2-D FINITE ELEMENT ANALYSIS AND OPTIMIZATION OF TEMPORARY STEEL STRUCTURE COVERING LARGE SPAN

[1]Chirag Javiya, [2]Satyen Ramani
[1]Post Graduate Student, [2]Associate Professor
Department of Civil Engineering
SAL Institute of Technology and Engineering Research, Ahmedabad

***Abstract*— Temporary steel structures are commonly used at various locations, with advantages on repeat construction, simple structure and light weight. For temporary steel structure there are not enough researches on the Finite element analysis and optimization. Finite element analysis of temporary steel structure by ANSYS® Workbench was proposed in this paper. Using this method various models of temporary steel structures for 25m span length were analyzed. The calculation results of objective and constrain conditions were obtained and discussed.**

***Index Terms*— FEM, Temporary steel structure, optimization, ANSYS® Workbench**

I. INTRODUCTION

Temporary structures defined here are those with short service lives. As their name suggests, they are subjected for a temporary function. They are usually made from lightweight components and are used for a wide variety of functions at private and public events. Temporary structures such as exhibitions, musical concerts, social occasions, scaffolds, shelters, tents, sporting events and facilities used during the reconstruction or repair of buildings and bridges, etc., are usually constructed for a limited-time use. Although the design of such structures to live and dead loads usually does not impose any particular challenge, their design for wind load requires more careful investigation. This is due to the fact that service life of a temporary structure is much shorter than a "permanent structure," and as such, the probability of load exposure to the temporary structure is substantially less.

The advantages of temporary steel structure is economical, easy dismantle and can use for repeat construction. The temporary structure can repeat construct and demolition with compare to the traditional temporary building. Recently, many reports can be read about temporary structure accidents, caused by wind.

Optimization is the act of obtaining the best result under given circumstances. In design, construction, and maintenance of any engineering system, engineers have to take many technological and managerial decisions at several stages. The ultimate goal of such decisions is either to minimize the effort required or to maximize the desired benefit. Optimization can be defines as the process of finding the conditions that give the maximum or minimum value of a function.

In structural optimization, design objectives are structural criteria used to evaluate the merit of a design such as minimum construction cost, minimum life-cycle cost, minimum weight, and maximum stiffness. IS (Indian standards) code provision, which provides safety and serviceability requirements to the structure, usually appear as the design constraints.

Finite Element Analysis is a good technique for the structures where the direct analysis is not possible.

ANSYS® Workbench is handy software for the FEM study. In ANSYS® Workbench project section various tools as Geometry, Static Structure, Modal, Parameter, Design of experiments, Response surface and optimizations etc. are available. In this paper the FEM study using ANSYS® Workbench was done for Temporary steel structure for various configuration and the comparison were carried out.

II. MODELLING AND LOADING

For the accurate analysis, the general purpose Finite Element Software ANSYS® Workbench is used. The Highlights and details for modeling and Load application are explained here.

LOADING:

DEAD LOADS

Dead loads are calculated by the IS-875(part- I)
Weight of roofing material = 0.131 KN/m^2 (clause-2.1, Table-1)
Weight of purlin = 0.125 KN/m
Self weight of the structure

LIVE LOADS

Live loads are calculated by the IS-875(part- II)
Live load = 0.75 KN/m^2 (clause-4.1, Table-2)

WIND LOADS

Wind loads are calculated by the IS-875(Part-III) by profile co-efficient method (clause-6.2, Table-4 & Table-5)
Location - Ahmedabad
Risk coefficients factor (k_1) – 0.92 m/s for temporary structure with design life of five year (Table-1)

MODELING:

ANSYS® Workbench provides very efficient interaction flowchart between several CAD design modules including geometry import from many supporting CAD design software with meshing import facility. Models for temporary steel structure have been developed with different span length, typical panel length and angle of truss bracing.

ANSYS® Workbench project schematic shown below can be developed for the analysis. General steps are as follows:
Step-1: Preparation of Geometry
Step-2: Meshing
Step-3: Boundary Conditions and Load Application
Step-4: Solutions and Results
Step-5: Parameter set
Step-6: Response surface optimization

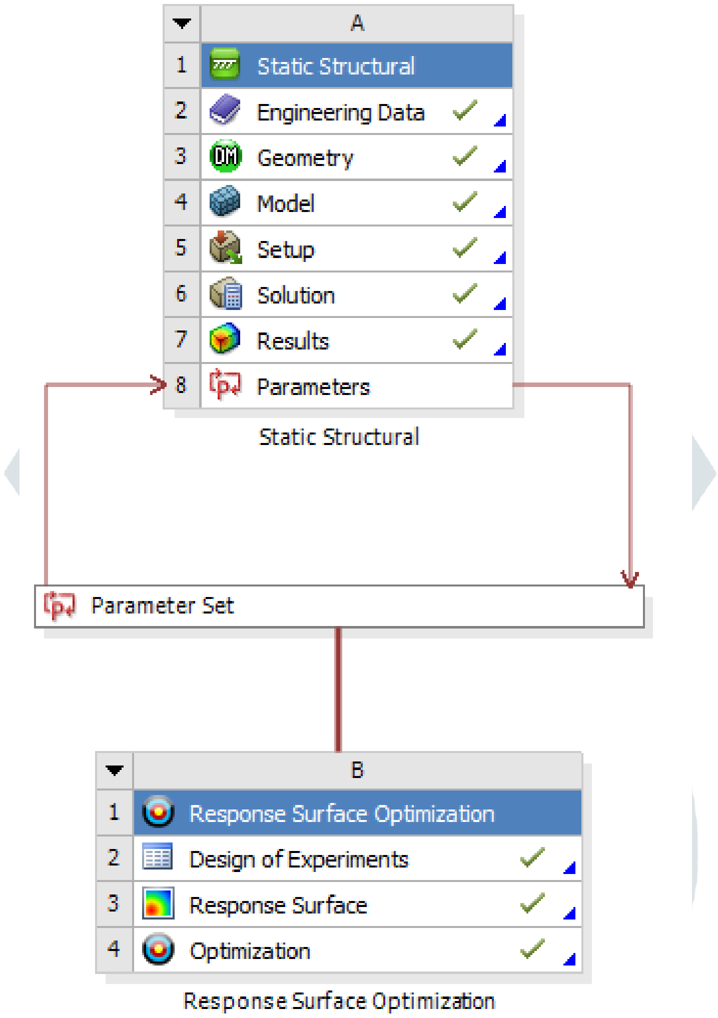


Fig.1: - Schematic chart of work

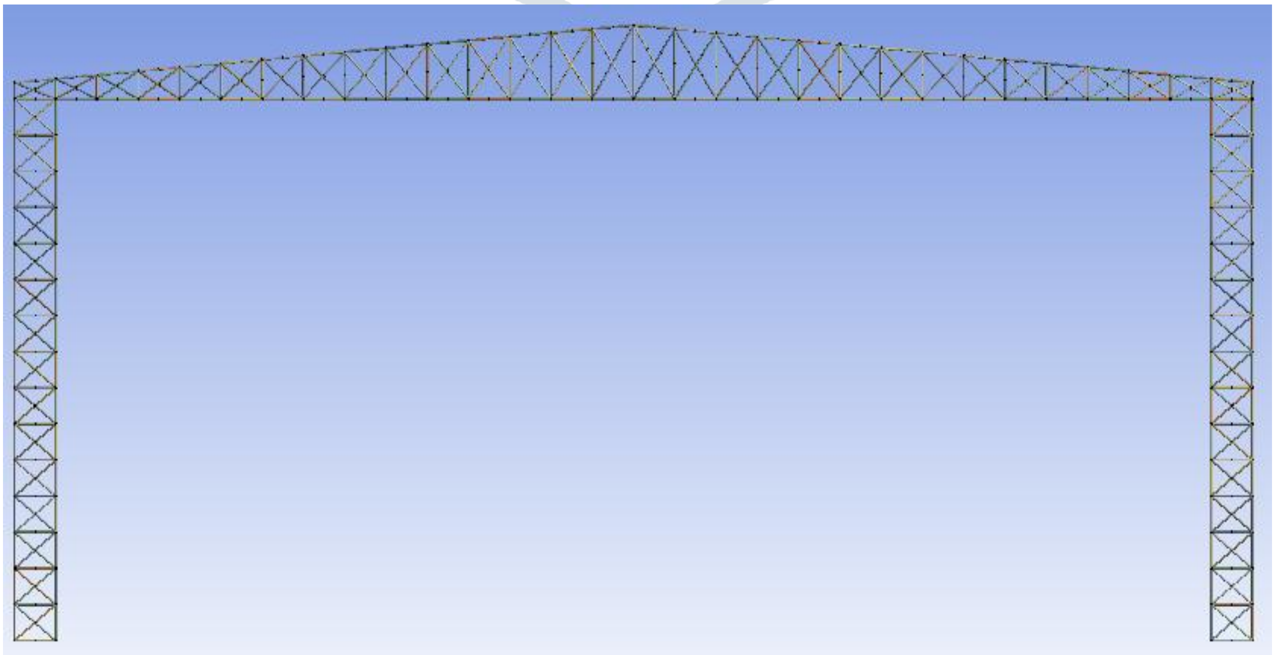


Fig.2: - Typical meshing of Structure

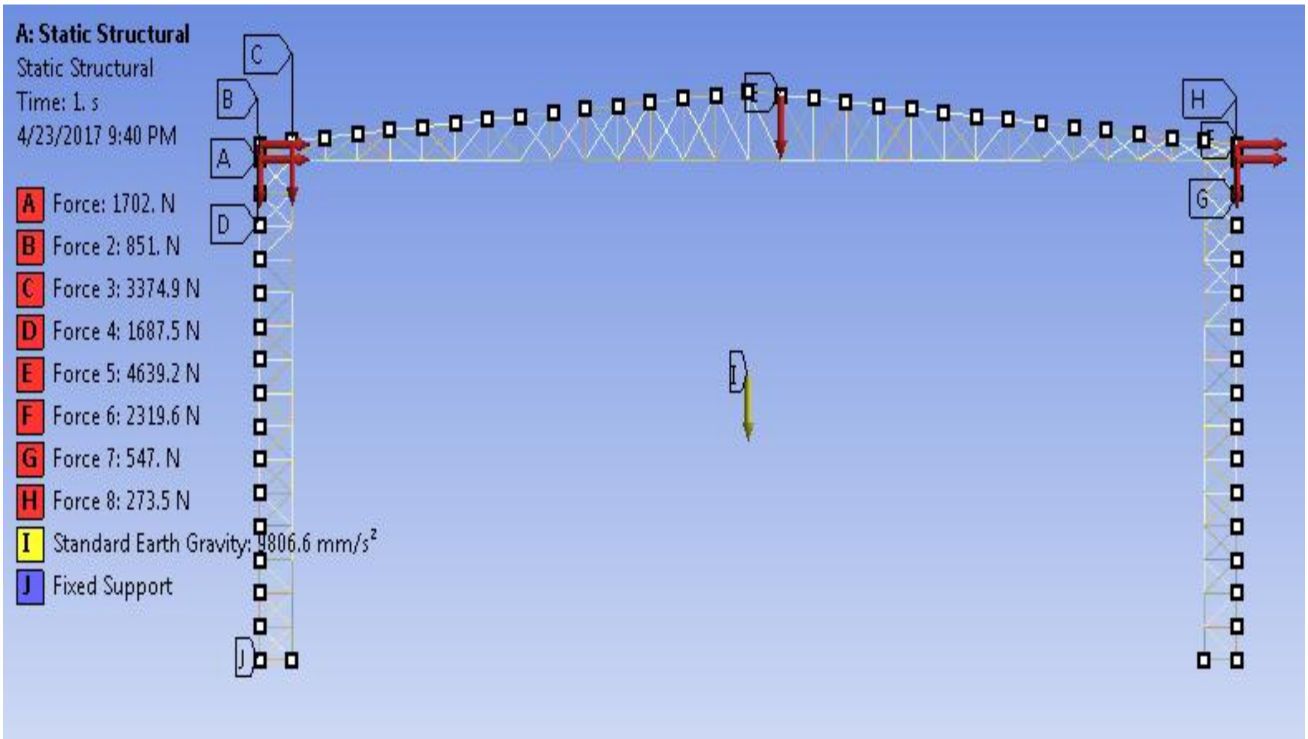


Fig.3: - Support condition and Loads

Basically, we can create required geometry in design modeler with more than one alterative, but it consume more time so to overcome this difficulty there is a option of 'run JAVA script file (JS)', by using this method we can develop the whole geometry within a single click and the properties are given by grouping of members. This generally affects our output either in terms of output quantity values or in terms of required computational time.

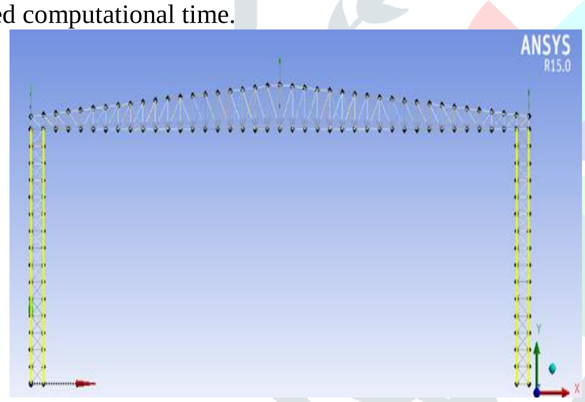


Fig 4: - column main member

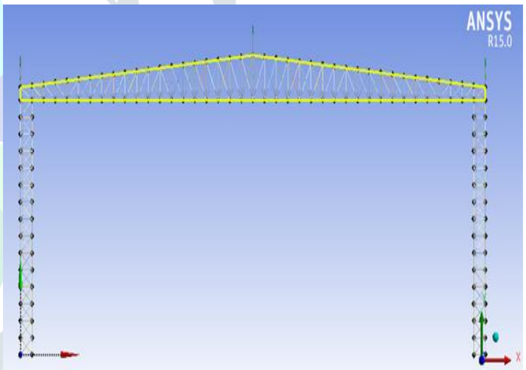


Fig 5: - Truss main member

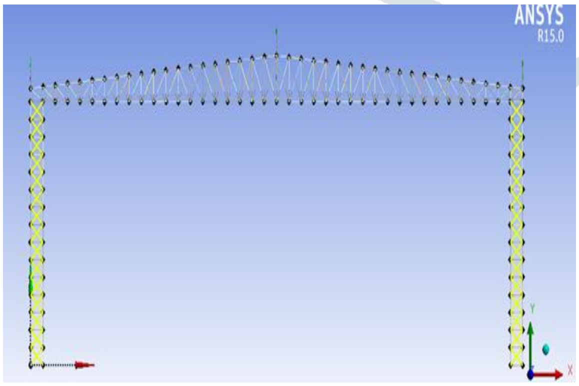


Fig 6: - Column bracing member

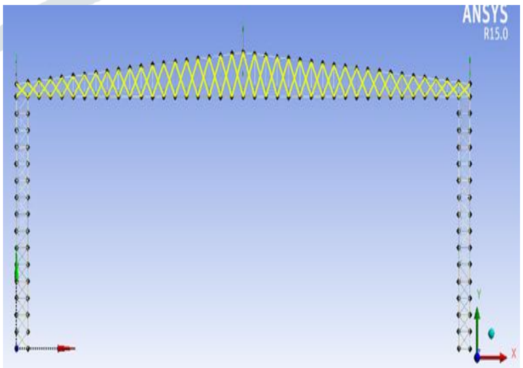


Fig 7: - Truss bracing member

PROBLEM DATA:

As described, the goal of the study is to optimize the weight of the structure on various structural analysis output quantities such as displacements and Normal stress. Where the criteria of the Displacements and Normal stress are as per IS: 800(2007).

Vertical displacement – span/180 (Table – 6, clause no. 5.6.1)
Lateral displacement – Height/150 (Table – 6, clause no. 5.6.1)
Normal stress – As per IS 800(2007)

We have considered here 6 models for 25m Length of spans (outer to outer of the column) for the analysis as per following section properties and structure configuration:

Table1: - configuration of the models

Span(L)	25(m)					
Typical panel length	L/30			L/35		
Angle of truss bracing	20′	30′	40′	20′	30′	40′

Table 2: - cross – sectional properties of the circular hollow section (CHS)

NAME	OD(mm)	T_w (mm)	Ax(cm^2)	I(cm^4)	Z(cm^3)	C(cm^3)	r_y(cm)
21.3x2 CHS	21.3	2	1.21	0.57	0.75	0.11	0.231346335
21.3x2.6 CHS	21.3	2.6	1.53	0.68	0.92	0.13	0.252685111
21.3x3.2 CHS	21.3	3.2	1.82	0.77	1.06	0.14	0.268887466
26.9x2.3 CHS	26.9	2.3	1.78	1.36	1.4	0.2	0.317986229
26.9x2.6 CHS	26.9	2.6	1.98	1.48	1.54	0.22	0.331718517
26.9x3.2 CHS	26.9	3.2	2.38	1.7	1.81	0.25	0.355519412
33.7x2.6 CHS	33.7	2.6	2.54	3.09	2.52	0.37	0.428232168
33.7x3.2 CHS	33.7	3.2	3.07	3.6	2.99	0.43	0.462222729
33.7x4 CHS	33.7	4	3.73	4.19	3.55	0.5	0.498662901
42.4x2.6 CHS	42.4	2.6	3.25	6.46	4.12	0.61	0.552011758
42.4x3.2 CHS	42.4	3.2	3.94	7.62	4.93	0.72	0.599528116
42.4x4 CHS	42.4	4	4.83	8.99	5.92	0.85	0.651196287
48.3x2.9 CHS	48.3	2.9	4.14	10.7	5.99	0.89	0.665630665
48.3x3.2 CHS	48.3	3.2	4.53	11.59	6.52	0.96	0.692760553
48.3x4 CHS	48.3	4	5.57	13.77	7.87	1.14	0.755106837

The ANSYS® WORKBENCH project schematic is prepared to carry out interconnected several type of analysis simulation related to static structural analysis and modal analysis. Fig.2 and Fig.3, represents the typical meshing and boundary conditions applied for each of modal study.

III. RESULTS AND COMPARISION

Several output predefined quantities are readily available in ANSYS results tabs within each simulating analysis type. The main focus was the Total Deformation, Directional Deformation, Axial forces and Normal Stress. Here the normal stress can be calculated by the equation of ($\sigma = P/A$) in the parameter set. In initial phase one model was solved for different load combination and the governing case is taken further for all models. The various graph plotted for the above solutions and the results of different load combinations are shown below:

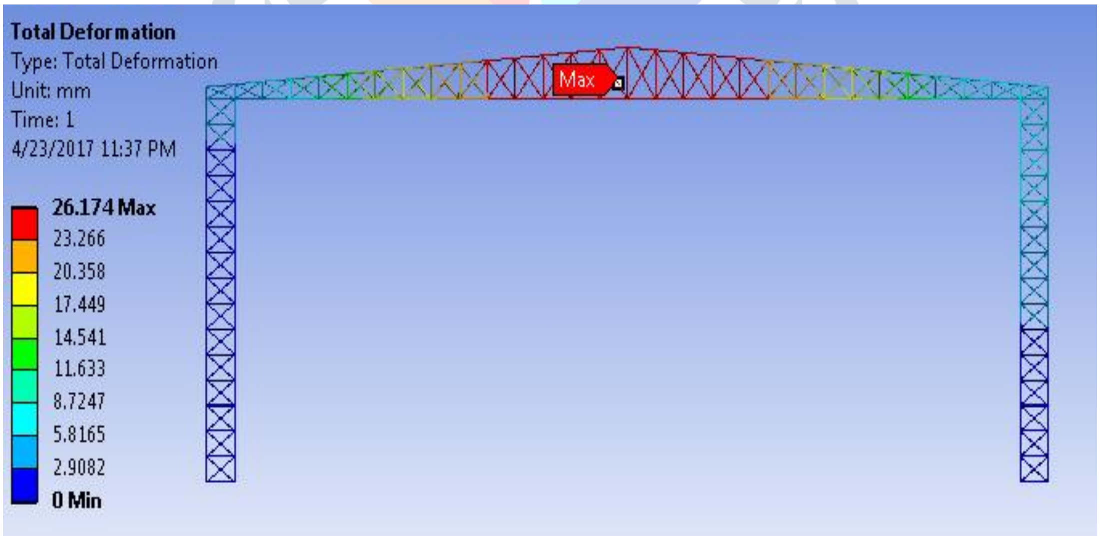
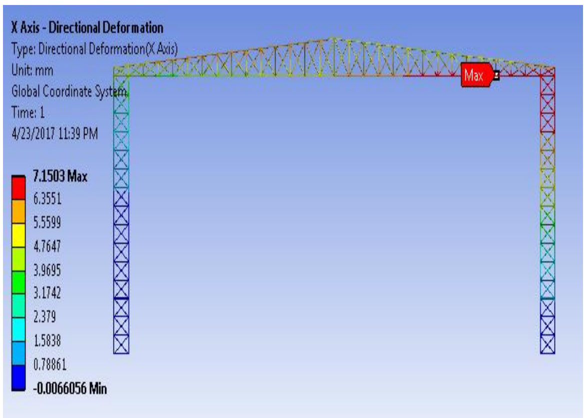


Fig.8: - Total Deformation



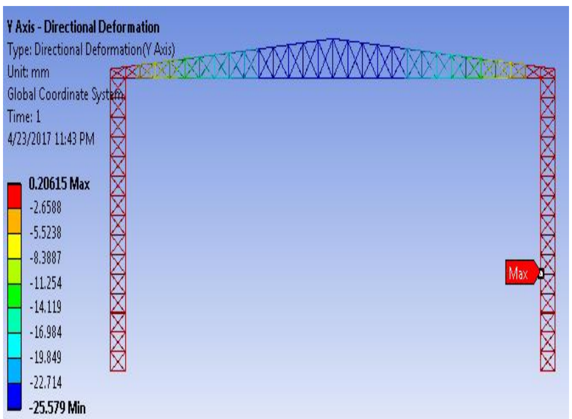


Fig.9: - X-axis Directional Deformation Fig.10: - Y-axis Directional Deformation

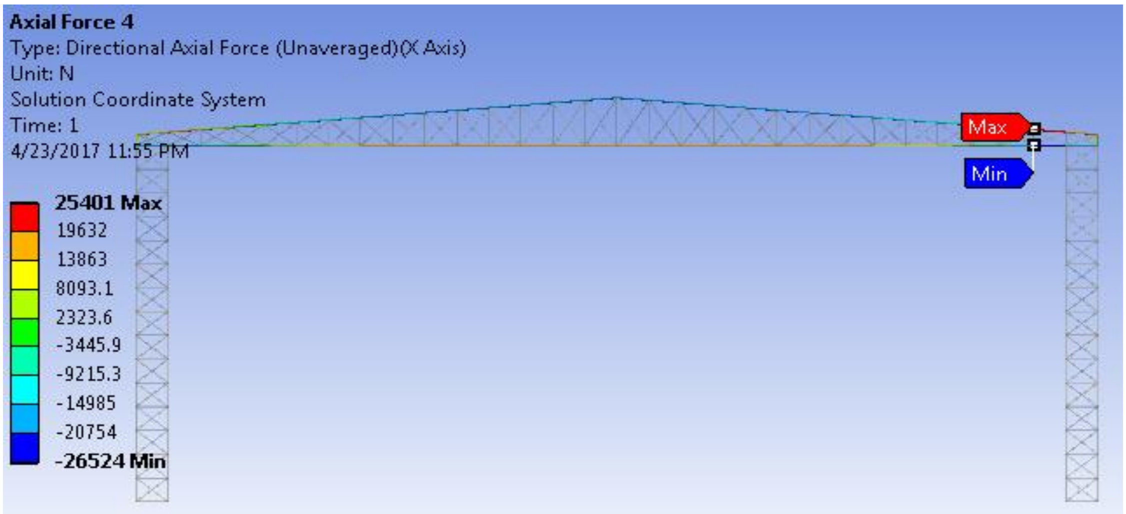


Fig.11: - Axial forces

➤ **The results obtained are as follows:**

Table 3: - results

Load combination	DL+LL	DL+LL+WL1	DL+LL+WL2	DL+WL1	DL+WL2
Total def. max(mm)	25.21	6.77	26.17	10.37	11.73
Total def. min(mm)	-25.21	0	0	0	0
X - Dir. min(mm)	-2.12	-4.07	0	-4.28	0
X - Dir. max(mm)	-2.00	0	7.15	0.013	3.7
Y - Dir. max(mm)	0.02	0.15	0.21	9.67	0.08
Y - Dir. min(mm)	-25.2	-5.6	-25.57	-0.16	-11.32
Axial force max(N)	16528	4724	25401	8048	8692
Axial force min(N)	-17945	-5817	-26524	-7462	-9957
Axial stress max(N/mm^2)	40	11.41	61.35	19.44	21
Axial stress min(N/mm^2)	-43.35	-14.15	-64.07	-18.03	-24.05

Design of experiments: For particular size of the tube, it is necessary to input the dimension of the tube manually by using 'CUSTOM' method in Design of experiments by giving them upper bound and lower bound limits. 'Design Points vs. Parameter' graph shows the variation of the results.

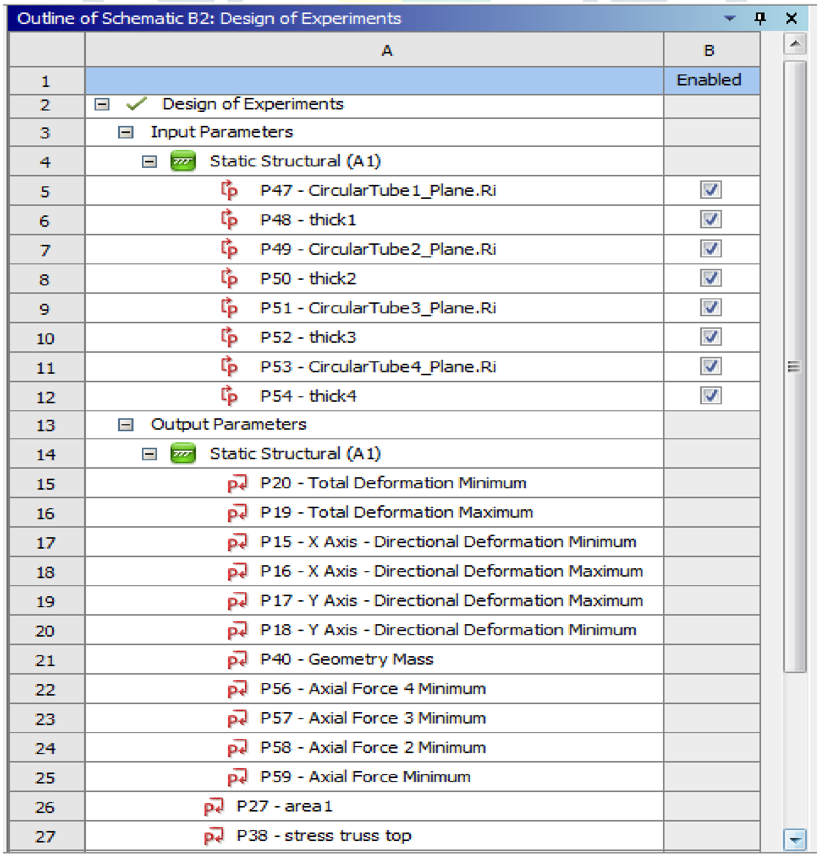


Fig 12: - Design of experiments

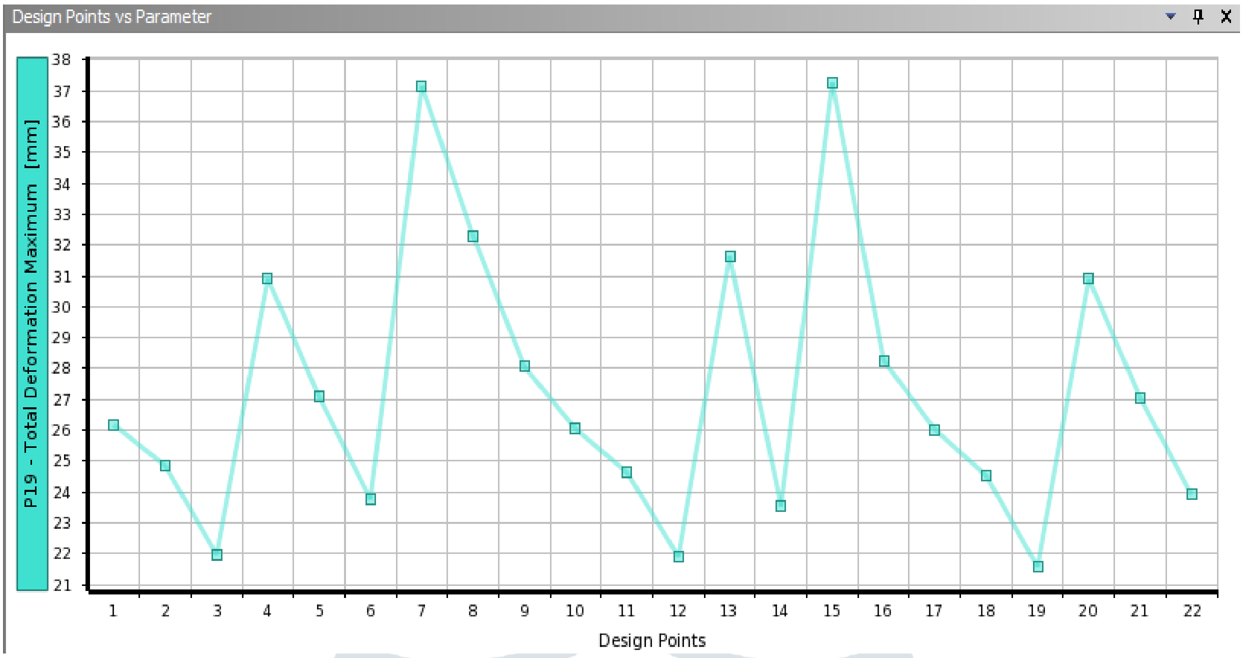


Fig 13: - Design points vs. parameters

Response surface: By using ‘KRIGING’ method in response surface, the 3-D schematic chart between various input and output parameter can be obtained. The ‘Goodness of fit’ represents accuracy of the results. ‘Local sensitivity’ represents the relation between input and output parameter in terms of factor.

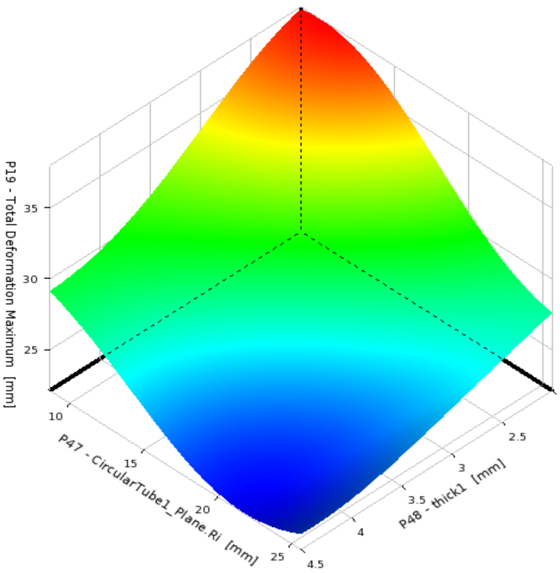


Fig 14: - Design of experiments

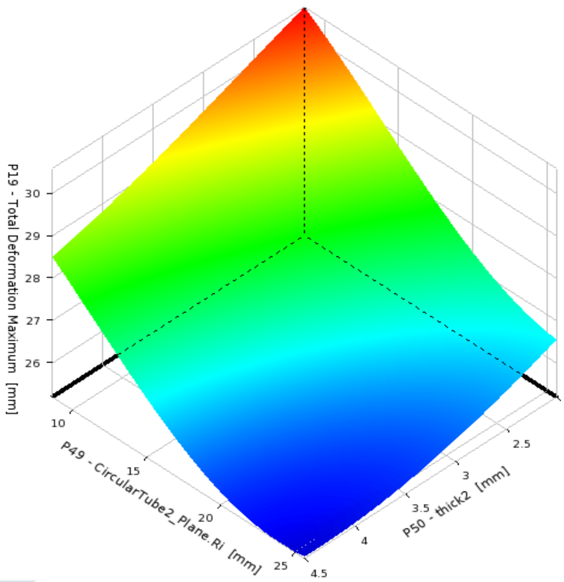


Fig 15: - Design of experiments

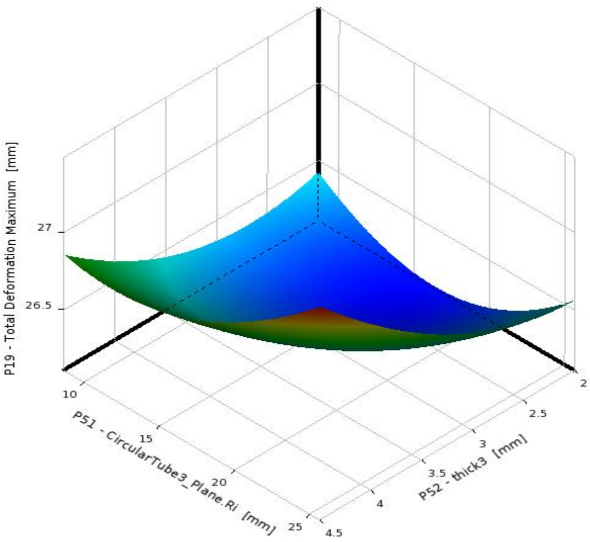


Fig 16: - Design of experiments

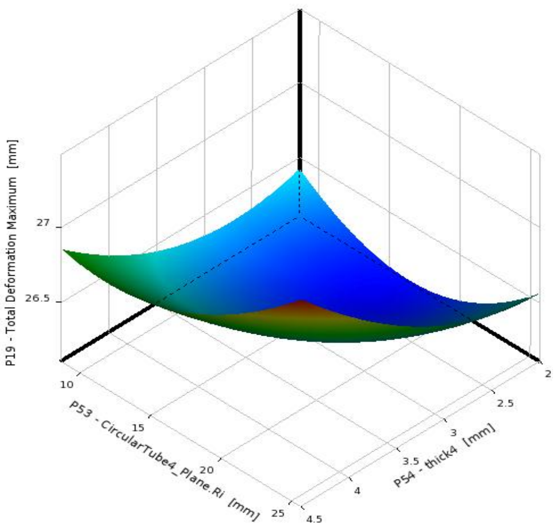


Fig 17: - Design of experiments

Table-4

3-D response surface graphs			
	Horizontal – 1 axis	Horizontal – 2 axis	Vertical – 3 axis
Fig14	Circular tube 1	Thickness 1	Total deformation 1
Fig15	Circular tube 2	Thickness 2	Total deformation 2
Fig16	Circular tube 3	Thickness 3	Total deformation 3
Fig17	Circular tube 4	Thickness 4	Total deformation 4

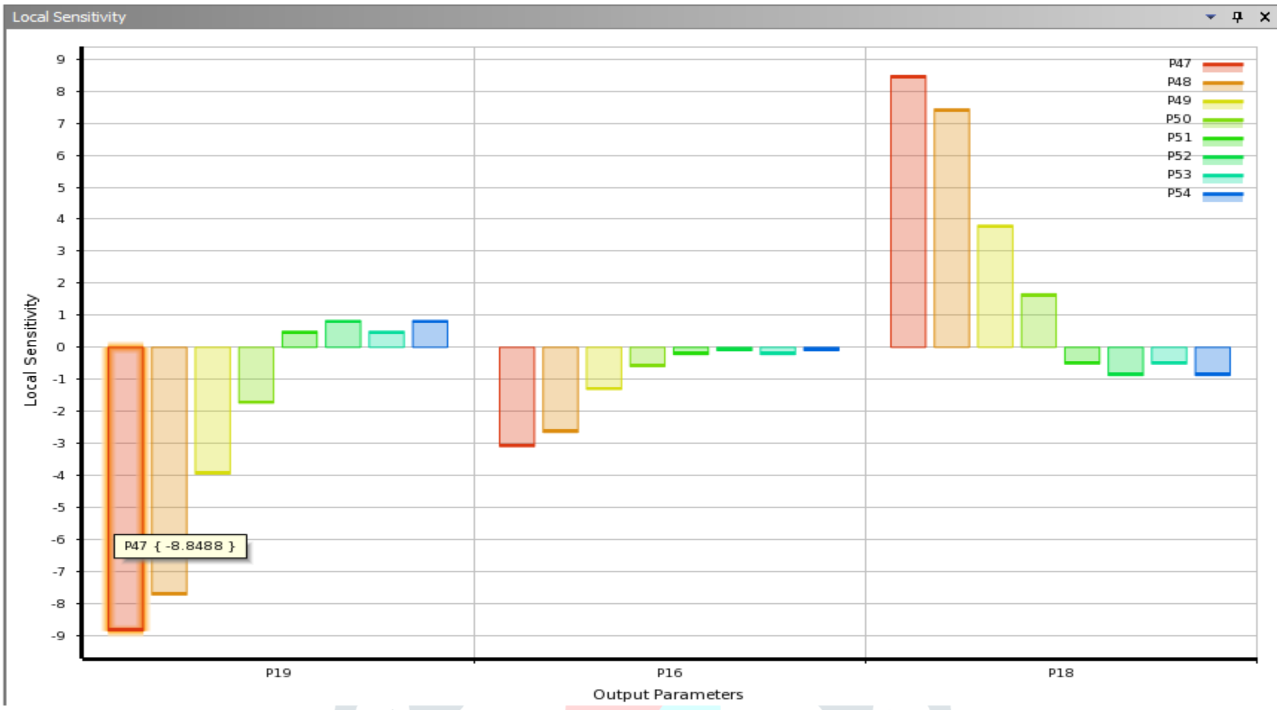


Fig 18: - Local sensitivity

Table of Outline A34: Goodness Of Fit

	A	B	C	D	E	F	G	H	I	J	K	L
1	Name	P20 - Total Deformation Minimum	P19 - Total Deformation Maximum	P15 - X Axis - Directional Deformation Minimum	P16 - X Axis - Directional Deformation Maximum	P17 - Y Axis - Directional Deformation Maximum	P18 - Y Axis - Directional Deformation Minimum	P40 - Geometry Mass	P56 - Axial Force 4 Minimum	P57 - Axial Force 3 Minimum	P58 - Axial Force 2 Minimum	P59 - Axial Force Minimum
2	Goodness Of Fit											
3	Coefficient of Determination (Best Value = 1)	1	1	1	1	1	1	1	1	1	1	1
4	Maximum Relative Residual (Best Value = 0%)	0	0	0	0	0	0	0	0	0	0	0
5	Root Mean Square Error (Best Value = 0)	0	1.786E-06	1.5456E-09	6.6669E-07	2.5535E-08	1.7281E-06	5.358E-07	0.0079968	0.0026938	0.0064428	0.0071655
6	Relative Root Mean Square Error (Best Value = 0%)	0	0	0	0	0	0	0	0	0	0	0
7	Relative Maximum Absolute Error (Best Value = 0%)	0	0	0	0	0	0	0	0	0	0	0
8	Relative Average Absolute Error (Best Value = 0%)	0	0	0	0	0	0	0	0	0	0	0
9	Goodness Of Fit for Verification Points											
10	Maximum Relative Residual (Best Value = 0%)	0	0	0	0	0	0	0	0	0	0	0
11	Root Mean Square Error (Best Value = 0)	0	1.7717E-06	1.4829E-09	6.4584E-07	2.5704E-08	1.7211E-06	5.5095E-07	0.009386	0.002905	0.0074886	0.0079628
12	Relative Root Mean Square Error (Best Value = 0%)	0	0	0	0	0	0	0	0	0	0	0
13	Relative Maximum Absolute Error (Best Value = 0%)	0	0	0	0	0	0	0	0	0	0	0
14	Relative Average Absolute Error (Best Value = 0%)	0	0	0	0	0	0	0	0	0	0	0

Fig 19: - Goodness of fit

Optimization: By using **Multi Objective Genetic Algorithm(MOGA)** method of optimization, the design points updated from the response surface should be optimized by selecting the objective function as weight, that is to be minimized and the constrains conditions such as Total deformation, Directional deformation and Normal stress by applying the limits. The graph clearly shows the optimized variation of the weight of the structure. The results of six models as per the Table: - 1 is shown below.

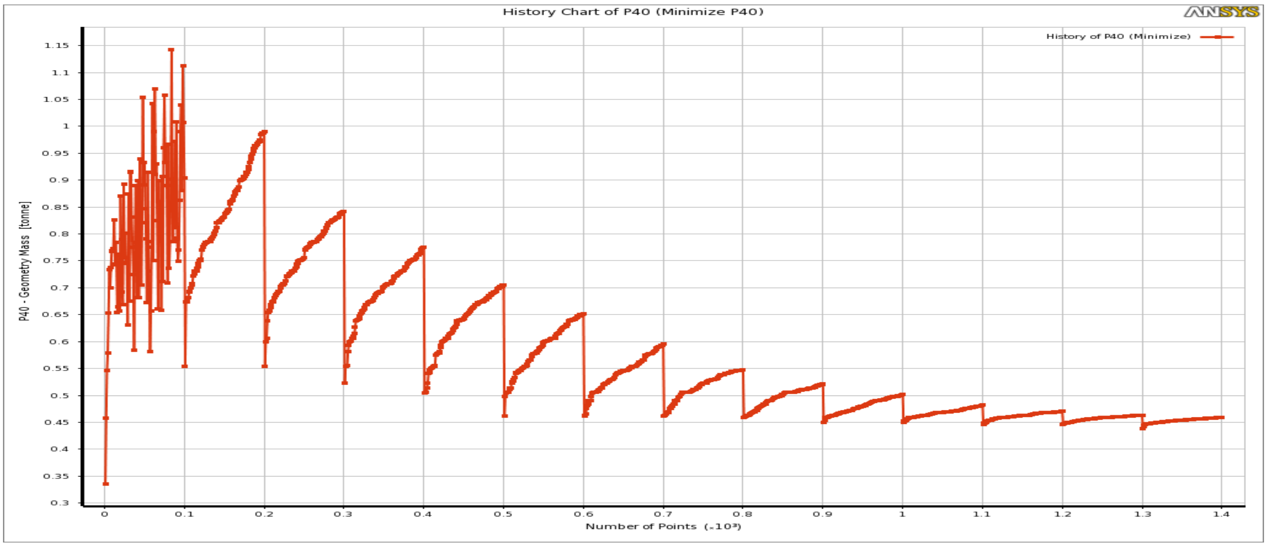


Fig 20: - Design of experiments

Table 5: - optimized results

Panel length	Bracing angle	Tube1 (Ri)	Tube2 (Ri)	Tube3 (Ri)	Tube4 (Ri)	T1	T2	T3	T4	Area1	Area2	Area3	Area4
		mm	mm	mm	mm	mm	mm	mm	mm	mm^2	mm^2	mm^2	mm^2
L/30	20'	18.3	8.9	10.7	8.8	2.3	2.0	3.2	2.0	280	127	247	124
	30'	11.6	9.2	9.6	9.2	2.6	2.0	2.3	2.0	208	129	155	129
	40'	11.4	8.9	9.6	8.9	2.5	2.1	2.2	2.0	199	129	150	127
L/35	20'	14.8	9.2	9.2	8.8	2.0	2.2	2.2	2.2	203	146	140	134
	30'	8.9	8.9	9.1	9.3	2.2	2.0	2.0	2.0	135	125	130	129
	40'	8.8	8.9	9.2	8.9	2.2	2.1	2.1	2.1	134	129	134	128

Panel length	Bracing angle	Mass	Total deformation Max.	X-axis deformation Max.	Y-axis deformation Min.	stress column bracing	stress column top	stress truss bracing	stress truss top
		Tone	mm	mm	mm	N/mm^2	N/mm^2	N/mm^2	N/mm^2
L/30	20'	0.412	58	13	-57	-27	-65	-58	-79
	30'	0.354	38	13	-37	-27	-80	-74	-76
	40'	0.348	37	14	-36	-21	-84	-75	-78
L/35	20'	0.329	62	21	-59	-31	-89	-111	-119
	30'	0.297	51	24	-46	-29	-122	-93	-126
	40'	0.314	42	22	-38	-26	-120	-80	-105

[Here (–ve) deformation indicate vertical downward deformation and (-ve) stress indicate compressive value]

IV. CONCLUSION:

Results show the optimized weight and various constrains in ANSYS® Workbench. The weight obtained in ANSYS® by using Response surface optimization toolbox is optimized successfully for the proposed models and the deflection and stress criteria are satisfied. It has been concluded that the optimization by using Response surface optimization toolbox gives more reliable and satisfying results. Hence it is advantageous to use this method for different varieties and configuration of the structure.

V. REFERENCES:

[1] Zheng Zhang, Xuefeng Cai: Finite Element Analysis on Frame Structure of Light Steel Temporary Buildings. Applied Mechanics and Materials Vols 351-352 (2013) pp 808-811 Trans Tech Publications, Switzerland

[2] Fankui Zeng, Changming Hu: Design Method of the Temporary Support Structure in Building Construction. Advanced Materials Research Vols 163-167 (2011) pp 406-409 Trans Tech Publications, Switzerland

[3] Michalis Fragiadakis, Nikos D. Lagaros, Manolis Papadrakakis: Performance-based multi-objective optimum design of steel structures considering life-cycle cost © Springer-Verlag 2006

[4] Nikolaos D. Lagaros, Manolis Papadrakakis: Structural optimization using evolutionary algorithms © 2002 Elsevier Science Ltd.

[5] IS 800: 2007, IS: 875 (Part 1) – 1987, IS: 875 (Part 2) – 1987 and IS: 875 (Part 3) - 1987