Dynamic Characterization of an Aluminum “F-Structure” with an Inclined Stiffener.

Prof.H.G.Vyas
Senior Assistant Professor ,Mechanical Engineering Department, Government Engineering College, Bhavnagar

Received: May 15, 2018 Accepted: June 30, 2018

ABSTRACT This paper contains prediction and validation of dynamic characteristics and behaviour of an aluminium ‘F’ structure, representing generalized machine tool structure, modelled using finite elements incorporating plane frame elements. Structural modifications are made by adding stiffeners at different orientations. A model of an aluminium ‘F’ structure is simulated in MATLAB for the prediction of natural frequencies, Mode shapes for an undammed structure. Physical response and Frequency Response Function (FRF) are plotted for different configurations of the structure. It is observed that addition of stiffener modifies the dynamic characteristics. an inclined stiffener restricts the motion of the end of arm but it is not doing so for the table arm end. The present model not only predicts the dynamic characteristics but also it gives significant analytical tool for further studies by an extension to mass modification. Simulation work is compared and validated with experimental results performed in the laboratory.

Keywords:

Accurate dynamic mathematical model of a structure is essential for simulating reliably the dynamic characteristics. Such a model would allow in improving the dynamic design of a structure at the computer level resulting in an optimized design apart from savings in terms of money and time. In practice a mathematical model can be derived by analytical approaches such as by finite element method [1]. Detailed work in dynamic design using updated finite element model was carried out by Modak et.al. They have used the mass modification and beam modification for updated model for the prediction of dynamic characteristics [1,2].

Increasing demand for accuracy and out put has caused vibration problems to gain importance .Both forced and sustained vibrations are known to be prejudicial to accuracy and out put .All machine tools give rise to vibration. Deterioration in machine condition always produces a corresponding increase in vibration levels. Vibration signals are one of the most reliable parameter used in machine health monitoring to check machine condition .The purpose of the vibration analysis of machine tool structure is to sustain the useful oscillations and eliminate the unwanted ones. In general, machine structures are very complex due to various functional elements or components. There fore modelling and analysis of actual structure is expensive and it requires more computational effort. To simplify the work, F-structure is considered as generalized machine structure [3].

Structural dynamic modification and its significance:

In machine tool structures condition of resonance leads to excessive difficulties, failure and inaccuracy during the phase of design a structure is generally given a suitable configuration and from strength point of view it is designed. The prototype manufactured from this design may lead to unwanted excessive vibrations. Due to the discrepancies arising in modelling and the actual performance of structure in real life, it is required to elaborate the method for checking the dynamic response of the structure so that we can tune physical model in order to obtain a nearly close tolerable limits of vibration. In addition the natural frequencies and mode shapes of the structure can be predicted and compared with the actual performance.

Methodology:

The present work aims at modelling and simulating the effect of providing inclinedstiffening members at the locations where there are more chances of vibrations. The inclinedstiffeners are located between the members representing the column and the arm of the structure at some known orientations. The effect of stiffeners is observed .The method of predicting the dynamic behaviour of the structure with the help of simulation in MATLAB is proposed. Experimental validation is made by FFT analyser and simulated and experimental results are compared.
Details of F-Structure:
Material: Aluminium.
Overall height and width: 800 mm X 400 mm
Dimension of cross section: 25mmx25mm
Mass density of Aluminum: 2700 Kg/m\(^3\)
Young’s modulus of elasticity of Aluminum: 6.9x10\(^{10}\)N/m\(^2\)
Mass moment of Inertia I:3.25521x10\(^{-8}\) m\(^4\)
Area of the elements:6.25x10\(^{-4}\) m\(^2\)
No. of elements:7
No. of nodes:7
No. of nodes per element: 2
Inclined stiffener : Length: 548 mm, angle: 46.12\(^\circ\)
Cross section of stiffener: 25mm x 5mm.

Assumption made while MATLAB simulation of structure:
- In practical case, the machine structure is three dimensional models but for simplification it is considered two dimensional.
- Joint and boundary conditions are considered to be rigid and fixed. The displacement at the grounded node is considered to be zero.
- Damping is neglected.
- Mass overlapping at cross joints are not considered.

Simulation Procedure:
1) The structure is divided into 7 two nodded plane frame elements and 7 nodes. Each node has two translational and one rotational degrees of freedom.
2) Elemental connectivity table is prepared for the structure, which shows the connectivity of each pair of nodes. Database is prepared for the elements, element connectivity, material properties, geometry of the model and boundary conditions,
3) Elemental mass matrices and stiffness matrices are computed. Assembly of all the element stiffness and mass matrices is done for obtaining global mass and stiffness matrices.
4) Modal analysis is done by exciting a structure with a constant force of 160 N at node 2
5) Natural frequencies, Mass normalised mode shapes and physical responses at each nodes are obtained.

<table>
<thead>
<tr>
<th>Excitation force of 160N at node 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>4 3 2</td>
</tr>
</tbody>
</table>

(a) Nodal representation of ‘F’-structure with cross stiffener.

Experimental setup. Figure 1
Table 1 Element connectivity table for ‘F’ structure with a cross stiffener

Cross stiffener is positioned between nodes 1 and 6 as shown in figure 4.18. Excitation of 160 N force is applied at node 2 by impact hammer and obtained natural frequencies are compared in table 2. We can see that simulated results are very close to experimental results. Here first natural frequency of structure is 15 Hz which is almost same after any structural modification. Numerical values of these natural frequencies are higher than bare structure as by addition of cross stiffener overall stiffness of structure will increase.

<table>
<thead>
<tr>
<th>Sr.No.</th>
<th>Experimental results (A)</th>
<th>Simulation results (B)</th>
<th>Error (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>15</td>
<td>15.48</td>
<td>3.2</td>
</tr>
<tr>
<td>2</td>
<td>45</td>
<td>60.34</td>
<td>34.08</td>
</tr>
<tr>
<td>3</td>
<td>-</td>
<td>135.73</td>
<td>-</td>
</tr>
<tr>
<td>4</td>
<td>-</td>
<td>432.90</td>
<td>-</td>
</tr>
</tbody>
</table>

Table 2 Frequencies comparison of aluminum ‘F’ structure with inclined stiffener.

Figure 2 (a) Simulated physical response

Figure 2 (b) Experimental physical response
Conclusion:

Figure 2 shows simulation, experimental physical response and FRF for 'F'-structure with cross stiffener (Aluminum). Nodes 2, 3 and 4 have very high negative response whereas node 6 shows 5x10^-6 m highest positive response at a moment which is higher than positive response at node one.

EMA gives 284.2x10^-6 m response at natural frequency of 15 Hz at node 1 whereas MATLAB simulation predict response of 300x10^-6 m same node. Which are very close than expected.

Response at node 6 (Channel 2) are not so closed which shows some error in result. But overall results are satisfactory that we can accept the FEA simulation modal model for prediction of dynamic characteristic of machine structure.

References:

For journal papers:

For books: