International Journal of Advances in Engineering and Management (IJAEM)
Volume 2, Issue 8, pp: 309-317 www.ijaem.net ISSN: 2395-5252

Thermo Mechanical Analysis of Upstream Production Pipes Used In High Temperature Oilwells Using Ansys

Velan Avudaiappan[1] T. Venkatajalapathi[2]
[1,] Senior Engineer, Cameron Manufacturing (India) Pvt. Limited, Coimbatore
[2]Associate Professor, Mechanical Engineering, SNS College of Technology, Affiliated to Anna University, Coimbatore,

---
Date of Submission: 10-10-2020 Date of Acceptance: 30-10-2020

ABSTRACT: This technical paper is mainly focused in developing a methodology to determine the temperature distribution in upstream production pipe assemblies employed in high temperature oil wells, in petroleum industry. The temperature of the fluids flowing inside the pipe is nearly 175^{o}C (448 K) and the pressure acting on the pipes is 70MPa. At this temperature, the pipes are subjected to thermal expansion and this expansion may affect the functionality of the assembly and interference between the pipes may occur, if the pipes expand more than the allowable limits. The allowable deformation of the pipes is 2mm, beyond which the pipes tend to touch each other resulting in malfunctioning. This paper emphasizes in studying the temperature gradient across the pipes for 100 seconds and to determine the expansion of the pipes using Finite Element Analysis Techniques. Thermo-mechanical analysis, widely known as Coupled field analysis methodology using ANSYS is adopted for this study.

KEYWORDS: Coupled Field Analysis, thermal expansion, ANSYS

I. INTRODUCTION

Petroleum or the Oil and Gas industry is one of the world's largest industry sector employing more than hundreds of thousands of workers worldwide. Employees' safety is the top priority for the entire industry. The Petroleum industry has three major areas, the Upstream, Mid-stream and Downstream. Upstream consists of finding the oil well either underground or underwater and drilling them to recover the crude oil. Midstream mainly deals with transportation to the refinery, storage, and processing of oil. Downstream refers to filtering of raw materials from crude oil.

In each of the stages a greater number of steel structures and assembly components are used to drill and transport the crude oil. Some wells may be at very high temperatures. These wells must be drilled very carefully.

Leakage of oil may lead to explosion and severe environmental hazards resulting in loss of life and spoiling nature. There are many governing bodies worldwide to monitor and ensure that the companies adhere to the safety norms very strictly.

Our project is focussed on upstream pipe assemblies subjected to very high temperature of 175^{o}C. Design of upstream pipes require in-depth knowledge of their behaviour under such extreme conditions. The components used in petroleum industry are huge in size demanding very high accuracy and precision in manufacturing. Considering the risk involved in drilling operations, most of the companies prefer a clear understanding of the functional aspects of the components, before installing them in wells for drilling.

Upstream production pipes are a combination of pipes that act as an interface with the oilwell and processing facility. A sectional view of the 3D model of the pipe assembly is shown in Figure 1 below.

International Journal of Advances in Engineering and Management (IJAEM)
Volume 2, Issue 8, pp: 309-317 www.ijaem.net ISSN: 2395-5252

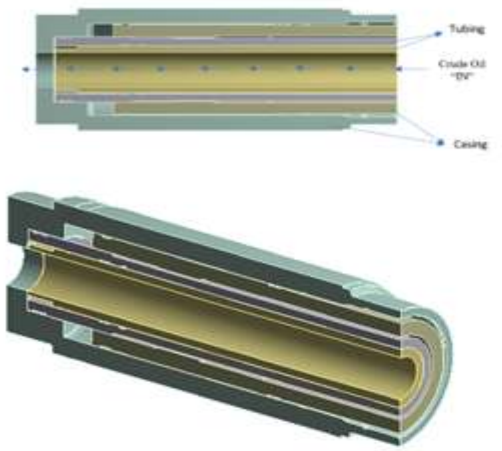


Figure 1: Pipe Assembly

Pipes which are directly in contact with the crude oil is called as "tubing". Pipes which surround the tubing is called as Casing. Heat transfer usually takes place by three modes, namely conduction, convection and radiation. For our pipe assembly, the mode of heat transfer is conduction. Conduction is the transfer of thermal energy in a substance due to difference in temperature. It occurs from a region of higher temperature to a region of lower temperature, and acts to balance temperature differences.

In this paper, modelling and analysis of pipe assembly will be performed using Finite Element Method. The commercial finite element package ANSYS version 2019 R2 is used for this analysis. Ansys-Design Modeler is used for creation of Pipe assembly. In order to reduce the computation time and have better control over the design parameters an Axisymmetric model is used for the analysis. The following assumptions were made in the analysis.

i. Nominal dimensions of the model were used
ii. Thread profiles were simplified
iii. Temperature surrounding the pipe assembly is maintained at 30^0C (303K) with the assumption that the entire heat is lost to atmosphere.

This paper presents the overall methodology for analysing components subjected to both thermal and structural load simultaneously. The effect of thermal loads on the model are critical if the gaps between the assembled parts are very minimum. Thermal expansion may lead to either interference or widen the gap between the components based on the direction of the temperature difference. In either way, the functionality is affected. With the present generation of high-performance computing machines, it is highly feasible for studying the thermal response over a longer period.

The same methodology can be applied to models with "N" number of components and different combination of thermal and structural loads.

II. MODEL DEFINITION AND MATERIAL SELECTION

The 3D solid model is developed and assembled in Design Modeler available with ANSYS. This 3D model is converted to axisymmetric model with negligible thickness as shown in Figure 2.

Figure 2: Axisymmetric model

Axisymmetric model helps the analyst to do iterations very quickly and have a better control over the parameters, which when assigned in 3D models are tedious to monitor. Non-critical items such as screws, plugs are not included in the FEA model.

Material selection is one of the key activities for components exposed to high temperature and pressure. Materials for high temperature applications are selected based on service requirements, mainly the yield strength. In these cases, the resistance of the material to corrosion may not be the primary design consideration. In considering materials, a detailed understanding of the service applications (bearing stresses; if fatigue loading is available; impact or erosion effects; thermal expansion) is needed. Pipe Assemblies must be robust and resilient to the thermal and structural loads and stresses imparted on them, which can include significant temperature changes and thermal gradients for many high-temperature applications. In making a choice, it is necessary to know what materials are available and to what extent they are suited to the specific application. The decision is quite involved, and the choice is significantly affected by the environment and the intended use of the component.

Petroleum industry has a vast classification of material specifications to select for the piping. Carbon and low alloy steels are the most common materials used in construction for equipments, tanks, and piping structures. High-temperature alloys are usually iron-, nickel- or cobalt-based alloys containing chromium, which helps to form a protection envelope against further oxidation. Material grades used in this study are mentioned in table 1. In ANSYS, these properties are entered in Engineering Data Tab in Ansys as shown below. Metric units are used in this analysis.

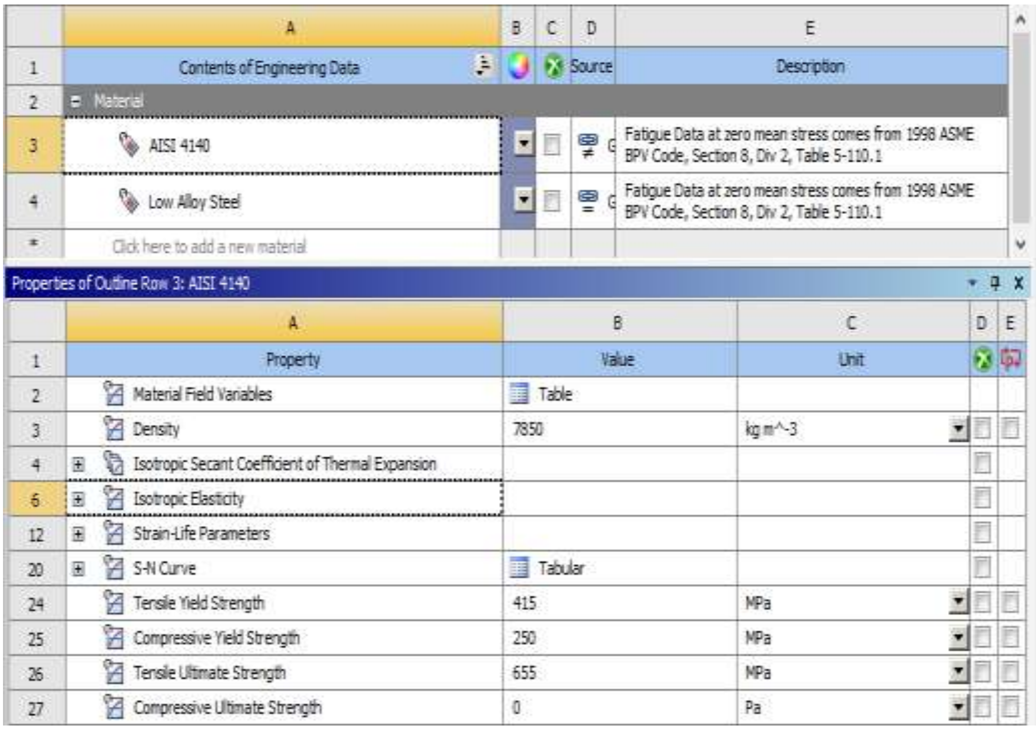


Figure 3: Material Input

S. No	Properties	AISI 4140	Low Alloy Steel
1	Tensile Strength	655 MPa	996 MPa
2	Yield Strength	415 MPa	715MPa
3	Bulk modulus	140 GPa	152 GPa
4	Shear Modulus	80GPa	82 GPa
5	Thermal expansion	12.2 µm/m°C	12.6 µm/m°C
6	Thermal conductivity	42.6 W/mK	46.5 W/mK

Table 1: Material Properties used for the Analysis

International Journal of Advances in Engineering and Management (IJAEM)
Volume 2, Issue 8, pp: 309-317 www.ijaem.net ISSN: 2395-5252

III. ANALYSIS METHODOLOGY

Meshing:

In order to capture the thermal gradient and the corresponding deformation of the model, minimum of 5 divisions are generated across the edges. This gives a fine and conformal mesh. PLANE77 element type is used for the analysis.

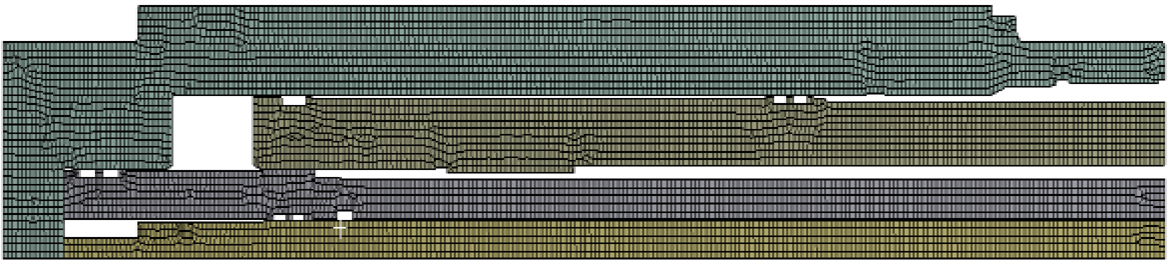


Figure 4: Meshed model

Loads and Boundary conditions:

The outer surface of the casing is maintained at 303K as shown in figure 5.

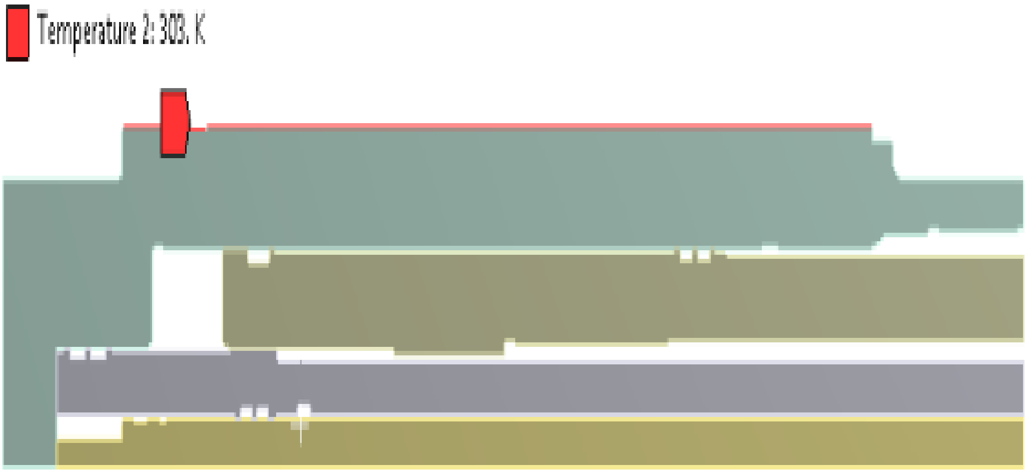


Figure 5: Temperature at the outer surface

The inner surfaces of the tubing are maintained at 448K as shown in figure 6.

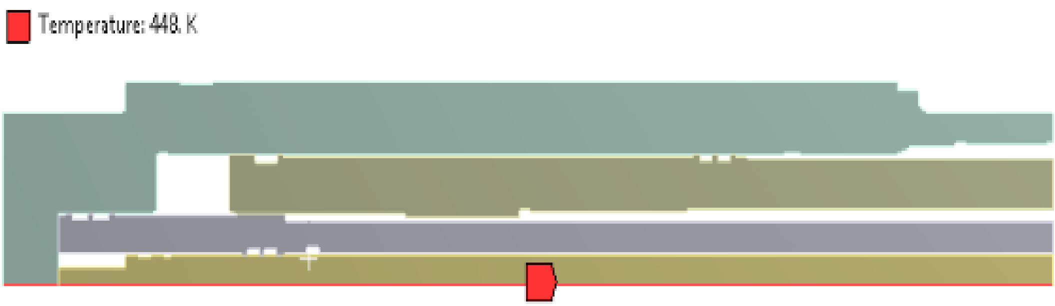


Figure 6: Temperature at the tubing inner surface

Internal Pressure of 70MPa is applied to the inner surfaces of the tubing pipe and the ends are fixed to maintain a model to avoid rigid body motion.

International Journal of Advances in Engineering and Management (IJAEM)
Volume 2, Issue 8, pp: 309-317 www.ijaem.net ISSN: 2395-5252

Figure 7: Pressure at the tubing inner surface

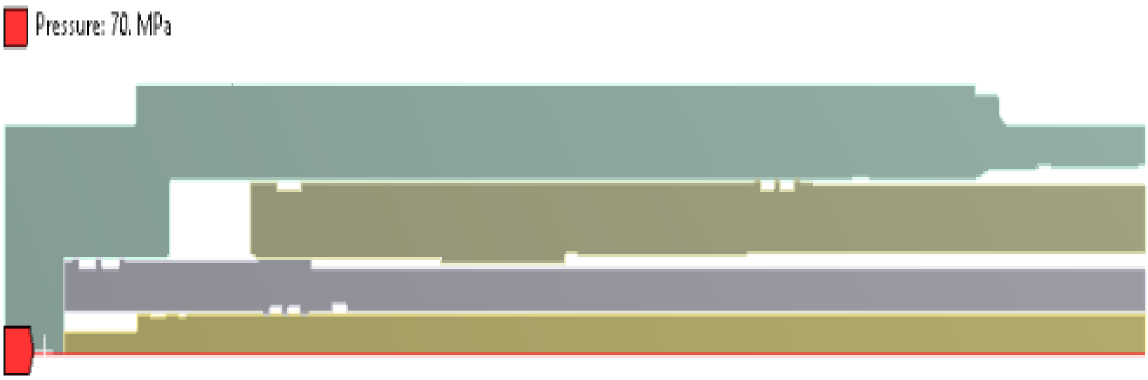


Figure 8: Fixed support at the ends

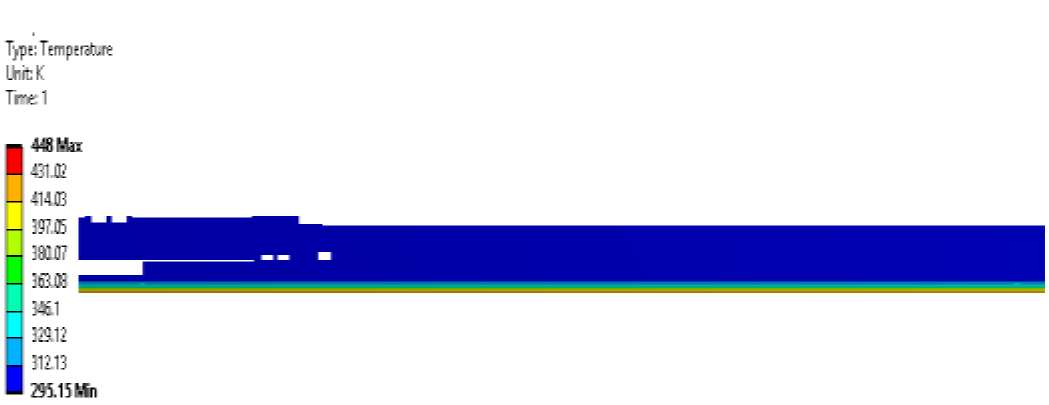
Sequence of analysis in ANSYS:

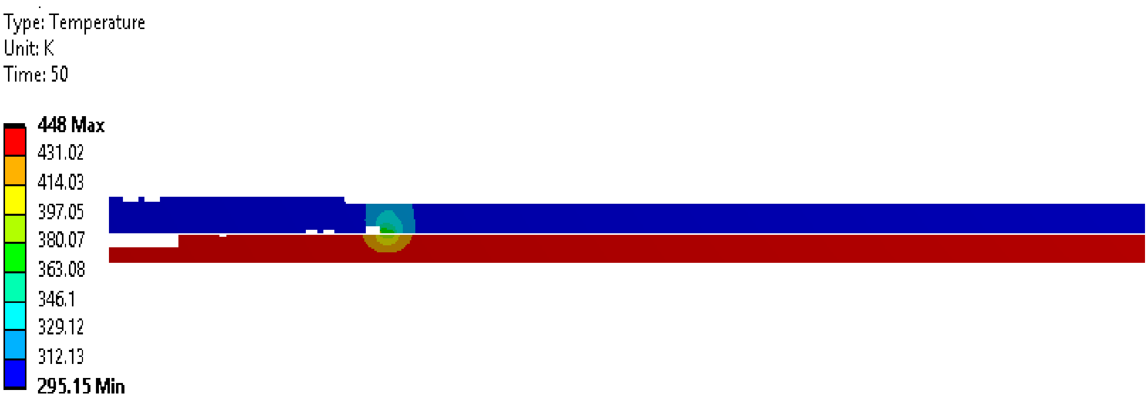
a. Transient Thermal Analysis is performed for the given boundary conditions.
b. The thermal analysis results are imported to the Structural Analysis.
c. The model is solved for the structural boundary conditions i.e. for an internal pressure of 70MPa.
d. Deformation of the model is studied at critical locations.

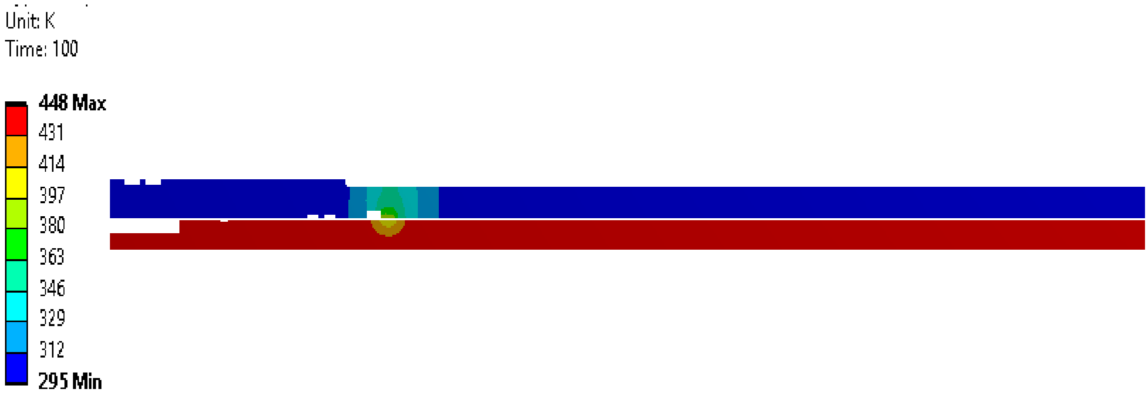
IV. DISCUSSION OF ANALYSIS RESULTS

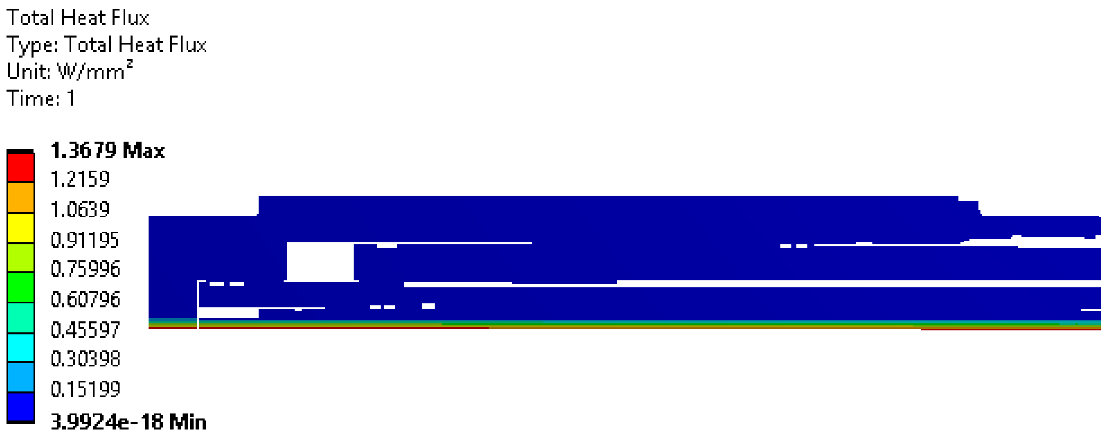
Thermal analysis results:

A transient analysis was performed for 100 seconds. In order to study the thermal gradient across the thickness of the pipes, Temperature distribution and thermal flux at 1 sec, 50 sec and the end of the analysis (100 sec) are plotted as shown below.




Type: Temperature
Unit: K
Time: 50
448 Max
431.02
414.03
397.05
380.07
363.08
346.1
329.12
312.13
295.15 Min


Unit: K
Time: 100
448 Max
431
414
397
380
363
346
329
312
295 Min


Total Heat Flux
Type: Total Heat Flux
Unit: W/mm²
Time: 1
1.3679 Max
1.2159
1.0639
0.91195
0.75996
0.60796
0.45597
0.30398
0.15199
3.9924e-18 Min


Total Heat Flux 2
Type: Total Heat Flux
Unit: W/mm²
Time: 50
0.54114 Max
0.48101
0.42089
0.36076
0.30063
0.24051
0.18038
0.12025
0.060127
1.0584e-17 Min

International Journal of Advances in Engineering and Management (IJAEM)
Volume 2, Issue 8, pp: 309-317 www.ijaem.net ISSN: 2395-5252

Figure 9: Temperature and Flux plots at 1 s, 50 s and 100 sec

Red colour in the legend depicts the maximum temperature and blue colour depicts the minimum temperature. From the above plots we can see that the temperature at the tubing pipes have reached the maximum value of 448K at 50 seconds. The outer casing remains at the ambient temperature. Plotting the variation of temperature with time will help us to visualize the point at which the model reaches steady state.

ANSYS stores the results at all the data points for a transient analysis. Hence it is easy to extract the results and identify the steady state point.

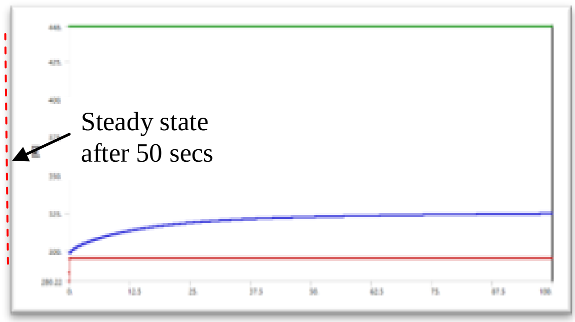


Figure 10: Graph of Time(s) vs Temperature (K)

From the temperature distribution graph, it is inferred that the model reaches steady state at 50 secs and the thermal effect increases with increase in time. This behaviour will have a huge impact on the structural behaviour of the pipe assembly. As these results are mapped to the structural analysis, plotting the deformation at these time points will give us a better understanding of the expansion of pipes.

Three critical locations A, B and C as seen in figure 10, were selected in the pipe assembly. The radial deformation at these locations were obtained from the analysis results and compared with the allowable deformation.

Figure 11: Critical locations

International Journal of Advances in Engineering and Management (IJAEM)
Volume 2, Issue 8, pp: 309-317 www.ijaem.net ISSN: 2395-5252

Structural Analysis Results:
The radial deformation at the critical locations are plotted in Ansys as shown in figure 11.

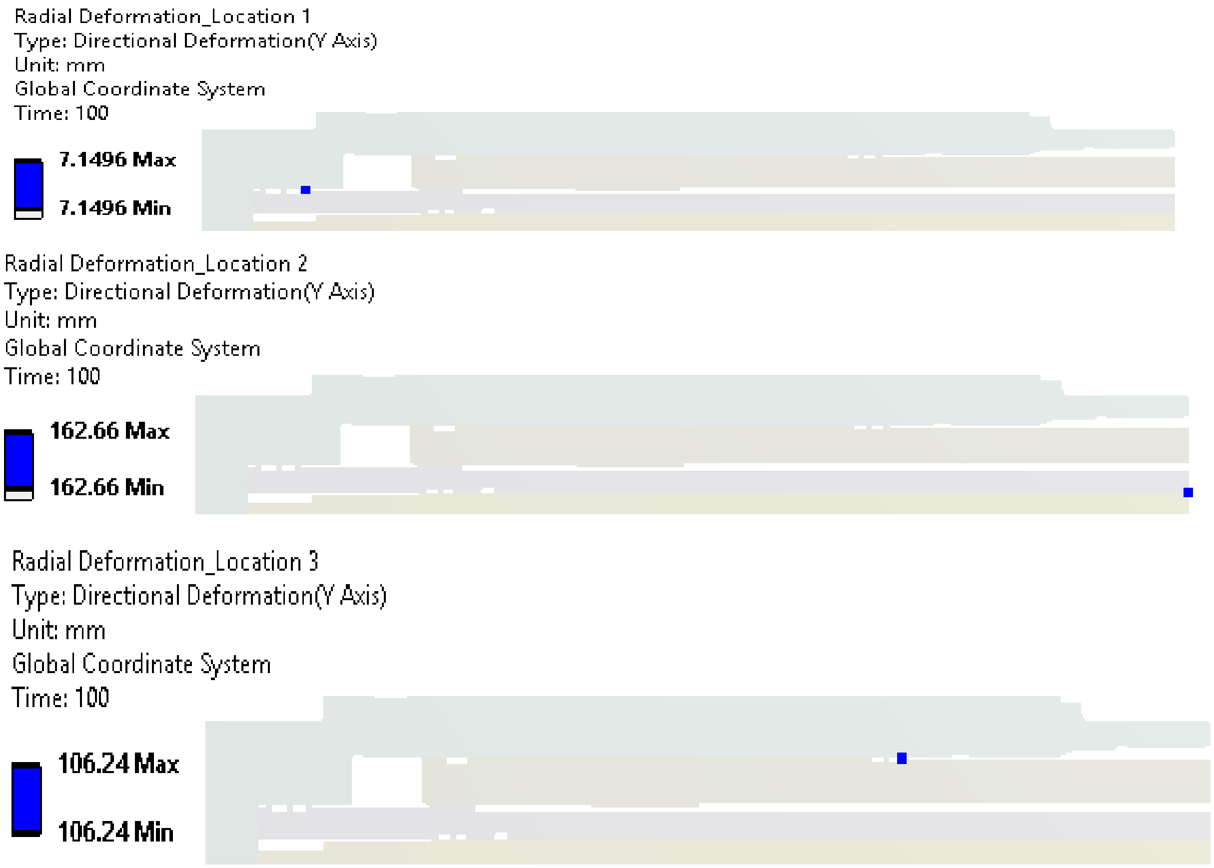


Figure 12: Radial Deformation

Location	Radial deformation from FEA mm	Allowable Radial deformation mm	Acceptable
1	7.15	2	NO
2	162.66	2	NO
3	106.2	2	NO

Table 2: Comparison of Results

V. CONCLUSION

A detailed analysis methodology to verify the deformation of components subjected to simultaneous thermal and structural loads has been developed and studied. From the results of thermal analysis seen in figure 9, as expected, the maximum temperature is reached at the inner surface of the tubing the minimum temperature is observed at the outer surface of the casing. In addition, by comparing the thermal flux results, shows that the heat flux flowing out of the assembly in axial direction is found to be higher than in the radial direction. As the time of fluid flow increases, the thermal flux gets reduced. This indicates that more amount of heat is lost to the surroundings along the length of the pipe assembly.

Deformation in the pipe assemblies as seen in Table 2, are very higher than the allowable deformation. This will be leading to interaction of metal surfaces resulting in malfunctioning of the equipment. Hence from the analysis results it is proved that this pipe assembly is not fit for service at this high temperature and pressure rating. It is recommended to reconsider the material selection criterion and use materials with high yield strength and thermal conductivity.

The same methodology can be adopted for different type of material combinations and

International Journal of Advances in Engineering and Management (IJAEM)
Volume 2, Issue 8, pp: 309-317 www.ijaem.net ISSN: 2395-5252

geometry. This article will serve as a best practise for performing coupled field analysis using ANSYS.

REFERENCES

[1]. Matweb.com for Material properties
[2]. ANSYS Help- Modeling and Meshing Guide.
[3]. Engineering Thermodynamics by P.K. Nag