Dynamic Characterization of an Aluminum "F-Structure" with a Vertical Stiffener.

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

Received: May 16, 2018 Accepted: June 30, 2018

ABSTRACT *This paper contains prediction 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 vertical stiffeners. A model of 'F' structure is simulated in MATLAB for the prediction of natural frequencies, Mode shapes for an un damped 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. If a vertical stiffener is added between the arm end and the table end the physical response of the nodes at and nearby the mid node is more and the same is much reduced at the nodes where stiffener is provided. This is because of the effect of constrained motion at the connecting nodes. 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 vertical stiffening members at the locations where there are more chances of vibrations. The vertical stiffeners 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
Vertical stiffener: Length: 395 mm, angle: 90^0
Cross section of stiffener: 25mm x 5mm.

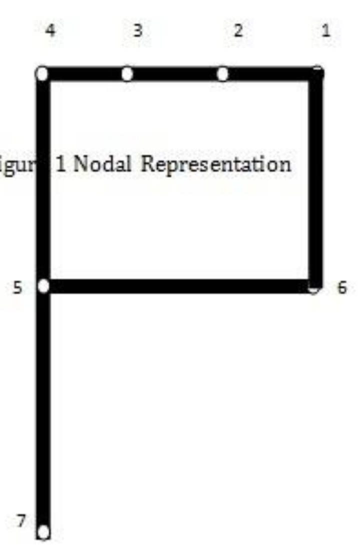
Assumption made while MATLAB simulation of structure:

- 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 7nodes. 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 150 N at node 2
5) Natural frequencies, Mass normalised mode shapes and physical responses at each nodes are obtained.

'F'-structure with a vertical stiffener

Excitation force of 150 N at node 2	
 Figur 1 Nodal Representation	 Experimental setup.

ELEMENT No.	NODE - 1	NODE - 2	Cross sectional area (m^2)	Moment of Inertia (m^4)	Element Length (m)	Angle with reference (X)
1	1	2	6.25E-04	3.25E-08	0.15	0
2	2	3	6.25E-04	3.25E-08	0.117	0
3	3	4	6.25E-04	3.25E-08	0.138	0
4	4	5	6.25E-04	3.25E-08	0.405	90
5	5	6	6.25E-04	3.25E-08	0.405	0
6	5	7	6.25E-04	3.25E-08	0.405	90
7	1	6	1.25E-04	2.60E-10	0.395	90

Table 1 Element connectivity table for 'F' structure with a vertical stiffener.

Vertical stiffener is positioned between nodes 1 and 6. Excitation of 150 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 vertical stiffener overall stiffness of structure will increase.

Comparison of first three natural frequencies (Hz)			
Sr.No.	**Experimental results(A)**	**Simulation results(B)**	**Error (%)**
1	15	16.20	08
2	52.48	65.34	24.50
3	122.68	135.73	10.63
4	-	432.90	-

Table 2 Frequencies comparison of aluminum 'F' structure with vertical stiffener.

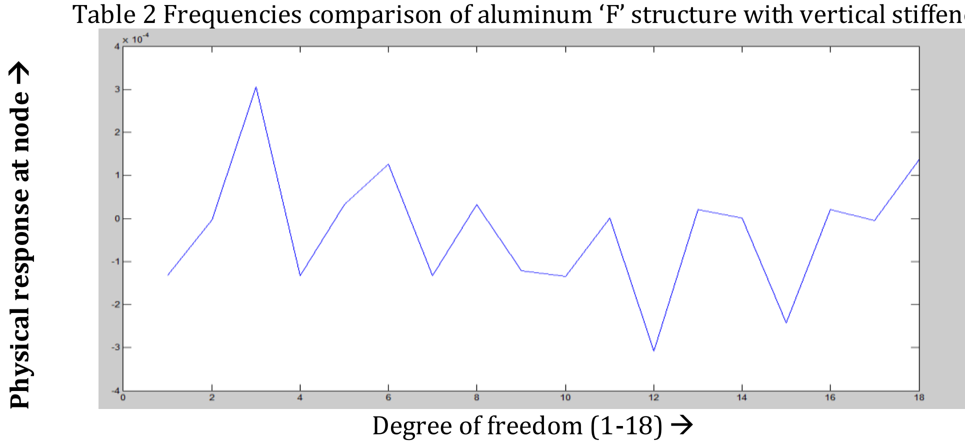


Figure 2(a) Simulated Physical response.

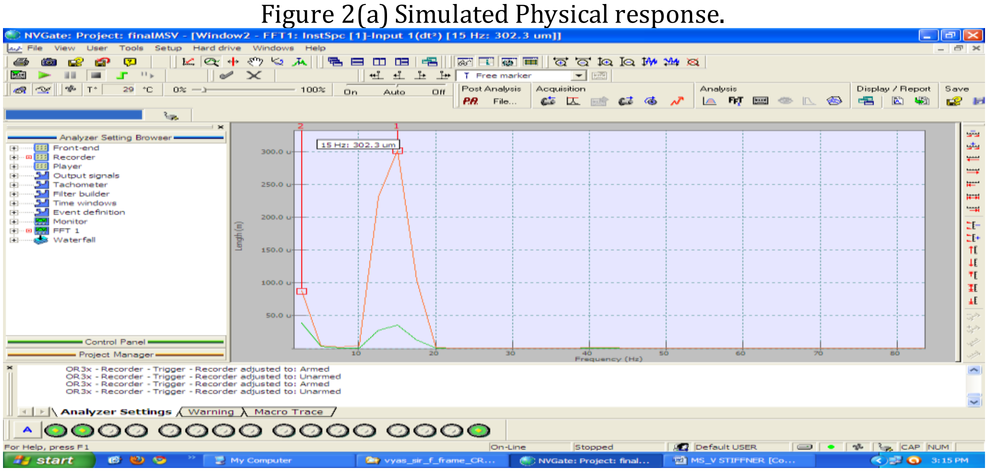
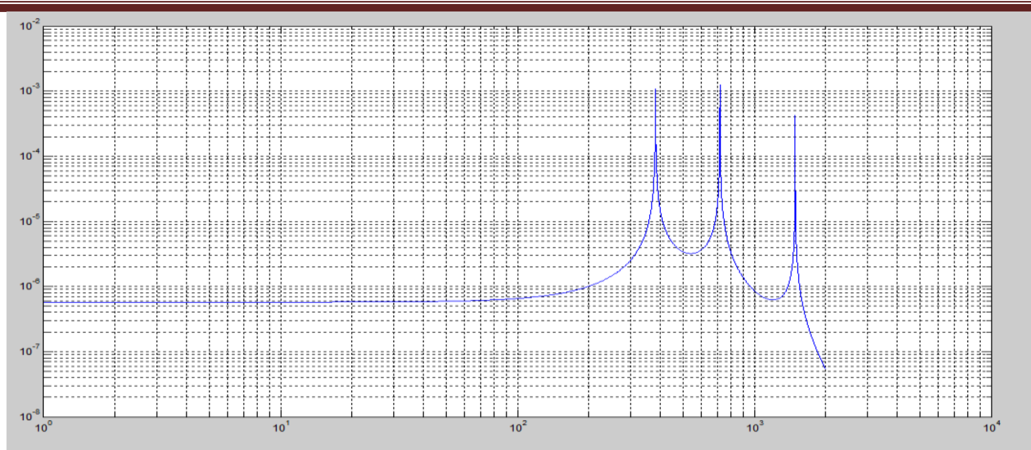


Figure 2 (b) Experimental physical response.

Inertance in db→



Frequency in Hz→

Figure 2 (c) Simulated FRF.

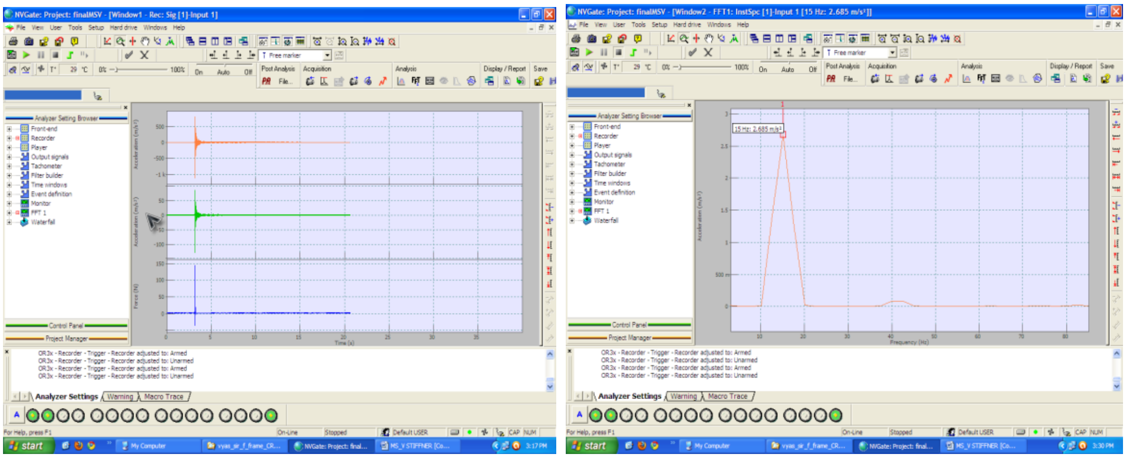


Figure 3 Experimental recorder and natural frequencies plots for 'F'-structure with vertical stiffener (Aluminum).

Conclusion:

Figure 2 (a) & 2(b) Simulation, experimental physical response and FRF for 'F'-structure with vertical stiffener (Aluminum). It is important to notice that now node three (3) shows negative response likes nodes 4 and 5. Response of node 2 is reduced with compared to node 1. Experimental modal analysis gives maximum response of 302.3×10^{-6}m at node 1 where as MATLAB simulation shows 300×10^{-6}m response at the same node which is exact what we required.

Simulation of modal analysis is showing very close results which indicate that FEA modeling of a modal model is behaving and responding like a real structure and predicting the dynamic properties in very close span of actual result. As we have no practical data of FRF we have no comparisons with prediction.

References:

For journal papers:

1. S.V.Modak et al. Prediction of dynamic characteristics using updated finite element models .Journal of Sound and Vibration (2002) 254(3), 447-467.
2. S.V.Modak et al. Use of an updated finite element model for dynamic design. Mechanical System and Signal Processing (2002) 16(2-3), 303-322.
3. J.S.Saini. Finding the Natural Frequencies and Mode Shapes for Vibration Analysis of Upright Drilling Machine. IE (I) Journal-MC, Vol 86, October 2005.

For books:

4. D.J.Ewins, Modal Testing: Theory, Practice and Application, Research studies press Ltd., 2000.
5. T.R. Chandrupatla, A.D.Belegundu, Finite Elements in Engineering, Pearson Education, 2005.
6. H.G.Vyas (Guided by prof.S.S.Pathan): Dynamic characterization of machine tool structures using simulated finite element model implemented in matlab and validation through experimentation. April - 2011(M.E.Dissertation)