Finite Element Modeling & Analysis of Support Structure of Switchgear in consideration of Actual Site Loading Conditions

Amit Shende[1], Vaibhav Bankar[2]
[1]M. Tech Student [2]Professor
[1,2]Department of Mechanical Engineering
[1,2]Vidarbha Institute of Technology, Nagpur, India

Abstract— The improvement in the virtual manufacturing of machine and its components provides elevated support to the designers for understanding the behaviour of design in real world conditions and making decisions to best fit for that situation. This knowledge helps designers to build better design in terms of efficiency, reliability and cost. There are various computer aided designing i.e. CAD tools available and are capable of building the exact model of the intended design for study and analysis purpose. The purpose of this paper is to finite element modelling and analysis of support structure of switchgear in consideration of actual loading conditions using ANSYS.

Keywords*:* Support Structure, FEA, Switchgear, CAD

I. INTRODUCTION

Traditionally, engineers have used laboratory testing to investigate the structural behavior of steel products and systems subject to the expected wind, dead and Operation loads to develop appropriate design rules. Laboratory testing was essential to develop new products and systems.

However, such reliance on time consuming and expensive laboratory testing has hindered progress in this area. The product manufacturers and designers often decide to go on with conservative designs in order to avoid expensive and time consuming laboratory testing. This may lead to product with higher manufacturing cost, higher assembly time, complicated design and nonessential higher safety factor. However, advances in the field of computer aided design and engineering during the last two decades have changed this situation significantly in many engineering industries. In the industry, the use of advanced finite element tools has not only allowed the introduction of innovative and efficient products, but also the development of accurate design methods. During the last decade, Queensland University of Technology researchers have undertaken many collaborative research projects with their industry partners using advanced computing facilities and finite element tools, resulting in considerable benefits to the industries. In this paper, design of support structure of switchgear has been prepared using CAD software such as PRO-E and analyzed by using ANSYS considering the entire site loading conditions individually as well as in combined form.

Switchgears are very expensive and vital equipments in substations, thereby calls for safe functioning of these equipments during limit conditions, like high speed wind, mechanical operations and short circuit loads and etc. Switchgear producers face major difficulty in integrating flexibility and resistance of the structure in order to achieve superior dynamic characteristics and load resistance. Also, switchgear should remain operational during different types of loadings and loading combinations in order to maintain power supply for large territorial areas. Hence, support structure is essential part of the equipment which upholds the switchgear under all the unfavorable conditions.

The verification of High Voltage Switchgear is performed according to IEC 67721-100 [3] using FEA software Ansys. The different loads combined and applied to get the cumulative effects of the all load on the support structure.

II. LITERATURE REVIEW

A. Mahen Mahendran (2007):

In this paper, four examples of successful applications of advanced CAD tools are presented and discussed. The structural performance of steel cladding systems, innovative hollow flange beams, plasterboard lined cold-formed steel stud walls and an innovative cold-formed steel building system was accurately simulated by finite element models and thus considerably reduced the number of time consuming and expensive large scale experiments required. Both local and overall failures of various building components were investigated. This paper presents a summary of the four detailed investigations using advanced finite element tools, and selected results.

It has described the extensive use of finite element methods in investigating fully the structural performance of a number of building components and systems. By using four examples it has demonstrated the significant benefits of using finite element tools and advanced computing facilities in obtaining safe and optimum building solutions without the need for expensive and time consuming laboratory testing. It has demonstrated how the use of finite element tools has not only allowed the introduction of innovative and efficient building products but also the development of accurate design methods for use by engineers, manufacturers and designers in the building.

B. Rupesh Daripa (2011):

In this paper the author made an attempt to perform analysis using finite element method in Modal and Spectrum analysis of the 145kV circuit breaker structure. The computational simulation was carried out using FEM and ANSYS program which offers a standard operating procedure to perform modal and spectrum analysis.

According to the results found in this paper, in the switchgear the most vulnerable parts are the insulator support columns made of porcelain. Although high stresses developed near its cemented joint, carefully developed finite element model in Ansys by response spectrum method showed acceptable seismic performance and adequate margin of safety. These results will be helpful in optimizing the circuit breaker.

III. FINITE ELEMENT MODEL

3D model of the switchgear is shown in fig.1. The model was developed in Pro-E Creo 2.0. The switchgear comprises of Interrupting unit elevated by Supporting pole unit which is placed on Baseframe fastened by hardwares. The mechanism unit is attached to Baseframe on its vertical face. The Baseframe and Control box unit are placed on the support structure by using of nuts and bolts of various metric sizes. ANSYS model of support structure is shown in fig. 2. The model is imported to ANSYS workbench for the analysis purpose. The support structure selected for this study is made up of 2 C channel sections kept parallel to each other by welding the top and bottom plates of steel. Battens are welded between the C channel sections at regular distance. Stiffeners are welded in between the channels and plates for strengthening. The structure is meshed fine in hexahedron solid elements with total number of nodes are 27167 and total number of elements are 11091.

– All the dead weights of Interrupting unit, Supporting pole unit, Mechanism box unit, Baseframe and control box unit are considered at the C.G. of equipment. The support structure is then analyzed by considering the dead weight as a point load acting on the top of the structure.

Fig. 1: Solid model of the Switchgear.

Fig. 2: Finite element model of Support structure.

The material used for support structure is steel and its properties are mentioned in Table I.

Property	Steel
Young's modulus (MPa)	21000
Poisson's Ration	0.3
Density (kg/mm3)	7.85E-6
Yield Strength (MPa)	250/355*

Table 1: Material Properties

The top and bottom plates of the support structure have yield strength of 355 MPa, while for channel material (IS: 2062-E250).

IV. TYPES OF LOADINGS

The structure has to withstand number of loads as per IS: 875 are listed below:

- Dead load
- Wind Load
- Imposed loads
  - Operating load
  - Cable load
  - Short Circuit load

A. *Dead load-*

It is the weight of the equipment carried by the support structure. The dead load is considered as a point load concentrated at the C.G. of the equipment applied vertically on the structure.

B. *Wind Load-*

This is a force acts on the equipment due to the wind pressures in the direction of wind movement. The magnitude of wind load is mostly dependant on the Wind speed (m/s), projected area of the equipment and shape of surface facing the wind force. As per IS: 875, part 3 in India the highest wind speed is observed to be 55 m/s which is considered for calculations in this paper.

As per IS: 875 part 3, clause 5.3, design wind pressure is expresses as,

$$V_z = K_1 K_2 K_3 V_b \quad \text{(m/s)}$$

Where,

K_1 is Risk coefficient

K_2 is a factor dependant on Terrain category

K_3 is a Topography coefficient depends on slope of site

V_b is basic wind speed.

Hence, Design wind pressure P_z is expressed as

$$P_z = 0.6\, V_z^2 \quad \text{(N/m}^2\text{)}$$

C. *Imposed Loads*

Generally, these loads coming on the switchgear are vertical operating load, horizontal operating load, cable load and short circuit load.

1. *Operating Load –*

These are the forces acting on the equipment due to mechanical operations performed. The driving energy stored in the mechanism box unit attached to the baseframe unit which carries the high energy compression springs for moving the contacts in the interrupter unit. The supporting unit carries the mechanical linkages which connects drive unit to the Interrupting unit. These loads act in vertical Z-direction as wel as horizontal X-direction on the top of the support structure.

2. *Cable Load-*

These loads acts on the equipment due to the transmission line cables connected to its terminals for incoming and outgoing of electricity. Table 9 of IEC

62271-100 provides examples of static horizontal and vertical forces for static terminal load test shown below in Table II. Here we have selected the load values for 420-800 kV voltage level.

Rated voltage range U_r (kV)	Rated current range I_r (A)	Static horizontal force F_{th}		Static vertical force (vertical axis-upward & downward) F_{tv} (N)
		Longitudinal F_{thA} (N)	Transversal F_{thB} (N)	
52 - 72.5	800 - 1250	500	400	500
52 - 72.5	1600 – 2500	750	500	750
100 – 170	1250 – 2000	1000	750	750
100 – 170	2500 – 4000	1250	750	1000
245 – 362	1600 – 4000	1250	1000	1250
420 – 800	2000 – 4000	1750	1250	1500

Table 2: Examples Of Static Horizontal And Vertical Forces For Static Terminal Load Test

3. *Short Circuit load-*

These are the loads generated in the equipment due to short circuit currents produced. Short circuit of the system is most non favorable situation. Because of this short circuit condition the repelling forces are generated in the system.

As per the best practice, magnitude of short circuit forces is consider double to that of cable forces.

V. LOADING CONDITIONS

As per IS: 800, clause 3.4.2.1, load combinations are:-

1) Dead loads + Imposed loads
2) Dead loads + Imposed loads + Wind load (X dir.)
3) Dead loads + Imposed loads + Wind load (Y dir.)

A. *Load case – 1*

In this case of loading the combination of loads is applied at same instant on the top of support structure. Typical loads are summarized and shown below in Table III.

Sr. No.	**Loading**	**Vertical KN**	**Horizontal X KN-m**	**Horizontal Y KN-m**
1	Dead load	27.5		
2	Operating load	26.2	6.3	
3	Cable load (static)	1.5	8.5	6.1
	Total	55.2	14.8	6.1

Table 3: Load Case - 1

B. *Load case - II*

Typical loads are summarized and shown below in Table IV.

Sr. No.	**Loading**	**Vertical KN**	**Horizontal X KN-m**	**Horizontal Y KN-m**
1	Dead load	27.5		
2	Operating load	26.2	6.3	
3	Cable load (Dynamic)	3.0	17.0	12.1
4	Wind load (X-dir)		3.8	
	Total	56.7	27.0	12.1

Table 4: Load Case - 2

C. *Load case – III*

Typical loads are summarized and shown below in Table V.

Sr. No.	Loading	Vertical KN	Horizontal X KN-m	Horizontal Y KN-m
1	Dead load	27.5		
2	Operating load	26.2	6.3	
3	Cable load (Dynamic)	3.0	17.0	12.1
4	Wind load (Y-dir)			17.5
	Total	56.7	27.0	12.1

Table 5: Load Case - 3

VI. ANALYSIS RESULTS

The analysis for all the 3 loading conditions is performed with ANSYS 15.0 Workbench. The results are displayed as below. Fig. 3, 4 & 5 shows graphical representation of Ansys model showing Equivalent stress and Total deformation. Table VI gives the summary of all 3 load cases comparison with permissible limit and design factor of safety (FOS).

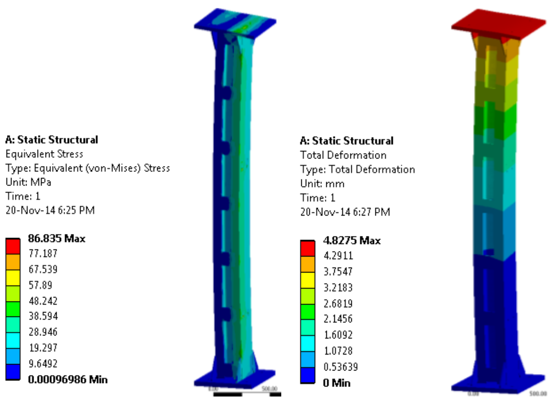

Fig. 3: Load Case – I: ANSYS results for Equivalent stress & Total deformation

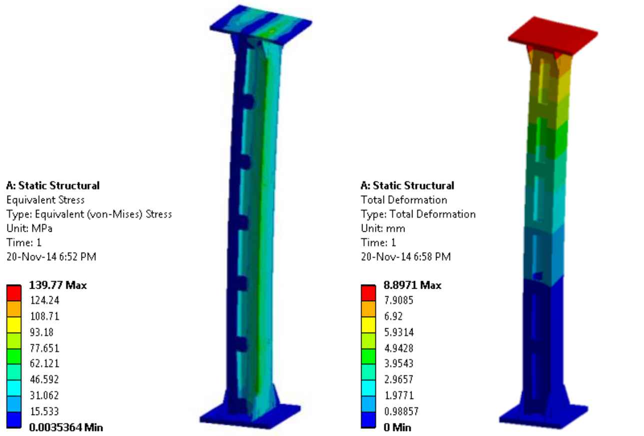

Fig. 4: Load Case – II: ANSYS results for Equivalent stress & Total deformation

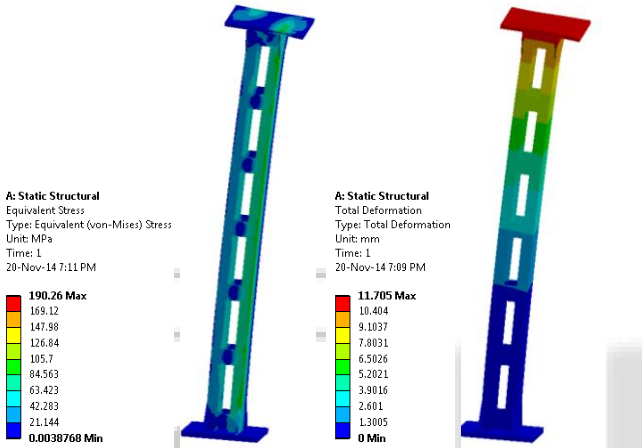

Fig. 5: Load Case – III: ANSYS results for Equivalent stress & Total deformation

	Equivalent (von-Mises) Stress (MPa)	Total Deformation (mm)	Permissible stress (MPa)	FOS
Load Case I	86.8	4.8	250	2.9
Load Case II	139.7	8.9	250	1.8
Load Case III	190.2	11.7	250	1.3

Table 6: Result Summary

VII. CONCLUSION

Support structures in any substation equipment can be designed and analyzed by using FEM simulation. The different loads can be considered as per the actual site conditions. Analysis results for different load cases can be used to evaluate if equivalent stress values are under the permissible stress and deformation is under control. The use of simulation software reduces the time & cost for prototype development and testing.

VIII. ACKNOWLEDGMENT

I would like to take this opportunity to express my profound gratitude and deep regard to my Prof. V.H. Bankar, VIT, Nagpur for his exemplary guidance, valuable feedback and constant encouragement throughout the duration of this research work. His valuable suggestions were of immense help throughout my work. His perceptive criticism kept me working to make this research paper in a much better way. Working under him was an extremely knowledgeable experience for me.

I would also like to express my sincere thanks to management of Siemens Ltd., India for allowing me to publish this paper and providing me the necessary facilities.

REFERENCES

[1] Mahendran Mahen, Application of finite element analysis in structural engineering, International conference on computer aided engineering, Chennai, India, (2007).
[2] Rupesh Daripa, Finite element method in modal and spectrum analysis of the 145kV circuit breaker structure, SWICON, India, (2011).
[3] IEC 62271-100 (2012-09), International standard for High-voltage switchgear and control gear.
[4] IS:875 (1987), Code of practice for design loads (other than earthquake) for building and structures.
[5] IS:800 (2007), General construction in steel – code of practice.