International Journal of Analytical, Experimental and Finite Element Analysis (IJAEFEA), Issue. 4, Vol. 2, Dec 2015.
e-ISSN: 2394-5141, p-ISSN: 2394-5133, pp 142-146

Seismic Analysis of Power Cabinet using FEA and Comparison with Experimental Validation

Ranjit A. Patil
ranjeetpat1@gmail.com

FE Analyst,
Spring Meadows,
Ambegaon Budruk,
Pune -411046
India.

***Abstract*—Power cabinets can be installed on various locations depending on the need like nuclear power plant, sea region, mountain regions. So it is necessary to validate the model against the seismic loads depending on region it is located, loading factors, standards followed. The given paper gives brief information on how to calculate static coefficient method equivalent to seismic analysis. It also gives information on how to calculate floor loading analysis which can be used to design given floor. The results are validated with experimental data and observed good correlation between FEA model and experimental results.**

***Index Terms*—Static equivalent method for seismic analysis, power cubicle, spectrum analysis, floor loading analysis.**

I. Introduction

For seismic simulation requirement, different methodologies can be applied such as static equivalent, modal, spectrum. The static equivalent methodology is often used as it provides conservative and fast results with the use of the automatic fastener assembly macro .This approach should be the most appropriate for most of the cases.This paper would consider that the static equivalent approach is used with respect to the North America level. The power cabinetshould withstand a level 1 as North America level referring to InternationalBuilding Code and AC156 standards.

Although the need for seismic-capable electrical equipment isknown, there is a lack of understanding of how to comply withcurrent code requirements. To suitably define the acceptability of the equipment to thedesignated codes, it is indispensable to present the equipment seismicrequisites and the equipment seismic capability data on thesame technical substructure. The equipment is considered acceptable, if it can withstand the seismic event and perform its function immediately afterward. To restore function of emergency management facilities as quicklyas possible, public officials have revised building codes to mandateimproved seismic design. This includes not only buildings, but alsothe electrical and mechanical equipment contained therein, as wellas machinery necessary for safe occupancy and normal operation.

Apart from seismic analysis, sometimes installers have requirements for determining pressure distribution on floor where power cabinet is going tobe installed. As most of power cubicles weight is high, there is requirement for installer to design floor according to weight and pressure distribution of power cabinet. We will study the methodology to determine the pressure distribution.

II. Static equivalent approach

According to the International Building Code (IBC) static loading approach, thefollowing equation is used to obtain the equivalent lateral accelerationrequirement that will be used in the side to side and front to back/back to frontdirections (for generic building) :

$$F_p = (0.4 \times a_p \times S_{DS} \times W_p / (R_p / I_p)) \times (1 + 2 \times Z/h) \quad (1)$$

Where,
a_p =1 (building amplification)
R_p=2.5 (Equipment response multiplication factor)
Ip =1.5 (Equipment importance factor)
Z/h= 1 (considering roof level)
S_{DS} = Design spectral response acceleration

TABLE I
SPECTRUM VALUES

S_{DS}	Level 1	Level 2	Level 3
Horizontal	1.25 g	1.78 g	2.46 g
Vertical	1.78 g	2.46 g	2.46 g

Fp	Level 1	Level 2	Level 3
Horizontal	0.9 g	1.28 g	1.77 g
Vertical	1.28 g	1.77 g	1.77 g

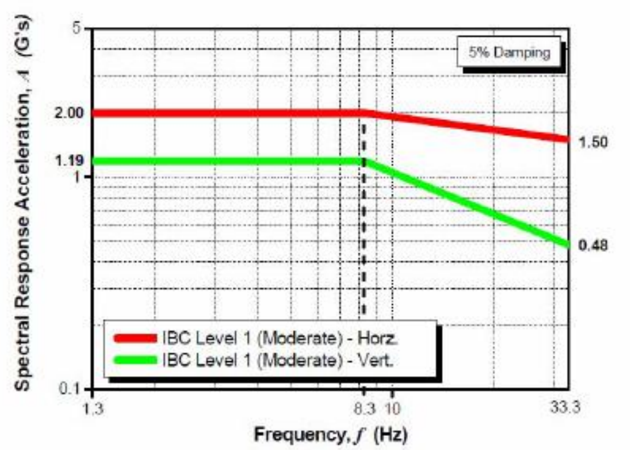


Figure 1:AC156 Preferred Qualification Testing IBC level 1 " Moderate"

III. Methodology related to FEA

A. Geometry

Seismic Analyses almost always imply the structural simulation of complex units with a highamount of interconnected pieces. Even though the FEM analysis requires thesimplification of these complex assemblies, the number of bodies that will finally remain for theANSYS solver to handle will be high.The structural performance of any unit depends heavily on the geometric features of its components and the interconnections between them. This, adding the material properties,determines the overall rigidity of the structure.This is the reason why the geometric configurationscannot be heavily simplified or represented throughprimitive geometric forms most of times in a seismic analysis.The geometry treatment starts even before using theDesign Modeler module from ANSYS, byidentifying and making some arrangements in acommon CAD modeler. This process is the most timeconsuming and hard to pull-out.Now, in seismic analyses, most of the internal component systems (breakers, smalltransformer components, for instance) are not required to be explicitly present in thesimulation, unless the client wants to see some specific stress or deformation results,which is rare. We will be almost always interested in the structure itself. We will representinternal components through lump masses.If not given other instruction, you should suppressany of these type of internal components. Be carefulnot to eliminate a whole subassembly at once, as itmay have components of interest like some sheetmetal covers and frame channels; unless you seethat a whole subassembly is unwanted, which isgreat. Leave elements like sheet metal, frames,connecting hardware, shelves.

B. Simulation

Place lump masses into the model. This can be done via the geometry tree and inserting pointmasses. This masses correspond to the bodies that were replaced when cleaning the CADmodel in the pre-treatment section. You can choose the number of point mass representationsyou want.You need the location of the centre of gravity of those group of components. It doesn’t have to beexact, but, try to enter a decent approximation. You also need to enter the mass amount we’retalking about. It is of common practice to ask directly the designer for these approximations. If the model is small and decided to go without screws, or, if it is noticed for many reasons (divergence, Modal Analysis explained later) a set of holes that doesn’t have any screws associated, it may create manual beam connections.Once define the beam completely, you will see it on screen. If you don’t like the orientation, you may change it by modifying the start and end reference coordinates. It’s notthat critical.

C. Loading

The loading is quite simple. It consists basically on two acceleration loadings: natural weight,and the equivalent static load seismic weight. The weight of the structure is explicitly availableas the Standard Earth Gravity. The seismic acceleration will be an input of the requirements(most of times 1.28 g’s). The direction is a requirement input, too. (side to side, front to back,back to front). There are three different directions in which the simulation could be done. Youshould always ask for this to be specified.Be sure to specify the fixed supports. If there’s notinformation about them, they should be veryevident as anchor points in the base of thestructure. If this is not the case, you should always ask for this information.

D. Convergence

If you can’t find an evident reason for the model not converging, use a Modal Analysis. It’s notmeant for correction processes, but, it is a resource that can tell the analyst about the flyingpieces in a very direct way. Drag a Modal Analysis Component in the Project window, and besure to associate it with the same model as the Static Structural is working.

IV. Pressure Distribution

Floor defined as rigid shell for contacts. Frictional contact is assumed between floor and bottom surface of Cabinet. Only structural components are considered for the simulation. Bolts are modelled as Beam elements. Linear Isotropic material properties are used.

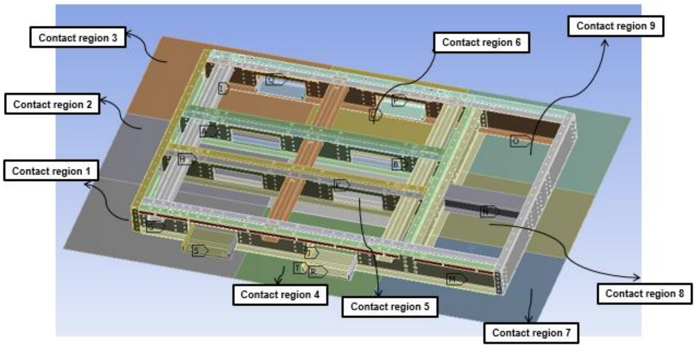


Figure 2:Dividing floor target region in 9 subparts to get pressure distribution values

Dividing the contact area in 9 regions helps us to give more clarity where maximum contact pressure willAppear. The floor has been modeled as rigid body. In above picture the color of floor has nothing to do withsimulation results.

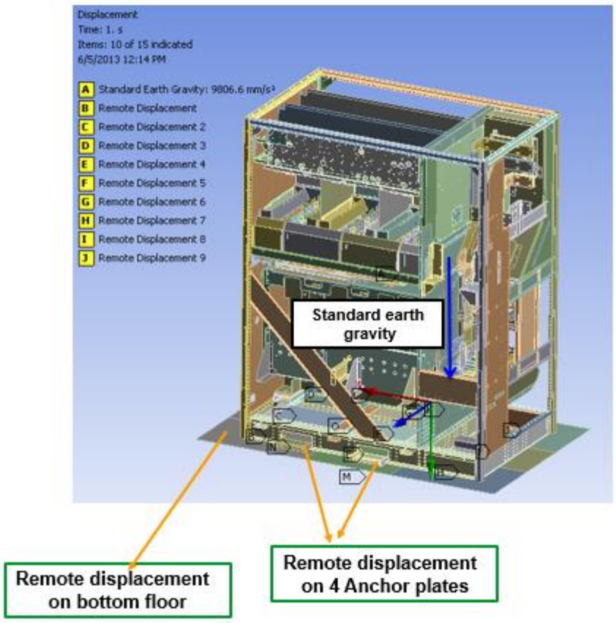


Figure 3:Boundary Conditions

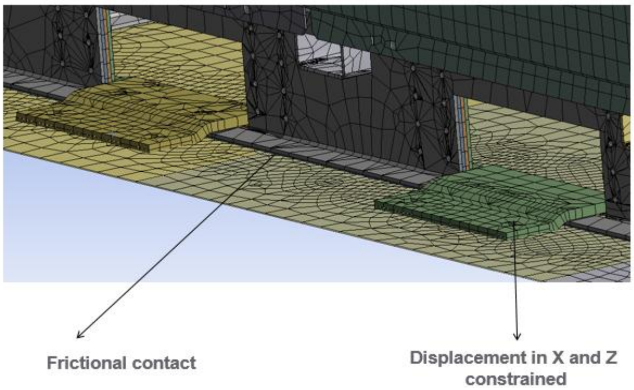


Figure 4:Frictional contact between rigid floor and base of cabinet

V. Boundary Conditions For Seismic

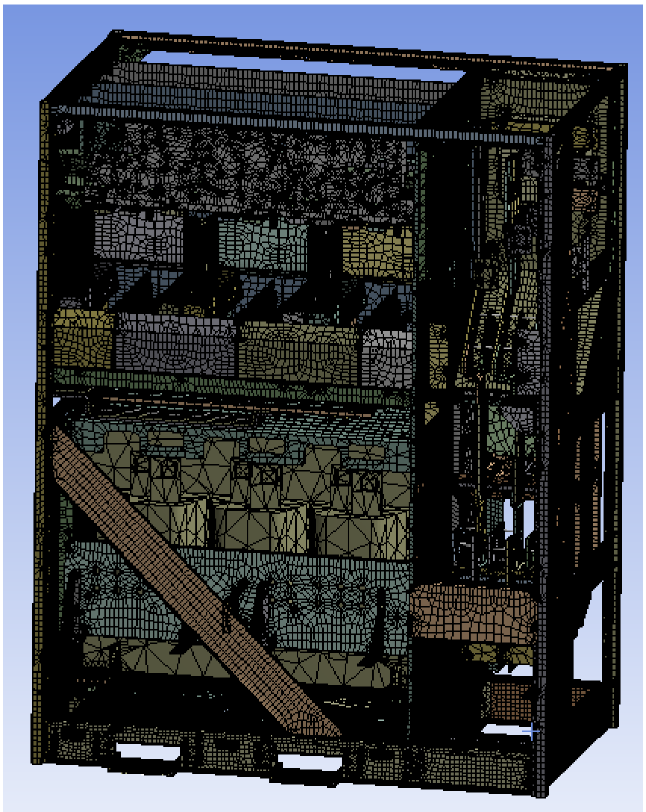


Figure 5:FEA Model in Ansys for Cabinet

All required sheet metal parts have been simulated using mid surface extraction. All subassemblies are simulated using point masses scoped on the appropriate locations. Total mass of the cubicle is around 4462kg.The finite element model was created in ANSYS Workbench v.13Finite element model is mainly composed by approximately 314736 Four-Node Finite Strain Shell 181Beam elements are used to simulate all bolted connections.Yield Stress is assumed as 340MPa (49313 psi).

A. Static Loading Conditions – Horizontal(X)

Lateral accelerations of 1.13g are prescribed in order to capture the behavior of the cubicle when subjected to seismic load in the following directions:X+ and X- corresponds to Right to Left and vice versa (Side to Side)Standard earth gravity is also acting on the vertical axis (y axis) direction.

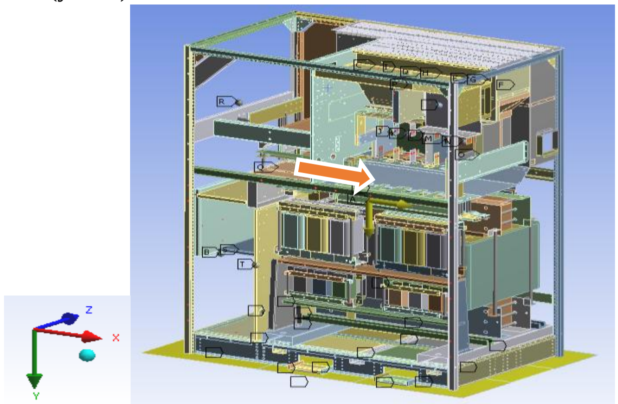


Figure 6:X direction loading

B. Static Loading Conditions – Horizontal (Z)

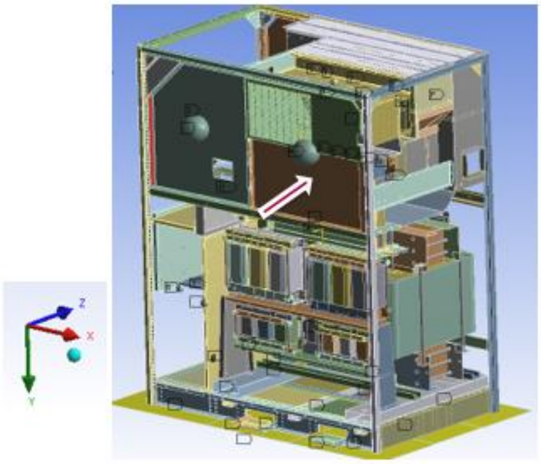


Figure 7:Z direction loading

Lateral accelerations of 1.13g are prescribed in order to capture the behavior of the cubicle when subjected to seismic load in the following directions. Z+ and Z- corresponds to Back to front and vice versaStandard earth gravity is also acting on the negative vertical axis (y axis) direction.

C. Static Loading Conditions –Vertical (Y)

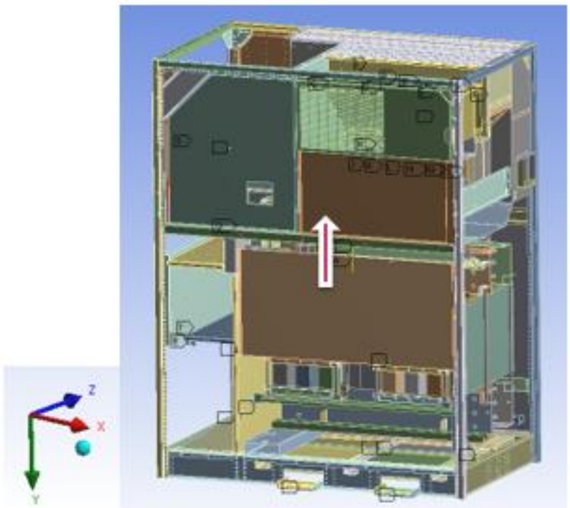


Figure 8:Y direction loading

V. RESULTS

A. For Pressure distribution

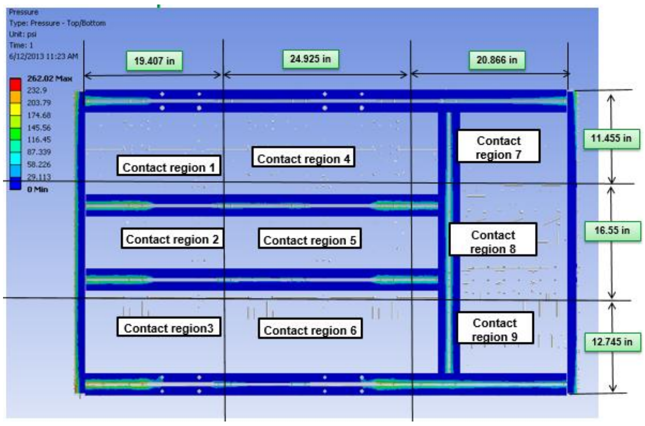


Figure9: Pressure distribution

It is clear from results that maximum pressure value of 262 psi has been observed for region 3.

From all values of pressure distribution and knowing the reaction forces at corresponding regions, we can determine the average pressure distribution for each divided region.

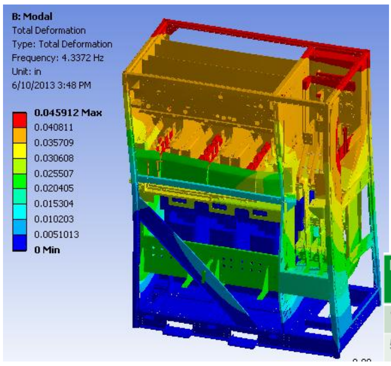
TABLE II
AVERAGE PRESSURE DISTRIBUTION

Reaction Force and Average contact pressure on contact regions						
Contact region	Vertical reaction force at Contact (lbf)	Area (in^2)	Average contact pressure (psi)	Maximum contact pressure (psi)	Average contact pressure (psf)	Maximum contact pressure (psf)
1	871	62.32	13.98	220.20	2012.58	31708.80
2	1910	118.37	16.14	183.80	2323.56	26467.20
3	1477.1	64	23.08	262.00	3323.48	37728.00
4	95	61.8	1.54	21.00	221.36	3024.00
5	1135	126.03	9.01	164.00	1296.83	23616.00
6	671.6	61.8	10.87	183.30	1564.89	26395.20
7	939	89.7	10.47	172.20	1507.42	24796.80
8	1141	80.47	14.18	134.00	2041.80	19296.00
9	1375	94.7	14.52	131.00	2090.81	18864.00

The regions 2,3,5,6 are the regions just below the transformer which shows more value for contact pressure.

B. For Seismic analysis

Modal Analysis and Frequency modes



Mode	Frequency, Hz	Description
1	4.33	side to side
5	7.95	Front to Back

Figure 10: Modal Results

It is clear from modal results that, horizontal loading (X and Z) will be more severe. Also it is noted that, major mass contribution was observed in Z and X direction with 66 % and 87 % respectively.

Direction loading results

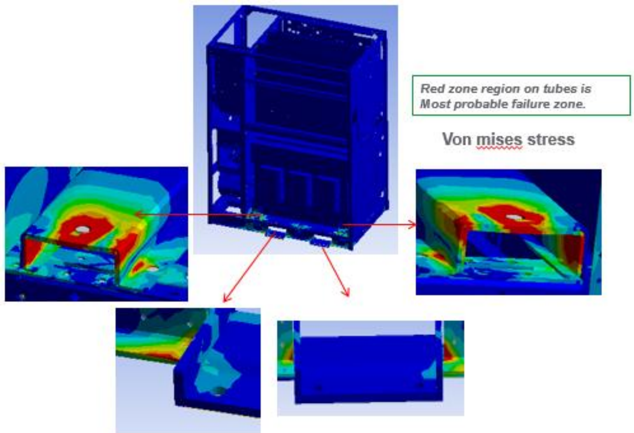


Figure 11:Z (Front to back) loading results

The red region shows that stresses are going above 340 MPa (49313 psi) limit, which are most probable zone for failure.

The tube channel on which Transformer sits is the weakest region. Also anchors which are attached to base of cabinet are showing some red region.

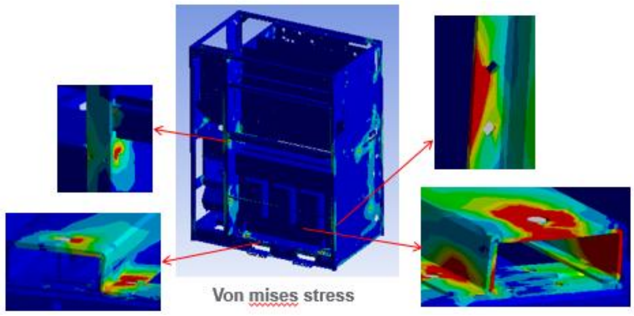


Figure 12:X(Side to Side) loading results

Apart from tube channels which are supporting transformer are going above yield limit but also, vertical channels are also showing stresses above safe limit.

VI. COMPARISON WITH TEST

Test was carried out on similar type of cabinet which has shown very similar results with FEA results. The failure regions for test and ansys model are showing same region.

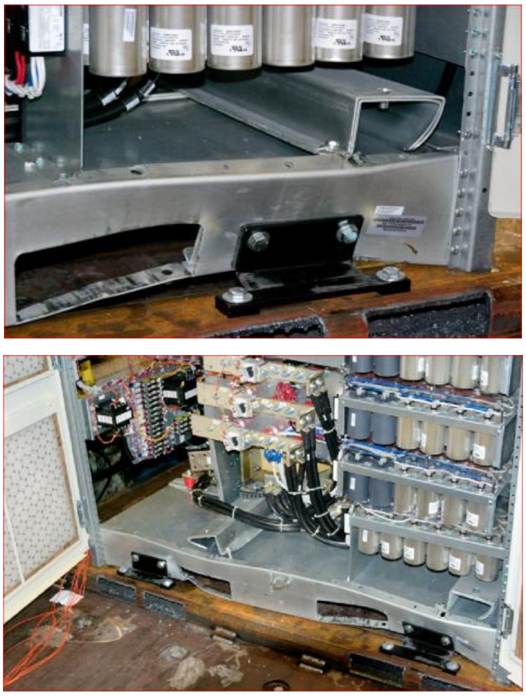


Figure 13: Seismic test done at laboratory

It is clear from test that; tube channels have shown considerable deformation causing failure. Also anchor points which are attached at base of cabinet are showing large deformation.

VII. Conclusion

Vertical loading shows small regions on Tube channel where stress regions are above yield limit.Front to back loading (Z loading) tubes are most probable region of failures. Also region near anchor plate is above yield limit.

Side to side loading (X) shows small stress regions developed near right front channel, partition panel. Tubes are also most probable regions of failure. There seems some lifting of Tube channel on which transformer is sitting.

The methodology described above has shown failure zones matching with test lab results. As seismic testing in laboratory is costly, probability of failure can be predicted using FEA method. It will not only show failures regions but also, it can be used for checking new iterations with modification in design. As most of power products needs to be certify for seismic compliances, this method will be very useful.

References

[1] https://law.resource.org/pub/us/code/ibr/icc.ibc.2009.pdf 2009 International Building Code.

[2] P. Busch, E. Carballo, H. Degree, “Seismic Design of Buildings Worked examples” Workshop “EC 8: Seismic Design of Buildings”, Lisbon, 10-11 Feb. 2011.

[3] Viral N. Talati1 M.Y.Patil 2 Chirag P. Mistry “Methodology for Seismic Qualification of Non-structural Components” IJSRD - International Journal for Scientific Research & Development| Vol. 2, Issue 03, 2014.

[4] Jingyao Zhang , Makoto Ohsaki and Atsushi Uchida “Equivalent static loads for nonlinear seismic design of spatial structures” The 14thWorld Conference on Earthquake Engineering October 12-17, 2008, Beijing, China

[5] Mehmed Causevic, Sasa Mitrovic “Comparison between non-linear dynamic and static seismic analysis of structures according to European and US provisions” Bull Earthquake Eng DOI 10.1007/s10518-010-9199-1.