International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 05 Issue: 07 | July 2018 www.irjet.net p-ISSN: 2395-0072

Development of Methodology for Rectification of Transformer Tank Failure by Finite Element Analysis

Mitali Lohar[1], Sunil Gaikwad[2]

[1]B.Tech Student, Dept.of Mechanical Engineering, Vellore Institute of Technology, Chennai
[2]Dy. Manager-Design and Technology, Power Transformer (T1), CG Power and Industrial solutions Ltd, Mumbai

---***---

Abstract - *Power transformers are used for transmission and distribution of electric power. Transformer tank plays vital role by providing adequate structural strength and shell for CCA (core coil assembly). Design of tank shall be robust enough to withstand full vacuum load and hydrostatic pressure as per IEC 60076- 1 standard. FEA based simplified model is developed to analyze failure of tank for vacuum load (735 mm of Hg) and verified with theoretical calculations as well as actual test results. The resulting parameters in all the cases have compared and found to be in acceptable range of error. Based on this FE methodology, optimum stiffening arrangement is done for rectification of failed tank to improve its structural strength to withstand specified load case.*

Key Words: Transformer tank, Finite element method, Section modulus, Ansys workbench, optimization

1. INTRODUCTION

The Power transformer is one of the most important components in a power transmission system. With an increase in their quality requirements, careful attention has to be paid to their structural stability. Transformer tanks have to be designed robust enough to withstand the static loads (vacuum- 735 mm of Hg) as per the IEC 60076- 1 standard.

The present paper demonstrates failure analysis methodology by numerical method validated with actual test data and FE analysis results. The present model of transformer is 63 MVA 230 kV 3 phase 5 limb type construction. Tank is made up of mild steel material as per standard IS-2062 grade E-250(FE410W) quality B or ASTM – A.36. The material is assumed to be linearly elastic and isotropic. Modeling and assembly of Transformer tank is done by using 3D modeling software NX as shown in Figure 1 and FEA analysis is done by using commercial software based on Finite Element method ANSYS workbench.

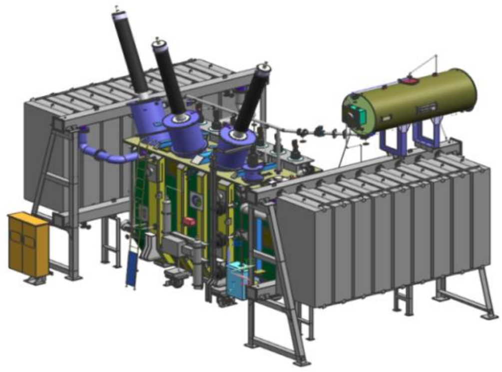

Fig -1: Transformer Assembly

2. VACUUM TEST REPORT AT FABRICATOR END

Vacuum test of Transformer tank is carried out for 735 mm of Hg at fabricator end and results are shown as below

VACUUM GAUGE: VG-01,02,03 Date of testing: 19/05/2018

Specified Vacuum: 735 mm of Hg		Vacuum applied: 735 mm of Hg		Duration : 1 hour	
Location for deflection measurement (see sketches)		Distance of the point from reference line (mm)			Permanent deflection (mm)
Location	Point	Before applying Vacuum	Under Vacuum	After releasing Vacuum	
Tank Wall on HV Side	1A	365	374	367	2
	1B	395	403	397	2
	1C	422	425	422	0
	2A	426	435	427	1
	2B	469	473	470	1
	2C	512	517	512	0
	3A	443	455	444	1
	3B	498	505	498	0
	3C	552	555	552	0
L.H.S (HV)	1A	552	556	552	0
	1B	557	561	557	0
	1C	538	541	538	0
Tank Wall on LV SIDE	1A	509	520	510	1
	1B	492	505	494	2
	1C	464	471	465	1
	2A	512	530	515	3
	2B	507	520	510	3
	2C	476	487	478	2
	3A	503	516	505	2
	3B	500	513	501	1
	3C	476	483	477	1
R.H.S (HV)	1A	265	266	265	0
	1B	307	312	307	0
	1C	1284	1289	1284	0
Tank Cover	A	287	290	287	0
	B	280	283	280	0
	C	290	293	290	0

Fig -2.1: Deflection results

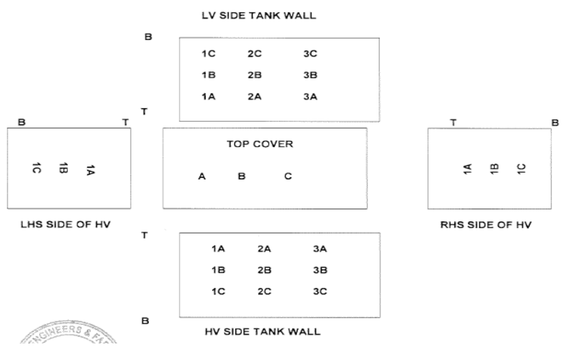

Fig -2.2: Deflection measurement points on tank wall

As seen in results, permanent deflection is subtraction of deflection after releasing vacuum and deflection before applying vacuum.maximum underload deflection observed is Maximum Permanent deflection observed is 3mm on LV side, which is exceeding allowable range as per IEC standard (Greater than 1mm for below 100 MVA Transformer). Hence it can be concluded that tank is failed to withstand vacuum test.

3. FINITE ELEMENT ANALYSIS OF FAILED TANK

FE analysis of failed tank has been carried out for vacuum load (735 mm of Hg) by using Ansys workbench and results are shown as below:

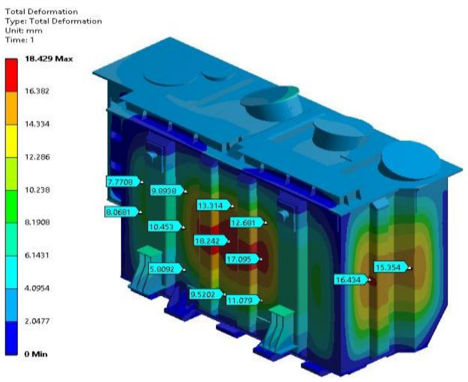

Fig -3.1: Deflection plot on LV Side (mm)

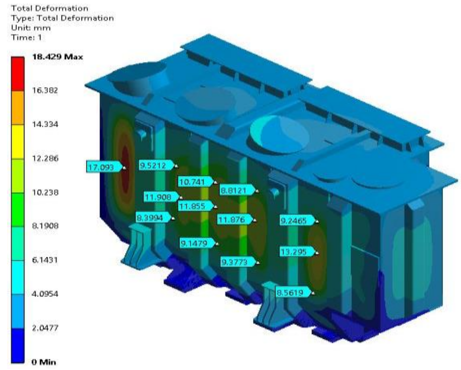

Fig -3.2: Deflection plot on HV Side (mm)

Maximum under load deflection observed is 18.4 mm on LV side and 17 mm on HV side

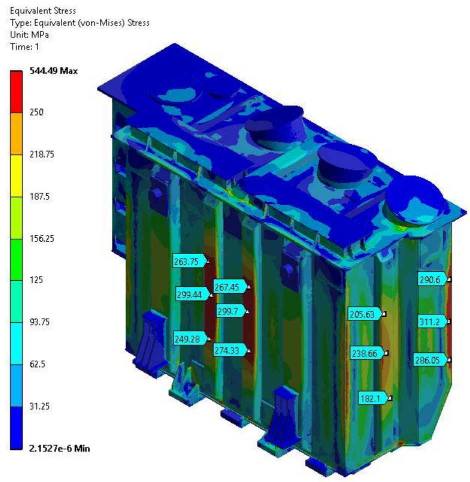

Fig -3.3: Stress plot on LV Side (MPa)

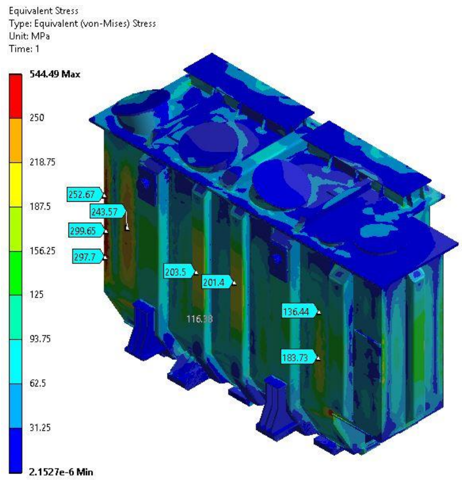

Fig -3.4: Stress plot on HV Side (MPa)

Maximum stress observed on tank wall is 544.5 MPA however this value is localized high stress due to numerical stress singularities in FEM calculations and therefore this stress value can be neglected. Average von misces stress observed on LV side is in range of 280-300 MPA. This stress is higher than yield limit of material mild steel (250 MPA) and hence it reached in plastic zone and will cause permanent deflection of tank wall. Tank wall is unable to regain its original position and will have permanent deformation greater than 1 mm causing failure of tank.

4. NUMERICAL CALCULATION OF FAILED TANK DESIGN

Conceptualized numerical method is developed to calculate deflection of tank wall panel based on section modulus of panel. Section modulus of box stiffener along with tank plate is calculated to withstand the specified load case (735 mm of Hg). Actual section modulus of tank panel and stiffener was less than required section modulus to withstand vacuum load.

4.1 Calculation of Section Modulus required withstanding vacuum load case (735 mm of Hg)

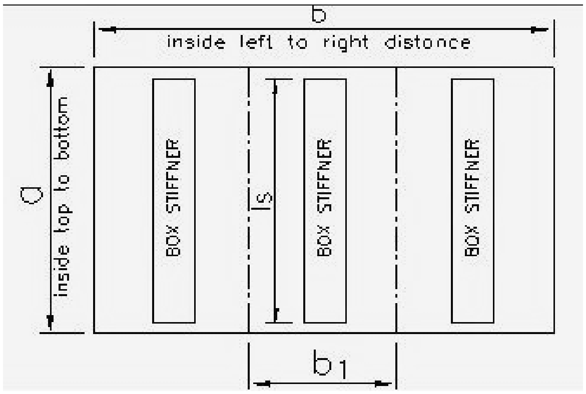

Fig -4.1: Schematic tank wall and panel dimensions

b_1 =Panel width = 75 cm

a =Height of the tank plate = 358.5cm

P_s =(P0+Pt) Pressure in kg/cm^2 or Vacuum

pressure whichever is higher

Section modulus required (Zr)

Z_r = (0.000059 x P_s x a^2 x b_1) cm^3

= 574.4 cm^3 -------------------------- Eq.1

4.2 Section Modulus of Panel (Box stiffener and tank wall)

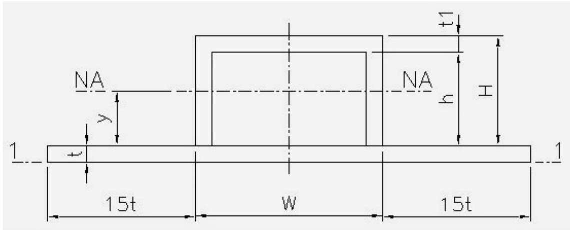

Fig -4.2: Section modulus of tank wall and box stiffener panel

Table -1: Tank wall and stiffener dimensions

	DESCRIPTION	VALUE	UNIT
W =	Width of box stiffener	25	cm
H =	Height of box stiffener	15	cm
h =	Inside height of box stiffener	14	cm
t =	Thickness of tank plate	0.8	cm
t_1 =	Thickness of box stiffener	1	cm
I_{1-1} =	Moment of Inertia about 1-1	7932.08	cm^3
Y =	Neutral axis	6.69	cm
I_{NA} =	Moment of Inertia about NA	3808.75	cm^3
Z =	Section modulus	417.	cm^3

$$Y = \frac{30t^3 + wt^2 + 2ht1(2t+h) + wt1(2t+2h+t1)}{2(30t^2 + wt + 2ht1 + wt1)}$$

$$Y = \frac{1233.16}{184.4}$$

$$Y = 6.69 \quad cm$$

$$I_{1-1} = \frac{(30t+w)t^3}{12} - \frac{t\,X\,h^3}{6} + \frac{w\,X\,t1^3}{12} - \frac{(30t+w)t^3}{4} + \frac{h\,X\,t1(2t+h)^2}{2} - \frac{t1\,X\,w(2t+2h+t1)^2}{4}$$

$$I_{1-1} = \frac{25.088}{12} - \frac{2195.2}{6} + \frac{25}{12} - \frac{25.088}{4} + \frac{3407}{2} - \frac{5852.3}{4}$$

$$I_{1-1} = 7932.08 \text{ cm}^4$$

$$I_{NA} = I_{1-1} - [(30t + w)\,t + 2\,h\,t1 + t1\,w] \,X\, Y^2$$

I_{NA} = 7932.08 – [39.2 + 28 + 25] X 44.722

I_{NA} = 7932.08 – 4123.32

I_{NA} = 3808.75 cm^4

$$Z = \frac{I\ NA}{(t+h+t1-Y)} \;\ldots\ldots \text{ if } Y \leq \frac{(t+t1+h)}{2}$$

$$Z = \frac{3808.75}{9.11} \;\ldots\ldots\ldots\ldots \text{ if } Y \;\leq 7.9$$

Z = 417.96 cm^3----------------------- Eq.2

Section modulus required to withstand vacuum load as per Eq. 1 is less than section modulus calculated based on failed design as per Eq.2

4.3 Calculation of panel deflection

E =Young modulus=2.1 X 10^6 Kg/cm^2

I_s =Moment of Inertia about Neutral Axis

d_{def} = Deflection of panel

$$d_{def} = \frac{Ps\ X\ b1\ X\ a^4}{E\ X\ Is} \text{ X } \frac{5}{384}$$

$$d_{def} = \frac{5}{384} \text{ X } \frac{1.25124E+12}{7998383552}$$

d_{def} = 0.013 X 156.43

d_{def} = 2.03 cm

- Under load deflection of panel as per actual measurement at vendor end is 18 mm.
- Under load deflection of panel as per FE Analysis is 18.4 mm
- Under load deflection of panel as per calculation is 20.3 mm.

All three results are found in good agreement with each other (within 10%). FE analysis results are more closure to actual test results and hence decided to go ahead with FE analysis for rectification of failed design. Permutation and combination of variety of stiffeners (T, Flat, C Channel, I beam) have been done to rectify final design to withstand specified vacuum load. Optimized stiffening arrangement is done to avoid failure of tank and substantial cost occurred. Rectification is done and highlighted as red circles shown in Fig 5.1 and Fig-5.2

5. FINITE ELEMENT ANALYSIS OF RECTIFIED TANK DESIGN

FE analysis of rectified tank has been carried out and simulation results are shown as below. Rectification is done for addition of flat stiffeners to minimize deflection and stress observed. Changes in design are highlighted as red circles in below simulation plots.

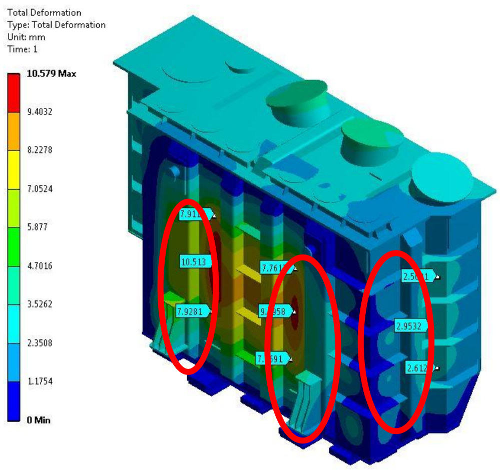

Fig -5.1: Deflection at LV side (mm)

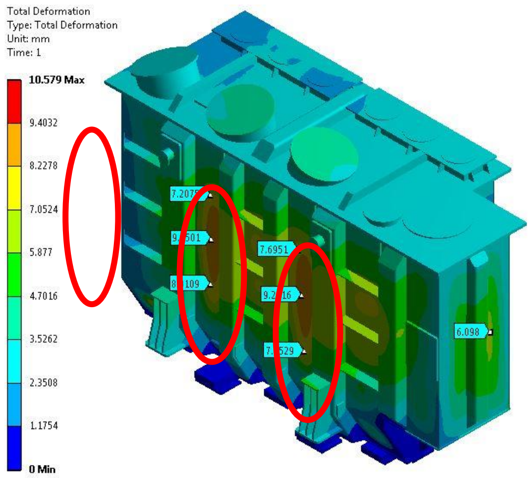

Fig -5.2: Deflection at HV side (mm)

Maximum under load deflection observed is 10.5 mm on LV side and 9.2 mm on HV side

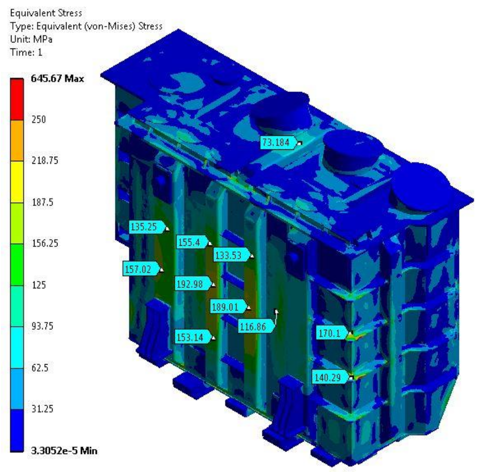

Fig -5.3: Stress at LV side (MPa)

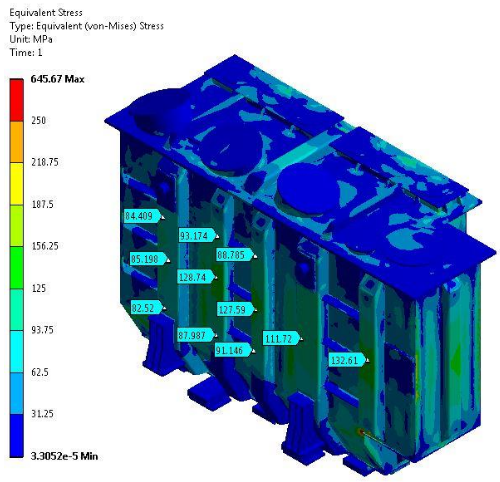

Fig -5.4: Stress at HV side (MPa)

Maximum stress observed on tank wall is 645.67 MPA however this value is localized high stress due to numerical stress singularities in FEM calculations and therefore this stress value can be neglected. Average von misces stress observed on LV side is in range of 180-200 MPA . This stress is well below yield limit of material mild steel (250 MPA) means it is in elastic zone and will deflection 10.5 mm will regain its original position after releasing vacuum load and shall not have any permanent deflection. This tank is safe to withstand vacuum load (735 mm of Hg)

6. COMPARISON OF RESULTS

Equivalent deflection and stress values of failed design and rectified design compared herewith in tabular format shown below.

Table -2: Comparison Table

	Failed Design			**Rectified Design**
	Actual results	FEA Results	Numerical results	FEA Results
Max. Def. (mm)	18	18.4	20.3	10.5

7. CONCLUSIONS

The present methodology defines failure analysis of transformer tank by FE analysis. The corresponding results between numerical methodology and FE analysis are in good agreement with each other (within 10% error). Optimized design of stiffener arrangement for failed tank is provided based on Finite Element Method. This conceptualized design methodology is competent and reliable for error free stiffener design.

ACKNOWLEDGEMENT

I would like to express my heartfelt gratitude to my project guide **Sunil Gaikwad**, Deputy Manager, Design Department Power Transformer (T1) CG Power and Industrial Solutions Ltd. For his committed guidance, valuable suggestions, and sustained help from the very early stage of this project work.

REFERENCES

[1] IEC 60076-1 Clause- 11.9

[2] Strength of materials vol.2 – advanced theory, Stephen Timoshenko

[3] Design, Operation & Maintenance of Core Type Oil Filled Power Transformers-Chiplunkar

[4] ASTM A-36

[5] IS-2062-2011 : TABLE-2

[6] Stresses in plates and shells, Ansel C Ugural

[7] CBIP : Manual on transformer , Section – A

[8] Ansys Help, Release 14.5 Documentation for ansys.

Author

MITALI LOHAR

B.Tech Student, Dept.of Mechanical Engineering, Vellore Institute of Technology, Chennai