ISSN 1 746-7233, England, UK
World Journal of Modelling and Simulation
Vol. 14 (2018) No. 3, pp. 174-180

Static Analysis of Stress Concentration and Elastic Follow-Up of Two Typical Power Plant Pipes with Structural Discontinuity*

Nasir Hasan Sk[1] , Sushovan Chaterjee[2†]

[1] Department of Mechanical Engineering, National Institute of Technology Durgapur, West Bengal, India
[2] Department of Mechanical Engineering, Cooch Behar Government Engineering College, Harinchawra, Coochbehar-736170, West Bengal, India

(*Received December 16 2016, Accepted February 01 2018*)

Abstract. The present paper deals with the static analysis of elastic follow-up and stress concentration of a typical power plant pipes having two different structural configurations (namely pipe bend and pipe with cross-hole) using ANSYS software. Two different pipe models with structural discontinuity are taken as (a) a single pass out of seven pass of boiler super heater pipe is taken as pipe bend model (b) pipe with existing cross holes. The stress analysis of the pipe bend has been conducted and stress factor is calculated along its circumference. In the pipe model with cross hole, the cross hole is a structural discontinuity, for which stress concentration near the hole is high. The elastic follow up in the body is quantified and calculated. The stress distribution of the entire mentioned model has been done for critical zone having risen in stress level.

Keywords: stress concentration, elastic follow-up, ANSYS, modelling, simulation

1 Introduction

Pipes are the integral parts of a power plant, failing of pipe can cause shut down of a plant. So safe designing of power piping is very much required. One can design a pipe by following the standard code or computationally [2]. If structural discontinuity is there then definitely the case of redistribution of stress will arise. During the design of pipes under practical working conditions, criticalities to be imposed specifically in structural irregularities such as cross-hole, bends etc. Stress analysis result gives some analytical estimation by which one can critically incorporate some design modification [10].

In this paper some preliminary studies [5, 7, 8, 12] have been carried on modelling of various structures in ANSYS. Solid 185 element was used from the library which has 6 degrees of freedom. Three boundary conditions are imposed on the pipe based on (a) displacement (b) force and (c) both force and displacement (mixed boundary condition). Stress profile of the geometrical model of the pipe has been plotted to check different critical zone having risen in stress level. This paper deals with the stress distribution in the pipe, which will allow us to see whether the pipe is in elastic or plastic deformation zone. Pipes are having almost all kinds of loads intentional and unintentional. Intentional loads are like fluid pressure inside the pipe, dead weight of the fluid and pipe and unintentional loads are like seismic load, sometimes torsional load etc. The stress distribution analysis is done on a pipe bend and pipe with structural discontinuity. According to this two different geometrical models have been created and analyzed in ANSYS. The brief description of the above mentioned model is given below.

* The authors are thankful to the computational facilities of Department of Mechanical Engineering, National Institute of Technology Silchar, Assam, India.
† Corresponding author. *E-mail address*: sushovan.chatterjee@gmail.com

Published by World Academic Press, World Academic Union

1.1 Pipe bend model

Geometrical model data for bend pipe is also taken from 15 MW co-generative thermal power plant thermal. For pipe bend model a superheated pipe is taken having 180o pipe bend and seven passes. Out of which only one pass is modelled and analyzed for stress analysis. Stress analysis in pipe bend started long time ago and Von-Karman is one pioneer analyzer in pipe bend elbow. He showed that in pipe bend the flexibility is more than the calculated value from the beam shell theory owing to the oval shape formation in the pipe bend. Based on his theory, work was also carried out by Mackenzie et al [9]. They have also worked on the pipe elbow element which will be elaborated afterwards. The modelling of the bend pipe and the analysis is done by ANSYS 14.0 software. For geometrical modelling the data were taken from a power plant as mentioned earlier also.

For modelling in ANSYS 14.0 APDL following procedure is followed:

- All the geometric parameter (length, inner diameter, outer diameter) are fed to ANSYS APDL as given in Table 1.
- The model is discretized using solid 185 element taken from ANSYS element library.
- Material properties (Youngs modulus, Poisons ratio) and load on the pipe (temperature gradient, pressure gradient) as well as boundary conditions has been provided.
- Finally stress, strain has been computed in post processing.

The bottom of the pipe is fixed and allowed to move 2 mm at the beginning of the pipe bend. The temperature distribution over the pipe is taken to be uniform.

1.2 Pipe containing cross-hole model

A pipe may contain opening in the surface due to many reasons like instrumentation, bursting caps, fluid transfer etc. This opening is obviously the structural discontinuity in the pipe which will lead to stress concentration near opening. Thus the limit pressure or load in the vessel will reduce. Now it is a very possible scenario that the stress developed near the opening is greater than the yield stress of the material. So only near the opening strain is plastic where the other part of the body is in elastic strain region. Chamfer may reduce the stress concentration at main portion. Maximum stress concentration move towards the intersection with as increasing cross hole to bore radius ratio [3]. According to ASME boiler and pressure vessel code Section VII. Div. 2, a small plastic deformation in the material is allowed if the cycle is low. Incremental plastic collapse is also not permitted. For the pipe model with structural discontinuity, a model has been made with 10 mm diameter cross hole at the mid potion of the pipe. The modelling is done by same procedure discussed in the pipe bend model.

1.3 Concept of elastic follow-up

Concept of elastic follow-up is given by Robinson in the relaxation of bolted joints because of creep [7]. He described that if flange of joint is rigid than the bolt, the elastic follow-up is insignificant. Only elastic follow-up occur when flange is elastic. Some author described magnitude of elastic follow-up differently from that suggested by Robinson[11]. The phenomenon of elastic follow-up arises when the highly stressed part exhibit any change in overall compliance weather through creep plasticity or creep growth [14]. The strain near the cross-hole is in plastic region where the other part of the body in elastic deformation zone then certain area of that portion will be subjected to strain concentration due to elastic follow-up of rest of the connected structure[1]. A similar setup was given by Smith [13] in his work on elastic follow up is manifested in figure 1. Where 1, 2 are flange holding the full setup 3 is central column on which elastic follow up is tested and 4 is side bar.

Generalization of definition of elastic follow-up factor according to Robinson.

$$Z = \frac{(\varepsilon_{final}) - [(\sigma_{final})/E]}{(\varepsilon_{infinal}) - [(\sigma_{final})/E]} \tag{1}$$

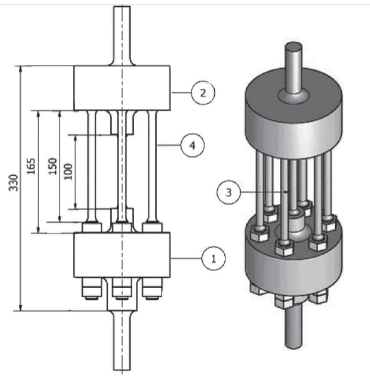

Fig. 1: Set up for elastic follow up calculation by D J Smith[13]

$\frac{\sigma_f inal}{E}$ = Reference Strain.

Where $\varepsilon_f inal$ the final total strain of central bar is, $\varepsilon_i nitial$ is the initial elastic strain that the central bar would achieve at maximum load if it had not exhibited plastic deformation.

When a small portion of the body undergoes plastic deformation and the whole part is in elastic deformation zone then Elastic Follow-Up (EFU) will set up. For quantifying elastic follow up (Z) we have to consider three boundary conditions:

- Displacement control boundary condition
- Force control boundary condition
- Mixed boundary condition

In displacement control boundary condition the body is subjected to a constant deformation. So the body does not experience any plastic deformation ($Z = 1$). In force control boundary condition the body is under constant load irrespective of displacement which will lead to deformation of the body until it collapse ($Z = \infty$). Real structure normally operate under mixed boundary condition i.e. the body is neither displacement control nor force control ($1 < Z < \infty$) [6, 13]. The analytical estimation of percentage elastic follow-up was also performed for other application like autoclave pressure vessel by using equivalent reduced elastic modulus procedure [4].

Table 1: Geometrical modelling parameter of pipe bend and pipe with cross hole

Parameter	Bend Pipe	Pipe with cross hole
Length of straight portion (mm)	2680	500
Outside radius (mm)	33.8	40
Inside radius (mm)	29.8	32
Young's modulus (GPa)	215	213
Poisson's ratio	0.33	0.3
Co-efficient of thermal expansion (/C)	11.6×10^{-6}	11.7×10^{-6}
Inlet pressures (MPa)	3.65	3.73
outlet pressures (MPa)	3.65	3.53
Inlet temperature (°C)	305	105
Outlet temperature (°C)	318.5	-
Ambient temperature (°C)	405	18
Bend Radius (mm)	60	-
Diameter of hole (mm)	-	10
Yield Stress (GPa)	650	750

2 Results and discussion

2.1 Pipe bend model

In pipe bend model boiler super heater pipes are vertically placed and the temperature of the pipe goes on increasing from inlet to the outlet with constant inside pressure. The inlet and outlet of the pipe is fully constrained, thus stress development near inlet and outlet is greater. This is due to the thermal stress at restrained ends. The maximum stress is at outlet region of the pipe because of maximum temperature. And there is a slight increase of stress near the bend of the pipe because of semi-restrained portion. The contour is shown in the figure 2 and figure 3. The maximum stress developed in the pipe is 548 MPa and the yield stress of the material (SA 213 T11) of the pipe is 650 MPa. So the pipe is safe. Since strain is directly proportional to the stress therefore the maximum strain developed in the maximum stress zone.

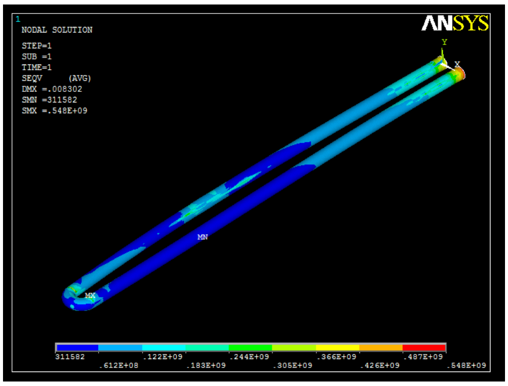

Fig. 2: Von Mises stress contour of pipe bend model

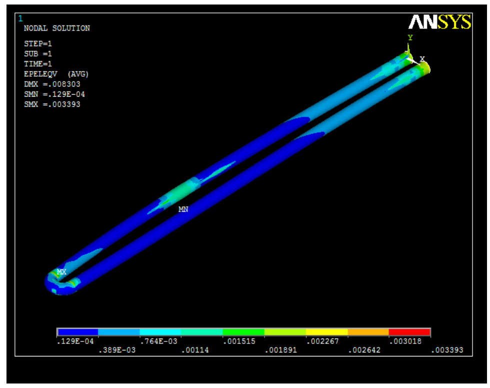

Fig. 3: Von Mises strain contour of pipe bend model

Validation of results of the above analysis is carried out using previous results and their contour. In the pipe stress factor can be calculated by the equation

$$\text{Stress factor}(\sigma *) = \sigma(I/Mr_0) \tag{2}$$

Where I is the second moment of area of pipe, M is the moment at the section at which the stress is measure or calculated and r0 is the outer radius of cross section and is von misses stress.

The element is enhanced to capture the general shell behavior of elbows. The formulation is based on the curved beam theory incorporating shear as well as cross-section ovalization effects. The ovalization of the cross-section varies cubically along the element length. The element PB1 developed by Mackenzie et al. [9]. They work on the Von-Karman theory of pipe bend and Vlasovs thin walled curved beam theory. According to Von-Karman theory of pipe bend, the flexibility in the pipe bend is greater than the calculated value from the beam shell theory due to the ovalization of the pipe bend. Stress factor is nothing but non-dimensionalized form of stress around the circumference of the pipe. It is seen from the previous literature[9] that stress factor goes on increasing with position 00 of the pipe circumference and drops to minimum at 900 position then starts increasing. The stress factor is same at position $\theta = 00$ and $\theta = 1800 - \theta$ as shown in Fig. 5. The present analysis gives the similar pattern of result manifested in Fig. 4. But the values are different due to the different loading on the model.

2.2 Pipe with cross hole

The input pressure in this pipe model is 3.73 MPa. The pipe experiences plastic deformation which is due to minimum stress on the left side of the hole. This is due to the fact that the stress line follow steam

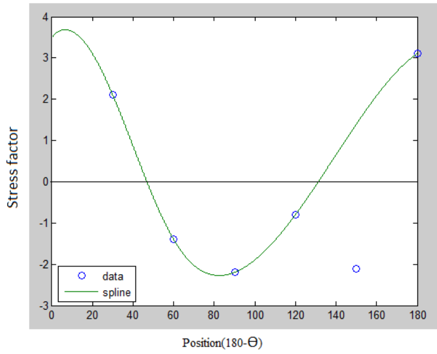

Fig. 4: Variation of stress factor in the circumference

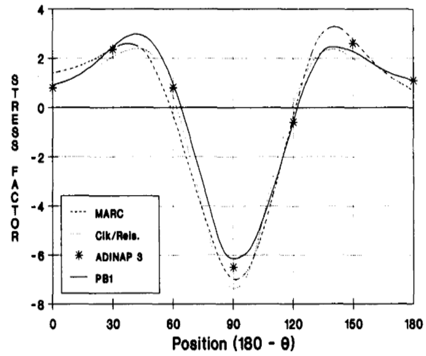

Fig. 5: Variation of stress factor in the circumference by Mackenzie et al.[9]

flow pattern. The stress from the entry side goes in axial direction to the exit side but due to the discontinuity of the model it goes around the hole. Though in plastic deformation of a linear elastic body the stress should have decreased, but the plasticity in the material in ANSYS has not been defined to see the maximum stress value. This is done to find the elastic follow up factor in the pipe. The maximum stress developed in the pipe is 922GPa whereas the yield stress of the material is 750GPa. Thus the pipe experiences plastic deformation but only near the hole. From the strain contour it can be seen that the stress development near the hole is higher than the other parts of the pipe. So by definition the strain near the hole must be greater than the other part.

From the figure 8 and figure 9 the variation of stress strain curve is similar. But due to the different geometrical structure and loading the output is different. The line A and B in the Fig. 8 shows the elastic deformation of the body if it does not exhibited plasticity and plastic deformation respectively. The yield stress of the material of the pipe is 750MPa. But the stress developed in the pipe is up to 922MPa. A small portion near the hole is in plastic deformation zone. By calculating the elastic follow up[13] in the pipe from the below equation the following results are obtained.

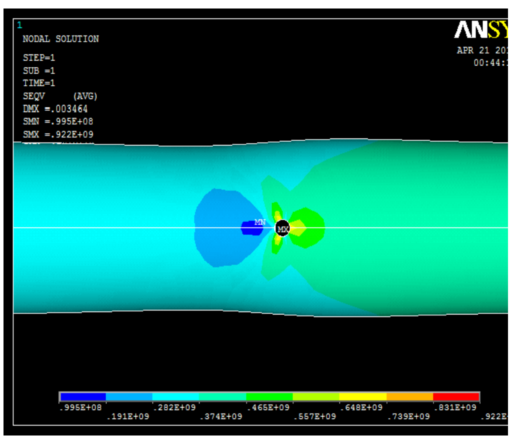

Fig. 6: Stress contour of pipe with cross-hole

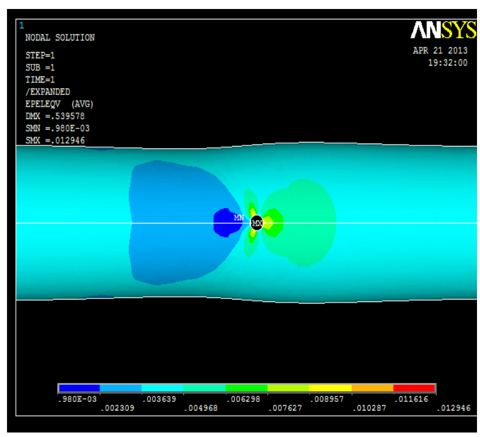

Fig. 7: Strain contour in pipe with cross-hole

$$Z = \frac{\varepsilon_{final} - (\varepsilon_{eq}^{el})_{final}}{\varepsilon_{initial} - (\varepsilon_{eq}^{el})_{final}} \tag{3}$$

Where ε = strain, $(\varepsilon_{eq}^{el})_{final} = \sigma_{final}/E$.
The value of the elastic follow up (Z) in the pipe is 1.4086.
The elastic follow-up factor Z, for the parallel assembly alone, is

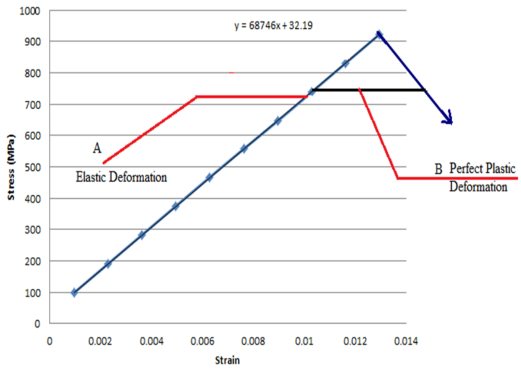

Fig. 8: Stress strain response for pipe with cross-hole

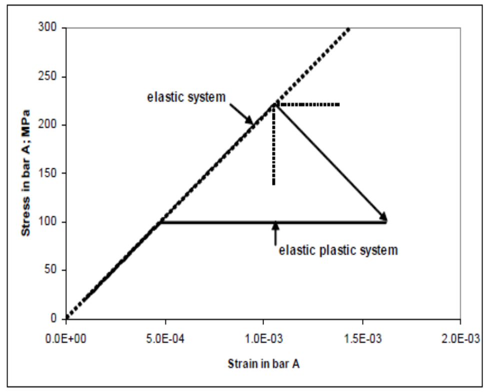

Fig. 9: Stress strain response of parallel bar[8]

$$Z = \frac{1+\alpha}{\alpha}. \tag{4}$$

Where α is stiffness ratio of two bars and that is

$$\alpha = \frac{K_B}{K_A}.$$

Now by drawing graph between the Elastic follow up factor and stiffness ratio. The graph is shown below in figure 10.

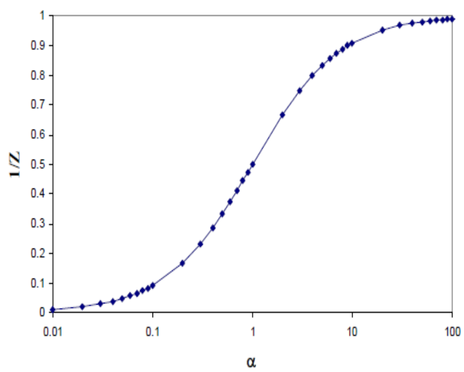

Fig. 10: Variation of inverse of Z (Elastic follow up factor) with stiffness ratio

3 Conclusion

The stress distribution pattern in the different pipe models depend on the geometrical structure of the pipe apart from the magnitude of the load. Elastic follow-up assessment has been calculated for piping system and the results are validated with comparison. The study on the elastic follow up shows that it is useful for relaxation of secondary stress and creep deformation.

References

[1] J. T. Boyle. A re-assessment of elastic follow-up in high temperature piping. *International Journal of Pressure Vessels & Piping*, 2013, s **108C109**(4): 7–12.

[2] Y. T. Chai, K. S. Varyani. An absolute coordinate formulation for three-dimensional flexible pipe analysis. *Ocean Engineering*, 2006, **33**(1): 23–58.

[3] T. Comlekci, D. Mackenzie, J. Wood. Elastic stress concentration at radial crossholes in pressurized thick cylinders. *Journal of Strain Analysis for Engineering Design*, 2007, **42**(6): 461–468.

[4] P. S. Dinkar, S. Chatterjee. Finite element based computational study for assessment of elastic follow-up in autoclave pressure vessel. *International Journal of Advance Research In Science And Engineering*, 2015, (4): 541–546.

[5] G. Dutta, S. K. Panda et al. Electro-magneto-elastic response of laminated composite plate: A finite element approach. *International Journal of Applied & Computational Mathematics*, 2016, (3): 1–20.

[6] S. Hadidi-Moud, D. J. Smith. Use of elastic follow-up in integrity assessment of structures. **in:** *ASME 2008 Pressure Vessels and Piping Conference*, 2008, 523–531.

[7] C. K. Hirwani, H. Mittal et al. Simulation study of stress and deformation behaviour of debonded laminated structure. **in:** *Materials Science and Engineering Conference Series*, 2017, 012005.

[8] S. P. K. Mehar. Elastic bending and stress analysis of cnt-reinforced composite plate: Experimental, numerical and simulation. *Advances in Polymeric Technology*.

[9] D. Mackenzie, J. T. Boyle. A simple pipe bend element for piping flexibility analysis. *International Journal of Pressure Vessels & Piping*, 1992, **51**(1): 85–106.

[10] R. Natarajan, J. A. Blomfield. Stress analysis of curved pipes with end restraints. *Computers & Structures*, 1975, **5**(2): 187–196.

[11] E. Robinson. *Robinson, The resistance to relaxation of materials at high temperature*. Eng, 2007.

[12] N. Sharma, T. R. Mahapatra et al. Numerical study of vibro-acoustic responses of un-baffled multi-layered composite structure under various end conditions and experimental validation. *Latin American Journal of Solids & Structures*, 2017, **14**(8): 1547–1568.

[13] D. J. Smith, J. Mcfadden et al. Elastic follow-up and relaxation of residual stresses. *ARCHIVE Proceedings of the Institution of Mechanical Engineers Part C Journal of Mechanical Engineering Science 1989-1996 (vols 203-210)*, 2010, **224**(4): 777–787.

[14] T. Teramae. A simplified method for elastic follow-up analysis of elevated temperature piping systems. *International Journal of Pressure Vessels & Piping*, 1981, **12**(1): 29–41.