Predictive Engineering Using DFSS of IBM Power9 System

Arshad Alfoqaha[1], Kevin O'Connell[1], Eric Campbell[1], Mehdi Hamid[1]

[1]International Business Machines Corporation, Rochester, MN, USA

1. Abstract

At IBM systems robust and reliable designs of servers and supercomputers are one of the main objectives. Predicting mechanical performance of servers, such as IBM Cognitive Systems' Power9 portfolio can be more challenging considering shorter development cycles, increasingly dense product design as well as advanced design features. The 2U version includes DDR4 RDIMM's, Power9 hybrid land grid array (HLGA) processor modules, PCIe Gen3 and Gen4 slots, blowers, hard drives, and internal storage controller slots.

Representative models were developed by applying Ansys Mechanical and LS-DYNA FEA tools to simulate drop test of Power9 systems for reliability and robustness. The simulation results were verified by testing that were conducted based on IBM shipping standards. Advantages of implementing predictive engineering methods at early stages of the product cycles are shorten the development time significantly, reduced number of physical builds as well as reduced development cost. Also, FEA simulations, both deterministic and probabilistic, of the main printed circuit board (PCB) were performed to evaluate the effect of plugging and unplugging components on the robustness and the reliability of the main PCB. Components plugged into the main printed circuit board include both PCIe, and RDIMM. Manufacturing tolerances, as well as dimensional and materials variation were taken into account to ensure accuracy. Initially, a deterministic FEA approach was employed to select the worst-case plug-in scenario. Then, a probabilistic approach of the worst-case plug-in scenario was employed using Design for Six Sigma (DFSS) techniques to generate a full factorial Design of Experiment (DOE), regression analysis, Response Surface Modeling (RSM), and to obtain the probability density function to predict the probability of meeting design requirement.

***keywords Supercomputers, FEA, Mechanical Performance, Design for Six Sigma**

2. Introduction

Long term reliability is one of the main contributors to success of a product in the high availability server industry marketplace. Withstanding the full range of shipping conditions is one of the key factors for defining the reliability of the servers. Another key factor to reliability is withstanding the plug-in and unplug forces of all cards attached to the main PCB.

IBM shipping standards contains rigorous tests to ensure servers reliability over transportation by air, truck, and rail. IBM systems utilize advanced finite element simulation across all the product range to meet the IBM standard shipping requirements. The FEA and design teams are working close together to provide timely feedback to reduce product iteration time. The result is a highly reliable product that can be integrated into a server rack and shipped anywhere in the world.

Deterministic and probabilistic FEA simulations, of the main PCB were also performed to evaluate the effect of card plug-ins and unplugs on the robustness and the reliability of the main PCB. Cards that plug-in and unplug from the main PCB included both PCIe, and RDIMM. Dimensional variation in components and material property variability were taken into account to ensure accuracy. Initially, a deterministic FEA approach was applied to identify the worst-case plug-in scenario. Then, a probabilistic approach of the worst-case plug-in scenario was implemented using DFSS techniques to generate the full factorial DOE, regression analysis, RSM, and to obtain the probability density function to predict the probability of meeting design requirement.

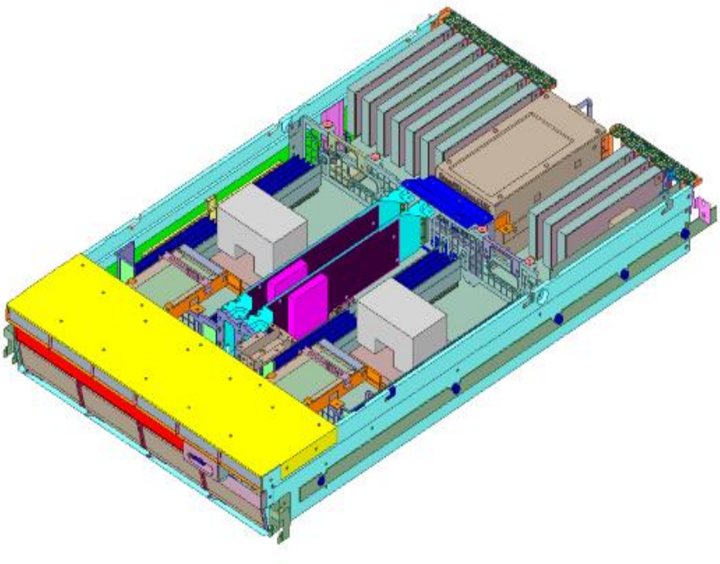
The paper will exhibit developing, analyzing and verifying steps of a transient FEA simulation and modeling for a Power9 system. Then, the paper will show how the physical server was instrumented to collect acceleration and strain values to verify the FEA model. Even though the IBM shipping standard includes several different tests, the data reported here will only capture the shock/drop portions of the

test, since this is when the highest forces are exerted on the equipment. Finally, the paper will discuss the reliability analysis using DFSS method.
The systems used as an example in this paper were part of IBM's 922 series of Power9 servers, Fig. 1. These servers were designed and optimized for both small-enterprise and scale-out computing applications [2]. The employment of FEA in early product design cycles helped to enhance product robustness and keep up with IBM's high demand standards.

Fig.1: 2U Power9 System (Cover off)

3. Finite Element Simulation Model

Several tools were used to build the FEA model for simulation. ANSYS SpaceClaim [3, 4] was used to create simplified 3D CAD models. This was required to reduce model size and complexity and improve solve times without compromising accuracy of the structural behavior. Non-critical features and geometry were removed including corner chamfers, or round edges, cable mounting holes and other non-structural punched or formed features. ANSYS Explicit Dynamics (LS-DYNA Export) and ANSYS Mechanical were applied to define the FEA portions and export the solver input file.



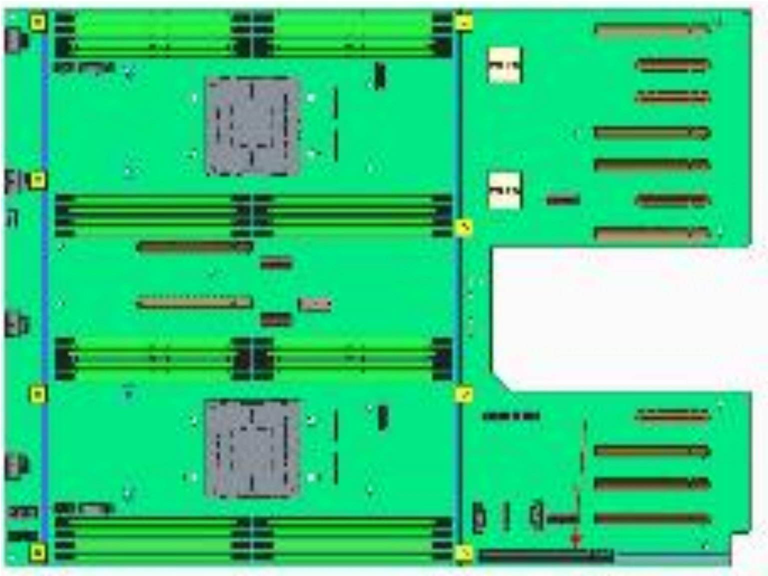


Fig.2: 2U & the main PCB Power9 systems SpaceClaim models

All sheet metal components, such as the top cover, the main chassis, the front and back bulkheads, were simplified in SpaceClaim as midplane surfaces with corresponding thickness and modeled using shell elements. Steel material properties were applied with a modulus of elasticity of 200 GPa and Poisson's ratio of 0.3. Also, a bilinear isotropic hardening stress vs strain curve, with a yield stress of 180 MPa and tangent modulus of 1.45 GPa was defined for the applied steel material. Bonded contact was assigned to the bodies with a face-to-face connection and an internal contact was assigned between all the bodies for nonlinear frictional contact. The T-pins on the side of the chassis were attached to the slide rails which were attached to the rack.
The main PCB was modeled as a midplane surface. Large components where exact structural behavior wasn't needed were modeled as mass blocks. The correct mass was represented by applying a modified density in Engineering Data that accounted for the total volume of the new mass block. The total mass of the 2U was about 66 lbs.
Two types of connections were used in the FEA model: (1) Tied/bonded contact which was assigned to all bodies with a face-to-face connection and was detected automatically within a specified tolerance value, and (2) body interaction contact which was assigned between all the bodies for nonlinear frictional and frictionless contact. For the frictional contact, the coefficient of friction (COF) was assumed to be equal to 0.1. Spacers were added to fill gaps between select components and the midplane surfaces they were bonded with to improve stability of the model. The spacers were excluded from the body interaction contact.
Rivets, bolts and screws were modeled using a combination of 1D beam element [5] and Constrained Nodal Rigid Bodies (CNRB). The beam element properties were defined using the *SECTION_BEAM card that assigns the appropriate section properties to the beam based on the diameter of the bolt. The CNRB's were created to rigidly connect the nodes around each hole to a central node. The center node of each of pair of holes was then connected together using the 1D beam element and the *ELEMENT_BEAM card.
The model of the 2U full system consisted of about 3,000 bodies, a combined shell and solid element count of about 500,000, and a node count of about 550,000. Four-noded tetrahedral and eight-noded hexagonal linear explicit solid elements, and three-noded triangle and four-noded quadrilateral explicit shell elements were used to mesh all bodies. The solid and shell element sizes were mostly between 1 mm and 4 mm, and the circular holes which represented the rivets and screws had at least 8 divisions (nodes), Fig. 3. As a good check of the mesh quality, the elements aspect ratios were targeted to be mostly below 15.

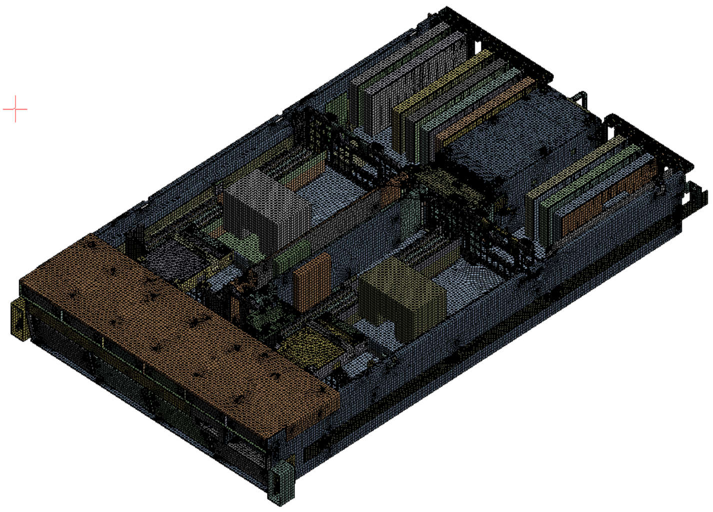


Fig.3: 2U Power9 System mesh (cover off)

The resulting FEA model was then exported as an LS-DYNA input file (*.k file) and solved using the nonlinear, explicit solver. The solver time step was set to approximately 1.0E-07 in order to handle the complex nonlinearities. This drove a solution time of 5 to 7 hours for 30 ms of real time and 15 to 20 hours for 90 ms of real time. The explicit solver was chosen due to its ability to handle complex contact and material nonlinearities. The computer used to solve the model utilized 18 cores of a 3.10 GHz Intel Xeon CPU along with 128 GB of RAM.
Two boundary conditions were used as input to the system: (1) Standard Earth Gravity of 9810 mm/s^2 applied to all bodies in the vertical direction, and (2) an acceleration profile from a 70 in/s palletized rack drop test (peak acceleration of ~50g for a time duration of 8.2 ms) which was applied on the rails, rear brackets, and front latches, Fig. 4.

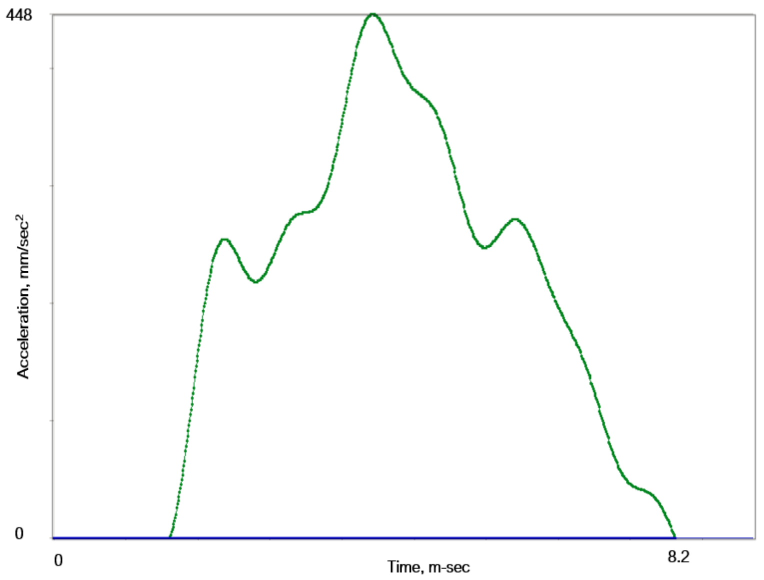


Fig.4: 70 in/s palletized rack drop profile

During the explicit solution, energy ratio, kinetic energy, total energy, internal energy, damping energy, and sliding energy were monitored to ensure accuracy of the model. The energy ratio was plotted to verify its value was stable and close to 1.0. The target for the other energies varied based on type.

Also, stress (such as von Mises and Principal), strain (such as von Mises, and maximum principal strain), internal energy, and nodal displacements and acceleration were obtained for all components in the system.

After collecting results, Fig. 5 and Fig. 6 revealed that the von Mises stress in the top cover and main chassis of the server were below the assumed yield stress of steel of 180 MPa [6]. High stresses around rivets and screws were ignored due to the localized nature of stresses around such features. As expected, plastic strain in the the top cover and main chassis reported very close to zero. Due to the low stress levels, plastic strain was low and confirmed that very little permanent deformation had occurred on the system. Similarly, other bulkheads and sheet metal parts were evaluated for stress and strain and indicated no significant material yield.

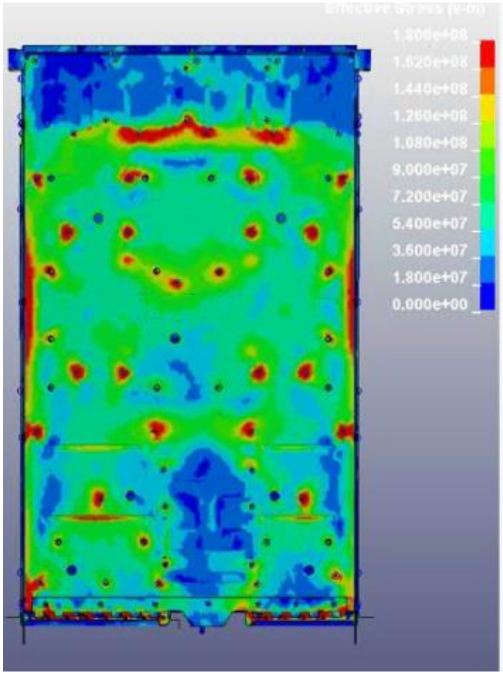


Fig.5: von Mises stress in bottom chassis for 2U system

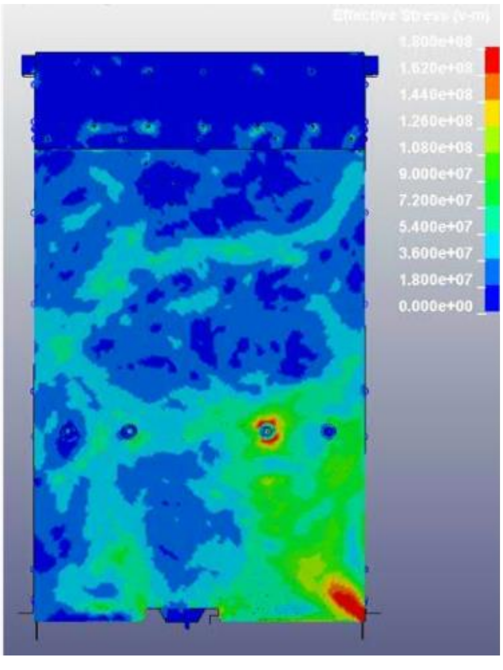


Fig.6: von Mises stress in top chassis for 2U system

Maximum principal strains were obtained for the main PCB of the 2U system from the FEA results. Critical areas around the two HLGAs (labeled 1 and 2 in Fig. 8) and cards that were attached to the board were closely monitored. The upper integration point (Ipt) maximum principal strains near the four corners of the HLGAs on the main PCB were probed and evaluated. FEA results showed that the upper Ipt maximum principal strains for HLGAs 1 and 2 were below 1,000, and 1,600 micro-strain, respectively, Fig. 7. Other components and PCBs of interest were the PCIe, SAS, and power supply, where FEA results for the 2U system showed that the upper Ipt maximum principal strains were low (below 400 micro-strain), Fig. 8, for all components.

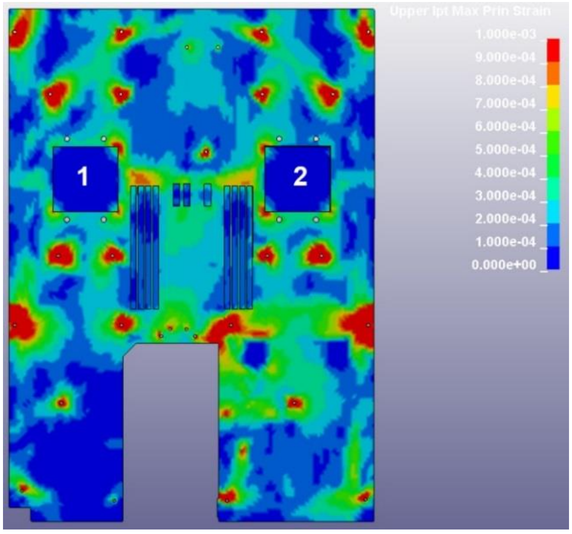


Fig.7: Upper maximum principal strain for the 2U main PCB. Locations 1 and 2 indicate the HGLAs on the PCB

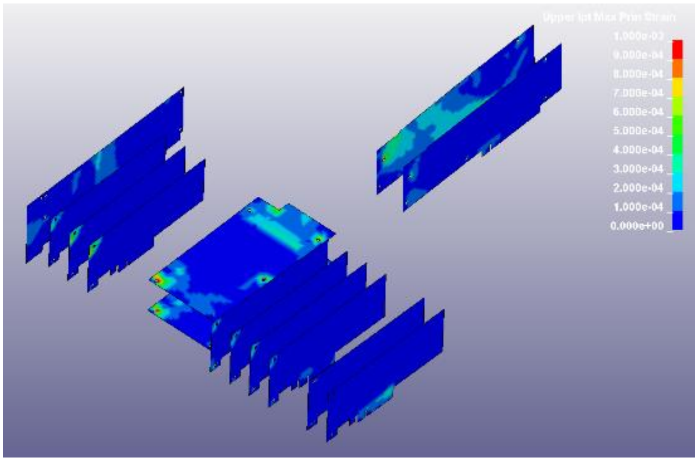


Fig.8: Upper Ipt maximum principal strain for the 2U SAS, PCIe, and power supply cards

4. Simulation validation: Drop test

Drop tests were performed on the 2U systems to validate the FEA results to ensure model accuracy. Strain gauges and accelerometers were attached at several locations on the main PCB, chassis, and the mounting points on the server rack, Fig. 9.

The main PCB of the 2U system was strain-gauged with 10 stacked rosette strain gauges to measure strains around the two HLGAs and other critical components on the PCB. Strain gauges [7] were placed on the PCB close to the corners of the HLGA sites, and close to the SAS and PCIe connectors, Fig. 10.

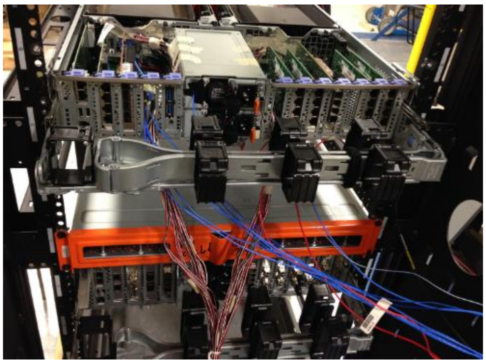


Fig.9: 2U Power9 systems with accelerometers and rosette strain gages

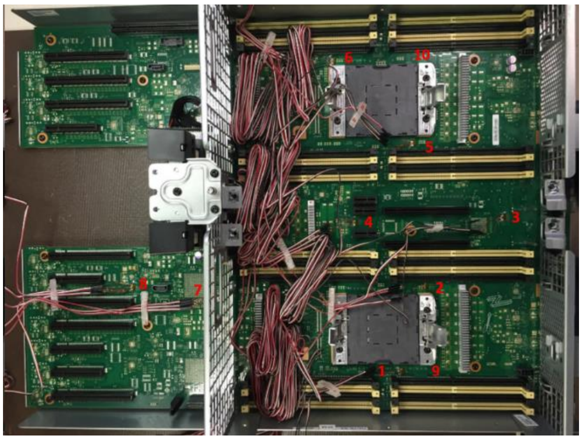


Fig.10: Main PCB with ten rosette strain gauges

Drop testing was performed on the 2U system, during which strain gauge measurements were collected. The measured strains from the drop tests were compared to the FEA strain results in Table 1. The principal strains obtained from testing and FEA were in good agreement.

Gage #	1	2	3	4	5	6	7	8	9	10
Test	374	987	255	493	968	370	510	314	839	892
FEA	438	946	359	411	912	351	518	499	863	885

Table 1: Maximum principal strain for 2U PCB: Testing vs. FEA results

5. Design for Six sigma (DFSS) Evaluation: RSM and Reliability Modeling

Three parameters were selected and assumed to have large variation: (1) plugging force, (2) mass density of heatsink, and (3) Young's Modulus of PCB material (FR4). All parameters were assumed to follow normal distribution and have a standard variation of 16 %. Also, three outputs were considered: (1) maximum stress, (2) maximum strain, and (3) maximum deflection.

For Response Surface Modeling (RSM), a DOE was generated using the Central Composite Design (CCD) sampling method. Therefore, 15 design points were generated (2n + 2n + 1), see Figure 11.

	A	B	C	D	E	F	G	H
1	Name	P1 - Board Young's Modulus (MPa)	P2 - Heatsink Density (kg m^-3)	P6 - Plugging Force (N)	P3 - Max Displacem... (mm)	P4 - Max Stress (MPa)	P5 - Max Strain mm/mm (mm mm^-1)	P7 - Max Strain
2	1 DP 0	83427	2700	94.3	0.20524	70.29	0.00018456	184.56
3	2	44953	2700	94.3	0.3809	70.29	0.00034252	342.52
4	3	1.219E+05	2700	94.3	0.14047	70.29	0.00012631	126.31
5	4	83427	1448.5	94.3	0.20525	70.291	0.00018458	184.58
6	5	83427	3951.5	94.3	0.20523	70.289	0.00018455	184.55
7	6	83427	2700	50.728	0.10993	37.657	9.8889E-05	98.889
8	7	83427	2700	137.87	0.30056	102.92	0.00027023	270.23
9	8	52146	1682.5	58.874	0.20439	43.759	0.00018385	183.85
10	9	1.1471E+05	1682.5	58.874	0.092917	43.759	8.358E-05	83.58
11	10	52146	3717.5	58.874	0.20436	43.757	0.00018382	183.82
12	11	1.1471E+05	3717.5	58.874	0.092904	43.757	8.3563E-05	83.563
13	12	52146	1682.5	129.73	0.45236	96.823	0.00040673	406.73
14	13	1.1471E+05	1682.5	129.73	0.20564	96.823	0.0001849	184.9
15	14	52146	3717.5	129.73	0.45233	96.821	0.00040669	406.69
16	15	1.1471E+05	3717.5	129.73	0.20563	96.821	0.00018488	184.88

Fig.11: DOE using central composite design (CCD) for Response Surface Modeling (RSM)

Taking into account all assumed variations, using Six Sigma analysis, it was found that the probability of achieving strain < 500 µε was greater than 99.99 %, the probability of achieving stress < 159 MPa is greater than 99.99 %, and the probability of achieving displacement < 1 mm was greater than 99.99 %. See Figure 12.

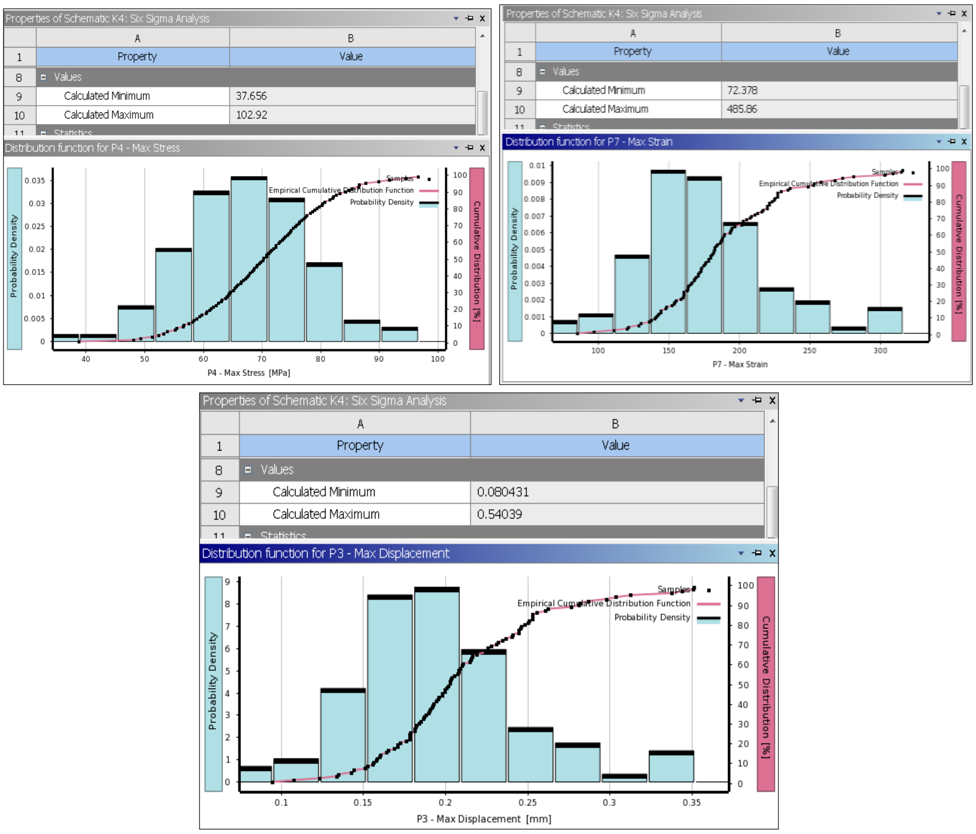


Fig.12: Probability of meeting strain, stress, and displacement targets

6. Summary and Conclusion

In this work a close correlation between simulation results and an IBM server rack drop test was provided. As part of server design process, a 3D model was generated. The generated model was simplified by Ansys Spaceclaim. Ansys Explicit dynamics (LSDYNA export) was applied to generate LSDYNA input file. The generated file from the former step was passed to LSDYNA solver for dynamic analysis. Moreover, the FEA results were verified by experimental testing results.
The observed close agreement between the measured strain results on the PCB from testing and the ones from FEA provided a strong validation of the simulation results and the modeling approach. The verified modeling approach was implemented to IBM's Power9 servers and reduced the number of test iterations in the design cycle.
Six sigma methods such as Response Surface Modeling (RSM), Design of Experiments (DOE) and reliability modeling were implemented to evaluate the design robustness and the reliability of the main PCB. Taking into account all assumed variations, it was found that the probability of meeting strain, stress, and displacement targets were close to 99.99%, assuring a robust and a reliable server design.

7. References

[1] LS-DYNA Keyword User's Manual Volume 1. Livermore Software Technology Corporation, Livermore, CA. 2017. https://www.dynasupport.com/manuals/ls-dyna-manuals/ls-dyna-manual-r10.0-vol-i.
[2] "IBM Enterprise Business Servers," accessed April 25, 2019, https://www.ibm.com/it-infrastructure/servers.
[3] ANSYS Mechanical User's Guide Release 18.0. ANSYS, Inc., Canonsburg, PA. 2017. https://support.ansys.com/portal/site/AnsysCustomerPortal/template.fss?file=%2Fprod_docu%2F18.0%2FANSYS+Mechanical+Users+Guide.pdf.
[4] ANSYS Theory Reference Release 5.6. ANSYS, Inc., Canonsburg, PA.

[5] Notohardjono, B. , Canfield, S. , Khambata, S. , and Ecker, R. , “Verification of Mainframe Computer Structure Finite Element Model Under Vibration and Seismic Tests.” Proceedings of ASME 2017 Pressure Vessels and Piping Conference. Prague, Czech Republic, July 15-20, 2018.
[6] “AISI 1008 Steel, CQ, DQ, and DQSK sheet, 1.6-5.8 mm thick,” accessed April 25, 2019, http://www.matweb.com/search/datasheet.aspx?matguid=145867c159894de286d4803a0fc0fe0f&ckck=1.
[7] “062WW General Purpose Strain Gages – Stacked Rosette,” last modified April 27, 2017, accessed April 25, 2019, http://docs.micro-measurements.com/?id=2560.
[8] Maass, E., McNair, P., “Applying Design for Six Sigma to Software and Hardware Systems,” Pearson Education, Inc, 2010.