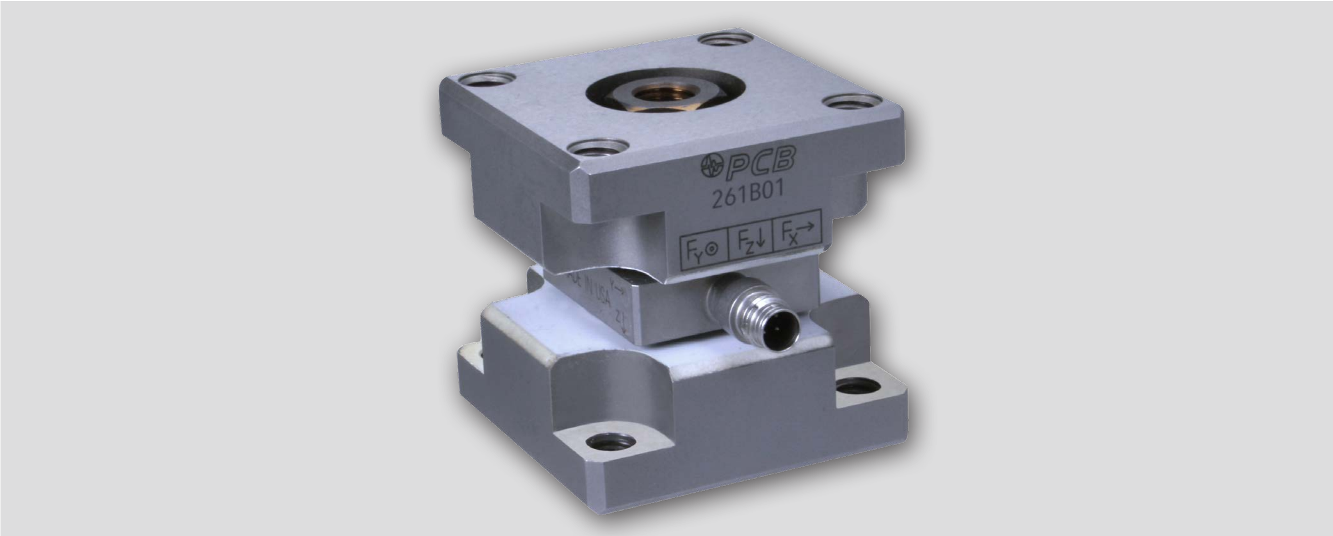
PCB PIEZOTRONICS
AN AMPHENOL COMPANY



Improved Stiffness Specifications for Piezoelectric Force Links

Written By

David Ort, Melissa Maze, and Jeff Dosch
PCB Piezotronics, Inc., Depew NY, USA

Improved Stiffness Specifications for Piezoelectric Force Links

David Ort, Melissa Maze, and Jeff Dosch
PCB Piezotronics, Inc., Depew, NY, USA

Contents

Abstract

When engineers integrate piezoelectric force link sensors into test systems, such as in force limited testing of satellites, the force link becomes a critical part of the system's overall dynamic performance. During the initial development of a test, it is often necessary to predict the dynamic response of the complete test system. An accurate and consistent method to make response predictions is essential and best performed with Finite Element Analysis (FEA). However, historically, the statements of force link stiffness provided by sensor manufacturers have not been practical for incorporation into FEA models.

PCB Piezotronics, Inc. has recently updated its force link specifications to include a complete set of stiffness coefficients that can be used to effectively represent the force link in a finite element model of a complete test system. This paper first defines the physical meaning of the stiffness coefficients and the FEA method used to determine them. Physical testing is shown to build confidence in the determined stiffness values. The usefulness of the updated stiffness coefficients is demonstrated by a modeling example of a basic force platform of the type used in a satellite testing. Finite element analysis is performed on this system, using a number of different methods to incorporate the influence of the force link's stiffness.

Introduction

Force links are measurement devices comprised of one piezoelectric 3-component force ring installed between two stainless steel mounting plates (Figure 1). The force link assembly is secured together via an elastic beryllium-copper stud under a specific preload value relative to the force sensor's full scale output. Applying the recommended preload provides several benefits: it tightly clamps the internal components, piezoelectric material, and housing together to obtain the best possible non-linearity (≤1%), allows for both tension and compression measurements in the Z-axis, and provides sufficient friction between components to transmit shear forces in the X and Y-axes.

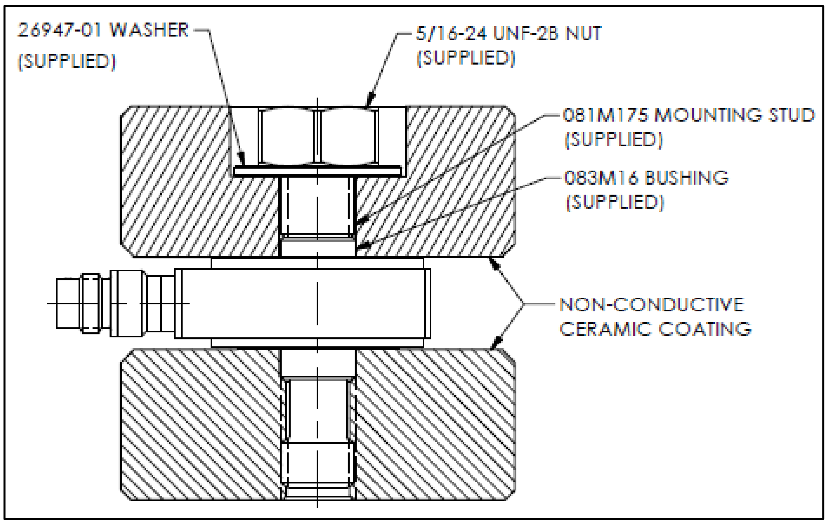


Figure 1: Force link construction of [PCB Model 261B01]

The link construction allows for convenient and immediate installation into a test fixture, simplifying a test setup by eliminating the need for on-site preloading. Force links offer a wide dynamic measurement range up to 10k lbs. (Z) full scale output, and outputs of multiple sensors can be summed to accommodate larger test needs. The link material and assembly result in the device having high stiffness, which is an important characteristic to consider when developing a system model to predict and understand dynamic response, as the presence and characteristics of the force link (or any measurement device) can affect the response of a test article.

Some of the most common force applications that can benefit from the use of force links include: force limited vibration (FLV) testing of satellites and subcomponents [1], bird strike testing, biomechanical studies, cutting tool forces and machine monitoring, reaction gyroscopes, vehicle dynamics, impact testing, and force plates. For FLV testing, multiple force links are used between the shaker and unit under test (UUT) to measure the input force from the shaker into the UUT (Figure 2). The summed force relates directly to the quasi-static acceleration of the structure's center of gravity. By using the vibration shaker's control loop, the force signal is compared to established force limits in the controller, and at frequencies where the measured force exceeds the limit (at high Q resonances), the controller output (i.e., the input signal to the shaker) is "notched." By reducing the input to the shaker at these frequencies, the reaction force between the test fixture and structure is maintained within specified limits and protects the UUT from over testing.

Force Limited Vibration Testing: NASA-HDBK-7004C [1] speaks to the importance of using piezoelectric 3-component force sensors paired with shaker controllers that offer real-time response limiting. The handbook also highlights the importance of understanding the complete test system, including UUT, fixtures, and sensors; proper modeling and simulation of all components must be conducted to fully understand the dynamics of the test article, specifically if any test fixtures or force sensors have an influence on response. We will next discuss the parameters of force links (i.e., stiffness coefficients) that must be considered for proper modeling, how the values were determined for our 261 series force links, and how to apply these parameters in FEA modeling.

Definition of Stiffness Coefficients

The force link's stiffness can be expressed as a 6x6 matrix with only five independent stiffness coefficients. These five coefficients are what is published on PCB Piezotronics' force link's data sheet [3]. The definition of these coefficients and their relation to the stiffness matrix is described here.

Figure 2 shows a cross section view of a typical force link. Its stiffness can be attributed to several materials arranged in a complex geometry. For the purposes of modeling and defining stiffness, we will simplify the geometry to a beam element with a node at the center of each mounting surface. Each node has six DOF (degrees of freedom). See Figure 2 for orientation of the beam element in relation to the force link and for definition of the nodal displacements.

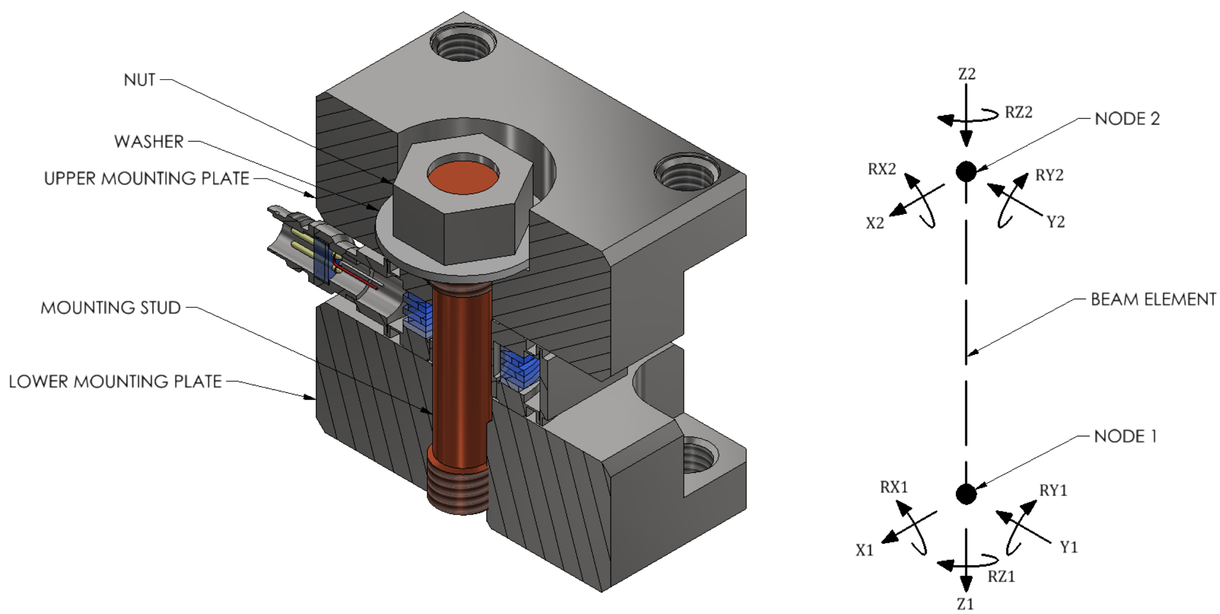


Figure 2: PCB Model 261B01 force link cross section and beam element

The beam element stiffness is characterized by its deflection, which is the difference between the nodal displacements. Equation 1 defines the three translational and three rotational deflections.

$$
\begin{Bmatrix} u_X \\ u_Y \\ u_Z \\ \theta_{RX} \\ \theta_{RY} \\ \theta_{RZ} \end{Bmatrix} = \begin{Bmatrix} \mathrm{X1} - \mathrm{X2} \\ \mathrm{Y1} - \mathrm{Y2} \\ \mathrm{Z1} - \mathrm{Z2} \\ \mathrm{RX1} - \mathrm{RX2} \\ \mathrm{RY1} - \mathrm{RY2} \\ \mathrm{RZ1} - \mathrm{RZ2} \end{Bmatrix}
$$

Equation 1: Beam Deflection

The stiffness matrix's symmetry about the main diagonal and the large number of coefficients with zero value are the result of the force link's relatively high symmetry about the Z axis and from top to bottom. Because of this high degree of symmetry, the simplified model described by Equation 2 has a stiffness matrix containing only five independent stiffness coefficients. The nomenclature for the coefficients is provided in Table 1 and is the same nomenclature used on PCB Piezotronics' published data sheets [3]. Note the minus sign preceding one of the K_C terms in Equation 2. Only the magnitude of K_C is provided on the data sheet, and care must be taken to provide the correct sign when inputting values for analysis.

Table 1: Stiffness coefficients

Name on specification sheet	Symbol
Stiffness (X or Y axis)	K_T
Stiffness (Z axis)	K_A
Stiffness (RX or RY)	K_{RT}
Stiffness (RZ)	K_{RA}
Coupled stiffness (X-RY or Y-RX)	K_C

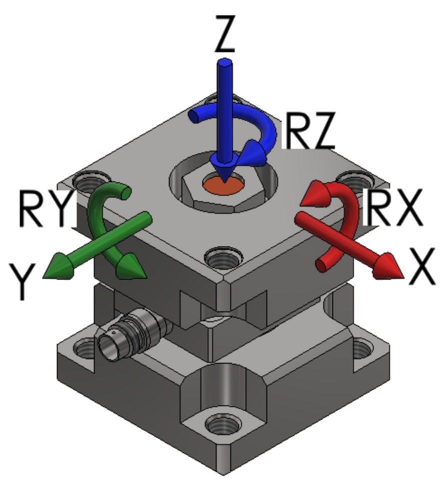


Figure 3: Coordinate system

Translational

$$
\begin{Bmatrix} F_x \\ F_y \\ F_z \\ M_{RX} \\ M_{RY} \\ M_{RZ} \end{Bmatrix} = \begin{bmatrix} K_T & - & - & - & - & - \\ 0 & K_T & - & - & - & - \\ 0 & 0 & K_A & - & - & - \\ 0 & K_C & 0 & K_{RT} & - & - \\ -K_C & 0 & 0 & 0 & K_{RT} & - \\ 0 & 0 & 0 & 0 & 0 & K_{RA} \end{bmatrix} \begin{Bmatrix} u_X \\ u_Y \\ u_Z \\ \theta_{RX} \\ \theta_{RY} \\ \theta_{RZ} \end{Bmatrix}
$$

Translational / rotational coupled

Rotational

Equation 2: Equivalent stiffness model

Method to Determine Stiffness Coefficients and Physical Testing

Determination of Stiffness Coefficients

The stiffness values on PCB's updated datasheets are determined from a detailed finite element model. Here we show the process of extracting values from the model using the 261B01 force link as an example. The same process is followed for other force link models.

Figure 4 shows the detailed geometry used to determine the stiffness coefficients for the model 261B01 force link. This finite element model is a fully coupled piezoelectric-structural simulation that includes the anisotropic material properties. To mimic the nodes of the beam model, the boundary conditions are applied to the upper and lower mounting faces through remote points. This method uses constraint equations to tie the finite element's nodes from a face to a single point. The lower point has all DOF fixed. The upper point has a small deflection/rotation applied in each DOF, one at a time, while holding all other DOF at zero (Figure 5). With applied deflection, output of the analysis is the resulting forces and moments. The stiffness coefficients are calculated as force and moment reactions divided by applied deflection. When a deflection in the X or Y direction is applied, both a force and moment reaction is generated. This creates the coupled stiffness coefficient.

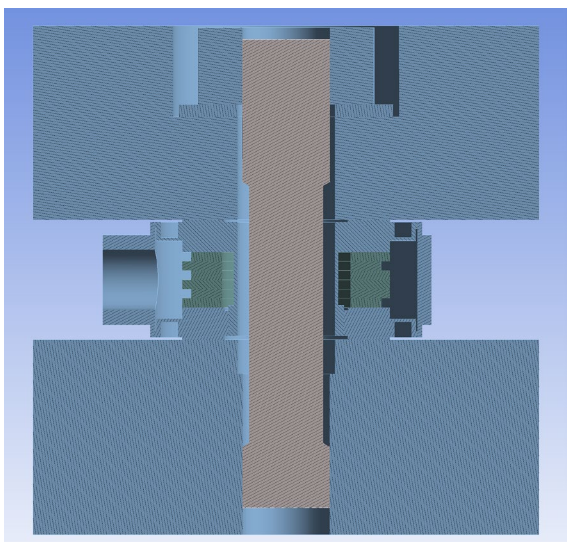
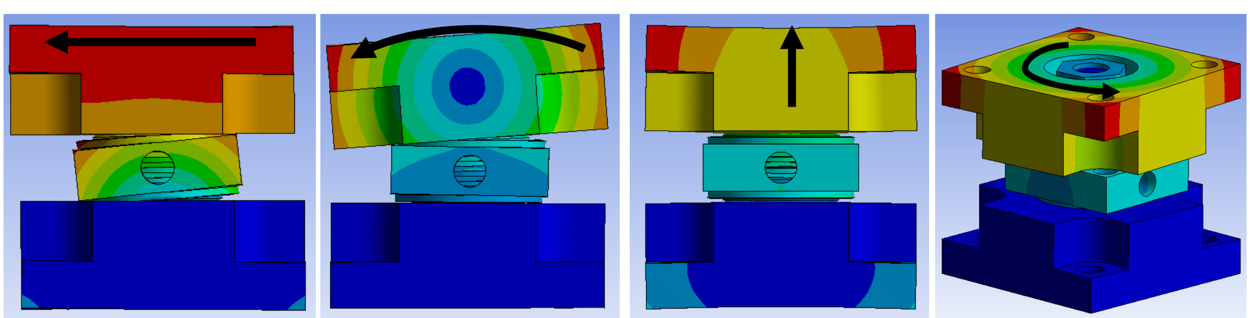


Figure 4: Cross section of force link model



Transverse - translation   Transverse - rotation   Axial - translation   Axial - rotation

Figure 5: Deflection in each direction

Confidence in Stiffness Values

To build confidence in 261B01's finite element model, we compare measured and simulated values of the force link's natural frequencies. A close match between these frequency values is a good indication that the model will provide the correct stiffness coefficients. This is because the modal stiffness is directly related to the square of the natural frequency.

Natural frequencies were measured by impact of the sensor under free-free boundary conditions. During testing, we placed the device on a pad of soft foam to produce a response free of external support. Then we used a steel ball bearing to impact the structure along the axis of interest to excite its natural frequency. Sensor output is acquired from the axis corresponding to the impact direction (the 261B01 is a three-axis sensor with three outputs). Figure 6 shows the time domain response of the sensor (top graph) and the fast Fourier transform (FFT) of the signal (bottom graph). In this example, the sharp peak in the FFT easily identifies the natural frequency. We repeated this process for each axis.

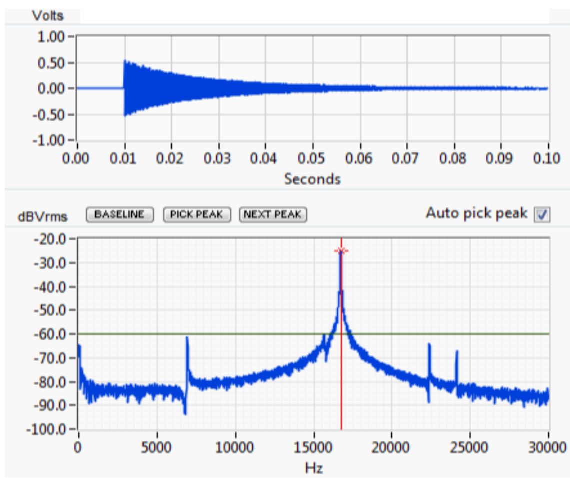


Figure 6: Natural frequency test data

The FEA model is a fully coupled piezoelectric-mechanical model that can provide voltage response from each of the sensor's three axes. Using the FEA model to perform modal analysis, with no mechanical boundary conditions applied to the structure, the voltage from each axis is exported for all of the modal frequencies (Figure 7). For each of the three sensor outputs, the frequency at maxiumum voltage can be compared to the natural frequency obtained by the impact response.

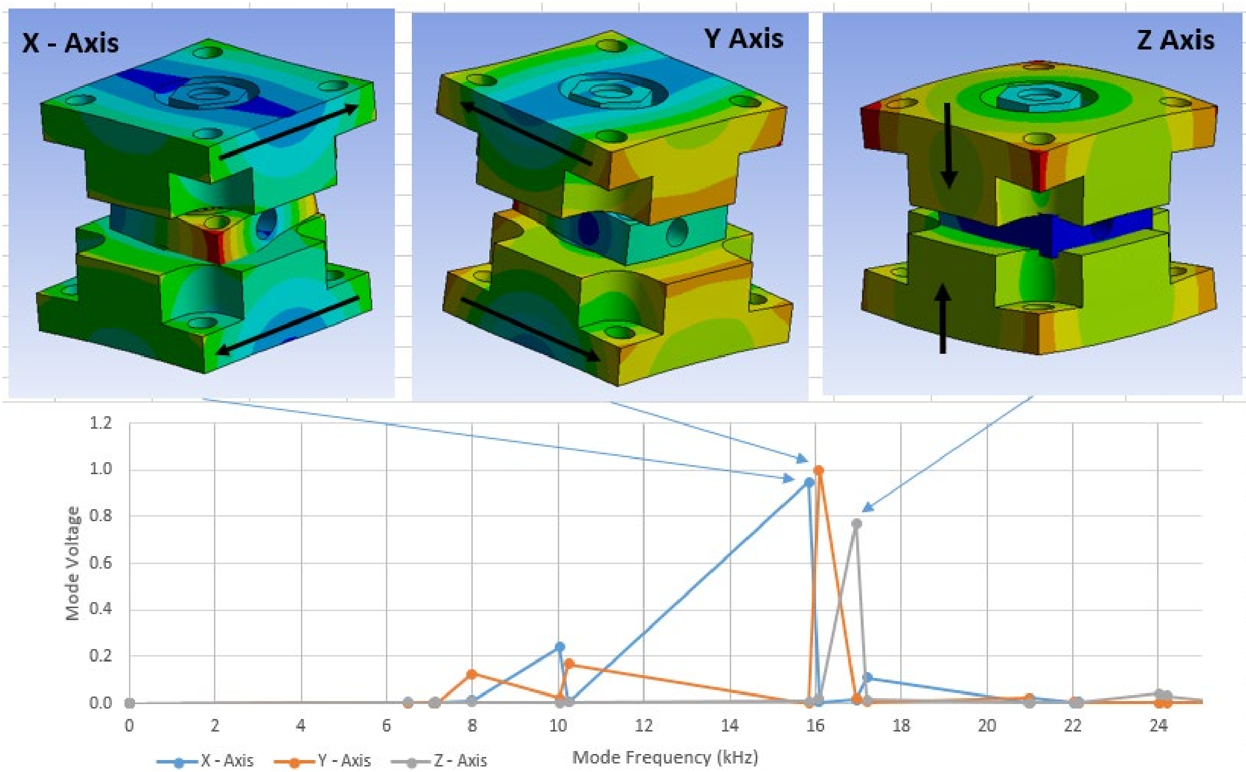


Figure 7: Voltage vs. mode frequency (bottom) and mode shapes of maximum voltage (top)

Table 2 compares the measured and simulated natural frequencies. The measured and predicted values closely match, providing confidence that the stiffness coefficients generated by the FEA model are reasonable.

Table 2: Comparison of Measured and Model Natural Frequencies

Axis Direction	Measured (kHz)	FEA Model (kHz)	% Difference
X	15.39	15.86	3.1%
Y	15.71	16.09	2.4%
Z	16.73	16.96	1.3%

Example Analysis of a Test System Including Force Link Stiffness

In this section, the predicted response of a force limiting vibration table demonstrates the use of the updated stiffness specifications in analysis. Four approaches to incorporate the influence of the force link are examined: full detail model, reduced order model (ROM), equivalent beam element, and lumped stiffness. With the exception of the full detail model, stiffness coefficients used in the analysis can be obtained from the force link's specifications.

A basic configuration for a force limiting vibration table is an aluminum plate supported at four corners by force link sensors. The object under test is mounted to the aluminum plate. The base of the assembly is driven by a shaker and the output of the force sensors is used by a control system. The object under test should be included in the analysis, but it is ignored in this example for simplicity. We perform modal analysis to simulate the first four mode shapes and frequencies of the system. The only boundary condition is a fixed support applied to the bottom of the lower plate.

FEA with Full Detail Force Link Model

As a baseline for comparison, the system is modeled with full-detail force links (Figure 8). The force links in this model are the same as the model used to extract the stiffness coefficients. The mode shape plots show that both the stiffness of the force links and aluminum plate play a role in the system performance. This model contained >1M elements and had a high computational cost. It is important to note that analysis also requires knowledge of the internal construction of the force links.

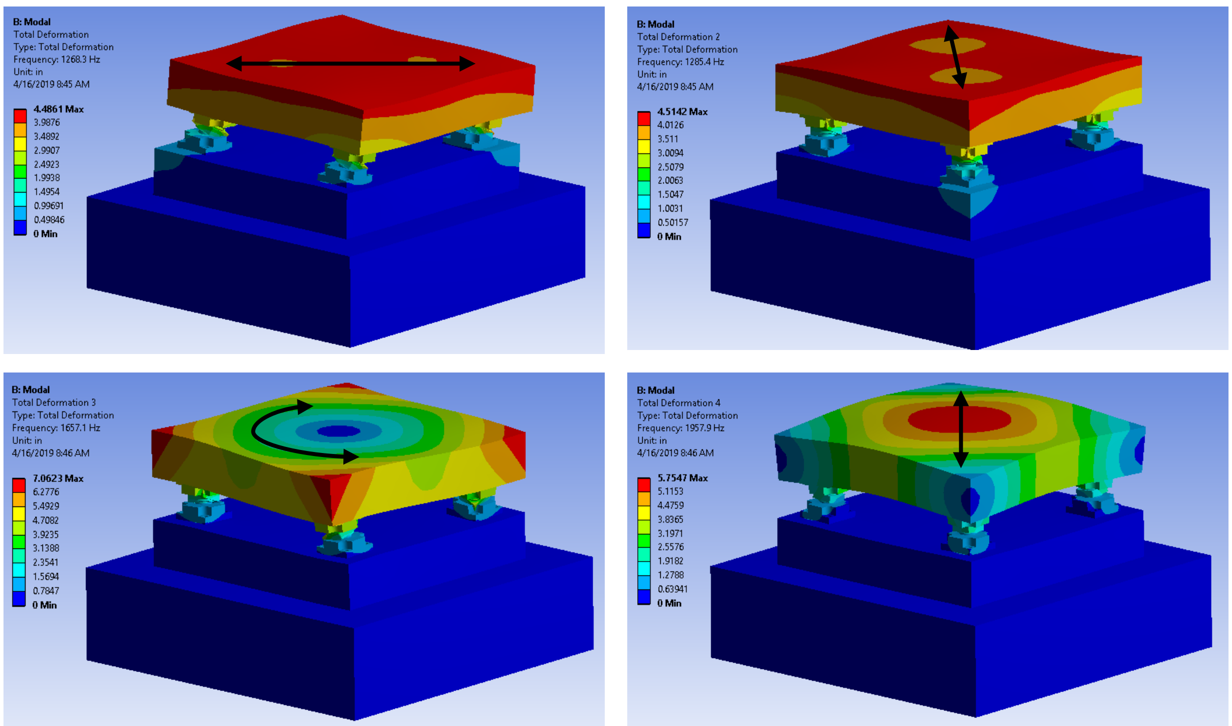


Figure 8: System mode shapes with full force link model

FEA with Equivalent Beam Elements

Some simulation packages, for example Ansys™, allow a connection between two bodies to be defined by a stiffness matrix (see joint, bushing). The stiffness matrix from Equation 2 is directly entered into the software. The software creates a beam element connecting the two bodies. The connection is scoped between areas on each body equal to the mounting faces of the force link. The scoped area's behavior is set to deformable. One beam connection is created for each force link. This allows for representation of the stiffness of the force links without explicitly modeling them, which results in a huge savings on computational cost. In this model the mass of the force link is ignored. In applications where the force link contributes a larger portion to the total mass, it should be included. Figure 9 below shows the mode shapes with beam elements connecting the two aluminum plates. The mode shapes are similar to the baseline, full detail force link model.

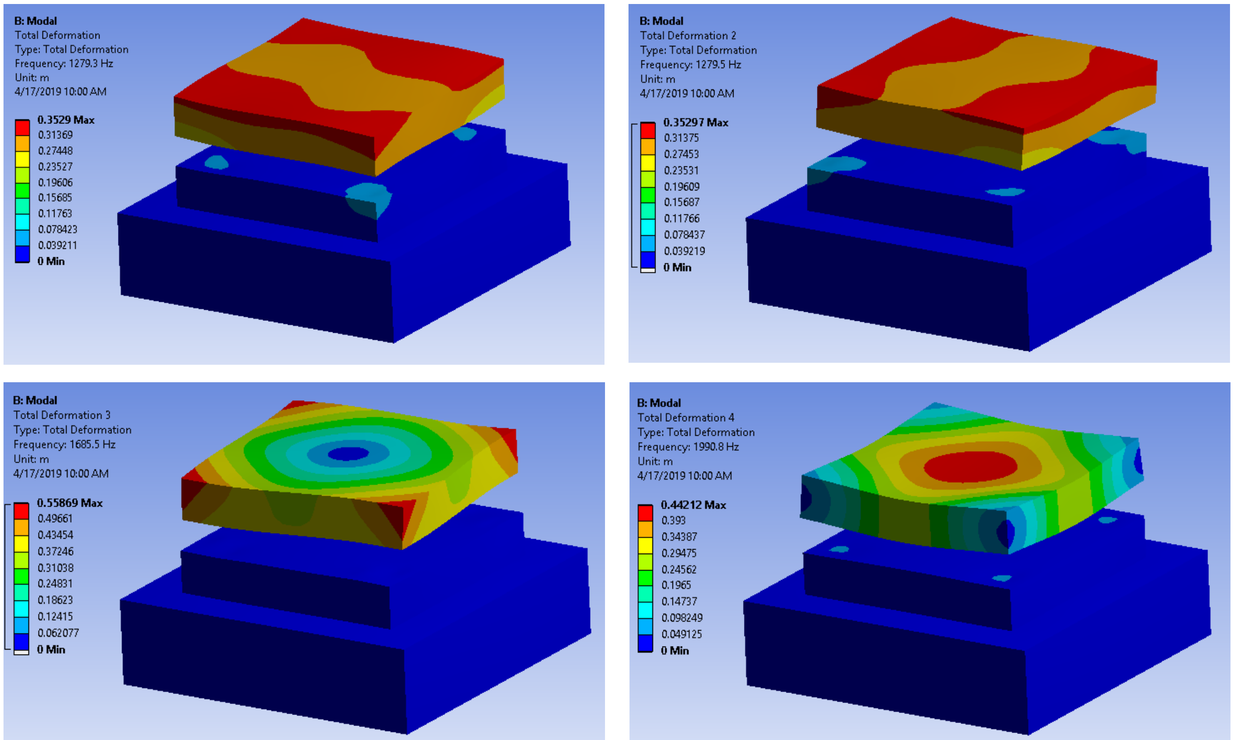


Figure 9: System mode shapes with equivalent beam force links

FEA with Reduced Order Model (ROM) Force Link

It is important to note that not all FEA packages allow the direct entry of the stiffness matrix. If this happens to be the case, another possibility is to create a reduced order model (ROM). There are many ways to construct a ROM; this example shows how we did it. The center stud is excluded and the geometry of the mounting plates is simplified. Each device's outline drawing contains enough information to reproduce this model [3]. The sensor portion of the force link is replaced with a rectangular toroid, see Figure 10. The same FEA approach that was used to determine the stiffness coefficients is performed on the ROM – that is, all but one of the DOF is constrained while each DOF is solely displaced to determine resulting force. The inside/outside diameters and material properties (Young's modulus and Poisson's Ratio) of the toroid is iterated until the stiffness matches all of the PCB stiffness specifications. This process can be performed manually but we found a gradient based optimization algorithm quickly converges on a solution. Figure 11 shows the mode shapes of the test system with the ROM force link. It can be seen they are similar to the baseline, full detail force link model.

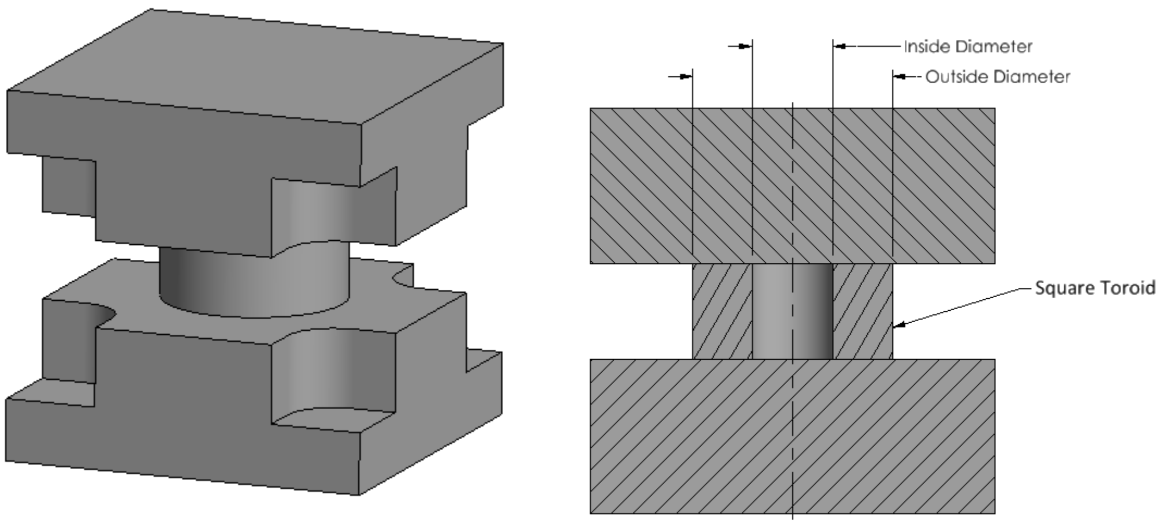


Figure 10: Force link ROM

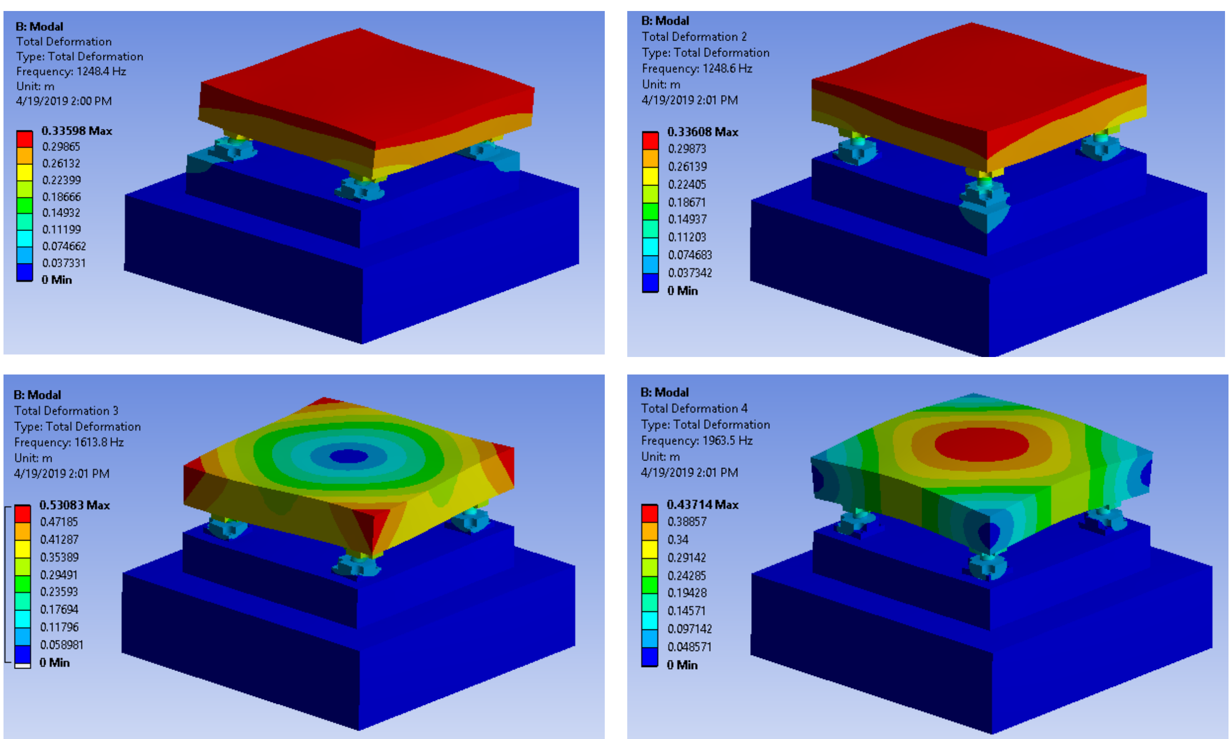


Figure 11: System mode shapes with simplified force links

Lumped Spring-Mass System Model

Historically, to predict system performance, some engineers would simplify the analysis to a 1D spring mass system, Figure 12. We use Equation 3 for two separate calculations: one in the Z direction, and the other in the X or Y. In the simplest form, only the stiffness of the force link would be used. If several force links are used in parallel, their stiffness should be summed. The mass is the combined mass of the parts on top of the force links. For this example, only the mass of the aluminum plate is used. To calculate the frequency of mode 1 and 2, X and Y stiffness is used from the product specification (K_T in Table 1). To calculate mode 4, the Z stiffness is used from the product specification (K_A in Table 1). With these inputs, it would be expected that the lumped model "hand" calculation would predict a higher frequency compared to the other methods.

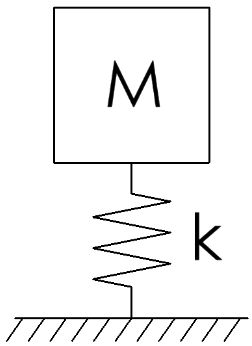


Figure 12: 1D spring mass system

$$f_n = \frac{1}{2\pi}\sqrt{k/m}$$

Equation 3: Natural Frequency

Model Comparison

Table 3 compares the calculated modal frequencies with each method. Both the reduced order model and the equivalent beam model agree with the fully detailed model. Although not accurate, the hand calculation can be a useful tool to acquire ballpark estimates quickly.

Table 3: Comparison of mode frequencies

Mode Number	Full Model	Reduced Order Model		Equivalent Beams		Simple Lumped-mass	
	(Hz)	(Hz)	% difference	(Hz)	% difference	(Hz)	% difference
1	1268.3	1248.4	1.6%	1279.3	-0.9%	1882.2	48%
2	1285.4	1248.6	2.9%	1279.5	0.5%	1882.2	46%
3	1657.1	1613.8	2.6%	1685.5	-1.7%	-	-
4	1957.9	1963.5	2.6%	1990.8	-1.7%	3208.9	64%

Conclusion

PCB Piezotronics has updated their stiffness specifications to provide greater usefulness in the analysis of complex test systems incorporating force links. The updated stiffness is described by five independent stiffness coefficients that define a 6x6 stiffness matrix. The meaning and definition of these stiffness coefficients has been presented in this paper. Two methods for using PCB's stiffness specifications in the modeling of complex test systems have been presented. The first method is direct input of the stiffness coefficients into the finite element analysis package as a beam element matrix. A second method, utilizing a Reduced Order Model (ROM), was demonstrated for finite element packages that don't have provisions for beam elements. Both the beam element and ROM methods were shown to produce similar results in the example of a finite element analysis of a simple vibration table instrumented with force links. Both yielded modal frequencies close to those of a fully detailed model of the force link. An analysis using a simple lumped-mass hand calculation predicted significantly higher frequencies, demonstrating the importance of using finite element analysis to more accurately predict test response.

References

[1] B. Metz and M. Grimaldi, "Optimizing 3-component Force Sensor Installation For Satellite Force Limited Vibration Testing," in *28th Space Simulation Conference-Extreme Environments: Pushing the Boundaries*, Baltimore, 2014.

[2] NASA, *Force Limited Vibration Testing (NASA-HDBK-7004C)*, NASA Technical Handbook, Washington, DC, 2012.

[3] PCB Piezotronics, [Online]. Available: www.pcb.com.

PCB PIEZOTRONICS
AN AMPHENOL COMPANY

3425 Walden Avenue, Depew, NY 14043 USA

pcb.com | info@pcb.com | 800 828 8840 | +1 716 684 0001

WPL_88_0822