Structural strength of voltage breaker cabinet assembly using Finite Element Method

Praveen S[1*], Sahadeva G N[2],

[1*]PG Scholar, Department of Mechanical Engineering, East Point College of Engineering and Technology, Bangalore-560049, Karnataka, India

[2]Professor, Department of Mechanical Engineering, East Point College of Engineering and Technology, Bangalore-560049, Karnataka, India

Abstract

The low voltage breaker utilized in and after the transformer are tried dependent on the IEC 60068-3-3 (International Electrotechnical Commission) for example seismic norms. The structural strength of the low voltage breaker is contemplated utilizing FEA where the whole assembly of the breaker is modelled through FEA discretization called nodes and elements.The entire get together is energized biaxially for example in two ways, the directions are considered as X & Y axis.

ANSYS recreation programming used to pre-process the get together for example for model structure reason and the post preparing movement did in the equivalent.

The goal of the examination is to investigations the disappointment area, correspond with the test outcomes and propose another attainable plan to withstand the standard excitation according to IEC report.

The structural strength or a vibration study is a non-linear analysis, also the non-linear finite part analyses, material non-linearity is turning into a benchmark study and thence the analyses is disbursed to derive a target answer.

The check results

are diagrammatically delineated and mentioned within the later a part of the report. The projected possible style is insight of a case study wherever the client has all the authority to simply accept new design. the whole case study is internal, and also the simulation results are related to IEC commonplace document and boundary conditions

are supported the check setup.

Key words: IEC 61439-1-2, FEA, ANSYS

1. Introduction

The low voltage breaker used to cut off the voltage during in and out of the transformer. The power feeder of the equipment utilized in the FEA study has 6300A capacity. The motor control is up to 250kw and main busbar is up to

7000A and also rated voltage up to 690V

It optimizes the motor performance and also efficiently save in terms of energy and fault prevention. The equipment is certified as per IEC 61439-1 & -2. It also has thermal monitoring. The equipment is good with medium voltage hardware and canalis control busways.

It contributes in the security reason however and also in unwavering quality and coherence of administration of the electrical establishments. It also a return to investment by helping to make electrical networks reliable which helps out in improving energy efficiency

The low voltage breaker has range of applications in the field of large industrial sites, commercial buildings & infrastructures. As the application of the instrument in heavy industries, it has to sustain seismic standards i.e. up to the certain level of Richter scale, the assembly has to withstand without any friction between the cabinets. If there is any friction of the cabinets then the chance of short circuits of busbars and fire catch up in the breaker and hence resulting in the destruction of the whole building or location where the breaker placed.

As per a seismic standard the get together of breaker should have the capacity to withstand the level 3 excitement where the assembly is experienced to the vibration along two directions i.e. biaxial. The Ansys model taking as the reference, it is considered biaxial in terms of X & Y axis.

The failure criteria or location is calculated using FEA method & new design proposal given to withstand the failure. The below Fig01 is the FEA model of low voltage breaker assembly

The challenge in the case-study to identify the high effected regions in the assembly when it is subjected to biaxial excitation considering X→side to side and Y→ vertical excitation. The examination is to know the auxiliary quality of the low voltage breaker get together for IEC 60068-3-3 Seismic principles.

<u>Bi-Axial (X and Y Direction):</u> (X-Side to Side and Y-Vertical Excitation) - Level 3

- Maximum deformation of **"30.06mm"** is at the Cabinet-3 outer panel right side location.
- High stress is observed at the **"Fixing locations, N32 breaker mass location and rear panel brackets and bolt location"** locations as the frequency of 5.06Hz is having mass contribution in this direction.
- The above observations are made for the frequency range between 0 to 26Hz in performing the seismic analysis.

Bi-Axial (Z and Y Direction): (Z-Front to Back and Y-Vertical Excitation) - Level 3

- Maximum deformation of **“20.75mm”** is at the cabinet-3 location (Inside plate).
- High stress is observed at the **“Fixing locations, N32 breaker mass location and N16 breaker mass location”** locations as the frequency of 7.79Hz is having mass contribution in this direction.
- The above perceptions are made for the recurrence go between 0 to 26Hz in playing out the seismic examination.

We have simulated for Biaxial seismic loads as previous testing conducted for Biaxial loads. In any case, notwithstanding this we have recreated for Uniaxial and Triaxial as a feature of extra demand from the originator and abridged in reference section.Stresses are calculated for Linear material which is not the real material behavior after the yield limit. To take the non-linear material effect into account, Neuber formula has been used to calculate the plastic strain.Plastic strain of the material is within the material breaking limit, this shows that material will yield to plastic region and have permanent deformation within material breaking.

2. Experimental Work

Materials used

The Linear material property is considered for the Structural Steel material. Linear Steel Property is used for beam connection.Total weight of the voltage breaker assembly = 1625kg.The above mass details are given based on the inputs from the designer.

The weights of the Copper busbar are calculated based on material density (ρ = 8.96 g/cm3

The structural material details are tabulated as below.

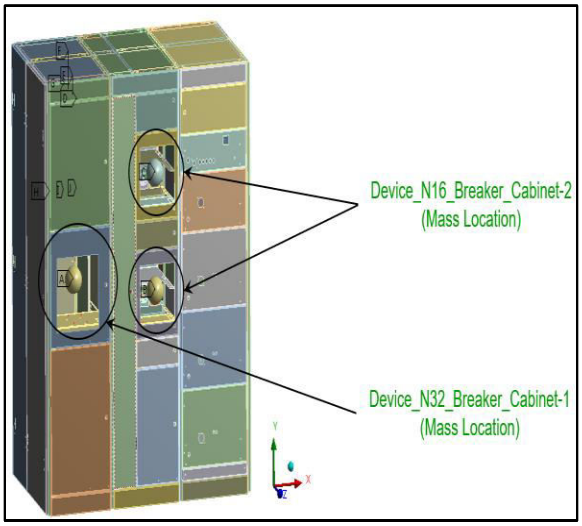
Table1. Structural steel material mechanical properties

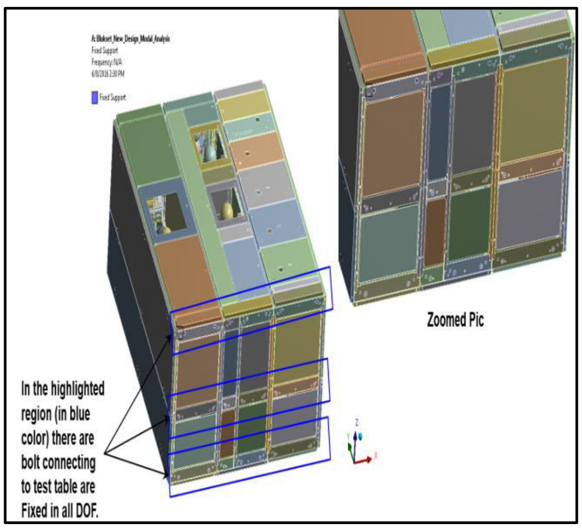
Young’s modulus	Poisson’s ratio	Yield strength	Density
2e5 MPa	0.3	235 MPa	7.85 g/cc

Assumptions and Boundary conditions used in analysis

Linear material properties are considered. Contacts between parts are not included.

Rigid Fixations are considered at mounting location. Fasteners are represented as beam elements. Point mass is used as representing structures for devices or any other structure based on simulation engineer experience. Test duration per frequency: Testing Time will not be considered in the simulation. Fatigue effect will not be included in the simulation.Stress which is higher than the material yield is not the actual values because of linear material assumption. Therefore,material crossing yield are considered as weaker section.





The above figure indicates the location of the masses considered in the simulation. In cabinet-1, we have Device_N32_Breaker (location of this mass is shown above) and in cabinet-2, we have Device_N16_Breaker (Location of mass is shown above). In Addition to this, we have considered busbar weights as point mass in cabinet-1 = 202.5kg, cabinet-2 = 226kg and cabinet-3 = 60kg respectively. The fixation areas are the regions where the parts are connected through bolts. We can find the fixation areas in the above figure highlighted.

Load cases

Horizontal Spectrum		
Frequency	DSD Level Acc. (g)	SDD Level Acc. (g)
0.5	0.1	0.2
1	0.3	0.7
3	2.5	5
10	2.5	5
36	0.5	1
100	0.5	1
Duration of corresponding accelerogram		20s

Vertical Spectrum		
Frequency	DSD Level Acc. (g)	SDD Level Acc. (g)
0.5	0.04	0.1
1	0.2	0.35
4	2	4
15	2	4
32	0.4	0.8
100	0.4	0.8
Duration of corresponding accelerogram		20s

Table → Seismic tests (DSD SDD levels)

X, Y and Z – Direction (X and Z Horizontal, Y is vertical)Reference Standard: IEC60068-3-3 and IEC 60068-2-57. The response loads (Frequency vs Acceleration) is taken from "IEC60068-3-3 and IEC 60068-2-57" standard document for test qualification. The loads highlighted in red box, is used for the seismic simulation in the blokset new design model.

Simulation is done for Biaxial seismic loads as previous testing are conducted for Biaxial loads. However, in addition to this we have simulated for Uniaxial and Triaxial as part of additional request from the designer and summarized in appendix. With reference to the assembly orientation, X and Z are considered as horizontal directions and Y is considered as vertical direction.

Modal results

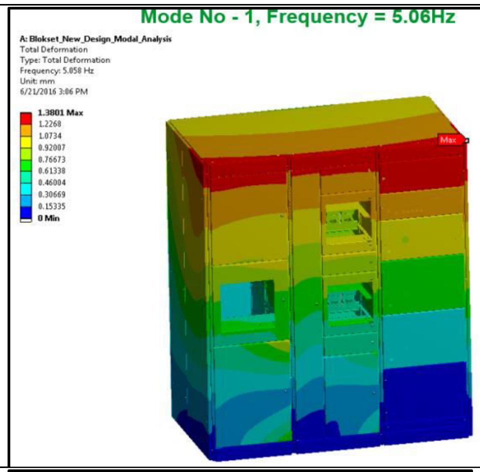
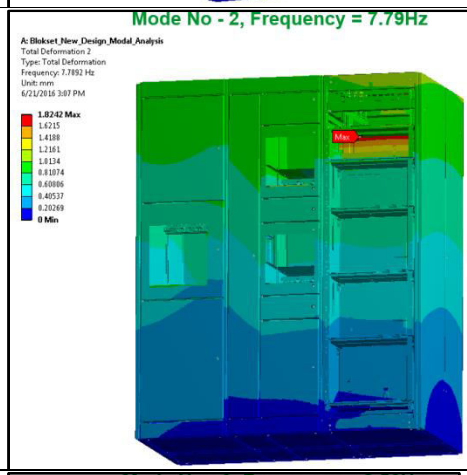
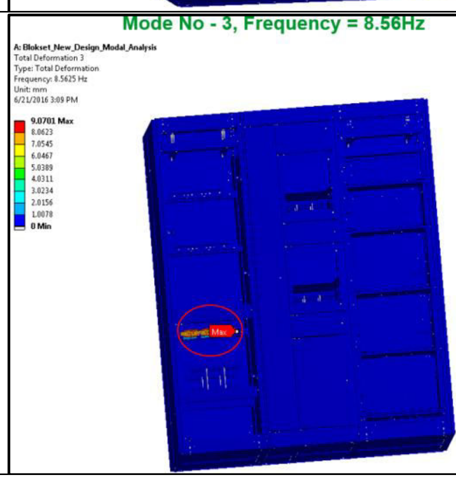
The total modal mass participation shown above are extracted for 60modes, which is 87.0%, 45.5% and 79.7% respectively. The max. participation is observed in the 1st mode for X-direction and 2nd mode for Z-direction (Horizontal directions). The max. participation for Y-direction is observed in 32nd mode (Vertical directions). Therefore,the seismic analysis between 0.5Hz to 26Hz. will be performed.

3. Results and Discussion

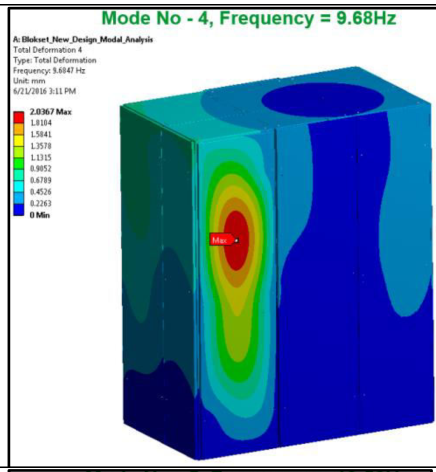
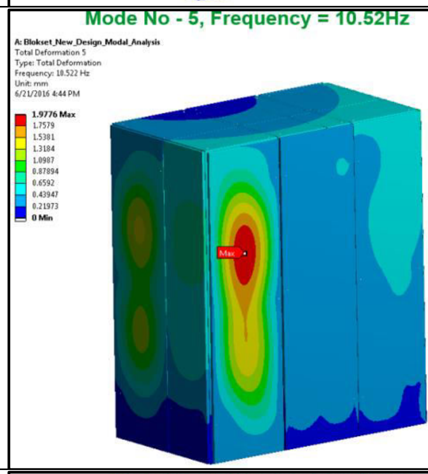
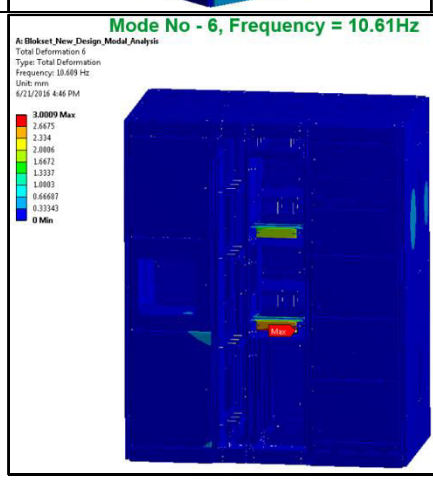
Mode No.	Frequency (Hz)	Mass Participation in X-Dir (%)	Mass Participation in Y-Dir (%)	Mass Participation in Z-Dir (%)	
1	5.06	77%	0%	0%	Mass Dominant in X-Direction Side to Side
2	7.79	0%	2%	71%	Mass Dominant in Z-Direction Front to back
3	8.56	0%	3%	2%	
4	9.68	1%	0%	0%	
5	10.52	0%	0%	1%	
6	10.61	0%	0%	0%	
7	10.87	0%	0%	1%	
8	11.41	0%	0%	1%	
9	11.87	0%	0%	1%	
10	12.22	0%	0%	0%	
11	12.32	0%	0%	1%	
12	12.76	0%	0%	0%	
13	13.08	0%	0%	0%	
14	13.61	2%	0%	1%	
15	14.48	0%	0%	0%	
16	15.95	0%	0%	0%	
17	16.97	0%	0%	0%	
18	17.22	4%	0%	0%	
19	17.59	0%	0%	0%	
20	18.03	0%	0%	0%	
32	19.9	0%	12%	1%	Mass Dominant in Y-Direction
57	25.8	0.0	0.0	0.0	
58	25.9	0.0	0.0	0.0	
59	26.0	0.0	0.0	0.0	
60	26.1	0.0	0.0	0.0	
Total Mass Participation up to 60 modes (%)		87.0%	45.5%	79.7%	

Fig1→ Mass participation factor

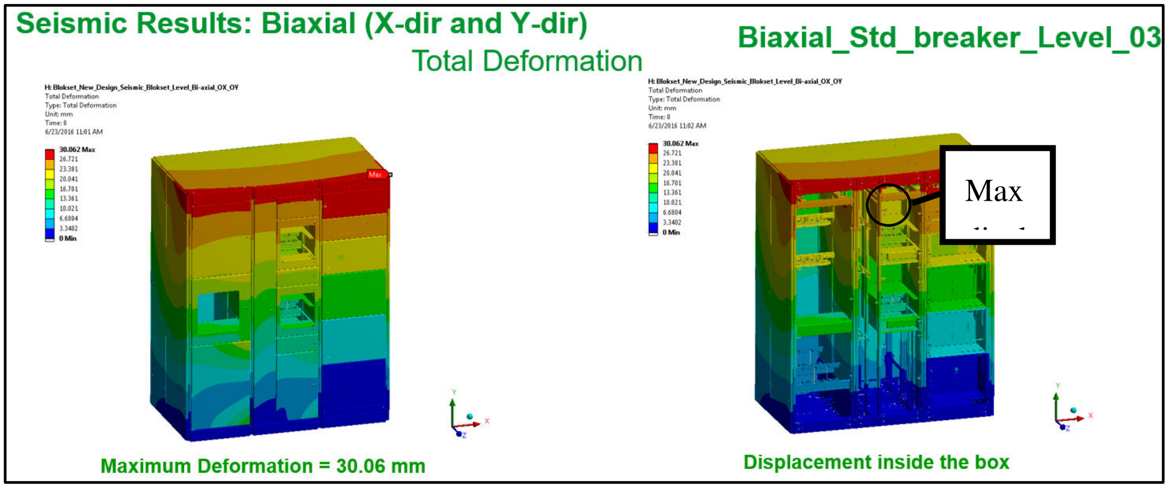
Fig1 explains the mass participation factor of the assembly

MODE1		**This mode is observed at the Cabinet-3, top location**
MODE2		**This mode is observed at the Cabinet-2, Inner part**
MODE3		**This mode is observed at the Cabinet-1, Bus bar location and it's a local mode.**

International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 03 Issue: 09 | Sept -2019 ISSN: 2590-1892

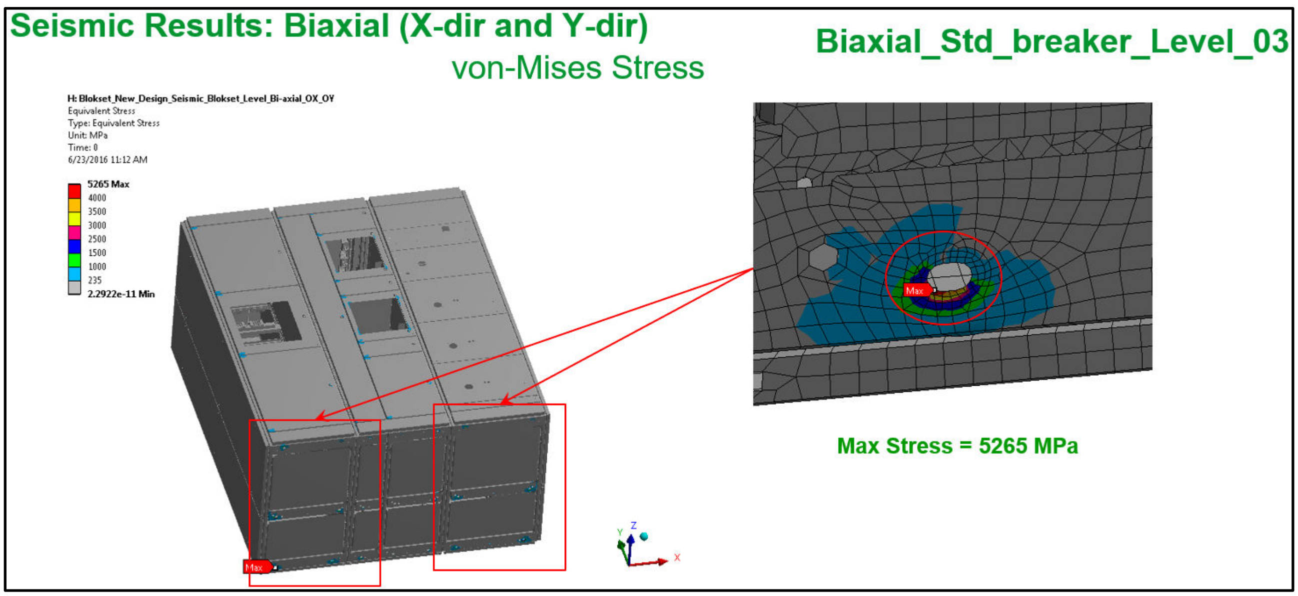
MODE4		**This mode is observed at the Cabinet-3, Outer panel Rear side location.**
MODE5		**This mode is observed at the Cabinet-3, Outer panel Rear side location**
MODE6		**This mode is observed at the Cabinet-2, N16 Breaker Mass location**

TOTAL DEFORMATION

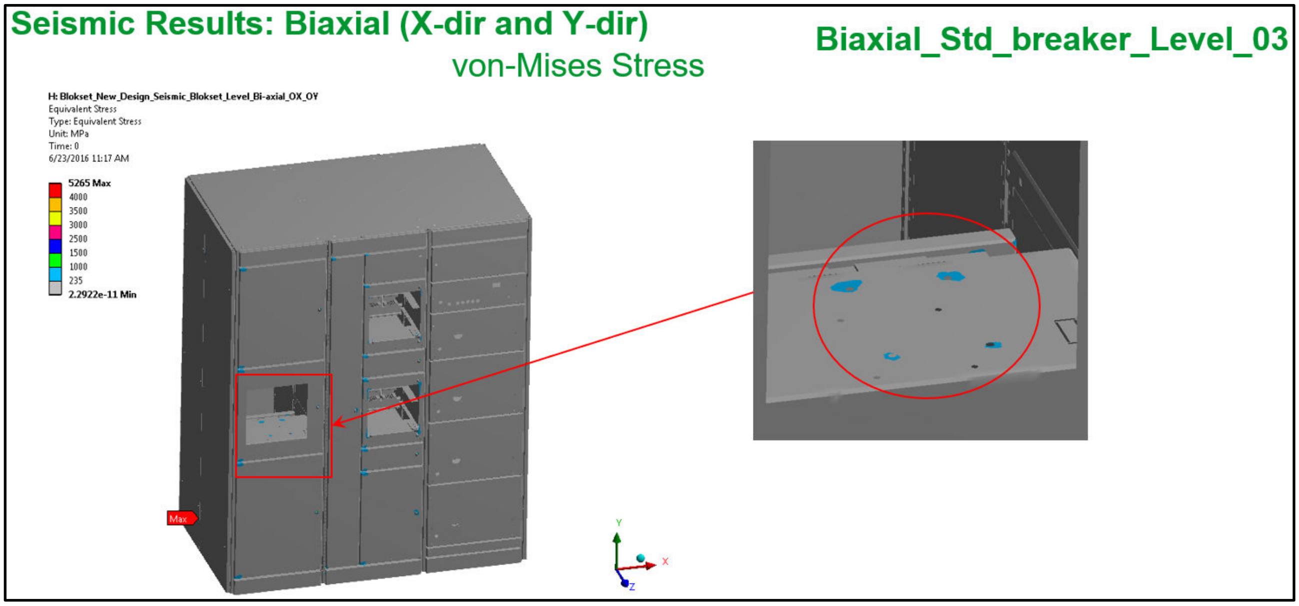


The maximum displacement is observed at the Cabinet-3 outer panel right side location

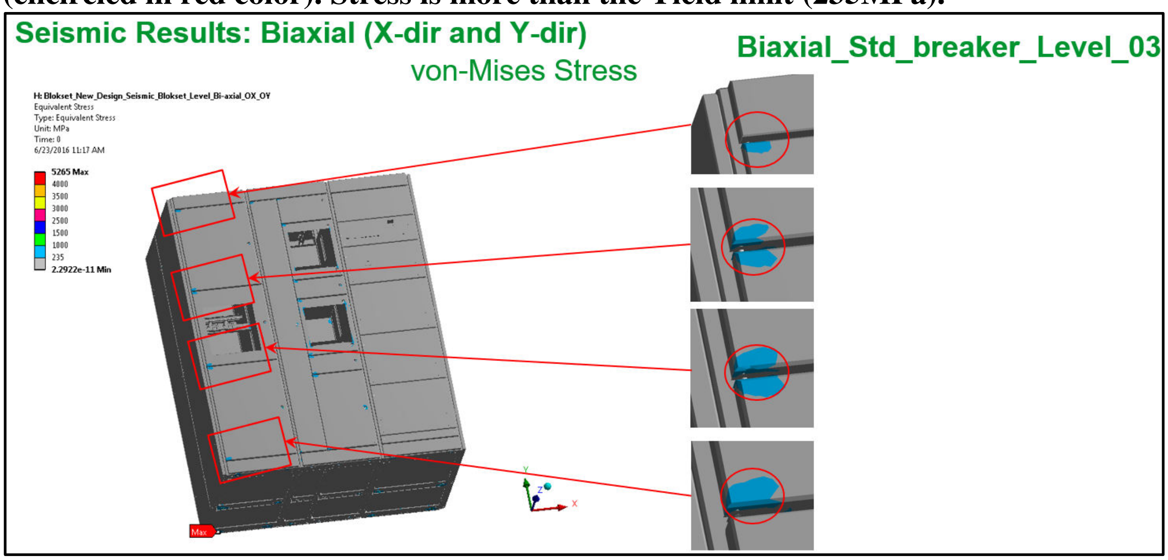
VON-MISES STRESS



- **Maximum Stress is observed at the fixing location on cabinet-1 is 5265MPa. Stress is more than the Yield limit (235MPa).**
- **Plastic strain at the maximum stress (5265MPa) is 56% which is more than the breaking limit (20%).**

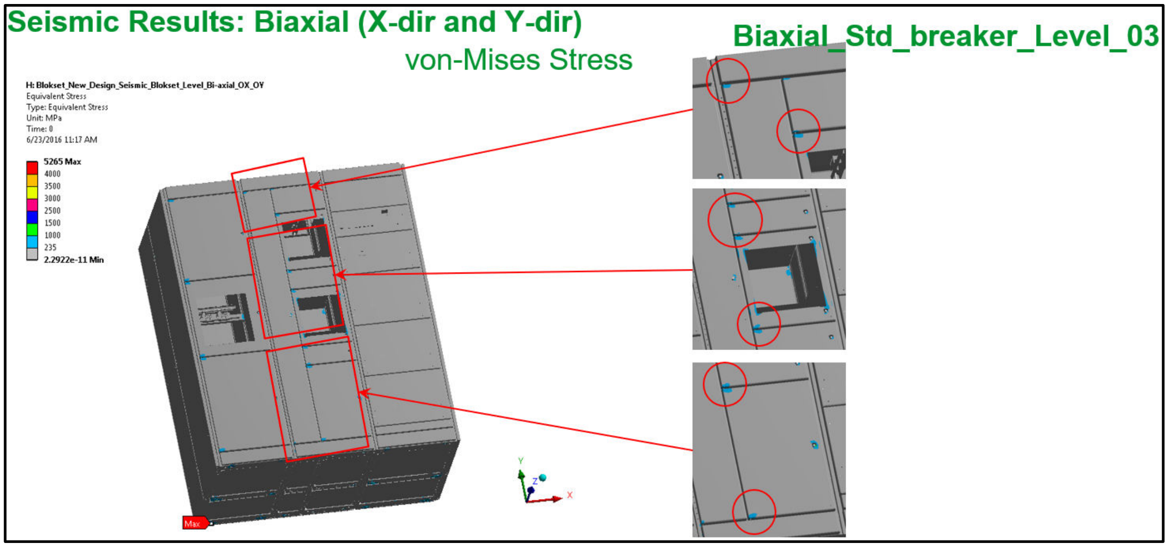


Peak Stress is observed on the fixing of N32 breaker in cabinet-1 (encircled in red color). Stress is more than the Yield limit (235MPa).

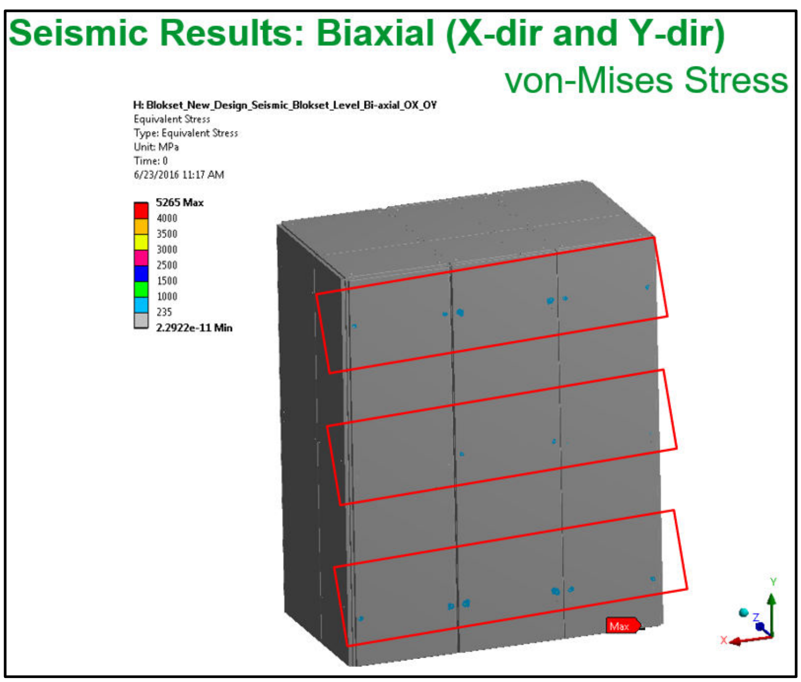


Peak Stresses are observed on the door hinges of cabinet-1 (encircled in red color). Stresses are more than the Yield limit (235MPa).

International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 03 Issue: 09 | Sept -2019 ISSN: 2590-1892

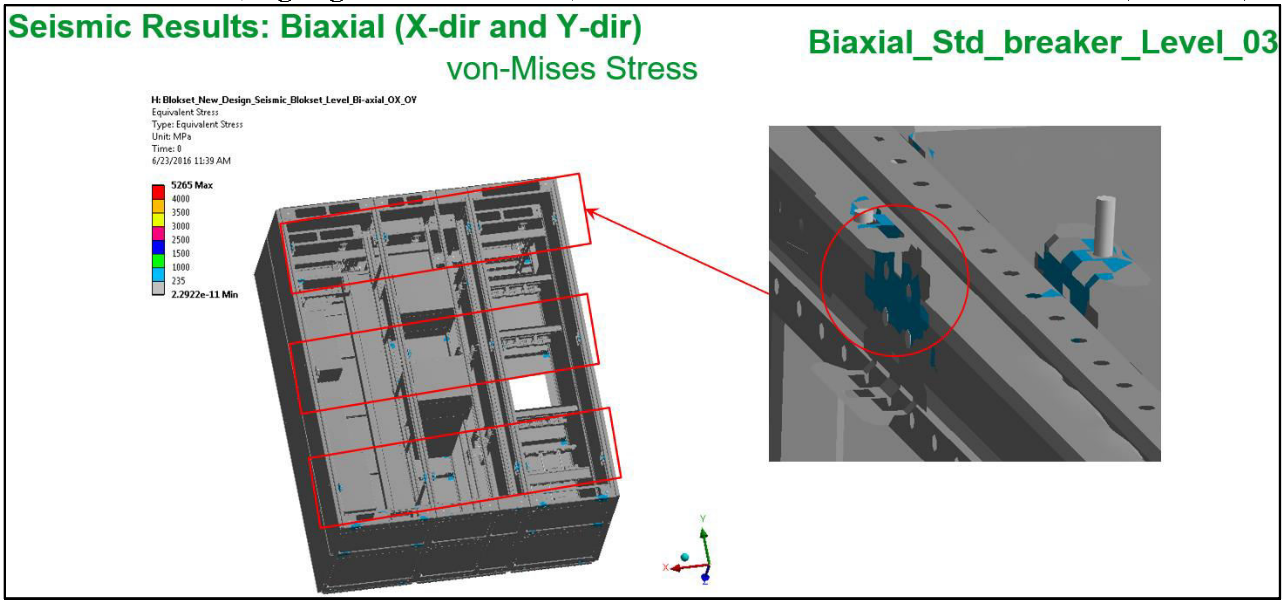


Peak Stresses are observed on the door hinges of cabinet-2 (encircled in red color). Stresses are more than the Yield limit (235MPa)



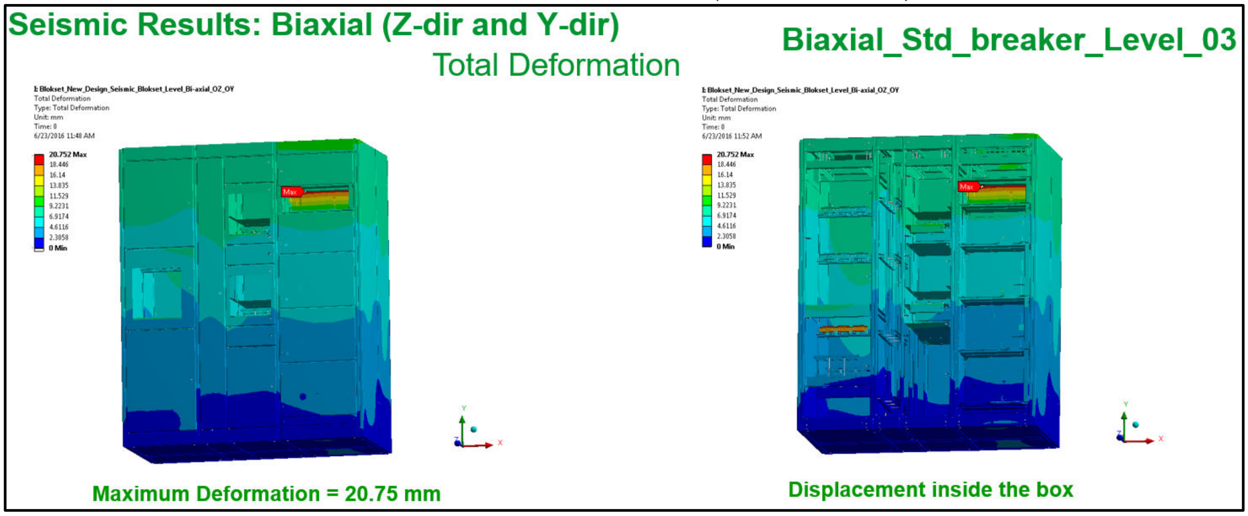
Peak Stresses are observed on the rear panel bolt location in cabinet-1, cabinet-2 and

cabinet-3 (Highlighted in red color). Stresses are more than the Yield limit (235MPa).



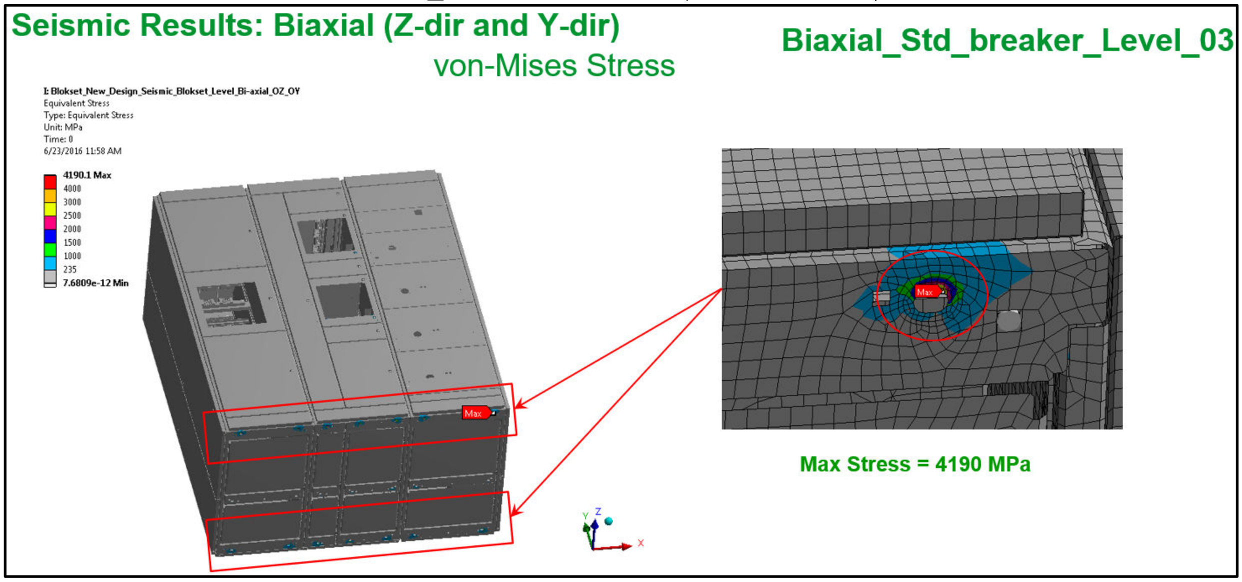
Peak Stresses are observed on the rear panel brackets in cabinet-1, cabinet-2 and cabinet-3 (Highlighted in red color). Stresses are more than the Yield limit (235MPa).

TOTAL DEFORMATION (Z-Y direction)

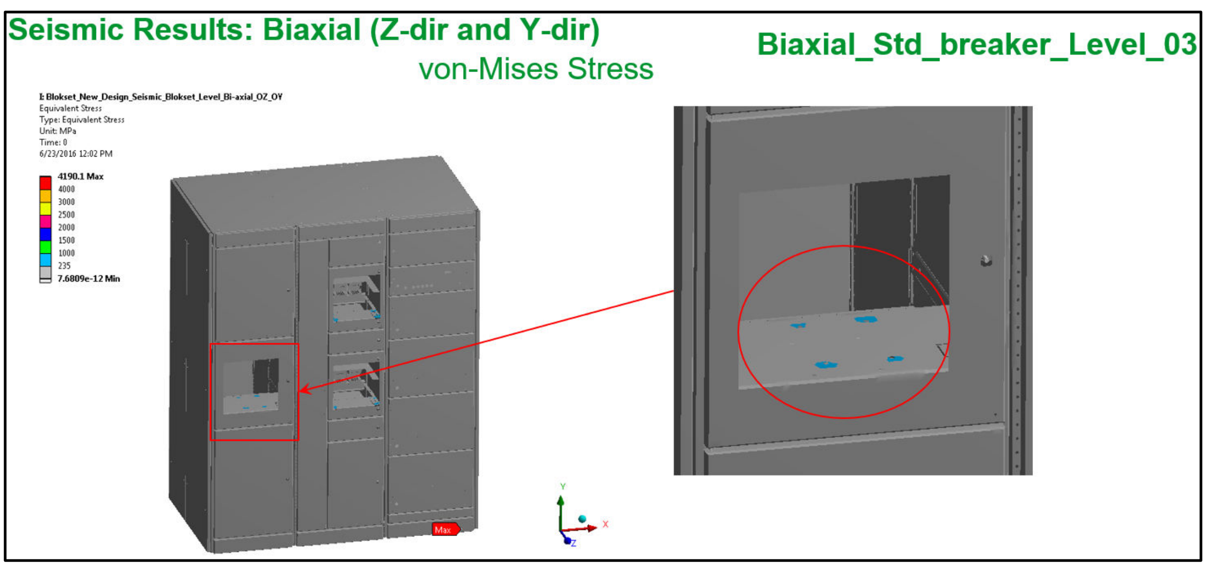


The maximum displacement is observed at the cabinet-3 location (Inside plate)

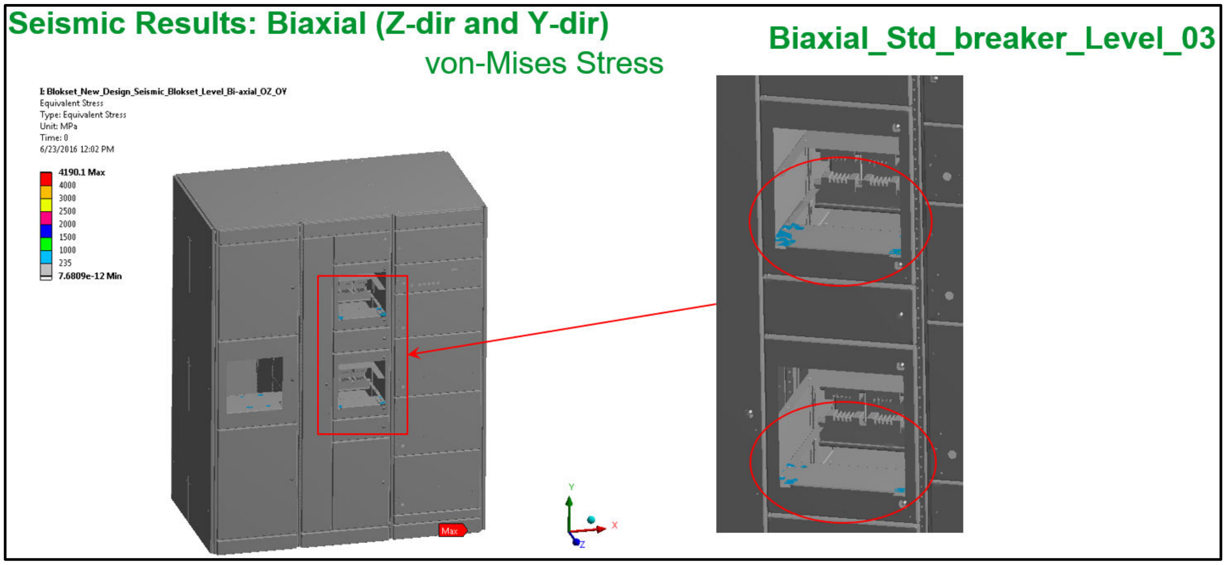
VON_MISES STRESS (Z-Y direction)



- **Maximum Stress observed at the fixing location on cabinet-3 is 4190MPa. Stress is more than the Yield limit (235MPa).**
- **Plastic strain at the maximum stress (4190MPa) is 36% which is more than the breaking limit (20%).**



Peak Stress is observed on the fixing of N32 breaker in cabinet-1 (encircled in red color). Stress is more than the Yield limit (235MPa).



Peak Stress is observed on the fixing of two N16 breakers in cabinet-2 (encircled in red color). Stress is more than the Yield limit (235MPa).

4. Conclusions

The present work on vibrational and structural strength analyses the peak stress is observed at the cabins of voltage breaker during biaxial loading stage.FEA results of the stress analyses and frequency of modal analysis is matching with the failures occurred in the test conditions. Design proposals is been given to overcome the failures at the cabinet areas as shown in the FEA simulation result. Future of scope is to modify the deign conditions and subject it to FEA analysis for the detailed study. ANSYS workbench is used to solve and post process the FEA results of the assembly. In view of the above plan recommendations it tends to be presumed that the FEM by and large and business FEA programming ANSYS specifically is a brought together approach for dynamic investigation (vibration study and structural strength analysis). In this study correlation of the numerical results of FEA was restricted to limited experimental results (Test results not disclosed as per NDA commitments).Be that as it may, there is a genuine requirement for advanced test examinations to check every one of the outcomes exhibited here. This is recognized as future work.

REFERENCES

[1] Sahadeva G N, "Professor, Head of Mechanical Engineering Department, EPCET"

[2]IEC standard document "IEC system of conformity assessment schemes for Electrotechnical equipment and composites"

[3]Anon., 2011. Stochastic Simulation and

Lagrangian Dynamics. [Online] Available at: http://stochasticandlagrangian.blogspot.fi/2011/07/what-doesshape-function-mean-in-finite.html [Accessed 30. January 2018].

[4]ANSYS Inc., 2017. ANSYS Help Guide, s.l.: s.n.

[5] Liu, G. R. &Queck, S. S., 2003. The Finite Element Method: A Practical Course. s.l.:Butterworth-Heinemann

[6]Pataik, S. N. & Hopkins, D. A., 2004. Strength of Materials: A Unified Theory. s.l.:Butterworth-Heinemann.

[7]Petyt, M., 2010. Introduction to Finite Element Vibration Analysis. 2 ed. New York: CamebridgeUiversity Press.

[8]Rieg, F., Hackenschmidt, R. &Alber-Laukant, B., 2014. Finite Element Analysis for Engineers. Munich: Carl Hanser Verlag.

[9]Steinke, P., 2015. Finite-Elemente-MethodeRechnergestützteEinführung. 5. ed. Steinfurt: Springer Verlag.

[10]Wallin, M., 2014. A Finite Element tool for buckling analysis. Linköping: s.n.

[11]Nithin S Gokhale, 2007. Practical Finite Element Analysis. First Edition. Publisher : Finite to Infinite