Research Paper

Volume : 2 | Issue : 6 | June 2013 • ISSN No 2277 - 8179

Spectrum/Seismic Analysis of High Voltage 145Kv Circuit Breaker Using Finite Element Method

Engineering

KEYWORDS :

D.M.Garasiya	ME CAD/CAM (Mechanical), IVth Semester, GEC Dahod (Gujarat), India
K. K. Bhabhor	Asst.Professor, Mechanical dept., GEC Dahod (Gujarat), India
A. B. Damor	Asst.Professor, Mechanical dept., BVM Engg.College (Gujarat), India

ABSTRACT

Circuit breakers are expensive and vital equipments in substations; thereby calls for safe functioning of these equipments, especially during limit conditions, like earthquakes. This paper describes the verification of HV circuit breaker with OM according to IEC TR 67721-300 AF5. The verification has been performed using FEA software Ansys. The paper describes the basic facts about the analyzed breaker as well as the considered load cases and its results. Seismic analysis by well-known 'response spectrum method' yields reliable results only when correct values of natural frequency and damping or fraction of critical damping of the structure are available. Therefore, it is important to first extract the natural frequencies and Corresponding mode shapes for first few dominant modes.

Introduction

Circuit breakers are expensive and vital equipments in substations; thereby calls for safe functioning of these equipments, especially during limit conditions, like earthquakes. Circuit breaker producers face major difficulty in integrating flexibility and resistance of the structure in order to achieve superior dynamic characteristics and Earthquake resistance. Also, circuit breakers should remain operational during different types of earthquake in order to maintain power supply for large territorial areas and prevent fire Disasters.

This paper describes the verification of HV circuit breaker with OM according to IEC TR 67721-300 AF5 [1]. The verification has been performed using FEA software Ansys.

The paper describes the basic facts about the analyzed breaker as well as the considered load cases and its results.

The seismic verification as per IEC TR 67721-300 includes the different load cases (i.e. Static Terminal forces, Wind load, dead weight and Dynamic response spectrum analysis). The different load cases calculated separately and results combined to get the cumulative effects of the all load cases.

Finite Element Model (Mesh Model)

ANSYS model of the circuit breaker in shown in Fig 1. The model was developed in PRO-e WILDFIRE-4, which was imported to ANSYS for analysis. Solid part of the circuit breaker, interrupter porcelain, support porcelain have been modeled by solid element (SOLIDSL95) while, circuit breaker frame has been modeled by shell element (SHELL181).All parts less than 20 mm meshed with 4 node shell elements and other regions meshed with 8 node hexahedron solid elements. The bolts are simulated with Beam and rigid elements. Drive unit and Mechanism Housing masses has been considered as point masses at the C.G. of the units and connected with rigid elements to simulation. Bolts are meshed with solid circular section beam element of the respective diameter of each bolt.

- Drive unit and Mechanism Housing masses has been considered as point masses at the C.G. of the units and effect of the masses transferred with the rigid elements connected to the respective regions, where effect has to applied.
- All the geometry (plates) less than 20 mm meshed with 4 node shell elements and other meshed with 8 node hexahedron solid elements.
- All the contact regions meshed with frictionless contact, however contact status in the dynamic analysis considered based on the initial status of the contacts.

The insulators simulated with shell elements, the density of the Insulators adjusted to set the required mass of the Insulators with breaker units. The parts on which effect of the mass applied. Bolts are meshed with solid circular section beam element of the respective diameter of each bolt. All the contact regions meshed with frictionless contact. The insulators simulated with shell elements with appropriate thickness of the Insulator, the densities of the insulators adjusted to consider the mass of the breaking unit and insulator rod.

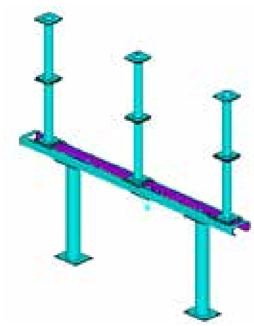

Figure 1.
Solid model for 145kv circuit breaker.

Assumptions and materials

There are following assumptions made for the seismic calculation of the HV breaker with OM.

- The mechanical housing flanges, Terminal flanges and all the insulator flanges are considered of aluminum.
- All the other parts are considered of Steel.
- All the bolts are simulated with Beam and Rigid elements, No pretention considered for the

Modal analysis by F.E.M

Modal analysis is the first step in earthquake simulation as it gives out natural frequencies of vibration of the structure and corresponding mode shapes, which are used in subsequent spectrum analysis.

The equation of motion for an un-damped system, expressed in matrix notation is:

$$[M]\{\ddot{u}\}+ [K]\{u\}= \{0\} \quad (1)$$

Where:
[M] - the Mass matrix;
[K] - the stiffness matrix

For a linear system, free vibration will be harmonic of the form:

$$\{u\} = \{\varphi\}_i COS\, \omega_i t \quad (2)$$

Where:
$\{\}_i$ = eigenvector representing the mode shape of the i_{th} natural

frequency;
ω_i -i_{th} natural circular frequency (radians per unit time);

Thus, equation (1) becomes:
$(-\omega_i^2 [M] + [K])\{\}_i = \{0\}$ (3)

Or
$|[K]-\omega^2[M]| = 0$ (4)

This is an eigen value problem which may be solved for up to n values of ωi and n eigenvectors $\{\}_i$ which satisfy equation (3) where n is the number of DOFs.

The natural frequencies (f) are:
$f_i = /2\pi$ (5)

Where f_i = ith natural frequency (cycles per unit time).

ANSYS program has several methods to perform modal analysis of a FEM model, but the best results of the simulation are offered by the Block Lanczos method which is suitable for processing models with big number of elements.

Material Properties
The material properties of the different parts of the breaker are summarized in Table 1.

Table-1
Material property table.

Property	Steel	Aluminium	Polymer /Porcelain
Young modulus (Mpa)	210000	70000	69000
Poisson's ratio	0.3	0.33	0.3
Density (Tonnes/mm3)	7.85E-09	2.70E-09	***
Yield (Mpa)	220/355*	220**	-

*The top plates of the support legs have yield strength of 355 Mpa, while the pole beam and leg material (EN 10025-S235JR)

** For the aluminum casted flange material (EN 1706-AC-43100)

*** Density adjusted to consider the weight of the breaker units

Loading conditions
The loading conditions according to IEC/TR 62271-300-AF5 [1], need to investigate two load cases, these are:

Load case-1:
- 100 % Seismic response spectrum in horizontal direction1(X-direction as per the FEA model in Ansys).
- 50 % Seismic response spectrum in vertical direction (Y-direction as per the FEA model in Ansys).
- Static Terminal forces.
- Wind load (10 m/s in positive X-Direction).
- Self-weight of the complete breaker (Gravitation acceleration effect in vertical downward).

Load case-2:
- 100 % Seismic response spectrum in horizontal direction-2 (Z-direction as per the FEA model in Ansys).
- 50 % Seismic response spectrum in vertical direction (Y-direction as per the FEA model in Ansys).
- Static Terminal forces.
- Wind load (10m/s in negative Z-Direction).
- Self-weight of the complete breaker (Gravity effect in vertical downward).

In the all the above load cases the all degrees of freedom fixed at the bottom foundation plate. The Terminal forces and gravity load remain same in the both load cases.

The static terminal forces are the forces given in IEC62271-100 for rated voltage 100-170 KV and rated current 2500-4000 A. As stated in IEC 62271-100, these forces are multiplied with a factor 0.7 in order to account for a wind load of only 10 m/s. The resulted terminal forces considered for the calculation.

The wind pressure p on an object can be calculated based on the following formula.

p= CD*(Density* U02)/2(5)
Table 2
Calculation of wind pressure

	Flat surface	Cylindrical surface
Drag coefficient C_D	1.5	0.8
Density of air	1.2 kg/m3	1.2 kg/m3
Speed U_0	10 m/s	10 m/s
Pressure p	90 pa	48 pa

The Breakers own weight is taken into account and the gravitational acceleration of 9810 mm/s2 applied in the positive vertical direction, in Ansys the effect of the self-weight transfer to the opposite direction of the applied gravitational acceleration, effect in this case vertical downward direction.

The response spectrum need to apply at the fixed support(Ground /Foundation) and the required response spectrum calculated from the response curve as give in the figure 8.The response of the response spectrum on the structure calculated differently and then combined using SRSS method. To consider the dynamic effect due to its free vibration the modal analysis and Eigen mode calculation done prior to spectrum analysis, the number of enough Eigen modes calculated to consider at least 90 % of full effective mass.

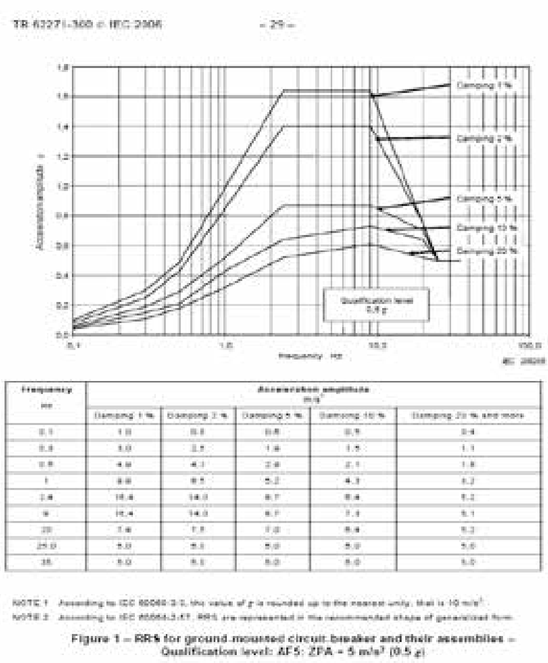

Figure 2
RRS for ground mounted circuit breaker and their assembly-Qualification level: AF5: ZPA = 5m/s²(0.5g)

Volume : 2 | Issue : 6 | June 2013 • ISSN No 2277 - 8179

Ansys APDL command

The following Ansys APDL command shows the apply spectrum value and direction.

For load case-1:

FREQ,0.1,0.3,0.5,1,2.4,9,20,25,35 ! Frequency points for SV vs. freq. table

SV,0.02,800,2500,4300,8500,14000,14000,7500,5000,5000
!Spectrum values associated with frequency points
!
SED, 1, 0, 0 ! 100 % in X direction
!
SED, 0, 0.5,0 ! 50 % in Y direction

For load case-2:

FREQ,0.1,0.3,0.5,1,2.4,9,20,25,35 ! Frequency points for SV vs. freq. table
!
SV,0.02,800,2500,4300,8500,14000,14000,7500,5000,5000
!Spectrum values associated with frequency points
!
SED, 0, 0, 1 ! 100 % in Z direction
!
SED, 0, 0.5,0 ! 50 % in Y direction

Results and discussion

Seismic analysis by well-known 'response spectrum method' yields reliable results only when correct values of natural frequency and damping or fraction of critical damping of the structure are available. Therefore, it is important to first extract the natural frequencies and Corresponding mode shapes for first few dominant modes.

The results comparison for the both load cases are in the following table 3 and 4, the maximum equivalent stresses for the component of structure compared with the yield stresses of the respective materials.

Table 3
Results comparison for load case-1

Components	Max. Stress (Mpa)	Yield Stress (Mpa)	Safety factor
Support Leg	50	220	4.4
Top support leg Plate	70	355	5.07
Pole beam	151	220	1.45

Table 4
Results comparison for load case-2

Components	Max. Stress (Mpa)	Yield Stress (Mpa)	Safety Factor
Support Leg	153	220	1.43
Top support leg Plate	105	355	3.38
Pole beam	153	220	1.44

Observations and conclusion

From the above results comparisons, the following are the observations and conclusions for the seismic verification of HV breaker with OM.

The max. Von-mises stress in the pole beam is 250 Mpa, which is localized due to rigid connection for bolt support and stress concentration, it might not appear in actual case.

Load case-1:

The actual maximum von-mises stresses after removal of stress concentrated elements is 151 Mpa in the support leg and 50 Mpa in the drive support leg.

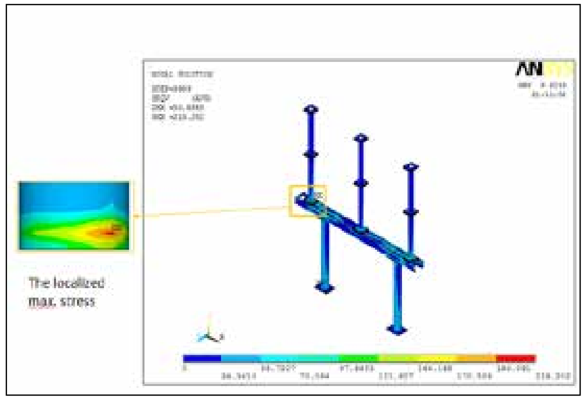

Figure 3.
von-mises stresses

Load case-2:

The actual maximum von-mises stresses after removal of stress concentrated elements is 153 Mpa in the pole beam and 84 Mpa in the support leg.

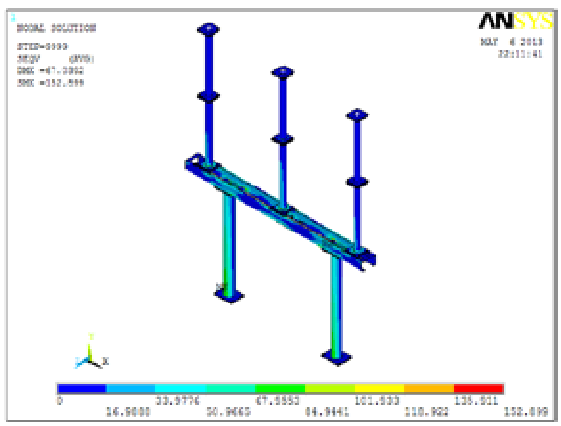

Figure 4.
von-mises stresses

From the above results, the maximum stresses in the bodies within the yield limit of the respective materials

REFERENCE

[1]IEC TR 62271-300, "High voltage Switchgear and Control gear, Part-300: Seismic qualification of Alternating Current Circuit-Breakers." | [2] IEC 62271-100, "High-voltage switchgear and control gear, Part-100: Alternating-current circuit-breakers." | [3]Ansys-14 User guide and Help. | [4] Seismic Analysis of a Telecommunications Cabinet, Midwest Ansys User GroupMay 8, 2007 by Rick Fischer Emerson Network Power. | [5] Comparison of different methods in seismic qualification of electrical equipment by K.A. Kleine-tebbe, German Aerospace Research Establishment | [6] Singiresu S. Rao, Mechanical Vibrations, 3rd Ed., Prentice Hall, 1995, Englewood Cliffs, New Jersey