International Journal of Advances in Applied Sciences (IJAAS)
Vol. 14, No. 4, December 2025, pp. 1402~1410
ISSN: 2252-8814, DOI: 10.11591/ijaas.v14.i4.pp1402-1410

Design and analysis of a portal frame test rig for vertical load testing of goalpost pipeline support

Tsaqif Al Farrel Ghazali[1], Mohd Shukri Yob[1], Mohd Juzaila Abd Latif[2], Ojo Kurdi[3], Fudhail Abdul Munir[4]
[1]Laboratory of Applied Mechanical Design, Universiti Teknikal Malaysia Melaka, Melaka, Malaysia
[2]Faculty of Mechanical and Technology Engineering, Universiti Teknikal Malaysia Melaka, Melaka, Malaysia
[3]Department of Mechanical Engineering, Diponegoro University, Semarang, Indonesia
[4]Department of Mechanical Engineering, Universiti Teknologi Petronas, Perak, Malaysia

Article Info

Article history:

Received Mar 8, 2025
Revised Oct 18, 2025
Accepted Nov 7, 2025

Keywords:

Deflection
Finite element analysis
Goalpost pipeline support
Load test
Safety factor
Test rig

ABSTRACT

Pipe support is a crucial infrastructure in the oil and gas industry, requiring robust designs to withstand various loads and maintain operational stability. While numerical analysis is commonly used to assess the interaction between pipelines and supports, experimental testing remains essential for validation. However, field testing is often costly and difficult due to safety constraints. To overcome this, a reliable test rig with minimal deflection is needed to ensure accurate experimental results. This study uses finite element analysis (FEA) to evaluate both a goalpost pipeline support and a newly developed portal frame test rig. The test rig was analyzed under two conditions: the failure load of the goalpost support and an amplified load with a factor of 2.5 to simulate unexpected scenarios. Results show the test rig can safely withstand loads up to 40 kN, meeting the EN 1990 safety factor requirement of 1.5. Furthermore, critical components remained within the deflection limit specified by the British Constructional Steelwork Association (BCSA), which is under L/1,000 of the beam length. These results confirm the structural integrity and suitability of the portal frame test rig for accurate testing of the goalpost pipeline support structure.

Corresponding Author:

Mohd Shukri Yob
Laboratory of Applied Mechanical Design, Universiti Teknikal Malaysia Melaka
St. Hang Tuah Jaya, Durian Tunggal, 76100, Melaka, Malaysia
Email: mshukriy@utem.edu.my

1. INTRODUCTION

Pipe supports are steel structures widely implemented in the oil and gas industry, serving to support networks of pipelines as well as electrical and mechanical equipment [1]–[4]. As outlined by Kawade and Navale [3], these structures are designed to accommodate and stabilize critical infrastructure for the pipeline system, while Meshram and Prasad [4] highlighted the function of pipe racks in withstanding various loads from the piping system. Structurally, pipe supports consist of multiple frame grids interconnected by longitudinal beams [1], and their configuration can be varied from single to multi-leveled designs, depending on the operational requirements of the plant [2]. Ensuring the stability of the pipe racks is essential to mitigate potential accidents and failures that occur within the pipeline system. Some researchers have noted the importance of analyzing support structures within the piping system [5]–[9]. Then, numerous researchers performed experimental testing to analyze the strength, load capacity, and buckling strength of the pipe support structures subjected to the vertical or horizontal load [10]–[12]. These tests provided a detailed understanding of the load-carrying capacity and the influence of local slenderness on structural stability.

Development of a test rig is necessary because performing field tests is usually expensive and difficult to organize because of the need to safety criteria [13], [14]. Therefore, a robust test rig is necessary for performing experimental testing. A test rig must be designed to withstand loads sufficient to fail the specimen and minimize its own deflection to prevent inaccuracies in the measurements. Previati *et al*. [15] observed that neglecting the deflection of the test rig can lead to measurement errors of about 10-20%. Several studies about test rig development have explored various configurations based on the size and geometry of the specimen being tested [16]–[18].

Finite element analysis (FEA) is widely used to optimize the test rig and model stress distribution and deflection precisely [19]. Asyraf *et al*. [20] evaluated the design of a creep test rig by comparing safety factors and deflection. On the other hand, Kondayya [21] explored the design of test rigs, prioritizing an optimal stiffness-to-weight ratio.

Several studies have further examined test rig design in relation to safety factor and allowable deflection. The research by Ramly *et al*. [22] developed a wing box static test rig with a safety factor of 2.5 and a deflection of 1 mm, while Raj *et al*. [23] analyzed a ladder frame test rig, achieving a safety factor of 1.25 and deflection targets of 1.66% between the test rig and specimen. Also, Dell'Orto *et al*. [24] and Shinde *et al*. [25] advanced new test rig designs, ensuring that the stress of each component remained under yield strength with deflection less than 0.5 mm.

Several standards also specified minimum safety factors and allowable deflection: EN 1990 [26] and Poutanen *et al.* [27]. Meanwhile, for allowable deflection, the British Constructional Steelwork Association (BCSA) [28] specified a deflection limit of test rig is L/1,000, where L is the length of the beam. Other than that, according to ASTM D790 [29], the total elastic deflection of the system does not exceed 1% of the total deflection of the test specimen during testing.

In this research, both the specimen and the test rig are modeled using computer-aided three-dimensional interactive application (CATIA), with safety factors and deflection analysis performed through FEA. These results aim to verify the capacity of the portal frame test rig to withstand the required load to fail the specimen and to ensure that the deflection of the test rig does not affect experimental accuracy.

2. METHOD

The flowchart in Figure 1 outlines the design and analysis process of the portal frame test rig. First, a preliminary FEA was performed to obtain the maximum load that would fail the goalpost pipe support. Following this, the portal frame test rig was evaluated using maximum load to fail the goalpost pipe support to assess both safety factors and allowable deflection. Once all criteria were satisfied, both the goalpost pipeline support and the portal frame test rig were deemed ready for fabrication and experimental testing.

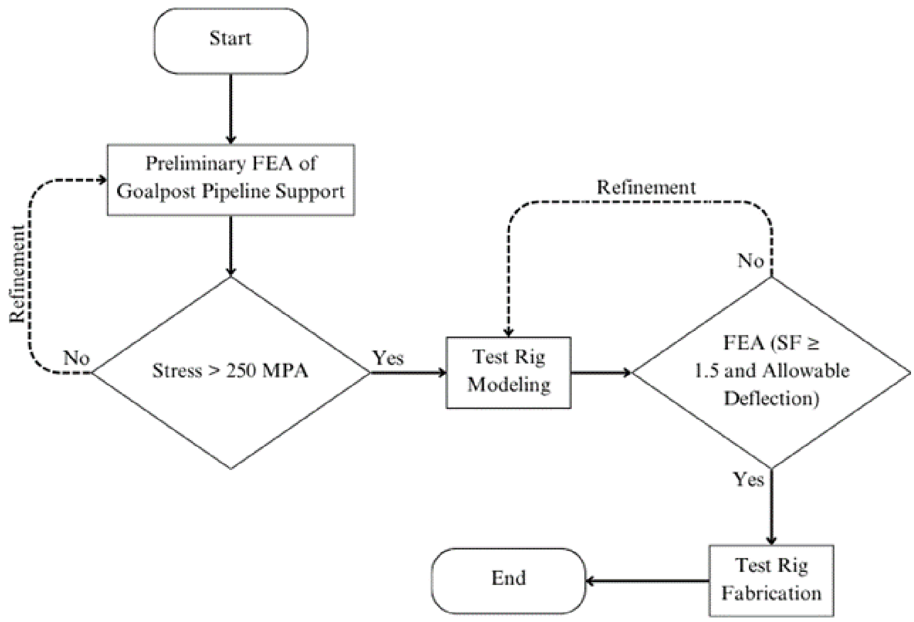

Figure 1. Research flowchart of portal frame test rig development

2.1. Material selection

In this research, mild steel material, which is ASTM A36, was used for the goalpost pipeline support and the test rig. The material was used because its availability in the market and its wide applicated in several industries, particularly that using steel structures. The material properties of mild steel can be seen in Table 1.

Table 1. Material properties of ASTM A36

Modulus elasticity, E (GPa)	Yield strength, σ_y (MPa)
200	250

2.2. Goalpost pipeline support model

The model of the goalpost pipeline support, as illustrated in Figure 2, consisted of a column and a transverse beam. The column was made from a 152×152 mm I-beam with a length of 500 mm, and the transverse beam was made from a built-up I-beam of 100×100 mm with a thickness of 2 mm and a length of 1,200 mm. In addition, endplates were welded to the transverse beam for to column connection using bolting.

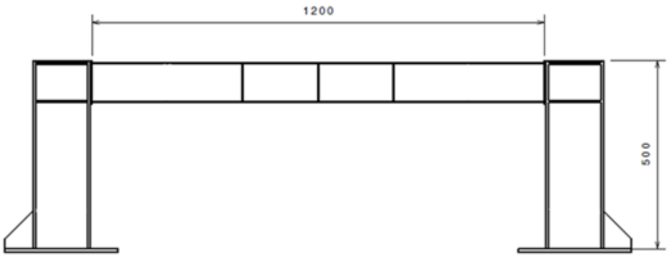

Figure 2. Model of goalpost pipeline support

2.3. Preliminary FEA of goalpost pipeline support model

Preliminary FEA was performed to estimate the maximum load to fail the goalpost pipeline support. Then, the maximum load data would be used to develop the test rig, evaluating the safety and allowable deflection of the test rig structure. The number of elements in this research is about 1129851 and 2013826 nodes. Other than that, the boundary condition of the goalpost pipeline support model can be seen in Figure 3. The boundary condition was fixed support at the baseplate, while the load at the stiffener surfaces was applied as a loading condition. After that, various load values were applied to determine the maximum load to fail the goalpost pipeline support. The applied load was initiated with 5 kN, then the load was increased by 5 kN increments until the stress on the pipe support was above the yield strength of its material. Based on Table 2, the stress of 252.8 MPa occurred in the goalpost pipeline support when the load was about 15 kN. The value of stress exceeds the selected material yield strength of 250 MPa, indicating the goalpost pipeline support will fail under a load of 15 kN.

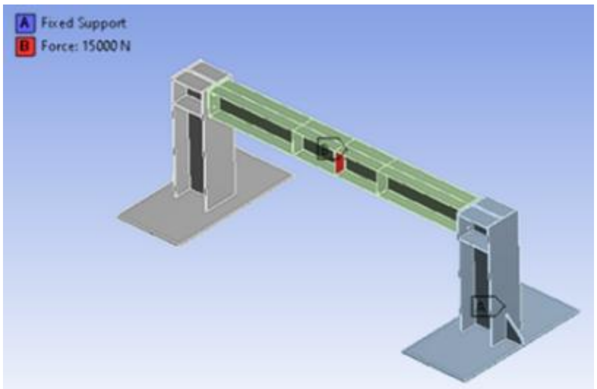

Figure 3. Boundary condition of the goalpost pipeline support model

Table 2. The stress resulting from the goalpost pipeline support

Force, F (kN)	Stress, σ (MPa)
5	84.2
10	168.5
15	252.8

2.4. Portal frame test rig design

The portal frame test rig design, which is shown in Figure 4, consists of four main components. First, the column to support the beam of the test rig. The column was made of a 152×152 mm I-beam, which has multiple bolting holes for beam-to-column connection. These multiple holes allow for adjusting the elevation of the beam. Also, there was a baseplate that was welded to the column as a support for the strong floor.

Second, a 2,000 mm length beam that would be attached to the column using a bolting connection. The beam of the portal frame also supported the hydraulic cylinder while testing was performed. The beam was made of a 152×152 mm I-beam and had stiffeners to strengthen the beam and prevent local failure. At the end of the beam, there were endplates that would be used to connect the beam to the column using a bolting connection.

Third, the load distributor that will transfer the applied load evenly from the hydraulic cylinder to the specimen. The load distributor was made of a 152×152 mm I-beam, and there were two attachments that allowed the load distributor to attach the load cells during testing. Also, stiffener plates were applied to prevent the load distributor from locally failing when the testing was conducted. Lastly, a flat bar was used to prevent unnecessary movement of the portal frame and goalpost pipeline support during the concentrated load test. The flat bar had holes for a lead screw to lock the portal frame and goalpost pipeline support when testing was conducted.

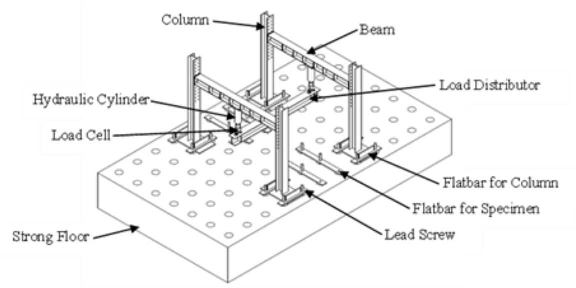

Figure 4. Portal frame test rig design

2.5. FEA of portal frame test rig

Once the test rig for load test was designed, then FEA was performed to evaluate the safety factor and allowable deflection of the test rig for load test. The load would be applied to evaluate the designed test rig according to the required load to fail the goalpost pipeline support. Each component of the test rig would be evaluated with different loads in accordance with the load distribution that occurred in the test rig structure. The load distribution of the test rig for the concentrated load test can be seen in Figure 5. In addition, the mesh size of 5 mm was used for each component of the test rig.

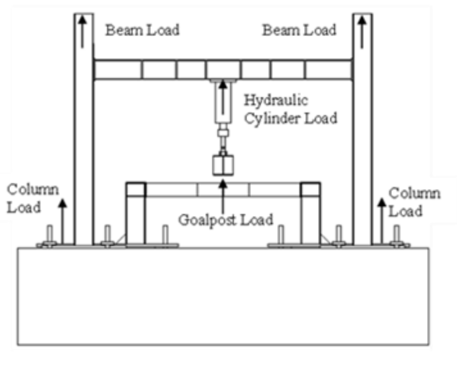

Figure 5. Load distribution of the portal frame test rig

3. RESULTS AND DISCUSSION

3.1. Safety factor of test rig

The stress of the test rig component was used to evaluate its safety factor to avoid test rig failure during testing in accordance with EN 1990 [26]. The minimum safety factor of the structure must be 1.5, or the maximum stress is about 166.6 MPa. According to Table 3, all the components of the test rig produced the stress under 166.6 MPa, or the safety factor was more than 1.5 under the required load to fail the goalpost pipeline support. The load distributor produced the highest stress among the other components, which was 81.8 MPa. The beam of the test rig showed lower stress around 29.3 MPa. Moreover, the column and flat bar had the lowest stress around 9.5 MPa. The stress and safety factors of each component are shown in Figure 6, where Figure 6(a) shows the load distributor, Figure 6(b) shows the beam, Figure 6(c) shows the column, and Figure 6(d) shows the flat bar.

Table 3. Safety factor of portal frame test rig component under the load to fail the goalpost pipeline support

Component	Yield stress of material, σ_y (MPa)	Stress, σ (MPa)	Safety factor
Load distributor	250	81.8	3.0
Beam		29.3	8.5
Column		9.5	26.2
Flat bar		9.4	26.3

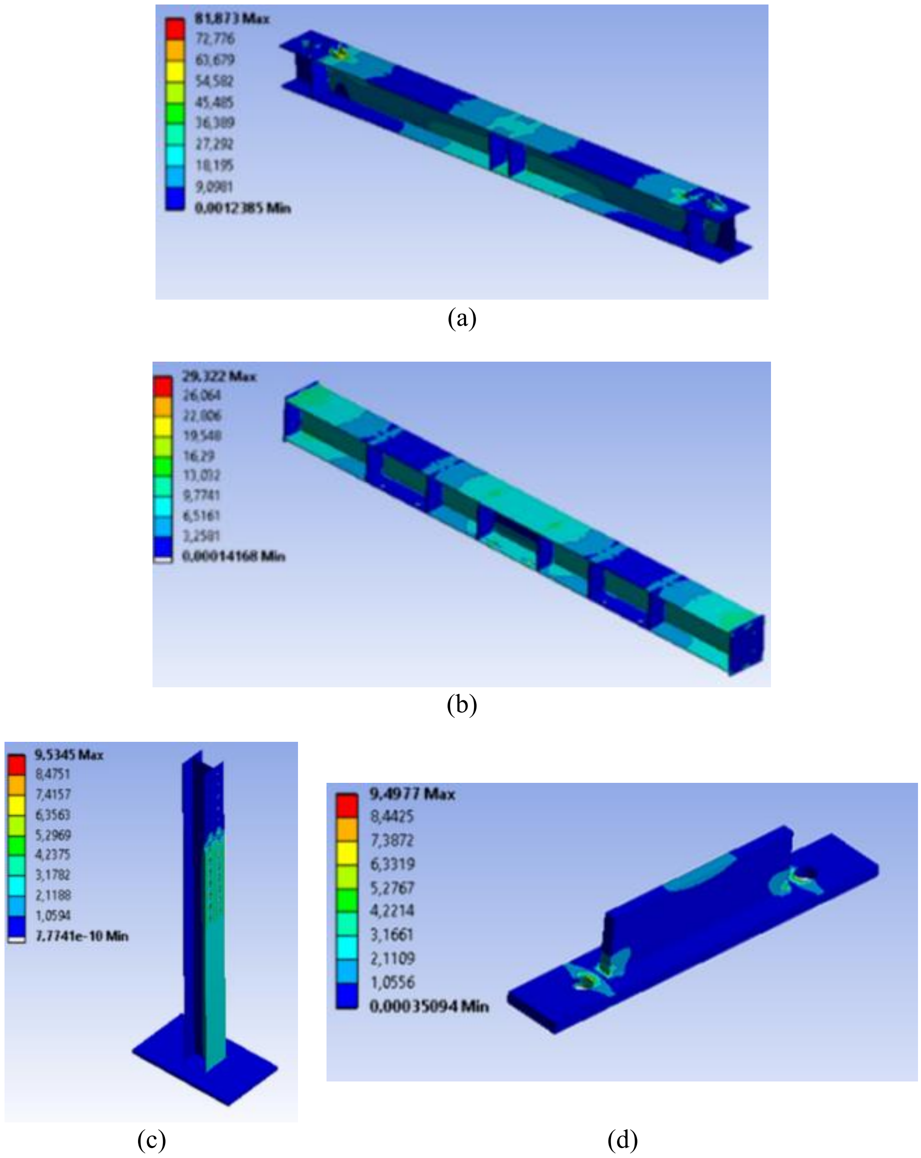

Figure 6. Stress of each test rig component of (a) load distributor, (b) beam, (c) column, and (d) flat bar

An FEA of the test rig component was also conducted using a load factor of 2.5 to prevent uncertainties in the experiment. Hence, a load of approximately 40 kN would be applied to evaluate the safety factor of the test rig structure. As shown in Table 4, the stress result of each test rig component with a 2.5 load factor complied with the requirements of EN 1990. Compared with similar test rigs reported in the literature, such as the large-scale bending and torsional test rig by Hamid *et al*. [16] and the ladder frame test rig by Raj *et al*. [23], the developed portal frame test rig demonstrates sufficient strength and suitability for experimental testing. These results indicate a robust and safe design, ensuring that the structure maintains its integrity and does not fail during testing.

Table 4. Safety factor of the portal frame test rig component with a 2.5 load factor

Component	Yield stress of material, σ_y (MPa)	Stress, σ (MPa)	Safety factor
Load distributor	250	163.7	1.5
Beam		77.7	3.2
Column		15.8	15
Flat bar		23.6	10.5

3.2. Deflection of test rig

The deflection of the portal frame was analyzed to ensure that the critical components deflect within the specified limits when testing was conducted. The deflection of the columns and flat bars was considered negligible, as these components have minimal influence on the test results. In contrast, the beam and load distributor were identified as critical components that could affect the testing result. According to the BCSA [28], the allowable deflection for a beam was limited to L/1,000, where L is the beam length. Table 5 summarizes the deflection of these critical components to fail the goalpost pipeline support and with a load factor of 2.5. Both deflection values were within the allowable limits, indicating that the test rig deflection complied with the BCSA criteria.

Table 5. Allowable deflection of the portal frame test rig component

Component	Length of beam, l (mm)	Allowable deformation, δs (mm)	Deformation, δ (mm)	Deformation with load factor, δLF (mm)
Load distributor	2100	2.1	0.4	2.0
Beam	2000	2	0.1	0.5

After the evaluation of the test rig's safety and deflection, the portal frame test rig and the specimen of the goalpost pipeline support were finally fabricated and set for the experiment, as shown in Figure 7. Goalpost pipeline support would be placed on the test rig and detained using flat bars. The load cells were placed between hydraulic cylinders and a load distributor using bolting. The data logger was placed in a location that was easily visible to the tester to enable viewing of load readings during the load test.

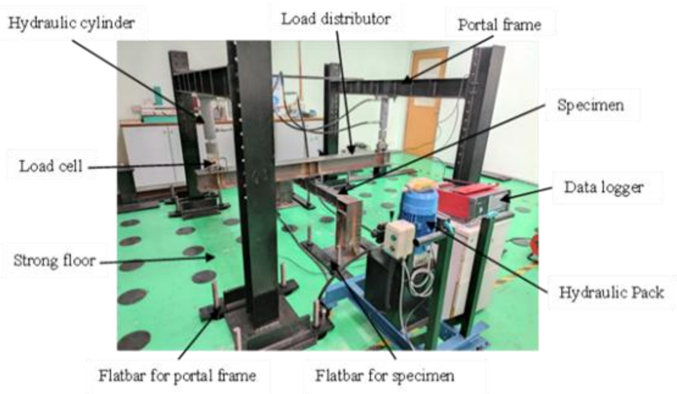

Figure 7. Testing setup for vertical load testing of the goalpost pipeline support

4. CONCLUSION

The portal frame test rig was evaluated using FEA to determine the safety factors and deflection in performing testing. This portal frame test rig can be used to withstand a load of up to 40 kN to perform the testing according to EN 1990. In addition, the deflection of the critical components of the portal frame test rig is within the allowable deflection according to BCSA. It is recommended to use a larger size I-beam or add reinforcement to increase the load capacity and reduce the error due to the deformation that occurred on the portal frame. For future work, it is suggested to explore the test rig under dynamic loading, a test rig with multi-point load, and the fatigue of the test rig to further enhance the performance of the test rig.

ACKNOWLEDGMENTS

In this journal, a portal frame test rig was evaluated in the framework of a research study at the Faculty of Mechanical and Technology Engineering, Universiti Teknikal Malaysia Melaka (UTeM). The authors would like to express gratitude to the Applied Mechanical Design Laboratory for providing the facility. The authors gratefully acknowledge UTeM and Kesidang Scholarship for providing access to open-access articles, which supported this research.

FUNDING INFORMATION

The authors would like to express sincere gratitude to the industrial partner for their generous financial support through the research grant (INDUSTRI(MTUN) HRSB/2021FKM/I100052), which significantly contributed to the successful completion of this study.

AUTHOR CONTRIBUTIONS STATEMENT

This journal uses the Contributor Roles Taxonomy (CRediT) to recognize individual author contributions, reduce authorship disputes, and facilitate collaboration.

Name of Author	C	M	So	Va	Fo	I	R	D	O	E	Vi	Su	P	Fu
Tsaqif Al Farrel Ghazali	✓	✓	✓	✓	✓	✓		✓	✓	✓			✓	
Mohd Shukri Yob	✓	✓		✓	✓	✓		✓	✓	✓	✓	✓		✓
Mohd Juzaila Abd Latif	✓		✓	✓			✓			✓	✓	✓	✓	✓
Ojo Kurdi						✓	✓	✓		✓		✓		
Fudhail Abdul Munir			✓		✓		✓			✓	✓			

C : **C**onceptualization
M : **M**ethodology
So : **So**ftware
Va : **Va**lidation
Fo : **Fo**rmal analysis
I : **I**nvestigation
R : **R**esources
D : **D**ata Curation
O : Writing - **O**riginal Draft
E : Writing - Review & **E**diting
Vi : **Vi**sualization
Su : **Su**pervision
P : **P**roject administration
Fu : **Fu**nding acquisition

CONFLICT OF INTEREST STATEMENT

The authors declare that they have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper.

DATA AVAILABILITY

The data that support the findings of this study are available on request from the corresponding author, [MSY]. The data, which contains information that could compromise the privacy of research participants, is not publicly available due to certain restrictions.

REFERENCES

[1] R. M. Drake and R. J. Walter, "Design of structural steel pipe racks," *Engineering Journal*, vol. 47, no. 4, pp. 241–251, 2010, doi: 10.62913/engj.v47i4.991.

[2] O. Bedair, "Rational design of pipe racks used for oil sands and petrochemical facilities," *Practice Periodical on Structural Design and Construction*, vol. 20, no. 2, 2015, doi: 10.1061/(asce)sc.1943-5576.0000224.

[3] M. G. Kawade and A. V Navale, "Optimization of pipe rack by study of braced bay," *International Journal of Research in Engineering, Science and Management*, vol. 2, no. 2, pp. 451–455, 2019.

[4] P. D. Meshram and R. K. Prasad, "Design of optimum pipe rack for various bays," *International Research Journal of Modernization in Engineering Technology and Science*, vol. 3, no. 4, pp. 1935–1944, 2021.

[5] R. Saikia and J. Pathak, "Seismic response of steel braced pipe racks and technological platforms in oil refineries," *15th Symposium on Earthquake Engineering*, pp. 1–9, 2014.

[6] N. J. Singh and M. Ishtiyaque, "Optimized design and analysis of steel pipe racks for oil and gas industries as per international codes and standards," *International Journal of Research in Engineering and Technology*, vol. 05, no. 10, pp. 16–28, 2016, doi: 10.15623/ijret.2016.0510004.
[7] P. Zakian, B. Ordoubadi, and E. Alavi, "Optimal design of steel pipe rack structures using PSO, GWO, and IGWO algorithms," *Advances in Structural Engineering*, vol. 24, no. 11, pp. 2529–2541, Aug. 2021, doi: 10.1177/13694332211004116.
[8] K. A. Rajai and S. Kulkarni, "Comparative analysis of pipe rack structure with different bracing system," *International Research Journal of Modernization in Engineering Technology and Science*, vol. 4, no. 8, pp. 910–915, 2022.
[9] K. Avrithi and H. H. M. Kim, "Optimization of piping supports and supporting structure," *Journal of Pressure Vessel Technology, Transactions of the ASME*, vol. 139, no. 4, Aug. 2017, doi: 10.1115/1.4036144.
[10] E. Shirai *et al.*, "Inelastic seismic test of the small bore piping and support system (part 2: static failure test for piping support equipment)," *American Society of Mechanical Engineers, Pressure Vessels and Piping Division (Publication) PVP*, vol. 8, pp. 239–245, 2008, doi: 10.1115/pvp2008-61351.
[11] B. Zheng, B. Wu, M. Zhang, J. Wang, G. Shu, and Q. Jiang, "Pseudo-static test on stainless steel frame with high-strength bolted extended endplate joint," *Journal of Constructional Steel Research*, vol. 207, Aug. 2023, doi: 10.1016/j.jcsr.2023.107980.
[12] X. Yun, Y. Zhu, Z. Wang, and L. Gardner, "Benchmark tests on high strength steel frames," *Engineering Structures*, vol. 258, p. 114108, May 2022, doi: 10.1016/j.engstruct.2022.114108.
[13] H. Westermann *et al.*, "A new test rig for experimental studies of drillstring vibrations," *SPE Drilling and Completion*, vol. 30, no. 2, pp. 119–128, Jun. 2015, doi: 10.2118/176019-PA.
[14] S. J. Cox, A. Wang, C. Morison, P. Carels, R. Kelly, and O. G. Bewes, "A test rig to investigate slab track structures for controlling ground vibration," *Journal of Sound and Vibration*, vol. 293, no. 3–5, pp. 901–909, Jun. 2006, doi: 10.1016/j.jsv.2005.08.051.
[15] G. Previati, M. Gobbi, and G. Mastinu, "Measurement of the mass properties of rigid bodies by means of multi-filar pendulums–influence of test rig flexibility," *Mechanical Systems and Signal Processing*, vol. 121, pp. 31–43, Apr. 2019, doi: 10.1016/j.ymssp.2018.11.011.
[16] M. F. A. Hamid, M. S. Yob, J. S. Vishnu, and M. M. Izahar, "Development of large-scale bending and torsional test rig using strong floor anchorage system for structural analysis," *Journal of Physics: Conference Series*, vol. 2784, no. 1, Jun. 2024, doi: 10.1088/1742-6596/2784/1/012021.
[17] M. I. Rokhim, M. S. Yob, M. J. B. A. Latif, and F. B. A. Munir, "Design and fabrication of a three-point bending test rig to evaluate strength and stiffness of I-beam," *Universal Journal of Mechanical Engineering*, vol. 7, no. 6, pp. 380–385, Nov. 2019, doi: 10.13189/ujme.2019.070609.
[18] H. Seifi, A. R. Javan, X. Lin, and Y. M. Xie, "An innovative and inexpensive experimental setup for testing connections in gridshell structures," *Engineering Structures*, vol. 207, Mar. 2020, doi: 10.1016/j.engstruct.2020.110257.
[19] N. Swamy G and M. AE, "Design optimisation and fatigue analysis of stroke endurance test rig using finite element analysis," *International Journal of Engineering Research and Technology*, vol. 2, no. 8, pp. 1718–1725, 2013.
[20] M. R. M. Asyraf *et al.*, "Evaluation of design and simulation of creep test rig for full-scale crossarm structure," *Advances in Civil Engineering*, vol. 2020, no. 1, Jan. 2020, doi: 10.1155/2020/6980918.
[21] D. Kondayya, "Structural design of a load test rig and design evaluation using FEM," *International Journal of Innovative Research in Technology*, vol. 2, no. 9, pp. 1–6, 2016.
[22] R. Ramly, W. Kuntjoro, W. Wisnoe, and R. E. M. Nasir, "Design and analysis for development of a wing box static test RIG," *CSSR 2010-2010 International Conference on Science and Social Research*, pp. 113–117, 2010, doi: 10.1109/CSSR.2010.5773704.
[23] V. Y. Raj, M. S. Yob, M. J. A. Latif, O. Kurdi, M. S. Kassim, and M. M. Izhar, "Design and analysis of four-point bending test rig for ladder frame," *Journal of Applied Science and Engineering*, vol. 26, no. 3, pp. 323–329, 2023, doi: 10.6180/jase.202303_26(3).0003.
[24] G. Dell'Orto, F. M. Ballo, G. Mastinu, and M. Gobbi, "Bicycle tyres–development of a new test-rig to measure mechanical characteristics," *Measurement: Journal of the International Measurement Confederation*, vol. 202, Oct. 2022, doi: 10.1016/j.measurement.2022.111813.
[25] R. S. Shinde, S. B. Gharge, R. B. Shinde, and A. S. Shinde, "Design and analysis of hydro test rig for GGCB forged valve with 4 stations," *International Journal of Engineering Research and*, vol. 5, no. 04, 2016, doi: 10.17577/ijertv5is040730.
[26] Technical Committee CEN/TC 250 "*EN 1990: Eurocode-basis of structural design*," CEN National Members, 2002.
[27] T. Poutanen, T. Länsivaara, S. Pursiainen, J. Mäkinen, and O. Asp, "Calculation of safety factors of the eurocodes," *Applied Sciences*, vol. 11, no. 1, pp. 1–9, Dec. 2021, doi: 10.3390/app11010208.
[28] BCSA and SCI, "*National structural steelwork specification for building construction*," 5th ed, The British Contructional Steelwork Association Ltd., 2007.
[29] ASTM D790, "Test methods for flexural properties of unreinforced and reinforced plastics and electrical insulating materials," *ASTM International*, West Conshohocken, PA, pp. 1–12, Jul. 01, 2017, doi: 10.1520/D0790-17.

BIOGRAPHIES OF AUTHORS

Tsaqif Al Farrel Ghazali is currently a master's student in Mechanical Engineering at Universiti Teknikal Malaysia Melaka (UTeM). Also, He holds a bachelor's degree in Mechanical Engineering from Diponegoro University (2024). He can be contacted at email: al.farrel.ghazali16@gmail.com.

Mohd Shukri Yob has a Ph.D. degree in Mechanical Engineering from Universiti Teknologi Malaysia (UTM), Malaysia (2016), a master's degree from University Kebangsaan Malaysia (2010), and a bachelor's degree from Universiti Teknologi Malaysia (2007). His research interest includes structural engineering, stress analysis, and FEA. He can be contacted at email: mshukriy@utem.edu.my.

Mohd Juzaila Abd Latif holds a Ph.D. degree from the University of Leeds (2011), a master's degree from the University of Wales Swansea (2005), and a bachelor's degree from Plymouth Stata College, Plymouth. He can be contacted at email: juzaila@utem.edu.my.

Ojo Kurdi holds a Ph.D. from Universiti Teknologi Malaysia (UTM) (2016), a Master of Engineering from Bandung Institute of Technology (ITB) (2002), and a Bachelor of Engineering from Diponegoro University (1997). His research interests include material strength, failure analysis, statics, machine elements, and numerical methods, with a focus on stress analysis in electric buses and fiber optic sensors. He can be contacted at ojokurdi@ft.undip.ac.id.

Fudhail Abdul Munir holds a Ph.D. degree from Yamaguchi University, Japan (2016), a master's degree from Universiti Teknologi Malaysia (2009), and a bachelor's degree from the International Islamic University Malaysia. He can be contacted at email: fudhail.munir@utp.edu.my.