International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395 -0056
Volume: 02 Issue: 04 | July-2015 www.irjet.net p-ISSN: 2395-0072

Analysis of Angle Type Fixed Section Support Structure in Switchgear

Amit Shende[1], Vaibhav Bankar[2]

[1] Student, Department of Mechanical Engineering. , VIT, Nagpur, Maharashtra, India
[2] Professor, Department of Mechanical Engineering. , VIT, Nagpur, Maharashtra, India

---***---

Abstract - ***The requirement of performing the detailed designing and analysis of support structure observed due to failures in the site conditions. In the initial days it had to be checked with manual calculations or laboratory testing. But 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 Straight Angle section support structure of switchgear in consideration of actual loading conditions using ANSYS. The angle section is used because it is low in weight and affordable cost.***

Key Words: *Computer aided designing (CAD), Support structure, Finite element analysis (FEA), Switchgear, Angle Section.*

1. INTRODUCTION

Switchgears are very important and critical part in power transmission and distribution network. It is used to connect the power source with the consumers via various transmission equipments. Being one the most critical equipment the switchgear has a role to serve of carrying, making and braking the current flowing through it in normal as wel as short circuit conditions.

The Support structure of the switchgear is the most basic and very important part. It looks very simple but has play an important role in the equipment. It has to be rigid enough to carry the dead load of the switchgear, keep it at the elevated height and withstand the various loads acting on the equipment such as Wind Load, Operation Load and Cable load in both static as wel as dynamic condition.

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.

1.1 Finite Element Model of Support Structure

Actual Fig. 1 of the switchgear equipment in working condition is shown below. 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 4 ISA angles of size 75x75x8 mm thick sections kept parallel to each other by welding the top and bottom plates of steel. Battens are welded between the parallel angles 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 24211 and total number of elements are 10237.

Fig -1: Switchgear in working condition

Fig -2: Finite Element model of the support structure

The material used for support structure is steel and its properties are mentioned in Table 1.

Table -1: Material Properties

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

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

1.2 Type 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

2. 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 2.

Table -2: Summary of Load Case - 1

SN	**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

B) Load case – 2

Typical loads are summarized & shown below in Table 3.

Table -3: Summary of Load Case - 2

SN	**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)		3.8	
	Total	56.7	27.1	12.1

C) Load case – 3

Typical loads are summarized & shown below in Table 3.

Table -4: Summary of Load Case - 3

SN	**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)			17.5
	Total	56.7	23.3	29.6

3. FEM ANALYSIS

The analysis for all the 3 loading conditions is performed with ANSYS 16.0 Workbench. The results are displayed as below. Fig. 3-11 shows graphical representation of Ansys model showing Equivalent stress and Total deformation. Table 5 gives the summary of all 3 load cases comparison with permissible limit and design factor of safety (FOS).

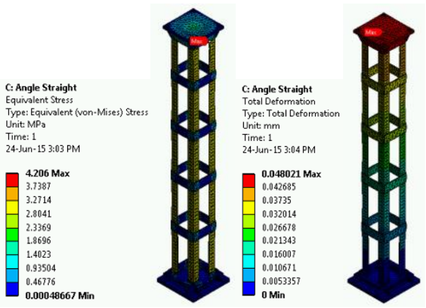

Fig -3: Load case 1- Vertical Load

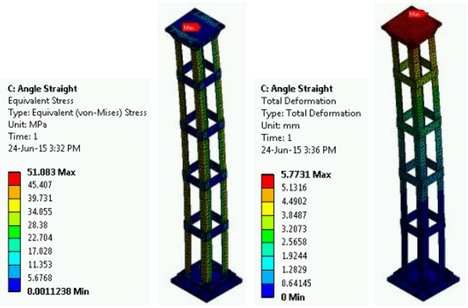

Fig -4: Load case 1- Bending Moment (X)

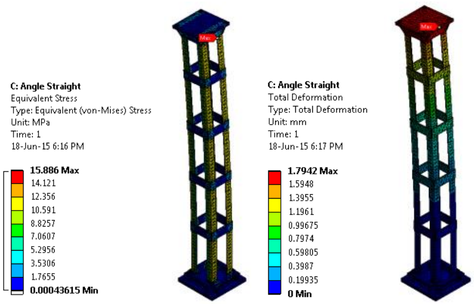

Fig -5: Load case 1- Bending Moment (Y)

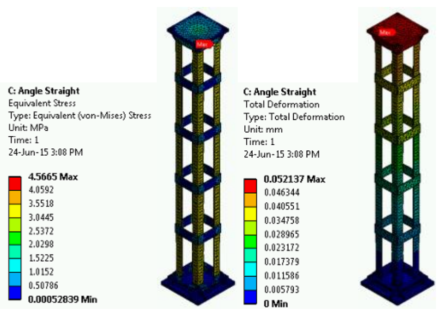

Fig -6: Load case 2- Vertical Load

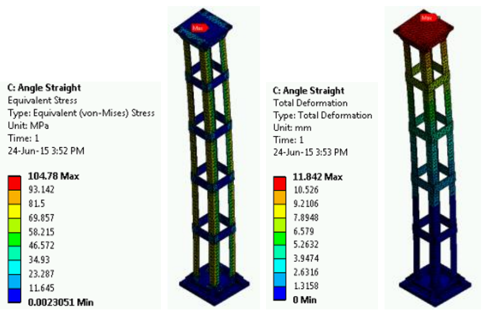

Fig -7: Load case 2- Bending Moment (X)

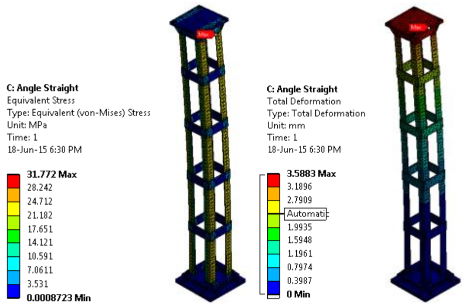

Fig -8: Load case 2- Bending Moment (Y)

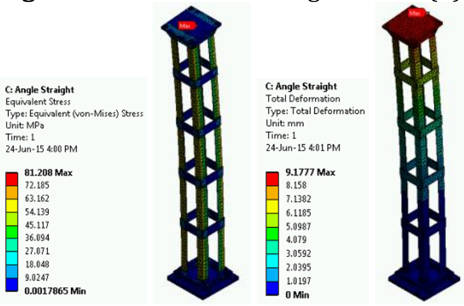

Fig -9: Load case 3- Vertical Load

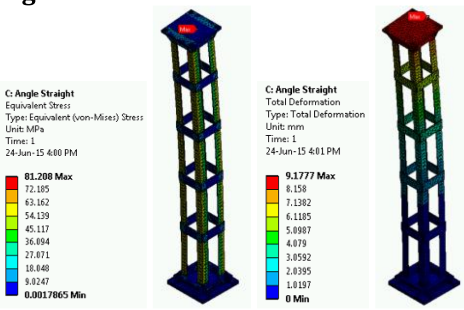

Fig -10: Load case 3- Bending Moment (X)

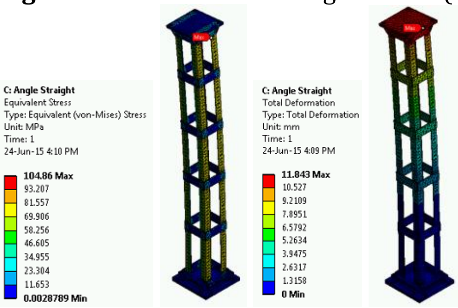

Fig -11: Load case 3- Bending Moment (Y)

Table -5: Summary of Results

SN	Maximum Load condition	Load	Stress (MPa)	Deflection (mm)	Stress Limit (MPa)	FOS
1	Vertical load-Case 2	56.7 kN	4.6	0.05	114.7	25
2	Bending moment(X)-Case2	27.1 kN-m	104.7	11.7	165	1.6
3	Bending moment(Y)-Case2	29.6 kN-m	110	13	165	1.5

The limit stress values are considered as per IS: 800- 2007 [1] and Chap no. 6, page no. 117, Design of steel structure, second edition [2].

4. CONCLUSIONS

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.

Further to this work, the stresses are wel within the limit. The deformation needs to be controlled by further redesigning the support structure.

ACKNOWLEDGEMENT

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] IS:800 (2007), General construction in steel – code of Practise.

[2] L. S. Negi, “Design of steel structures”, Second Edition, Tata McGraw-Hill Publishing Company Ltd., New Delhi, ISBN-10: 0-07046205-2

[3] IS:875 (1987), Code of practice for design loads (other than earthquake) for building and structures.

[4] Mahendran Mahen, Application of finite element analysis in structural engineering, International conference on computer aided engineering, Chennai, India, (2007).

