibration. A mass's vibration might be 'in and out,' 'to and fro,' or 'oscillation.' In the case of maritime vessels, vibration may be roughly categorized into two types:

- Vibrations in the Hull

Hull vibrations comprise the main hull, substructures (such as the deck house, uptakes, masts, deck, bulkheads, and so on), and local structures (such as panels, plated and minor structure members).

Another structural problem inherent in vibrations is the effect of waves and the sea on the hull and consequently the structure above it. This is called Slamming and represents the induced forces that are impact forces on the ship's bow (or stern) when the relative motion and velocity exceed a threshold amount. The hull has a transitory elastic reaction following the collision. The last but not less important problem is called Springing. Is a resonant hull girder vibration that occurs in a steady condition as a result of wave exciting forces. When they occur, they are frequently associated with second order effects on the wave exciting forces that excite the two node vertical vibration of mono-hulls.

- Vibrations in machinery

The ship's propeller, shafting, thrust block, main engine, and other gear all create vibrations.

The worst thing that happens is that hull and machinery vibration interact that means that one can cause excitement in the other.

Vibrational Effects on Ships Vibrations are undesirable on board ships and must be avoided. Vibrations can cause hull construction damage, mechanical failure or malfunction, personal discomfort, loosening of joints, nuts, bolts, fatigue fractures, and stress fluctuation.

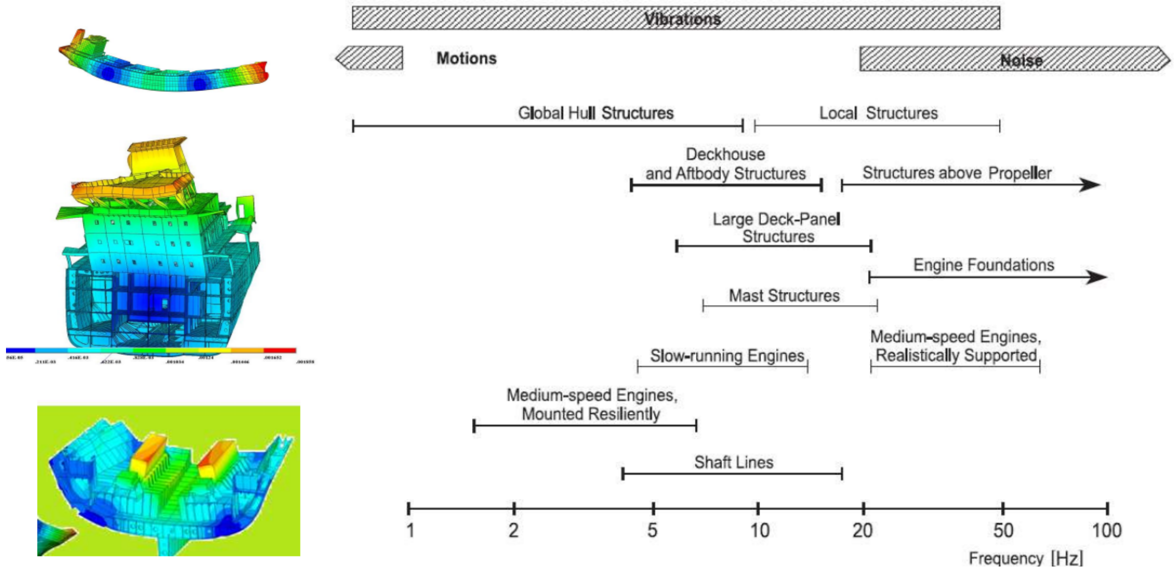


Figure 15. Frequency of vibration contributions in a ship

Excessive vibration can cause construction fatigue failure, reduce transportation efficiency, harm human health, and shorten the service life of equipment. As a result, precisely forecasting the ship hull natural frequencies during the design stage is critical for minimizing hazardous vibration, particularly total vertical vibration.

This may be due to the fact that the natural frequencies of a ship hull are determined on the basis of the free-free bar corresponding to a prismatic ship hull, in which the following basic parameters are accounted for: effect of variable cross section and load distribution, virtual added mass of water surrounding ship, shear deflection and rotatory inertia of the section of ship hull, effective breadth of the section or the reduction of rigidity.

FEM method calculation process

Considering the calculation of the vertical flexural vibrations considered in our study, i.e. that of a ship with certain physical and geometric characteristics, it is possible to develop a calculation analysis using FEM and appropriate ANSYS software.

The aim of the analysis is to calculate and discover the first three natural frequencies and modes of the ship system, considered as a non-uniform beam.

This induces the implementation of a calculation system that starts from the geometry of the water lines in the vertical plane, by means of aerial calculations, and consideration of a hull immersed in a fluid that receives a contribution of added mass that can be defined as hydrodynamic added mass. Through the latter, which is added to the total mass of the ship, it is possible to know under the calculation of determinant values dependent on geometric studies, the hydrodynamic weight and consequently, according to the subdivisions of the ship, the total weight of the ship and its sections (divided into 20 areas).

Much more specifically, the procedure we used to obtain the final results was divided into steps, with an initial part dedicated to the analytical calculation part and then implementation of the values in the FEM method.

The first part was to start with the reasoning, using initial design values, of a ship as a non-uniform beam, which is described by geometric surfaces that identify the areas in its length between the perpendiculars.

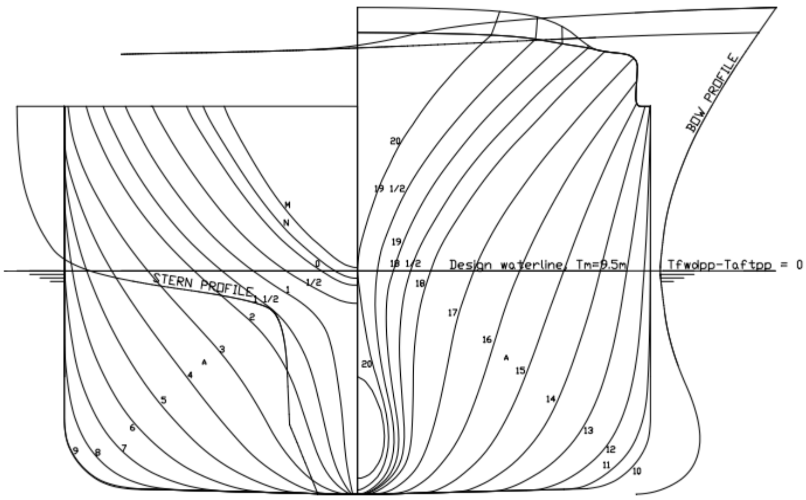


Figure 16. Bodylines of the ship

As can be seen from the diagram above, the areas are identified by calculating the surface area present between two water lines, starting from the stern and ending with the area representing the bow of the ship.

Length over all, $L_{OA\ (m)}$	166.5
Length between perpendiculars, $L_{PP\ (m)}$	157.5
Beam, *B (m)*	22.86
Draft at mid-ship, *T (m)*	8.55
Trim by stern (m)	0
Displacement, *D (t)*	18036.9
LCG section	-1.77
Gravity center height, *KG (m)*	8.55
Metacentric height, *GM (m)*	0.9

Table 12. Geometrical characteristics

In our case we have tried to calculate the areas between the water lines, up to the limit of the immersed surface that is T, using a graphic modeling software, in which the image represented above was imported, traced through polylines the various water lines and then set as reference measures the width of the ship B and the draught D, in order to set a scale 1:1 that agreed with the measures defined in reference.

It was possible to select, using the "area" command, the surface concerned and subsequently its geometric calculation with true-to-scale measurements.

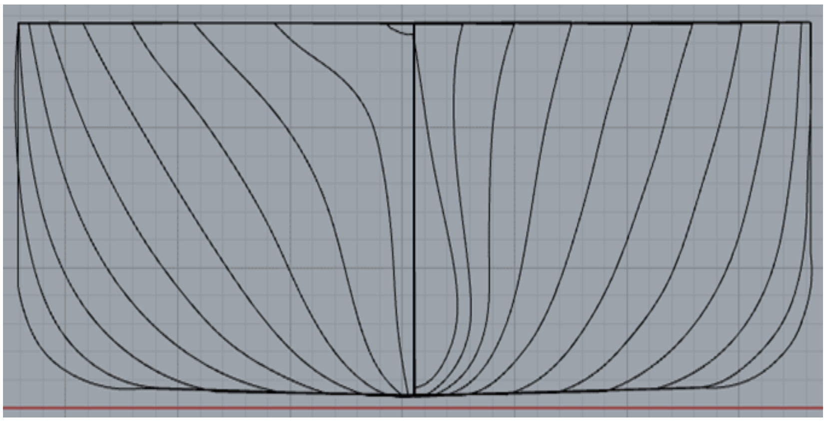


Figure 17. Representation of bodylines for calculating areas in Rhino

As we can see, the representation of the areas and their calculation was carried out in Rhino, then the distances of the respective areas, such as B and T, were also calculated geometrically from the drawing scale and shown in the following table.

Wet Areas (Rhino)		Area * 2	B/2	B	T	A/(B*T)	B/T
A0	0.167	0.334	0.875	1.75	0.31	0.615668	5.645161
A1	9.578	19.156	4.137	8.274	8.647	0.267747	0.956864
A2	25.011	50.022	6.538	13.076	8.763	0.436549	1.492183
A3	39.889	79.778	8.27	16.54	8.731	0.552438	1.894399
A4	54.123	108.246	9.765	19.53	8.632	0.642093	2.262512
A5	66.932	133.864	10.865	21.73	8.521	0.722959	2.55017
A6	77.526	155.052	11.346	22.692	8.607	0.793876	2.636459
A7	85.477	170.954	11.532	23.064	8.607	0.861178	2.679679

A8	92.876	185.752	11.37	22.74	8.57	0.953152	2.653442
A9	94.78	189.56	11.37	22.74	8.57	0.972692	2.653442
A10	94.23	188.46	11.37	22.74	8.57	0.96704	2.653442
A11	92.137	184.274	11.37	22.74	8.57	0.945568	2.653442
A12	87.357	174.714	11.245	22.49	8.57	0.906478	2.624271
A13	77.521	155.042	10.786	21.572	8.434	0.852168	2.557742
A14	66.45	132.9	9.764	19.528	8.519	0.798875	2.292288
A15	52.569	105.138	8.231	16.462	8.484	0.752795	1.940358
A16	39.559	79.118	6.563	13.126	8.521	0.707379	1.54043
A17	28.301	56.602	4.761	9.522	8.55	0.695244	1.113684
A18	18.575	37.15	2.752	5.504	8.372	0.806216	0.65743
A19	11.451	22.902	1.357	2.714	8.501	0.99264	0.319257
A20	5.901	11.802	1.259	2.518	8.107	0.578149	0.310596

Table 13. Calculation of wet ship geometries

We can also see that there are other values within the table, which have been used to determine fundamental parameters in the calculation of the hydrodynamic mass, such as the ratio B/T and A/(B*T), which are fundamental for the determination of the added mass coefficient, and will be used in the calculation of the total added mass m(x).

In this respect, the objective of this analytical part is reflected in the calculation of the latter, which is defined as the force of inertia of the water that is directly proportional to the acceleration of the ship's surface, as if it were a liquid mass accelerating together with the ship.

In a ship vibration analysis, this factor cannot be ignored, which is why its value is calculated using a method called Lewis. The value of the added mass according to this method is calculated as:

$$m(x) = \pi/8 * \rho * B^2(x) * C(x) * Jn$$

where ρ is the density, B(x) is the section beam, C(x) is the section 2-D mass added coefficient and Jn is the Lewis j factor that represents the reduction factor 2D on added mass compared to 3D dimensional vibration induced flow.

Two graphs are used to determine these two constants Jn and C(x), in which the fundamental ratios of the geometric characteristics of the hull are used.

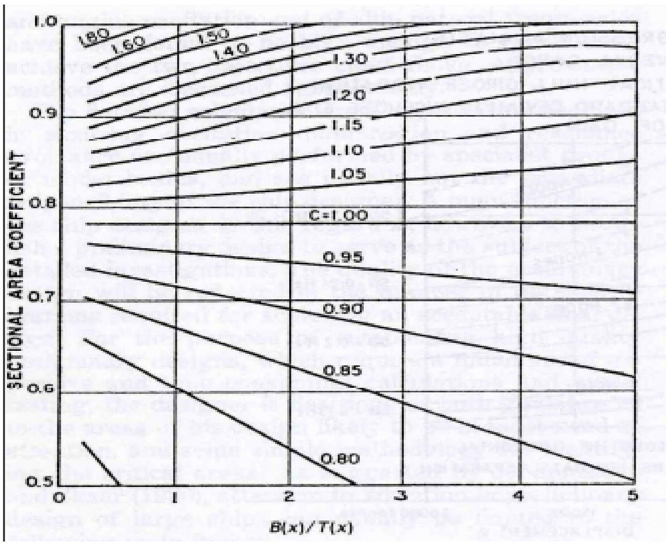


Figure 18. Graph for calculation and selection of two-dimensional coefficient

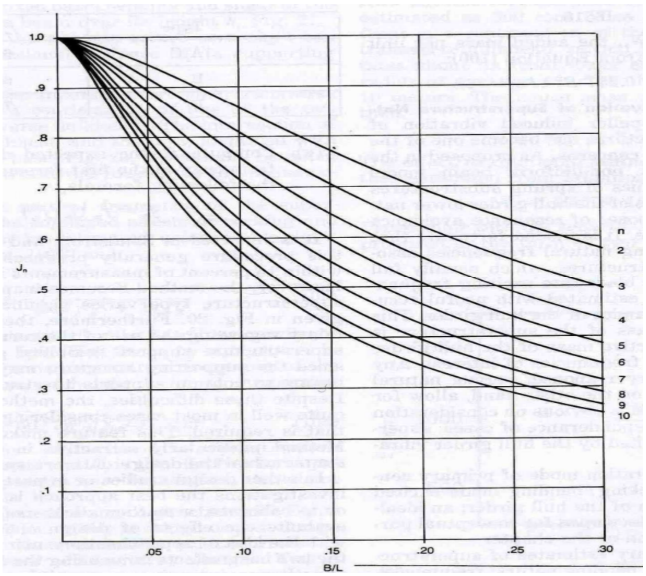


Figure. 19 Graph for calculation of coefficients Jn

In this way, using these two graphs it is possible to calculate and evaluate the constants.

In our case we have obtained:

Ship Dimensions			c (related to the number of areas)
Length AO	166.5	m	0.95

Ship Dimensions			c (related to the number of areas)
LPP	157.5	m	0.74
B	22.86	m	0.76
T	8.55	m	0.87
Trim by stern	0	m	0.9
D	18036.9	m	0.95
LCG	-1.77	m	1.1
KG	8.55	m	1.17
GM	0.9	m	1.24
Density	1.025	t/m3	1.37
B/L	0.137	-	1.37
J2	0.78		1.37
J3	0.72		1.16
J4	0.66		1.1
			1.08
			1
			0.91
			0.92
			0.95
			1.07
			0.77

Table 14. Coefficients C and Jn obtained from calculations

Deepening the concept of the value Jn, it defines an important point of the analysis that directly influences the calculation of the added mass, but we will see later on that the density useful to the calculation of the total mass of the compartment, at the variation of the three values J2,J3,J4 will not influence much on the natural frequency of the system, reason for which for simplification of the problem we can consider only the values calculated and used for J2.

Then, by applying all the necessary data in the calculation of the formula m(x) of the added mass, we obtain the following table:

m(x) [t/m]
0.913435858
15.90527223
40.79827208
74.72546338
107.7768161
140.8385067
177.8347336
195.4040111
201.3172363
222.4230755
222.4230755
222.4230755
184.2108701
160.7133161
129.3056918
85.0831477
49.22486638
26.18921828
9.035637715
2.474467865
1.53278249

Table 15. Results of added mass calculation

Accordingly, having this value the iteration of the method continues with the aim of determining the values inherent in the subdivision of the vessel into compartments: having a length between the perpendiculars, it is possible to divide by the number of required subdivisions deltax, i.e. areas (20) and calculate the weight of the additional mass for each subdivision having the same length, but different density, since the castle of weights and wetted surfaces, describe how each section has different density and consequently volume, which in the end allows to find the total weight of the subdivision along the x-axis.

Length of each compartment:

h [m]
7.875

The following table shows the results obtained from this procedure:

m section	V [m3]	W [T/m]	W section	Weight section	Volume(for dens)	density	considered area	tot mass
8.409354042	76.741875	17	27.5	216.5625	1539.192375	146.1622717	195.453	134.7884
28.35177215	272.388375	38	48.5	381.9375		266.5613986		315.1553
57.76186773	511.0875	59	66	519.75		375.2044755		505.4233
91.25113975	740.3445	73	80.5	633.9375		471.1487995		649.6005
124.3076614	953.308125	88	95.5	752.0625		569.3701292		800.7768
159.3366202	1137.60675	103	110	866.25		666.3147744		951.9635
186.6193724	1283.648625	117	124.5	980.4375		758.2267761		1099.21
198.3606237	1404.529875	132	139	1094.625		840.0415989		1234.904
211.8701559	1477.791	146	146	1149.75		884.632862		1351.067
222.4230755	1488.45375	146	146	1149.75		891.4890028		1372.173
222.4230755	1467.640125	146	146	1149.75		891.4890028		1372.173
203.3169728	1413.51525	146	146	1149.75		879.0759328		1372.173
172.4620931	1298.41425	146	140.5	1106.4375		830.8900264		1333.961
145.0095039	1133.771625	135	129	1015.875		754.2166416		1223.838
107.1944197	937.274625	123	117	921.375		668.2526736		1097.931

67.15400704	725.508	111	105	826.875		580.8429288		959.2081
37.70704233	534.3975	99	93.5	736.3125		502.8738155		828.8499
17.612428	369.1485	88	82	645.75		430.9808434		719.1892
5.75505279	236.45475	76	63.5	500.0625		328.6252979		607.5356
2.003625177	136.647	51	39	307.125		200.8381994		404.0995
-	-	27	-	-	-	-		214.1578

Table 16. Calculations procedure

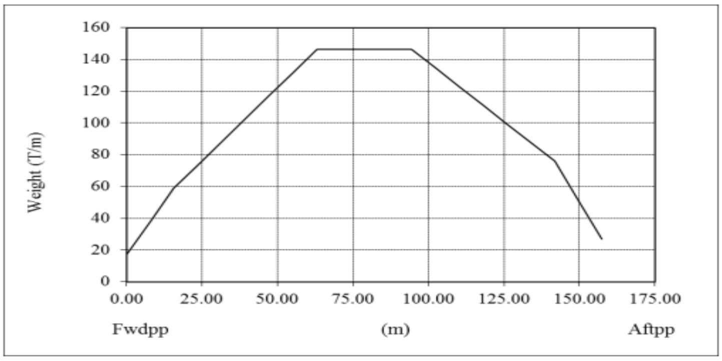


Figure 20. Distribution of masses

As we can see, at the end of the calculation of the distributions of weights, compartmental masses, volumes and added masses, we obtain the values for the total masses, which will be of fundamental importance for the calculation of the section density, which will then be entered into the ANSYS code in order to find the natural frequency values.

All these values are defined on the basis of the 20 areas calculated above.

As a final calculation procedure before initializing the ansys code with all the variables required by the macro, we need to obtain the values inherent to the moment of inertia I and the young modulus E, which are sequenced and calculated using the table that relates the cross section and vertical bending E*I.

Cross section position (m)	Vertical Bending EI (GNm^2)
0.0	3543
18.0	5905
36.0	7086
54.0	7676
108.0	7676
144.0	6495
157.5	4724

Table 17. Distribution of bending stiffness

This must be expanded and iterated for all values of the cross section, which are not specified in the table, but only graphically. So we get:

h [m]	Line	Distance	Vertical Bending Stiffness	E [Pa]
7.875	0	0	3543.00	2975622460
Izz	1	7.875	4576.38	3843512344
1190.675244	2	15.75	5609.75	4711402229
	3	23.625	6274.06	5269331440
	4	31.5	6790.75	5703276382
	5	39.375	7196.63	6044154386
	6	47.25	7454.75	6260943137
	7	55.125	7676.00	6446762067
	8	63	7676.00	6446762067
	9	70.875	7676.00	6446762067
	10	78.75	7676.00	6446762067
	11	86.625	7676.00	6446762067
	12	94.5	7676.00	6446762067
	13	102.375	7676.00	6446762067
	14	110.25	7602.19	6384769933

h [m]	Line	Distance	Vertical Bending Stiffness	E [Pa]
	15	118.125	7343.84	6167797462
	16	126	7085.50	5950824991
	17	133.875	6827.16	5733852520
	18	141.75	6568.81	5516880049
	19	149.625	5757.08	4835141539
	20	157.5	4724.00	3967496614

Table 18. Calculation of Young modulus E and Inertia moment I

These are always defined for the 20 selected areas and considering the moment of inertia of the session, in order to obtain the exact value of E young modulus. The attached graph represents the vertical bending stiffness in all the cross sections.

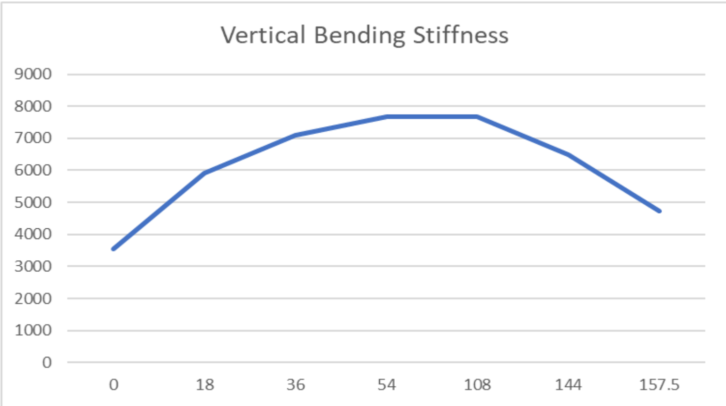


Figure 21. Representation after iteration of load distribution along the length

Once the total calculation procedure has been completed in order to obtain the values of all the variables inherent in the sections and the initialisation of the ANSYS code, we can begin to introduce the FEM as we have already defined and introduced in past sections, analysis in the reflection of a non-homogeneous beam, which is properly represented by sections formed by different densities, young modulus and distance.

We now see how variables play a fundamental role in the code in order to achieve our two main objectives: Natural Frequencies and Modal Analysis.

```
/clear
! -- Start of Macro
/FILNAME, VIBRATION
/TITLE, VIBRATION ANALYSIS
/PREP7
N,1,0
N,21,157.5
!---------------------------------------------!
!-- Define Section ''0-1'' Properties-----!
!---------------------------------------------!
et,1,3
MP,EX,1,2975622460
MP,PRXY,1,0.3
MP,DENS,1,146.1622717
R,1,195.453,1190.67524,8.55,,,0
N,2,7.857
MAT,1
REAL,1
E,1,2
```

Figure 22. Ansys code

We note that the macro is initially defined by node numbers that identify the number and position in the longitudinal system of the ship (from 1 to 21) the number 0 in the first line indicates the position of the location in reference to the ship length, it starts to increase section by section considering that the deltax is calculated and fixed for each section at 7.875m.

Then the section is defined, in which the elements and their beams are defined, on which the following definitions are based that affect the material properties, such as the young modulus E, the poisson coefficient (0.3) which remains fixed and defined and finally the density calculated with reference to the section considered.

The code line "R" then defines the characteristics of the cross-section such as calculated area, moment of inertia Izz, immersion T and added mass.

The procedure is repeated n times depending on the number of sections and changing from section to section the Young's modulus, the density and the location of the nodes, until node 21 which indicates the last subdivision. Finally, everything is plotted according to the required command and boundary condition.

In completing all the variables required in the code to satisfy the calculation of natural frequencies, the system generates a FEM analysis which is then analyzed in order to find the unknown of the problem.

The sequence includes the generation of the general postprocessor in which the read results function will be required and also read for peak, that means only the most accurate values are considered and also will allow us to obtain the three natural frequencies of the system.

Available Data Sets:

Set	Frequency	Load Step	Substep	Cumulative
1	0.0000	1	1	1
2	6.06831E-07	1	2	2
3	1.4243	1	3	3
4	3.5209	1	4	4
5	6.6433	1	5	5

Table. 19 Natural frequencies obtained from analysis

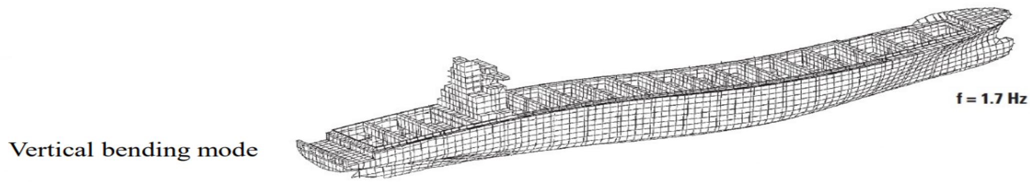


Figure 23. Example of vertical flexural response in mode one

As we can see from the frequencies obtained, the first two frequency values are too small to express a correct modal analysis as the displacement produced on the ship system is infinitesimal. For this reason, our choice was to consider sets 3, 4 and 5, which best represent the modes of the system and its respective vertical flexural vibrations.

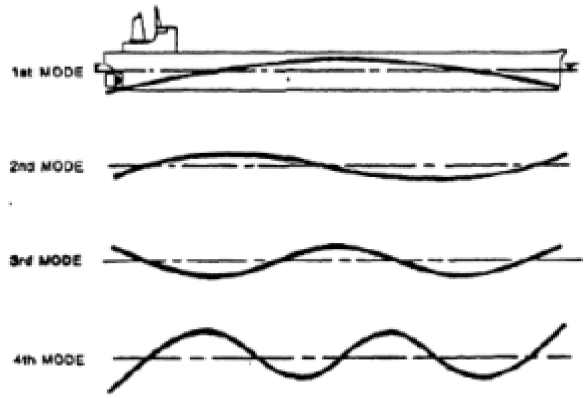


Figure 24. Different ship response depending on analysis modes

We can also understand from the figure above, how the ship's system has different responses depending on the intensity of the selected frequency and the number of modes that divide the section longitudinally, which also gives a verified reason for our choice to discard the first two frequencies.

Once the three natural frequencies have been selected, it is possible to proceed with the FEM modal analysis which allows us to understand the total displacement distributions on the structure.

The sequence to generate the following representative meshes with respect to set frequency 3, set frequency 4 and set frequency 5 respectively, is as follows:

- General postprocessor
- Read results (select the interested frequency)
- Read by peak
- Plot results
- Nodal solution
- DOF analysis
- Plot sum displacement vector.

By carrying out this procedure, we obtain the following results for the three different frequencies and their modes.

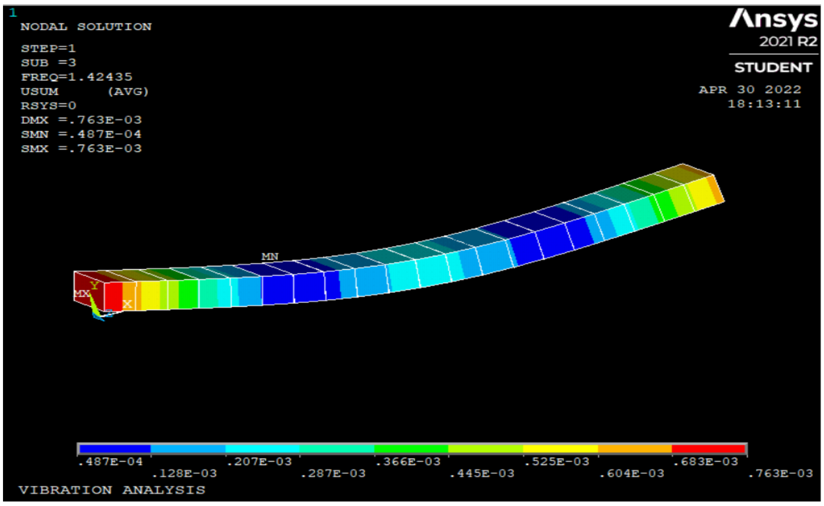


Figure 25. Nodal solution analysis for mode one

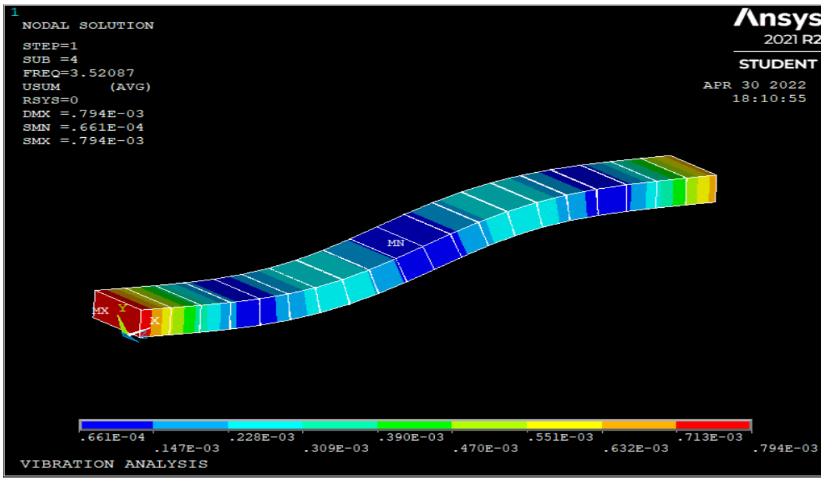


Figure 26. Nodal solution analysis for mode two

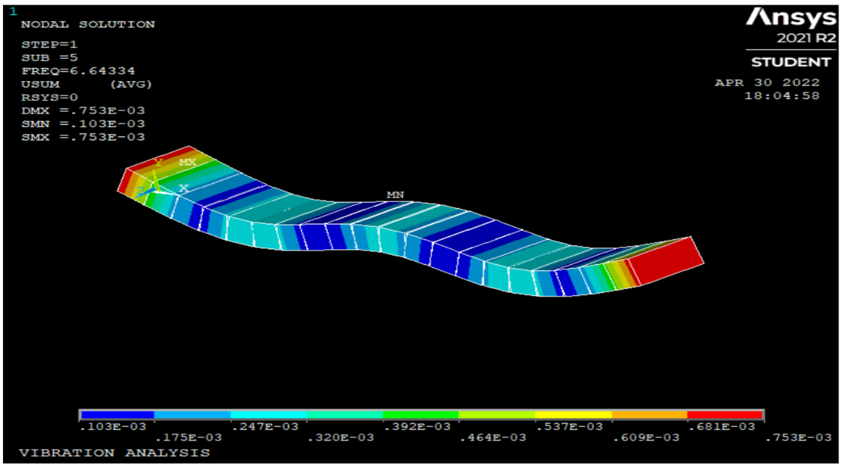


Figure 27. Nodal solution analysis for mode three

As we can observe graphically from the results obtained in the mesh, the trend of the vertical bending vibrations of the ship depends on the number of modes taken into consideration and identifies different bending trends. In mode one the flexural displacement identifies a downward variation trend which is directly proportional to the intensity of the applied frequency, then as the frequency and therefore the modes increase, the distribution of the displacement vector follows the number of flexures obtained from the modal analysis, therefore in mode two a flexure is created and in node three a double downward curvature.

To conclude, it is important to contextualize the trend of the displacements along the length of the ship. As we can see, the ship is divided into behaviors with different intensities as they have been defined separately according to their young modulus E, density and added hydrodynamic mass, so we can deduce that the stress is directly proportional to the wetted surface stressed which depends on geometric factors of the hull and weight distribution by length. Doing so creates maxima and minima of intensity which are justified by the characteristics listed above and the fact that the ship is considered as a non-uniform beam.

3 Calculations based on Myklestad method for a ship

In many real-world vibration problems, parameters are not uniformly distributed and boundaries can become sufficiently complex so that it is not possible to find closed-form solutions. In these circumstances, approximate techniques emerge as alternative methods. These techniques model distributed parameter systems as discrete systems, dividing the length of the system into small increments. The distributed mass between these increments is then lumped at geometric or mass center, and lumped masses are assumed to be connected with each other by massless springs with stiffness given by an analysis of flexibility influence coefficients.

More precisely, the points in which the masses are lumped are named stations, and the massless elastic segments between lumped masses are named fields. Then, the equations of motion (derived from Newton’s laws or Lagrangian mechanics) relate the existing forces on the right side of a station with the forces on the left side, considering the same displacement on both sides. And the well known force-deformation relation for springs relates the displacement on the right side of a field with the displacement and force on the left side, considering the same resulting force on both sides.

This approach provides an iterative process relating displacement and force at the right end of the system with force at the left end. Imposing adequate edge boundary conditions, one can derive a characteristic equation that yields the natural frequencies and, using recursive relations, calculate useful physical parameters (like translational displacement and bending moment) on stations and edges of the system. Algorithmically, this method can be explained in a two-part step-by-step process:

Algorithmically explanation of Myklestad method

Part 1 - Calculating transfer matrices

1. Discretize the system in terms of lumped masses $m(x)$, lumped bending stiffness $EI(x)$ and length of fields dx;
2. For every station:
   2.1. Using the *flexibility influence coefficients* (which are derived from considerations of the forces involved and principles from resistance of materials), calculate the *field transfer matrix* - a matrix that encompasses information of force-deformation relation on the left and right side on the field preceding that station;
   2.2. Calculate the *station transfer matrix* - a matrix that encompasses information of existing forces on the left and right side of that station;
   2.3. Calculate the transfer matrix relating station vector on left side of station with index i+1 to station vector on the left side of station i by multiplying the field transfer matrix with the station transfer matrix

The flexibility influence coefficients are given by:

$$a_i^{YM} = \frac{dx_i^2}{2EI_i}$$

$$a_i^{YQ} = \frac{dx_i^3}{3EI_i}$$

$$a_i^{\psi M} = \frac{dx_i}{EI_i}$$

$$a_i^{\psi Q} = \frac{dx_i^2}{2EI_i}$$

where a_i^{YM} is displacement at i + 1 due to a unit moment at i + 1; a_i^{YQ} is the displacement at i + 1 due to a unit force at i + 1; $a_i^{\psi M}$ is the slope at i + 1 due to a unit moment at i + 1; and $a_i^{\psi M}$ is the slope at i + 1 due to a unit force at i + 1.

Then the field transfer matrix is calculated as:

$$TF_i = \begin{bmatrix} 0 & dx_i & a_i^{YM} & a_i^{YQ}/2 \\ 0 & 1 & a_i^{\psi M} & a_i^{\psi Q} \\ 0 & 0 & 1 & -dx_i \\ 0 & 0 & 0 & 1 \end{bmatrix}$$

The station transfer matrix is calculated as:

$$TS_i = \begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 \\ 0 & 0 & 1 & 0 \\ -w^2 m_i & 0 & 0 & 1 \end{bmatrix}$$

And the transfer matrix on the station is simply:

$$T_i = TF * TS$$

Part 2 - Considering boundary conditions:

1. Using the transfer matrices calculated on every station on part 1 and considering the edge boundary conditions, calculate the *overall transfer matrix* - a matrix that encompasses the relationship between the *station vectors (vectors* containing the translational displacement, angular displacement, bending moment and shearing force on every station);
2. Mathematically analyzing the overall transfer matrix, derive the characteristic equation and calculate the natural frequencies of the system
3. Using the natural frequencies calculated above, and considering the edge boundary conditions and calculated, station, field and overall transfer matrix, calculate the station vectors (on both stations and edges)

To perform part 2, it is necessary to abandon generalities and consider specific cases depending on the edge boundary conditions. For the studied case, we consider first an uniform beam with both ends clamped and then a ship, which is modeled as a structure free in both ends. The details of the discretization process and implementation of part two of the method are given in the corresponding sections below.

3.1 Validation of the Myklestad method by considering a uniform beam

For a uniform beam, we consider a beam made of the same material in all places, so, by definition, the masses and Young modulus are constant along all dimensions of the beam. Because the given width and thickness of the beam are also constant, the calculated area moment of inertia section is also constant. It was also specified that the validation has to be made considering a beam clamped on both edges. All relevant beam information is then summarized in the table below.

p [kg/m^3]	E [N/m^2]	L [m]	Cross Section B.H [m^2]	Edge boundary conditions
7850	$2x10^{11}$	30	$1.2\,x\,0.4$	Clamped-Clamped

The process of discretization consisted in dividing the beam in segments of equal length that could be calculated depending on the number of stations specified. The figure below illustrates the process of transforming the continuous beam into a discretized set of stations and fields:

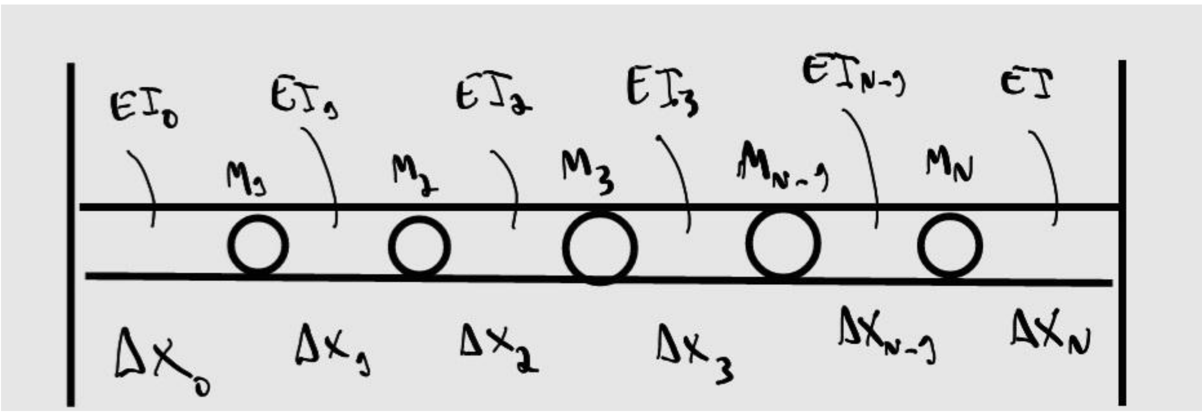


Figure 28. Lumped model for a clamped-clamped beam

Since the beam is uniform, and has constant cross sectional area, all bending stiffness and length of fields are equal. This yields:

$$dx = \frac{L}{n+1}$$

$$EI = E\,\frac{BH^3}{12}$$

where n is the number of stations (and, by inference, $n + 1$ the number of fields), E is the Young modules, B is the width of the beam and H the thickness of the beam.

3.1.1 Solve the uniform beam problem of (a) with the Myklestad method. Carry out a convergence analysis. Calculate the first three bending natural frequencies and Eigenfunctions

Given the established lumped model, to solve the proposed problem with a number of stations n using the Myklestad method for the clamped-clamped case, the following overall transfer matrix is considered:

$$T = T_n T_{n-1} ... T_2 T_1 T F_0$$

The station vector at the left edge and the state vector at the right edge are given by:

$$v_0 = \begin{bmatrix} Y_0 \\ \psi_0 \\ M_0 \\ Q_0 \end{bmatrix}; \; v_n = \begin{bmatrix} Y_n \\ \psi_n \\ M_n \\ Q_n \end{bmatrix}$$

Their relate with each other by means of the overall transfer matrix:

$$v_0 = T\, v_n$$

We know that, for clamped beams, $Y_0 = \psi_0 = Y_n = \psi_n = 0$ and $M_0, Q_0, M_n, Q_n \neq 0$. So, applying these boundary conditions and considering the result of the matrix multiplication on the expression above, we find:

$$T_{13} M_0 + T_{14} Q_0 = 0$$

$$T_{23} M_0 + T_{24} Q_0 = 0$$

This system of equations has a nontrivial solution provided the determinant of the coefficients is equal to zero. So we deduce the following frequency equation:

$$det \begin{bmatrix} T_{13} & T_{14} \\ T_{23} & T_{24} \end{bmatrix} = 0$$

from where a set of n natural frequencies w_n is found. From the same system, we have:

$$Q_0 = -\frac{T_{13} M_0}{T_{14}}$$

Arbitrating M_0 to be 1, we have the state vector at the left edge v_0 fully described.

Subsequently, the calculated natural frequencies can be used to recursively find the station vectors on each station. The formula to calculate the station vectors is:

$$v_1 = TF_0 v_0$$

$$v_2 = T_1 v_1$$

$$\ldots$$

$$v_n = T_{n-1} v_{n-1}$$

$$v_{n+1} = Tv_0$$

Remembering that T_i is the transfer matrix at a given station, not the overall transfer matrix (which is just T). The obtained station vectors are represented in the figure below:

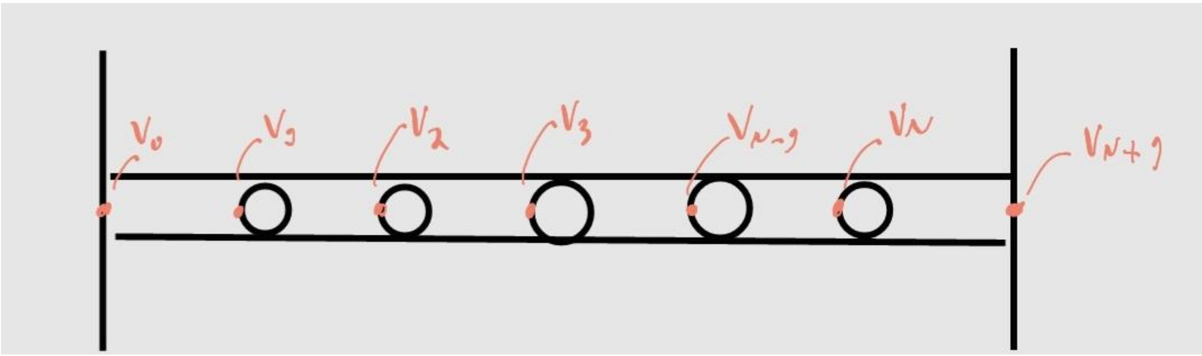


Figure 29. Representation of position of station vectors on clamped-clamped beam

For our purposes and a sufficiently small number of n, the eigenfunctions can be obtained by simple extracting the first element Y of each state vector.

Following the described procedure, a convergence analysis was made considering an increasing number of stations, yielding the following results for the first three natural frequencies and eigenfunctions:

n	ω_1 [rad/s]	ω_2 [rad/s]	ω_3 [rad/s]
6	13.41	36.900	71.73
8	13.65	37.63	73.60
10	13.81	38.07	74.57
12	13.992	38.36	75.18
20	14.14	38.97	76.40

Table 20 - First three natural frequencies for given number of stations

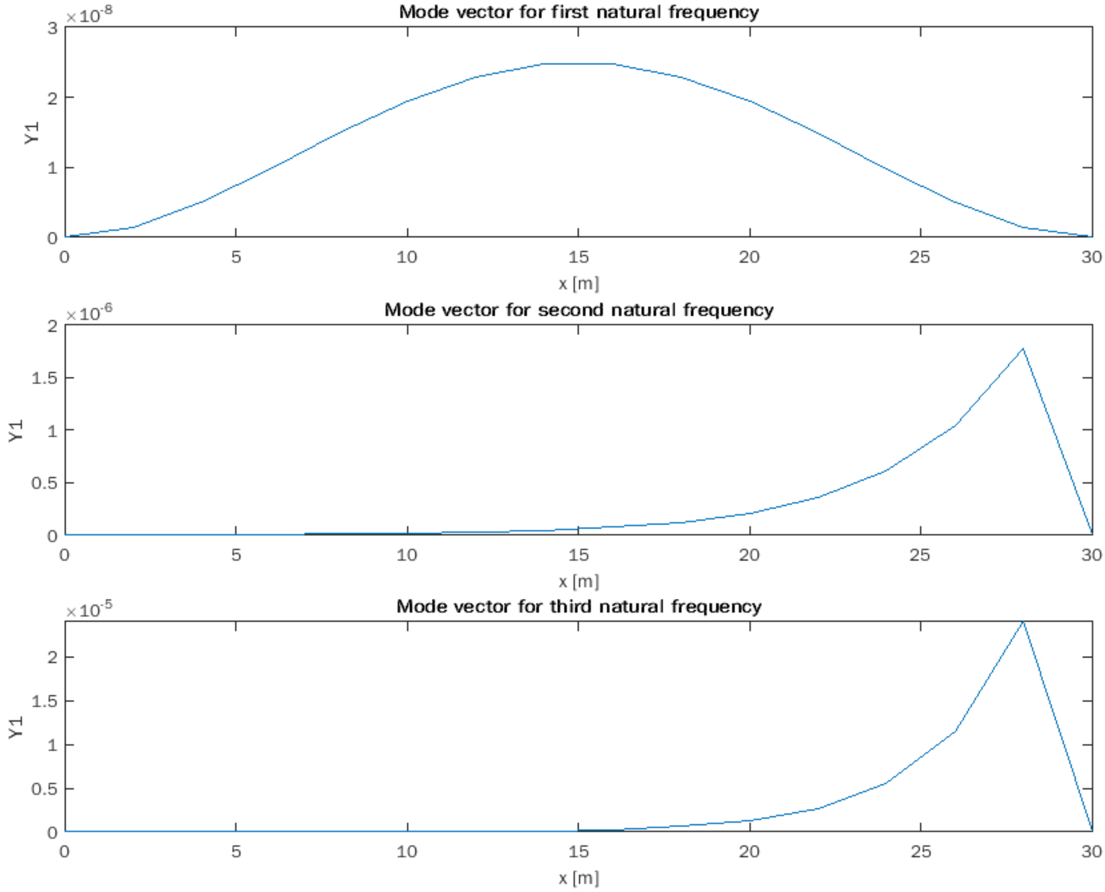


Figure 30. Mode vectors for first three natural frequencies and 20 stations

3.1.2 Compare the Above Results

With the analytical solution, it was found:

	ω_1 [rad/s]	ω_2 [rad/s]	ω_3 [rad/s]
Numerical (n=20)	14,14	38,97	76,40
Analytical	14,38	39,95	78,30
Relative Discrepancy	1,68%	2,45%	2,42%

Table 21- Comparison of natural frequencies with analytical results

3.2 Calculate the vertical flexural vibrations of a ship using Myklestad method

For a ship, the lumped model had to be developed considering a given mass and bending stiffness distribution, and the motion of the ship. The figures 20 and 21 present these distributions. The corresponding data is presented in figures below:

x(m)	W(T/m)
0.00	17
7.88	38
15.75	59
23.63	73
31.50	88
39.38	103
47.25	117
55.13	132
63.00	146
70.88	146
78.75	146
86.63	146
94.50	146
102.38	135
110.25	123
118.13	111
126.00	99
133.88	88
141.75	76
149.63	51
157.50	27

Figure 31. Data for distribution of mass

Cross section position (m)	Vertical Bending EI (GNm^2)
0.0	3543
18.0	5905
36.0	7086
54.0	7676
108.0	7676
144.0	6495
157.5	4724

Figure 32 .Data for distribution of bending stiffness

However, unlike the case of the beam validated, we must consider the impact of the ship on the surrounding fluid. We know that the water inertia forces are proportional to ship surface acceleration, implying the need to consider an effective fluid mass imagined to accelerate along with ship mass. The formula to calculate the added mass, per unit length, on a given longitudinal coordinate of the ship is given by:

$$m(x) = (\pi/8)pB^2(x)C(x)J_n$$

where p is the density of the water, $B(x)$ is the section beam, $C(x)$ the section 2-dimensional added mass coefficient and J_n the Lewis J-Factor, a reduction factor on the 2-dimensional added mass for 3-dimensionality of the vibration-induced flow.

Taking these factors into account, the process of discretization consisted in first also dividing the ship in segments of equal length that could be calculated depending on the number of stations specified. Considering the ship as a structure with free edges on both sides, the process of transforming the continuous ship into a discretized set of stations and fields was aimed to arrive at the following configuration:

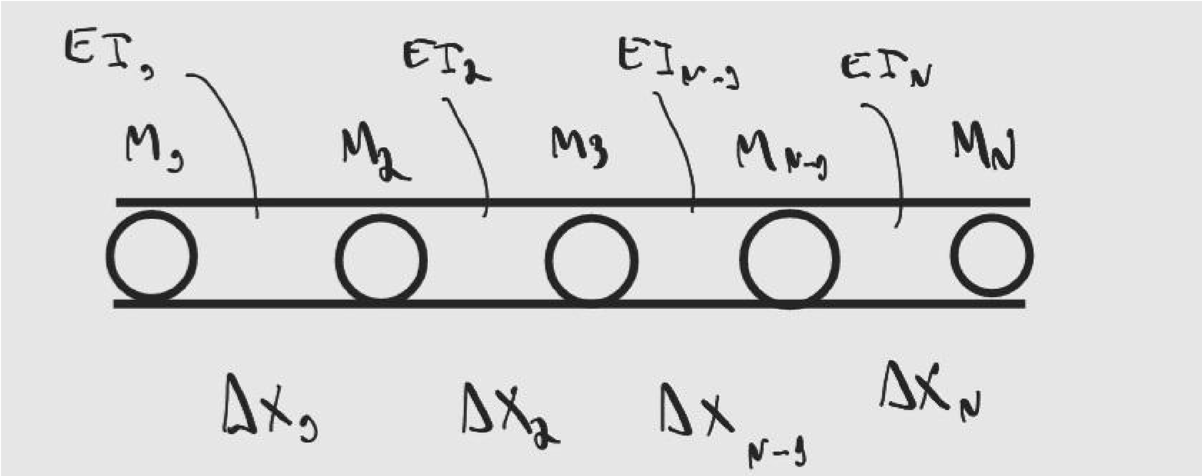


Figure 33. Lumped model considering the ship as a structure with free edges

Since we are considering segments of equal length, we also have::

$$dx \ = \ \frac{Lpp}{n+1}$$

where Lpp is the length between perpendiculars of the ship and n is the number of stations (and, by inference, $n - 1$ the number of fields).

To arrive at the proposed model, the continous masses on the segments had to be distributed along the proposed stations. Considering the added mass, for a given number of stations specified, a step-by-step process had to be carried:

Algorithmically approach to find masses on each station

1. From the data of the mass distribution and the formula to calculate the added mass, find a polynomial fit that would give the mass per unit length $W(x)$ on any given longitudinal coordinate;
2. For every segment dx, discretize the mass of the continuous segment into two lumped masses at the left and right ends of the segment - stations i and i+1. Noting that the left mass on a given iteration it's on the same position of the right mass of the previous iteration, we make sure the masses for the same station are added in the process.

To be able to discretize the mass of the continuous segment into two lumped masses at the left and right ends of the segment, we impose that the center of gravity $\bar{x}$ of the segment in both cases remains the same. This means that:

$$m_{i+1}(x_{i+1} - \bar{x}) \ = \ m_i(\bar{x} - x_i)$$

where m_{i+1} is the mass at the end station, m_i on the start station, and x_i, x_{i+1} are the longitudinal coordinate of the left and right ends of the segment, respectively.

We also know that the sum of the masses on both ends has to be equivalent to the mass of the entire segment, meaning:

$$M \ = \ m_{i+1} + m_1$$

So we find that:

$$m_i = M \ * \ (x_{i+1} \ + 2x_i - \bar{x}) \ / \ (x_{i+1} + x_i)$$

$$m_{i+1} = M \ * \ (\bar{x} - x_i) \ / \ (x_{i+1} + x_i)$$

To find the center of gravity of the segment and its corresponding mass, we have:

$$\bar{x} = \int_{x_i}^{x_{i+1}} W(x)x \, dx/M$$

$$M \ = \int_{x_i}^{x_{i+1}} W(x)dx$$

To find the center of gravity, $W(x)$ was considered to have a trapezoidal form, from where we deduce:

$$\bar{x} = \frac{dx}{3} \left(\frac{2W(x_{i+1})+W(x_i)}{W(x_{l+1})+W(x_i)}\right)$$

3.2.1 Calculate the first three natural frequencies and modes for the non-uniform ship vertical flexural vibrations with Myklestad method. Carry out a convergence analysis

Given the established lumped model, to solve the proposed problem with a number of stations n using the Myklestad method for the free-free case, the following overall transfer matrix is considered:

$$T \ = TS_n T_{n-1} ... T_2 T_1$$

We know that, for free beams, $Q_0 = M_0 = Q_n = M_n = 0$ and Y_0, ψ_0, Y_n, $\psi_n \neq 0$. So, applying these boundary conditions and considering the result of the matrix multiplication on the expression above, we find:

$$T_{31}Y_0 + T_{32}\psi_0 = 0$$

$$T_{41}Y_0 + T_{42}\psi_0 = 0$$

This system of equations has a nontrivial solution provided the determinant of the coefficients is equal to zero. So we deduce the following frequency equation:

$$det\begin{bmatrix} T_{31} & T_{32} \\ T_{41} & T_{42} \end{bmatrix} = 0$$

from where a set of n natural frequencies w_n is found. From the same system, we have:

$$Y_0 = -\frac{T_{32}\psi_0}{T_{31}}$$

Arbitrating ψ_0 to be 1, we have the state vector at the left edge v_0 fully described.

Subsequently, the calculated natural frequencies can be used to recursively find the station vectors on each station. The formula to calculate the station vectors is:

$$v_1 = T_1 v_0$$

$$v_2 = T_2 v_1$$

$$...$$

$$v_{n-1} = T_n v_{n-1}$$

$$v_n = Tv_0.$$

Following the described procedure, a convergence analysis was made considering an increasing number of stations, yielding the following results for the first three natural frequencies and eigenfunctions:

n	ω_1 [hz]	ω_2 [hz]	ω_3 [hz]
6	2.40	4.53	7.62
8	2.16	5.21	8.53

10	1.91	5.02	9.04
12	1.76	4.72	8.9
20	1.57	4.14	8.02

Table 22 - First three natural frequencies for given number of ship stations

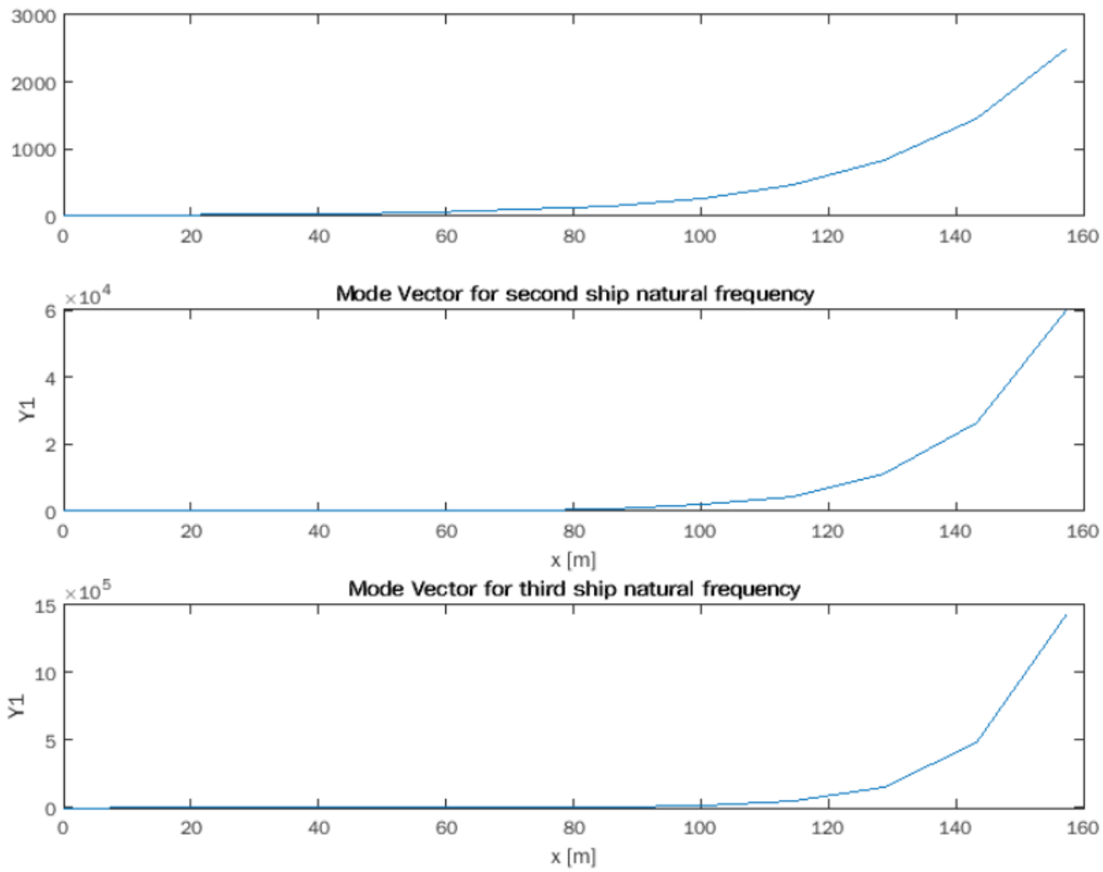


Figure 34 - Mode vectors for first three natural frequencies and 20 ship stations

3.2.2 Calculate the first three natural frequencies by using a simple formula

Kumai's formula gives the natural frequencies of the ship in terms of ship parameters:

$$N_{2v} = 3.07\ 10^{6} \sqrt{I_v/\Delta_i L^3}$$

$$\Delta_i = (1.2 + B/3T_M)\Delta$$

From where we find:

ω_1 [hz]	ω_2 [hz]	ω_3 [hz]
2.3729	4.8120	7.2768

3.2.3 Compare the estimations of natural frequency and natural modes from the FEM method (1.3), the results from Mykelstad method (2.2a) and the empirical calculation (2.2b)

Method	ω_1 [hz]	ω_2 [hz]	ω_3 [hz]
Numerical (n=20)	1.57	4.14	8,02
Simple Formula	2.3729	4.8120	7.2768
FEM	1.42	3.520	6.64

Table 23 - Comparison for natural frequencies for different methods

4 Conclusions

This work consists of the study of natural frequencies and normal modes using two different methodologies: using software and applying numerical methods. It is noted that the methodology using the ANSYS software seems to have led to consistent results when compared with the calculated analytical values. On the other hand, the application of numerical methods with Myklestad was clearly unsuccessful - although producing consistent results for natural frequencies, it was not possible to obtain minimally reliable mode vectors/eigenfunctions.

There are a myriad of possible causes for the errors found in the application of Mykelstad's method. Although applying the method with problems proposed in books has generated compatible results, it is quite possible that there was some kind of error in the process of discretization and calculation of state vectors.

It is hoped that, to correct such errors, it will be possible to validate the results with other similar existing problems. This would imply an iterative process of trial and error until the error is found, either in the formulation of the problem or in the implementation of the algorithm itself in MATLAB.

List of Figure Captions

Reference List

[1] Project 2/Dynamics and Hydrodynamics of Floating Bodies, https://fenix.tecnico.ulisboa.pt/disciplinas/VN/2021-2022/2-semestre/project-2

[2] Fundamentals of Vibration, http://www.iust.ac.ir/files/fnst/ssadeghzadeh_52bb7/files/EB__Fundamental_of_Vibration.pdf

[3] Principles of Naval Architecture Series: Vibration, https://www.sname.org/principles-naval-architecture

[4] Mechanical Vibrations, Rao
https://xn--webducation-dbb.com/wp-content/uploads/2019/01/Mechanical-Vibrations-in-SI-Units-6th-Global-Edition.pdf

MATLAB Code

All used code is available at https://github.com/leonardolombardi07/VN-Project2

main

```
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
%%%%
% Cleaning Workspace & Importing Functions
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
%%%%
close all; % Closes all figures
clear; % Clears variables from workspace
clc; % Clears all text from command window

% Importing functions from folders with given relative path
addpath(genpath("./analytical"));
addpath(genpath("./myklestad"));
addpath(genpath("./kumai"));
addpath(genpath("./ship"));


%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
%%%%
% Constants to be reused throughout this file
```

```
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
%%%%

E = 200e9; % Young Modulus [N/m^2]


%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
%%%%

% 2.1) Validation of the Mykelstad method by considering a uniform beam

%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
%%%%


% Beam constants

L_beam = 30; % Length of beam [m]

p_beam = 7850; % Density of beam [kg/m^3]

B_beam = 1.2; % Width of beam [m]

H_beam = 0.4; % Thickness of beam [m]

A_beam = B_beam * H_beam; % Sectional Area of beam [m^2]

m_beam = p_beam * A_beam * L_beam; % Mass of beam [kg]

I_beam = B_beam * (H_beam^3) / 12; % Area moment of inertia of beam
section [m^4]

EI_beam = E * I_beam; % [N.m^2]


%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
%%%%

% 2.1 a) Solve the uniform beam problem of (a) with the Mykelstad

% method. Carry out a convergence analysis. Calculate the first three

% bending natural frequencies and Eigen functions
```

```
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
%%%%


% 2.1.a - I) Solving the uniform beam problem of (a) with the Mykelstad
method

% 2.1.a - II) Convergence analysis

% 2.1.a - III) Calculating the first three bending natural frequencies
and Eigen functions


num_of_stations = 14;

num_of_fields = num_of_stations + 1;


% Field length discretization

dx = (L_beam / (num_of_fields)); % Length of Field [m]

dxs = dx * ones(num_of_fields, 1); % Field Length Vector


% EIs discretization

EIis = EI_beam * ones(num_of_fields, 1); % Lumped Stifness Vector


% Mass discretization

mi = m_beam / num_of_stations; % Mass per station [kg/st]

mis = mi * ones(num_of_stations, 1); % Lumped Masses Vector


% Vector with increasing numbers of stations, for convergence analysis

% UNCOMMENT TO RUN CONVERGENCE ANALYSIS FOR BEAM

% convergent_num_of_stations = [6, 8, 10, 12, 20];
```

```
% for i = 1:length(convergent_num_of_stations)
%     num_of_stations = convergent_num_of_stations(i);
%     num_of_fields = num_of_stations + 1;

%     % Field length discretization
%     dx = (L_beam / (num_of_fields)); % Length of Field [m]
%     dxs = dx * ones(num_of_fields, 1); % Field Length Vector

%     % EIs discretization
%     EIis = EI_beam * ones(num_of_fields, 1); % Lumped Stifness Vector

%     % Mass discretization
%     mi = m_beam / num_of_stations; % Mass per station [kg/st]
%     mis = mi * ones(num_of_stations, 1); % Lumped Masses Vector

%     [v_myklestad, wn_myklestad] = myklestad_clamped_clamped(mis,
EIis, dxs);
% end

%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
%%%%
% 2.1 b) Comparing the above results
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
%%%%
```

```
% Analytical results for comparison

% UNCOMMENT TO GET ANALYTICAL SOLUTION FOR BEAM

% x_analytical = 0:dx:L_beam;

% [Y_analytical_beam, wn_analytical_beam] = analytical(x_analytical,
EI_beam, p_beam, A_beam, L_beam);


%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
%%%%

% 2.2) Calculating the vertical flexural vibrations of a ship using

% Mykelstad method

%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
%%%%


% Ship constants

Lpp = 157.5; % Length between perpendiculars [m]

LOA = 40; % Length over all [m]

B = 22.86; % Beam [m]

D = 11; % Depth [m]

T = 8.55; % Draft [m]

displacement = 18036.9; % [t]


%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
%%%%

% 2.2 a) Calculating the first three natural frequencies and modes for
the

% non-uniform ship vertical flexural vibrations with Mykelstad method.

% Carry out a convergence analysis.
```

```
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
%%%%


% 2.2.a - I) Calculating the first three natural frequencies and modes
for the

% non-uniform ship vertical flexural vibrations with Mykelstad method

% 2.2.a - II) Convergence analysis


% UNCOMMENT TO PLOT NORMAL MODES FOR


% num_of_stations = 20;

% num_of_fields = num_of_stations - 1;


% % Field length discretization. Considering a uniform ship here

% dx = (Lpp / (num_of_fields)); % Length of Field [m]

% dxs = dx * ones(num_of_fields, 1); % Field Length Vector


% % Vector with the position of every station

% x_ship = 0:dx:Lpp;


% % EIs discretization

% EIis = get_lumped_stiffness_vector(x_ship);


% % Mass discretization

% mis = get_lumped_mass_vector(x_ship); % we consider the addded mass
here
```

```
% [v_ship, wn_ship] = myklestad_free_free(mis, EIis, dxs);

% Y1 = extract_mode_from_station_vector(v_ship{1});
% Y2 = extract_mode_from_station_vector(v_ship{2});
% Y3 = extract_mode_from_station_vector(v_ship{3});

% figure;
% subplot(3, 1, 1);
% xlabel("x [m]")
% ylabel("Y1")
% title("Mode Vector for first ship natural frequency")
% plot(x_ship, Y1);

% subplot(3, 1, 2);
% plot(x_ship, Y2);
% xlabel("x [m]")
% ylabel("Y1")
% title("Mode Vector for second ship natural frequency")

% subplot(3, 1, 3);
% plot(x_ship, Y3);
% xlabel("x [m]")
% ylabel("Y1")
% title("Mode Vector for third ship natural frequency")
```

```
% Vector with increasing numbers of stations, for convergence analysis
% UNCOMMENT TO RUN CONVERGENCE ANALYSIS FOR SHIP
% convergent_num_of_stations = [20];

% for i = 1:length(convergent_num_of_stations)
%     num_of_stations = convergent_num_of_stations(i);
%     num_of_fields = num_of_stations - 1;

%     % Field length discretization. Considering a uniform ship here
%     dx = (Lpp / (num_of_fields)); % Length of Field [m]
%     dxs = dx * ones(num_of_fields, 1); % Field Length Vector

%     % Vector with the position of every station
%     x_ship = 0:dx:Lpp;

%     % EIs discretization
%     EIis = get_lumped_stiffness_vector(x_ship);

%     % Mass discretization
%     mis = get_lumped_mass_vector(x_ship); % we consider the addded
mass here

%     [v_ship, wn_ship] = myklestad_free_free(mis, EIis, dxs);
```

```
% end


%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
%%%%

% 2.2 b) Calculating the first three natural frequencies by using a
simple

% formula

%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
%%%%

wn_kumai = kumai(B, D, T, Lpp, displacement)


%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
%%%%

% 2.1 c) Compare the estimations of natural frequency and natural modes

% from the FEM method (1.3), the results from Mykelstad method (2.2a)
and

% the empirical calculation (2.2b)

%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
%%%%
```

myklestad_clamped_clamped

```
function [v, wn] = myklestad_clamped_clamped(mi, EIi, dx)

    % Myklestad Method


%-----------------------------------------------------------------------
----

    % Iterates through lumped masses over stations and their nearby
fields,
```

```
    % relating displacement and force at the right end of a station to
those at
    % the left end of the same station to find out station vectors
    % Returns a list of station vectors (one for every station) and a
vector
    % containing the natural frequencies of the system
    %
    % Input
    % ----------
    %       [mi] :      Lumped Masses Vector
[n,1]
    %       [EIi] :     Lumped Stifness Vector
[n+1,1]
    %       [dx]:       Field Length Vector
[n+1,1]
    %
    % Output
    % ----------
    %       [v]:        Station Vectors
[n,1]
    %                    - each station vector has the form [Y, psi, M,
Q],
    %                    where "Y" stands for translational
displacement, "psi"
    %                    for angular displacement, "M" bending moment
and "Q"
    %                    shearing force
    %
```

```
    %       [wn]:       Natural Frequencies Vector
[n,1]

    %

    % Notes

    % ----------

    % For a clamped-clamped case, we have something like

    % |--FIELD0--STATION1--FIELD1--STATION2--FIELD2--|,

    % where "|" stands for edges. In this case, we have n stations and
n+1 fields



    % Declaring variables

    num_of_stations = length(mi);

    syms w; % Symbolic variable representing natural frequency



    % TODO: prealocate the cells with 4x4 matrices to improve
perfomance

    % We need to declare this variables as cell arrays because

    % this is the only way to store matrices in a MATLAB data structure

    % See:
https://www.mathworks.com/matlabcentral/answers/496101-matrix-of-matric
es-matrix-with-matrices-inside-it

    TF = {}; % Field Transfer Matrices

    TS = {}; % Station Transfer Matrices

    Ts = {}; % Transfer Matrices - cell array containing a transfer
matrix at each station



    % TODO: make this comment clearer or find a better way to access
```

```
    % data from fields
    % We start counting at i=2 because we skip the first field, which is not used
    % to calculate Ts{i}. Remember, for a clamped-clamped case, we have:
    % |--FIELD0--STATION1--FIELD1--STATION2--FIELD2--|
    % For STATION1, we calculate the transfer matrix as Ts{1} = TF{2}*Ts{1}
    % For STATION2, we calculate the transfer matrix as Ts{2} = TF{3}*Ts{2}
    % So there's no need to access FIELD0 in the loop
    for i = 2:(num_of_stations + 1)
        % Flexibility influence coefficients
        a_YM = (dx(i)^2) / (2 * EIi(i)); % displacement at i+1 due to a unit moment applied at i+1
        a_YQ = (dx(i)^3) / (3 * EIi(i)); % displacement at i+1 due to a unit force applied at i+1
        a_psiM = dx(i) / EIi(i); % slope at i+1 due to a unit moment applied at i+1
        a_psiQ = (dx(i)^2) / (2 * EIi(i)); % slope at i+1 due to a unit force applied at i+1

        % Transfer Matrix at nearby field
        TF{i} = [1 dx(i) a_YM -a_YQ / 2;
            0 1 a_psiM -a_psiQ;
            0 0 1 -dx(i);
            0 0 0 1];
```

```
        % Transfer Matrix at station

        TS{i - 1} = [1 0 0 0;

                0 1 0 0;

                0 0 1 0;

                -w^2 * mi(i - 1) 0 0 1];


        % Transfer matrix relating station vector on left side of
station i+1

        % to station vector on the left side of station i

        Ts{i - 1} = TF{i} * TS{i - 1};

    end



%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
%%%%

    % CALCULATING OVERALL TRANSFER MATRIX (T)


%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
%%%%



    % TODO: abstract method to calculate a field matrix

    % so we don't repeat this here and inside the loop

    % Initial Field Transfer Matrix

    a_YM = (dx(1)^2) / (2 * EIi(1));

    a_YQ = (dx(1)^3) / (3 * EIi(1));

    a_psiM = dx(1) / EIi(1);

    a_psiQ = (dx(1)^2) / (2 * EIi(1));
```

```
    TF{1} = [1 dx(1) a_YM -a_YQ / 2;

        0 1 a_psiM -a_psiQ;

        0 0 1 -dx(1);

        0 0 0 1]; % Field Matrix at first field


    % Overall Transfer Matrix (T = Ts{n}*Ts{n-1}*...Ts{2}*Ts{1}*TF{1})

    T = TF{1};


    for i = 1:length(Ts)

        T = Ts{i} * T;

    end



%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
%%%%

    % CALCULATING NATURAL FREQUENCIES (wn)

%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
%%%%


    % This is found by carefull analysis of vRn = T*vL0

    % (where vRn is the station vector at the end edge

    % and vL0 is the station vector at the start edge)

    % Knowing that, for a clamped edge, Y=0, psi=0, M≠0, and Q≠0

    % we find that the determinant below must be equal to 0

    symbolic_wn = solve(det([T(1, 3) T(1, 4);

                        T(2, 3) T(2, 4)]) == 0, w);
```

```
    % Converting the natural frequencies from symbolic to numeric
values

    all_wn = double(subs(symbolic_wn)); % This can contain negative
numbers

    wn = sort(all_wn(all_wn > 0)); % Keep only positive numbers, sorted
from lowest to highest



%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
%%%%

    % CALCULATING STATE VECTORS (v) FOR MULTIPLE MODES

%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
%%%%


    % TODO: this can be a parameter of the function

    % or even selecting exactly what modes the user wants

    max_num_of_modes = 3;

    v = {}; % TODO: prealocate the cells with arrays to improve
performance


    for i = 1:max_num_of_modes

        % Replacing the symbolic natural frequency with the calculated
numeric

        % natural frequency for the overall matrix

        T = double(subs(T, w, wn(i)));
```

```
        % We'll have one station vector for every station + 2 station
vectors:
        % one at the start/left edge, and one at the end/right edge
        % If we have: |--FIELD0--STATION1--FIELD1--STATION2--FIELD2--|
        % We get:     v0-------v1-----------------v2----------------v3

        % Preallocating the variable that will contain the station
vectors
        % with 4x1 vectors

        v{i} = {};

        for j = 1:(num_of_stations + 2)
            v{i}{j} = zeros(4, 1);
        end

        % TODO: determine M0 and Q0 in the right way. How to arbitrate
M0, instead of 1?
        Y0 = 0; psi0 = 0; M0 = 1; Q0 = -T(1, 3) * M0 / T(1, 4);
        v{i}{1} = [Y0; psi0; M0; Q0]; % station vector at start/left
edge (wall)
        v{i}{2} = TF{1} * v{i}{1}; % station vector at first station

        for j = 3:(length(v{i}) - 1)
            % Creating a transfer matrix at station with symbolic
natural frequency
            % converted to numeric natural frequency
```

```
            Ts_iminus2 = double(subs(Ts{j - 2}, w, wn(i)));

            v{i}{j} = Ts_iminus2 * v{i}{j - 1};

        end


        v{i}{length(v)} = T * v{i}{1}; % station vector at end/right
edge (wall)

    end


end
```

myklestad_free_free

```
function [v, wn] = myklestad_free_free(mi, EIi, dx)

    % Myklestad Method


%----------------------------------------------------------------------
----

    % Iterates through lumped masses over stations and their nearby
fields,

    % relating displacement and force at the right end of a station to
those at

    % the left end of the same station to find out station vectors

    % Returns a list of station vectors (one for every station) and a
vector

    % containing the natural frequencies of the system

    %

    % Input
```

```
    % ----------
    %       [mi] :      Lumped Masses Vector
[n,1]
    %       [EIi] :     Lumped Stifness Vector
[n-1,1]
    %       [dx]:       Field Length Vector
[n-1,1]
    %
    % Output
    % ----------
    %       [v]:        Station Vectors
[n,1]
    %                    - each station vector has the form [Y, psi, M,
Q],
    %                    where "Y" stands for translational
displacement, "psi"
    %                    for angular displacement, "M" bending moment
and "Q"
    %                    shearing force
    %
    %       [wn]:       Natural Frequencies Vector
[n,1]
    %
    % Notes
    % ----------
    % For a free-free case, we have something like
    % STATION1--FIELD1--STATION2--FIELD2--STATION3
    % where "|" stands for edges. In this case, we have n stations and
n-1 fields
```

```
    % Declaring variables

    num_of_stations = length(mi);

    syms w; % Symbolic variable representing natural frequency


    % TODO: prealocate the cells with 4x4 matrices to improve
perfomance

    % We need to declare this variables as cell arrays because

    % this is the only way to store matrices in a MATLAB data structure

    % See:
https://www.mathworks.com/matlabcentral/answers/496101-matrix-of-matric
es-matrix-with-matrices-inside-it

    TF = {}; % Field Transfer Matrices

    TS = {}; % Station Transfer Matrices

    Ts = {}; % Transfer Matrices - cell array containing a transfer
matrix at each station


    for i = 1:(num_of_stations - 1)

        % Flexibility influence coefficients

        a_YM = (dx(i)^2) / (2 * EIi(i)); % displacement at i+1 due to a
unit moment applied at i+1

        a_YQ = (dx(i)^3) / (3 * EIi(i)); % displacement at i+1 due to a
unit force applied at i+1

        a_psiM = dx(i) / EIi(i); % slope at i+1 due to a unit moment
applied at i+1

        a_psiQ = (dx(i)^2) / (2 * EIi(i)); % slope at i+1 due to a unit
force applied at i+1
```

```
        % Transfer Matrix at nearby field

        TF{i} = [1 dx(i) a_YM -a_YQ / 2;

            0 1 a_psiM -a_psiQ;

            0 0 1 -dx(i);

            0 0 0 1];


        % Transfer Matrix at station

        TS{i} = [1 0 0 0;

            0 1 0 0;

            0 0 1 0;

            -w^2 * mi(i) 0 0 1];


        % Transfer matrix relating station vector on left side of
station i+1

        % to station vector on the left side of station i

        Ts{i} = TF{i} * TS{i};

    end


%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
%%%%

    % CALCULATING OVERALL TRANSFER MATRIX (T)

%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
%%%%


    % Overall Transfer Matrix (T = TS{n}*Ts{n-1}*...Ts{2}*Ts{1})
```

```
    T = Ts{1};


    for i = 2:(num_of_stations - 1)

        T = Ts{i} * T;

    end


    % TODO: abstract method to calculate a station matrix

    % so we don't repeat this here and inside the loop

    % Final Station Transfer Matrix (TS{n})

    TS{num_of_stations} = [1 0 0 0;

                           0 1 0 0;

                           0 0 1 0;

                           -w^2 * mi(num_of_stations) 0 0 1];


    T = TS{num_of_stations} * T;



%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
%%%%

    % CALCULATING NATURAL FREQUENCIES (wn)


%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
%%%%


    % This is found by carefull analysis of vRn = T*vL0

    % (where vRn is the station vector at the end edge

    % and vL0 is the station vector at the start edge)
```

```
    % Knowing that, for a free edge, Y≠0, psi≠0, M=0, and Q=0

    % we find that the determinant below must be equal to 0

    symbolic_wn = solve(det([T(3, 1) T(3, 2);

                        T(4, 1) T(4, 2)]) == 0, w);



    % Converting the natural frequencies from symbolic to numeric
values

    all_wn = double(subs(symbolic_wn)); % This can contain negative
numbers

    wn = sort(all_wn(all_wn > 0)); % Keep only positive numbers and
sorted from lowest to highest



%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
%%%%

    % CALCULATING STATE VECTORS (v) FOR MULTIPLE MODES


%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
%%%%



    % TODO: this can be a parameter of the function

    % or even selecting exactly what modes the user wants

    max_num_of_modes = 3;

    v = {}; % TODO: prealocate the cells with arrays to improve
performance



    for i = 1:max_num_of_modes

        % Replacing the symbolic natural frequency with the calculated
numeric
```

```
        % natural frequency for the overall matrix
        T = double(subs(T, w, wn(i)));


        % We'll have one station vector for every station:
        % If we have: STATION1--FIELD1--STATION2--FIELD2--STATION3
        % We get:     v1----------------v2----------------v3


        % Preallocating the variable that will contain the station
vectors
        % with 4x1 vectors
        v{i} = {};


        for j = 1:(num_of_stations)
            v{i}{j} = zeros(4, 1);
        end


        % TODO: determine Y0 and psi0 in the right way. How to
arbitrate psi0, instead of 1?
        psi0 = 1; Y0 = -T(3, 2) * psi0 / T(3, 1); M0 = 0; Q0 = 0;
        v{i}{1} = [Y0; psi0; M0; Q0]; % station vector at start/left
edge (wall)


        for j = 2:(length(v{i}))
            % Creating a transfer matrix at station with symbolic
natural frequency
            % converted to numeric natural frequency
            Ts_iminus1 = double(subs(Ts{j - 1}, w, wn(i)));
```

```
            v{i}{j} = Ts_iminus1 * v{i}{j - 1};

        end


    end


end
```

get_lumped_mass_vector

```
function [m] = get_lumped_mass_vector(x)

    % x is a vector with the position of each station

    % mi is the vector with corresponding lumped masses for each
station

    % we consider the added mass here



    % Position of stations

    x_of_stations = [0; 7.88; 15.75; 23.63; 31.50; 39.38; 47.25; 55.13;
63.00; 70.88; ...

                    78.75; 86.83; 94.50; 102.38; 110.25; 118.13;
126.00; 133.88; 141.75; ...

                    149.63; 157.50];



    % Mass distribution [kg/m]

    % From table on project assignment

    % W_table_data = 1000 * [17; 38; 59; 73; 88; 103; 117; 132; 146;
146; 146; 146; 146; ...

    %                         135; 123; 111; 99; 88; 76; 51; 27];

    W_table_data = 1000 * [17; 38; 59; 73; 88; 103; 117; 132; 146; 146;
146; 146; 146; ...

                            135; 123; 111; 99; 88; 76; 51; 27];



    % Result with ANSYS for j=2

    % Ideally, we should use this:

    % get_added_mass = @(x) (pi / 8) * p * B(x) * C(x) * Jn;

    % and consider the right B(x), C(x) and Jn for each case
```

```
    W_added_mass_data = 1000 * [0.91; 28.35; 57.76; 91.25; 124.30;
159.33; 186.61; 198.36; ...
                                211.87; 222.42; 222.42; 203.31; 172.46;
145; 107.19; ...
                                67.15; 37.70; 17.61; 5.75; 2.00; 0];

    W_total_data = W_table_data + W_added_mass_data;

    % Regression to obtain the polynomial responsible for describing
weigth per length against x
    W = fit(x_of_stations, W_table_data, 'linearinterp'); % Weight per
length [kg/m]

    m = zeros(length(x), 1);

    for i = 1:(length(x) - 1)
        x_left = x(i); x_right = x(i + 1);
        dx = x_right - x_left;

        xx = [x_left, x_right];
        M = trapz(xx, [W(x_left), W(x_right)]);
        x_CG = (dx / 3) * ((2 * W(x_right) + W(x_left)) / (W(x_right) +
W(x_left)));

        m_left = M * (x_right + 2 * x_left - x_CG) / (x_right +
x_left);
        m_right = M * (x_CG - x_left) / (x_right + x_left);
```

```
        m(i) = m(i) + m_left;

        m(i + 1) = m_right;

    end


end
```

get_lumped_stiffness_vector

```
function [EIs] = get_lumped_stiffness_vector(x)

    % x is a vector with the position of each station

    % EIs is the vector with corresponding lumped bending stifness for
each station


    % Position of stations [m]

    x_of_stations = [0; 18; 36; 54; 108; 144; 157.5];


    % Bending Stiffness distribution [N.m^2]

    EI_data = (10^9) * [3543; 5905; 7086; 7676; 7676; 6495; ...

                        4724];


    % Regression to obtain the polynomial responsible for describing

    % the bending stiffness at any x

    EI_fit = fit(x_of_stations, EI_data, 'linearinterp');
```

```
    EIs = zeros(length(x), 1);


    for i = 1:(length(x))
        EIs(i) = EI_fit(x(i));
    end


end
```

extract_mode_from_station_vector

```
function [Y] = extract_mode_from_station_vector(v)
    Y = zeros(length(v), 1);


    for i = 1:length(v)
        Y(i) = v{i}(1);
    end


end
```

kumai

```
function [wn] = kumai(B, D, T, Lpp, displacement)
```

```
    % Calculate the natural frequencies by using the simple Kumai's
formula

    % B = Beam

    % D = Depth

    % T = Draft

    % Lpp = Length between perpendiculars

    % displacement = displacement of ship


    % Area moment of inertia of ship's midsection [m^4]

    % Depth Kumai correction

    D = D * 0.5;

    I = B * (D^3) / 12;


    % Displacement including virtual mass [t]

    displacement_with_vmass = (1.2 + B / (3 * T)) * displacement;


    % First natural frequency [hz]

    wn(1) = (1/60) * (3.07 * 10^6) * sqrt(I / (displacement_with_vmass
* Lpp^3));


    num_of_natural_modes = 3;

    alpha = 1.02; % considering a tankero ship


    for n = 3:(num_of_natural_modes + 1)

        wn(n - 1) = wn(1) * (n - 1)^alpha;

    end
```

```
end
```

analytical

```
function [Y, wn] = analytical(x, EI, p, A, L)

    beta1_times_L = 4.7300041;

    beta2_times_L = 7.853205;

    beta3_times_L = 10.995608;

    beta1 = (beta1_times_L / L);

    beta2 = (beta2_times_L / L);

    beta3 = (beta3_times_L / L);


    %  Calculating first three natural frequencies

    square_root = sqrt(EI / (p * A * (L^4)));

    wn = [(beta1_times_L^2) * square_root,

        (beta2_times_L^2) * square_root,

        (beta3_times_L^2) * square_root];


    %  Calculating first three eigenfucntions/mode shapes

    function [Y] = mode_shape(x, beta, L)

        Cn = 1;

        alfa_n = (sinh(beta * L) - sin(beta * L)) / (cos(beta * L) -
cosh(beta * L));
```

```
        Y = Cn * (sinh(beta * x) - sin(beta * x)) + alfa_n * (cosh(beta * x) - cos(beta * x));
    end

    Y = [mode_shape(x, beta1, L),
        mode_shape(x, beta2, L),
        mode_shape(x, beta3, L)];
end
```