Tuijin Jishu/Journal of Propulsion Technology
ISSN: 1001-4055
Vol. 46 No. 4 (2025)

Failure Analysis and FEM Study of a Spline Plug in a High-Pressure Flow Regulation Valve

Ganesh Gawande[1], Nitin A. Kharche[2], Santosh R. Shekokar[3], Yugesh A. Kharche[4], Ajay Varma[5], Achal R Gole[6]

[1]*P.G. student, Department of Mechanical Engineering, Padm. Dr. V. B. Kolte College of Engineering, Malkapur, Maharashtra, India*

[2,3,4,5,6]*Department of Mechanical Engineering, Padm. Dr. V. B. Kolte College of Engineering, Malkapur, Maharashtra, India*

Abstract

In the offshore oil and gas business, valve failure is a big risk that can cost a lot of money. It leads to very bad things happening, like losing property or production because a company has to shut down, as well as health, safety, and environmental (HSE) problems like oil and gas spills, pollution, and even death. In the Norwegian offshore industry, different kinds of valve failures have happened for a number of reasons, including bad material choice, corrosion, high stresses and loads on valve parts, not coating them, and not inspecting them. This report talks about an SS 420 plug part that was broken while it was in use. It is a valve plug stem. It is necessary to look into the type of failure and its most likely cause. In the early stages of the study, it was thought that the failure might have been caused by a sudden impact on the stem. Since the part went through all of the NDT tests before it installed, there is almost no chance that the stem will break because of a flaw in the material. This work focuses on the results of a thorough analysis of failure that was done in line with the requirements. The investigation of failure plug will be carried by using finite element analysis (FEA). The FEA will include stress and modal analysis to investigate the actual failure regions and deformation in the vibration due to high pressure.

Keywords: Failure, Oil, and gas industry, FEA

Introduction

The offshore oil and gas industry plays a crucial role in global energy production, but it also faces significant risks, particularly in the area of equipment reliability. One of the most critical components in this industry is the flow regulation valve, which is essential for controlling high-pressure fluids. Failure of such valves can lead to catastrophic consequences, including production loss, environmental pollution, and even fatalities. The potential for such failures underscores the need for thorough investigation and analysis to prevent them. This project focuses on a failure investigation and finite element analysis (FEA) of a failed spline plug used in a high-pressure flow regulation valve. Specifically, the study investigates an SS 420 valve plug stem that fractured during operation. Given that the part passed all non-destructive testing (NDT) before installation, it is unlikely that the failure was due to material defects. Initial assessments suggest that the failure may have been caused by sudden impact or high stress acting on the stem.

Techniques of failure analysis

In study of any failure, the analyst must consider a broad spectrum of possibilities or reasons for the occurrence. Often a large number of factors, frequently interrelated, must be understood to determine the cause of the original, or primary, failure. The analyst is in the position of Sherlock Holmes attempting to solve a baffling case. Like the great detective, the analyst must carefully examine and evaluate all evidence available, then prepare a hypothesis—or possible chain of events—that could have caused the “crime”. The analyst may also be compared to a coroner performing an autopsy on a person who suffered an unnatural death, except that the failure analyst works on parts or assemblies that have had an unnatural or premature demise. If the failure can be duplicated under controlled simulated service conditions in the laboratory, much can be learned about how the failure actually occurred. If this is not possible, there may be factors about the service of the part or assembly that are not well understood. Fractures, usually the most serious type of failure, will be studied here in some detail. Usually undesired and unexpected by the user, fractures can have disastrous results when a load-bearing member suddenly loses its ability to carry its intended load. Distortion, wear, and corrosion failures also are important, and sometimes lead to fractures. However, these types of failure can be reasonably well predicted and prevented.

To comprehensively understand the root cause of the failure, this project will conduct a detailed finite element analysis. The FEA will include both stress and modal analysis to identify failure regions and deformation caused by high-pressure vibrations. The findings from this investigation will contribute to enhancing the reliability and safety of valve components in the offshore oil and gas industry. Figure 1 shows the high-pressure valve schematics along with internal plug.

In the high-pressure flow regulation valves used in critical industries like oil and gas, the failure of components such as the SS 410 plug stem poses significant operational challenges. The failure of this plug stem during service led to unexpected downtime, potentially jeopardizing safety and reliability. The plug stem, manufactured in-house (JP12682), was identified as the source of the problem after thorough review and inspection. Despite following standard procedures, the failure occurred, indicating the need for a comprehensive investigation into the root cause.

This research aims to address the critical need for understanding the causes of such failures in high-pressure environments. By utilizing advanced investigative techniques and modern tools such as ANSYS for stress distribution and modal analysis, the project will provide insights into the failure mechanisms. These findings will not only enhance the reliability of similar components in the future but also contribute to the development of more effective failure prevention strategies

The methodology for this research involves a systematic approach to investigating the failure of the SS 410 plug stem used in a high-pressure flow regulation valve. The following steps outline the process:

- Failure Analysis and Initial Inspection:
  The investigation begins with a detailed inspection of the failed SS 410 plug stem, which was identified as the valve plug stem (JP12682). Site images of the crack generation and failure are examined to understand the nature of the damage. A thorough review of the manufacturing process, material selection, and in-service conditions is conducted to gather initial insights.
- Application of Modern Tools:
  Modern tools and software, particularly ANSYS, are utilized to simulate the failure conditions and analyze the component under operational stresses. Stress distribution and modal analysis are conducted using finite element analysis (FEA) in ANSYS to identify critical stress points and understand how the failure occurred under high-pressure conditions.
- Cause Identification:
  Based on the results from the FEA and other investigative methods, the likely causes of the failure are determined. Factors such as high stress, improper material handling, or unforeseen operational conditions are considered as potential contributors to the failure.
- Recommendations and Preventive Measures:
  The final step involves developing recommendations to prevent similar failures in the future. This includes proposing design improvements, material selection changes, and enhanced inspection protocols to ensure the reliability of high-pressure flow regulation valves.

Literature Review

A variety of factors can contribute to ball valve failure including poor design (chemical compatibility, rated pressure/flow rate, etc.), faulty installation, and/or improper operation. Understanding valve failure mechanisms can provide useful information to designers and operators to improve valve reliability, which is essential for smooth plant operation and a safe working environment. Guidelines are available in literature to properly design a ball valve. Kelso et al. [7] measured the surface pressure distribution. Kerh et al. [8] employed finite element method to simulate transient interaction of fluid and structure in a control valve.

Kirk and Driskell [9], Similarly, Ota and Itasaka [12] measured the surface pressure distribution behind a blunt body to understand the structure of the recirculation behind a blunt body.

Pearson [13] have provided design guidelines to prevent failure of these valves. Understanding flow behavior inside the ball valve can help manufacturers improve design from failure a prevention perspective. Furthermore, computational fluid dynamics is being used to characterize the flow behavior and understand valve performance. Davis and Stewart [4] used FLUENT to investigate flows in globe valves. Huang and Kim [5] also used fluent to simulate turbulent flows in a butterfly valve. Seals are another vulnerable area in valves and can cause failure.

Ridha et al. [10] investigated the failure analysis of ball valve seals. They concluded that valve failure is caused by multiple mechanisms such as wear and plastic deformation of seals. Now days design changes in the mechanical systems are rapidly taking place. The changes in the small parameters in the system may affect the overall performance of the system which may also cause failure of the system. Following literatures summarizes discussion regarding failure and analysis of various mechanical systems.

Vera et. al [16] studied failure of Co–Cr casting hip resurfacing prosthesis. The presence of cracks in this prosthetic device result in a catastrophic fatigue failure in operation. The study shows that due to hot tearing in stem of hip resurfacing causes fracture during manufacturing.

Tawancy et. al [17] done analysis of corroded elbow section of carbon steel piping used for an oil–gas separator vessel. The corrosion is due to chlorination and sulfidation reactions. The chlorination and sulfidation reactions are associated with calcium chloride and hydrogen sulfide present in crude oil.

Upadhyay et. al [18] focuses Rolling Contact Fatigue (RCF) occurring due to cyclic stress during its operation. Due to vibration or sliding oscillation false Brinelling occurs which tends to damage bearing surface within a short period. It is suggested that bearing life need to be improved.

Pantazopoulos et. al [19] focuses on failure analysis of a machinable brass connector in a boiler unit installation. The analytical study is carried with the help of Visual examination, light and scanning electron microscopy coupled with local elemental energy dispersive X-ray spectroscopy. The study suggests that failure is due to progressive cracking, resulting into fatigue failure. The main suggested changes are change the alloy and quality assurance of tubing assembly process during installation.

Nauman A. Siddiqui et. al [20] has investigated the failure of bevel gears in an aircraft engine. The mode of failure was contact fatigue due to micro structural variations in the gear material. The excessive wear and removal of hardened case at driven gear teeth occurred by simultaneous rolling and sliding action of meshing teeth.

Souvik Das et. al [21] studied problem of Central bursting by means of metallurgical check.. In this study three broken wires which failed during production were investigated. The analysis shows that first two wires breaksdue to formation of hard and brittle phase and the third wire fails due to wrong drawing operations.

Suman Mukhopadhyay et. al [22] studied Premature failure of Heat Trace Tube used in blast furnace to carry waste hot gases. The failure is due corrosion which is caused due to reaction of sulfuric acid and moisture with tube material.

Pantazopoulos et. al [23] investigated low alloy steel welded pipes buried in the ground. Failure of pipes was not caused by tensile ductile loading but resulted from low ductility fracture in the weld, which also contains multiple intergranular secondary cracks. Random surface cracks or folds were found around the pipe. Analytical techniques such as Chemical analysis, visual inspection, and optical microscopy were used for the study.

Venkateswarlu et. al [24] investigated Failure analysis and optimization of thermo-mechanical process parameters of titanium alloy (Ti–6Al–4V) fasteners for aerospace applications. The titanium alloy (Ti–6Al–4V) fasteners is subjected to fatigue failure as Socket head hole piercing into shank. For optimizing strain rate experimentation is done. Metallurgical test with optimum process parameters shows no proof of heterogeneity in microstructure.

Delavar et. al [25] investigated cracking causes in ISOMAX unit such as reactor, valves tubings etc. The study is attributed by four parameters which may lead to failure and detailed investigation shows that the failure was due to the stress corrosion cracking (SCC) caused by the presence of chloride in the used anti-seize grease.

Ortiz et. al [26] studied spark plug failure due to combined effect of strong magnetic field and undesirable fuel additives. The magnetic field causes short circuit which in turn lead to inefficient combustion and deposition of soot on the insulator surface. On the other hand organ metallic anti-knocking agents present in low-grade fuel results into failure. The complete remedy for the problem is to action taken by government for imposing compliance with fuel composition norms.

Bhagi et. al [27] studied fracture of low pressure (LP) steam turbine blade of a 110 MW thermal power plant. These blades were made from chrome alloy steel X20Cr13 (Tempered martensitic stainless steel). The study consists of the visual examination, SEM fractography, chemical analysis, hardness measurement, and micro-structural characterization. The cause of failure is corrosion-fatigue.

El-Batahgy et. al [28] studied fatigue failure of thermowells in feed gas supply downstream pipeline at a natural gas production plant. Due to High flow velocity of the pipeline medium increased the wake frequency above the natural frequency of the used straight type thermowell. This results into a resonance where large amounts of energy is absorbed and high stresses are produced. The problem was solved by installing new modified truncated conical-type thermo wells.

Park and Park [29] investigated the influence of austenitizing treatments on the corrosion resistance of 14Cr-3Mo martensitic stainless steel. Their study revealed that variations in the austenitizing temperature significantly affect the microstructure and corrosion resistance. Higher temperatures led to grain growth and dissolution of precipitates, which improved the uniformity of the martensitic structure and reduced susceptibility to localized corrosion. The authors emphasized the importance of optimizing heat treatment parameters to enhance the corrosion performance of martensitic stainless steels in aggressive environments.

Garcia de Andres et al. [30] explored the role of carbide-forming elements on the thermal treatment response of X45Cr13 martensitic stainless steel. The study demonstrated that elements like vanadium and molybdenum influence carbide precipitation behavior during tempering, which in turn affects hardness and mechanical strength. The addition of these elements improved the tempering resistance and stability of the microstructure. This work provided insight into how alloying strategies could be used to tailor heat treatment responses for desired mechanical properties.

Nasery-Isfahany et al. [31] studied the effects of various heat treatment processes on the mechanical properties and corrosion behavior of AISI 420 martensitic stainless steel. Their findings indicated that while hardening improved strength and wear resistance, tempering at appropriate temperatures was critical to balance strength with corrosion resistance. Over-tempering led to carbide coarsening and increased susceptibility to corrosion, highlighting the need for precise control over thermal cycles in industrial applications.

Vander Voort and James [32] provided a comprehensive overview of the metallography and microstructural characteristics of wrought stainless steels in the ASM Handbook. They described the typical features of martensitic, ferritic, and austenitic stainless steels, along with heat treatment effects on grain structure, phase transformations, and mechanical properties. Their work serves as a foundational reference for understanding how processing variables influence the microstructure and performance of stainless steels in various applications.

Cihal [33] addressed the mechanisms and conditions leading to intergranular corrosion in steels and alloys. The study emphasized the role of chromium depletion at grain boundaries caused by carbide precipitation during improper heat treatments, which significantly compromises corrosion resistance. This phenomenon, especially relevant in stainless steels, underscores the importance of post-weld heat treatments and alloy design in mitigating intergranular corrosion risks.

Greeff and Du Toit [34] examined the sensitization behavior of 11–12% chromium EN 14003 stainless steel during welding. The research demonstrated that welding-induced thermal cycles can promote carbide precipitation and chromium depletion along grain boundaries, leading to sensitization and increased susceptibility to intergranular corrosion. They suggested that careful control of welding parameters and possible post-weld heat treatments are necessary to prevent degradation in corrosion resistance of such steels.

Bhambri [35] investigated intergranular fracture mechanisms in 13 wt.% chromium martensitic stainless steel. The study linked the occurrence of intergranular fractures to prior thermal exposures and microstructural changes such as carbide precipitation along grain boundaries. This brittle fracture mode was especially prominent after tempering treatments that caused significant chromium segregation. The research highlighted the delicate balance between heat treatment parameters and fracture behavior in martensitic stainless steels.

Failure Investigation

ASTM A275 Type SS 420 plug component identified as a valve plug stem, which was broken while in service. The details site images of crack generation and failure is as shown in Fig. 1.

Figure 1. Failed plug stem

To thoroughly understand the problem and investigate the likely cause of failure, a detailed study was conducted on the broken plug stem. Through inspection, the plug stem material was identified as stainless-steel grade SS420, a martensitic stainless steel known for its good corrosion resistance and high hardness, but also some brittleness, especially in high-stress applications. The plug in question, designated as JP12682, was manufactured in-house specifically to meet the unique requirements of this project. In this particular project, five valves in total were produced, each utilizing a similar design. However, out of these five valves, only the stem for this specific plug was manufactured in-house, while the remaining four were machined by external vendors. The failure point on the plug stem occurred at the threaded portion near the top, an area recognized as the weakest section due to the reduced cross-sectional area and stress concentration typically associated with threading.

Upon close inspection, the stem showed a slight bend, which indicated it may have been subjected to an excessive load or improper handling that resulted in bending stresses. Additionally, the run-out (the deviation from the true rotation axis of the part) was measured to be between 0.15 and 0.17 mm, a figure that could be indicative of misalignment or uneven machining, leading to uneven load distribution during operation. This run-out deviation, combined with the bending observed, suggests that the component might have been subjected to non-uniform or excessive forces. These forces, particularly if cyclic or dynamic in nature, could have weakened the stem further over time, leading to fatigue and eventual breakage at the thread. The investigation implies that the material properties of SS420, combined with possible machining inaccuracies or misalignment, may have contributed to the failure of the plug stem under operational loads.

Finite Element Analysis by Ansys

Finite Element Analysis (FEA) is an essential tool for understanding and addressing the failure mechanisms in components like the plug stem. By using FEA, engineers can simulate the complex stresses and loads that the plug stem experiences during operation, providing insights into areas of potential weakness without requiring physical testing, which can be costly and time-consuming. For the plug stem, FEA helps visualize stress concentrations, especially around the threaded portion where the actual failure occurred, which is challenging to observe with traditional inspection methods.

One of the key advantages of FEA is its ability to model complex geometries and loading conditions accurately. The plug stem has intricated features, such as threads, that can act as stress concentrators and are susceptible to failure. FEA allows engineers to apply precise boundary conditions, including the 1.3 MPa pressure on the exterior surface and the fixed constraints at the threaded end, to mimic real-world operational conditions. By doing so, FEA provides a comprehensive understanding of how different forces interact with the geometry of the plug stem, revealing weak points where stress accumulates. This is particularly valuable in predicting failure points and understanding the root cause of breakage. The importance of FEA in this context lies in its ability to evaluate material behaviour under various loads without physically testing each scenario. For example, using FEA, one can assess how SS420, the material of the plug stem, responds to different stress levels, shedding light on its brittleness under high stress or its potential to withstand cyclic loading. This predictive capability is essential for materials like martensitic stainless steels, which may exhibit brittle behaviour under stress. Additionally, FEA allows for the visualization of deformation patterns, such as bending or twisting, which were observed in the actual plug stem failure. By reproducing these deformations in a controlled virtual environment, FEA assists in confirming the observed failure modes and validating theoretical assumptions.

Another significant advantage of FEA is the ability to conduct parametric studies, where various conditions and material properties can be modified to observe different outcomes. For the plug stem, FEA can simulate changes in the material, load magnitude, or design modifications, allowing engineers to explore ways to strengthen the stem and avoid failure. This feature is crucial in assessing whether alternative materials or design changes could improve performance.

FEA also plays an instrumental role in optimizing design, as it enables iterative testing without requiring a physical prototype for each iteration. With FEA, engineers can make small adjustments to the plug stem design, such as altering the thread profile or adjusting the wall thickness, to reduce stress concentrations and minimize potential failure risks. This not only saves time and cost in the design phase but also leads to safer, more reliable products.

One of the most important benefits of FEA in failure analysis is its capability to predict the behaviour of the plug stem under both static and dynamic conditions. In real-world applications, components often face dynamic forces that vary over time, and FEA can help simulate these conditions. For example, if the plug stem is subject to cyclic loads, FEA can analyze fatigue behaviour, estimate the component's life span and identifying when and where failure is likely to occur.

In the case of the plug stem failure, FEA's ability to generate visual outputs, such as stress and deformation contours, provides a clear representation of critical areas. This visual data helps engineers and decision-makers to understand complex failure mechanisms more easily and make informed decisions regarding material selection, design improvements, and manufacturing adjustments. Moreover, FEA allows engineers to validate analytical results with experimental data, strengthening the reliability of the analysis. By comparing the FEA model results with the physical observations of the broken plug stem, such as the bending and run-out measurements, the accuracy of the model is confirmed, making FEA a reliable method for failure prediction.

Static stress analysis of plug stem

In the finite element analysis setup, specific static boundary conditions are applied to simulate the operational environment of the plug stem. A uniform pressure of 1.3 MPa is exerted on the exterior surface, representing the external loading conditions that the stem experiences. Furthermore, the threaded portion at the end of the plug stem is fully constrained, meaning it is fixed in all degrees of freedom. This constraint prevents any movement or rotation in that section, accurately replicating how the stem would be held in place during actual use. Figure 13 provides a visual representation of these boundary conditions, highlighting both the applied pressure and the fixed constraints on the threaded region.

Figure 2 shows the CAD geometry of pressure vessel. Figure 6 shows the mesh model with 14545 number of nodes and 7917 number of elements.

Tuijin Jishu/Journal of Propulsion Technology
ISSN: 1001-4055
Vol. 46 No. 4 (2025)

Different quality parameters are the measure of how far a given element deviates from the ideal shape. Following are the quality parameters to be checked.

- Element quality
- Aspect ratio
- Max corner angle
- Jacobian ratio

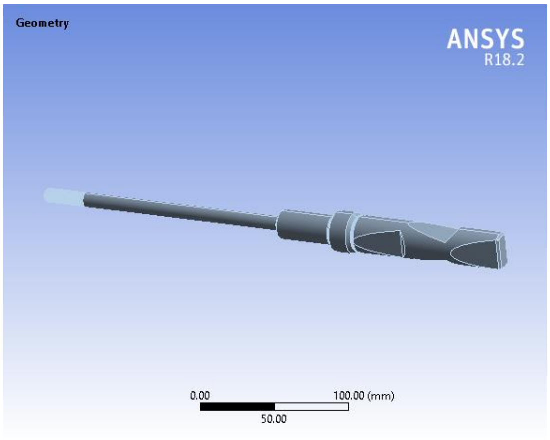

Figure 2. CAD model of stem

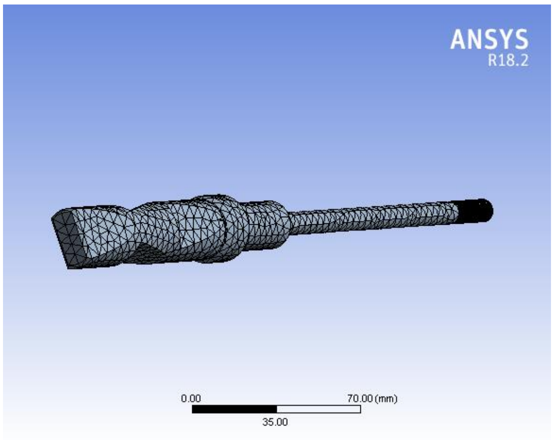

Figure 3. Mesh model

Results and Discussion

The analysis of the plug stem was carried using analysis tool based on finite element method. In the further section, detailed results and discussion were presented.

Static stress analysis of plug stem under the loadings of pressure

Static boundary conditions are mention below-

- Pressure on exterior surface is 1.3 Mpa.
- The end of plug stem i.e. threaded portion is fixed in all degrees of freedom as shown in Figure 4.

Figure 2 shows the CAD geometry of pressure vessel. Figure 3 shows the mesh model with 14545 number of nodes and 7917 number of elements.

Static analysis

Static structural analysis determines the stress, strain and deformation of a component or assembly can be investigated under a range of load conditions to ensure that expensive failures are avoided at the design stage. Static stress analysis is arguably the most common type of structural analysis using FE method. Structural loads are typically one, or a combination of the following:

- External forces such as clamping force in subsea connectors.
- Surface loads, e.g. pressure loading in pressure vessels
- Body forces (gravity, acceleration such as centrifugal force in rotating machines)

The structural response to more complex loads, for instance those arising from thermal analysis, can also be simulated using the multi-physics approach.

Tuijin Jishu/Journal of Propulsion Technology
ISSN: 1001-4055
Vol. 46 No. 4 (2025)

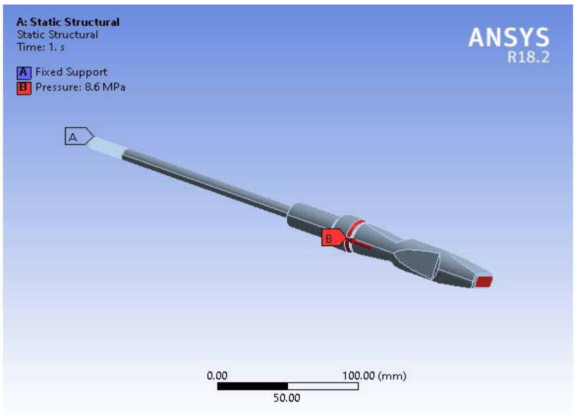

Figure 4 Static condition

Total deformation gives a maximum and minimum displacement occurs in model as shown in Figure 5. As per the given material and boundary condition it is observed that deformation (displacement) in stud is smaller. The maximum deformation is 0.050 mm occurred at the end portion of plug stem and minimum deformation 0 mm occurs at the treaded portion. Maximum deformation having smaller value for given boundary condition. So, the plug stem model is safe in total deformation (displacement).

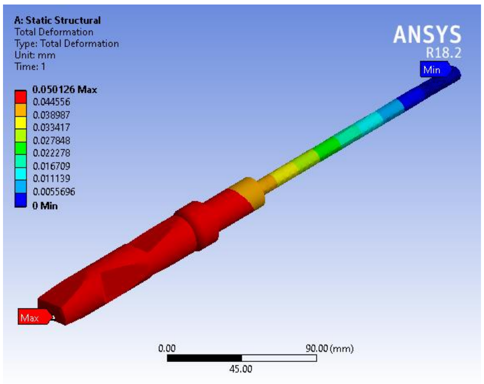

Figure 5 Total deformation

The equivalent stresses i.e Von Mises Stress is observed within stress limit in structural analysis. For the pressure in axial direction of 8.6 Mpa the plug stem is found in safe zone in stress analysis. Maximum Stress occur in the stud is 72.225 Mpa. So, plug stem is safe for given material as shown in Figure 6 and Figure 7.

The maximum principal stress was found to be 32.10 Mpa, it is within the permissible limit. It is observed that the current analysis results are within safe zone of the allowable stress. This proves that the current FE model is not accurate for saddles fixed in all degrees of freedom. The summary of all results is mentioned in Table 1. The deformation in the vessel is 0.050 mm which is very small for given boundary conditions.

Table 1 Results of FEA

Total deformation (mm)	Equivalent stress (Mpa)	Maximum principal stress (Mpa)
0.050	72.25	32.10

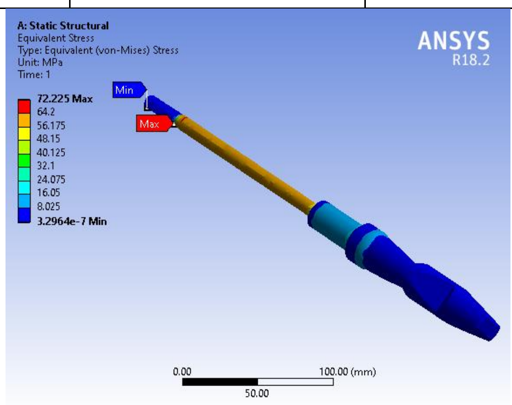

Figure 6 Equivalent stress

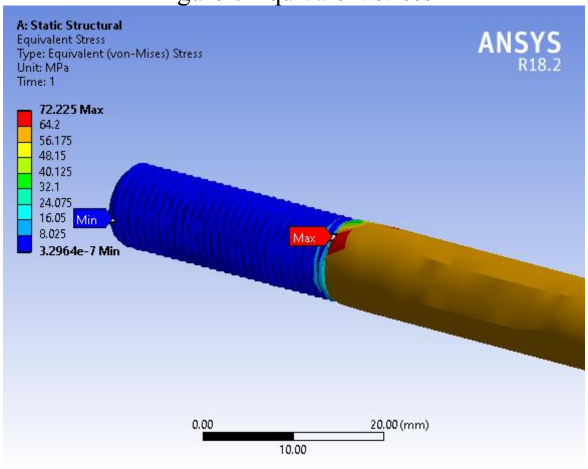

Figure 7 Equivalent stress (Maximum stress zone)

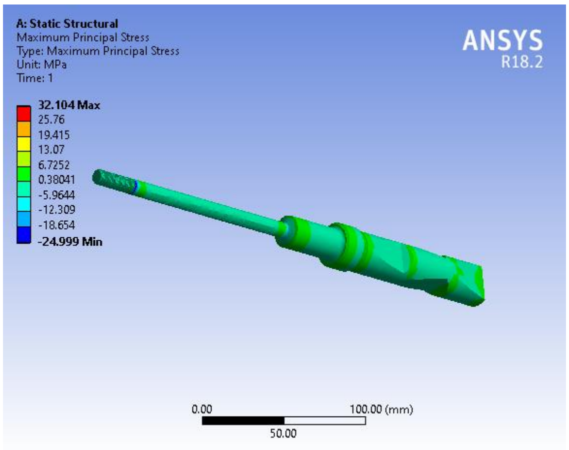

Figure 8 Max principal stress

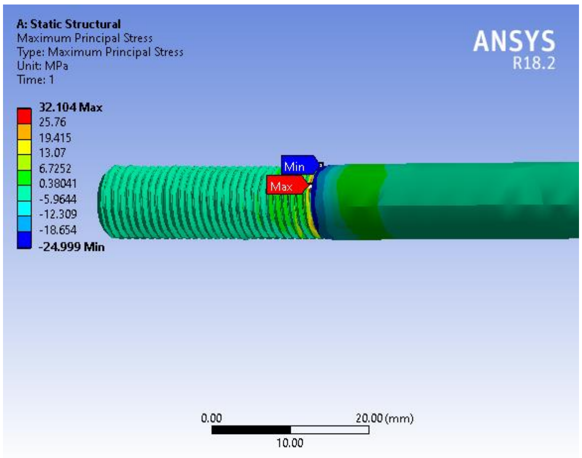

Figure 9 Max principal stress (maximum stress zone)

Modal Analysis

Modal analysis is a powerful tool to identify the dynamic characteristics of structures. Every structure vibrates with high amplitude of vibration at its resonant frequency. It is imperative to know the modal parameters- resonant frequency, mode shape and damping characteristics of the structure at its varying operating conditions for improving its strength and reliability at the design stage. The theoretical modal analysis technique has also been investigated using finite element method (FEM). Modal analysis has been performed after creating the pressure vessel finite element model and meshing in free-free state and with no constraints. The results have been calculated for the first 6 frequency modes. In this analysis we have made use of subspace method in ANSYS. The three modes are related to the stem plug displacement in x, y and z directions and three modes are related to plug stem rotation about x, y and z axes. In Figure 8-12 related natural frequencies and mode shapes for plug stem with maximum displacement in each mode have been shown. The first, modes frequency shows principal frequency called as natural frequency of vibration.

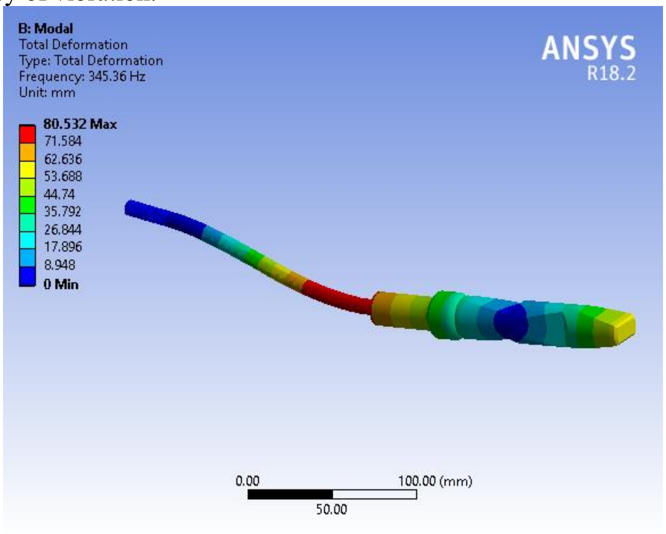

Figure 10 Mode shape 1

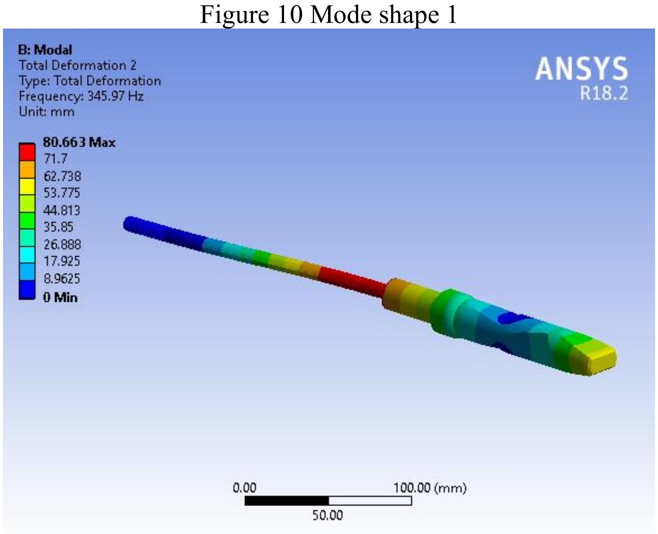

Figure 11 Mode shape 2

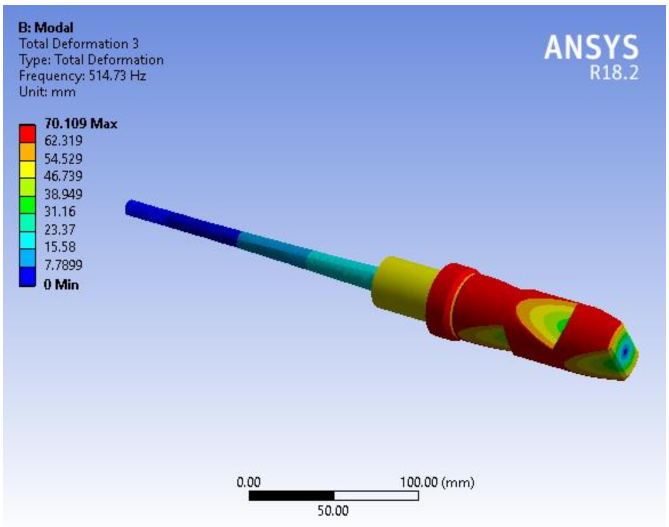

Figure 12 Mode shape 3

Tuijin Jishu/Journal of Propulsion Technology
ISSN: 1001-4055
Vol. 46 No. 4 (2025)

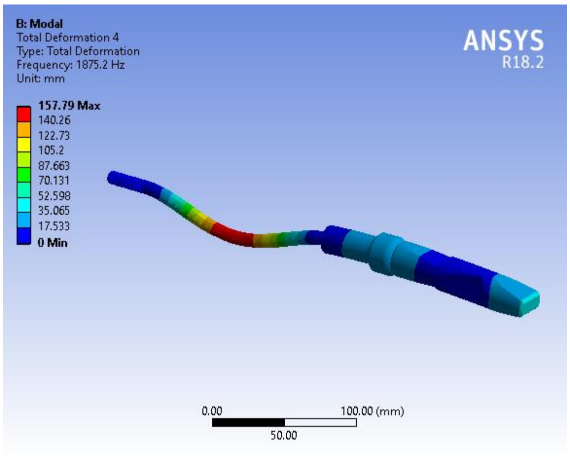

Figure 13 Mode shape 4

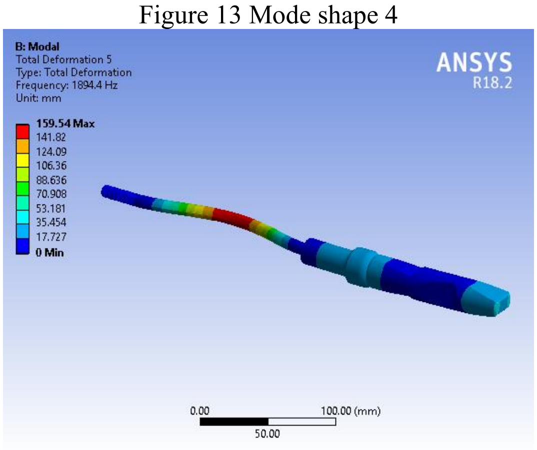

Figure 14 Mode shape 5

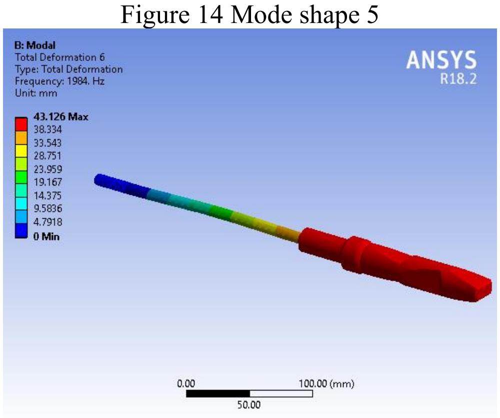

Figure 15 Shape 6

Results of modal analysis

The first mode shape with 345.36 Hz frequency as a principal frequency of vibration. The second mode is a vertical bending at 345.97 Hz. The third and fourth modes are localized bending modes at 514.73 Hz and 1875.2 Hz. The plug stem also experienced big translation at fifth mode which is a localized mode. The sixth mode is the torsion mode at 1984 Hz with maximum translation at bottom end of the plug stem.

Table 2 Mode shape number and corresponding natural frequency

Mode	Frequency [Hz]
1.	345.36
2.	345.97
3.	514.73
4.	1875.2
5.	1894.4
6.	1984

Conclusion

This study addresses the failure investigation and finite element analysis (FEA) of a spline plug, designated as JP12682, utilized in a high-pressure flow regulation valve. The plug, which was manufactured in-house to meet specific project requirements, experienced a stem breakage during operation. Upon inspection, the plug stem was found to be made of stainless steel (SS410), a material known for its hardness and corrosion resistance, though prone to brittleness under high stress. The purpose of this investigation is to understand the nature and cause of the failure, focusing on the weak areas that led to the breakage. The primary objectives of this investigation include studying and applying various failure analysis approaches, evaluating modern tools and techniques used in failure investigations, and pinpointing the root causes of the plug stem's failure. To accomplish these objectives, a combination of physical inspection, material analysis, and FEA is employed. The ANSYS software is used to perform a detailed stress distribution and modal analysis, replicating the operational conditions to identify areas of high stress concentration and determine the component's vibrational characteristics under load. This combined approach will provide insights into potential design or material weaknesses, enabling the identification of preventive measures and improvements to enhance the reliability and durability of similar high-pressure valve components. Static stress analysis revealed a maximum von Mises stress of 72.25 MPa, localized at the threaded region, confirming it as the critical failure zone. Total deformation was minimal (0.050 mm), indicating the failure was primarily stress-driven rather than deformation-induced. Modal analysis identified natural frequencies (e.g., 345.36 Hz for the first mode), suggesting potential resonance risks under dynamic loads.

References

1. Nesbitt Brian, Handbook of valves and actuators: valves manual international, 1st edn. (Elsevier, Oxford, 2007)
2. P. Smit, R.W. Zappe, Valve selection handbook, 5th edn. (Elsevier, New York, 2004)
3. K. Sotoodeh, Optimized valve stem design in oil and gas industry to minimize major failures. J. Fail. Anal. Prev. (2020). https://doi. org/10.1007/s11668-020-00891-0
4. K. Sotoodeh, Valve failures, analysis and solutions. Valve World 23(11), 48–52 (2018)
5. Chendong Huang, Rhyn H. Kim, Three-dimensional analysis of partially open butterfly valve flows, J. Fluids Eng. 118 (3) (1996) 562 sep.
6. Inc, ANSYS FLUENT User Guide Manual, (2015).
7. R.M. Kelso, T.T. Lim, A.E. Perry, The effect of forcing on the time-averaged structure of the flow past a surface-mounted bluff plate, J. Wind Eng. Ind. Aerodyn. 49 (1–3) (1993) 217–226 Dec.
8. Tienfuan Kerh, J.J. Lee, L.C. Wellford, Transient fluid-structure interaction in a control valve, J. Fluids Eng. 119 (2) (1997) 354 jun.
9. M.J. Kirik, L.R. Driskell, Flow Manual for Quarter-Turn Valves, (1986).
10. Ridha Mnif, Mourad Chokri, Ben Jemaa, Riadh Elleuch, Tribological failure analysis of ball valve seals Tribological failure analysis of ball valve seals, Tech. Rep. 5 (2013).
11. S.F. Moujaes, R. Jagan, 3D CFD predictions, and experimental comparisons of pressure drop in a ball. EBSCOhost, J. Energ. Eng. 134 (1) (2008).
12. Terukazu Ota, Masaaki Itasaka, A separated and reattached flow on a blunt flat plate, J. Fluids Eng. 98 (1) (1976) 79 Mar.
13. G.H. Pearson, Valve Design, Mech. Eng. Publ, London, 1978.
14. Hunter Vegas, Gregory McMillian, Why You Should Not Use On-Off Valves as Control Valves, (2012).
15. Yaofu Zhang, Dezincification, and Brass Lead Leaching in Premise Plumbing Systems: Effects of Alloy, Physical Conditions and Water Chemistry, PhD thesis, Virginia Polytechnic Institute, 2009.
16. Alvarez-Vera, J.H. Garcia-Duarte, A. Juarez- Hernandez, R.D. Mercado-Solis, A.G. Castillo, M.A.L. Hernandez-Rodriguez "Failure analysis of Co–Cr hip resurfacing prosthesis during solidification ", Case Studies in Engineering Failure Analysis, Vol. 1, no.1, pp. 1-5, January, 2013.
17. H.M. Tawancy, Luai M. Al-Hadhrami, F.K. Al- Yousef "Analysis of corroded elbow section of carbon steel piping system of an oil–gas separator vessel",Case Studies in Engineering Failure Analysis, Vol. 1, no.1, pp. 6-14, January, 2013.
18. R.K. Upadhyay, L.A. Kumaraswamidhas, Md.Sikandar Azam "Rolling element bearing failure analysis: A case study",Case Studies in Engineering Failure Analysis, Vol. 1, no.1, pp. 15-17, January, 2013.
19. George A. Pantazopoulos, Anagnostis I. Toulfatzis"Failure analysis of a machinable brass connector in a boiler unit installation", Case Studies in Engineering Failure Analysis, Vol. 1, no.1, pp. 18- 23, January, 2013.
20. Nauman A. Siddiqui, K.M. Deen, M. Zubair Khan, R. Ahmad "Investigating the failure of bevel gears in an aircraft engine", Case Studies in Engineering Failure Analysis, Vol. 1, no.1, pp. 24-31, January, 2013.

21. Souvik Das, Jitendra Mathura, Tanmay Bhattacharyya, Sandip Bhattacharyya "Metallurgical investigation of different causes of center bursting led to wire breakage during production",Case Studies in Engineering Failure Analysis, Vol. 1, no.1, pp. 32-36, January, 2013.
22. Suman Mukhopadhyay, Piyas Palit, Souvik Das, Nilotpal Dey, Sandip "An analysis Premature failure of heat trace stainless tube", Case Studies in Engineering Failure Analysis, Vol. 1, no.1, pp. 37-42, January, 2013.
23. George Pantazopoulos, Athanasios Vazdirvanidis "Cracking of underground welded steel pipes caused by HAZ sensitization",Case Studies in Engineering Failure Analysis, Vol. 1, no.1, pp. 43-47, January, 2013.
24. Vartha Venkateswarlu, Debashish Tripathy, K. Rajagopal, K. Thomas Tharian, P.V. Venkitakrishnan "Failure analysis and optimization of thermo-mechanical process parameters of titanium alloy (Ti–6Al–4V) fasteners for aerospace applications",Case Studies in Engineering Failure Analysis, Vol. 1, no.2, pp. 49-60, January, 2013.
25. A.N. Delavar, M. Shayegani, A. Pasha "An investigation of cracking causes in an outlet RTJ flange in ISOMAX unit", Case Studies in Engineering Failure Analysis, Vol. 1, no.2, pp. 61-66, January, 2013.
26. Armando Ortiz, Jorge L. Romero, Ignacio Cueva, Víctor H. Jacobo, Rafael Schouwenaars"Spark plug failure due to a combination of strong magnetic fields and undesirable fuel additives", Case Studies in Engineering Failure Analysis, Vol. 1, no.2, pp. 67-71, January, 2013
27. Loveleen Kumar Bhagi, Pardeep Gupta, Vikas Rastogi"Fractographic investigations of the failure of L-1 low pressure steam turbine blade", Case Studies in Engineering Failure Analysis, Vol. 1, no.2, pp. 72-78, January, 2013.
28. Abdel-Monem El-Batahgy, Gamal Fathy "Fatigue failure of thermo wells in feed gas supply downstream pipeline at a natural gas production plant", Case Studies in Engineering Failure Analysis, Vol. 1, no.2, pp. 79-84, January, 2013.
29. J.Y. Park and Y.S. Park, Effects of Austenitizing Treatment on the Corrosion Resistance of 14Cr-3Mo Martensitic Stainless Steel, Corrosion, 2006, 62(6), p 541–54
30. C. Garcıa de Andre´s, L.F. Alvarez, V. Lopez, and J.A. Jimenez, Effects of Carbide-Forming Elements on the Response to Thermal Treatment of the X45Cr13 Martensitic Stainless Steel, J. Mater. Sci., 1998, 33, p 4095–410.
31. Nasery-Isfahany, H. Saghafian, and G. Borhani, The Effect of Heat Treatment on Mechanical Properties and Corrosion Behavior of AISI, 420 Martensitic Stainless Steel, J. Alloys Compd., 2011, 509(9), p 3931–3936
32. G.F. Vander Voort and H.M. James, Wrought Stainless Steels, in ASM Handbook—Metallography and Microstructures, Vol 9, Ohio, ASM International, 1992, p 279–296
33. V. Cihal, Intergranular Corrosion of Steels and Alloys, Mater. Sci. Monogr., 1984, 18, p 79–8
34. M.L. Greeff and M. du Toit, Looking at the Sensitization of 11-12% Chromium EN 14003 Stainless Steel During Welding, Weld. J., 2006, 85(11), p 243s–251s
35. S.K. Bhambri, Intergranular Fracture in 13 wt.% Chromium Martensitic Stainless Steel, J. Mater. Sci., 1986, 21, p 1741–1746