STATIC, FATIGUE AND CREEP ANALYSIS OF BOILER SHELL WITH CIRCUMFERENTIAL RIVETED JOINT

Capstone Project-II Report submitted

Partial fulfillment for the award of the

Degree of

BACHELOR OF TECHNOLOGY

Submitted by

TAFHIM EQBAL

(18021011958)

VAIBHAV GUPTA

(18021011970)

IN

MECHANICAL ENGINEERING

DEPARTMENT OF MECHANICAL ENGINEERING

Under the Supervision of

Dr. SUDHIR KUMAR SINGH

GALGOTIAS UNIVERSITY

(Established under Galgotias University Uttar Pradesh Act No. 14 of 2011)

MAY-2022

DEPARTMENT OF MECHANICAL ENGINEERING

BONAFIDE CERTIFICATE

This is certified that this project report on **"STATIC, FATIGUE AND CREEP ANALYSIS OF BOILER SHELL WITH CIRCUMFERENTIAL RIVETED JOINT"** is the bonafide work of **"Tafhim Eqbal (18021011958), Vaibhav Gupta (18021011970)"** who carried out the project work under my supervision.

SIGNATURE OF HOD

SIGNATURE OF SUPERVISOR

APPROVAL SHEET

This project report entitled **“STATIC, FATIGUE AND CREEP ANALYSIS OF BOILER SHELL WITH CIRCUMFERENTIAL RIVETED JOINT”** by **Tafhim Eqbal-18021011958, Vaibhav Gupta-18021011970** is approved for the Degree of Bachelor of Technology in Mechanical Engineering.

Examiners

____

____

____

Supervisor

____

HOD

____

Date: _____________________

Place: _____________________

Statement of Project Report Preparation

1. Project report title: **“**STATIC, FATIGUE AND CREEP ANALYSIS OF BOILER SHELL WITH CIRCUMFERENTIAL RIVETED JOINT”**.**
2. Degree for which the report is submitted: BACHELOR DEGREE OF TECHNOLOGY.
3. Project Supervisor was referred to for preparing the report.
4. Specifications regarding thesis format have been closely followed.
5. The contents of the thesis have been organized based on the guidelines.
6. The report has been prepared without resorting to plagiarism.
7. All sources used have been cited appropriately.
8. The report has not been submitted elsewhere for a degree.

Name:(Tafhim Eqbal) (Signature of the student)

Roll No: (18021011958)

Statement of Project Report Preparation

1. Project report title: **“**STATIC, FATIGUE AND CREEP ANALYSIS OF BOILER SHELL WITH CIRCUMFERENTIAL RIVETED JOINT”**..**
2. Degree for which the report is submitted: BACHELOR DEGREE OF TECHNOLOGY.
3. Project Supervisor was referred to for preparing the report.
4. Specifications regarding thesis format have been closely followed.
5. The contents of the thesis have been organized based on the guidelines.
6. The report has been prepared without resorting to plagiarism.
7. All sources used have been cited appropriately.
8. The report has not been submitted elsewhere for a degree.

Name:(Vaibhav Gupta) (Signature of the student)

Roll No: (18021011970)

ACKNOWLEDGEMENT

I have taken efforts in this project. However, it would not have been possible without the kind support and help of many individuals and organizations. I would like to extend my sincere thanks to all of them.

I am highly indebted to **Dr. P.K.S. NAIN (HOD, DOME) & Dr. SUDHIR KUMAR SINGH (PROFESSOR, DOME)** for their guidance and constant supervision as well as for providing necessary information regarding the project & also for their support in completing the project.

I would like to express my gratitude towards member of **GALGOTIAS UNIVERSITY** for their kind co-operation and encouragement which help me in completion of this project.

My thanks and appreciations also go to my colleague in developing the project and people who have willingly helped me out with their abilities.

Tafhim Eqbal (18021011958)
Vaibhav Gupta (180201011970)
(Department of Mechanical engineering)

ABSTRACT

Boilers and pressure vessels with riveted joints are used to contain pressurized fluids and are subjected to complex loads under static, dynamic and creep situations. Pressure vessel failure happens both circumferentially and longitudinally. Circumferential riveted joints are critical to the design of pressure vessels because the circumferential stress is double the longitudinal stress. Boiler shell with circumferential riveted joints is studied at pressures between 2.5 MPa and 5 MPa for structural analysis. Boiler shell with circumferential riveted joints is studied at pressures 2.5 MPa and varying temperature (300 °C, 400 °C, and 450 °C) for creep analysis. The riveted joint was created with SolidWorks software and Ansys software. The proposed joint is investigated to examine how structural steel, titanium alloy, and nickel-cobalt-chromium alloy affect the vessel's performance. The results are displayed, and a comparison is made in order to determine which material is the best. Static results indicate that Boiler shell joints of nickel-cobalt-chromium alloy have a smaller total deformation (0.067mm) and lower Von-misses stress (63.37MPa) than structural steel and titanium alloy at an internal pressure (2.5 MPa). The Maximum shear stress of titanium alloy (30.63 MPa) shows better result as compared to Structural steel (33.52 MPa) and Nickel-Cobalt-Chromium alloy (34.11 MPa). Data for fatigue life, damage, safety factor and sensitivity for candidate materials are provided by Ansys software. The findings demonstrate that the boiler shell with circumferential riveted joint of Titanium alloy has a good fatigue life, low fatigue damage, and a high safety factor at high internal pressure when compared to the circumferential riveted joint of structural steel, Nickel-cobalt-chromium alloy. The final result shows that selected materials will survive and function well, while titanium alloy and nickel-cobalt-chromium alloy surpass structural steel. From the Creep analysis equivalent creep strain of structural steel at constant load (2.5MPa) and varying temperature (300 °C, 400 °C and 450 °C) are 0.0016895, 7.2834e-6, and 6.2525e-5 respectively. The finding demonstrates that the best load range for greater creep life of structural steel was shown to be 2.5 MPa, and the optimum temperature range was discovered to be 450^0 C. The final result demonstrates that as the temperature is gradually increased to its maximum limit, the creep strain rate of the specified material (structural steel) increases or decreases.

Table of Content

List of Abbreviations

1	FEA	Finite Element Analysis
2	FEM	Finite Element Method
3	ASME	American Society of Mechanical Engineers
4	IBR	Indian Boiler Regulations
5	MPa	Mega Pascal

List of Figures

List of Tables

CHAPTER -1

INTRODUCTION

Boiler and pressure vessels are used in steam generation plants and industrial plants. Pressure vessels, such as boilers, have a cylindrical form and can bear high internal pressure and temperature. Boiler design depends on capacity and working condition of fluid. Boilers contain fluids subjected to high internal pressures, temperature and a complex loading that can be static or dynamic or fatigue. The design of pressure vessels in terms of the shape, material used, chemical composition, environmental conditions of vessels have a distinct impact on the vessel’s performance. The fluid being kept inside the vessel may change state as in a steam boiler. Boiler should be constructed with extreme caution in accordance with ASME or IBR rules and specifications. Rupture of the pressure vessels implies an explosion, which may result in loss of life or property.

The estimated service life of the circumferential riveted joint of the boiler under creep condition is one of the primary considerations in the high-temperature and high-pressure design of these boilers. Creep failure is one of the most common failure modes of a boiler's circumferential riveted joint. Creep is a time-dependent inelastic deformation that occurs at high temperatures and under mechanical stress. The creep phenomenon is one of the most important destructive elements that may be impacted by structural geometry, boundary conditions, and material qualities. High-temperature components, such as boiler joints, frequently fail due to a creep or stress-rupture process. The primary reason may not be increased temperatures, since fuel-ash corrosion or erosion may diminish wall thickness, causing creep and creep failures of boiler joints to occur sooner than predicted. The rupture of the pressure vessels suggests an explosion, which may result in death or property destruction. According to the Economics Time News Report [1], six persons were murdered and 11 were injured when the boiler at a food processing facility burst on Sunday morning, December 26, 2021, in Bihar's Muzaffarpur district.

A boiler's geometrical variables are length and diameter. The length of the boiler may be raised by circumferentially linking the two shells, and the diameter of the boiler can be increased by longitudinally bending the number of sheets using riveted joints. A thin cylindrical pressure vessel fails in two ways. It may fail circumferentially or longitudinally. Circumferential riveted joints are considered in pressure vessel design because the circumferential or hoop stress is double the longitudinal stress. As a result, circumferential joints lose strength while longitudinal joints gain strength [2]. To avoid operational breakdowns, it is critical to foresee the possibility of unintentionally critical boiler zones, such as circumferential riveted joints.

CHAPTER -2

LITERATURE REVIEW

Boiler shells with riveted joints were examined by Pathan et al. [3]. The FEM was used to do a stress analysis on the inside surface of the boiler shell. The author determined that a boiler shell composed of structural steel with riveted seams was much safer than a boiler shell constructed of an aluminum alloy. The boiler shell with riveted joints was analysed using FEM by Singh et al [4]. They used structural steel and aluminium alloy to design and analyse a boiler shell with riveted joints using Ansys software. Under the same operating conditions, the author found that Structural steel boiler shell with riveted connections is safer than aluminum alloy boiler shell. Bhansali et al. [5] investigated the numerical modelling and stress analysis of several boiler shell materials. For commonly used materials in boiler shells, the results and conclusions were derived using FEA software such as Ansys. Titanium Alloy was slightly safer than the other two materials in terms of static stress conditions such as equivalent stress, shear stress, and Von-Mises stress. On the basis of strength, availability, safety factor, and cost, Gopal and colleagues [6] evaluated the performance of AISI 4130, stainless steel, and aluminium alloy for riveted lap joints. The author's research and survey data showed that 45C8 carbon steel was best suited for the production of rivets because of higher FOS.

Analytical and experimental methods were used to study the riveted and bolted joint by Kumar et al [7]. They used two different materials for bolted and riveted joint. Both techniques were subjected to stress and tension testing, which were then compared to single-lap riveted and bolted connections. When it comes to tension and stress levels, the authors' tests reveal that rivet joints outperform bolted ones. An investigation by Shankar et al. [8] looked on leak-proof adhesive applications using single lap riveted connectors. Finite element analysis was used to analyse the work. Authors result showed that the strength of riveted bonded joints was better to that of riveted joints. The tool developed by the Abid et al [9] automates the design study of rivets for boiler shells. The goal of creating the software was to offer a way for efficiently using general-purpose programmes in design analysis. Kondayya [10] analysed a fire tube boiler for static and thermal loading. CATIA was used to develop a geometric model of the boiler shell while the structural and thermal analysis were carried out using Ansys software. A researcher compered the structural and thermal results with ASME results and found that the given boiler was safe to use under the present loading conditions.

The study of Balbudhe et al. [11] focuses on stress and fracture analysis of riveted lap joints under both residual stress and external tensile loading. Modelling of the joint was done by CATIA V 16.0 and analysed by Ansys 14. They compared the FEM result with the analytical result and found that each result was close to the other. Research work of More et al. [12] deals with the analysis of a single lap joint under tensile load and the stress distribution in the joint components under various design parameters. For the stress distribution characteristics of a single lap joint, the Ansys FEA tool was employed. The results were interpreted in terms of displacement and frequency. Stresses operating on the boiler drum wall during start-up were computed by Bohdan et al [13] using the Finite Element Method. During the steam boiler's operating cycle, the researchers measured the highest circumferential, axial, and Von Misses stresses. Dwivedi et al [14] predicted the bursting pressure level in pressure vessels using finite element analysis. The primary objective of the research was to suggest several finite element methods for estimating the burst strength of pressure vessels. Ansys Workbench was used by Koksal et al [15] to conduct fatigue analysis on a notched cantilever beam. Author discovered that the largest damage, equivalent alternating stress, and lowest factor of safety take place near the notch's tip.

The purpose of Drastiawati et al [16] research was to determine the reason of the fracture of the reheater tube. Tensile and impact tests on the failure and replacement tubes were used in the mechanical analysis. The author found that material reheater tube failure happens as a result of a combination of stress and toughness. Using SolidWorks and Ansys software, Kumar et al. [17] created a 3D model of a pressure vessel with multiple stiffeners. Among the stiffener models tested, the rectangular stiffener model exhibited the best fatigue characteristics. Shankar et al. [18] studied the structural analysis of the aircraft wing using Ansys software and CATIA was used to create a 3D model. The result revealed that the highest stress was found near the wing tip, which was lower than the material's yield strength. Shrivastava et al. [19] used SolidWorks to model a pressure vessel. Ansys software was used to analyze thermal, fatigue, and creep. Stress increased with higher temperature. An increase in the effect of fatigue failure at low temperatures. Fajri et al. [20] used Ansys Workbench to analyze total deformation, fatigue life, and fatigue damage on a notched cantilever beam. A significant amount of stress was found to be concentrated in the notch, causing a short fatigue life and a poor safety factor.

Drinovska et al. [22] investigated simply supported beams using Ansys software. The three-calculation technique for creep analysis on the simply supported beam was performed using FEA. The author discovered that all three calculating approaches (step by step method, transient analysis, and static analysis) performed admirably. Katerina et al. [23] Determine the creep effect's influence on the beam's strain. Ansys and some other software were used to investigate the deflection and modelling of the creep characteristic in this investigation. In this article, the software solution and experimental observation are compared, and it is discovered that both outcomes are near to one other. Tran et al. [24] investigated the creep model of a concrete beam. The study was carried out using Ansys software and based on the notion of viscoelasticity via Prony's series in finite element analysis. The results reveal that the FEM-derived beam model fits the analytical method's results rather well.

After completing thermomechanical and creep experiments on boiler tubes, Nguyen et al. [25] offer an approach for boiler tube design optimization depending on cost. Super 304H austenitic stainless steel and nimonic alloy 80A, a Ni-based alloy, were employed in this investigation. The author discovered the employed material's thermal efficiency and creep lifespan. The findings also revealed that determining the creep lifespan is critical and should be factored into the optimization process. The research of Ashcroft et al. [26]. It focuses on fatigue and creep, as well as the environmental consequences of structural adhesively bonded joints, and discusses the primary methodologies used to study these phenomena. The author addresses how creep and fatigue may be accounted for in industrial design.

Pandey et al. [27] use XFEM to simulate elastoplastic creep behaviour using Finite Element Method to model damage accumulation mechanics. The creep crack model and explicitly integrated time approach are used to predict Various materials' creep strain and damage factors at various temperatures (550, 600, and 650 0C). This study compares experimental data and finite element solutions to see if the extended finite element technique results are valid. In this study, Liu et al. [28] presented a computational technique for evaluating the life of superalloy turbine blades depending on the Lemaitre-Chaboche creep deformation model. The creep damage impact of the turbine blade is considered using Ansys APDL software. The author discovered that the turbine blade life prediction approach is practical, and the study is decided by creep fracture instead of creep deformation. Scattina et al. [29]

investigated creep in carbon fiber reinforced structure under high pressures of bolt connection between composites and aluminium plates. Furthermore, this research work demonstrates that to reduce the likelihood of joint tightening reduction.

To evaluate the degradation process, Fujimitsu [30] researched creep failure using a welded joint and performed creep damage identification studies. The results demonstrate that the creep analysis of welded joints made of Mod.9Cr-1Mo steel is lowered by 30–40% against the base material in the range of temperature of 575–600 °C. In this article, two different materials of P91 steel are simulated for various strength using the finite element method and the creep analysis of uniaxial and grooved bar was performed. The steels were tested at 650°C, and 625°C temperature. Hyde et al. [31] employed single phase parameter and third phase parameter creep deformation constitutive models in their analysis. The author was successfully got the result for creep deformation of P91 steel. Ayyanar et al. [32] investigate the creep behaviour of aluminium alloy at constant and varying stress in order to develop mechanical and physical characteristics of aluminium alloy at high temperatures. Ansys APDL, version 14.5, was used for specimen modelling and analysis. The author was analysed the creep rate or strain rate and found that the creep rate continues to increase or decrease when the temperature is steadily increased to the maximum limit, and it was also discovered that the creep stain rate was constant at 350 °C. Increasing the load or temperature over the present limit (350 °C, 100MPa) increased the strain rate, causing the specimens to shatter. Hore et al. [33] investigated the creep analysis of 2.25Cr-1Mo steel. The continuum damage process was used to build the model. The findings show that simulated graphs properly represent experimental data and have a character that is quite similar to that of real plots.

Masse et al. [34] analyse the creep deformation and damage processes of 9Cr1Mo steel at temperatures ranging from 450 to 650 degrees Celsius using finite element analysis software. The results of their massive literature research were utilised to define characteristics for a creep strain rate of viscoelastic simulation that included the effect of degradation. Duda et al. [35] give a determination of the maximum permitted operating temperature for steam superheaters that is restricted by the creep phenomena. The analysis was carried out using Ansys software using finite element method and result was compared between analytical creep test and Ansys result. The conclusions were based on creep testing, which demonstrated that pressure is the primary source of the stresses that regulate the creep process. Bhansali et al. [39] investigated the numerical modelling and stress analysis of several boiler shell materials. For commonly used materials in boiler shells, the results and conclusions were derived using FEA software such as Ansys. Titanium Alloy was slightly safer than the other two materials in terms of static stress conditions such as equivalent stress, shear stress, and Von-Mises stress.

Previous investigations and studies done in structural and creep analysis of pressure vessels draw attention to the dearth of relevant research on boiler shell with circumferential riveted joint under the three-condition static, dynamic and creep. Analyses performed so far are based on static conditions with two materials only [3]. This investigation would fill a gap in the design of boiler shell with circumferential riveted joint that currently exists. This research has five goals: (I) Create a boiler shell model with riveted joint (II) Investigate the modelled boiler shell with riveted joint for deformation, Von-Mises stress using three materials (III) Examine the boiler shell with riveted joint for Fatigue life and damage for three materials under consideration. (IV) Investigate the modelled boiler shell with riveted joint for creep behaviors of structural steel under constant and variable stress. (V) Examine the boiler shell with riveted joint for creep fracture and creep deformation with varying temperature and time for one material under consideration.

In this study, the solid models of the cylindrical boiler shell are created using the SolidWorks software 2021. The Fatigue, deformation and stresses of developed models are assessed using Ansys 2019R3 and Ansys APDL are used to evaluate the creep behaviors, creep damage, and stresses of produced models. The paper is structured in the following manner: Section 1 begins with a short introduction followed by the literature review. Section 2 discusses the methodology used in the study, while Section 3 presents the findings and discussions. Finally, section 4 draws the key conclusions.

CHAPTER -3

MATERIALS AND METHODS

Three materials namely Titanium, Structural Steel, and Nickel-cobalt-chromium alloy are chosen for circumferential riveted joint of boiler shell. These materials are suitable for the intended operation and may be tailored to meet specific needs. Table 1 highlights the properties of selected materials.

Table 1: Properties of selected Material [20]

Property	Behavior	Density (kg/m^3)	Poisson's Ratio	Modulus of Elasticity (MPa)	Yield Strength in Tension (MPa)	Ultimate Strength in Tension (MPa)	Yield Strength in compression (MPa)
Structural Steel	Isotropic	7850	0.30	2×10^5	250	460	250
Titanium Alloy	Isotropic	4620	0.36	9.6×10^4	930	1070	930
Nickel-Cobalt-Chromium Alloy	Isotropic	8000	0.28	2.2×10^5	826	1170	826

3.1 A Model Development for static and fatigue analysis

A model of the boiler shell with a circumferential chain riveted joint has been created from three different materials. The study's major purpose is to determine the optimal material for the boiler shell joint under static and dynamic situations. Table 2 shows the circumferential riveted joint dimensions. SolidWorks software is used to model the Boiler shell riveted joint as depicted in Fig 1. The static and fatigue performance of the boiler shell joint is studied using the model designed for this purpose.

Fig 1. 3D CAD model of boiler shell joint

Table 2: Circumferential riveted joint dimension [21]

S.No	Description	Dimensions(mm)
1	Diameter of boiler shell	1600
2	Diameter of rivets	33
3	Number of rivets	90
4	Pitch of rivets	113.7
5	Distance between rows of Rivets	69
6	Margin	52
7	Thickness of boiler	28
8	Hole of rivets	34.5

3.2 Computational domain, Meshing and Boundary conditions for static and fatigue analysis

It is possible to perform a broad variety of finite element analyses using the Ansys software, ranging from basic static to complex dynamic simulations. In Ansys, there are three primary processes to solve any problem: 1. Create the model, 2. Apply loads, 3. Obtain the solution and review the results.

Model analysis relies on component meshing since it is the foundation for component analysis in any software program that supports finite element techniques. A general meshing capability in Ansys Mechanical can be used to mesh the created model. Figure 2 shows the mesh generated joint model which is further utilized in Ansys analysis.

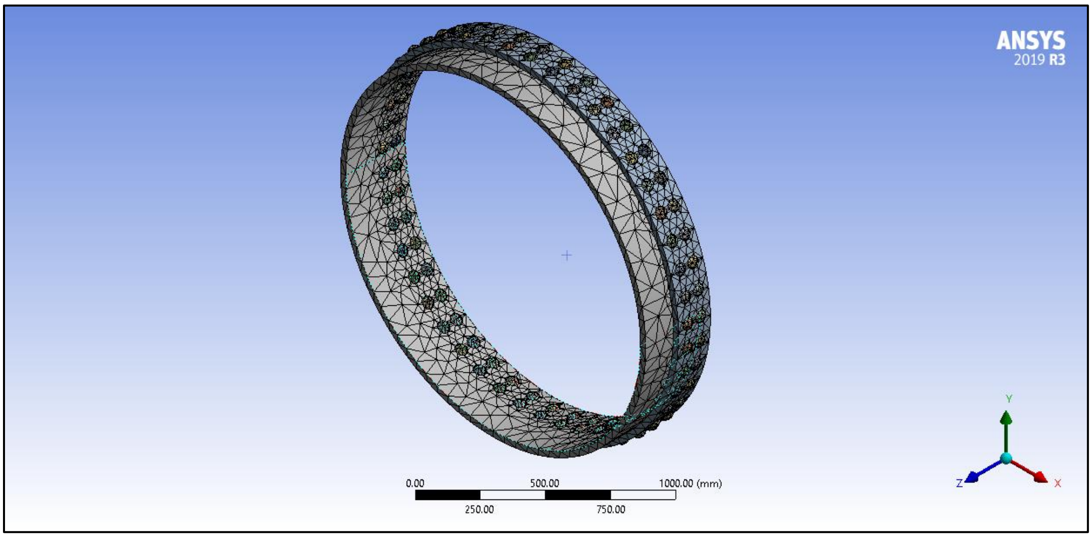


Fig.2. Meshing of boiler shell

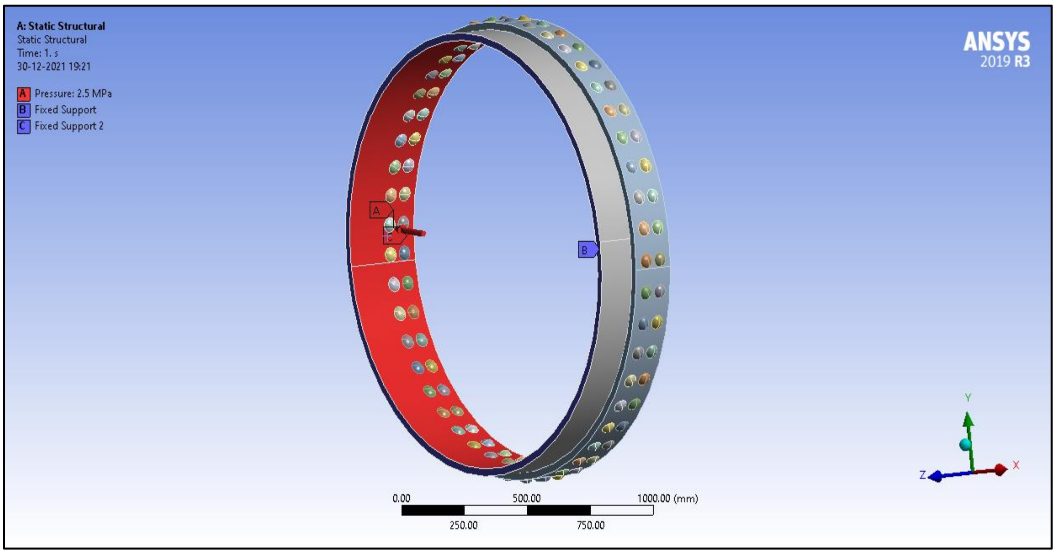


Fig.3. Boundary condition in boiler shell

A significant level of internal pressure exists inside boilers. The 2.5 MPa pressure applied to the inner surface of the boiler shell is shown in Fig. 3, and the value is obtained from the boiler's normal operating circumstances [19]. The inner surface of the boiler shell is applied to two displacements and a 2.5 MPa pressure force as the boundary conditions.

Validation of a boiler with a circumferential joint is very complicated, and it is very difficult to formulate mathematically. For the purpose of validation and completion of the presented work, a problem is introduced for the purpose of validating the FEA result [21].

$$\text{Maximum principal stress } (\sigma_1) = 75 \text{ MPa}$$

$$\text{Minimum principal stress } (\sigma_3) = 2.5 \text{ MPa}$$

$$\text{Von-Mises Stress } (\sigma_e) = \sqrt{[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2]/2} = 71.195 \text{ MPa}$$

$$\text{Maximum Shear Stress} = (\sigma_1 - \sigma_3)/2 = 36.25 \text{ MPa}$$

After the mesh is finished, 2.5 MPa pressure is applied inside the boiler shell. For all circumferential joints, structural static analysis and fatigue analysis are done with the same boundary conditions. These are necessary steps for analysing and to get accurate results.

Structural Steel is chosen for circumferential riveted joint of boiler shell. These materials are suitable for the intended operation and may be tailored to meet specific needs. Table 3 highlights the properties of selected materials.

Table 3: Properties of selected Material [20]

Property	**Behavior**	**Density (kg/m^3)**	**Poisson's Ratio**	**Modulus of Elasticity (MPa)**	**Yield Strength in Tension (MPa)**	**Ultimate Strength in Tension (MPa)**	**Yield Strength in compression (MPa)**
Structural Steel	Isotropic	7850	0.30	2×10^5	250	460	250

3.3 A Model development for creep analysis

A model of the boiler shell with a circumferential chain riveted joint has been created from structural steel. The study's major purpose is to determine the creep behaviour of the boiler shell joint under high temperature and pressure. Table 2 shows the circumferential riveted joint dimensions. SolidWorks software is used to model the Boiler shell riveted joint as depicted in Fig 4. The Creep performance of the boiler shell joint is studied using the model designed for this purpose.

Fig 4. 3D CAD model of boiler shell joint

Under a creep load and temperature, creep is a rate dependent material deformation. Creep is classified as primary, secondary, and tertiary in numerical theory. The first two levels, primary and secondary, are where modelling competency is achieved. The third stage is frequently ignored since it indicates approaching failure under any load and at any temperature. Interpolating and predicting the creep deformation of materials showing time-dependent, inelastic deformation has been established using time-dependent, inelastic constitutive models. Norton's power law equation is the most essential and extensively used constitutive equation for predicting secondary creep. It is possible to write it as:

$$\frac{d\varepsilon cr}{t} = \frac{d\varepsilon cr}{dt} = C_1 \sigma^{C_2} e^{-C_3/T}$$

Where, σ is applied stress, T is applied temperature, C_1, C_2 & C_3 are creep coefficients and $d\varepsilon cr/dt$ is creep strain rate.

It is vital to note that the Norton Creep Law is simply an estimate of structural steel creep behaviour at a boiler's circumferential riveted joint. The creep strain events are predicted using time and strain hardening formulae at various load and temperature levels. In this circumstance, the implicit creep approach was chosen although it is reliable, quick, and accurate, and it is recommended for widespread usage. It also supports temperature-dependent creep constants and simultaneous linkage with isotropic harden plasticity models.

3.4 Computational domain, Meshing and Boundary conditions for creep analysis

It is possible to perform a broad variety of finite element analyses using the Ansys software, ranging from basic static to complex dynamic simulations.

In Ansys, there are three primary processes to solve any problem: 1. Create the model, 2. Apply loads, 3. Obtain the solution and review the results.

Model analysis relies on component meshing since it is the foundation for component analysis in any software program that supports finite element techniques. A general meshing capability in Ansys Mechanical can be used to mesh the created model. Figure 5 shows the mesh generated joint model which is further utilized in Ansys analysis.

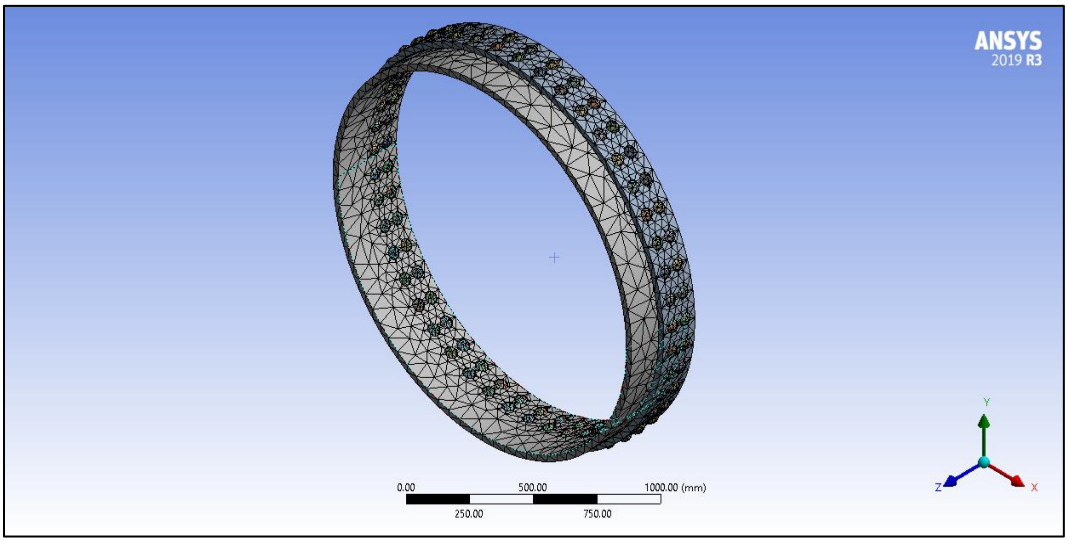


Fig.5. Meshing of boiler shell

A significant level of internal pressure and temperature distributions within the circumferential riveted joint of boiler shell were determined. The 2.5 MPa pressure and varying temperature (300 °C, 400 °C, and 450 °C) applied to the inner surface of the boiler shell is shown in Fig. 6, and the value is obtained from the boiler's normal operating circumstances [19] and The Norton Power law was utilised to analyse the findings produced by the Ansys 2019 model. The inner surface of the boiler shell is applied to two displacements and a 2.5 MPa pressure force and temperature as the boundary conditions. To perform creep analysis, creep effects needs to be turned in analysis settings and end time for load step 2 is need to be specified and creep ratio limit is set to

Thermal and mechanical parameters are discussed in table, including thermal conductivity, specific heat, elastic modulus, Poisson's ratio, and thermal expansion coefficient. Temperature has an impact on these variables.

Table 4: Properties of structural steel [32]

Property	Specific heat (C)	Temperature (T)	Thermal conductivity (K)	Thermal expansion (α)	Stress (σ)	Time period (t)
Structural steel	434 J/Kg K	300 °C, 400 °C, 450 °C	60.5 W/m K	1.2 E-5 /K	2.5 MPa	10 hrs

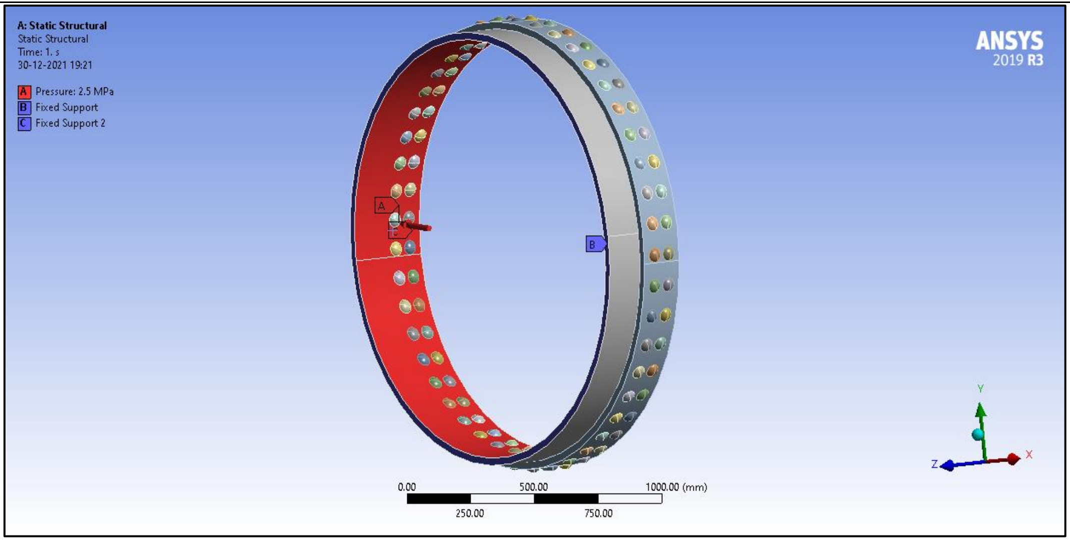


Fig.6. Boundary condition in boiler shell

Validation of a boiler with a circumferential joint is very complicated, and it is very difficult to formulate mathematically. For the purpose of validation and completion of the presented work, a problem is introduced for the purpose of validating the FEA result [37].

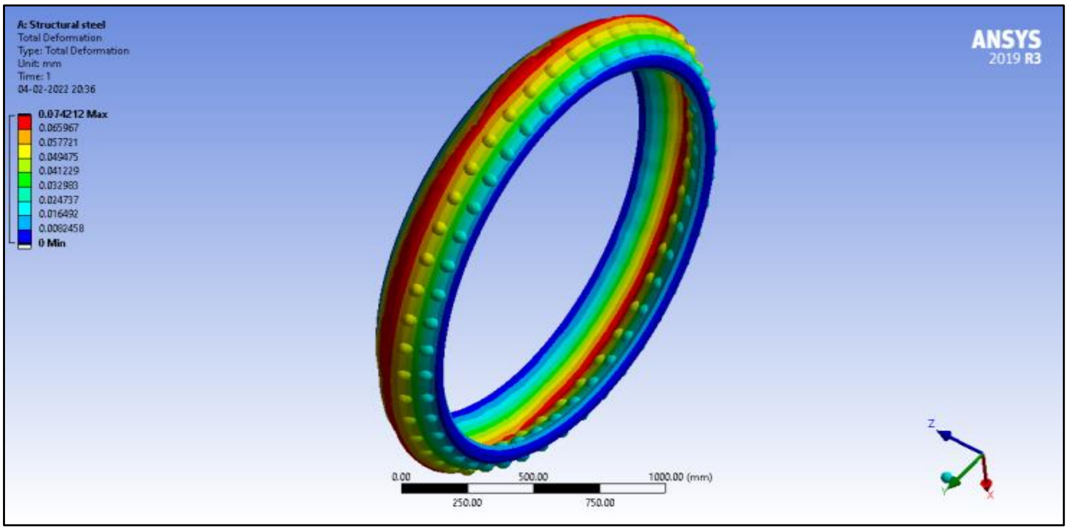
After the mesh is finished, 2.5 MPa pressure and varying temperature (300 °C, 400 °C, and 450 °C) is applied inside the boiler shell. For circumferential joints, Creep Behaviour analysis are done with the same boundary parameters. These are appropriate steps for analysing and to get accurate findings.

CHAPTER -4

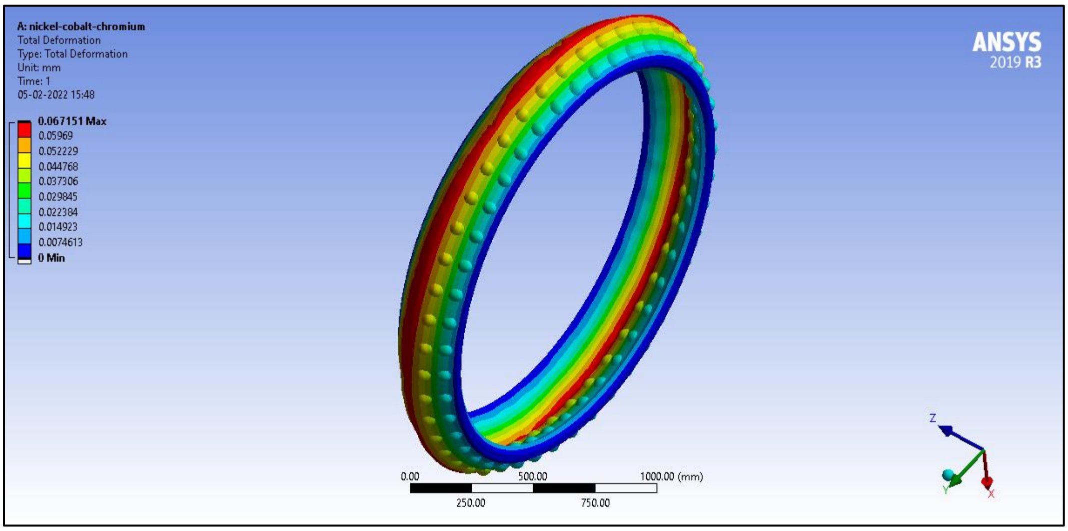
RESULT AND DISCUSSION

4.1 Results and discussion for static and fatigue analysis

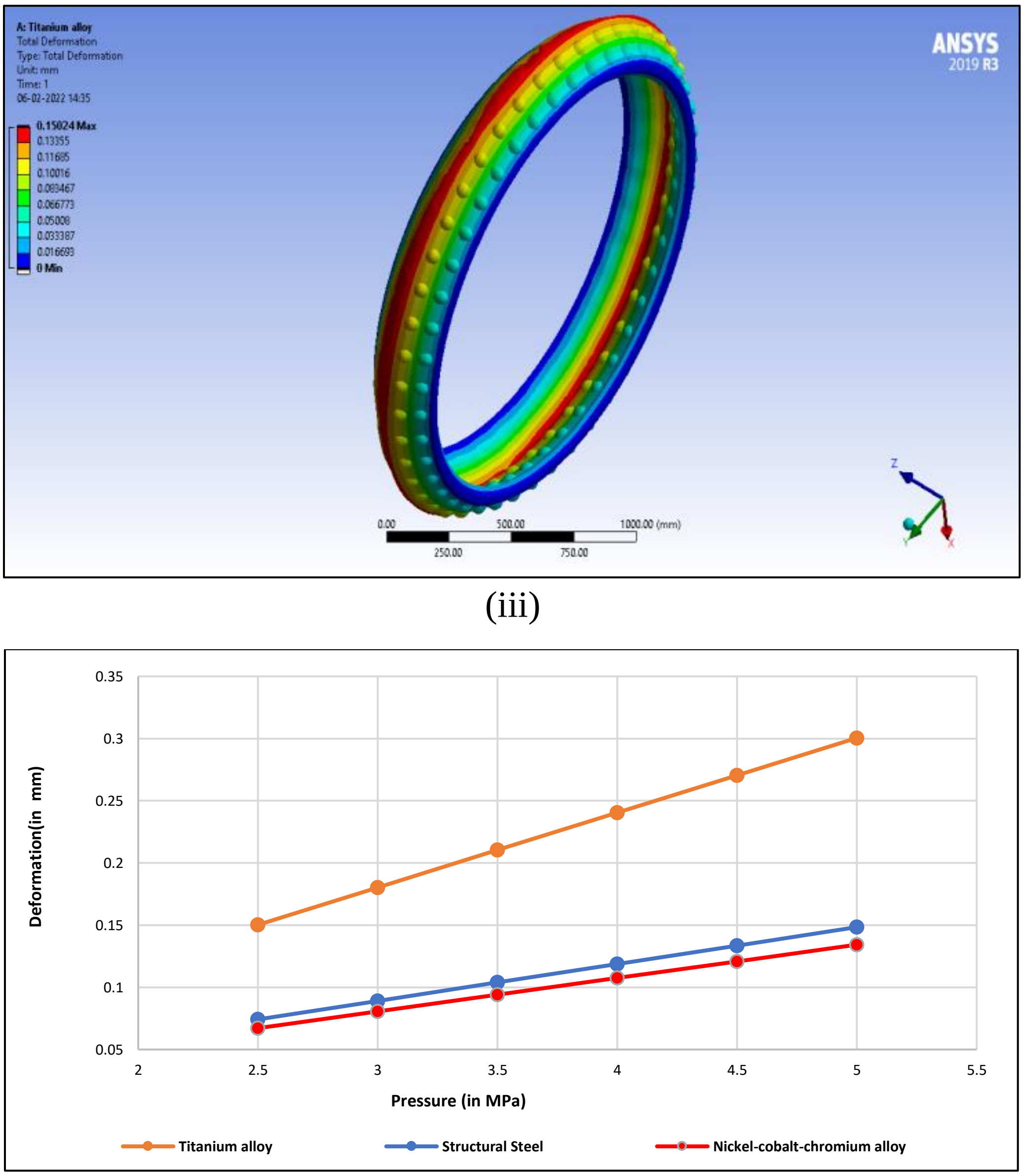
Applied Forces and constraints on a boiler shell create total deformation, which is defined as the vector sum of all directional displacements. Figure 7 illustrates the contour of the boiler shell joint's total deformation at 2.5 MPa internal pressure. In the contour chart, the maximum deformation is shown in red, while the least distortion is shown in blue. Increasing the internal pressure of the boiler shell results in an increase in total deformation. Titanium alloy's lower density means that its circumferential riveted connection deforms more than that of structural steel and nickel-cobalt-chromium alloy. At any given pressure, nickel-cobalt-chromium alloy has the lowest overall deformation among the three materials.



(i)



(ii)

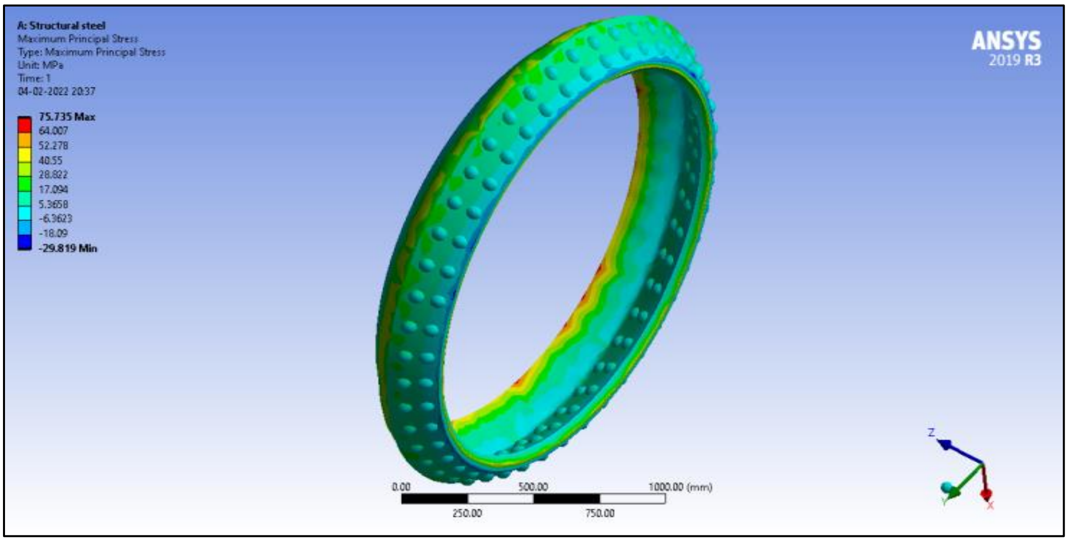


(iii)

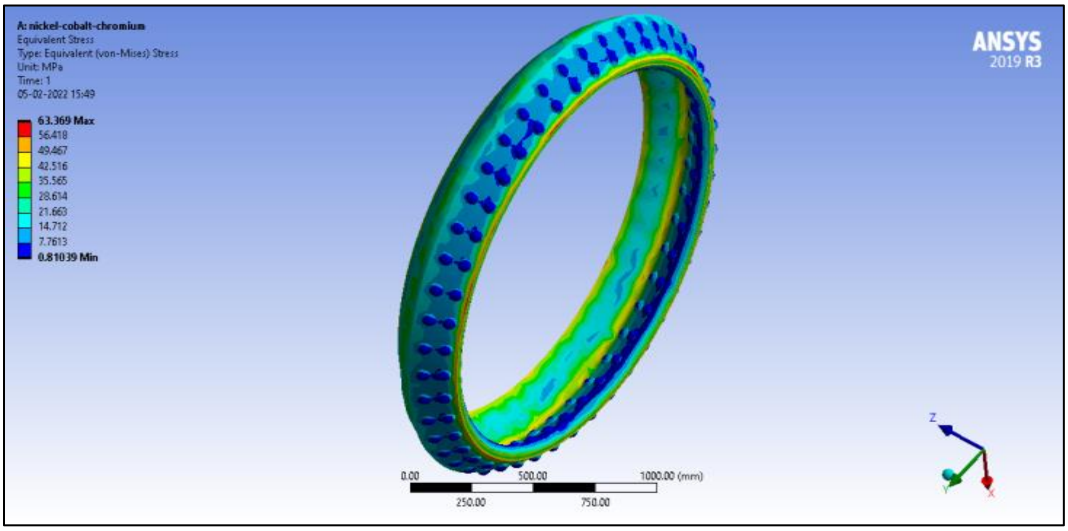
(iv)

Fig. 7: Total Deformation at 2.5MPa internal pressure for (i)Structural steel, (ii) Nickel-cobalt-chromium alloy, (iii) Titanium alloy and (iv) Comparison of total deformation at increasing pressure

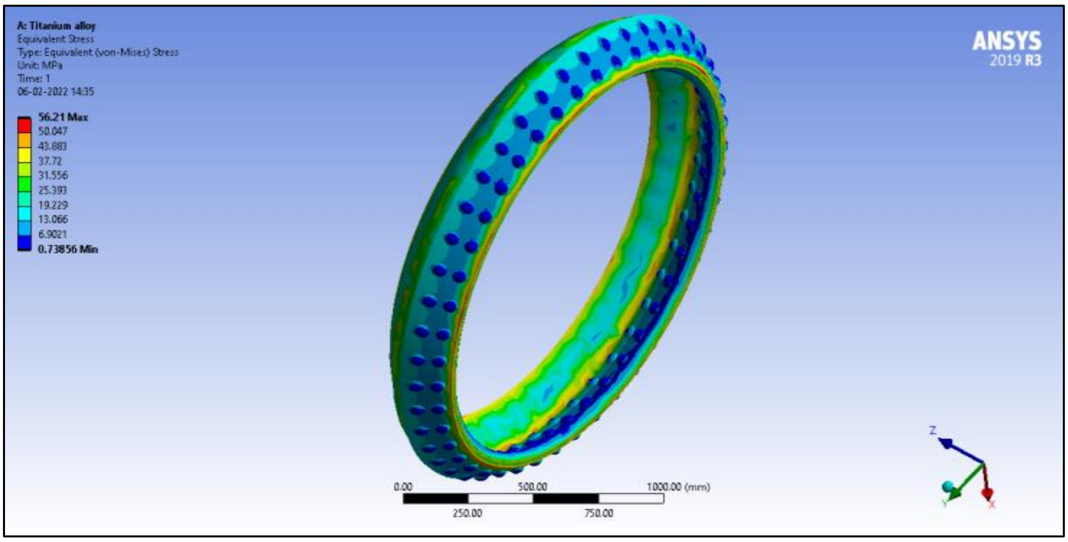
Figure 8 depicts the Von-Mises stress in a circumferential riveted joint manufactured of structural steel, nickel-cobalt-chromium alloy, and titanium alloy. The Equivalent stress value is used to assess whether a material will yield or fracture while under stress. However, owing to the use of the safety factor, this number is lower than the actual value. Fig. 5 Colour variation depicts the various stresses at various points in the joint model. The Nickel-Cobalt-Chromium alloy exhibits the highest equivalent stress of 63.67 MPa and the lowest equivalent stress of 0.81 MPa at 2.5 MPa internal pressure, as shown in Fig. 8. Nickel-Cobalt-Chromium alloy, as illustrated in Fig. 5(iv), had the maximum Von-Mises stress at the same pressure as Titanium Alloy and structural steel. Nickel-Cobalt-Chromium alloy looks to be more promising, despite the fact that the value is modified by the safety factor.



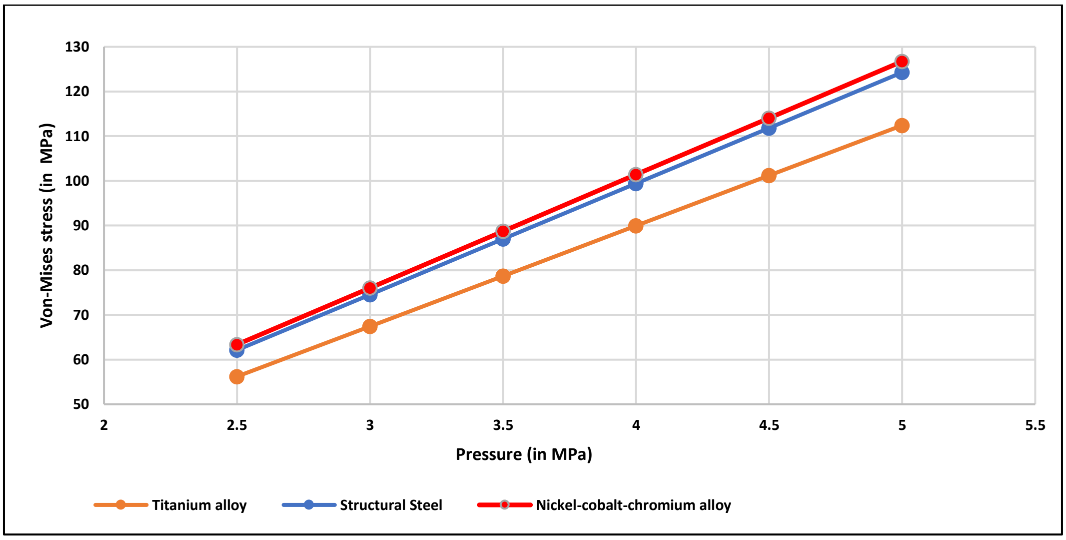
(i)



(ii)



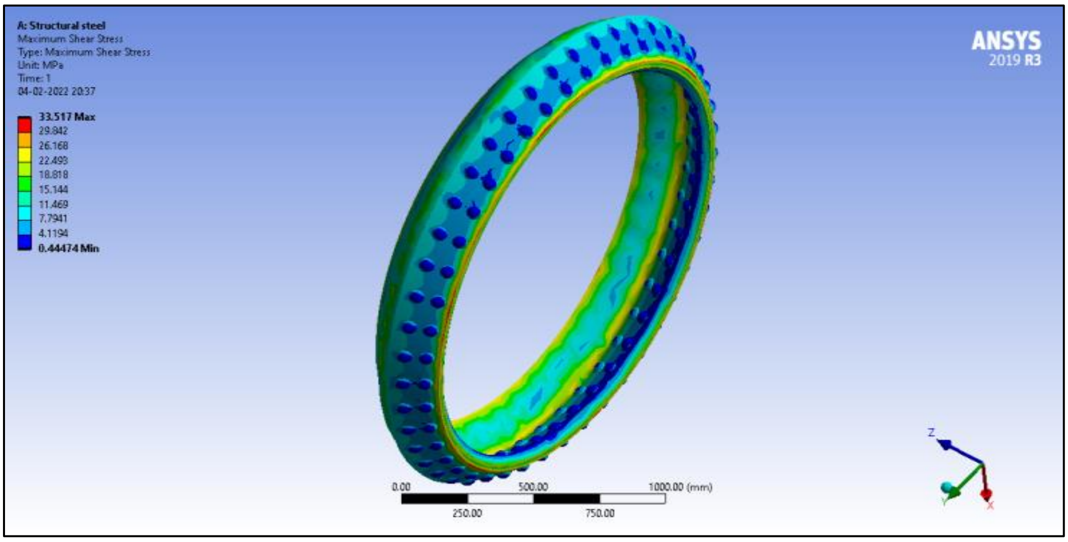
(iii)



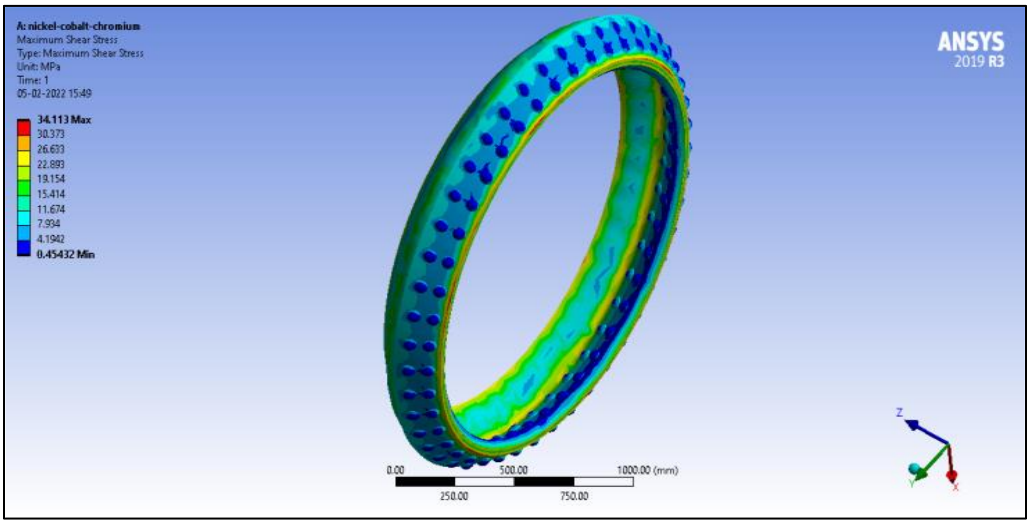
(iv)

Fig.8: Von-Mises stress at 2.5MPa internal pressure for (i) Structural steel, (ii) Nickel-cobalt-chromium alloy, (iii) Titanium alloy and (iv) Comparison of Von-Mises stress at increasing pressure

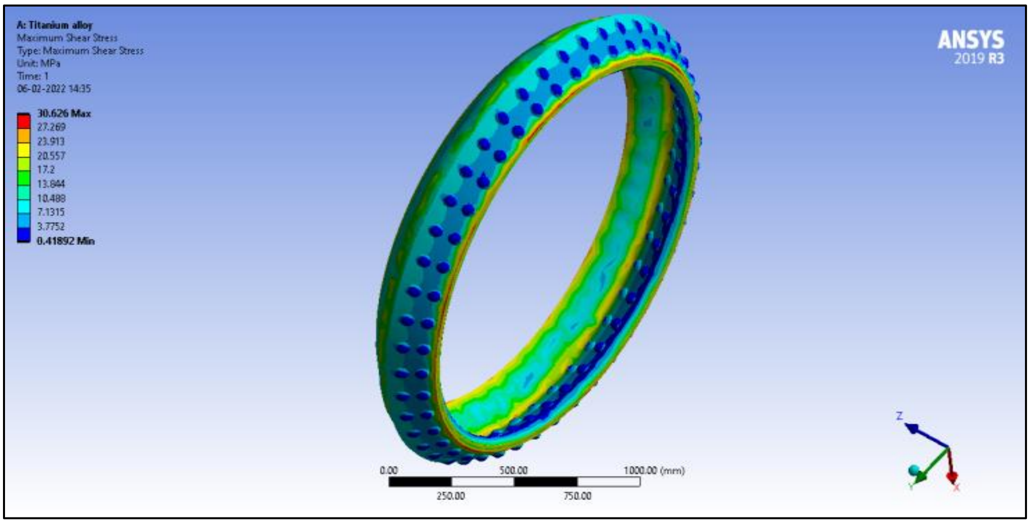
Figure 9 shows the maximum shear stress at 2.5 MPa internal pressure in a structural steel, nickel-cobalt-chromium alloy, and titanium alloy boiler shell joint. "Maximum shear stress" describes the unequal distribution of maximum shear force due to its concentration in a limited location. Stronger structures are required because unbalanced shear stresses might result in disastrous situations. According to the findings, the shear stress in a Titanium alloy circumferential riveted joint is more promising. Figure 9 depicts the shear stress differential between structural steel and nickel-cobalt-chromium alloy (iv).



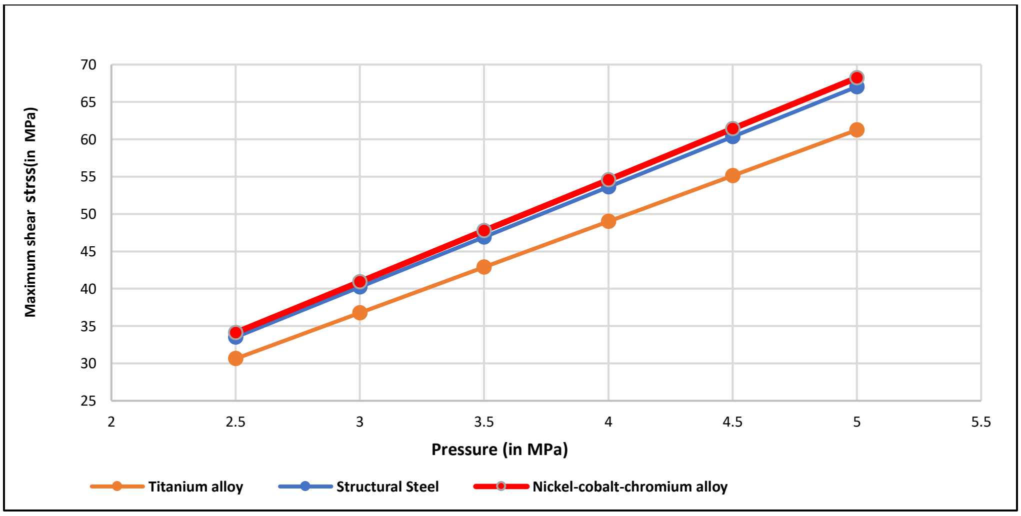
(i)



(ii)



(iii)



(iv)

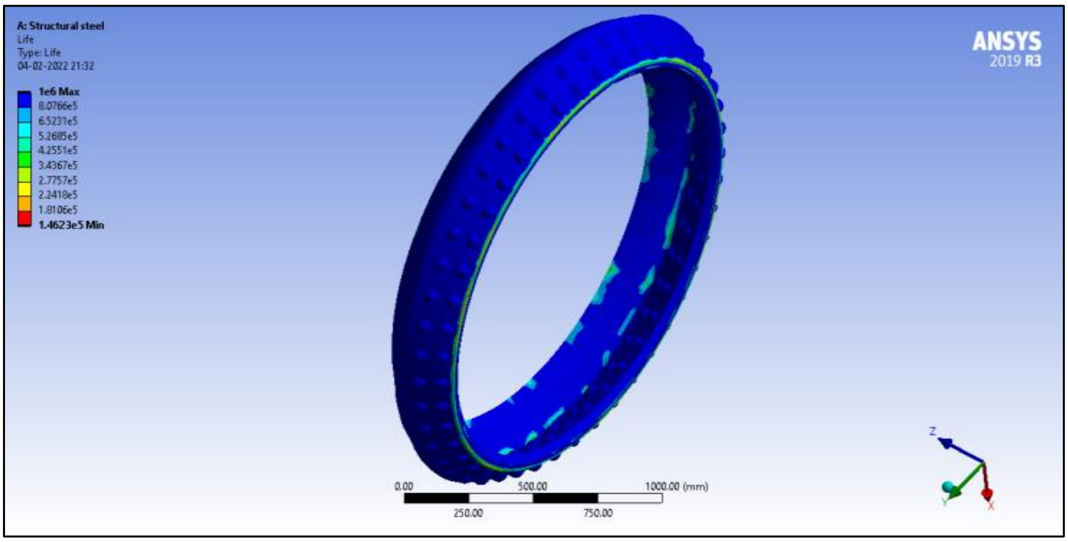
Fig.9: Maximum shear stress at 2.5MPa internal pressure for (i) Structural steel, (ii) Nickel-cobalt-chromium alloy, (iii) Titanium alloy and (iv) Comparison of maximum shear stress at increasing pressure.

The FEA result of the riveted joint modal of all three materials at internal pressure 2.5 MPa is compared in Table 5 with the analytical results. Von-Mises stress and maximum shear stress are less in FEA findings than in analytical results for all three materials. According to the aforementioned comparison, the FEA result of Von-Mises stress and maximum shear stress is 12% and 6% lower than the analytical result, respectively. As a result, the boiler shell's joint will survive and fulfil its function successfully. The Ansys Mechanical module is used for the fatigue study of the model created. The S-N curve is used in stress life fatigue analysis. The fatigue data are extracted from the software result for the fatigue analysis of the boiler shell model. This study also covers the fatigue factors sensitivity.

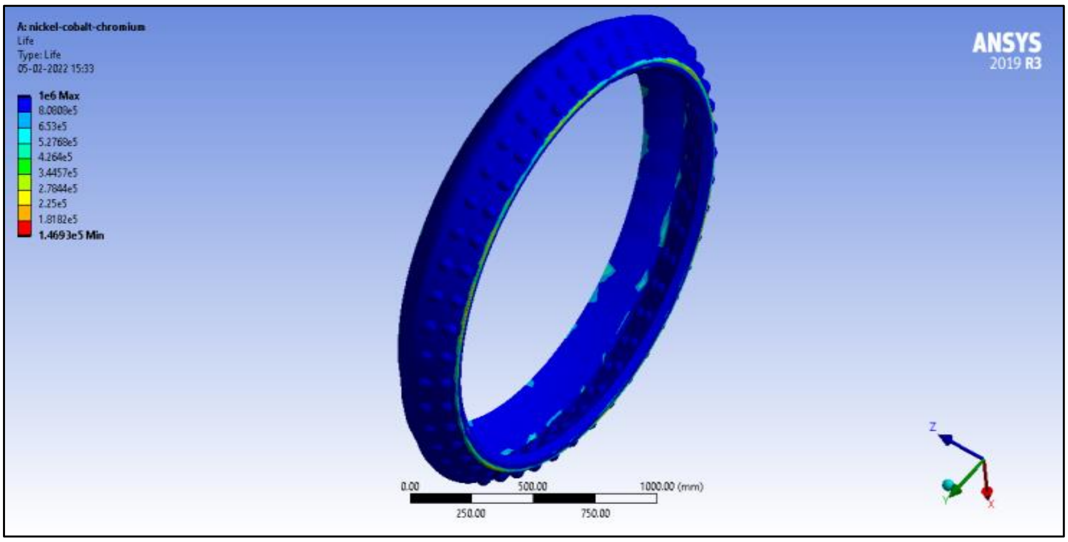
The functional life of a model before it fails due to fatigue is referred to as the "fatigue life." A number of cycles illustrates the fatigue life at particular stress level. The components subjected to the maximum stress have the minimum life expectancy. Figure 10 shows the boiler shell joint fatigue life at 5 MPa internal pressure. In the 2.5 MPa range, all three materials have same minimum and maximum fatigue life (1 x 10^6 cycles). Minimum fatigue life for structural steel is 1.4623x10^5 cycles when pressure is doubled(5MPa). Contour map shows maximum and minimum lifespans, with blue indicating maximum and red showing minimum lifespans. Structural steel and nickel-cobalt-chromium alloy have shorter fatigue life than titanium alloys as depicted in fig10(iv).

Table 5 Comparison of analytical findings with FEA findings of Structural steel, Nickel-cobalt-chromium alloy and Titanium alloy at 2.5 MPa

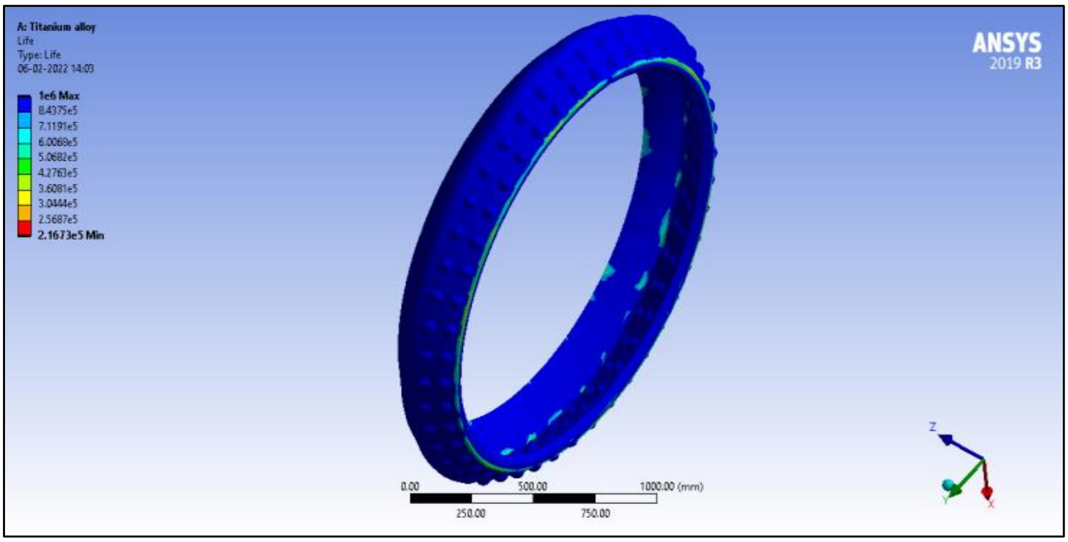
S. No	**Variables**	**Analytical Findings**	**FEA findings for structural steel**	**FEA findings for Nickel-cobalt-chromium alloy**	**FEA findings for Titanium alloy**
1	Von-Mises stress (in MPa)	71.95	62.14	63.37	56.21
2	Maximum shear stress (in MPa)	36.25	33.52	34.11	30.63


A: Structural steel
Life
Type: Life
04-02-2022 21:32
1e6 Max
8.0766e5
6.5231e5
5.2685e5
4.2551e5
3.4367e5
2.7757e5
2.2418e5
1.8106e5
1.4623e5 Min
ANSYS
2019 R3
0.00
500.00
1000.00 (mm)
250.00
750.00
Z
Y
X

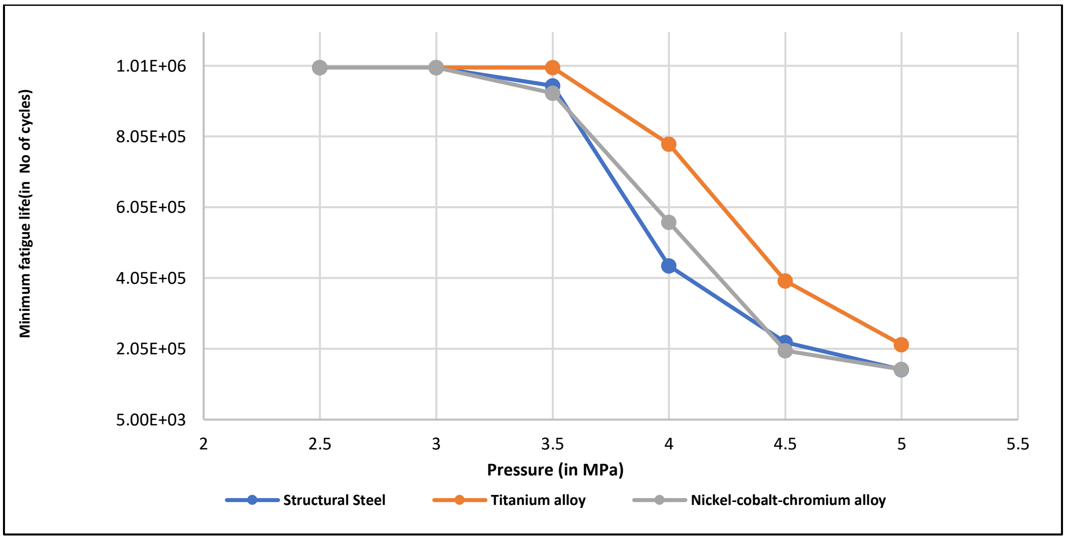
(i)


A: nickel-cobalt-chromium
Life
Type: Life
05-02-2022 15:33
1e6 Max
8.0808e5
6.53e5
5.2768e5
4.264e5
3.4457e5
2.7844e5
2.25e5
1.8182e5
1.4693e5 Min
ANSYS
2019 R3
0.00
500.00
1000.00 (mm)
250.00
750.00
Z
Y
X

(ii)


A: Titanium alloy
Life
Type: Life
06-02-2022 14:03
1e6 Max
8.4375e5
7.1191e5
6.0068e5
5.0682e5
4.2763e5
3.6081e5
3.0444e5
2.5687e5
2.1673e5 Min
ANSYS
2019 R3
0.00
500.00
1000.00 (mm)
250.00
750.00
Z
Y
X

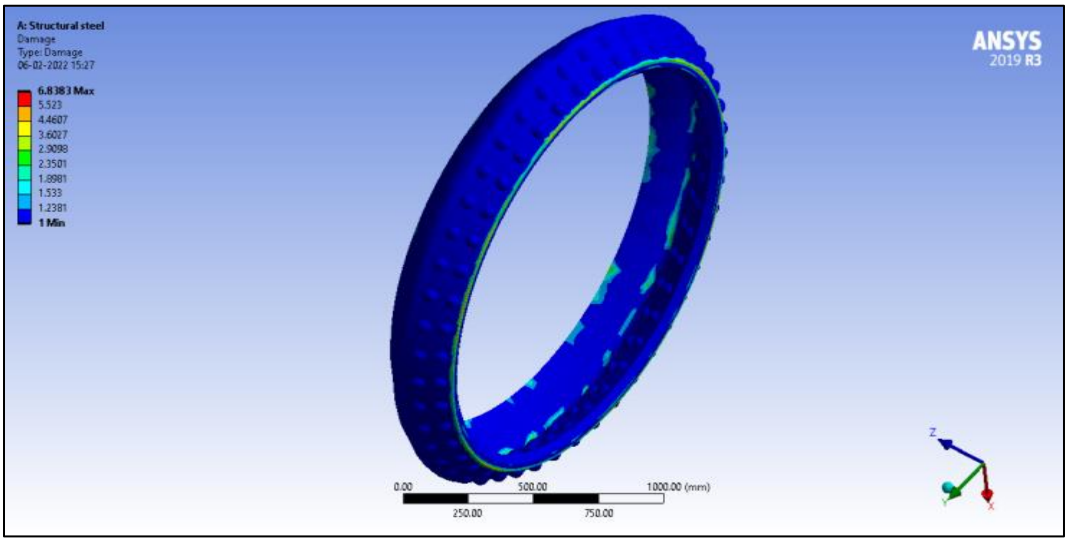
(iii)



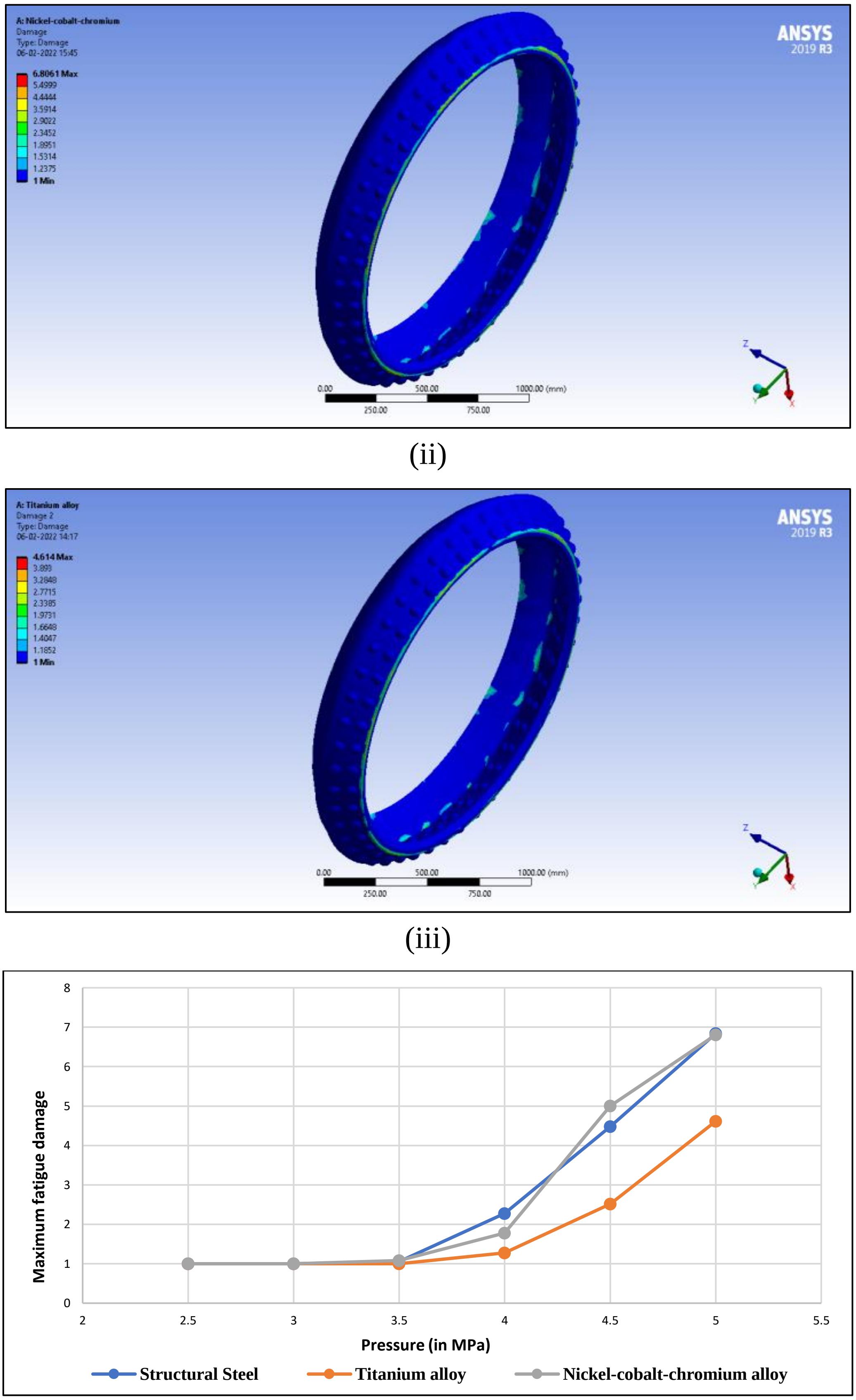
(iv)

Fig.10: Fatigue life for (i) Structural steel, (ii) Nickel-cobalt-chromium alloy, (iii) Titanium alloy and (iv) Comparison of minimum fatigue life at increasing pressure

The ratio of design life to available life is known as fatigue damage. In the contour plot as shown in Fig.11, the blue colour shows the minimum fatigue damage, and the red colour shows the maximum amount of damage. At an internal pressure of 5 MPa, the boiler shell joint's Fatigue damage is depicted in Fig. 11. From 2.5 MPa to 3.5 MPa, there is no evidence of Fatigue damage to the boiler shell joint, as illustrated in Fig.8 (iv). When the internal pressure is raised to 5 MPa, structural steel shows the maximum fatigue damage of 6.83 at the inner shell's edge. In contrast, at 5 MPa internal pressure, titanium alloy shows a minimum damage of 4.6.



(i)



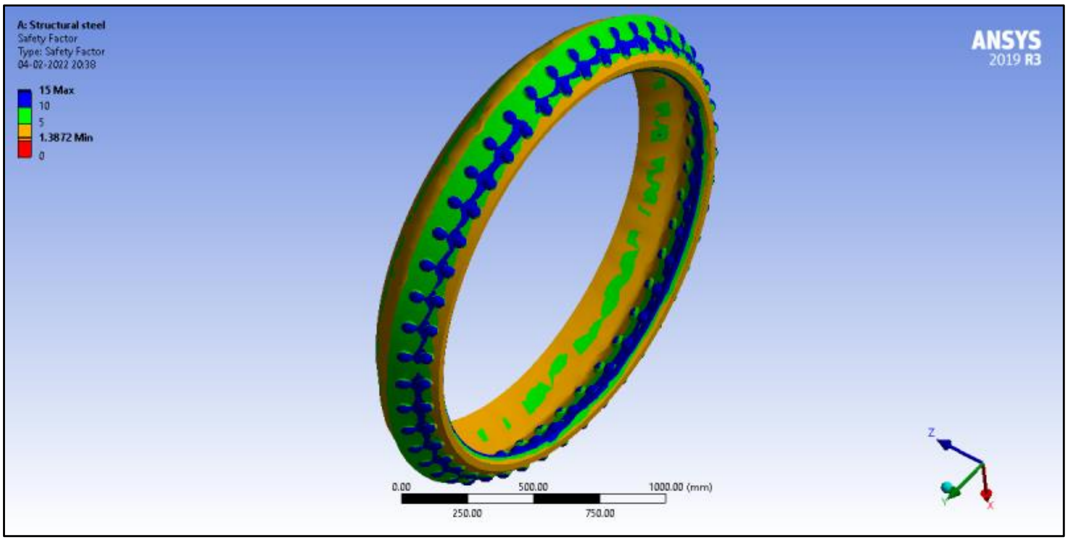
(ii)

(iii)

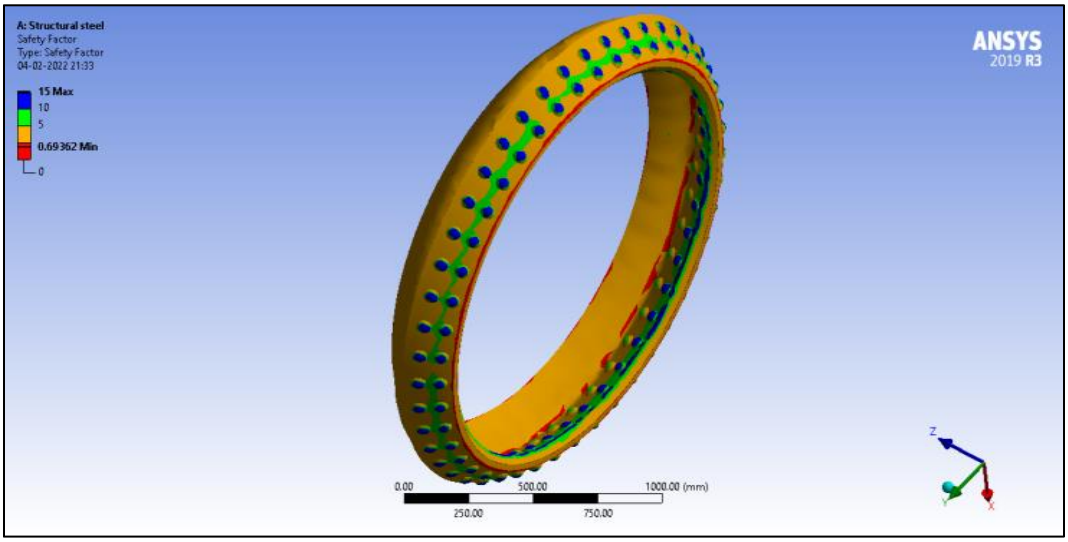
(iv)

Fig. 11: Fatigue damage (i) Structural steel, (ii) Nickel-cobalt-chromium alloy, (iii) Titanium alloy and (iv) Comparison of maximum fatigue damage at increasing pressure

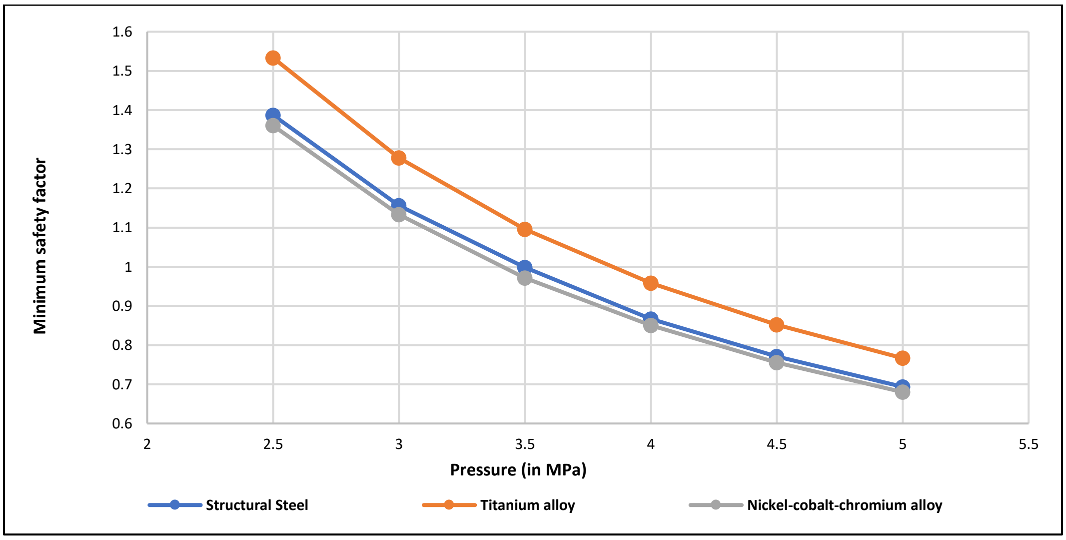
The Fatigue factor of safety is a contour plot representing the factor of safety with reference to a fatigue failure at a particular design life. The highest safety factor is fifteen. Fatigue Safety Factor values of less than one suggests that the component will fail before its design life has reached. An internal pressure of 2.5 MPa results in maximum safety in titanium Fig. 12 (iii). The safety factor of all materials decreases as the pressure increases. Boiler shell joint of three different materials with an internal pressure of 4 MPa are not safe to use, since their safety factor is below 1.



(i)



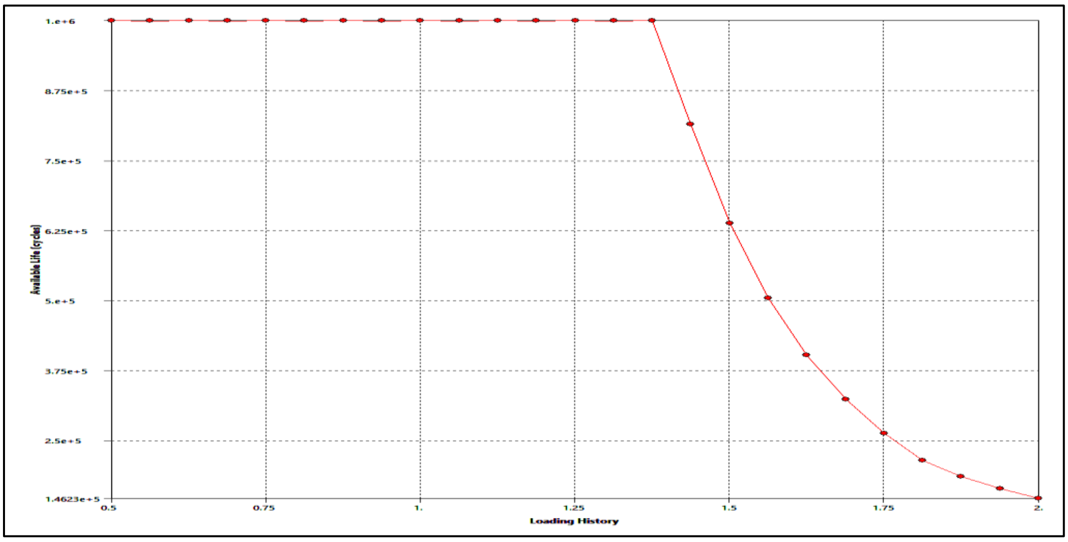
(ii)



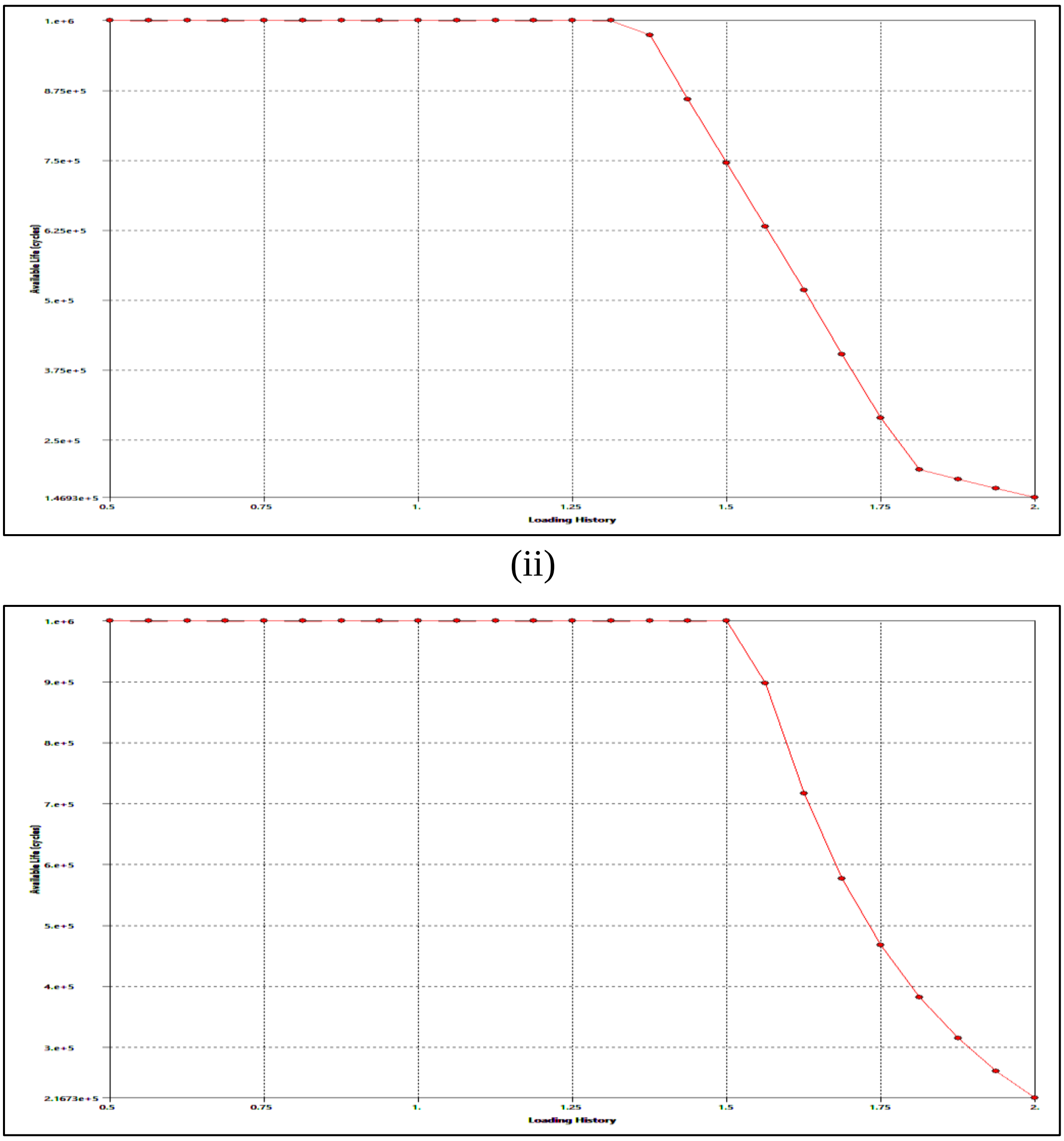
(iii)

Fig. 12: Safety factor for Structural steel at (i)2.5 MPa, (ii) 5 MPa, and (iii) Comparison of minimum safety factor at increasing pressure for all three materials

The graph of fatigue sensitivity depicts the relationship between the fatigue outcome and the variation in loading. The sensitivity of the joint model life is measured from 50 percent to 200 percent of the base load (2.5MPa). Titanium alloy's minimum service life stays unchanged at 150 percent of the base load. titanium alloy has a maximum life compared to structural steel and nickel-cobalt-chromium alloy at 200 percent base load.



(i)



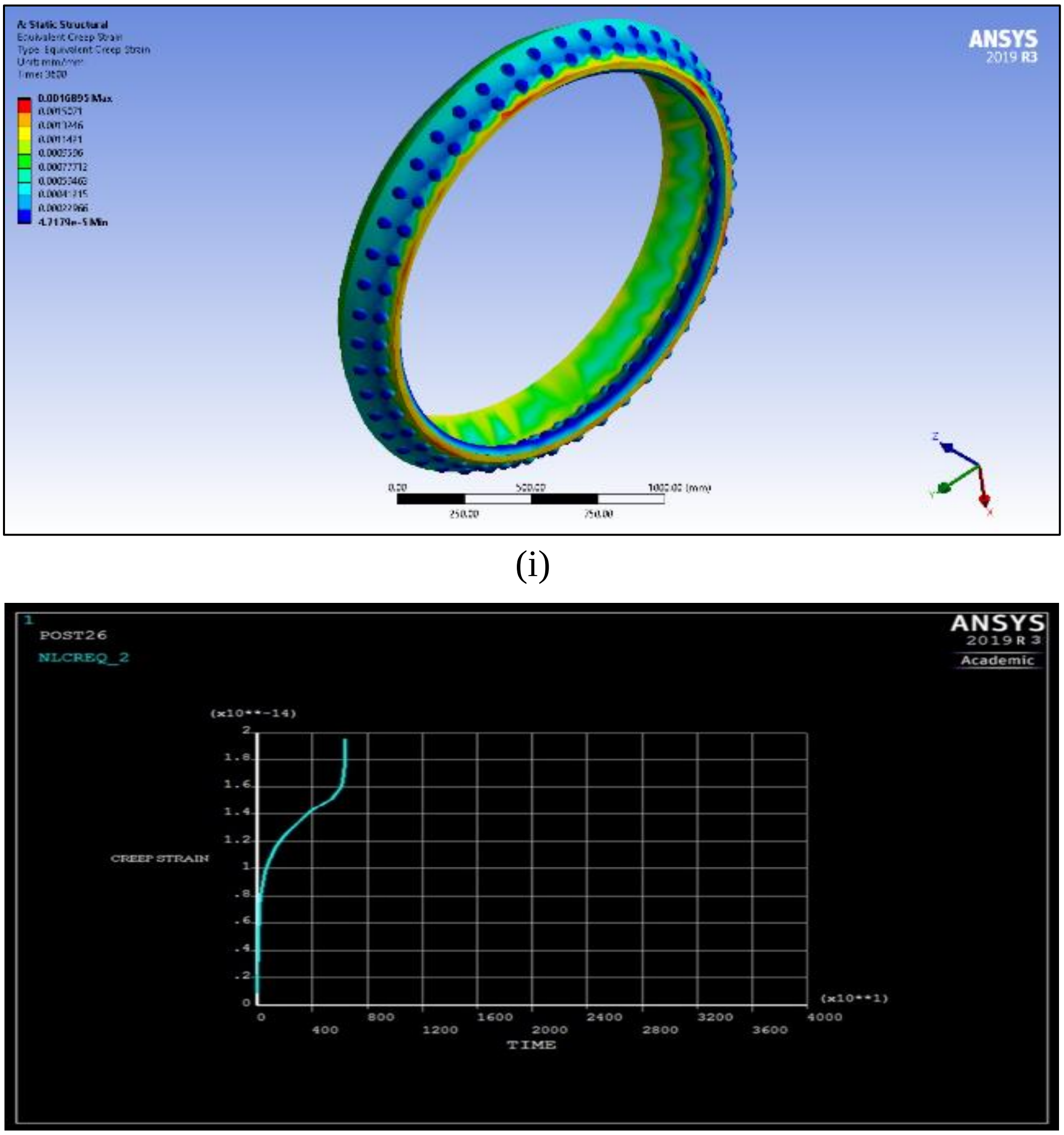
(ii)

(iii)

Fig. 13: Fatigue sensitivity at 2.5MPa internal pressure for (i) Structural steel, (ii) Nickel-cobalt-chromium alloy, and (iii) Titanium alloy

4.2 Results and discussion for static and fatigue analysis

Equivalent creep strain is the persistent plastic deformation of a structure under steady load over an extended period of time. It is a subatomic, micro-level phenomenon. At 300 °C and 2.5 MPa, Figure 14(i) displays the equivalent creep strain of the boiler shell's circumferential riveted joint. The greatest value of equivalent creep strain of the boiler's circumferential riveted joint is 0.0016895, and Fig 14(ii) illustrates creep strain-time variation with two distinct phases. The strain gradually increases with time in the first phase, which corresponds to transitory creep. The results show that with constant applied stress and temperature, the creep rate rises. Because stress causes gaps to shift, displacement and displacement jog. As a consequence, the period of the second phase in comparison to the first phase may have an effect on the calculation of model properties, for which the curve produced at 2.5 MPa and 300 °C is superior.



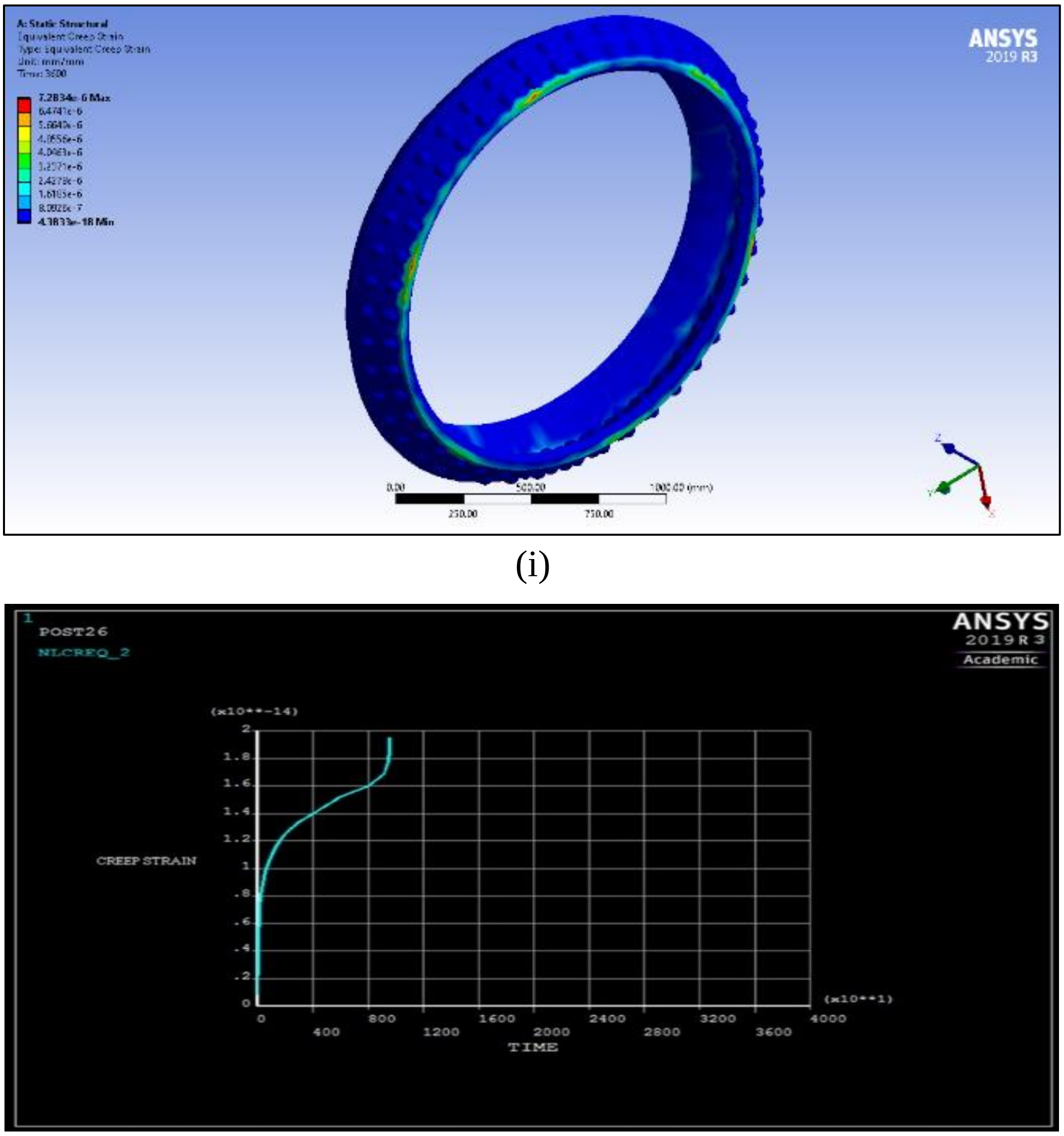
(i)

(ii)

Fig.14: At 300 °C (i) Equivalent creep strain of boiler shell at 3600 s and (ii) Graph of creep strain vs time (in seconds) of boiler shell

At 400 °C and 2.5 MPa, Figure 5a displays the corresponding creep strain of the boiler shell's circumferential riveted joint. The maximum equivalent creep strain of the boiler's circumferential riveted joint is 7.2834e-6. Figure15(i) depicts strain-time variations with clearly defined creep phases. The strain gradually increases with time in the first phase, which correlates to transitory creep. The second phase is the minimal creep rate, in which the strain grows linearly over time. The third phase is rapid creep, which results in material failure at various intervals. As a result, the period of the third phase in comparison to the preceding example may have an effect on the identification of model parameters for which the curve produced at 400 °C. The creep curve for a load of 2.5 MPa and 400 ^{0} C temperature is better than the curve for 300 °C temperature, 2.5 MPa pressure. At constant temperature, the findings reveal that creep rate increases with increasing stress.

Figure 15(ii) depicts the third phase, which displays rapid creep leading to material failures over time. As a result, the period of the third phase in comparison to the prior example may have an effect on the identification of model components for which the curves created at 300 °C. The data show that creep strain rate increases as temperature rises.

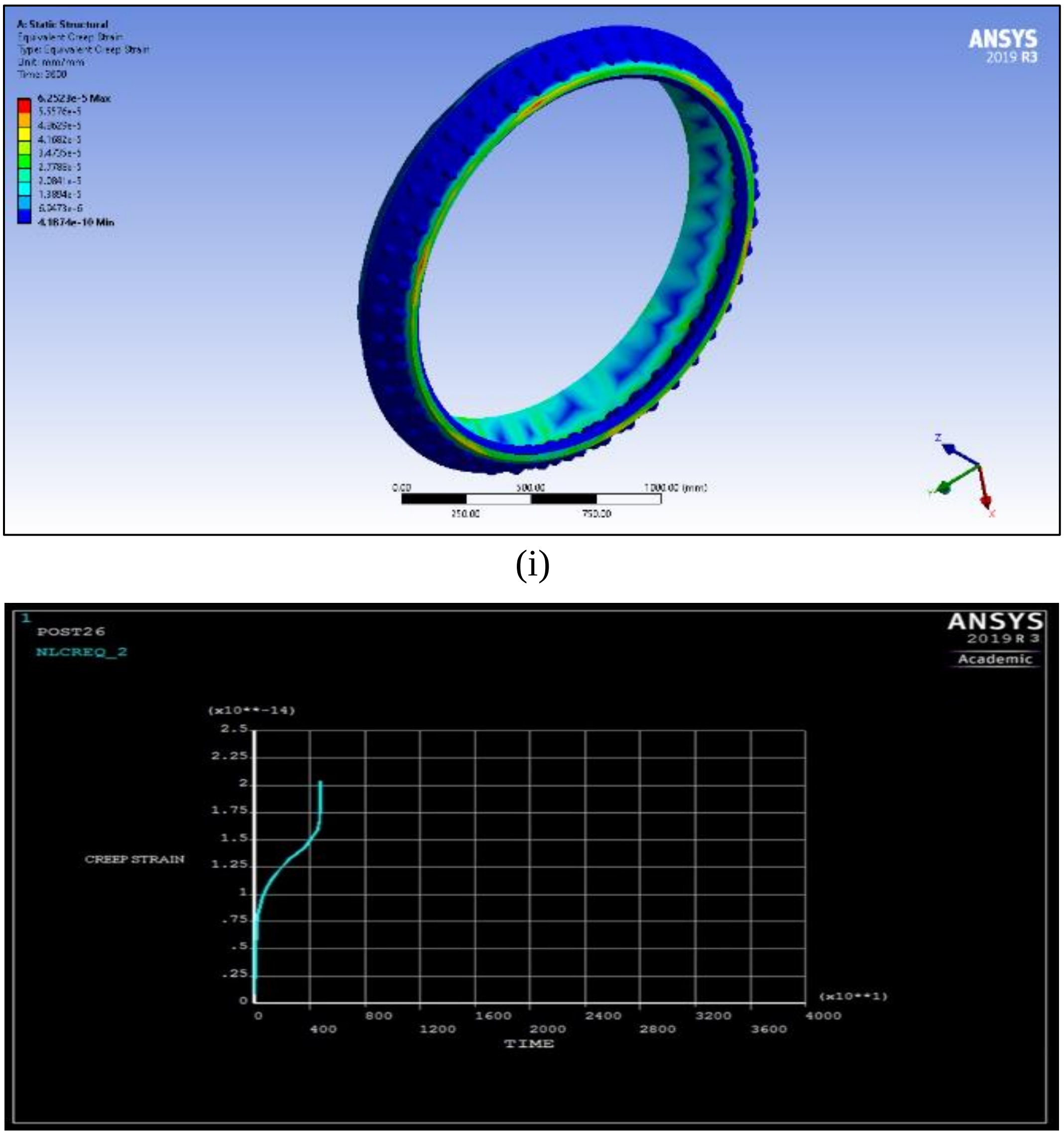


(i)

(ii)

Fig.15: At 400 °C (i) Equivalent creep strain of boiler shell at 3600 s and (ii) Graph of creep strain vs time (in seconds) of boiler shell

The maximum equivalent creep strain of the boiler's circumferential riveted joint is 6.2525e-5 at 450 °C and 2.5 MPa applied stress, as illustrated in Figure 16(i) Figure16(ii) depicts strain-time variation with well-defined creep phases. The instantaneous creep is the initial stage, in which the strain quickly rises over time. The minimal creep rate is the second phase, in which the strain grows linearly over time. Accelerated creep is the third phase, which results to materials degradation at varied time intervals. As a result, the period of the third phase in comparison to the prior scenario may have an impact on the model parameters determined for the curve acquired at 450 °C. The creep curves for a load of 2.5 MPa is better than that of 300 °C at 2.5 MPa. The findings show that when stress and temperature remain constant, the creep rate rises.



(i)

(ii)

Fig.16: At 450 °C (i) Equivalent creep strain of boiler shell at 3600 s and (ii) Graph of creep strain vs time (in seconds) of boiler shell

The preceding results show that altering the load causes a short-term failing of the creep component at a constant temperature of 450 °C. For a load of 2.5 MPa, the superior creep curve is achieved. The ideal load range for greater creep life of structural steel was determined to be 2.5 MPa, and the optimum temperature range was determined to be 450 °C, based on the above findings.

CHAPTER – 5

CONCLUSION

The findings of these investigations were used to analyse the static, fatigue and creep behaviour of boiler shell joint with the best shell material. The study led to the following conclusions:

- Boiler shells made of structural steel, titanium alloy, and nickel-cobalt-chromium alloy were created using SolidWorks software.
- A nickel-cobalt-chromium alloy boiler shell joint has a lower total deformation (0.067mm) and lower Von-Mises stress (63.37MPa) when subjected to an internal pressure (2.5 MPa) than structural steel and titanium alloy.
- When comparing the maximum shear stress of titanium alloy (30.63 MPa) to structural steel (33.52 MPa) and nickel-cobalt-chromium alloy, the titanium alloy obtains a best performance (34.11 MPa).
- The boiler shell with a circumferential riveted joint of Titanium alloy has a longer fatigue life than the circumferential riveted joint made of structural steel, Nickel-cobalt-chromium alloy at high internal pressure.
- The static and fatigue study of the circumferential riveted joint of the boiler shell reveals that all three materials will survive and perform the function efficiently, however Nickel-cobalt-chromium alloy and titanium alloy exhibit superior results than structural steel.
- The Equivalent creep strain of structural steel at constant stress (2.5MPa) and varying temperature (300 °C, 400 °C and 450 °C) are 0.0016895, 7.2834e-6, and 6.2525e-5.
- In comparison to other parameters, the better creep strength was reached at temperature of 450 °C and 2.5MPa applied stress, according to the numerical analysis findings.
- In the remaining situations, Creep behaviour of structural steel reached the tertiary phase in a short duration of time.
- The best load range for greater creep life of structural steel was shown to be 2.5 MPa, and the optimal temperature range was discovered to be 450 °C.

CHAPTER-6

LIST OF PUBLICATION

1. Singh, S.K., Eqbal, T. & Gupta, V., (Static and Fatigue Analysis of Boiler Shell with Circumferential Riveted Joint). 3rd Biennial International Conference on Future Learning Aspects of Mechanical Engineering (FLAME - 2022). Accepted
2. Singh, S.K., Eqbal, T. & Gupta, V., (Static and Fatigue Analysis of Boiler Shell with Circumferential Riveted Joint). 3rd Biennial International Conference on Future Learning Aspects of Mechanical Engineering (FLAME - 2022). Communicated

REFRENCES

1. The Economics Times, Dec 26, 2021, six people were killed and 11 injured when the boiler at a food processing factory exploded in Bihar’s Muzaffarpur. https://economictimes.indiatimes.com/news/india/boiler-blast-kills-six-in-bihars-muzaffarpur/articleshow/88505933.cms?utm_source=contentofinterest&utm_medium=text&utm_campaign=cppst

2. Bhandari, V. B. *Design of Machine Element.* New Delhi: Tata McGraw- Hill Publishing company Limited. (2020).

3. Pathan, A. k., Sheikh Rauf, S. K., & Nandanwar, N. V. Static Structural Analysis of Boiler Shell with Riveted Joints Using CAE Tool's. *International Journal of Engineering Research and devlopment, 10*, 76-83 (2014).

4. Singh, K., Chattarjee, S., & Pandey, K.. Modeling and structural Analysis of Boiler Shell with Riveted Joint. *Journal of Material Science and Mechanical Engineering (JMSME), 4*, 103-108 (2017).

5. Bhansali, G., Singh, G., Kumar, S. R., & Wagadre, M. K.. Comparative assessment and numerical simulation of boiler shell materials. *International Conference on Functional Material, Manufacturing and Performances* (2021).

6. Gopal, M., Ashutosh, & Mishra, R. . Static and dynamic analysis of riveted joint considering different Material. *International Conference on Advancement in Materials, Manufacturing and Energy* (2021).

7. Kumar, K. S., Selvakumar, P., Jagadeeshwari, R., Dharmaraj, M., Uvanshankar, K., & Yogeswaran, B.. Stress analysis of riveted and bolted joints using analytical and experimental approach. *Materials Today: Proceedings, 42*, 1091-1099 (2021).

8. Shankar, P., & Kumar, L. . Stress Analysis of Single Lap Riveted Joint for Leak Proof Applications by Adhesive Bonding Using Finite Element Method. *International Journal of Advanced Mechanical Engineering, 4*, 395-404 (2014).

9. Abid Muslim, F. K., & Esmail, E. L. . Computer Aided Design Of Rivets For Steam Boiler Shells. *Al-Qadisiya Journal For Engineering Sciences, 5*, 377-393 (2012).

10. Kondayya, D.. Structural and Thermal Analysis of Boiler Using Finite Element Analysis. *I PASJ I nternational Journal of M echanical Engineering (I I JM E), 4*, 1-5 (2016).

11. Balbudhe, S. W., & Zaveri, S.. Stress Analysis of riveted lap joint. *International Journal of Mechanical Engineering and Robotics Research*, 137-143 (2013).

12. More, D. A., Dhadave, A. R., Khutwad, J. S., Bhilare, S. S., & Saste, A. A. . Design and Analysis for Different Shapes of Rivet Joint. *Journal of Analysis and Computation (JAC)*, 1-10 (2019).

13. Bohdan, W., Paweł, O., & ANDRZEJ, M. . Monitoring of the stress state in the boiler drum using Finite Element Method. *Advanced Materials Research Vols 875-877, 875-877*, 1176-1182 (2014).

14. Dwivedi, N., Shrivastava, A., Kumar, V., & Nareliya, R. . Burst Pressure Assessment of Pressure Vessel Using Finite Element Analysis: A Review. *Journal of Pressure Vessel Technology, 135*, 044502-1 (2013).

15. Koksal, N. S., Kayapunar, A., & Cevik, M. . Fatigue Analysis Of A Notched Cantilever Beam Using Ansys Workbench. *Fourth International Conference on Mathematical and Computational Applications*, 111-118 (2013).

16. Drastiawati, N. S., & Soekrisno, R.. Failure Analysis on Reheater Tube Boiler Furnace Base on Tensile and Impact Test. *International Joint Conference on Science and Engineering (IJCSE 2020), 196*, 355-360 (2020).

17. Kumar. A, E., Krishna, G. S., Santosh, R., & Najaraju, M. . Fatigue Analysis of Pressure Vessels Using Various Stiffners. *International Journal of Civil Engineering and Technology (IJCIET), 8*, 1296-1304 (2017).

18. Shankar, S., Jain, P. S., Chandru, V., Gowda, K. P., & Jain, S. K. . Fatigue Analysis on Wing Structure. *3rd International Conference on "Advancements in Aeromechanical Materials for Manufacturing"*, 040013-1-040013-8 (2021).

19. Shrivastava, A., & kala, P. . Creep and Fatigue Analysis of Pressure Vessels. *International Journal of Recent Research Aspects, 4*, 129-133 (2017).

20. Fajri, A., Prabowo, R. A., Surojo, E., Imaduddin, F., Sohn, J. M., & Adiputra, R.. Validation and Verification Of Fatigue Assessment using FE Analysis: AStudy Case on the Notched Cantilever Beam. *IGF26 - 26th International Conference on Fracture and Structural Integrity, 33*, 11-18 (2021).

21. Khurmi, R., & Gupta, J. . *Machine Design*. Ram Nagar, New Delhi: Eurasia Publishing House (PVT) LTD (2007).

22. Drienovská, J., & Tvrda, K. Structural and Physical Aspects of Construction Engineering. *Elsevier Ltd, 190*, 459 - 463. (2017).

23. Katarina, T., & Jana, D. Analysis of Creep Effects in a Concrete Beam Using Various Software. *Key Engineering Material, 738*, 79-88. (2017).

24. Tran, K. B., Nguyen, Q. S., & Truong, T. T. Creep Analysis of Concrete Beams And Flat-Slabs by Using Finite Element Method. *Journal of Science and Engineering, 01*, 46-51. (2019).

25. Nguyen, V. P., Lee, S., & Choi, S. T. Effect of creep lifetime on geometric optimization of boiler tubes for thermal power plants. *Material at high Temperature*, 01-09. (2019).

26. I. Ashcroft, & Briskhan, P. Designing adhesive joints for fatigue and creep load conditions. *Woodhead Publishing Limited, 16*, 469-515. (2010).

27. Pandey, V., Singh, I., Mishra, B., Ahmad, S., Rao, A., & Kumar, V. Creep Crack Simulation using continuum damage mechanics and extended finite element method. *International Journal of Damage Mechanics, 28(I)*, 3-34. (2019).

28. Liu, D., Li, H., & Liu, Y. Numerical Simulation of Creep Damage and Life Prediction of Superalloy Turbine Blade. *Methematical Problem in Engineering, 2015*, 01-10. (2014).

29. Scattina, A., Roncato, D., Belingardi, G., & Martino, G. Investigation of creep phenomenon on composite material for bolt Connections. *Composite Structures, 134*, 378-383. (2015).

30. Masuyama, F. Creep degradation in welds of Mod.9Cr-1Mo steel. *International Journal of Pressure Vessels and Piping, 83*, 819-825. (2006).

31. Hyde,T.H., Becker,A.A., Sun, W., Williams, J.A. Finite-element creep damage analysis of P91 pipes. International Journal of Pressure Vessel and Pipes, 83, 853-863. (2006).

32. Ayyanar, C., Suresh, M., ArunRaj, P., & Rajmadhan, M. Modeling and Creep Strain Analysis of Aluminum Alloy 6061-T6 By Creep Tensile Test. *Materials Today, 5*, 14345-14354. (2018).

33. Hore, S., & Ghosh, R. Computer simulation of the high temperature creep behaviour of Cr–Mo steels. *Material Science and Engineering A, 528*, 6095-6102. (2011).

34. Masse, T., & Lejeail, Y. Creep behaviour and failure modelling of modified 9Cr1Mo steel. *Nuclear Engineering and Design, 246*, 220-232. (2012).

35. Duda, P., Felkowski, L., & Duda, A. An Analysis of Creep Phenomena in the Power Boiler Superheater. *MDPI, 892*, 01-12. (2015).

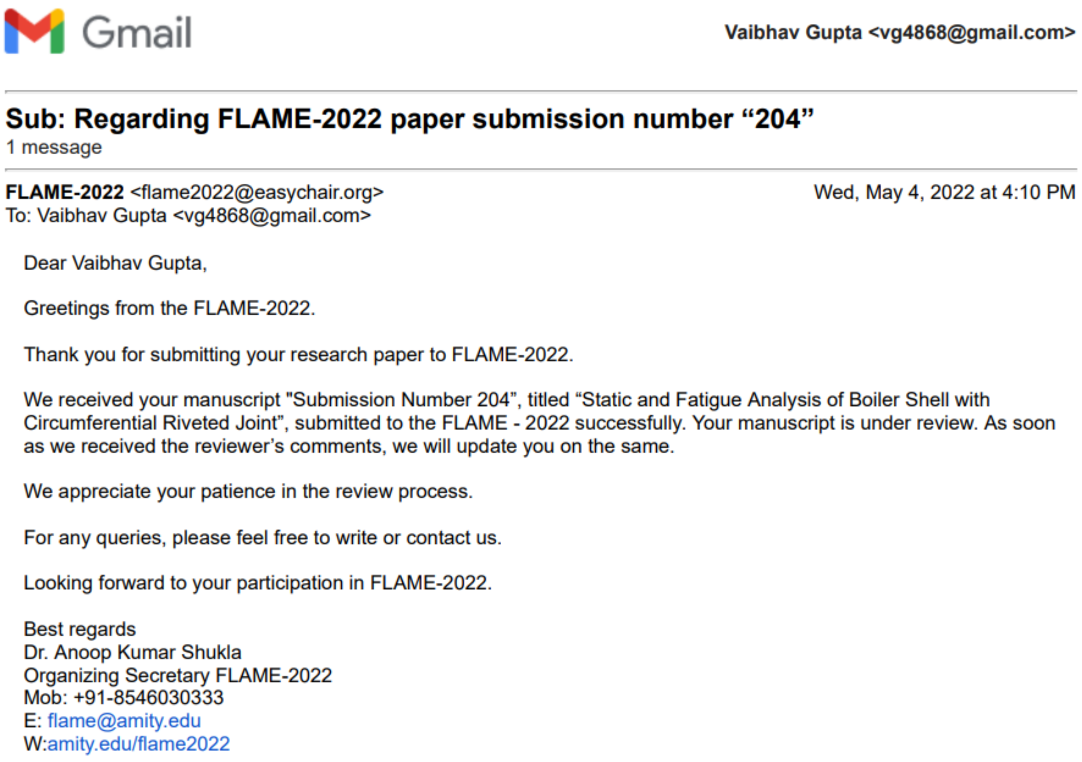
36. Bolton, J. Analysis of structures based on a characteristic-strain model of creep. *International Journal of Pressure Vessels and Piping, 85*, 108-116 (2008).

37. Khurmi, R., & Gupta, J. . *Machine Design.* Ram Nagar, New Delhi: Eurasia Publishing House (PVT) LTD (2007).

38. Fajri, A., Prabowo, R. A., Surojo, E., Imaduddin, F., Sohn, J. M., & Adiputra, R.. Validation and Verification Of Fatigue Assessment using FE Analysis: AStudy Case on the Notched Cantilever Beam. *IGF26 - 26th International Conference on Fracture and Structural Integrity, 33*, 11-18 (2021).

39. Bhansali, G., Singh, G., Kumar, S. R., & Wagadre, M. K.. Comparative assessment and numerical simulation of boiler shell materials. *International Conference on Functional Material, Manufacturing and Performances* (2021).

APPENDIX



Vaibhav Gupta <vg4868@gmail.com>

Sub: Regarding FLAME-2022 paper submission number “204”

1 message

FLAME-2022 <flame2022@easychair.org>
To: Vaibhav Gupta <vg4868@gmail.com>

Wed, May 4, 2022 at 4:10 PM

Dear Vaibhav Gupta,

Greetings from the FLAME-2022.

Thank you for submitting your research paper to FLAME-2022.

We received your manuscript "Submission Number 204”, titled “Static and Fatigue Analysis of Boiler Shell with Circumferential Riveted Joint”, submitted to the FLAME - 2022 successfully. Your manuscript is under review. As soon as we received the reviewer’s comments, we will update you on the same.

We appreciate your patience in the review process.

For any queries, please feel free to write or contact us.

Looking forward to your participation in FLAME-2022.

Best regards
Dr. Anoop Kumar Shukla
Organizing Secretary FLAME-2022
Mob: +91-8546030333
E: flame@amity.edu
W:amity.edu/flame2022