IJIREEICE ISSN (O) 2321-2004, ISSN (P) 2321-5526

International Journal of Innovative Research in Electrical, Electronics, Instrumentation and Control Engineering

Impact Factor 7.047 ⸎ Vol. 10, Issue 4, April 2022

DOI: 10.17148/IJIREEICE.2022.10449

Design, Analysis and Simulation of Cantilever Beam Using Different Tools.

Shanmugavalli M[1], Priadharshini M[2], Sharmila Rani A[3], Swetha R[4]

Professor, Instrumentation and Control Engineering, Saranathan College of Engineering, Trichy, India[1]

Student, Instrumentation and Control Engineering, Saranathan College of Engineering, Trichy, India[2]

Student, Instrumentation and Control Engineering, Saranathan College of Engineering, Trichy, India[3]

Student, Instrumentation and Control Engineering, Saranathan College of Engineering, Trichy, India[4]

Abstract: The article concerns the vibration control of a cantilever beam using LabVIEW. The cantilever beam is designed using different software to compare the modal analysis of the beam. Four different types of software were used in this project. One is for experimentation using LabVIEW software with the help of myRIO tool kit. The other three are ANSYS, COMSOL and Intellisuite. The cantilever beam is designed using its dimensions and the frequency response for the applied changes is measured by Intellisuite software. The changes in the potential, stress mises and displacement occurred in the region of beam for the applied pressure is analysed using Intellisuite software. The design and analysis of cantilever beam for different Eigen Frequencies are done using COMSOL. By using this software the modal analysis of the cantilever beam is done. The comparison of modal analysis of the cantilever beam is done between ANSYS, COMSOL and Intellisuite. The vibration controlled in the cantilever beam using LabVIEW is explained in detail.

Keywords: Vibration Control, Intellisuite, LabVIEW, ANSYS, COMSOL.

I. INTRODUCTION

The control of flexible structures vibration is an important issue in industries. Many engineering applications required to maintain stability. The flexible materials having low rigidity and having very small damping ratio are susceptible to vibration. The main causes are imbalance, misalignment, wear and looseness. Vibration suppression has become one of the major issues for modern transportation. The active vibration control of smart structures has received a lot of attention in the field of vibration suppression. A smart active control system comprising of piezoelectric materials, signal conditioning circuits and the embedded hardware is proposed in this paper [1]. This paper deals with the use of Compact RIO control system developed by National Instruments as a control system for active vibration control of a test device. The Compact RIO control system is programmed using the LabVIEW software tool [2]. The design of the cantilever-based structure is done using Comsol Multiphysics and different parameters like deflection, stress, strain and voltage generated are analysed. The dynamic response is also obtained to analyse deflections under the application of dynamic pressure at different modal frequencies [3]. This study is focused on the presentation of the laboratory test model, designed for the experiments with active vibration control on the beam structure. . In the paper, there is also a part dedicated to the identification procedure of the laboratory model as well as the creation of corresponding mathematical model. At the end, the control algorithm is presented and evaluated in the simulation and on the real structure. [4]. In order to solve the problems of parameter optimization effectiveness and low control accuracy of traditional piezoelectric control method for rigid flexible hybrid manipulator, a new active piezoelectric control method for rigid flexible hybrid manipulator based on PSO is proposed in this paper. The experimental results show that the proposed method can effectively optimize the vibration parameters of the manipulator, and has high vibration control accuracy and strong practicability [5]. Active vibration control of thin plates using piezo-ceramic actuators is done. Paper describes method of identification of the laboratory model and also creation of approximation mathematical model together with basic algorithm which is designed the first four modes. As a result is presented good effect for first, third and four modes, and zero effect on second mode, which is probably caused by piezo-actuator position [6]. Active control strategies have attracted more and more attention because of the high adaptive capacity. However, during control, it is difficult to obtain the vibration signal of the cutting position of the work piece. In this paper, a modified Filtered-x Least Mean Square (MF x LMS) algorithm is constructed considering the deviation between sensor position and the cutting position of the work piece [7]. In allusion to the complex nonlinear vibration characteristics of the casting flash cutting machine, this paper was to analyse the vibration mode of the whole machine in the working process with the finite element software-ANSYS Workbench. It was found that the machine tool was severely deformed at the first, second and fifth frequencies. The excitation frequency of the machine tool had a certain

IJIREEICE ISSN (O) 2321-2004, ISSN (P) 2321-5526

International Journal of Innovative Research in Electrical, Electronics, Instrumentation and Control Engineering

Impact Factor 7.047 ⋇ Vol. 10, Issue 4, April 2022

DOI: 10.17148/IJIREEICE.2022.10449

difference from the 6th-order natural frequency value obtained from the modal analysis [8]. In this paper, modelling active vibration control for a thin mechanical structure as a cantilever beam with bonded piezoelectric patches is analysed. The transversal linear force produced by a linear piezo-actuator is replaced by the moment of force produced by the patch piezo-actuator, which is glued to the surface of the thin structure. Beam deflection measurement is replaced by measuring the elongation or shortening of the surface of the deformed beam. To analyse the effect of various controllers on vibration damping, a model using Matlab Simulink is created [9]. In this article, a new two-dimensional piezoelectric actuator driven vibration stage is proposed and prototyped. A closed-loop control system is developed and proposed based on the LabVIEW program consisting of a data acquisition device and capacitive sensors. Machining experiments have been carried out to evaluate the performance of the vibration stage and the results show good agreement with the tooltip trajectory simulation results, which demonstrates the feasibility and effectiveness of the vibration stage for vibration-assisted micro milling [10].
In this paper, a cantilever beam is fixed at one end and it undergoes forced vibration. This beam is made of aluminium and piezo-electric sensors are fixed to measure the vibration created in the beam. It is controlled by using LABVIEW and myRIO tool kit and the output is displayed in a Digital Storage Oscilloscope where the input signal and filtered signal will display. By comparing the two signals the supressed amount vibration is calculated. In addition, this cantilever beam is designed in three soft wares to analyse and simulate the beam. It includes ANSYS which is Analysis System, Intellisuite to design and fabricate the cantilever beam to its dimensions, and COMSOL for modal analysis of the cantilever beam. This paper is about comparing the modal analysis, vibration control of the cantilever beam in different tools.

II. BLOCK DIAGRAM

The block diagram of vibration control in a cantilever beam is shown below. Fig. 1

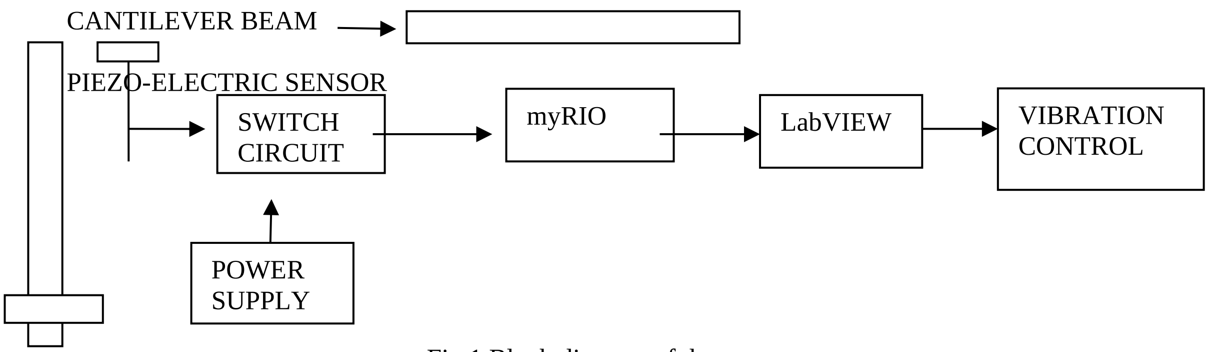

Fig.1 Block diagram of the system

III. VIBRATION CONTROL USING LABVIEW

In the LabVIEW platform the coding for vibration control of the cantilever beam is done. This software consists of two windows. One is for block diagram and the other one is front panel. In block diagram window usually the function and the logical parts are included. And in the front panel window it includes indicators and controllers for the requirement of the problem for programming. In our LabVIEW program, the front panel consists of various controllers like the Input signal and filtered signal waveform chart, indicators to indicate the value of the input signal and the filtered signal, two switches to reset the filter and to reset the signal, IIR specifications which consists of topology, type, PB ripple, SB attenuation, Lower Fc and Upper Fc. It also includes signal parameters to know the signal type and frequency of the signal. Sampling info is also included. The block diagram panel consists of the digital input module where the vibration from the cantilever beam is given as digital input to the LabVIEW program through the buffer circuit where 555 IC is used to boost up the minimum voltage from the piezo sensor. This digital input is involved in the program by using for loop where the reset signal, sampling info, signal parameters and IIR filter specifications are used. These are given into the case structure. The output of the filtered signal is taken as analog output and it is displayed in the module. There will be two signals are displayed in two different colours. The blue colour waveform indicates the input signal and the red colour waveform indicates the filtered signal. These are displayed in the graph between time and amplitude in x axis and y axis respectively. To know the difference between the input and filtered signal values, the data of the signals are exported to excel sheet. Where we can see the tabulation of amplitude and time values of input signal and filtered signal is displayed clearly. By right clicking the waveform chart in the front panel, there will be option for export. Click the export and select export to excel sheet. We can get the difference

IJIREEICE ISSN (O) 2321-2004, ISSN (P) 2321-5526

International Journal of Innovative Research in Electrical, Electronics, Instrumentation and Control Engineering

Impact Factor 7.047 ⋇ Vol. 10, Issue 4, April 2022

DOI: 10.17148/IJIREEICE.2022.10449

between the amplitude and time values of the signals are calculated. So this is how the vibration is controlled using LabVIEW programming.

IV. HARDWARE DESCRIPTION

1. DESIGN OF CANTILEVER BEAM

In this project a cantilever beam is made up of aluminium material and piezoelectric sensor is attached to it. This cantilever beam is fixed at one end to a stand set up and the other end is free. The following tabulation describes about the dimensions and material properties of the cantilever beam used in this experiment of vibration control.

DIMENSIONS AND MATERIAL PROPERTIES OF CANTILEVER BEAM

S. No	SYMBOL	PARAMETERS (with unit)	VALUES
1.	L	Length (m)	0.35
2.	B	Width (m)	0.025
3.	H	Thickness (m)	0.003
4.	P	Density (kg/m^3)	2700
5.	E	Young’s Modulus (N/m^2)	7.1*10^{10}
6.	V	Poisson’s Ratio (unit less)	0.34

2. PIEZO-ELECTRIC SENSOR

A piezoelectric sensor is a device which is used to measure the changes in the physical quantity such as (acceleration, temperature, force, etc.) into an electrical quantity (charge). In this project the piezoelectric sensor is fixed to the beam to sense the vibration occurred in the beam when it undergoes forced vibration. This signal is passed to the myRIO kit to store the data in it and then it is monitored by using LabVIEW programming. The dimension and material properties of the piezoelectric sensor is given below in the table.

DIMENSIONS AND MATERIAL PROPERTIES OF CANTILEVER BEAM

S. No	SYMBOL	PARAMETERS (with unit)	VALUES
1.	Lp	Length (m)	0.0765
2.	B	Width (m)	0.0127
3.	Ta	Thickness (m)	0.0005
4.	Ep	Young’s Modulus (Gpa)	47.62
5.	d 31	Piezoelectric strain constant (mV^{-1})	-247x10^{-12}
6.	g31	Piezoelectric stress constant (Vm N^{-1})	-9x10^{-3}
7.	P ρ	Density (kg/m^3)	7500

V. SOFTWARE DESCRIPTION

1. LABVIEW:

Install the Labview software. The installer is configured to install only the software components you need to get started. Restart your computer when prompted. Connect power to the myRIO device, and then connect the USB cable from the myRIO device to the computer. Allow 20 to 30 seconds for the computer to recognize the myRIO device. Launch the Getting Started Wizard and follow the prompts to install software on the device. Use the test panel to test the on-board devices. Connect I/O accessories and start using myRIO. The steps are shown below Fig.2

IJIREEICE ISSN (O) 2321-2004, ISSN (P) 2321-5526

International Journal of Innovative Research in Electrical, Electronics, Instrumentation and Control Engineering

Impact Factor 7.047 ⁖ Vol. 10, Issue 4, April 2022

DOI: 10.17148/IJIREEICE.2022.10449

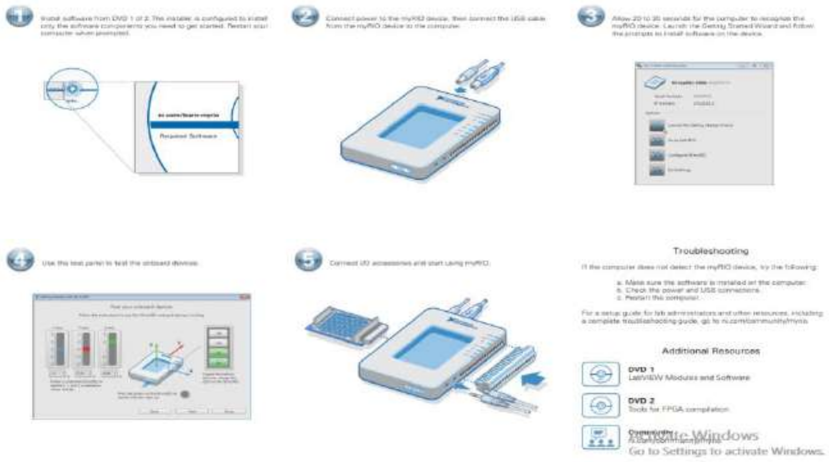

Fig.2 Procedure to connect myRIO

If the computer does not detect the myRIO device, try the following:

a. Make sure the software is installed on the computer.
b. Check the power and USB connections.
c. Restart the computer.

Once My Rio is configured, then configure the I/O port which is connected to the vibration sensor. Acquire the data from vibration sensor in LabVIEW using My Rio tool kit as shown below in Fig. 3

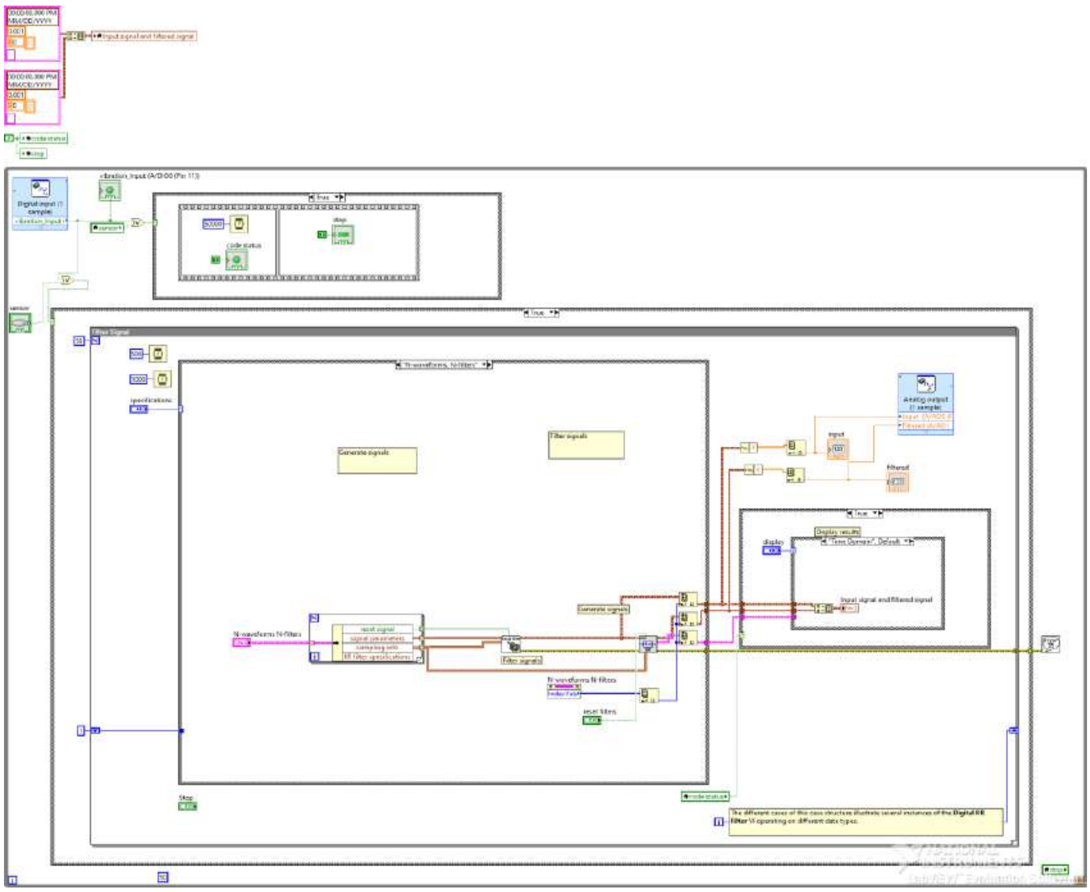

Fig.3 Code used in the LabVIEW using myRIO tool kit

ISSN (O) 2321-2004, ISSN (P) 2321-5526

International Journal of Innovative Research in Electrical, Electronics, Instrumentation and Control Engineering

Impact Factor 7.047 ⁜ Vol. 10, Issue 4, April 2022

DOI: 10.17148/IJIREEICE.2022.10449

After acquiring the data, it is sent as an input to waveform generator. By providing the signal parameters and sampling information, the waveform generator converts the Input data to waveform. Simple Array waveform generator is shown below in Fig.4

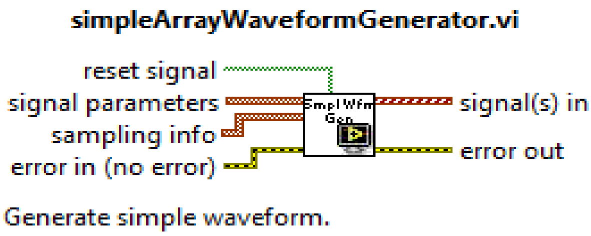

Fig. 4 Simple array waveform generator

By using IIR filter, the processed data is suppressed by using chebyshev filtering technique. The filtered output and the original input is segregated using index array and build tools and the output is projected as waveform as shown below in Fig.5

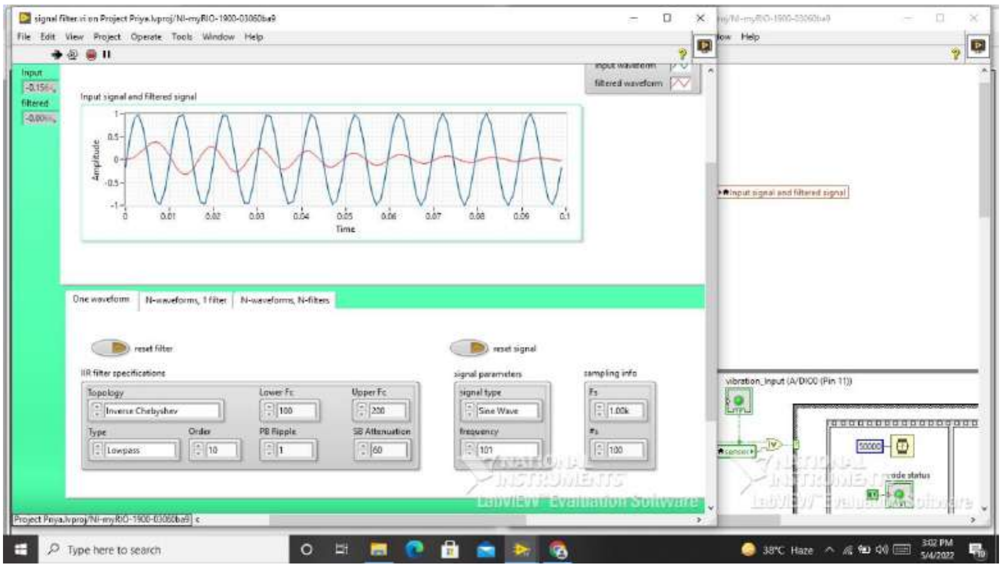

Fig. 5 The filtered output waveform using LabVIEW

2. INTELLISUITE

The design of cantilever beam using Intellisuite software is done when the pressure is applied to the beam and the corresponding displacement, stress and potential values of the cantilever beam for the applied pressure is displayed for all regions in the beam. To design the cantilever beam with piezo electric the following procedures are followed.

- In Intellisuite there will be a blueprint module. In this blueprint we are going to create geometry. Select 2 layers. Click layer 1 and choose specific colour and give coordinate as 0, 0 and 300,100 to create a rectangle and for layer 2 repeat the same coordinates.
- Close the blueprint and open the 3D builder module in Intellisuite. Here we are having two layers and click the each layer modify its height for our analyses. Then save the program in a folder.
- In the saved file that is TEM analysis window click the simulation and simulation setting and select piezoelectric and apply. Choose piezoelectric material in material window and apply stress to it by giving values 0.001, -0.001, 0.03 and click ok.
- To apply pressure, go to load and click the layer and give 1e-3 value to it. Now click the one side and fix the boundary and the other end is free.
- To mesh the model, click mesh and give value as 10 to maintain a very course mesh. Then go to analysis, click start static analysis.

IJIREEICE ISSN (O) 2321-2004, ISSN (P) 2321-5526
International Journal of Innovative Research in Electrical, Electronics, Instrumentation and Control Engineering
Impact Factor 7.047 ⋇ Vol. 10, Issue 4, April 2022
DOI: 10.17148/IJIREEICE.2022.10449

- To check the displacement values for the applied pressure, click result and choose displacement and select z axis. The displacement values are displayed for the applied pressure is observed in the following picture.

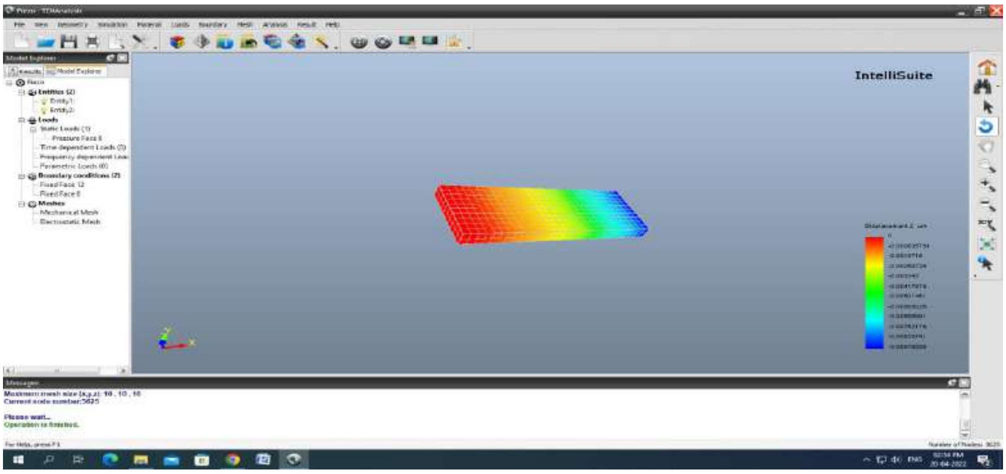

Fig.6 Changes in the displacement value for the applied pressure

- To see the potential generated for the applied pressure in the cantilever beam go to result, and click the potential to see the results. The following picture describes the potential generated in the beam for the applied pressure.

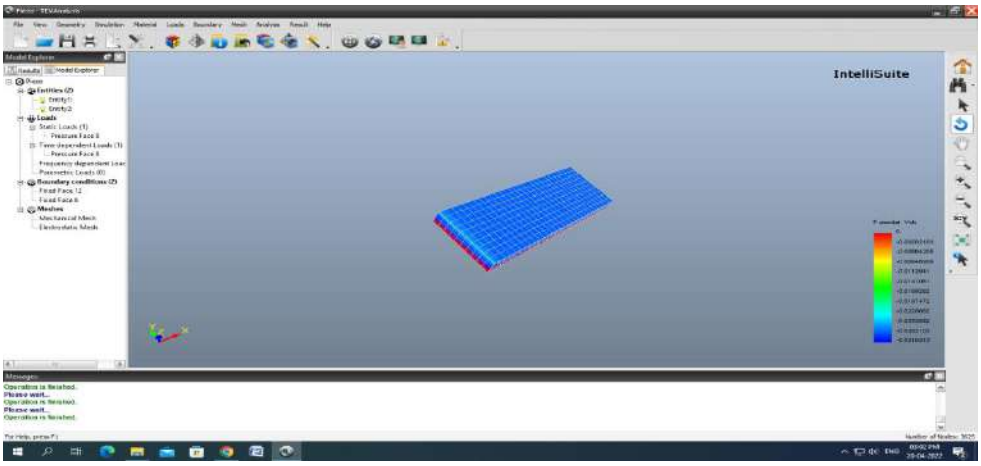

Fig. 7 Changes in the potential value for the applied pressure

To do the dynamic analysis, now select the simulation settings and select dynamic, stress displacement (direct integration) and for transient (fixed time increment) enter the time value as 0.1 second for every increment (100) and click ok. To apply piezoelectric material go to material property and define it. Go to load, to apply face pressure give the pressure value in the table and fix the boundary conditions. After completing the dynamic analysis in result choose the stress invariant to see the stress mises.

Impact Factor 7.047 ⁘ Vol. 10, Issue 4, April 2022

DOI: 10.17148/IJIREEICE.2022.10449

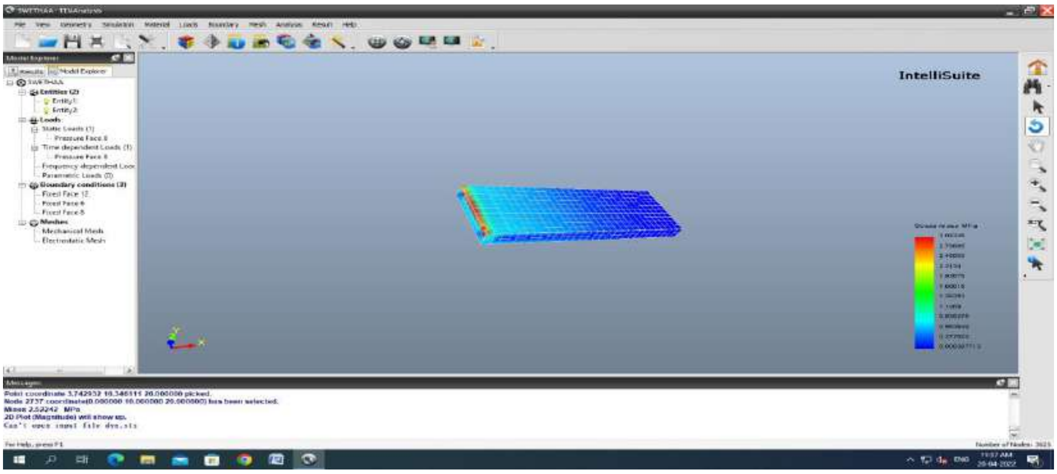

Fig. 8 Changes in the stress mises for the applied pressure

The stress is more in the area where the beam is fixed. To see the response of potential with time select the plot in the beam at different planar geometry. The changes in the displacement value for the applied pressure is displayed in Fig. 6 and changes in the potential value for the applied pressure over the beam is shown in Fig. 7. The changes occurred in the stress mises in all over the region of the beam is shown in Fig. 8. This is how the simulation is done for a cantilever beam with piezo electric using Intellisuite.

3. ANSYS:

Ansys is a 3D design platform which is used for engineering simulation. ANSYS refers to analysis system. By using Ansys we can get the structural analysis software solutions that enable engineers to solve complex structural engineering problems faster and more efficiently. Many kind of analysis are done by using this software like harmonic analysis, modal analysis, etc. In this paper we have done the modal analysis for the cantilever beam.
Modal analysis is the study of the dynamic properties of the system in the frequency domain. It is process of testing structures under vibrational excitation.

- First Open Mechanical APDL in Ansys then click on Preferences in Main Menu after that Preference for GUI Filtering will open. In that click Structural then click OK. Click Pre-processor then Element type and then click Add/ Edit/ Delete. It will open Element type in that click None defined, Add it will open one box of Library of Element Types in Structural Mass click Beam and 2 node 188 then click OK then close Element type.
- Now click Material Props in that click Material Models, define Material Model Behaviour box will show in that click Structural than linear, Elastic, Isotropic. Here we are doing Ansys Modal analysis for Cantilever beam so give the Beam dimensional details. Then in the Rectangular box give Young's Modulus (EX) value as 7.1*1010 and then Poisson's ratio (PRXY) as 0.3 and click OK. Density there, selects on density, give Cantilever beam density (DENS) value as 2700 and click OK and close the box.
- Now go to Sections in Material Props in that select beam then go to Common Sections, give width (B) 0.025 and Thickness (H) value as 0.003 and click OK. Here every measurement values are metre (m) unit. Now go to Modelling then Key points in that click In Active CS Create Key-points in Active Coordinate system will show up in that simply give apply because key-points were generated at the origin and in that X, Y, Z Location in active CS give 1 in first box which is the distance of one metre then click OK. X, Y, Z points on the different places on the Ansys workbench will show up.
- Now go to lines then Straight lines. In Ansys workbench window click first point X and second point Y then click OK in the left bottom box. Now go for meshing then click mesh tool then click mesh on Right side box then click OK on left side box again do the same as above click mesh then select the X and Z line then click OK after that click raise hidden which is in the above in that click close.
- Now go to Loads then Define Loads, Apply in that click Structural then go to Displacement then click on nodes then click OK on the left side box. After that Apply U, ROT en Nodes will show up in that click All DOF (degree of freedom) then click OK. Now go to Solution, click Analysis type then click New Analysis, select Modal, click OK. Now go to Analysis Options in that No. of modes to extract give 5 and No. of modes to expand give 5 then click OK. Now Block Lanczos Method will show up in that click OK.

IJIREEICE ISSN (O) 2321-2004, ISSN (P) 2321-5526

International Journal of Innovative Research in Electrical, Electronics, Instrumentation and Control Engineering

Impact Factor 7.047 ⁜ Vol. 10, Issue 4, April 2022

DOI: 10.17148/IJIREEICE.2022.10449

- Now go to solution then solve in that click current LS, click OK. Now Solution is done will pop up then click close. Now go to General Postproc there select on Read Results then click First set and now go to plot result then go to Deformed Shape in that already Def. shape only is selected then click OK. In Ansys workbench it will show the first displacement response and the frequency value. In this picture you can see the light difference in the line which is the displacement in Fig. 9.

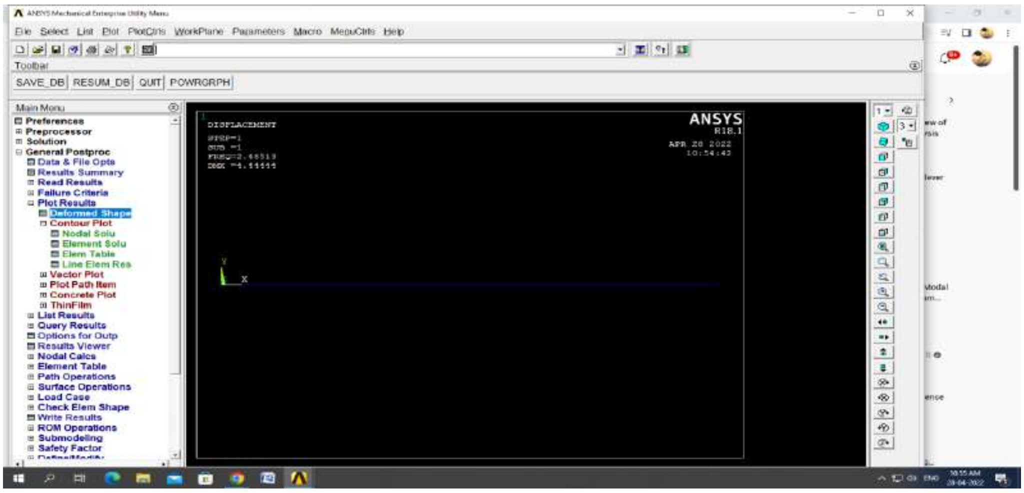

Fig.9 The response for the first mode

For 2, 3, 4, 5 set of response values do the same as go to Read Results then click Next Set ,go to Plot results select Deformed shape there select Def+undeformed and then click OK. There you can see the second set of response. You can see the variation in the frequency value by checking all the sets of values.

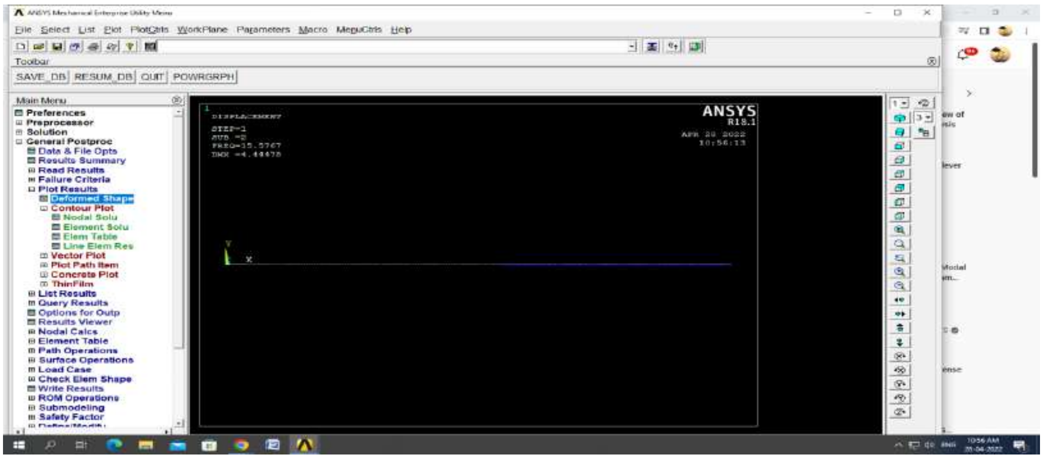

Fig. 10 The response for the second mode

IJIREEICE ISSN (O) 2321-2004, ISSN (P) 2321-5526

International Journal of Innovative Research in Electrical, Electronics, Instrumentation and Control Engineering

Impact Factor 7.047 ⋊ Vol. 10, Issue 4, April 2022

DOI: 10.17148/IJIREEICE.2022.10449

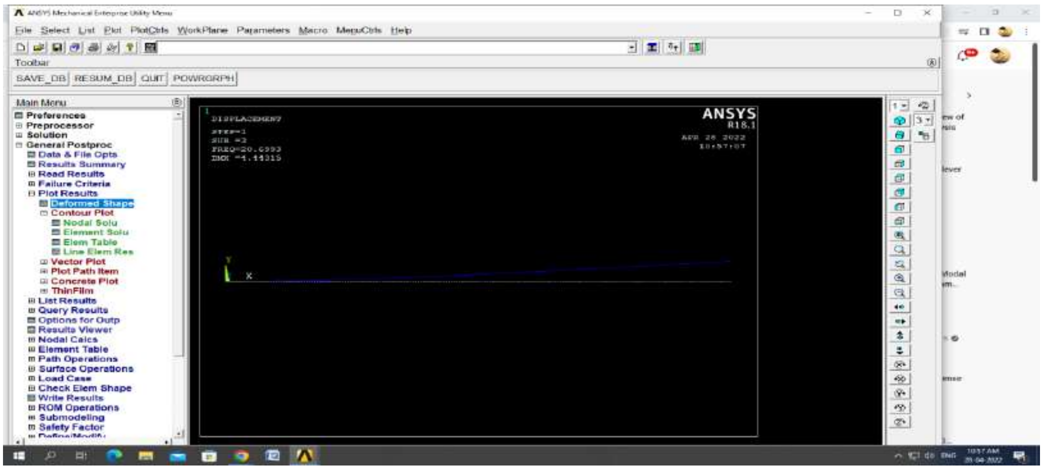

Fig.11 The response for the third mode

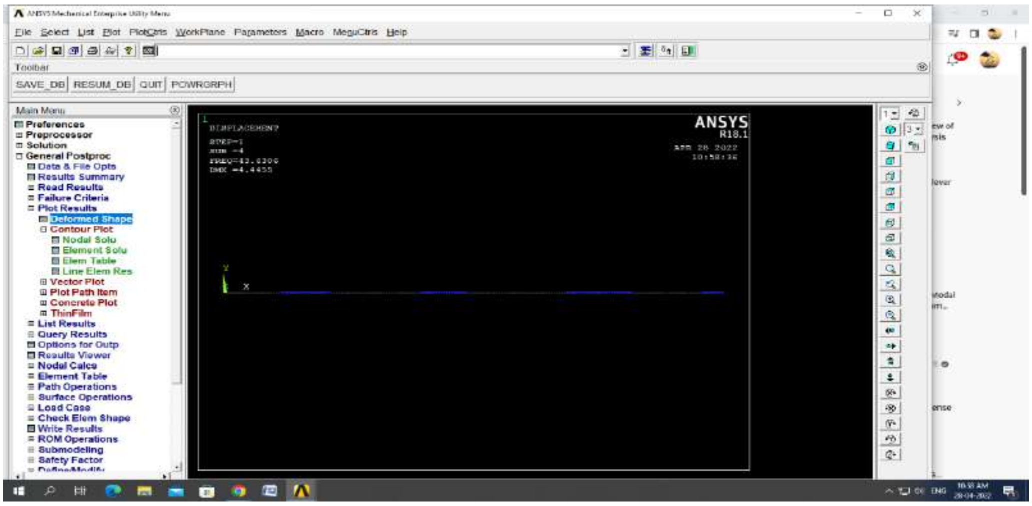

Fig. 12 The response for the fourth mode

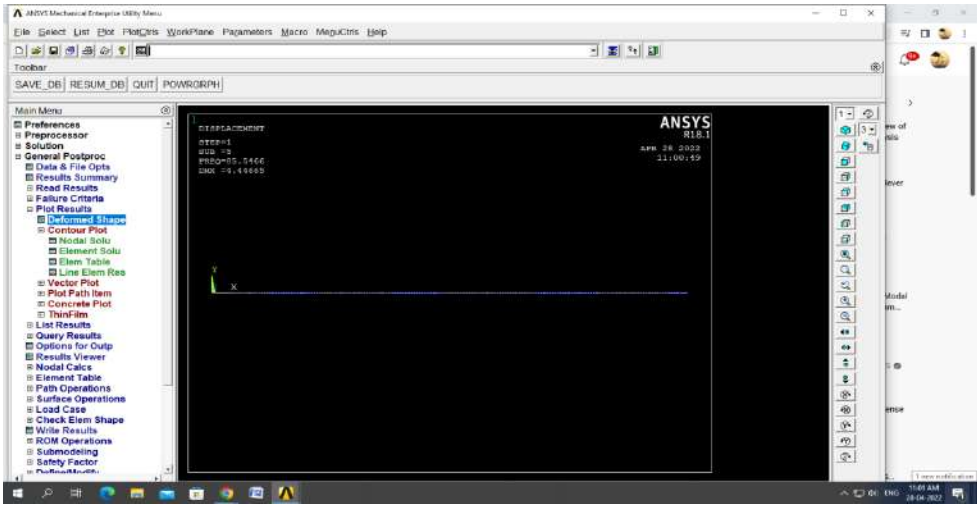

Fig.13 The response for the fifth mode

IJIREEICE ISSN (O) 2321-2004, ISSN (P) 2321-5526

International Journal of Innovative Research in Electrical, Electronics, Instrumentation and Control Engineering

Impact Factor 7.047 ⋇ Vol. 10, Issue 4, April 2022

DOI: 10.17148/IJIREEICE.2022.10449

Thus the modal analysis of the cantilever beam is done for our required dimensions and material properties. The given photographs are the modal analysis of the beam for the given 5 modes and their responses are displayed. For the first mode the frequency value is 2.485 Hz is shown in Fig. 9. For the second mode the response gives 15.576 Hz is shown in Fig. 10. For the third mode, the frequency will be around 20.699 Hz is displayed in Fig. 11. In Fig. 12, the response for the fourth mode is shown where the frequency is 43. 630 Hz. For fifth mode the response is shown in Fig. 13 where the frequency is 85.546 Hz.

4. COMSOL

COMSOL is software which is used in all field of engineering to simulate the designs of devices and processes. COMSOL multiphysics a platform for simulation that provides single-physics and multi-physics modelling.
By using this software here the structural analysis of cantilever beam is done. The design and analysis of cantilever beam for different Eigen frequencies is done. In this analysis we will be using a horizontal bar which is made up of Aluminium having the following dimensions. Length (L) 0.35 m, Width (B) 0.025m, and Thickness (H) 0.003m. The cantilever beam is designed using the given dimensions by following procedure.

- First go to the model result. In this we have to select 3D model. In the 3 D model go for structural mechanics following by solid mechanics. To add the physics interface double click the solid mechanics and engage it to study of Eigen frequency. Then click on done and then save the file.
- The interface will open where we can have model builder, settings and graphics respectively. In the settings, change the length unit to micro meter. In model builder there will be global definitions. By right clicking the global definition a dialog box will open. Select parameters and then in settings enter the dimensions and its expression and value for our cantilever beam. Fill the tabular column by giving name as Beam L and its expression as 350000um, Beam W with 25000um and Beam H with 3000um.
- Now in the global definition we have the geometry. Right click on geometry and select the block. In the block section change the size and shape quantities. For width enter Beam W, for depth Beam L and Beam H for height. And finally click on build all objects.
- Then to add materials go to material tab and click add material. In add material column there will be a list of materials. Select aluminium material in built-in option and then click on add to selection and close the material page. Now go to material in model builder. There will be two layers of material is added. By right clicking it remove one aluminium layer and allow only first material to be added. Then in selection we can see the entire material is selected.
- Then in model builder select the solid mechanics. By right clicking it choose fixed constraint. In the fixed constrain select the phase 2. To proceed on meshing, in the model builder by right clicking the mesh 1 we can select the free tetrahedral and select size. In the size we have to set the element size as normal in predefined column.
- To build the entire mesh click build all and as a result the entire mesh is seen in the graphical window. Following which go to the Eigen frequency of the model builder and R click to study and click compute. So, thus we are able to observe the Eigen frequencies and linear displacement of analysis of cantilever beam.
- For result analysis, we can see the data sets in result. In the result click Mode shape (solid) and surface and then deformation. The actual deformation occurred in the cantilever beam is displayed in the graphical window with the Eigen frequency value of 20.258 and the total displacement occurred in the surface in um is shown below in Fig. 14. Thus the design and analysis of cantilever beam is done using Comsol.

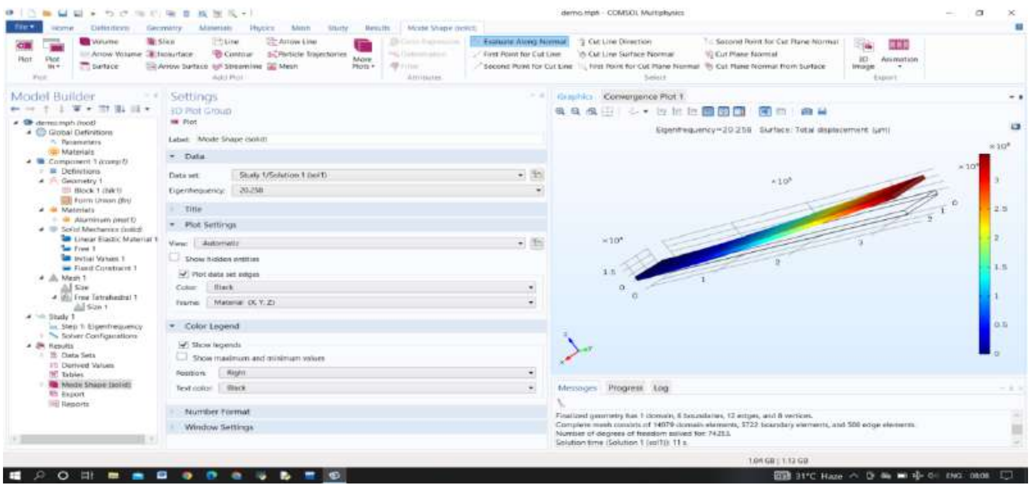

Fig 14 Modal analysis of cantilever beam using Comsol

VI. EXPERIMENTAL SETUP / HARDWARE PROTOTYPE

The below figure depicts the hardware prototype that has been developed to control the vibration from the cantilever beam. The process is done using the below experimental setup in Fig. 15.

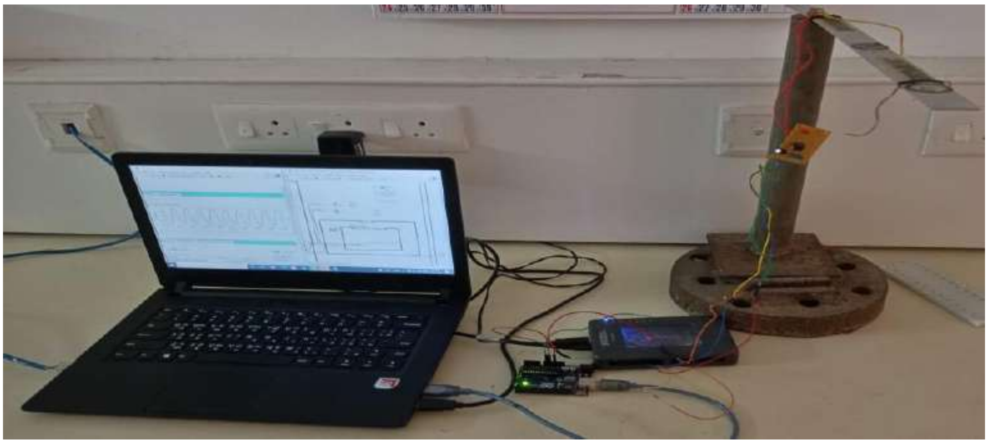

Fig.15 Experimental Setup of vibration control using cantilever beam

VII.PROCESS DESCRIPTION

In the LabVIEW platform the coding for vibration control of the cantilever beam is done. This software consists of two windows. One is for block diagram and the other one is front panel. In block diagram window usually the function and the logical parts are included. And in the front panel window it includes indicators and controllers for the requirement of the problem for programming. In our LabVIEW program, the front panel consists of various controllers like the Input signal and filtered signal waveform chart, indicators to indicate the value of the input signal and the filtered signal, two switches to reset the filter and to reset the signal, IIR specifications which consists of topology, type, PB ripple, SB attenuation, Lower Fc and Upper Fc. It also includes signal parameters to know the signal type and frequency of the signal. Sampling info is also included. The block diagram panel consists of the digital input module where the vibration from the cantilever beam is given as digital input to the LabVIEW program through the buffer circuit where 555 IC is used to boost up the minimum voltage from the piezo sensor. This digital input is involved in the program by using for loop where the reset signal, sampling info, signal parameters and IIR filter specifications are used. These are given into the case structure. The output of the filtered signal is taken as analog output and it is displayed in the module. There will be two signals are displayed in two different colors. The blue color waveform indicates the input signal and the red color waveform indicates the filtered signal. These are displayed in the graph between time and amplitude in x axis and y axis respectively. To know the difference between the input and filtered signal values, the data of the signals are exported to excel sheet. Where we can see the tabulation of amplitude and time values of input signal and filtered signal is displayed clearly. By right clicking the waveform chart in the front panel, there will be option for export. Click the export and select export to excel sheet option. We can get the difference between the amplitude and time values of the signals are calculated.

VIII.RESULT

The Cantilever beam is designed, developed using different software such as Intellisuite, Ansys, Comsol, and the experimentation is done by using Lab View where the vibration of the cantilever beam is controlled and their frequency response for the Modal Analyses are calculated. The frequency range is around 20.235 Hz in Intellisuite when the beam is simulated and designed. In Ansys, the modal analysis of the beam having five modes is displayed with the frequency range of 2.485 Hz for first mode, 15.576 Hz for the second mode, 20.699 Hz for the third mode, 43.630 Hz for the fourth mode and finally, 85.546 Hz for the last mode. The corresponding changes in the shape are also displayed in the Ansys. In Comsol the frequency range of cantilever beam for the given dimension is 20.258 Hz. In LabVIEW the input signal is sensed in the range of 0.156 Hz and the controlled vibration signal frequency is about 0.006 Hz. The change in the vibration signal is also displayed in excel sheet for the continuous signal of vibration. The difference between the ranges of frequency from the controlled vibration is compared between software used in this project and the values are tabulated below.

Parameter	Intellisuite	Ansys	Comsol
Modal Analysis (Hz)	20.235	2.485 15.576 20.699 43.630 85.546	20.258

XI.CONCLUSION

As a conclusion, the vibration of the cantilever beam is controlled using LabVIEW and the amplitude and time is displayed in excel sheet for the input and filtered signal. Also the cantilever beam is designed and analysed by different tools such as Intellisuite, Comsol and Ansys. In Ansys software, for modal analysis we extracted five modes and corresponding frequency in Hz (2.485, 15.576, 20.699, 43.630, and 85.546). We get 20.258 Hz for modal analysis in Comsol software. Respectively in Intellisuite software we get 20.235 Hz for modal analysis. The above values are analysed and tabulated. Thus the vibration is controlled from
-0.1564 to -0.0066 in experimentation.

REFERENCES

1. Smart Active Vibration Control System Using Piezoelectric Materials Mao Cui; Wei Tang; Yuying Han; Ze Li 2020 Chinese Control and Decision Conference (CCDC) Year: 2020 | Conference Paper | Publisher: IEEE
2. CompactRIO control system for active vibration control Miroslav Pawlenka; Miroslav Mahdal 2018 19th International Carpathian Control Conference (ICCC) Year: 2018 | Conference Paper | Publisher: IEEE
3. Design and Analysis of Piezoelectric Cantilever Based Vibration Sensor S Bhuvana; Hoode Prathiksha; VT Sindhu; Hegde Vasudha 2018 IEEE International Conference on System, Computation, Automation and Networking (ICSCA) Year: 2018 | Conference Paper | Publisher: IEEE
4. Experimental Model for Active Vibration Control on Lattice Structure Pavel Šuránek; Srikanth Sreethar; Radek Štramberský 2021 22nd International Carpathian Control Conference (ICCC) Year: 2021 | Conference Paper | Publisher: IEEE
5. Piezoelectric active vibration control method of rigid flexible hybrid manipulator based on PSO Meixuan Li 2021 13th International Conference on Measuring Technology and Mechatronics Automation (ICMTMA)Year: 2021 | Conference Paper | Publisher: IEEE
6. Thin Plate Active Vibration Control Vaclav Pavelka; Pavel Suranek; Radek Strambersky 2021 22nd International Carpathian Control Conference (ICCC) Year: 2021 | Conference Paper | Publisher: IEEE
7. Active Vibration Control of Thin Plate Milling Using Piezoelectric Actuator Chentong Cai; Tong Zhao 2021 4th World Conference on Mechanical Engineering and Intelligent Manufacturing (WCMEIM) Year: 2021 | Conference Paper | Publisher: IEEE
8. Vibration Mode Analysis of Casting Flash Cutting Machine Fan Yang; Yunfeng Pan 2020 5th International Conference on Mechanical, Control and Computer Engineering (ICMCCE) Year: 2020 | Conference Paper | Publisher: IEEE
9. Design, Analysis, and Control of a Two-Dimensional Vibration Device for Vibration-Assisted Micromilling Lu Zheng; Wanqun Chen; Dehong Huo; Xiaoyu LyuIEEE/ASME
10. Transactions on MechatronicsYear: 2020 | Volume: 25, Issue: 3 | Journal Article | Publisher: IEEE
11. Modeling the Use of the Patch Piezo-Actuators for Active Vibration Control Jiri Tůma; Radek Štramberský; Václav Pavelka 2021 22nd International Carpathian Control Conference (ICCC) Year: 2021 | Conference Paper | Publisher: IEEE

