26th International Congress of Mechanical Engineering
COBEM 2021
ABCM Associação Brasileira de Engenharia e Ciências Mecânicas
November 22-26, 2021, Virtual Congress, Brazil

COBEM2021-1832 VIBRATION CHARACTERIZATION OF BEAMS RIVETED LAP JOINT USING SPECTRAL ELEMENT METHOD

Amanda Aryda Silva Rodrigues de Sousa
Post-Graduate Program - Integrity of Engineering Materials, University of Brasília. Campus Universitário Darcy Ribeiro, Faculdade de Tecnologia - Asa Norte, Brasília - DF, 70910-900
Jefferson da Silva Coelho
Department of Mechanical Engineering, University of Brasília. Campus Universitário Darcy Ribeiro, Faculdade de Tecnologia - Asa Norte, Brasília - DF, 70910-900
Marcela Rodrigues Machado
Department of Mechanical Engineering, University of Brasília. Campus Universitário Darcy Ribeiro, Faculdade de Tecnologia - Asa Norte, Brasília - DF, 70910-900
Ariosto Bretanha Jorge
Post-Graduate Program - Integrity of Engineering Materials, University of Brasilia, Brazil
eng.amandaaryda@gmail.com, jeffersoncoelho@ufam.edu.br, marcelam@unb.br, ariosto.b.jorge@gmail.com

Abstract. *Mechanical structures are commonly jointed using fasteners such as rivets or bolts arranged in various configurations depending on the required performance. They are an important part of the system and can influence its dynamic response. In this paper, two beams are connected in a single row rivet lap joint and modelled using the spectral element method (SEM). The equivalent spectral model represents the rivet joint and the beams. The identification of dynamic characteristics of the system is validated by comparing the results to the commercial finite element- ANSYS. It is found that the vibration responses obtained with SEM are in good agreement with the dynamic responses estimated via FEM. The advantage of using SEM are the reduced number of elements in the mesh and high accuracy in the vibration response prediction.*

Keywords: *Vibration characterization, Rivet lap joint, Spectral element method, Finite element method*

1. INTRODUCTION

The dynamic behaviour of structures and components is considered major importance to engineering, which depends on several factors like shape, loads e geometrics and mechanics properties. Therefore, a dynamic structural characterization in the design intends to avoid excessive vibrations and reduce the system's useful life. Mechanical structural components are commonly jointed using fasteners such as rivets or bolts arranged in various configurations depending on the required performance. They are an important part of the system and can influence dynamic response inducing local flexibility e damping (Ahmadian and Jalali., 2007). The rivets are largely used in several industries, including aerospace, naval construction and civil construction, and aircraft components. Also in the automotive industry, the saelf-piercing riveting (SPR) are largely used to assembly components and construct of vehicle bodies. Identifying this joint's parameters is critical once an excessive number of rivets is present in such a structure. Aside from identifying to model the fasteners is a challenge because of their non-linear behaviour.

The exposure of structures to a certain environment can lead to problems in the rivets resistance loss as the heat fluctuations and vibrations changes in their operation while at the same time induce weakening or corrosion, causing an abrupt failure on the rivets (Wang *et al.*, 2018). The finite element method (FEM) is a common method used to numerical simulate structures and systems. However, for some system and analysis, FEM will demand time-consuming and demands large computers memory and processor. Dourado and Meireles (2016) used to model the riveted joints connecting two aluminium plates as a spring-damper. The authors used FEM to the numerical simulation and validate the results by comparing them to experimental data. Abulfaiz and Shelke (2018) studied the vibration analysis of rivets overlapped joints to solve the geometric influence pattern problem of the four rivets on the damping riveted joints. They used CATIA to model the different riveting patterns and ANSYS to analyse the vibration. He *et al.* (2007) discussed the theoretical investigation using FEM to dimensionalize to beams assembled by the components parts through the joints and presented a vibration characterizing. They found out that the natural system frequencies increased for a single joint because of the rigidity up a level, but small changes happened in the Poisson coefficient. J. *et al.* (2009) addressed a finite element analysis in airships contain rivet joints. Bagale *et al.* (2019) used FEM to analyze the rivets joint with several diameters for all the materials and tension.

Aside from FEM, the spectral element method (SEM) is the easy tool with a direct access to the matrix and general formulation, reduced number of elements in the mesh propitiating inclusion of uncertainties in parameters and non-linearities without losing accuracy and low computational cost. SEM is a mesh method similar to FEM, where the functions of approximate forms of the element are replaced by functions of the exact solution of differential equations of government. Therefore, a single element is enough to model any continuous and uniform part of the structure. This feature significantly reduces the number of elements required in the structure model and improves the accuracy of the dynamic system solution. An extensive study of the fundamentals and a variety of new applications of SEM, such as composite laminates, periodic structure, damage detection was presented in (Lee, 2009; Doyle, 1997) and the behaviour of the waves in a conductor cable of transmission line in (Dutkiewicz and Machado, 2019a,b; Machado *et al.*, 2020). Lee (2001a) presented the model of spectral bolt applied in a cantilevered beam structure with a bolt-joint. Choi and Inman (2014) modelled a cable-harnessed structure by means of SEM, where the used a double beam connected by springs to simulate a cable-harnessed structure.

This work explores the numerical model via SEM and FEM of two Euler–Bernoulli beam connected with one and three rivet in the mid-span. The models were performed using the FEM analysis via ANSYS-Workbench. SEM model employed the use of a beam spectral element and the spectral rivet element. The solution provides the frequency response function of the connected system and its modal parameters. The resonance frequencies were compared with the analytical solution of a continuous beam.

2. FINITE ELEMENT BACKGROUND

The finite element method is a numerical tool that uses variational formulations and uses interpolation methods to solve difficult analytical formulation (Petyt, 2010). The concept of the method is to solve a problem of balance where a complex global structure is divided into partitions, called elements. The elements are then connected, where boundary conditions of movement or rotation restriction and loading can be applied. By solving the set employing numerical methods of approximation of result. The finite element formulation is, to begin with, a variational principle related to total potential energy as follows.

$$\int_V \mathbf{B}^T\mathbf{D}\mathbf{B}dV + \int_V \mathbf{N}^T\rho\mathbf{N}dV - \int_x \mathbf{N}\mathbf{P}_x dx = 0 \tag{1}$$

where $\mathbf{N}$ is matrix of shape functions,$\mathbf{B}$ is the strain matrix, $\mathbf{D}$ is the matrix of material constants, and $\mathbf{P}_x$ vector of distributed load. Equation 1 is the basic equation for the finite element discretization and can be converted to algebraic equations of motion in the dynamic case as

$$(\mathbf{K} - \omega_i^2\mathbf{M})\mathbf{u}_i = \mathbf{f} \tag{2}$$

For the static approach, eq (1) and (2) reduce to the expressions

$$\int_V \mathbf{B}^T\mathbf{D}\mathbf{B}dV - \int_x \mathbf{N}\mathbf{P}_x dx = 0 \tag{3}$$

$$\text{and} \qquad \mathbf{K}\mathbf{u}_i = \mathbf{f} \tag{4}$$

2.1 ANSYS Mechanical simulation

In general, the analysis process in ANSYS is divided into three main steps: pre-processing, processing and post-processing. The pre-processing step includes defining the types of elements, the system mesh, assigning material properties to each element, defining boundary conditions and loads. Understanding the physical principles behind the problem is always more important than the modelling technique itself. If the boundary conditions are incorrect or the element type is not chosen correctly, the results may be incorrect. The processing step is where the MEF is applied computationally in the model made previously. The pre-processed model is imported into a solver that will then assemble the stiffness matrix and calculate all degrees of freedom (GDLs) and solve the problem of extracting the result given the choice analysis.

Modal analysis is used to determine the vibration characteristics (natural frequencies and mode shapes) of a structure or a machine component while it is being designed. Can also serve as a starting point for another, more detailed, dynamic analysis, such as transient dynamic analysis, a harmonic response analysis, or a spectrum analysis (ANSYS, 2009b). The natural frequencies and mode shapes are important parameters in the design of a structure for dynamic loading conditions. They are also required to do a spectrum analysis or a mode superposition harmonic or transient analysis.(ANSYS, 2009b). There are a great number of numerical methods for extracting the eigenvalues and eigenvectors. The main ones are Block Lanczos, PCG Lanczos, Supernode, Reduction (Householder), as presented by (ANSYS, 2009a).

In this work, the Block Lanczos extraction model was used, which is an acceptable eigenvalue extraction method for problems with a high number of nodes and for presenting a good convergence rate when applied to problems with symmetric matrices. Harmonic analysis is used to determine the response of a structure to time-varying cyclic loads. Therefore, gives youthe ability to predict the sustained dynamic behavior of your structures, thus enabling us to verify whether your designswill successfully overcome resonance, fatigue, and other harmful effects of forced vibrations. Analyzes can generate plots of displacement amplitudes at given points in the structure as a function of forcing frequency.

3. SPECTRAL ELEMENT METHOD

In dynamic system analysis and SHM, it is crucial to have an efficient and economic numerical technique. Finite Element Method (FEM) is one of the most common computational methods employed in several science areas. However, in medium and high-frequency wave propagation problems, this method requires a too high computational cost. The SEM was first proposed by Narayanan and Beskos in 1978 Narayanan and Beskos (1978), further improved and named SEM by DoyleDoyle (1997) and Lee Lee (2009). The SEM consists of the exact displacement of the wave equation of the analytical solution in the frequency domain. It is equivalent to an infinite number of finite elements. This characteristic and the spectral domain make SEM more suitable to solve the crack problem. The advantage of SEM is the reduced number of elements required to model the system as compared to other computational methods, as demonstrated in Figure 1.

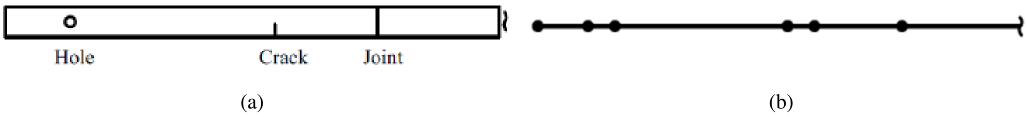


Figure 1: Representation: (a) Physical structure; (b) Spectral element model.

SEM is similar in style to the FEM, it is written in the frequency domain, and the element interpolation function is the exact analytical solution of the differential equation. These futures allows a no mesh element requirement associated with high accuracy in solve structural dynamic problems. The number of elements required for a spectral model will coincide with the number of discontinuities of the structure. Another significant advantage of using SEM is the throw-off element that consists of conduct to propagate energy out of the system, it works as an anechoic termination dissipating the remaining energy in the system.

3.1 Beam spectral element

The beam is assumed as slender with transversal and rotational nodal displacement, shear and momentum nodal forces. By neglecting shear deformations, the differential equation of movement in its spectral form can be written as

$$\frac{d^4\hat{v}}{dx^4} - k^4\hat{v} = F, \tag{5}$$

with the homogeneous solution given by

$$\hat{v}(x,\omega) = a_1e^{-ikx} + a_2e^{-kx} + a_3e^{-ik(L-x)} + a_4e^{-k(L-x)}, \tag{6}$$

where

$$\mathbf{e}(x,\omega) = \begin{bmatrix} e^{-ikx} & e^{-kx} & e^{-ik(L-x)} & e^{-k(L-x)} \end{bmatrix},$$

$$\mathbf{a} = \begin{bmatrix} a_1 & a_2 & a_3 & a_4 \end{bmatrix}^T,$$

for L being the beam length. The wavenumbers, k, k_1 and k_2 are given by

$$k^2 \equiv \sqrt{\frac{\omega^2\rho A}{EI}}, \quad k_1 = \pm k, \quad k_2 = \pm ik, \tag{7}$$

where ω is the circular frequency, E is the Young's modulus, A is the cross-section area, ρ is the density, I is the inertia moment, and $i = \sqrt{-1}$. By using a complex Young's modulus, $E_c = E(1 + i\eta)$, a internal structural damping is introduced where η is the hysteretic structural loss factor. Figure 2 illustrates a two nodes healthy beam spectral element model with two degrees of freedom (dof) per nodes. The nodal displacements are $\hat{v}$ and $\hat{\phi}$ and the nodal forces $\hat{V}$ and $\hat{M}$.

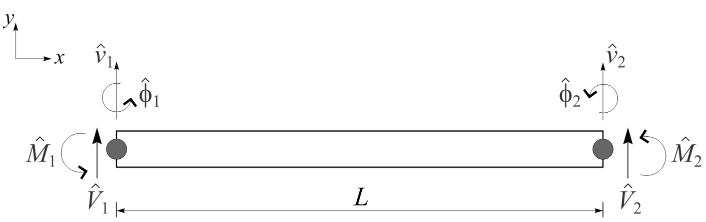


Figure 2: Two nodes beam spectral element.

The spectral nodal displacements and slopes of the finite beam element can be related to the displacement field as at node 1 ($x = 0$) and at node 2 ($x = L$)

$$d = \left\{ \begin{array}{c} \hat{v_1} \\ \hat{\phi_1} \\ \hat{v_2} \\ \hat{\phi_2} \end{array} \right\} = \left\{ \begin{array}{c} \hat{v}(0) \\ \hat{v}\prime(0) \\ \hat{v}(L) \\ \hat{v}\prime(L) \end{array} \right\} = \left\{ \begin{array}{c} e(0,\omega) \\ e'(0,\omega) \\ e(L,\omega) \\ e'(L,\omega) \end{array} \right\}, \tag{8}$$

where $\mathbf{a} = \mathbf{H_B}(\omega)^{-1}\mathbf{d}$, and

$$\mathbf{H_B}(\omega) = \begin{bmatrix} 1 & 1 & e^{-ikL} & e^{-kL} \\ -ik & -k & ike^{-ikL} & ke^{-kL} \\ e^{-ikL} & e^{-kL} & 1 & 1 \\ -ike^{-ikL} & -ke^{-kL} & ik & k \end{bmatrix}.$$

The frequency-dependent displacement within an element is interpolated from the nodal displacement vector d by eliminating the constant vector, it is expressed as

$$\hat{v} = e(x,\omega)H_B^{-1}(\omega)\mathbf{d}. \tag{9}$$

Shear forces and bending moments defined for the beam is related to the defined forces and moments in a spectral nodal form as

$$f = \left\{ \begin{array}{c} \hat{V_1} \\ \hat{M_1} \\ \hat{V_2} \\ \hat{M_2} \end{array} \right\} = \left\{ \begin{array}{c} -V(0) \\ -M(0) \\ V(L) \\ M(L) \end{array} \right\} = \left\{ \begin{array}{c} -\hat{v}(0)''' \\ -\hat{v}(0)'' \\ \hat{v}(L)''' \\ \hat{v}(L)'' \end{array} \right\}. \tag{10}$$

where by applying boundary conditions it has,

$$f = EI \left\{ \begin{array}{cccc} -ik^3 & k^3 & ie^{-ikL}k^3 & e^{-kL}k^3 \\ k^2 & -k^2 & e^{-ikL}k^2 & -e^{kL}k^2 \\ ie^{-ikL}k^3 & -e^{kL}k^3 & -ik^3 & k^3 \\ -e^{-ikL}k^2 & e^{-kL}k^2 & -k^2 & k^2 \end{array} \right\} \mathbf{a} = G(\omega)\mathbf{a}. \tag{11}$$

By relating the nodal forces to the nodal displacement, one has

$$\mathbf{f} = \mathbf{G}(\omega)\mathbf{H}_B^{-1}(\omega)\mathbf{d} = \mathbf{S}_B(\omega)\mathbf{d} \tag{12}$$

where $\mathbf{S}_B(\omega) = \mathbf{G}(\omega)\mathbf{H}_B^{-1}(\omega)$ is the dynamic stiffness matrix of the Euler-Bernoulli beam spectral element.

3.2 Beam spectral element connected by rivet

The spectral beam element connected by a rivet considered two overlapped beams jointed by a rivet as illustrated in Fig.3(a), and the spectral revit model (Lee, 2001b; Machado *et al.*, 2019) is equivalent to a lumped mass and a spring system. The lumped mass has its mass m and the spring consists of a linear spring kv, as shown in Fig.3(b). The lumped mass and the spring-system are connected in series.

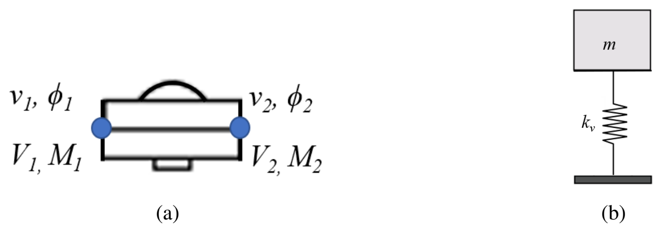


Figure 3: Representation: (a) Physical structure; (b) Spectral element model.

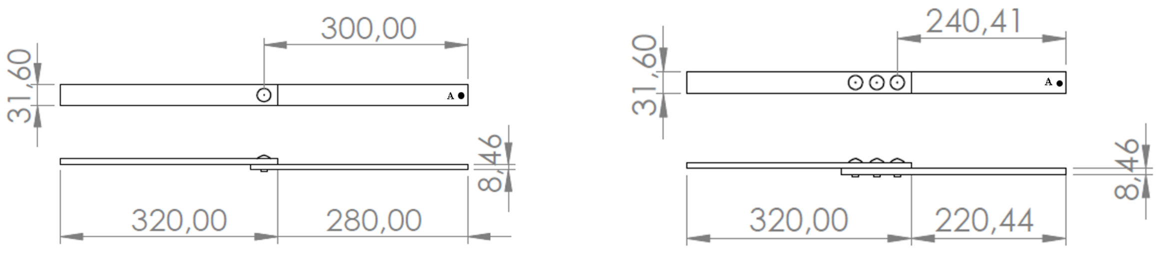
The symmetric dynamic spectral element matrix or the equivalent rivet-joint model, is expressed by (Lee, 2001b),

$$\mathbf{S}_b(\omega) = \begin{bmatrix} k_v & 0 & -k_v & 0 \\ 0 & 0 & 0 & 0 \\ -k_v & 0 & -m\omega^2 + k_v & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix}. \tag{13}$$

where ω is the circular frequency. One should note that the spectral element matrix $\mathbf{S}_b(\omega)$ includes of equivalent rivet-joint model parameters. Analogous to FEM, the SEM can be assembled to form a global structure matrix system.

4. NUMERICAL RESULTS

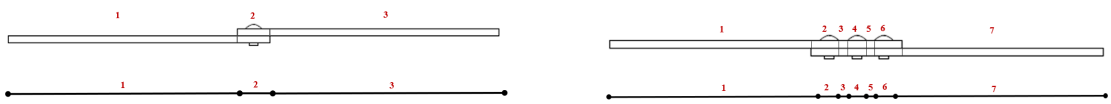
In the numerical analysis assumed two beams with a width of 31.6 mm, a thickness of 8.46 mm, and a length of 320 mm, joined with one and three rebits. The mechanical properties of the beams and rivets are Young's modulus of 200 GPa, density of 7850 Kgf/m^3, and Poisson's ratio of 0.3. Receptance responses and modal parameter were estimated using SEM and FEM. The structure was modelled in SolidWorks and used ANSYS WORKBENCH software to perform a modal analysis based on the finite element method. Figure 4(a-b) shows a 3D model of beams jointed by one and three revits and their dimensions. The structure is considered in clamped-clamped boundary condition and excited with a unitary force applied in in point A the beam.



(a) (b)

Figure 4: Draft of the beams assembly by (a) one rivet and (b) three rivets.

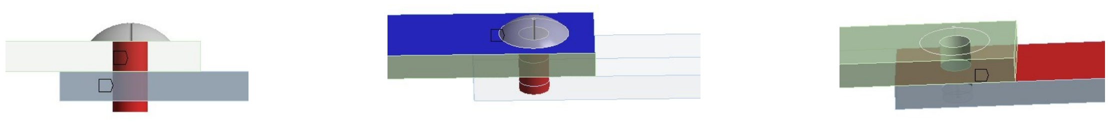
In the case with one rivet, assumed three spectral elements in the mesh in being a spectral rivet element (element 2) and two beam elements as shown in Fig. 5(a). The case with three rivets used three spectral rivet elements (elements 2, 4, and 6) and four spectral beams (elements 1, 3, 5, and 7) in total 7 elements. Elements 3 and 5 of the case with three rivets have a thickness of 16.92 mm. Otherwise, all beam elements are similar to the mechanical and geometrical parameters specified above. The rivet element has $k_v = 8.7266x10^6$ N/m, $m = 0.0185$ Kgf/m^3 and $I_{bolt} = 4.9087x10{-}10$.



(a) (b)

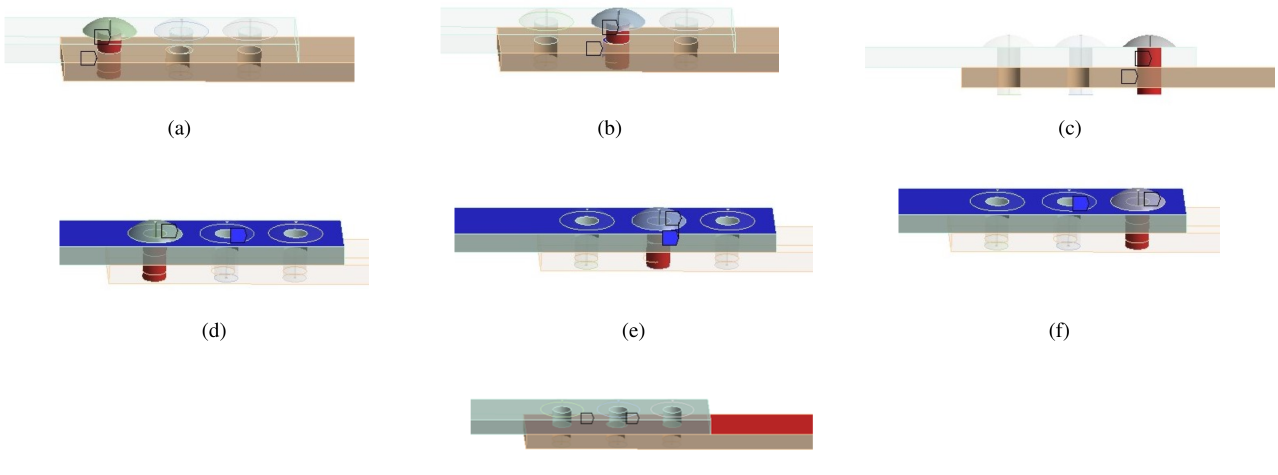
Figure 5: Spectral beams assembly by (a) one rivet and (b) three rivets.

In ANSYS-Workbench, the structure is modelled using SOLID186 three-dimensional elements. The element is defined by twenty nodes and each with three degrees of freedom, which exhibits quadratic displacement behaviour. In addition, a surface-to-surface contact element, which corresponds to contact elements (CONTAC174) and target segment elements (TARGE170), is used at the interfaces between the rivet head and the beam. In the interface, we used a tool is known as a fixed contact to decrease the preload of the rivet. Depending on the amount of preload, one would expect non-linear dynamic effects. A contact modelling can be performed by point-to-point, point-to-surface, or surface-to-surface elements. Figure 6 shows contact made between the beams and the rivet for the case with one rivet, and Fig. 7 shows the contact made between the three rivets. In this rivet model, virtual thermal strain application methods are employed to apply contact force on the rivet.



(a) (b) (c)

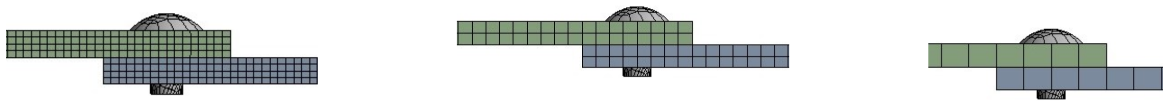
Figure 6: Representation: (a) Physical structure; (b) Spectral element model.



(a) (b) (c) (d) (e) (f) (g)

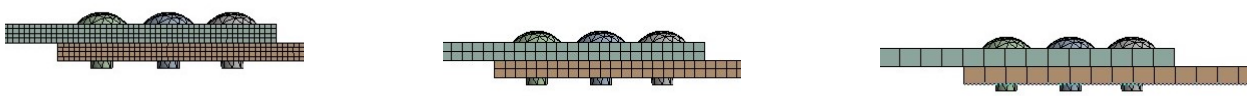
Figure 7: Representation: (a) Physical structure; (b) Spectral element model.

The element sizing is important to securer a good accuracy of the response maintains the optimal computational time. Elements have a sizing 2.5, 5, and 10mm, were tested in the mesh to demonstrate the convergence. Figure 8 shows the mesh density for one rivet, and Fig. 8 the mesh density for the three rivet. This criterion maintains a greater uniformity in the mesh distribution process during the calculation for the various models.



(a) (b) (c)

Figure 8: Mesh density for one rivet case: (a) 2.5mm; (b)5.0mm; (c)10.0mm.



(a) (b) (c)

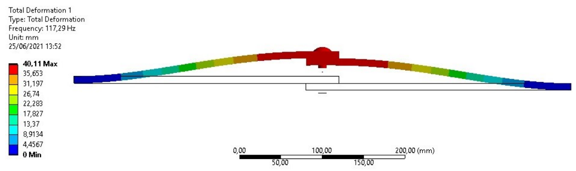
Figure 9: Mesh density for three rivet case: (a) 2.5mm; (b)5.0mm; (c)10.0mm.

Table 1 displayed discretization mesh used in FEM and SEM together with computational time, number of nodes and elements in both cases. The computational CPU times was acquired in an Intel(R) Core(TM) i5-7200U machine. In both cases, SEM had better performance reducing the computational time by 85% compared to the fast time of FEM. This study has shown that the mesh density of case A, B and C presented close modal parameters demonstrating the mesh convergence. Hence, the dynamic analysis assumed meshing of case C and explored the modal parameters (natural frequencies and modal shape) and receptance response of the clamped-clamped beams connected by the rivet.

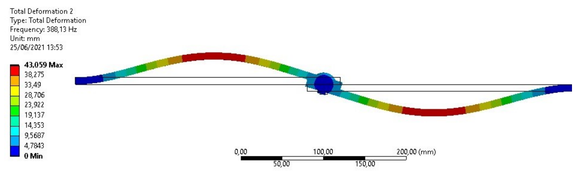
Table 1: Meshing comparison.

		FEM Case A	FEM Case B	FEM Case C	SEM
Beams joined by a rivet	Elapsed CPU time [s]	40.900	10.000	9.000	1.229
	No. of nodes	67792	15512	6904	4
	No. of elements	16459	4071	2513	3
Beams joined by three rivets	Elapsed CPU time [s]	631.000	11.000	16.000	2.663
	No. of nodes	73997	17397	8715	8
	No. of elements	21922	7314	5112	7

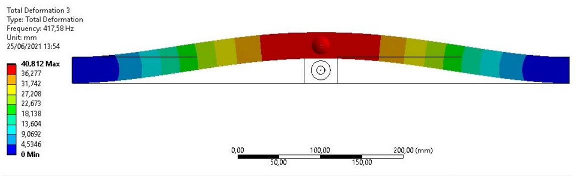
The first six mode shapes and natural frequencies of the beams connected by one rivet are shown in Fig. 10, and modal parameters of the beams connected by three rivets shown in Fig. 11. Modal parameters were extracted by using ANSYS-Block Lanczos method, which is suggested for solving large symmetric eigenvalue problems since has a fast convergence rate.



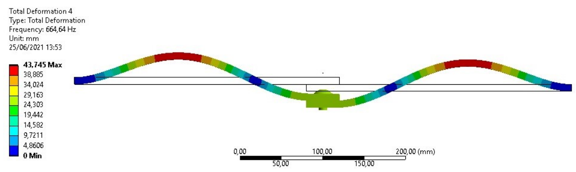
(a) 1st mode at 117.29 Hz.



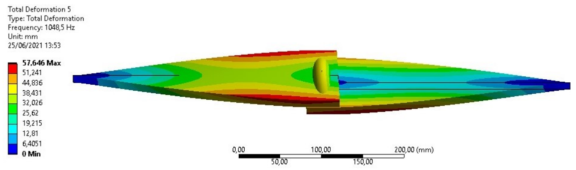
(b) 2nd mmode at 388.13 Hz.



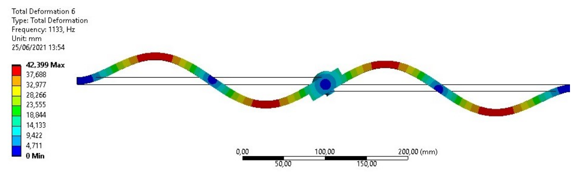
(c) 3rd mode at 417.58 Hz.



(d) 4th mode at 664.64 Hz.



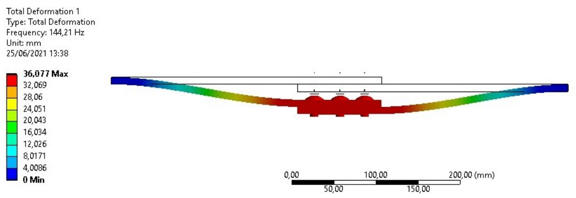
(e) 5th mode at 1048.5 Hz.



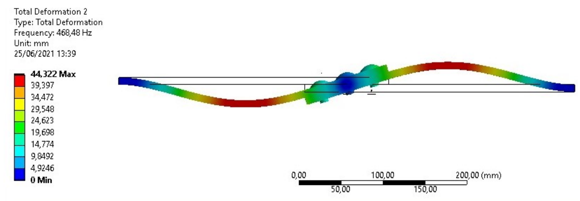
(f) 6th mode at 1133.0 Hz.

Figure 10: Modes Shape of the structure with one rivet extracted in ANSYS.

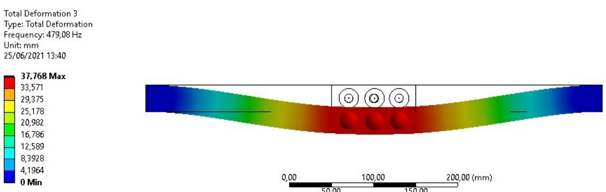
In both cases, the 1st, 2nd, 4th and 6th are flexural modes (out plane) and the 3rd and 5th are in-plane vibrations. Figure 10 illustrates the beams connected by one rivet vibrational and natural frequencies obtained in the simulation o, where the flexural modes are occurring at frequencies 117.29 Hz (Fig. 10a), 388.13 Hz (Fig. 10b), 664.64 Hz (Fig. 10d), and 1133.0 Hz(Fig. 10f). The in-plane vibration modes happened at 417.58 Hz (Fig. 10c) and 1048.5 Hz (Fig. 10e). Figure 11 illustrates the beams connected by three rivet vibrational and natural frequencies obtained in the simulation,



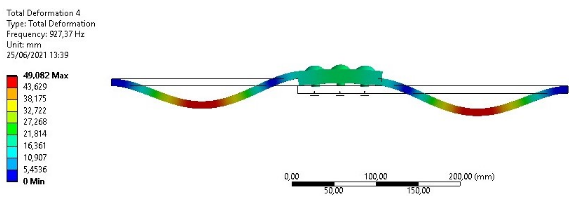
(a) 1st mode at 144.21 Hz.



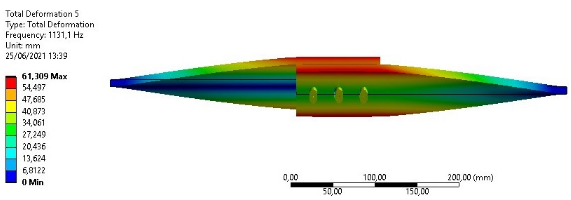
(b) 2nd mmode at 468.48 Hz.



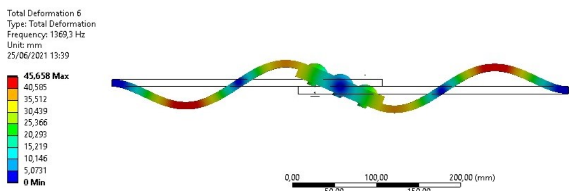
(c) 3rd mode at 479.08 Hz.



(d) 4th mode at 927.37 Hz.



(e) 5th mode at 1131.1 Hz.

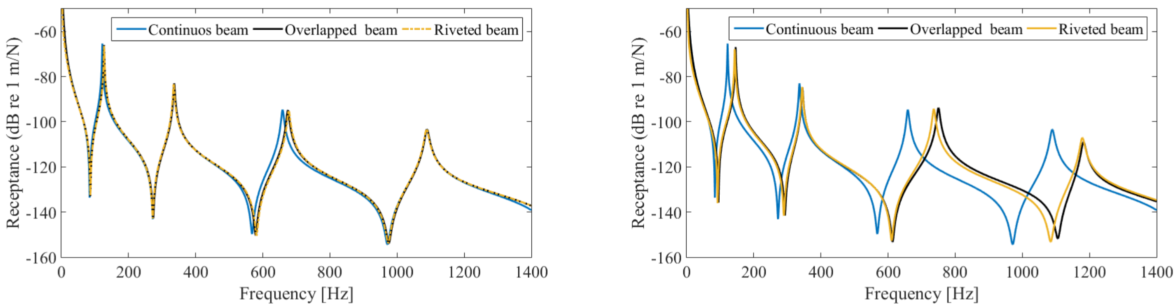


(f) 6th mode at 1369.3 Hz.

Figure 11: Modes Shape of the structure with three rivets extracted in ANSYS.

where the flexural modes are occurring at frequencies 144.21 Hz (Fig. 11a), 468.48 Hz (Fig. 11b), 927.37 Hz (Fig. 11d), and 1369.3 Hz(Fig. 11f). The in-plane vibration modes happened at 479.08 Hz (Fig. 11c) and 1131.1 Hz (Fig. 11e).

The SEM yields the transcendental eigenvalue problems, while the conventional FEM yields the linear eigenvalue problems (Lee, 2009). Transcendental eigenvalue problems are not direct and imply a proper solution method for the eigenvalue problems derived by the SEM. Hence, the estimation of the modal parameters of the system is not direct. Therefore, harmonic analysis is applied in this case, and only the flexural modes considered. A unitary force exciting the beam at the right-hand edge and the receptance frequency response function (FRF) estimated at both ends of the assembled beams in a frequency range of 0-1400Hz. Figure 12(a) shows the beam joined by one rivet FRFs for a continuous beam (31.6 x 8.46 x 600 mm), two overlapped beams as in fig 5 with and without the rivet. Demonstrates the changes in the receptance by the inclusion of the rivet. The 1st and 3rd flexural modes were the most affected by the connection because of its mid-spam location where the odd modes have a high amplitude, as shown in Fig 10a and 10c.



(a) (b)

Figure 12: Representation: (a) Physical structure; (b) Spectral element model.

Figure 12(b) shows the FRF of a continuous beam (31.6 x 8.46 x 600 mm) and the beam joined by three rivets with and without the rivets. The receptance response shift to the right with the inclusion of the rivets. In this case, all the four flexural modes presented some alteration, being greater at the 1st, 3rd, and 4th flexural modes.

Table 2: Natural Frequencies (Hz)

Mode	Analytical	Caso 1- FEM	Caso 1- SEM	Caso 2 - FEM	Caso 2 - SEM
1	122	117.29	127	144.21	124
2	336	388.13	334	468.48	344
3	-	417.58	-	479.08	-
4	659	664.64	678	927.37	740
5	-	1048.5	-	1131.1	-
6	1089	1133.0	1088	1369.3	1181

Table 2 listed a summary of natural and resonance frequencies obtained with SEM and FEM with one and three rivets compared analytical solution of a continuous beam. Results differ among the used method, related to the rivet interface connection and by the springs assumed values. Both numerical approaches presented a progressive increase in the frequencies as the rivet's number goes from one to three. A more detailed investigation must be performed in the numerical models.

5. CONCLUSION

This work explores the numerical model via SEM and FEM of two Euler–Bernoulli beam connected by a rivet joint in the mid-span. The models were performed using the FEM analysis via ANSYS-Workbench, combining in the mesh the SOLID186, CONTAC174, and the TARGE170 elements. SEM model employed the use of a beam spectral element and the spectral rivet element. An approximate solution for the dynamical behaviour of assembled structure was obtained using both methods. The solution provides the frequency response function of the connected system and its modal parameters. The resonance frequencies were compared with the analytical solution of a continuous beam. Results found by comparing analytical solutions with the numerical simulated via FEM and SEM differ by including one and three rivets. The error associated with the estimations can be due to some assumption made by the rivet parameters and because of the rivet interface connection model. It must be investigated, and some adjust implemented as in the parameters of the rivet joint interface and the bolt linear and rotational spring. However, using SEM is the easy access to the matrix and general formulation, reduced number of elements in the mesh propitiating inclusion of uncertainties in parameters and non-linearities without losing accuracy and low computational cost.

6. REFERENCES

Abulfaiz, K.A. and Shelke, R.D., 2018. "Analysis of riveted joints under vibration". *International Journal of Engineering Research and Generic Science*, Vol. 4, No. 5, pp. 2455–1597.

Ahmadian, H. and Jalali., H., 2007. "Identification of bolted lap joints parameters in assembled structures". *Mechanical Systems and Signal Processing*, Vol. 21, p. 1041–1050.

ANSYS, 2009a. *ANSYS User's Manual: Structural Analysis Guide*.

ANSYS, 2009b. "Theory reference for the mechanical apdl and mechanical application". www.anays.com. Accessed 12 January 2021.

Bagale, Y., Nagarkar, S., Attarde, L. and Bhadane, K., 2019. "Analysis of rivet joint for application of substation". *International Research Journal of Engineering and Technology*, Vol. 6, No. 3, pp. 2395–0072.

Choi, J. and Inman, D.J., 2014. "Spectrally formulated modeling of a cable-harnessed structure". *Journal of Sound and Vibration*, Vol. 333, No. 14, p. 3286–3304.

Dourado, M. and Meireles, J., 2016. "Obtaining a simplified finite element model for representing riveted joints in dynamic structural analysis using an update tool". *Rev. int. métodos numcálc. diseño ing*, Vol. 32, No. 3, pp. 173–180.

Doyle, J.F., 1997. *Wave propagation in structures : spectral analysis using fast discrete Fourier transforms*. Mechanical engineering. Springer-Verlag New York, Inc., New York, 2nd edition.

Dutkiewicz, M. and Machado, M.R., 2019a. "Measurements in situ and spectral analysis of wind flow effects on overhead transmission lines". *Sound & vibration*, Vol. 53, pp. 161–175.

Dutkiewicz, M. and Machado, M.R., 2019b. "Spectral element method in the analysis of vibrations of overhead transmission line in damping environment". *Structural Engineering and Mechanics*, Vol. 71(3), pp. 291–303.

He, X.C., Pearson, I. and Young, K., 2007. "Three dimensional finite element analysis of transverse free vibration of self-pierce riveting beam". *Key Engineering Materials*, Vol. 333, No. 344, p. 647–654.

J., K., W., W., J., J. and E., S., 2009. "Methods for fem analysis of riveted joints of thin-walled aircraft structures within the imperja project". *ICAF Symposium – Rotterdam, 27–29 May*.

Lee, U., 2001a. "Dynamic characterization of the joints in a beam structure by using spectral element method". *Shock and Vibration*, Vol. 8, No. Artigo ID 254020, pp. 1–10.

Lee, U., 2001b. "Dynamic characterization of the joints in a beam structure by using spectral element method". *Shock and Vibration*, Vol. 8, No. 6, pp. 357–366.

Lee, U., 2009. *Spectral element method in structural dynamics*. Wiley, Singapore, Inc.

Machado, M.R., Appert, A. and Khalij, L., 2019. "Spectral formulated modelling of an electrodynamic shaker". *Mechanical Research Communication*, Vol. 97, pp. 70–78.

Machado, M.R., Dutkiewicz, M., Mat, C.F.T. and Castello, D.C., 2020. "Dynamic response of overhead transmission line in turbulent wind flow with application of the spectral element method". *Mechanical Systems and Signal Processing*, Vol. 136, p. 106483.

Narayanan, G. and Beskos, D., 1978. "Use of dynamic influence coefficients in forced vibration problems with the aid of fast fourier transform". *Computers & Structures*, Vol. 9, No. 2, pp. 145–150.

Petyt, M., 2010. *Introduction to Finite Element Vibration Analysis*. Cambridge University Press; 2nd edition.

Wang, C., Wang, N., Ho, S., Chen, X., Pan, M. and Song, G., 2018. "Design of a novel wearable sensor device for real-time bolted joints health monitoring". *IEEE Internet of Things Journal*, Vol. 5, No. 6, pp. 5307–5316.

7. RESPONSIBILITY NOTICE

The following text, properly adapted to the number of authors, must be included in the last section of the paper: The author(s) is (are) solely responsible for the printed material included in this paper.