International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395 -0056
Volume: 04 Issue: 06 | July -2017 www.irjet.net p-ISSN: 2395-0072

Stress Analysis Of Buried Pipelines

Pranav Nair*, Dr. Shailendra Naik

Department of Petroleum Engineering, Maharashtra Institute of Technology, Pune, India-411038

---***---

***Abstract** : Oil and gas pipelines are usually buried in ground to provide protection and support. Buried pipeline may experience significant loading as a result of relative displacements of ground along their length In the case of a buried pipeline, forces are statically indeterminate because the characteristic of soil is not uniform (Watkins and Anderson, 2000). The present paper is to analyse the pipeline buried in soil using CAESAR-II software. Main aim of piping stress analysis is to provide adequate flexibility for absorbing thermal expansion, code compliance for stresses and displacement incurred in buried piping system. The design is safe when all these are in allowable range as per code. In this study, a pipeline buried in soil is considered for analysis as per power piping ASME B31.1 code and the results thus obtained are analysed. This paper also examines the typical pipeline behaviour caused by static load in accordance with soil types and a degree of saturation in considered soil. The finite element method (FEM) is selected as the examination method for the underground piping system.*

Key Words **: CAESARII, ASME B31.1,Code compliance , static load ,Finite Element Method**

1. INTRODUCTION

Underground or buried piping are all piping which runs below grade. In every process industry there will be few lines (Oil and gas), part of which normally runs underground. However buried piping or underground piping, appears for pipeline industry is used to carry fluids for long miles.
Analysing an buried pipe line is quite different from analysing plant piping. Due to unique characteristics of a pipeline some special problems are involved that are code requirements and techniques required in analysis. The elements of analysis include, pipe movements, anchorage force, soil friction, lateral soil force and soil pipe interaction.

To appreciate pipe code requirements and visualize problems involved in pipe line stress analysis, it is necessary to first distinguish a pipe line from plant piping. Various unique characteristics of a pipe line include:

• High allowable stress: A pipe line has a rather simple shape which is circular and very often runs several miles before making a turn. Therefore, the stresses calculated are all based on simple static equilibrium formulas which are very reliable. Since stresses produced are predictable, allowable stress used in plant piping are considerably higher.

• High yield strength pipe: In order to raise the allowable, the first obstacle is yield strength. Although a pipe line operating beyond yield strength may not create structural integrity problems, it may cause undesirable excessive deformation and possibility of strain follow up. Therefore, for pipeline construction high test line with a very high yield to ultimate strength ratio is normally used .In some pipe yield strength can be high as 80 percent of the ultimate strength. All allowable stresses are based on yield strength only.

• High pressure elongation: Movement of pipe line is normally due to expansion of a very long line at low temperature difference. Pressure elongation is negligible in plant piping ,which contributes much of the total movement and must be included in the analysis.

• Soil- pipe interaction: The main portion of a pipe line is buried underground. Any pipe movement has to overcome soil force, which can be divided into two categories: Friction force created from sliding and pressure force resulting from pushing. The major task of pipe line analysis is to investigate soil- pipe interaction which has never been a subject in plant piping analysis.

Common materials used for underground piping are Carbon Steel, Ductile iron, cast Iron, Stainless Steel and FRP/GRP.

2.0 BACKGROUND

2.1 Piping flexibility analysis

Piping stress analysis is a term applied to calculations, which address the static and dynamic loading resulting from the effects of gravity, temperature changes, internal and external pressures. The purpose of stress analysis is to ensure safety of piping and piping components as well as the safety of connected equipment and supporting structure . Flexibility as well as stress analysis for this piping system is done through CAESAR II software. Operating loads are calculated using self weight, operating pressure and temperature for the piping system, Sustained loads are by using self weight and operating pressure and Expansion loads are due to temperature differences.

2.2 Finite element method

The objective of the finite elements method is to obtain a formulation that allows the analysis of complex and / or irregular systems through computer programs, automatically. To achieve this goal, the method considers the global system as being equivalent to a group of finite elements, in which each of these is a simple continuous structure. Although the finite element method considering the individual elements as continuous, is in essence is a discretization procedure, which aims to transform an infinite-dimensional problem in finite-dimensional, ie a system with a finite number of unknowns. The resolution of the problem consists in decomposing or discretizes the area under study into small subdomains called "finite element" which are connected by means of discrete points, termed "nodes". The set of elements used in discretization is called mesh.

3. PIPE STRESS ANALYSIS

3.1 CAESAR II MODEL

The piping system taken for case study is VGO- HDT unit of a petroleum refinery . the piping system is modeled using software CAESAR II .The geometric properties of piping system are directly given to the software . the material is to be selected from material library.in .For piping stress analysis piping lay out is modeled first. ASTM A108 Grade B is the pipe material used for high temperature service .Second is to model the buried pipe by using underground modeler by CAESAR II by inputting which all node are buried by marking it in underground modeler input screen .

MECHANICAL PROPERTY	TERM	VALUE
ELASTIC PROPERTY	DENSITY (kg/m3)	78330.43
	YOUNGS MODULUS (M pa)	2.034×10^5
	POISSONS RATIO	0.29
PLASTIC PROPERTY	YEILD STRENGTH (M pa)	490
	TENSILE STRENGTH (M pa)	750

Table 1: Pipe modeling input

Now the task is to create the soil model and input data received from civil. On clicking Soil Models button you will get the window where you have to enter the data. The soil model type used is CAESAR II Basic Model. The modeler uses the values that you define to compute axial, lateral, upward, and downward stiffness, along with ultimate loads. E. Input all known the parameters. Fig 1 shows CEASER II model of the piping system.

Parameter	Value
Friction Coefficient	0.5
Soil Density (Kg/m3)	1600
Buried Depth To Top Of Pipe (mm)	1400
Friction Angle	45
Overburden Compaction Multiplier	8
Yield Displacement Factor	0.015
Thermal Expansion Coefficient	11.2131
Temperature Change (Deg c)	160
Young Modulus (M pa)	24
Poisson Ratio	0.29
Dilation Angle	2
Cohesive yield Strength (M pa)	100

Table 2 : Soil modeling inputs

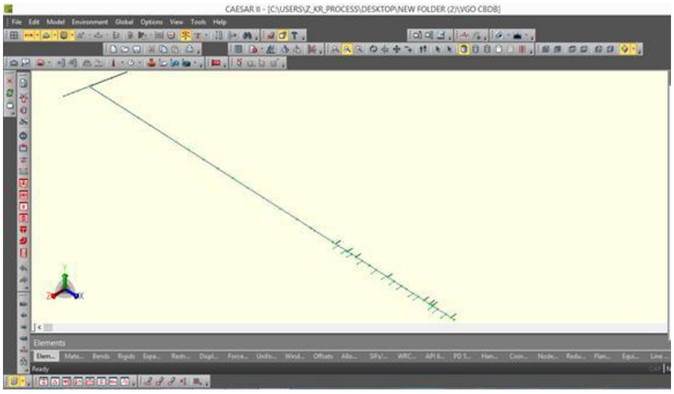

Fig 1: CAESAR II model

3.1 Load cases

To meet these objectives several load cases are required during stress analysis

Operating case: When operation starts working fluid will flow through the piping at a temperature and pressure. So accordingly our operating load cases will be as mentioned below:

L1= W+T1+P1

Alternate sustained load case: Alternate sustained load case depend upon the support configuration of a converged operating condition. This condition addresses systems where non-linear supports are active in some operating conditions and inactive in others

L2 = W+P1

Sustained Case: Sustained loads will exist throughout the plant operation. The sum of weight and pressure are known as sustained loads. So our sustained load case will be as follows:

L3=W+P1 SUS

Expansion load case : The expansion case is a combination case that results from subtracting the sustained case from the operating case. The expansion case represents the change in the piping system due to the effect of temperature, with the presence of other loads. This is important because the restraint status of the operating and sustained cases can be different if there are nonlinear restraints (such as +Y, -Z, any restraint with a gap, etc.), or boundary conditions (friction).Therefore expansion load case :

L4=L1-L3

4 .Numerical modelling of pipe-soil system

The three dimensional (3D) finite element (FE) analyses were carried out using ABAQUS/CAE 6.11 to obtain the pipe and soil stress distribution around the pipe. The pipeline model was aligned with soil model in the centre of soil model's width according to the buried depths of pipeline from the top surface of soil to the crest of pipeline such as 0.5m, 1m, 1.5m, 2m, 2.5m, 3m, 4m, 5m and 6 metres. The soil was represented by 8-noded brick reduced integration elements and the pipe was represented by 8-noded shell reduced integration elements. Constraint between pipeline and soil may cause elastic and plastic deformation during loading process. One constraint called Tie is adopted for simplicity to connect pipeline with soil, that pipeline and soil are fully bonded each other and the interface between pipeline and soil is perfect without defect. In 3D- FE models related to soil and pipeline, two boundary conditions of 3D-FE soil model need to be considered; bottom surface and four beside surfaces of 3D-FE soil model and it is also necessary to consider two boundary conditions of 3D-FE pipeline model; two end surfaces of pipeline and circumferential pipeline surface which comes into contact with soil. Fig 2 shows model dimensions. The appropriate dimensions and the mesh density of the model were to reduce the computational time. The parameters used for FEA analysis is same as shown in table 1 and table 2.the traffic load and internal pressure of pipe applied to this model is 1100 k pa and 414 k pa respectively.

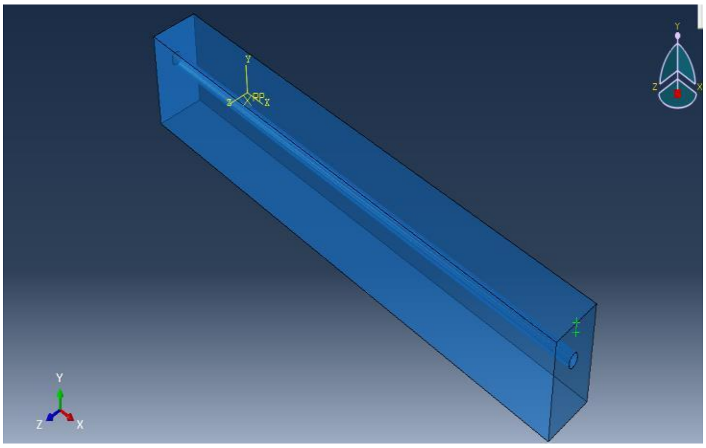

Fig 2: 3D-FE assembled model soil and pipeline with buried depth of 1.5 m

5 : RESULTS AND DISCUSSION

5.1 Experimental result using software CAESARII

FIG 1 Is a piping lay out which is buried with a depth of 1.5m is modeled using Caesar II software .The modeled is then simulated with sustained type of loading system and output parameters such as displacement, stress were analyzed. The output of stress analysis is shown below.

LOADCASE 1 (OPE) W+T1+P1

CODE STRESS CHECK PASSED
Highest Stresses: (N/mm2) LOADCASE 1 (OPE) W+T1+P1
Ratio (%): 0.0 : @Node 12
OPE Stress: 124.2 : Allowable Stress: 0.0
Axial Stress: 2.4 : @Node 95
Bending Stress: 123.8: @Node 12
Torsion Stress: 0.0 : @Node 14
Hoop Stress: 4.3 :@Node 12
Max Stress Intensity: 124.2 : @Node 12

LOADCASE 2 (Alt-SUS) W+P1

CODE STRESS CHECK PASSED
Highest Stresses: (N/mm2) LOADCASE 2 (Alt-SUS) W+P1
Ratio (%): 1.5 @Node 12
Code Stress: 2.1 : Allowable Stress: 137.3
Axial Stress: 2.1 : @Node 12
Bending Stress: 0.0 : @Node 12
Torsion Stress: 0.0 : @Node 14
Hoop Stress: 4.3 : @Node 12
Max Stress Intensity: 4.9: @Node 12

LOADCASE 3 (SUS) W+P1

CODE STRESS CHECK PASSED
Highest Stresses: (N/mm2) LOADCASE 3 (SUS) W+P1
Ratio (%): 1.5 : @Node 12
Code Stress: 2.1 : Allowable Stress: 137.3
Axial Stress: 2.1 : @Node 12
Bending Stress: 0.0 : @Node 12
Torsion Stress: 0.0 : @Node 14
Hoop Stress: 4.3 :@Node 12
Max Stress Intensity: 4.9: @Node 12

LOADCASE 4 (EXP) L4=L1-L3

CODE STRESS CHECK PASSED
Highest Stresses: (N/mm2) LOADCASE 4 (EXP) L4=L1-L3
Ratio (%): 36.3 @Node 12
Code Stress: 124.6 : Allowable Stress: 342.9
Axial Stress: 4.5 : @Node 95
Bending Stress: 123.8: @Node 12
Torsion Stress: 0.0 : @Node 12
Hoop Stress: 0.0 :@Node 12
Max Stress Intensity: 124.6: @Node 12

Based on the pressure and temperature, the stress value were changed at every node in the routing .The ratio of the pipe routing obtained in the analysis was 36.3 at node 12 the ratio of pipe routing is defined as

$$\text{Ratio} = \frac{\text{code stress}}{\text{allowable stress}}$$

The allowable stress is the maximum force per unit area that may safely be applied to a pipe .Allowable stress of a material is an important parameter in the stress analysis of piping system. The working stress (code stress)in the piping system should not exceed an allowable stress of the material for the selected code

and standard .Nodal displacement (DXin, DYin ,DZin,) and (RXdeg,RYdeg,RZdeg) in all three direction.

Code compliance evaluation of both piping system is passed i.e the maximum stress developed in the piping system is less than the allowable stress mentioned by the process piping code ASME B31.1.

5.2 Numerical results using software ABAQUS CAE

Here Analysis is dealt with the purpose of understanding of static behavior regarding buried pipeline under static loads. It is observed that due to static loads, which are operated vertically, caused the pipeline to be deformed as an oval shape and allowed the maximum stress of buried pipeline to occur at both the crest and invert of pipeline. High stress of pipeline, which comes close to the maximum stress of pipeline, was also examined at the spring-line of pipeline. While the maximum stress was generated by compression force caused by static loads acting vertically, the high stress which comes close to the maximum stress was caused by expansion force acting horizontally at spring-line of pipeline. Fig 3 is an isometric view of pipeline stress analysis under static load. .

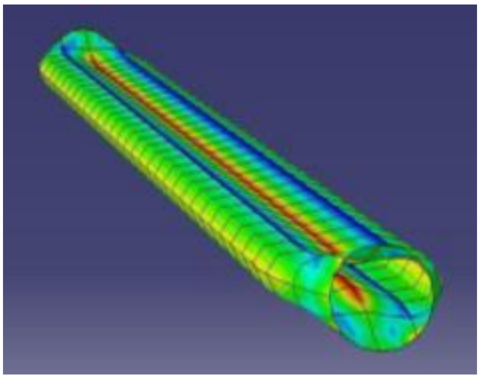

Fig 3: Pipe stress result under static loads

This meant that if there is no enough strength of pipeline for resisting static loads, the buckling will be generated at both the crest and invert of pipeline and the flexible pipeline will be totally collapsed when the bucking reaches critical statement caused by maximum stress of pipeline. Increase of pipeline's maximum stress was generally examined according as the buried depth of pipeline became deep without only the case of general moist cohesive soil as in chart 1. This meant that deep burying of pipeline affects the pipeline to generate high stress and makes the pipeline possible buckled more easily. Therefore, it is unreasonable to make a decision regarding that deeply buried pipeline is the safest for the only consideration of serviceability issue.

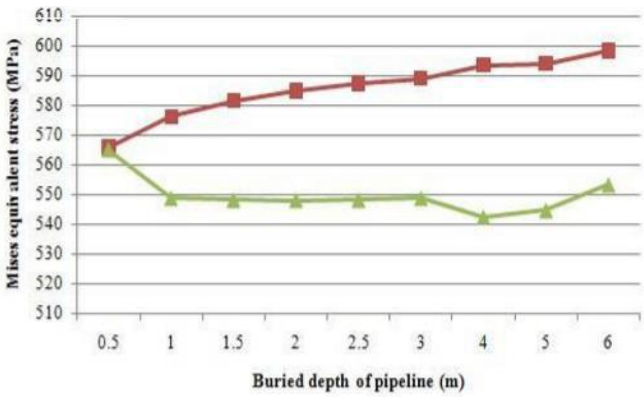

Chart 1 : Maximum stress of pipe under static loads

6. Conclusion

The analytical study of buried piping system is done using the power piping code ASME 31.1 and the piping system is modeled and analyzed using CAESAR II platform. . Modified layouts can be checked with the help of this software for same environmental and operating conditions and optimized layout can be selected. From finite element analysis by using ABAQUS CAE we can experimentally conclude that it is adaptable to consider both settlement and stress of pipeline for making a decision regarding to the safest buried depth. For choosing the exact safest depth is difficult we only can assume the medium buried depth such as 1.5m and 3m is a safe range of buried depth. The use of advanced finite element method software will allow user to achieve understanding of mechanical behavior of their piping system, increasing fatigue life calculation uncertainties allowing proper evaluation of nonstandard fittings. Additionally, the use of these techniques will make it possible for the piping industry to take advantage of the latest developments and trends of the simulation community.

REFERENCES

1. M. Jamuna Rani, K. Ramanathan "Design and Analysis of Piping System with Supports Using CAESAR-II" International Journal of Mechanical, Aerospace, Industrial, Mechatronic and Manufacturing Engineering Vol:10, No:5, 2016
2. ASME B31.1, Process Piping, ASME, New York-2006.
3. IITK-GSDMA Guidelines for SEISMIC DESIGN of BURIED PIPELINES, JULY 2001
4. Howard, A. K. (1977). "Modulus of Soil Reaction Values for Buried Flexible Pipe Design." ASCE, J. of Geotechnical Engineering, V103, GT1, January, pp 33-43.
5. Eugenia Marinou and Valdir de Souza, SPE-179944, —Review of Typical Challenges in Establishing the Integrity Status of Pipeline||, UK, May-2016.
6. Carlos Eduardo A. de C. e Sousa "Piping Flexibility Analysis and the Development of PCS – Pipe Calculation System for Abaqus" ,SIMULIA community conference ,2015
7. Pathmanathan Rajeev, D.j Robert Jayantha kodikara "Stress Analysis of Buried pipes" November 2015
8. E.Shashi Menon, "Piping Calculations Manual", 8th Edition, McGraw-Hill, 2009