Journal of Emerging Trends in Engineering and Applied Sciences (JETEAS) 6(5): 309- 314
 (ISSN: 2141-7016)
jeteas.scholarlinkresearch.com

Thermo-Elastic Stress Analysis of the GHARR-1 Vessel during Reactor Operation Using ANSYS 13.0

[1]Justice Darko and [2]NiiAllotey

Ghana Technology University College,
PMB 100, Accra, Ghana
[2]Research Scientist,
Ghana Atomic Energy Commission, AE1, Atomic, Ghana

Corresponding Author: **Justice Darko**

Abstract

A representation of the temperature and thermal stress distribution in the Ghana Research Reactor 1 (GHARR-1) Vessel is investigated. The specialized software package ANSYS is used to provide an engineering perspective on how a real event can be examined by evaluating data from the study. For the analysis, a finite element model of the GHARR-1 vessel is created using ANSYS 13.0 FEA program. A 3-D FEA shell analysis is performed using the sequential coupled thermal stress analysis approach; static structural analysis is implemented in determining the thermal stresses at different time steps. The present work quantified the temperatures and the subsequent thermal stresses in the vessel arising during operation. In general the complex nature of thermal-structural systems and more so due to the composite nature of structural components encountered in engineering in practice makes it difficult to analyze such systems. The emergence of advanced computational power and improvements in computer-aided designs provides understanding to enhanced levels in conjunction with graphic enhanced visualization for these complex systems.

Keywords: ANSYS, Finite Element Analysis (FEA), Ghana Research Reactor 1 (GHARR-1, Static Structural Analysis, Thermal Analysis

INTRODUCTION

Assessing the thermal stresses induced in the reactor vessel as a result of the various thermal loadings during operation is increasingly becoming one of the most important endeavors undertaken by engineers and researchers. By means of the finite element method, the difficulty encountered in performing this task is greatly reduced. In addition, the use of finite element propriety software packages to analyze the thermal stresses in reactor vessels provides a holistic representation of the stresses in the vessel as a result of the thermal loads and further provides a graphic representation of the properties being investigated. The finite element method now widely used in most engineering analysis is used to compute the thermal stresses induced in the GHARR-1 (Ghana Research Reactor -1) vessel as a result of flow of reactor coolant during operation. The analysis is conducted using the ANSYS 13.0 FEA package (2010).

GHARR-1 is a research version of the Prototype Miniature Neutron Source Reactor (P-MNSR) designed and manufactured by China Institute of Atomic Energy (CIAE), Beijing (Akaho et al, 2003). With a nominal power of 30 kW, the reactor is designed for use in universities, hospitals and research institutes, mainly for neutron activation analysis, production of short-lived radioisotopes, education and manpower development.

Ajay (2005) presented a research on thermal stress analysis of multilayered cylindrical molds made up of different materials. Heating and cooling cycles of 10 and 25 seconds are applied to the inside surface, while the outside surface is water cooled. A 2-D (Plane strain) coupled-field analysis is performed using a thermal elastic-plastic model accounting for the elastic, as well as, the plastic deformation with ANSYS. The results of the finite element analysis are validated against a widely accepted mathematical model result and empirical industrial data.

Chapelle and Bathe (1998) presented the fundamental considerations regarding the finite element analysis of shell structures. They reviewed some well-known results regarding the asymptotic behavior of a shell mathematical model. From their study, they concluded that as the thickness becomes small, the shell behavior falls into one of two dramatically different categories; namely, the membrane-dominated and bending-dominated cases. They further stated that the shell geometry and boundary conditions decide into which category the shell structure falls, and a seemingly small change in these conditions can result into a change of category and hence into a dramatically different shell behavior.

Irfan and Chapman (2009) studied the thermal stresses that are generated in radiant tubes. The analytical approach is verified using a finite element model. From the analysis, the axial temperature gradients are found not to be a source of thermal stresses as long as the temperature distribution is linear. Spikes in the axial temperature gradient are a source of high thermal stresses. Symmetric circumferential gradients also generated thermal stresses, which are low as compared to the stress rupture value of the radiant tube. Also, the radial temperature gradients created bi-axial stresses and are a major source of thermal stress in the radiant tube. Finally, it is concluded that a local hot spot generated stresses, which could lead to failure of the tube.

The complex nature of thermal-structural systems makes it intractable to physically analyze such system. Due to the importance attached to thermal stresses, the ability to properly assess the stability and performance of reactor vessels during operation is critical for safe operation. This paper provides an alternative method for validating the structural integrity of the GHARR-1 vessel under actual operating condition.

MATERIALS AND METHODS

The need to assess the thermal stresses in GHARR-1 Vessel during start-up and shut down is very important since these processes might result in significant temperature changes and thereby induce sizable thermal stresses.

Finite Element Model

The GHARR-1 Vessel which was considered in this study is made from aluminum alloy LT21; this belongs to the 3xx.x series of aluminum alloys. The structure was modeled as a thin shell since it satisfied the thin shell criteria ratio$\frac{h}{R} \leq \frac{1}{20}$, where h=thickness from the midsection of the shell and R=radius of curvature from the midsection of the shell.

For the thermal and structural analysis, a 3-D FEA shell analysis was performed using the sequential coupled thermal stress analysis approach. The thermal element SHELL 131 was used for the thermal analysis and then followed by SHELL 181 for the stress analysis. Both elements had a 0.01 m specification for shell thickness. Fig. 1 shows the finite element model for the GHARR-1 Vessel.

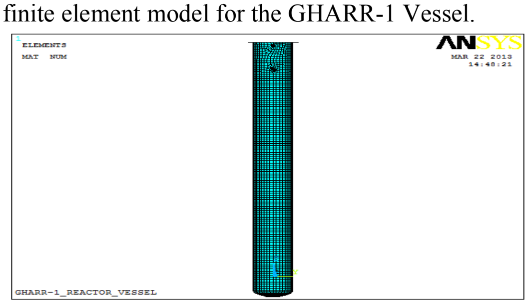


Fig. 1. Finite element model of the GHARR-1 Vessel

Transient Thermal Analysis

The three dimensional heat conduction equation without a heat source in cylindrical coordinate was written as

$$\frac{\partial^2 T}{\partial r^2} + \frac{1}{r}\frac{\partial T}{\partial r} + \frac{\partial^2 T}{\partial z^2} + \frac{1}{r^2}\frac{\partial^2 T}{\partial \theta^2} = \frac{1}{k}\frac{\partial T}{\partial t}$$

This transient problem can be solved by most of the available finite element techniques including weighted residual method. The general concept of the method of weighted residuals seeks an approximate solution of the form

$$y^*(x) = \sum_{i=1}^{n} c_i N_i(x) \qquad (1)$$

wherey^* is the approximate solution expressed as the product of $c_i$unknown, constant parameters to be determined and $N_i(x)$ trial functions. The method of weighted residuals requires that the unknown parameters $c_i$be evaluated such that

$$\int_a^b w_i(x) R(x) dx = 0 \qquad i = 1, n \quad (2)$$

where$w_i(x)$ represents *n*arbitrary weighting functions. Integrating Eq. 2, results in *n* algebraic equations, which can be solved for the *n* values ofc_i. This expresses that the sum (integral) of the weighted residual error over the domain of the problem is zero (Hutton 2004).

From thermal hydraulics studies and previous works done, it was concluded that the thermal transient process in the coolant of the vessel took approximately 20 seconds, and it was this time period that the analysis looked at. The core region of the vessel, also described as the beltline configuration was expected to undergo significant temperature changes while the upper section of the vessel assumes an approximately constant temperature throughout the analysis. Figure 2 shows the longitudinal section of the vessel, with the dimensions (millimeters) and region where the temperature change was expected to be significant.Figure 3 shows the variation of coolant temperature with time and axial distance in the inner part of the vessel as a result of natural convection.

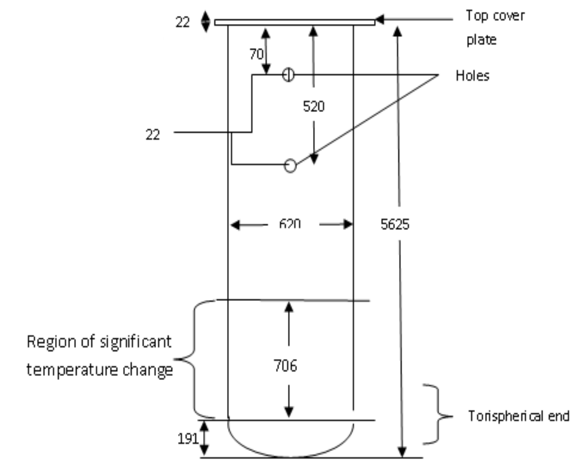


Fig. 2.Longitudinal section of GHARR-1 showing region of significant temperature change

The convection heat transfer was related to heat flux by Newton's law of cooling:

$$q/A = h(T_S - T_f) \qquad (3)$$

where q/A is heat flux out of the face, h is the heat transfer coefficient T_S is the temperature on the face and T_f is the bulk fluid temperature.

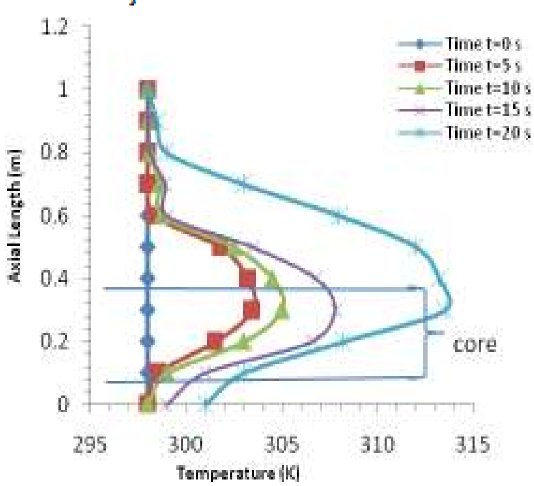


Fig. 3.Convective boundary condition developed in ANSYS

The temperature distribution for the model was determined at a time step of 0.1 s for 20 s.

This generates a total of 200 sub steps for the thermal analysis. The model was then solved again but this time with a step size of 10 s for 1000 s to determine the steady state temperature profile.

Thermo-Elastic Stress Analysis

The generalized three dimensional stress-strain equation can be expressed in matrix form as

$$\{\sigma\} = [D]\{\varepsilon^{el}\} \qquad (4)$$

Where $\{\sigma\}, [D]$ *and* $\{\varepsilon^{el}\}$ are the stress vector, material property matrix and strain vector respectively.

Considering the effects of temperature on the strain, then

$$\{\varepsilon^{el}\} = \{\varepsilon\} - \{\varepsilon^{th}\}$$

$$\{\varepsilon\} = total\ strain\ vector = [\varepsilon_r \varepsilon_\theta\ \varepsilon_z\ \gamma_{rz}]^T$$

and

$$\{\varepsilon^{th}\} = thermal\ strain\ vector = \alpha\Delta T[1\ \ 1\ \ 1\ \ 0]^T$$

$$\Delta T = T - T_{ref}$$

Where α, T and T_{ref} are the coefficient of thermal expansion, current temperature at the point in question and the reference temperature respectively.

The displacement due the thermal load was solved as

$$\{\delta\} = [K]^{-1}\{f\}_{th} \qquad (5)$$

Where $\{\delta\}$ is the displacement vector, $[K]^{-1}$ is the stiffnes matrix and $\{f\}_{th}$ the thermal load vector.

From the above equation the thermal stress was deduced from Eq. 4 as

$$\{\sigma\} = [D]\Big\{\{\varepsilon\} - \{\varepsilon^{th}\}\Big\}$$

This was written as

$$\{\sigma\} = [D][B]\{\delta\} - [D]\alpha\{\Delta T\} \qquad (6)$$

The ANSYS program performed the load transfer coupled analysis using the sequential coupling method. The temperature values obtained from the thermal analysis was imposed on the structural finite element model with a reference temperature of 298 K set for the analysis. In the thermal stress analysis, a static analysis approach was used were selected temperature values from the 1 to 1000 s solutions from the thermal analysis was imposed one at a time to solve for the stresses.

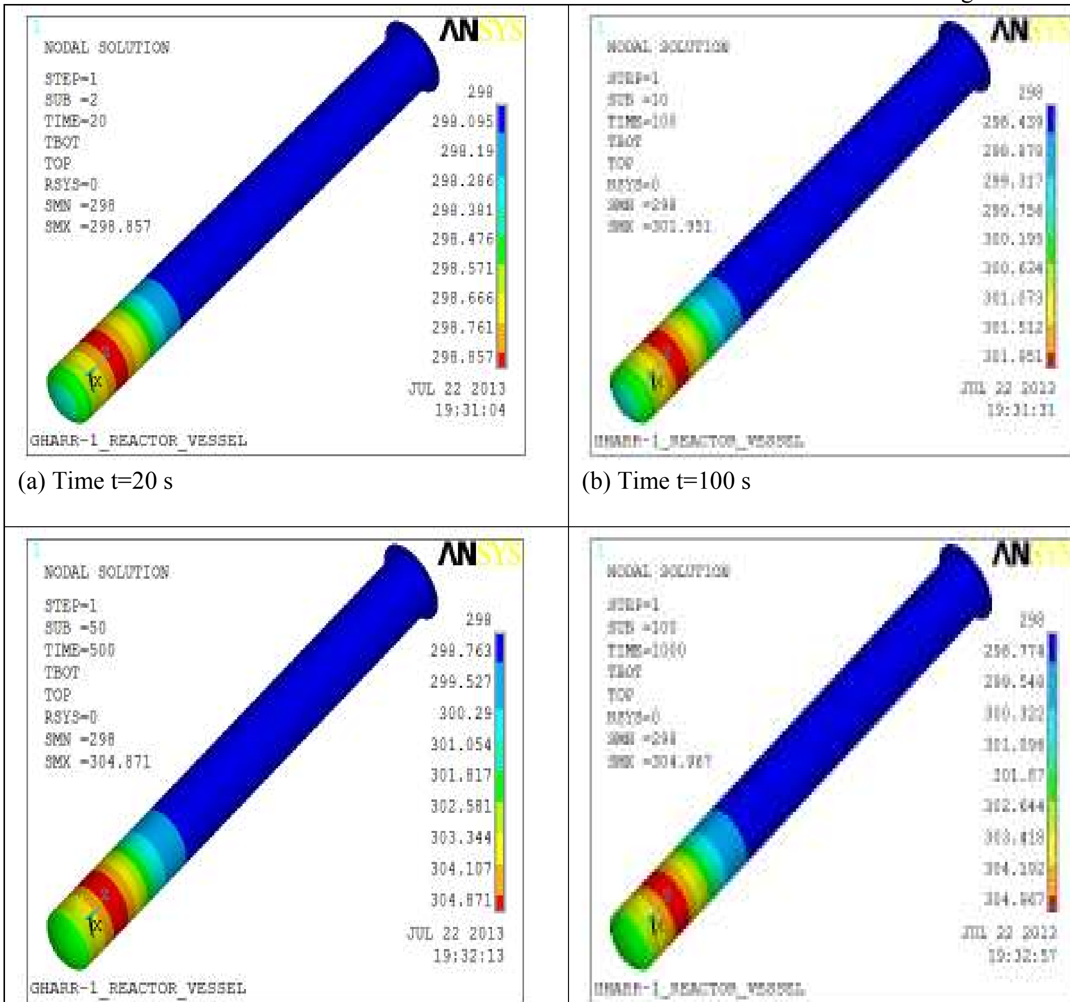
RESULTS AND DISCUSSION

The temperature distribution of the vessel at the twentieth, hundred, five hundred and one thousand seconds are represented in the contour plot of Fig. 4.

Fig. 4 (a) shows the temperature distribution of the vessel at the twentieth second of the analysis. This represents the end of the transient nature of the coolant temperature. Though the highest temperature in the coolant at this time is 314 K, the highest temperature which occurred in the lower section (beltline region) of the vessel was determined to be 298.9 K.

The temperature distribution of the vessel at time t= 100 seconds was presented in Fig. 4. (b). At this time, the maximum temperature of the vessel is found to be 301.94 K. The distribution pattern is expected to be the same, with the temperature increasing with each time step until the steady state profile is attained. This is as a result of the fact that the reactor coolant is imposing a constant temperature profile after the twentieth second. The maximum temperature of the vessel increased from 301.95 K to 304.87 K in the five hundredth second and this is shown in Fig. 4. (c). This indicated that more heat is still being transferred to the vessel and as such had not yet attained steady state.

From the five hundred to the thousand second, the temperature change is recorded as 0.09 K and this signaled the attainment of the steady state point. Fig. 4. (d) presents the temperature distribution at steady state with the maximum value recorded as 304.96 K. After this time, no significant temperature change was recorded in the computed temperature profile. In all the analysis presented, the minimum temperature is recorded at the upper section of the vessel as expected with the maximum located in the lower section of the vessel around the core region.



(a) Time t=20 s

(b) Time t=100 s

(c) Time t=500 s

(d) Time t=1000 s

Fig. 4. Plot of temperature distribution for different time-steps

STRESS RESULTS

To depict the dependence of stress on time, two elements are selected. The first element, element 1499 is located at the beltline region of the lower section of the vessel and this is done to probe the stress variation at a point which is expected to undergo significant temperature changes. The second element, element 2258 is located at a point 0.1 m above the lower section of the vessel and this is done to provide information on how stress is varying at a region were the change in temperature is expected to be insignificant.

Figures 5 and 6 are graphs of the various stress components plotted for the selected elements. In general, the radial and circumferential stresses were low as compared to the axial stresses.

Fig. 5 presents a plot for the radial, circumferential and axial stress variation with time for element 1499.

The maximum radial, circumferential and axial stresses were obtained as -2.79 MPa, -1.48 MPa and -11.51 MPa respectively.

The plots of radial, circumferential and axial stress variation with time for element 2258 is shown in Fig. 6. This plot also followed the trend observed with element 1499 but with lower values of stresses as compared with element 1499. Element 2258 was located in a region which had lower temperature values as compared with element 1499 and as such accounted for the low stress values.

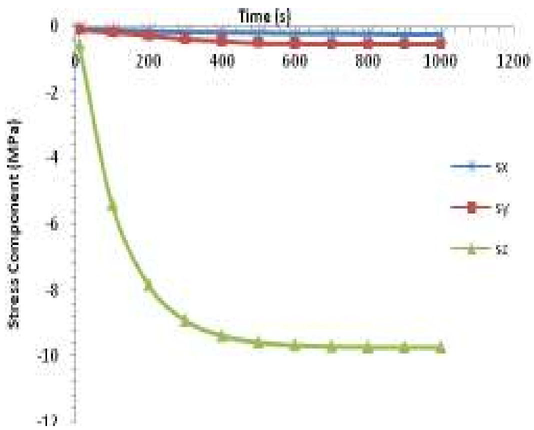


Fig. 5. Plot of thermal stress values with time for element 2258

The values -0.25 MPa, -0.52 MPa and -9.72 MPa are obtained as the maximum stress values for the radial, circumferential and axial stress components respectively.

Effective Stress Distribution Analysis at Steady State

The effective stress according to von Mises theory (Boley 1997) is given by

$$\sigma_v = [\sigma_\theta^2 + \sigma_r^2 + \sigma_z^2 - (\sigma_\theta\sigma_r + \sigma_\theta\sigma_z + \sigma_r\sigma_z)]^{1/2}$$

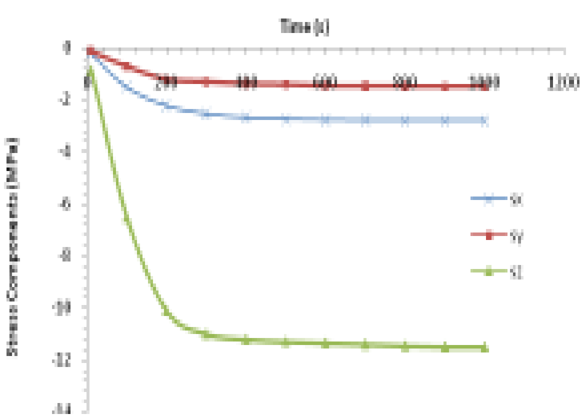


Fig. 6. Plot of thermal stress values with time for element 1499

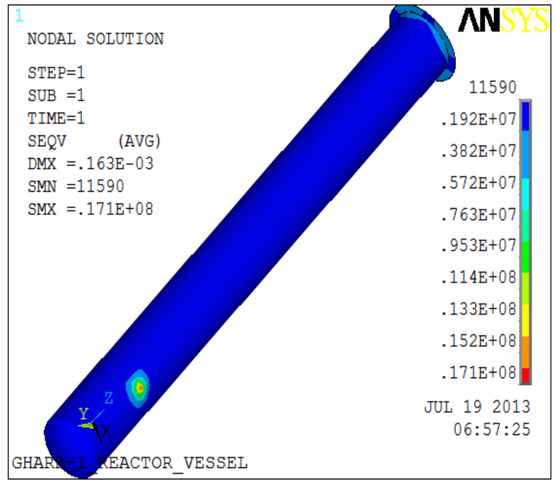


Fig. 7.Effective stress distributions at steady state

These stresses provide a means of assessing whether the induced stresses pose any danger to the material. Fig. 7 is the effective stress distribution obtained at steady state time of 1000 seconds. The dominant effective stress value was found to be within the range of 0.012 MPa to 1.92 MPa; the maximum effective stress at stress concentration regions was determined to be 17.1 MPa. All these values are considerably lower than the material yield stress of 480 MPa.

CONCLUSION

The thermo elastic stress profile of the GHARR-1 vessel during reactor operation is deduced and analysed in this paper. Using the finite element analysis software ANSYS 13.0., a holistic analysis of the GHARR-1 vessel is presented.

The steady state profile of the vessel is determined at about 1000 seconds or 17 minutes. This phase is of interest since this provides information on whether the vessel stresses were within acceptable range during the daily 4 or more hour operation period. The maximum temperature and effective stress value obtained is 304.96 K and 17.1 MPa respectively and these values are below the melting temperature and yield stress value for the vessel material. The stresses determined in all the analysis are lower than the yield stress of the material and it is therefore concluded that the reactor vessel is safe under normal reactor operation.

Also based on the results presented in this paper, one can conclude that a careful use of the finite element software ANSYS, can provide a holistic perspective in analyzing engineering problems.

REFERENCES

Akaho E. H. K, Maakuu B. T, Anim-Sampong S, Emi-Reynolds G, Boadu H. O, Osae E. K, Akoto-Bamford S, Dodoo-Amoo D. N. A. (2003), *GHARR-1 Safety Analysis Report*, GAEC, Kwabenya, Ghana.

Amuasi J. H, Schandorf C, Yeboah J. (2005), *Safety of Ghana Research Reactor (GHARR-1)*, Kwabenya, Ghana

Ansys Inc. (2010). *Ansys Release 13.0 documentation*, Pennsylvania.

Boley B.A and Weiner J. H. (1997), *Theory of Thermal Stresses*, Dover, second Edition.

Chapelle D. and Bathe K. J. (1998), Fundamental considerations for the finite element analysis of shell structures, *Computers and Structures*, Vol. 66, No. 1, pp. 19-36.

Hutton D. V. (2004), *Fundamentals of Finite Element Analysis*, McGraw-Hill Company, New York, USA, pp.1, 132, 202

Irfan M. A and Chapman W. (2009), Thermal stresses in radiant tubes due to axial, circumferential and radial temperature distributions, *Applied Thermal Engineering*, Vol. 29 pp.1913-1920.

Knop N. M. (2009), *Thermal analysis of a fireplace using ANSYS*, Msc Thesis, Iowa State University, Ames, Iowa, USA

Roy K. A. (2005), *Finite Element Analysis and Life Estimation of a Permanent Cylindrical Mold with ANSYS and Coffin-Mansion Approach,* Msc Thesis, Auburn University, Auburn, Alabama, USA.

Ventsel E and Krauthammer T. (2001), Thin Plates and Shells, Theory and Application, Marcel Dekker, New York, p. 300.