International Journal of Mechanical and Production Engineering, ISSN(p): 2320-2092, ISSN(e): 2321-2071
Volume- 6, Issue-4, Apr.-2018, http://iraj.in

PLATE HEAT EXCHANGER AND ITS STRUCTURE ANALYSIS FOR WITHSTANDING LOAD WITHIN ALLOWABLE LIMIT

[1]JI YUN KANG, [2]DONG-WON JUNG

[1]The Project Management Center Cultivating Smart Grid & Clean Energy Man power(CK-1)
690-756, Republic of South Korea
[2]Department of Mechanical Engineering, Jeju National University, 1 Ara 1-dong, Jeju-si, Jeju,
690-756, Republic of Korea
E-mail: [1]rkdwldbs7592@gmail.com, [2]jdwcheju@jejunu.ac.kr

Abstract - A plate heat exchanger is a type of heat exchanger that uses metal plates to transfer heat between two fluids. This has a major advantage over a conventional heat exchanger in that the fluids are exposed to a much larger surface area because the fluids spread out over the plates. Plate and frame exchangers offer the highest efficiency mechanism for heat transfer available in industry today. Plate heat exchanger is used in many industrial and household applications like water heater, cooling tower isolations, waste heat recovery and thermal storage systems. This paper presents a simulation that analyzes stress during different load condition in order to improve the design of a plate heat exchanger.

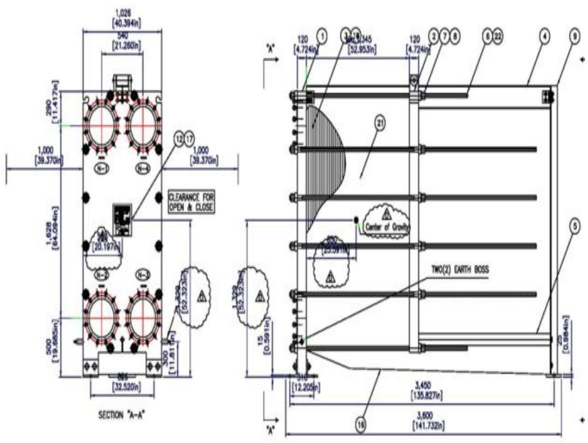
Keywords - Plate heat exchanger; Structure analysis; ANSYS; Numerical simulation; Seismic load.

I. INTRODUCTION

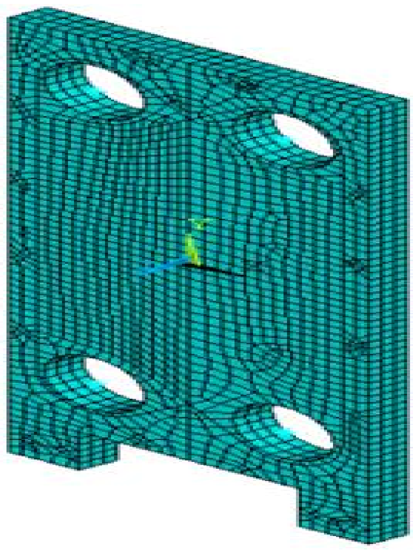
The design report demonstrates that the Plate Heat Exchanger is capable of withstanding the loads specified in material requisition [1] and will be performed in accordance with API 662[2] and ASME code [3, 4]. The numerical simulation provides results of the stress analysis for the proposed plate heat exchanger.

The ANSYS computer program was used to perform the dynamic analysis, and the structure was modeled by using finite element techniques [5-7]. The model was then simultaneously subjected to a dead load, a seismic load, and a nozzle load. The analysis calculates the stresses for various parts of the equipment due to dead, nozzle, and seismic loads. The parts include shell mid-surface, shell bottom and top surface. Therefore, we can have guidance during the design of the plate heat exchanger.

II. ANALYSIS DATA



(a) General configuration



(b) Solid model

Fig.1. The drawing of the plate heat exchanger

The two-dimensional drawing and the three-dimensional image of the plate heat exchanger is shown in Fig. 1. After modeling is finished in ANSYS, we can directly use the model to perform the simulation [8]. The properties of the material in different parts of the heat exchanger are shown in the Table1.

Unit: MPa

	Material	Allowable Stress (Sa)	Yield Strength (Sy)	Modulus of Elasticity (E)
Remove Frame	SA516 Gr.70N	138	192	198960
Fixed Frame	SA516 Gr.70N	138	192	198960
Plate	SB265 Gr.1	132	345	103960

Tight bolt / nut	SA193-B7 / SA194-2H	172	-	200960
Upper G/B	SA240-304	115	154	185880
Support	SA36	Note 3	230	198960
Anchor bolt	SA307-B	Note 3	-	198960

Table 1: Material properties of the different parts of the plate heat exchanger

Note)
1. Sy = yield strength
2. Values from applicable Table, part D [9]
3. Stress limits for anchor bolts and support frame shall be as per AISC (Ref.8)

The weight of the empty containment spray heat exchanger is 6,174 kg. During operation condition and testing its weight is 7,894 kg. The pressure and temperature for closed cooling water and sea water are shown in the Table 2. Nozzle loads are shown in Table 3, and the data are quoted from the technical specifications and the drawing of the outline [10]. Table 4 shows the different states of pressure for plate heat exchanger during test.

Table 2: Design pressure and temperature of the plate heat exchanger

	Closed cooling water	Sea water
Design Pressure, kg/cm^2 g	12.7	10.5
Design Temperature, °C	82.0	82.0

Nozzle name	Severe-service nozzle loading					
18”	F (kg/N)			M (kg-m/N-m)		
	Fx	Fy	Fz	Mx	My	Mz
Hot side (N1, N2)	2,901/28,430	2,901/28,430	2,901/28,430	4,993/48,931	4,993/48,931	4,993/48,931
Cold side (N3, N4)	2,901/28,430	2,901/28,430	2,901/28,430	4,993/48,931	4,993/48,931	4,993/48,931

Table 3: The loads for the nozzle

Note) 1.Fz : Radial force, Fx&Fy : Shear force,
Mz: Torsional moment, Mx& My: Bending Moment
2. Globe axes X, Y & Z are mutually perpendicular at the equipment base / foundation,
Where, axis Y coincides with the vertical direction

Table 4: Pressure from the test

	Hot side	Cold side
Test Pressure, kg/cm^2 g	16.5	13.7

III. ANCHOR BOLT DESIGN

Anchor bolt and Support frame design shall be performed considering dead load, nozzle loads, wind load or seismic load for normal condition and evaluation shall be done in accordance with the limit of AISC[11]

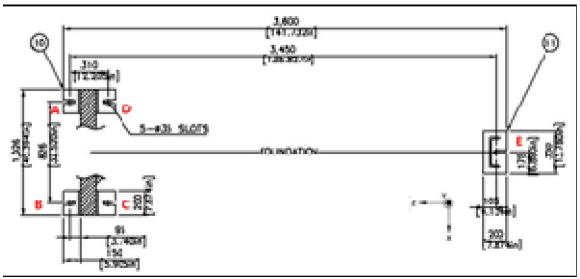


Fig.2. Anchor bolt arrangement

Table 5.Anchor bolt input data

Bolt Size	1.25	(in)
Number of Bolt (N)	5	(EA)
Number of threads per inch (n)	7	-
Bolt Material	SA307-B	-
Tensile Strength for Bolt	60,000	(psi)

The tension strength calculated in operation condition will be compared with allowable strength limit as shown in Table 6.

Unit: kg_f

		Design Condition						Judgement
		Bolt A, B		Bolt C, D		Bolt E		
		Calculated	Allowable	Calculated	Allowable	Calculated	Allowable	
Tensile	(Nu)	9,594	12,956	8,478	13,762	5,593	8,767	OK
Shear	(Vu)	3,379	7,914	2,949	7,914	5,613	7,914	OK

Table 6: Tension and shear strength check

IV. FINITE ELEMENT MODEL

Stress analysis for fixed frame under pressure and nozzle load, the ANSYS solid elements of SOLID45 (4-noded structural solid) were selected to model the apparatus, and its materials properties are shown in Table 7 below

Unit: MPa

	Design Temperature	Allowable Stress (Sa)	Yield Strength (Sy)
Fixed frame (SA516-70)	C	138.0	192

Table 7: Materials properties

Fig.3 shows the finite element model used for analysis. Fig.4 shows boundary conditions applied finite element model and Nozzle load shall be applied at the assumed flange face for pipe connection. For the boundary conditions, four nodes were pinned on a fixed support base (Fx, Fy, Fz fixed) and (Mx, My, Mz moments) [12]. ANSYS 'POST1' categorizes the stress into membrane, bending, membrane + bending, peak, and total stresses inside & outside the surface and the center of the cut section. SI unit shall be used for the final results values during evaluation.

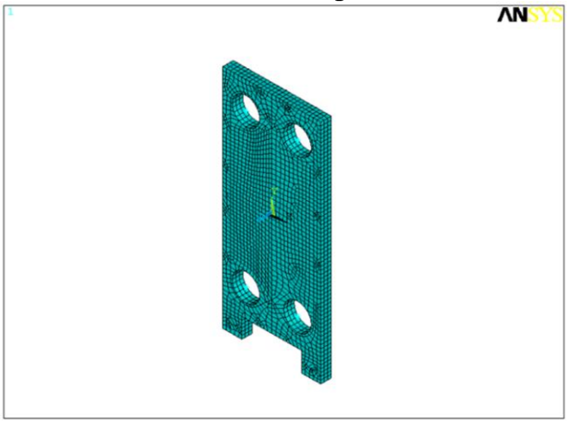


Fig.3. Finite Element model

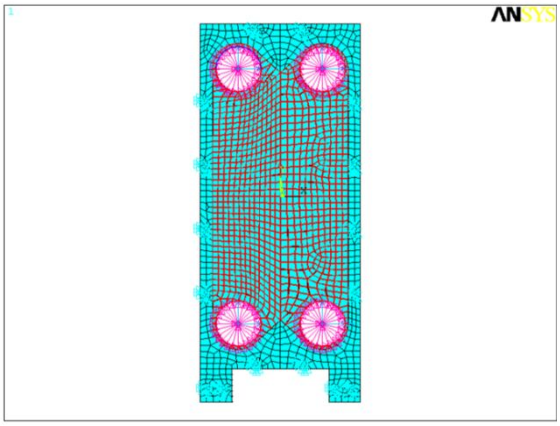


Fig.4. Boundary conditions applied tothe finite element model

V. UPPER GUIDE BAR AND SUPPORT DESIGN

This study we verify that upper guide bar and support would be adequate for the intended service. In order to accomplish this verification, all structures have been studied using the "ANSYS" computer program[10].Material properties are shows in Table 8.

Unit: MPa

	Material	Allowable stress (Sa)	Yield strength (Sy)	Allowable deformation limit
Upper G/B	SA240-304	108.2 / 147.5 (Note 1)	17	0.5% (Note 2)
Support	SA36	151.8 / 207 (Note 1)	230	0.5 % (Note 2)

Table 8: Material properties

Note1. Allowable bending stress is 0.66 x Sy for design condition and 0.9 Sy for design condition with wind or seismic loads per AISC.

2 The maximum deformation shall not exceed 6 inch per 100 ft. of length

VI. ANALYSIS OF THE SIMULATIONS IN ANSYS

Since the ANSYS program uses linear distribution of stress through the thickness of solid elements, equivalent values of shell mid-surface, shell bottom and top surface present linearized results. The mid-wall stress is a shell membrane stress while the top and bottom surfaces are membrane plus bending stresses. Hence, the mid-wall stress can be compared directly with the ASME code allowable stress for membrane stress and the two surface membranes plus bending stresses can be compared directly with the code allowable for membrane plus bending. Fig.4 shows the distribution of equivalent stress for design condition. Fig.5 shows path location of the center, edge of fixed frame and near the nozzle for the calculation of the linearized equivalent stress.

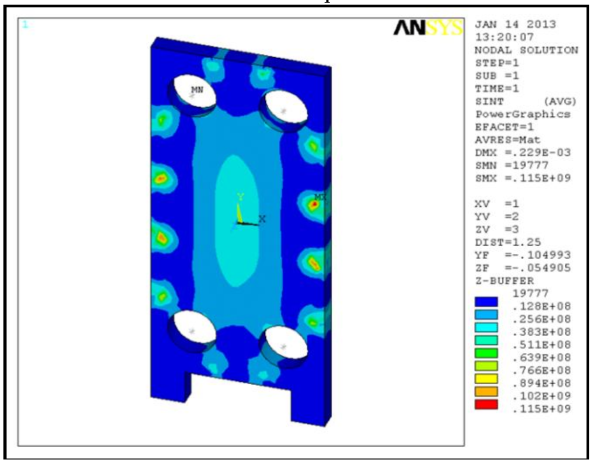


Fig.5. Distribution of equivalent stress on fixed frame for design condition

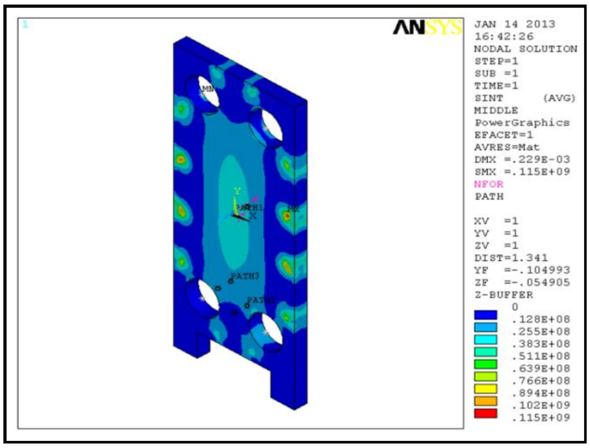


Fig.6. Path location on fixed frame

VII. FINITE ELEMENT MODEL

The finite element analysis code selected for this work was the ANSYS 11.0.ANSYS solid elements of SOLID45 (4-noded structural solid) and BEAM188 (2-noded linear finite strain beam) were selected for modeling. Fig. 6 is solid model and Fig. 7 is finite element model for analysis. Fig. 8 is boundary conditions applied finite element model. The units used in this analysis have been mm and N. Therefore SI unit shall be used for the final results during the evaluation.

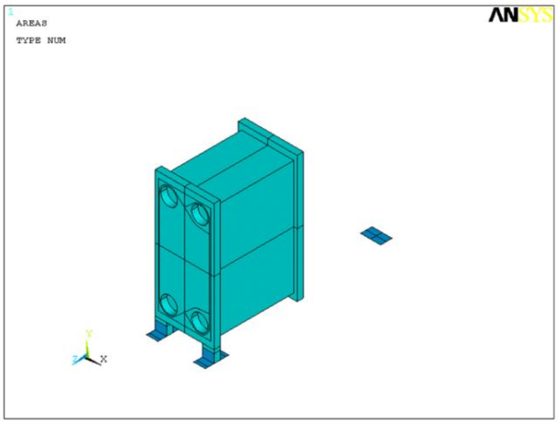


Fig.7. Solid model for FEM analysis

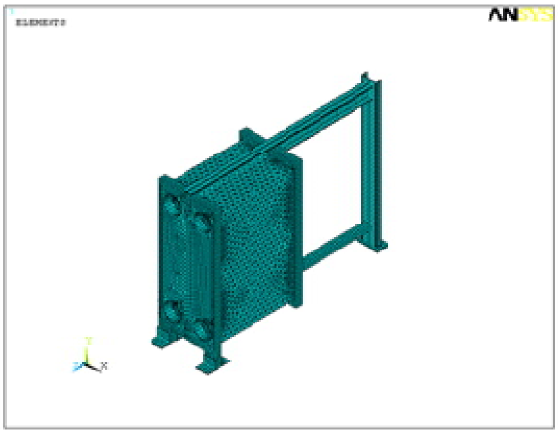


Fig.8. Finite element model for FEM analysis

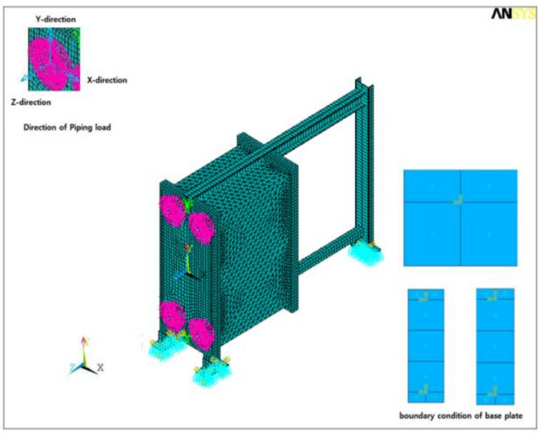


Fig.9. Applied boundary conditions for FEM analysis

VIII. ANALYSIS OF THE SIMULATIONS IN ANSYS

Since the ANSYS program uses linear distribution of stress through the thickness of shell elements, stress values of shell mid-surface, shell bottom and top surface present linearized results. The mid-wall stress is a shell membrane stress while the top and bottom surfaces are membrane plus bending stresses.

Hence, the mid-wall stress can be compared directly with the code allowable stress for membrane stress and the two surface membranes plus bending stresses can be compared directly with the AISC allowable values for membrane plus bending. Distributions of equivalent stress in the upper guide bar and support are shown from Fig. 9. to Fig. 14. Only maximum equivalent stresses are checked because the stress level is lower than the allowable stress as shown in Table 9.

Location	Loading	Max. Equivalent stress (MPa)	Deformation (DMX) (mm)	Allowable limit		Status / Remark	
				Stress (MPa)	Deformation (mm)		
Upper G/B	Nozzle loads	14.1	7.16	-	-	OK	Fig. 9
	Dead weight	7.66	3			OK	Fig. 10
	Sub total	21.75	-	151.8	-	OK	-
	Seismic loads	9.02	3.9	-	-	OK	Fig. 11
	Total	30.78	14.06	207	17.4	OK	-
Support	Nozzle loads	65.6	7.3	-	-	OK	Fig. 12
	Dead weight	35.8	3.27			OK	Fig. 13
	Sub Total	101.4	-	151.8	-	OK	-
	Seismic loads	42.4	3.958	-	-	OK	Fig. 14
	Total	143.8	14.53	207	14.6	OK	-

Table 9: Maximum equivalent stresses on upper guide bar and support

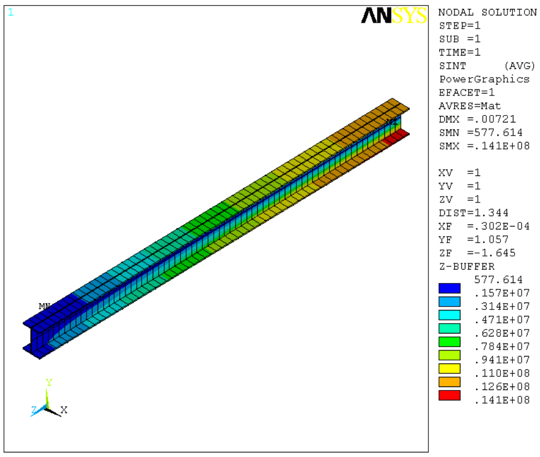


Fig.10. Distribution of equivalent stress dueto nozzle loads for upper guide bar

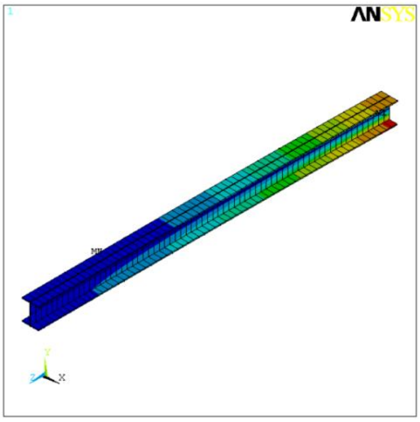


Fig. 11 Distribution of equivalent stress due to dead weight condition for upper guide bar

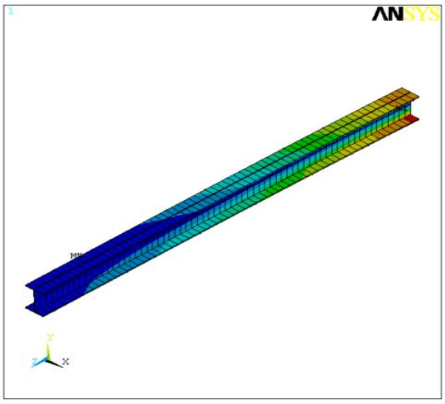


Fig.12. Distribution of equivalent stress due to seismic loads for upper guide bar

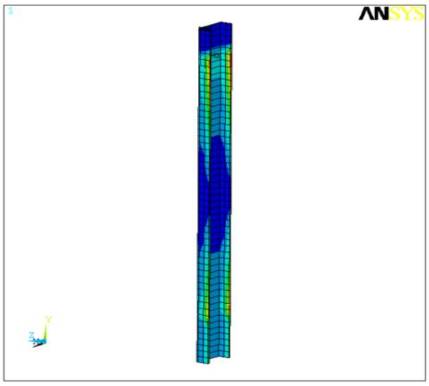


Fig.13. Distribution of equivalent stress due to nozzle loads for support

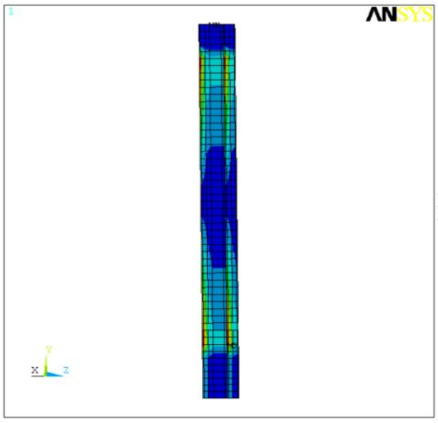


Fig.14. Distribution of equivalent stress due to dead weight condition for support

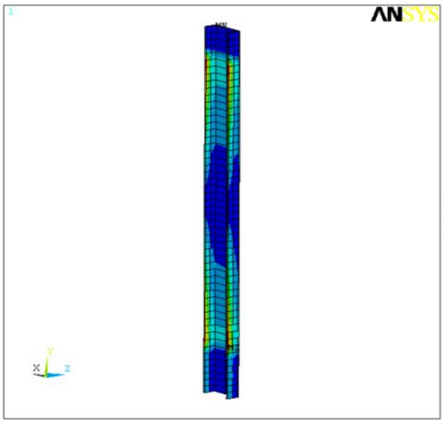


Fig. 15 Distribution of equivalent stress due to seismic loads for support

CONCLUSION AND DISCUSSION

This design report demonstrates that the Plate Heat Exchanger is capable of withstanding the loads specified in material requisition. After analyzing the results of the simulations in ANSYS, we can find The Plate type Heat Exchanger has been verified by the stress analyses that all the stresses were within the allowable limits [4,11,12].

ACKNOWLEDGEMENT

This research was financially supported by The Project Management Center Cultivating Smart Grid & Clean Energy Manpower (CK-1), JNU.

REFERENCES

[1] Material Requisition (MR-090768-13-212-001 Rev.A)
[2] API 662 / ISO 15547 2000 Edition
[3] ASME Section II, 2007 Edition with 2009 Add.
[4] ASME Section VIII Div.1, 2007 Edition with 2009 Add.
[5] Min Zeng, Ting Ma.Effects of lateral fin profiles on stress performance of internally finned tubes in a high temperature heat exchanger. AppliedThermal Engineering. Volume, 10 January 2013, Pages 886–895.
[6] E.K. Wang, M. Jagfeld. Sub structuring for Large Structures Using Finite Element Program ANSYS on a WorkStation. Control and Dynamic Systems. Volume 59, 1993, Pages 137–166.
[7] Y. Kaptan, E. Buyruk, A. Ecder. Numerical investigation of fouling on cross-flow heat exchanger tubes with conjugated heat transfer approach. International Communications in Heat and Mass transfer. Volume 35, Issue 9, November 2008, Pages 1153–1158
[8] Ting Ma, Yitung Chen, Min Zeng, Qiuwang Wang. Stress analysis of internally finned bayonet tube in a high temperature heat exchanger. Applied Thermal Engineering. Volume 43, October 2012, Pages 101–108.
[9] ASME Section II, 2007 Edition with 2009 Add.
[10] Jonghyeok Lee, Kwan-Soo Lee, Flow characteristics and thermal performance in chevron type plate heat exchangers. International Journal of Heat and Mass Transfer. Volume 78, November 2014, Pages 699–706.
[11] AISC Manual, Load and Resistance Factor Design, Third Edition
[12] ASME Section VIII Div. 2, 2007 Edition with 2009 Add.