Determination of optimal coupling stiffness using modal updating techniques for Stiffened Plates

Shrivastava, K.; Vijayan, Kiran; Arora, Vikas

Published in:
Procedia Structural Integrity

DOI:
10.1016/j.prostr.2019.05.068

Publication date:
2019

Document version
Final published version

Document license
CC BY-NC-ND

Citation for published version (APA):

Terms of use
This work is brought to you by the University of Southern Denmark through the SDU Research Portal. Unless otherwise specified it has been shared according to the terms for self-archiving. If no other license is stated, these terms apply:

- You may download this work for personal use only.
- You may not further distribute the material or use it for any profit-making activity or commercial gain.
- You may freely distribute the URL identifying this open access version.

If you believe that this document breaches copyright please contact us providing details and we will investigate your claim. Please direct all enquiries to puresupport@bib.sdu.dk

Download date: 24. Dec. 2019
2nd International Conference on Structural Integrity and Exhibition 2018

Determination of optimal coupling stiffness using modal updating techniques for Stiffened Plates

K. Shrivastava, K. Vijayan, V. Arora

Abstract

Stiffened structures are often utilized as structural members for numerous applications in various fields of engineering like Civil, Automobiles, Naval, Aerospace, etc. Thus analyzing stiffed structure for their structural integrity become pivotal. Simplest instance of a stiffened structure could be a plate with a beam acting as a reinforcement. The presented study focuses on Modal analysis of a stiffened plate comprising of a base plate and a beam attached to it in the longitudinal direction. Numerical analysis is carried out by formulating a FE model for the beam and plate. The plate and beam elements are coupled using discrete spring element. The capability of the model is verified by comparing the natural frequencies obtained through experimental techniques. A numerical based model updating was carried out to determine the optimal spring stiffness using Latin hypercube sampling. The optimal values of the spring stiffness were determined which reduced the error in the natural frequencies.

© 2019 The Authors. Published by Elsevier B.V.
This is an open access article under the CC BY-NC-ND license (https://creativecommons.org/licenses/by-nc-nd/4.0/)
Selection and peer-review under responsibility of Peer-review under responsibility of the SICE 2018 organizers.

Keywords: Stiffened Plates; Modal Analysis; Model Updating

1. Introduction

A major cause of structural failure could be attributed to structural vibration. In order to ensure the structural integrity of any body when subjected to harmful vibration, it is important to know its modal properties. These properties define the characteristics and behavior of the structure under dynamic conditions.

* Corresponding author. Tel.: +91-3222-282283; fax: +91-3222-255303.
E-mail address: krshrivastava@iitkgp.ac.in
**Modal properties of any civil structure or mechanical system could be easily identified experimentally with various Modal testing methods as illustrated by Schwarz (1999) and Ewins (2000). Experimental modal testing methods work by analyzing the response of the system to a known input excitation. These response signals help in defining the Frequency Response Function (FRF) which provides an estimation on the modal parameters. In order to determine these properties in the design phase of a system, there are many theoretical methods available to numerically model the system and then predict its modal properties.**

One such method is Finite Element (FE) method which discretizes the overall structural geometry into many smaller elements with material and structural properties assigned to each element. Assembling these elements together gives an estimate on the global response of the system. But like various other numerical methods, FEM also gives approximated results due to various uncertainties relating to boundary conditions, joints, material property, etc. In order to provide an estimate even under these uncertainties and enhance the output results, the FE numerical model could be updated.

Finite element model updating (FEMU) is a technique which could be employed to enhance the predictions of a theoretical model and making them closer to the experimental observations from a physical structure. However, Imregun et al. (1995) established that this is only possible if the measured data is sufficiently acquired to represent the actual behavior of the structure and is free of noise to an acceptable threshold.

The present work is an extension to work carried out on modelling and performing modal analysis on plate and beam structures by Haldar et al. (2017). Current work is towards combining these models together to form a stiffened structure. For simplicity, damping has been ignored from the numerical modelling. Authors have also attempted to redefine the joint between the plate and beam FE models. A modelling approach similar to the one carried out by Shankar et al. (1995), Liu et al. (2001) and Vijayan et al. (2013) was considered. The welded joint was considered as a flexible joint having finite stiffness which coupled the continuous beam and plate with a pair of discrete lumped translational and rotational joint stiffnesses.

### 2. Experimental Setup

An experimental study was initiated on a test sample plate made up of a L-section angle having 50x50mm flange and web width and 6mm overall thickness respectively. This angle is welded on a plate measuring 310x230x12mm (LxBxT), to form the stiffened plate sample. The angle is welded longitudinally and runs for the complete length of the plate. The sections are welded together using electric arc weld technique. The material of both the sections is Mild Steel and the standard material properties are assumed for theoretical modelling. The entire sample was supported symmetrically on soft Styrofoam sheets, which provide minimum stiffness and mimics a Free-Free boundary condition as shown in Fig. 1.
The experimental setup consists of an accelerometer attached to the sample plate and an impact hammer to induce an impulse excitation at specific points on the structure. These specific points are a part of grid which is similar to the mesh generated by the nodal points in FE analysis. The response to this impulse excitation is measured together with the forcing signal through the hammer impacting at a fixed location and an accelerometer moving to different grid points. The sensors are connected to the computer through a Data Acquisition System (DAQ) which converts the input signals from time domain to frequency domain using Fast Fourier Transform (FFT), as stated in eq. 1. Then the modal parameters are identified by a Multi-Analyzer in the frequency domain using frequency response functions (FRF) or transfer functions $H(\omega)$ as stated in eq. 2. Since the response is collected in terms of acceleration, this FRF is known as accelerance (Ewins 2000).

$$F(\omega) = \int_{-\infty}^{\infty} f(t)e^{-i\omega t} dt$$  \hspace{1cm} (1)

where, $F(\omega)$ is the Frequency response and $f(t)$ is the impulse response signal of the system.

$$H_{jk}(i\omega) = \frac{X_j(i\omega)}{F_k(i\omega)}$$  \hspace{1cm} (2)

where, $X_j(\omega)$ is the harmonic response at point $j$, caused by an impulse force $F_k(\omega)$ at point $k$.

The above system could be considered as a continuous multi DOF system, which would have infinite modes. However, for this analysis we have extracted first 5 modes (excluding rigid modes) of the system and compared them with the modal values obtained by FE numerical model. The dynamic properties of the system are listed in Table 1; these nature frequencies were estimated from the peak amplitude as in the FRF. These FRF were generated by carrying out a roving accelerometer test on three different points distributed throughout the plate.
The experimental setup consists of an accelerometer attached to the sample plate and an impact hammer to induce an impulse excitation at specific points on the structure. These specific points are a part of a grid which is similar to the mesh generated by the nodal points in FE analysis. The response to this impulse excitation is measured together with the forcing signal through the hammer impacting at a fixed location and an accelerometer moving to different grid points. The sensors are connected to the computer through a Data Acquisition System (DAQ) which converts the input signals from time domain to frequency domain using Fast Fourier Transform (FFT), as stated in eq. 1. Then the modal parameters are identified by a Multi-Analyzer in the frequency domain using frequency response functions (FRF) or transfer functions as stated in eq. 2. Since the response is collected in terms of acceleration, this FRF is known as accelerance (Ewins 2000).

\[
F(\omega) = \int_{-\infty}^{\infty} f(t) e^{-i\omega t} dt
\]

\[
H(\omega) = \int_{-\infty}^{\infty} F(\omega) e^{i\omega t} dt
\]

The dynamic properties of the system are listed in Table 1; these nature frequencies were estimated from the peak amplitude as in the FRF. These FRF were generated by carrying out a roving accelerometer test on three different points distributed throughout the plate.

<table>
<thead>
<tr>
<th>Mode</th>
<th>Experiment (\omega_n) (Hz)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>488</td>
</tr>
<tr>
<td>2</td>
<td>-</td>
</tr>
<tr>
<td>3</td>
<td>928</td>
</tr>
<tr>
<td>4</td>
<td>1098</td>
</tr>
<tr>
<td>5</td>
<td>1393</td>
</tr>
</tbody>
</table>

### 3. Numerical Modeling

The next step in analyzing the stiffened plate for modal parameters was carried out by establishing a simple numerical model. The preliminary numerical model of the Stiffened Plate was created by coupling the Plate and Beam Finite Element model through Longitudinal and Torsional 1-D springs. The schematic model of the system is depicted in Fig. 2.

The plate was modelled using Mindlin plate theory formulated by Mindlin et al. (1956). The energy expression forming the basis of FE model for plate element is given by eq. 3. The terms in the energy expression are contribution from bending and shear.

\[
U_{\text{plate}} = \frac{1}{2} \int_V \sigma_b \varepsilon_b dV + \frac{\alpha}{2} \int_V \tau_s \gamma_s dV
\]

where, \(\sigma_b\) and \(\varepsilon_b\) are bending stress and strain respectively,

\(\tau_s\) and \(\gamma_s\) are transverse sheer stress and strain respectively,

and the \(\alpha\) is sheer correction factor and is taken as \(\alpha=0.833\).

The plate was discretised using bilinear four-noded quadrilateral (Q4) element as illustrated by Ferreira (2009) and Cook et al. (2003). Each node has three DOF,

\[
u = z_0 \quad v = x\theta_z \quad w = y\theta_z
\]

where, \(u\) is the translational whereas \(v\) and \(w\) are rotational (bending) DOF of the plate.
The material and sectional properties of the plate are provided in Table 2. These properties were taken presuming standard mild steel.

<table>
<thead>
<tr>
<th>Material Property</th>
<th>Value</th>
<th>Sectional Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \rho ) (kg/m(^3))</td>
<td>7850</td>
<td>L (m)</td>
<td>0.31</td>
</tr>
<tr>
<td>( E ) (N/m(^2))</td>
<td>2.1e11</td>
<td>B (m)</td>
<td>0.23</td>
</tr>
<tr>
<td>( \nu )</td>
<td>0.3</td>
<td>T (m)</td>
<td>0.012</td>
</tr>
<tr>
<td>( G ) (N/m(^2))</td>
<td>75e9</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

The L-Section Stiffener is modelled as beam using a 1-D beam element having one node at each end and two DOF at each node as illustrated by Ferreira (2009) and Cook et al. (2003). This element is based on Timoshenko Beam theory formulated by Timoshenko (1921), which includes effects of shear deformation in the cross section about the central axis. Table 3 enumerates the material and sectional properties of the beam element.

<table>
<thead>
<tr>
<th>Material Property</th>
<th>Value</th>
<th>Sectional Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \rho ) (kg/m(^3))</td>
<td>7850</td>
<td>L (m)</td>
<td>0.31</td>
</tr>
<tr>
<td>( E ) (N/m(^2))</td>
<td>2.1e11</td>
<td>Width of Web (m)</td>
<td>0.05</td>
</tr>
<tr>
<td>( \nu )</td>
<td>0.3</td>
<td>Thickness of Flange (m)</td>
<td>0.006</td>
</tr>
<tr>
<td>( G ) (N/m(^2))</td>
<td>75e9</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

The assumed DOF for the beam elements were,

\[ \begin{align*}
  x &= z_0 \\
  y &= x \dot{\theta}_z 
\end{align*} \]

where, \( x \) is the deflection \( y \) is the slope of the beam.

### 3.1. FEM Discretization

The mesh size was determined by convergence study of the model. An error tolerance limit was set for first five natural frequencies excluding rigid modes. The convergence test provided the optimum number of elements required for the numerical analysis as 200. The meshing pattern is depicted through a schematic diagram in Fig. 3.

[Fig. 3. FEM Discretization]
The beam was decided to be attached in the longitudinal direction over the plate and thus the number of beam elements were equal to the plate seeding in the longitudinal direction. Each beam node was coupled with the plate node by a discrete spring element, making independent coupling spring elements equal to beam elements. The plate and beam elements were coupled in \( u \) and \( x \) DOF longitudinally and torsionally in \( v \) and \( y \) DOF respectively.

The Global Stiffness and Mass matrix were generated by assembling and coupling the elemental stiffness and mass matrices of plate and beam together. The boundary condition selected for this analysis was Free-Free. The governing equation of motion for the entire structure for free vibration response given in eq. 4. The natural frequency and the mode shapes of the numerical model are obtained by solving the eigenvalue problem for the given global mass and stiffness matrices. Solution for the eigenvalue problem is given by eq. 5.

\[
M^g(\ddot{x}) + K^g(x) = \begin{bmatrix} M_p^g & 0 \\ 0 & M_b^g \end{bmatrix}(\ddot{x}) + \begin{bmatrix} K_p^g + [K_f + K_t] & -[K_f + K_t] \\ -[K_f + K_t] & K_b^g + [K_f + K_t] \end{bmatrix}(x) = 0
\]

where, \( M^g \), \( M_p^g \) and \( M_b^g \) are Assembled, Plate and Beam Global Mass matrices respectively, \( K^g \), \( K_p^g \) and \( K_b^g \) are Assembled, Plate and Beam Global Stiffness matrices respectively, \( K_f \) and \( K_t \) are Longitudinal and Torsional Stiffness.

\[
K^g * (Z) = M^g \omega_n^2 * (Z) \quad \Rightarrow \quad (K^g/M^g - \omega_n^2 I)^* (Z) = 0
\]

4. Model Updating

A substantial work in updating the numerical model with the experimental FRF data could be found in literature by Mottershead (1993) and Braz-César (2017) including multiple techniques and involving various modal parameters such as mass, stiffness and damping matrices. In context to the present study, modal updating is implemented on the coupling between the beam and plate. This is implemented as an optimization problem. The objective of the optimization is in minimizing the error in first five natural frequencies, excluding the rigid body modes.

The aforementioned optimization was subjected to finding proper stiffness values of two variables \((K_f, K_t)\). In the initial stages the two methods considered for this optimization were Monte Carlo simulations (MC) and Latin Hypercube sampling (LHS). A Monte Carlo simulation involved generation of random sample values for each variable and then pairing these values for numerical model simulation. MC has a disadvantage; the number of sample points might be high for the optimization loop.

LHS is also sample generation technique that spread the sample points more evenly across all possible values. It partitions each input distribution into \( n \) intervals of equal probability, and selects one sample from each interval. It shuffles the sample for each variable so that there is no correlation between the variable. For the present study LHS algorithm developed by McKay M., et al. (1979), was selected due to its ability to greatly reduce univariate variance.

All the longitudinal and torsional spring elements were assumed to have constant value throughout the weldline, whereas in real life scenario these values could change along the weldline subjected to weld type, welding parameters and material properties. The initial assumption for the variables \( K_f \) and \( K_t \) is enumerated in Table 4.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Initial Assumption</th>
<th>Lower Limit</th>
<th>Upper Limit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Longitudinal Spring Stiffness ((K_f)) ((N/m))</td>
<td>2.1e6</td>
<td>1e5</td>
<td>9e7</td>
</tr>
<tr>
<td>Torsional Spring Stiffness ((K_t)) ((Nm/rad))</td>
<td>8.0e3</td>
<td>1e2</td>
<td>9e4</td>
</tr>
</tbody>
</table>
Fig. 4 shows a flowchart for the overall numerical analysis and implementation of LHS for optimizing the spring stiffnesses.

5. Results

The error in the first five natural frequencies were estimated on the values of initial assumption provided in Table 4. Their comparison with experimental natural frequencies is given in Table 5. The error could be due to the gap in the exact and assumed values of material, spring stiffness and boundary condition. This indicates the need for a model updating including more optimization parameters and also to take damping into consideration. The absence of second mode from the experimental results indicate that the experimental modal analysis need to be carried out covering more number of grid points.

Table 5. Natural Frequencies before optimization

<table>
<thead>
<tr>
<th>Mode</th>
<th>$\Omega_n$ (Exp) (Hz)</th>
<th>$\Omega_n$ (FEM) (Hz)</th>
<th>$\Omega_n$ (Exp) - $\Omega_n$ (FEM) (Hz)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>488</td>
<td>328.1737</td>
<td>159.8263</td>
</tr>
<tr>
<td>2</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>3</td>
<td>928</td>
<td>729.214</td>
<td>198.786</td>
</tr>
<tr>
<td>4</td>
<td>1098</td>
<td>833.0925</td>
<td>264.9075</td>
</tr>
<tr>
<td>5</td>
<td>1393</td>
<td>998.1634</td>
<td>394.8366</td>
</tr>
</tbody>
</table>

Table 6 provides the optimized values for spring stiffnesses obtained from LHS algorithm. Table 7 enumerates the natural frequencies obtained from numerical modal analysis after spring stiffness optimization and its comparison with the experimental values. It also shows the reduction in error due to optimization.

Table 6. Optimized variable values obtained from LHS

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Longitudinal Spring Stiffness ($K_l$) (N/m)</td>
<td>1.02e7</td>
</tr>
<tr>
<td>Torsional Spring Stiffness ($K_t$) (Nm/rad)</td>
<td>1.94e4</td>
</tr>
</tbody>
</table>
Table 7. Natural Frequencies after optimization

<table>
<thead>
<tr>
<th>Mode</th>
<th>ωn (Exp) (Hz)</th>
<th>ωn (FEM) (Hz)</th>
<th>ωn (Exp) - ωn (FEM) (Hz)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>488</td>
<td>487.98</td>
<td>0.013513</td>
</tr>
<tr>
<td>2</td>
<td>-</td>
<td>692.34</td>
<td>-</td>
</tr>
<tr>
<td>3</td>
<td>928</td>
<td>927.17</td>
<td>0.827766</td>
</tr>
<tr>
<td>4</td>
<td>1098</td>
<td>1099.07</td>
<td>-1.07818</td>
</tr>
<tr>
<td>5</td>
<td>1393</td>
<td>1393.15</td>
<td>-0.15739</td>
</tr>
</tbody>
</table>

6. Conclusion

The present work showcases an experimental modal analysis of a stiffened plate in order to determine its dynamic properties. A roving accelerometer test was carried out to determine the natural frequency of the sample by obtaining its FRF. A numerical model was also developed through FEM for the same sample plate. The FE model was generated by coupling plate and beam elements by discrete longitudinal and torsional spring elements representing a welded joint. Later the numerical model was updated by optimizing the spring stiffnesses through LHS technique in order to better represent the experimental system.

Upon updating the model by optimizing the spring stiffnesses the error in natural frequencies between the experimental and numerical analysis was reduced. However, it was found that the roving accelerometer test on three different points provided insufficient data and failed to capture the contribution of all the modes, resulting in absence of a natural frequency from the experimental modal analysis.

Future work would be carry out model updating using more number of optimization parameters. The experimental analysis can also be improved by considering more grid points.

Acknowledgements

The author gratefully acknowledge the support and funding received for the presented work from SERB under Project no - ECR/2016/001531

References


Halder, S., Shrivastava, K., Vijayan, K., Arora, V., 2017. Uncertainty in the modal parameters of the structural elements within a stiffened structure, 5th International Conference on Ship and Offshore Technology ICSOT, IIT Kharagpur, India.


