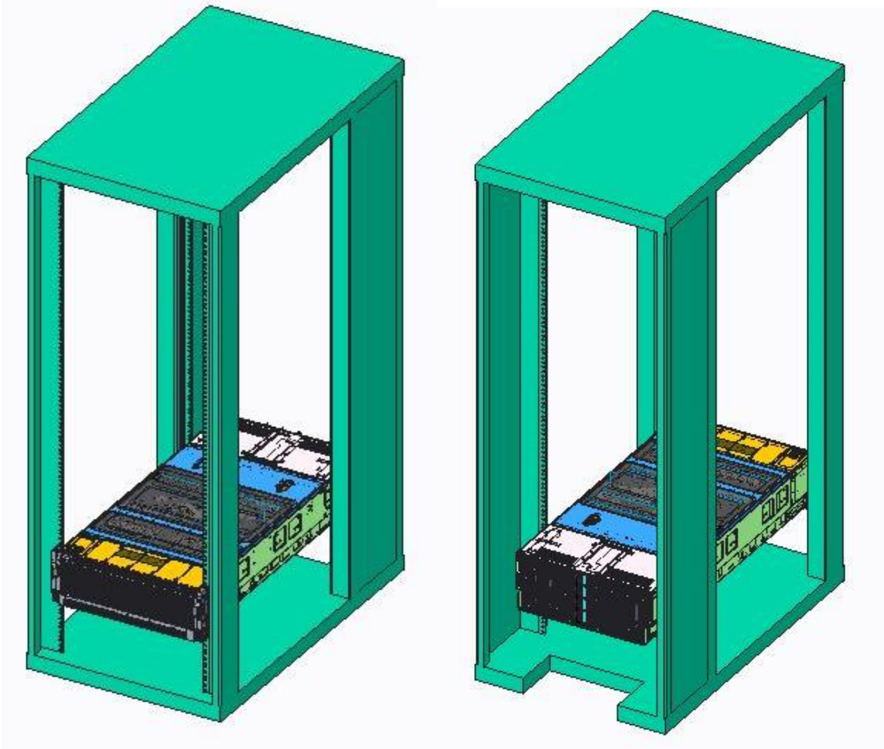
Mainframe Computer Connector Wear Correlation and Prediction Analysis

Shawn Canfield, Budy Notohardjono, Richard Ecker, Suraush Khambati

IBM Corporation

1 Background

Mainframe computers are expected to be highly reliable and available. To achieve this high level of reliability and availability, care must be taken from the initial development cycles to insure robust software and hardware. Here, the discussion will be focused on the structural aspect, namely the hardware assembly. A mainframe computer's hardware structure consists of the rack, processor drawer, cooling assembly, input and output (I/O) assembly, power supply assembly, memory assembly and storage drawers. A typical mainframe computer with a single drawer installed is shown in Figure 1. The total height is 2.0 m where a total of 42 units (U) of many different types of mountable assemblies or drawers can be installed in the rack; 1U is 44.45 mm in vertical height. The height of the assemblies varies from 88 mm to 440 mm. The rack is an EIA (Electronic Industries Alliance) standard 19-inch-wide rack (482.6 mm), where the actual width of the mounting rails where the assemblies or server drawer is installed is 17 ¾" (450.85 mm). The total width of the rack is equal to 600 mm, which provides space to accommodate the cabling and vertical structure outside the width of the server drawer. The rack depth is 1070 mm. The drawer shown in Figure 1 is a 4U server drawer installed in the bottom of the rack, with a total drawer mass of 73 kg.



Front view   Rear view

Fig.1: Typical mainframe computer structure with only one server drawer shown installed in the rack

All of the server drawers and assemblies installed in the rack have many internal connections as well as drawer to drawer connections with different types of connector systems. From manufacturing to being installed at its final destination, the mainframe computer is subjected to environmental conditions such as vibration that can adversely affect the reliability of the system. One of these conditions is shipping vibration while enroute to its destination as well as operational vibration after the mainframe has been installed at its final destination. Shipping vibration is about an order magnitude higher than the operational vibration; therefore, shipping vibration can induce connector wear that affects the long-term reliability of the mainframe. Most of connections are not positively retained, allowing sliding micro-motion between the male and female parts of the connector pair during shipping vibration. This paper describes a process to predict the wear (and thereby, the performance) of connector pairs under shipping vibration in a server drawer shown in Figure 2a. This is a server drawer with integrated storage, memory and I/O units. Specifically, the focus of this paper is the finite element (FE) analysis of connector contact wear of the busbar connector assembly shown in Figure 2b.

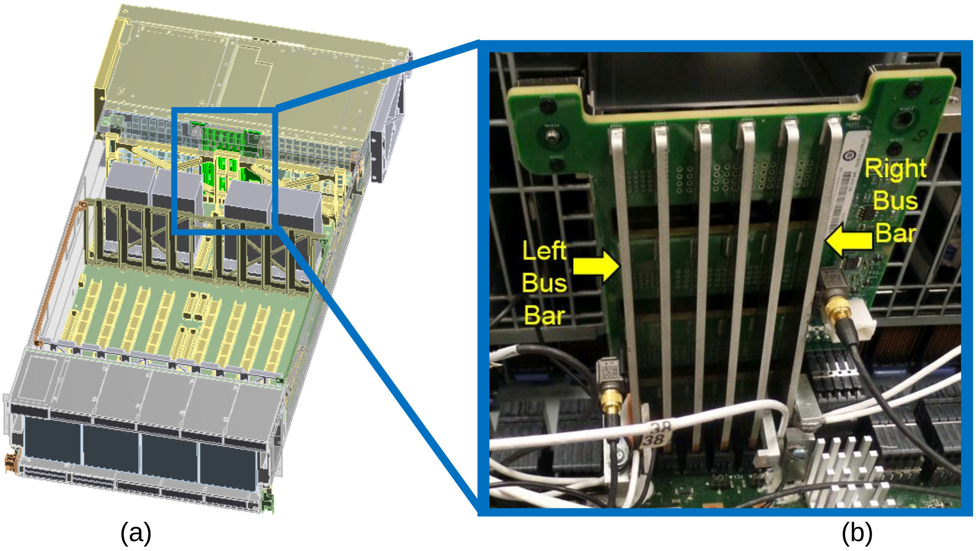
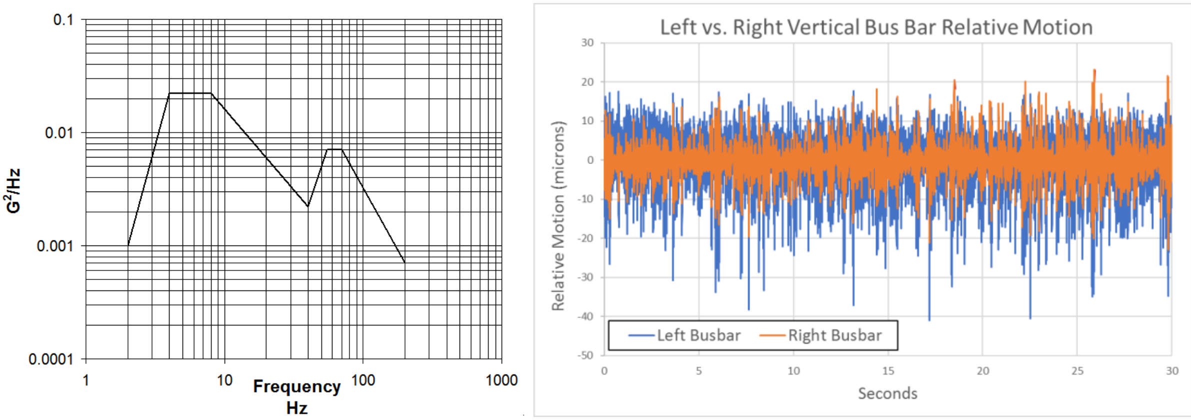


Fig.2: (a) Simplified Geometry of 4U Server Drawer (b) Detail of Busbar Connector Assembly

As part of the development process, the rack with a single server drawer was bolted to the shock table and subjected to vertical shock at 55 m/s velocity change followed by a vertical random vibration at 0.8 Grms for 15 minutes. The power spectral density of the random vibration to simulate the air, truck and rail shipping condition is shown in Figure 3a. The input vibration to the rack is the same, but the actual response of the relative motion within the drawer will differ depending on the mechanical support structure within the server drawer. The shaker and drop tables are hydraulic machines capable of producing low frequency vibration content as low as 2Hz. During the tests, the response acceleration and especially relative motion at connector pairs along all three axes were recorded. Button type capacitive gap sensors by Capacitec model HPB-150A-A-L20-B with amplifier Capteura 520 XL-4Khz were utilized to monitor the motion during vibration. Nominally, 1 micron of relative motion between the two halves of the connector pair produces 4 mV output. The maximum peak to peak displacement of the capacitive sensor is 2.5 mm. The recorded data represents the response of the specific connector system design to a standard shipping random vibration profile. A sample of the relative connector motion, in the vertical direction, is depicted in Figure 3b, where data was taken at both the left-most busbar as well as the right-most busbar (as indicated in Figure 2b).

The connector contact has three layers. The outer layer is 0.8-micron of gold followed by a 1.25 microns layer of nickel. These two layers protect copper from corrosion due to oxidation and insure long term reliability of the connection. After the vibration test, connector contact wear can be evaluated under a scanning electron microscope (SEM) to determine the remaining thickness of the gold layer. Significant wear of the gold layer is considered a risk for long term connector reliability.

This paper discusses the FE analysis of the wear mechanism of typical mainframe connectors. A finite element model (FEM) of the mainframe's processor drawer with its many connector pairs was created. The recorded data from the vibration test, specifically, the relative displacement over time between the connector pair halves was used as an input to a detailed, sub-model of the mated connector pair. Consequently, the LS-DYNA wear model constants were derived based on the measured gold layer wear under known relative motion conditions. Incorporating the LS-DYNA wear model at the connector into a detailed sub-model of the connector assembly then provides a way to estimate the amount of connector wear over time for future design improvement cases, that have not or cannot be tested. Actual connector pair inspection under different design cases was used to validate the model subjected to an explicit, time domain shipping vibration profile. The benefit of explicit LS-DYNA models is that it allows the evaluation of several structural design concepts and reduces destructive hardware testing, shifting towards more learning from FE analysis early in the design process. In addition, LS-DYNA provides a quantitative measure of connector contact wear instead of simply a pass/fail post-test inspection with a microscope. During the development cycle, the ability to predict the performance of connectors under shipping vibration is critical in optimizing the design to prevent connector wear which would impact long term connection reliability.



(a) (b)

Fig.3: (a) Truck, Air, Rail, Vibration Spectrum [1] (b) Test Data Comparison of the Relative Motion of the left and right sides of the Busbar Connector Assembly

2 Global Finite Element Model - Server Drawer

A CAD model was obtained containing the relevant cast and sheet metal parts with printed circuit board (PCB) components and connectors for FEA modeling. Preparation involved removing non-structural details and simplifying geometry for an efficient and stable mesh. Sheet metal and PCBs were mid-surfaced and meshed as 4-node quadrilateral elements with all PCB soldered components offset and bonded with spacer bodies [2]. All individual component material models, both linear and non-linear are then assigned. The global assembly is held together with fasteners represented as beam elements created with an in-house MATLAB script [3]. The final FEA model matched the 4U tall 73 kg test lab model, consisting of 1.7 million elements and 790 thousand nodes, as shown in Figure 4.

Each connector's geometry was simplified with its mating surfaces idealized to provide plug/unplug frictional force. For this, a MATLAB script was developed to create non-linear springs at the connectors that provide the stick-slip frictional force of plugging and unplugging equal to a total of 53.4 N for the entire busbar connector assembly. The non-linear spring force deflection curve was defined to resist motion in either direction until an axial force equal to 53.4 N was reached and then the non-linear spring curve provided no additional resistance allowing for a stick-slip condition approximating the actual frictional forces experienced in the actual connector system when fully plugged. The goal was to simulate the relative motion between connector mating pairs in response to a vertically applied 0.8 Grms random vibration lab test profile. This input profile was of a two second duration chosen out of the 15-minute time history recorded in the lab; representing the highest magnitude as seen in test.

The workstation used for this simulation contained two, 12 core Xeon Gold 6136 CPUs @ 3.0GHz. Using the Intel MPP distributed solver with 22 cores enabled, the typical solve time for the two second random vibration explicit analysis was approximately two weeks. This length of solve time led to the need for developing a sub-model so as to reduce the small details needed in the connector region of this global model. These small elements, dictate the critical time step size, thereby resulting in longer solve times in the larger FE model. They also posed problems in the dynamic relaxation phase of the solution when trying to get the proper/equal connector contact preload.

Fig.4: Global Finite Element Model of the Server Drawer

3 Finite Element Sub-model – Busbar Connector Assembly

As we have just seen with the construction of the global FEM of the server drawer, the connector pair of interest is represented with simplified geometry with nonlinear springs acting between the connector halves. These nonlinear springs provide the nominal frictional force that would be present between the actual connector contacts during the unplugging and plugging that would occur during a random vibration shipping environment test. The global server drawer model is only capable of outputting the relative motion, in all three dimensions, between the connector pair of interest. The real interest here is to simulate the volumetric wear that would occur at the connector contacts during this same random vibration input. To expand further and to get a measure of the wear depth at the connector contacts due to random vibration, more detail of this connector pair of interest is needed along with defining a wear law between connector contacts to produce volumetric wear results. To model the actual female connector contacts would force many small elements into the global model which would further extend the already long solve time of the global drawer model (2 weeks for a 2 second explicit, random vibration input profile), so a sub-model approach is taken when modelling the detailed connector pair.

The finite element sub-model of the connector pair of interest is shown in Figure 5a. The sub-model consists of a portion of the mother card, the daughter card, the busbars and the detailed geometry of the female busbar connectors. The female busbar connectors each have 14 copper contacts (7 on each side) with an electroplated gold finish as detailed in Figure 5b. The geometry comes from the original CAD model of the connector which was then defeatured in ANSYS SpaceClaim to remove any non-structural features, such as drafts and chamfers, that would interfere with getting a consistent, hexahedral swept mesh. The portion of the PCB cards retained in the FE sub-model were modelled as shells and assigned a linear PCB material with a modulus of 21 GPa, which was derived from four-point bend tests using prototype boards. The solid connectors were attached to the shells using a spacer body and face-to-face connections of type **`*CONTACT_TIED_SURFACE_TO_SURFACE_OFFSET`**. The spacer bodies span the distance between the midsurface of the shell bodies to what would be the actual physical 3D top surface of the PCB on which the components would sit on. The connector bodies were assigned an LCP Vectra K130 liquid crystal polymer linear material property with an elastic modulus equal to 16.0 GPa. The connector contacts of the FE sub-model were meshed with solid hexahedral elements with a 0.2 mm element size. All other parts in the assembly were meshed with a 0.5 mm element size. The resulting mesh contains 120 thousand nodes and 90 thousand elements.

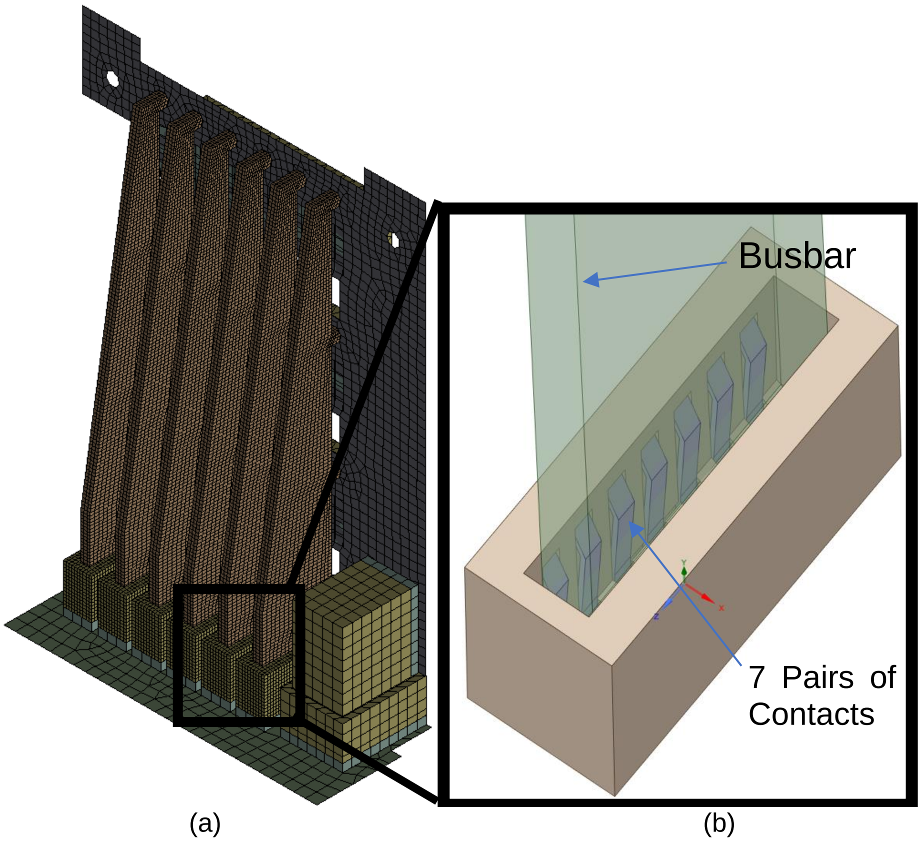


Fig.5: (a) Finite element sub-model of the connector pair of interest. (b) Detail of a single female busbar connector and busbar with two contacts shown.

A ***CONTACT_SURFACE_TO_SURFACE** contact definition was created between the connector contacts and the busbar with a constant static and dynamic friction coefficient of 2.0 [4] to simulate the gold to gold sliding interface. Preload of the connector contacts onto the bus bar blade is necessary to accurately simulate wear and was applied using the geometrical interference that exists in the CAD geometry and a dynamic relaxation step before the explicit solution to resolve the geometric interference between the busbar blades and their 14 contacts, as seen in Figure 5b. The subsequent normal force at each connector contact is approximately 0.31 N. When coupled with the 2.0 coefficient of friction of the gold-on-gold interface, this results in an unplug force equal to 52.5 N which is in line with the nominal unplug force for this connector assembly.

3.1 Wear Coefficient Calculation

A ***CONTACT_ADD_WEAR** definiton was added to the surface-to-surface contact that involves the connector contacts and mating busbar face. The standard Archard wear model in Equation 1, was chosen which relates the rate of material volume loss to contact interface pressure and the relative sliding velocity at the wear interface. The hardness was taken as the hardness of the copper contact (120 HV) and the time scale factor is the ratio of analysis test cycle time (15 minutes) versus the analysis cycle time (2 seconds). The interface pressure and relative velocity results are internal to the FE sub-model based on the model setup and driving inputs. The wear coefficient for this connector assembly was derived by driving the FE sub-model using a 2 second sample of the relative displacement test data taken at the left and right-most busbar locations. The relative displacement test data used as input was in all three directions. Figure 3b shows the test data in the vertical direction.

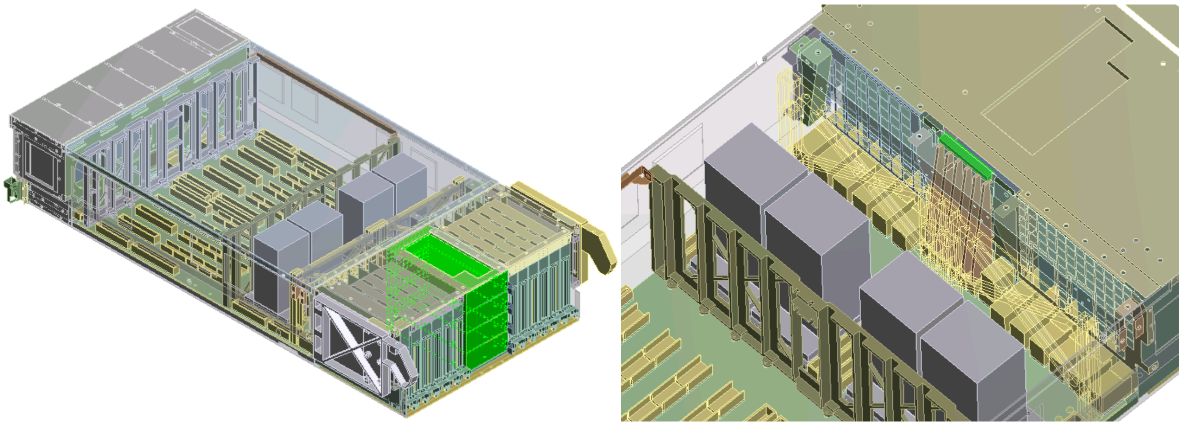
Importantly, this particular test data set represents a case where the increased motion and/or pressure at the left-most busbar produced a marginal, failing wear condition while the right-most bus bar passed the connector wear test. The pass-fail criterion is specified as a presence of exposed nickel which lies under the 0.8 micron gold layer. Values of the wear coefficient were iterated on until the resulting wear depth at the left-most busbar reached 0.8 microns which corresponds to the exposure of the nickel layer under the gold plating. The FE sub-model coupled with this wear model can now be used to explore other server design changes using as input the relative displacement output from the global drawer model at the busbar connector assembly.

$$\dot{w} = \frac{K_{time}K_{wear}}{H}\ Pv_{rel} \tag{1}$$

K_{time} = time scale factor
K_{wear} = wear coefficient
P = contact interface pressure
v_{rel} = the relative sliding velocity
H = material hardness

4 Validation of FE approach based on FE Design Cases and test verification

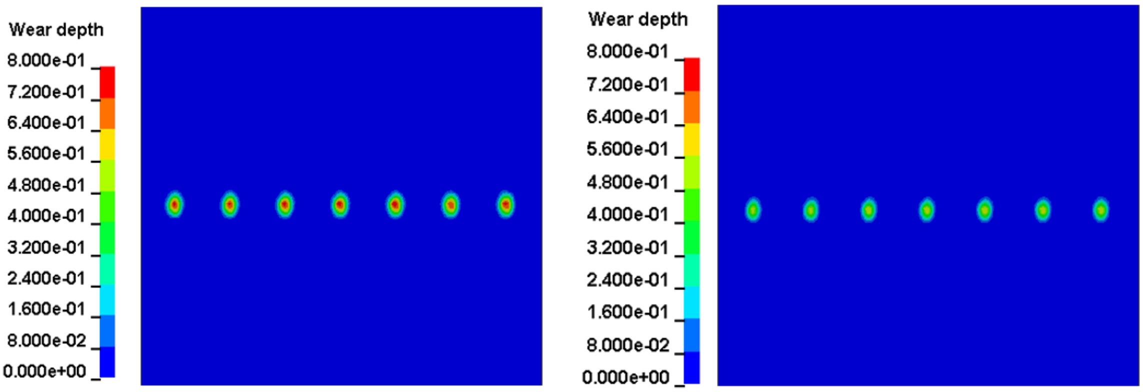
Now that the sub-model's wear law constants have been established using test data, subsequent design modifications can be made to the global FE server model to explore the design space to reduce motion of the busbar connector system. The procedure to predict connector wear is as follows. Input to the modified FE global models is a 2 second, time domain 0.8 Grms random vibration acceleration test data sample taken at the frame mounting location of the server drawer, as shown in Figure 6. During the explicit solution of the global FE model, the idealized connector assembly in the global model with its nonlinear springs approximates the frictional stick/slip of the connector contacts during the plugging/unplugging that occurs during the random vibration event. Solve time for the entire 2 second time history is approximately 2 weeks. The 3D, relative displacement of the left-most and right-most busbar is the output of the global model which is then used as input to drive the detailed sub-model of the busbar connector assembly. The 2 second sub-model analysis time takes approximately 8 hours. At the conclusion of that solution, the wear depth is examined across the busbars to determine if any areas experience a wear depth that is greater than the 0.8-micron connector contact gold layer thickness. If wear depth results greater than 0.8 microns exist, then that is deemed a failure, and if no wear depth results exceed 0.8 micron that is considered a pass.



(a) (b)

Fig.6: (a) Design Alternative #1- Removal of four power supplies reducing total drawer weight by 4.5 kg. (b) Design Alternative #2- Addition of elastomer layer to provide preload on busbar connector assembly

Using this procedure and wear pass/fail criteria two design modifications were examined to validate this methodology. The first design modification is shown in Figure 6a, where the four highlighted power supply units were removed from the assembly reducing the total server drawer weight by 4.5 kg. These power supplies are plugged into the rear of the busbar connector daughter card and influence the motion of the daughter card during a random vibration shipping event and thus the amount of motion of connector motion at the busbar connector. Figure 7a shows the maximum resulting wear depth at the busbar connector for this design alternative whose magnitudes reach more than 1.2 microns which is a failing condition. The next design modification is shown in Figure 6b, where the highlighted body is an elastomer block introduced on top of the daughter card. This elastomer block is nominally in compression adding a 75.6 N preload on the top of the daughter card. The preload is an attempt to hold the busbar connector assembly firmly at the top and bottom limiting any motion that would occur at the connector assembly. After modification of the global FE model with the elastomer, the 2 second random vibration time history profile was applied as an input. The relative motion of the busbar daughter card was then used as input to the FE sub-model with the previously established wear law. The maximum connector wear depth occurs on the left-most busbar and is shown in Figure 7b. The magnitude of the maximum wear depth is 0.49 microns making this a passing condition. Both design modifications were tested in actual hardware and the pass/fail connector wear results are consistent with the FE analyses, giving validation to this method predicting connector contact wear.



(a) (b)

Fig.7: (a) Maximum Wear Depth Result for Design Alternative #1 exceed 0.8 microns. (b) Maximum Wear Depth Result for Design Alternative #2 does not exceed 0.8 microns.

5 Summary

This paper presents a finite element analysis method to predict connector contact wear in a server drawer through the use of a detailed FE sub-model of a connector assembly in conjunction with a global server drawer FE model. A wear law was applied to connector contacts of the FE sub-model with the wear coefficients derived through test data correlation and a measured pass-or-fail wear evaluation. Once the wear law is setup with wear coefficient K_{wear}, FEA can then be used effectively to explore the effectiveness of design iterations for additional improvements while having a direct measure of the connector contact wear depth to compare the performance of each proposed design alternative.

Future work items for this methodology are to explore an update by which the connector contact preload is generated. Herein, the contact interference resolution during the dynamic relaxation step was used which sometimes produced non-uniform pressures on either side of the same busbar at the start of the explicit run. Alternatively, a pseudo thermal expansion of the power blade in the thickness direction only using a **`*LOAD_THERMAL_LOAD_CURVE`** would serve the same purpose of engaging the connector contacts against the busbar surface. This method would eliminate the need for the dynamic relaxation step to produce the connector contact preload and could be done entirely inside the explicit run before the application of the random vibration input. Also, work will continue in applying this connector wear model with current wear coefficients to other connector types in the server drawer to see if a good approximation can be made as is or if each individual connector type must be treated individually with a calibration test hardware correlation.

6 Literature

[1] IBM Corporate specification C-H 1-9711-015: Distribution and Fragility Tests for Rack Products Test Levels and Procedures", 2013-10, 25-31.

[2] J. Torok, S. Canfield, S. Khambati, R. Mullady, B. Notohardjono, and J. Zhang, I/O Printed Circuit Board Assemblies; Recent Learning in Mechanical Stress Analysis, Verification Testing, and Post-test Analysis Techniques and Results, IMAPS 2015, Orlando, FL USA

[3] B. Notohardjono, R. Ecker and S. Canfield, DYNAMICS MODELING AND ANALYSIS OF RIVETED MAINFRAME COMPUTER STRUCTURES, Proceedings of the ASME 2017 PVP Conference, July 16-20, 2017, Waikoloa Village, Hawaii, USA

[4] G. L. Horn and W. A. Merl, Friction and wear of electroplated hard gold deposits for connectors, IEEE Transactions on Parts, Hybrids and Packaging, 1974-10, 53-59.