Stress Analysis of thick wall bellows using Finite Element Method

Digambar J. Pachpande[1]
Post Graduate Student
Department of Mechanical Engineering
V.J.T.I. Mumbai, India

Prof. G. U. Tembhare[2]
Assistant Professor
Department of Mechanical Engineering
V.J.T.I. Mumbai, India

Mr. Sangeev M. Wagle[3]
Director
CADEM Services, Pune

Abstract— This paper mainly focuses on Stress Analysis of thick wall bellows in heat exchanger using Finite Element Method. The main Purpose of bellows is to withstand axial thermal deflection and equivalent internal shell side Pressure. For analysis, thick wall bellows is considered as axisymmetric object. In this paper we are calculating stresses in the thick wall bellows by FEM program using C# programming language and comparing the results with FEA software (ANSYS) results.

Keywords—8 node axisymmetric Quadrilateral elements; FEA; Gauss elimination method; penalty approach; Thick wall bellows

1. INTRODUCTION

The research on the determination of stresses in thick wall bellows has never stopped because of their importance in heat exchanger. The main purpose of thick wall bellows is to withstand with axial thermal deflection and equivalent internal shell side pressure. The TEMA-9th edition gives importance of FEA for thick wall bellows because of drawbacks of old design procedure which may cause overestimation and overstress in knuckle region.

In this paper we calculate the results for two operating condition of bellows, the first is Differential Expansion and second is Shell side pressure + tube side pressure + differential thermal expansion. Actually There are seven operating condition on which stress are calculated. These operating conditions are calculated by different combination of shell side pressure, tube side pressure and differential thermal expansion. From this different combination, the shell stress is calculated and these shell stress are used to calculate axial deflection which applied to thick wall bellows. The Differential expansion condition mean only the effect of thermal expansion is considered and shell side pressure + tube side pressure + Differential expansion mean the effect of shell side pressure , tube side pressure and thermal expansion is considered.

For analysis, the thick wall bellows are modeled by 8 node axisymmetric Quadrilateral elements [6] as shown in fig. 1. This type of element is called serendipity element. In this axisymmetric problem, the model is revolved about axis of revolution 'z' and these problems subjected to axisymmetric loading. The thick wall bellow subjected to internal pressure and axial displacement.

Finite element method is used to find the displacement and stresses at each nodes by using C# (computer programming language). Results obtained by FEM program are then compared with FEA software (ANSYS) result and the comparison is discussed.

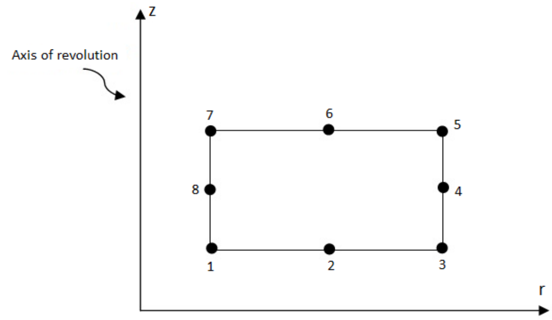


Fig.1 8 node axisymmetric quadratic element

2. PROCEDURE FOR DESIGN AND ANALYSIS

The Following procedure is to be carried out for design and analysis of thick wall bellows to obtain the desired result.

2.1 *Model description*

The following *Fig.2* shows of a two-dimensional model of a thick wall bellows. We consider the thick wall bellows are static, solid, structural, linear and isotropic material model. In geometry definition the cylinder length *li* should not be less than $3.6(G.ts)^{1/2}$ [6].

The dimensions of the bellows are given below:

- O.D of outer cylinder=19.6875 inch,
- I.D of outer cylinder= 19.3125 inch,
- G=32.125 inch,
- Length of outer cylinder, lo=2 inch,
- Length of inner cylinder, li=13.4962 inch,
- Thickness of bellows, t = 0.375 inch,
- Thickness of shell, ts = 0.4375 inch
- Radius of inner and outer knuckle ra=rb=1.125 inch.

(This work is licensed under a Creative Commons Attribution 4.0 International License.)

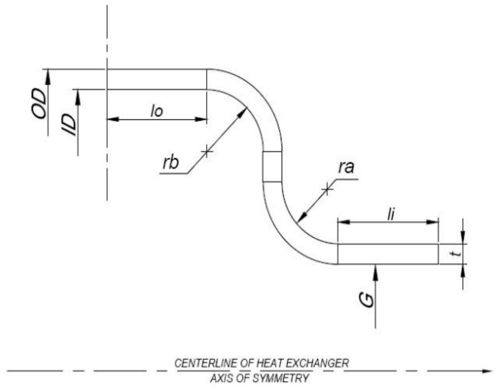


Fig.2 two dimensional model of thick wall bellow

2.2 Design data

Design code – TEMA, 9th edition, section -5, RCB-8

Internal Design Pressure, Pi = 250 psi (Shell Side)

Corrosion allowance = 0 mm

2.3 Material data

The Material Properties has been taken from ASME Sec II Part D. The Mechanical & thermal properties of material used for thick wall bellows for various components are given below:

- Material - SA 516 GR. 70
- Allowable stress - 38000 psi
- Mean metal temperature for the shell - 129^{o} F
- Mean metal temperature for tube- 187^{o} F
- Modulus of elasticity E - 27400000 psi
- Shell stress for case-1 = 838.72 psi
- Shell stress for case-2 = 1129.5 psi
- Poisson's ratio υ - 0.3

2.4 Load Cases

Case 1- Differential thermal expansion

Case 2- Shell side pressure + Tube side pressure + Differential expansion

2.5 Boundary condition and loading condition

In *Fig.3* shows the boundary condition and loading condition in the thick wall bellows. The internal pressure of thick wall bellows is acting on shell side shell. The smaller diameter end is unrestrained in axial direction & restrained in radial direction and the large diameter end are restrained in the axial direction & unrestrained in radial direction.

The loading in the axial direction is entered as displacement which is calculate as $\delta = (Ss \times As) / K_{FES}$ where Ss= shell stress, As=shell cross-sectional area and K_{FSE}=spring rate of main shell. In general, $\delta applied = \delta\ (1/2N_{FSE})$ where δ=amount of applied displacement and N_{FSE} is total number of bellow.

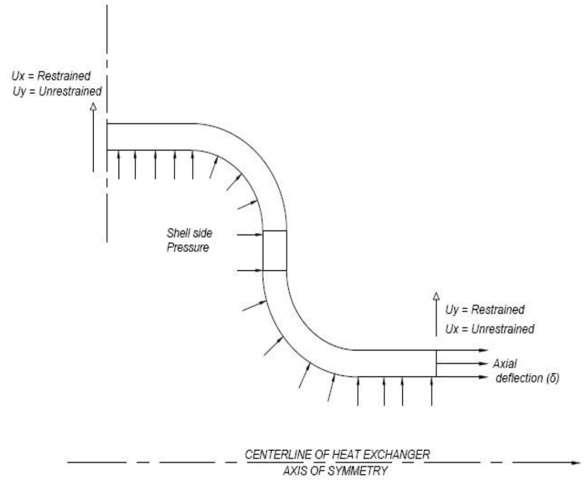


Fig.3 Boundary and loading condition

The calculated applied displacement from shell stress for both case-1 and case-2 are given Table 1:

Table 1 calculated displacement for both condition

Sr.NO.	Operating condition	Shell side pressure (psi)	Shell stress (psi)	Applied displacement(δ) inch
1	Differential expansion	250	838.72	0.01517
2	Shell side pr +Tube side pr +Differential expansion	250	1129.58	0.02043

2.6 Meshing of thick wall bellow

The meshing in finite element models is the very important step in during analysis because it affects the accuracy and the economy of the solid model. The mesh is developed for thick wall bellows are structured mesh and uses eight node quadratic axisymmetric elements. The meshing model of the thick wall bellows are shown in Fig.2

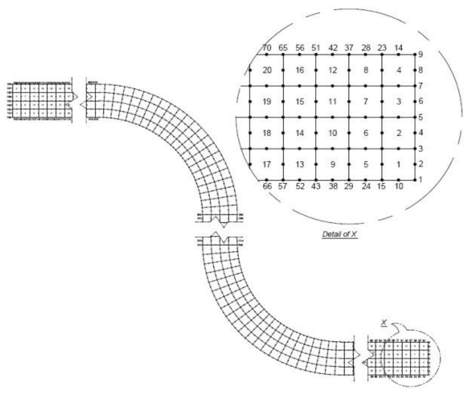


Fig.2 meshing model of thick wall bellows

3. ANALYSIS OF THICK WALL BELLOWS USING FEM PROGRAM

The Finite Element Method (FEM) is one of the most commonly used numerical analysis method for the structural analysis. We perform Fem program using computer programming language C#. The steps FEM program is given below:

(This work is licensed under a Creative Commons Attribution 4.0 International License.)

i. This integrated program takes inputs pertaining basic dimension & material properties, then discretizes the geometry as per 8 node quadratic element.

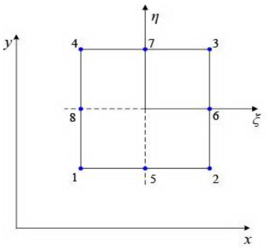


Fig. 4 8 node quadratic element

ii. After meshing, calculate stiffness matrix [Ke] and load vector [Pe] for each element using shape function [N] of 8 node quadratic element and its derivative's.

iii. After calculating stiffness matrix [Ke] and load vector [Pe], transform this local matrix into the global matrix [Kg] and [Pg].

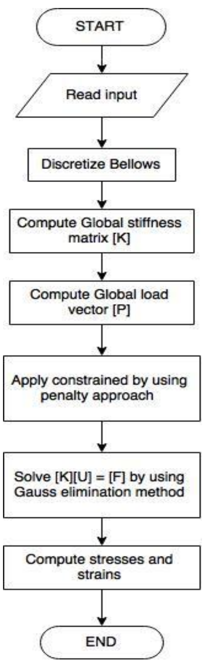
iv. Once we have [Kg] [U] = [Pg], apply boundary condition and modify the [Kg] and [Pg] using penalty approach.

v. Now, solve [Kg] [U] = [Pg] by using Gauss elimination method which gives output as unknown displacement of each node. In Gauss elimination method, the result is calculated by making lower triangle to zero.

vi. Once we know unknown displacement of each node, we can easily calculate stress and strain of each element.

vii. After calculating stress, compare the output with allowable stress to check whether design is safe or not.

Flowchart for FEM program:



4.

Fig.4 Flow chart of FEM program

5. ANALYSIS OF THICK WALL BELOW USING FEA SOFTWARE (ANSYS)

Software – ANSYS 14.5

Analysis type – Structural analysis

Element type – Solid elements

Element – 8 nodes 183

The results obtain from ANSYS for case-1 and case-2 are given below:

Case-1: Differential expansion

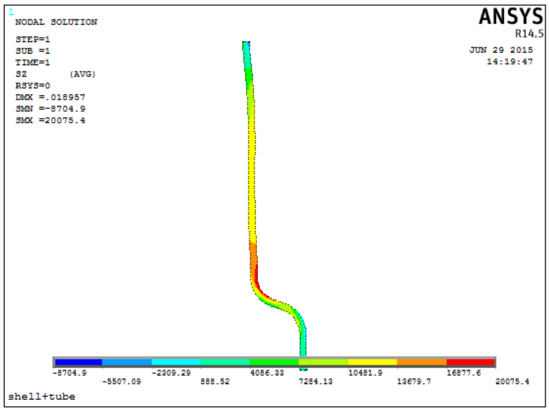


Fig.5 Circumferential stress for case-1

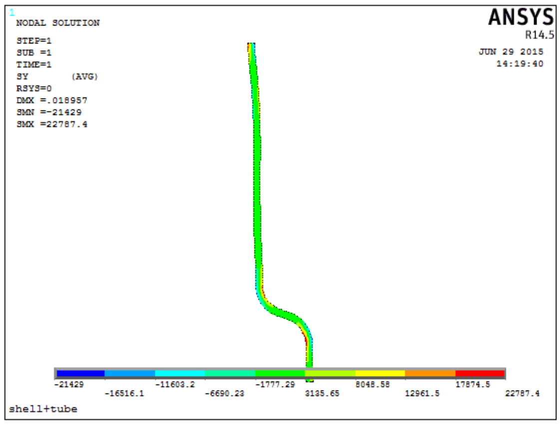


Fig.6 Radial stress for case-1

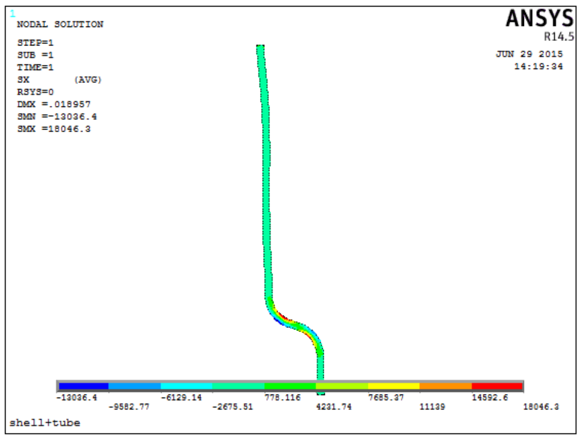


Fig.7 Axial stress case-1

(This work is licensed under a Creative Commons Attribution 4.0 International License.)

Case-2: Shell side pr + Tube side pr + Diff. expansion

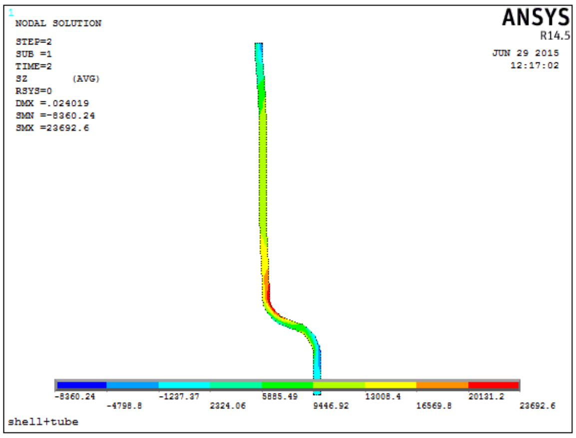


Fig.8 Circumferential stress for case-2

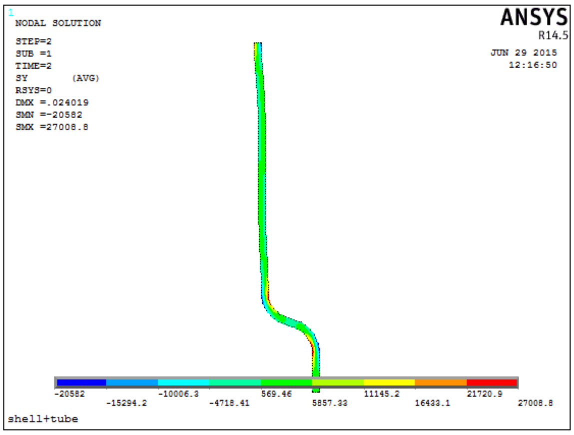


Fig.9 Radial stress for case-2

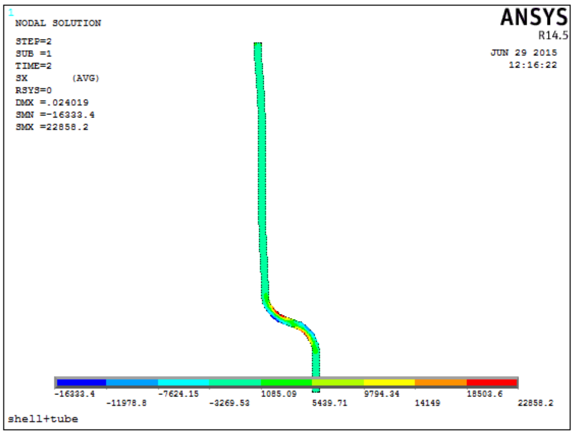


Fig.10 Axial stress for case-2

6. RESULT

The result obtains from program are as shown in following Table 2. The stresses are radial stress (σr), circumferential stress (σθ) and axial stress (σz). The influencing stress generally in the thick wall bellows is circumferential stress. The results obtained from the program are shown in Table 2.

Table 2 Fem program output

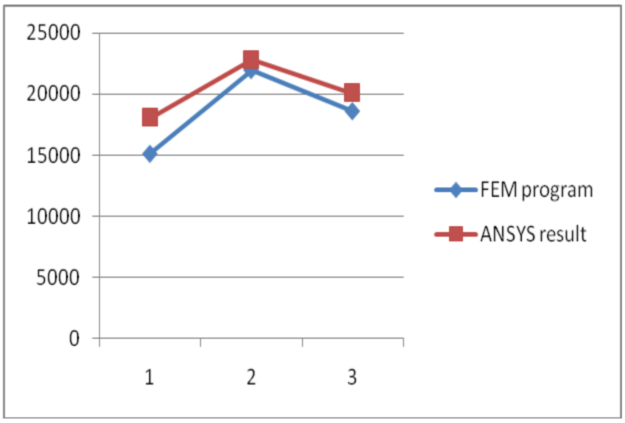
Sr. no.	Load case	Stress	FEM program
1	Differential expansion	σr	15137.99
		σθ	21939.81
		σz	18602.74
2	Shell side pr +Tube side pr +Differential expansion	σr	20395.26
		σθ	25653.27
		σz	25131.4

After calculating stresses in FEM program and ANSYS software, we compared the results obtained from both methods. The comparisons of results are shown in following Table 3.

Table 3 Comparison of results between FEM program and ANSYS

Sr.No.	Load case	Stress	ANSYS (psi)	Allow. stress	Safe or unsafe
1	Differential expansion	σr	18046.3	38000	Safe
		σθ	22787.4	38000	Safe
		σz	20075.4	38000	Safe
2	Shell side pr. +Tube side pr. +Differential expansion	σr	22858.2	38000	Safe
		σθ	23692.6	38000	Safe
		σz	27008.2	38000	Safe

The result of stress plots obtained from FEM program and ANSYS are as shown in following figure 11, 12.



7.

Fig.11 Comparison plot of result between Fem program and ANSYS for case-1.

(This work is licensed under a Creative Commons Attribution 4.0 International License.)

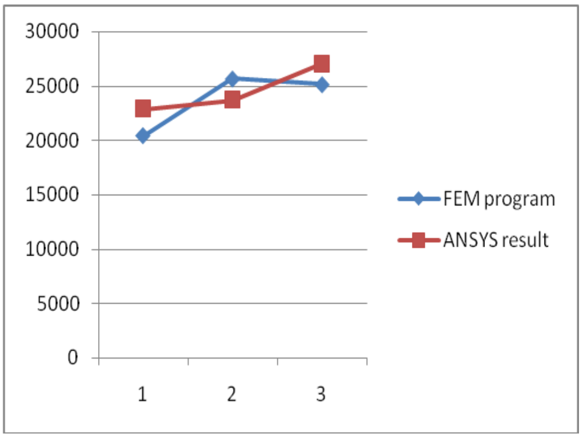


Fig.12 Comparison plot of result between Fem program and ANSYS for case-2

8. Result discussion

From result table we can conclude that result obtain from program are closer to results obtain from ANSYS FEA. We can further optimize the results of the program by decreasing the mesh size so that we can get fine meshing and accurate results. We calculated results for two operating conditions consisting of 1) differential expansion and 2) Shell side pr. + tube side pr. + differential expansion. The results obtained from FEM program is less than allowable stress, hence the design is safe.

9. Conclusion

In this paper we established logic and methodology for FEM program for analysis of thick wall bellows. By using this FEM program for we can save cost and time required for analysis of thick wall bellows on FEA software.

10. Reference:

Journal:

[1] Mohite and Edlabadkar "Analysis of Expansion Joint in heat exchanger using Finite Element Analysis Method" Volume 2 (9): 436-449, 2014.
[2] Mircea Cristian "Experimental Stress Analysis in Bellows Subjected To Axial Loads".
[3] Ergatoudis, B. M. Irons and 0. C. Zienkiewicz "Curved, Isoparametric Quadrilateral elements For Finite Element Analysis", Vol. 4, pp. 31 to 42, 1968.
[4] J.D. Claytona, J.J. Rencisb, "Numerical integration in the axisymmetric finite element formulation" 1999.
[5] Sharaban Thohura, Dr. Md. Shahidul Islam, "Study of the Effect of Finite Element Mesh Quality on Stress Concentration Factor of Plates with Holes" Volume 3, Issue 6, December 2013.
[6] TEMA-9th, section-5, RCB-8
[7] TEMA-8th, section-5, RCB-8

Books:
[8] G. Lakshmi Narasaiah, "Finite Element Analysis", BS publication.
[9] "Fundamentals of computer programming with C# - The Bulgarian C# book," Svetlin Nakov & Co.

(This work is licensed under a Creative Commons Attribution 4.0 International License.)