Finite Element Simulation and Analysis for the Design of a Pressure Vessel with Expansion Joint

Rafiq Rustam, Ahmad Faiz Ahmad Nazri, Md Rizal Md Tasliman, and Jamaluddin Mahmud

***Abstract*—At present, design and analysis of a pressure vessel is usually performed using a customise software. This study demonstrates the usefulness of general finite element simulation and analysis for the design of a pressure vessel with expansion joint. There are two case studies involved; where the first case study focuses on validating the finite element model. The deformation and stress distribution on the pressure vessel using finite element method is compared to analytical method and this can be used for validation purposes. The second case study involves finite element simulation and analysis for the design of a pressure vessel with expansion joint (horizontal vessel on saddle point). The deformation and stress distribution on the pressure vessel using finite element method is compared to previous design work. In both cases, the results are found to be in good agreement to the analytical and previous work respectively. Therefore, it could be concluded that finite element simulation and analysis could be a useful tool in designing a pressure vessel with expansion joint. Moreover, the knowledge gained from it has increased the understanding about the mechanics and design of pressure vessels with expansion joint.**

***Index Terms*—Finite element simulation, design and analysis, pressure vessel, stress distribution.**

I. Introduction

Pressure vessel is a closed container designed to hold gases or liquid at a pressure substantially different from ambient pressure [1]. There are six types of pressure vessel that is horizontal vessel on saddle support; vertical vessel on leg support; tall vertical tower; vertical reactor, spherical pressurized storage vessel and vertical vessel on lug support [2]. Pressure vessels are widely used in reactor technology, the chemical industry, marine and space engineering. They often operate under extreme condition, which is high and low temperatures and in high pressure. Advances made to the material used in their fabrication. Concrete and composite materials are becoming more frequently used compared to conventional steels. Modern design of pressure vessels might also lead to the development of supporting features. Even though it is not quite common in Malaysia, a feature namely below expansion joint is sometimes need to be considered in the pressure vessel design. The addition of a bellow expansion joint allows pressure balanced design. An expansion joint provides an equal and opposite reaction force to the pressure thrust; typically used to allow axial movement but not subject the piping system to the pressure thrust force. Merrick et al stated that a metal bellows expansion joint is a highly engineered, precisely manufactured piping system component [3]. Therefore, the analysis of modern pressure vessels becomes more sophisticated. History shows that many fatal accidents have occurred mainly due to the design issues. During the last three decade, considerable advances have made into application of numerical technique to analyse pressure vessel [4]-[8]. Finite Element Analysis (FEA) has been used in various branch of engineering in design and development of a product such as for machine elements [9]-[11]. Recent trend shows that FEA has also been adapted for biomechanics application [12], [13]. Common commercially available FEA software includes ANSYS, NASTRAN, ALGOR, ABAQUS and HYPERWORKS.

At present, design and analysis of a pressure vessel by companies in Malaysia, is usually performed using a tailored and specific customise software. Apart from design, fabrication is also a very important aspects in terms of safety. The fabrication of a pressure vessel involves a lot of welding tasks. Therefore, the welding tasks need to be performed by competent welders. Before approval, the pressure vessel needs to be inspected by authority. The quality of the welds is one of the main attentions and inspected using non-destructive testing (NDT) techniques such as x-ray images. In terms of physical tests, the common method used to test the strength of the pressure vessels is the hydrostatic test [14]. However, this method is could be destructive as it could destroy the sample and if this rework happens, it will lead to high cost. Therefore, in analysing a complex and sophisticated structure, FEA has been preferred as an alternative tool to replace physical tests. Moreover, FEA is capable to produce useful visualization and animation to simulate the deformation of the structure such as a pressure vessel. At least it will allow to visualise the predicted deformation and stress in coloured contours [15]. This information is very important for design optimisation, such as in minimising weight, materials, costs and safety [16], [17]. Nevertheless, the results of the simulation must be validated to ensure its reliability, especially when designing a structure that possible to induce fatal accidents, such as a pressure vessel. Therefore, this study attempts to demonstrate the usefulness of general finite element simulation and analysis for the design of a pressure vessel with expansion joint.

Manuscript received October 17, 2017; revised January 11, 2018. This work was supported by the Ministry of Higher Education (MOHE) Malaysia and Universiti Teknologi MARA under the Fundamental Research Grant Scheme, grant no FRGS/1/2014/TK01/UITM/02/7 (UiTM File. No. 600-RMI/FRGS 5/3 (80/2014)).

Rafiq Rustam, Ahmad Faiz Ahmad Nazri, and Jamaluddin Mahmud are with the Faculty of Mechanical Engineering, Universiti Teknologi MARA, 40450 Shah Alam, Selangor, Malaysia (e-mail: rafiqrustam@gmail.com, ahmadfaizahamadnazri@gmail.com, jm@salam.uitm.edu.my).

Md Rizal Md Tasliman is with MMC Tepat Teknik Sdn Bhd, Shah Alam, Selangor Malaysia (e-mail: rizaliza75@gmail.com).

doi: 10.18178/ijmmm.2018.6.4.389

II. Methodology

There are two case studies presented. The first case study focuses on validating the finite element model of a pressure vessel. The deformation and stress distribution on the pressure vessel using finite element method is compared to analytical method. For the analytical method, the thin pressure vessel formula is used to compute the stresses. The second case study involves finite element simulation and analysis for the design of a pressure vessel with expansion joint (horizontal vessel on saddle point).

A. Analytical Approach

In general, analytical approach could be considered as the fundamental tool in solving problems related to engineering mechanics. Therefore in this study, for analytical approach, the fundamental equations were used to determine the axial stress and longitudinal stress for the thin walled pressure vessel respectively. The mechanics of thin walled pressure vessel is adapted as the computed inner radius to wall thickness ratio, r/t for the structure analysed in this study is greater than 10.

Before proceeding to the calculations, the fundamental mechanics related to thin walled pressure vessel is presented. For cylindrical vessel, a gauge pressure *p* is developing within the vessel by contained gas or fluid, which assumed to have negligible weight. Fig. 1 shows the stress direction for a thin walled cylindrical pressure vessel subjected to the normal stress σ_1 in the *circumferential or hoop direction* and σ_2 in the *longitudinal or axial direction*.

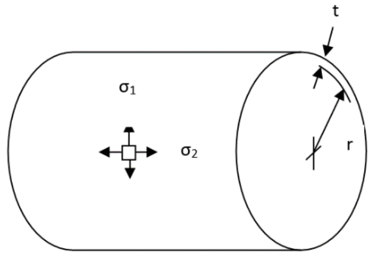

Fig. 1. Stress direction.

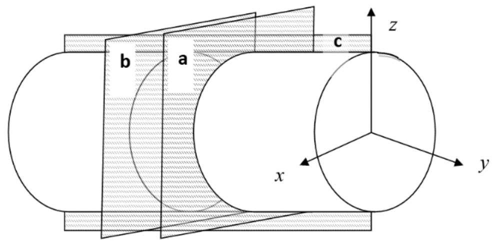

Fig. 2. Hoop stress.

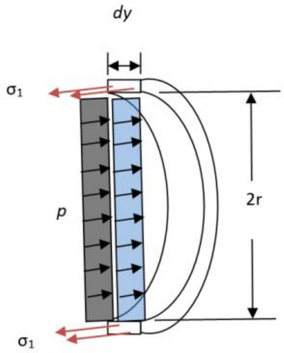

Fig. 3. Cross Section with pressure applied and hoop stress.

To elaborate more about the hoop stress, the figure of a cylindrical vessel that are sectioned according to planes **a**, **b** and **c** is presented shown in Fig. 2. The specific section **c** is further analysed and shown in Fig. 3.

Fig. 3 shows the back segment along with the contained gas or fluid. This loading conditions are developed by the uniform hoop stress, σ_1 acting through the vessel wall and the pressure acting on the vertical face of the sectioned gas or fluid.

For equilibrium in the *x*-direction, the sum of forces on the incremental segment of width *dy* equal to zero as shown in Eq. 1. Eq. 1 is expanded to include all the forces acting in the *x*-direction that leads to Eq. 2. Finally, re-arranging Eq. 2 leads to Eq. 3.

$$\sum F_x = 0 \tag{1}$$

$$2[\sigma_1 t dy] - p2xdy = 0 \tag{2}$$

Next, the longitudinal stress for a cylindrical vessel is considered. For this purpose Fig. 4 is presented to denote the direction of the axial stress acting on the segment on planes **a** and **b** as shown in Fig. 2 earlier.

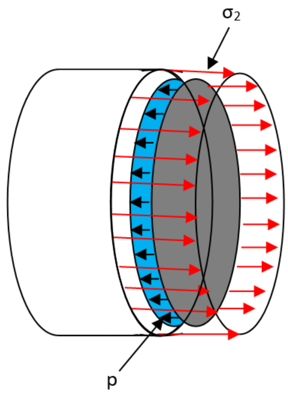

Fig. 4. Axial stress on a cylindrical vessel.

In order to obtain the longitudinal stress, σ_2, the left portion of section **b** of the cylinder is considered. As observed in Fig. 4, σ_2 acts uniformly throughout the wall and the pressure, *p* acts on the section of gas or fluid. Since the mean radius is approximately equal to the vessel inner radius, equilibrium in *y*-direction could be achieve by summing all the forces in the *y*-direction to zero, as shown in Eq. 3. Eq. 3 is expanded to include all the forces acting in the *y*-direction that leads to Eq. 4. Finally, re-arranging Eq. 4 leads to Eq. 6.

$$\sum F_y = 0 \tag{3}$$

$$\sigma_2[2\pi r t] - p(\pi r^2) = 0 \tag{4}$$

Therefore, from Eq. 1 to Eq. 4, the normal stress σ_1 in the *circumferential or hoop direction* and σ_2 in the *longitudinal or axial direction* are represented by Eq. 5 and Eq. 6 respectively.

$$\sigma_1 = \frac{pr}{t} \tag{5}$$

$$\sigma_2 = \frac{pr}{2t} \tag{6}$$

Eq. 5 and Eq. 6 were used for calculating the stresses and these stresses data were used as the basis for validation in Case Study 1.

B. Finite Element Simulation

The first stage of this study was conducted for validation purposes. The FE model was developed using a commercially available finite element software, ANSYS R16.0 installed at the Computer Lab, in the Faculty of Mechanical Engineering, Universiti Teknologi MARA.

A circular section of a cylindrical pressure vessel with a radius of 600 mm and thickness of 12mm was modelled. The vessel was made of Carbon Steel with Modulus of Elasticity of 200000 MPa and the Poisson's Ratio was 0.30. The circlular section was meshed into 20 elements using linear elastic 4 noded 2D shell elements. Various values of pressure form 3.0 N/mm^2 to 4.0 N/mm^2 was applied inside the vessel. A boundary condition was applied in the centre of the cylinder. Once the FE model was validated, the second stage of the study was conducted that was deformation and stress analysis of a pressure vessel with expansion joint.

C. Case Study 2: Deformation and Stress Analysis of a Pressure Vessel with Expansion Joint

The deformation and stress distribution on the pressure vessel using finite element method is compared to previous design work [17]. The material and design specifications are based on a real design of a pressure vessel with expansion joint as presented in Table I, Table II and Table III.

TABLE I: MATERIAL SPECIFICATION

Pressure vessel	Material
Shell	Carbon Steel Plate
Bellow	Seamless pipe at line no 5 and 35

TABLE II: MATERIAL PROPERTIES

Modulus of Elasticity, E for Carbon Steel (C < 0.30 %)	200000 MPa
Poisson's Ratio	0.30

TABLE III: PRESSURE VESSEL DESIGN

Design Pressure	0.386 MPa
Design Temperature	60 ℃
Vessel Thickness	6.35 mm
Inner radius	325 mm
Bellow thickness	8.41 mm
corrosion allowance	1.0 mm

Initially, analysis was performed to verify whether the pressure vessel was a thick walled or thin walled vessel using inner radius to wall thickness ratio, r/t. The result shows a ratio of 51.18, which verify the vessel is thin walled; otherwise, if the ratio is less than 10, then it is thick walled. The inner radius of the vessel is 325 mm and thickness, 6.35 mm.

$$\frac{r}{t} = \frac{325\text{mm}}{6.36\text{mm}} = 51.18$$

III. RESULTS AND DISCUSSION

A. Case Study 1: Finite Element Method versus Analytical Approach

For Case Study 1, the simulated results are shown in Fig. 5 and tabulated in Table IV. Fig. 5 shows the sample results, i.e. the stress contour of the undeformed and deformed shape of the shell with respect to the applied pressure. It could be observed in Fig. 5 that when pressure was applied in the vessel, the diameter of the cylindrical pressure vessel expanded uniformly in the radial direction.

For validation, the simulated axial stress values for various pressure value (between 3 – 4 N/mm^2) are presented in details in Table IV. The results from ANSYS are compared to the analytical results.

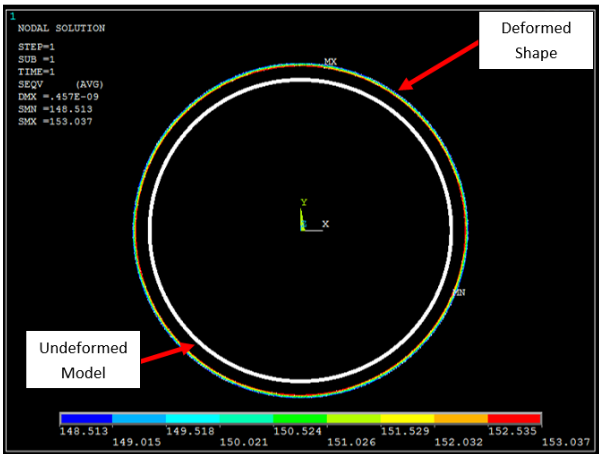

Fig. 5. Sample of simulated results.

TABLE IV: SIMULATION VERSUS ANALYTICAL RESULTS

Pressure (N/mm^2)	Axial Stress (analytical) (N/mm^2)	Average Axial Stress (FEA) (N/mm^2)	Error (%)
3.0	150	150.78	0.51
3.2	160	160.83	0.51
3.4	170	170.88	0.51
3.6	180	180.93	0.51
3.8	190	190.98	0.51
4.0	200	201.03	0.51

In case study 1, the sample of a vessel is modelled using 2D geometry. The geometry is analysed with the circumferential pressure only. One important finding to note is that if, the boundary condition is not applied correctly, the values of the simulated results are correct but the simulated model could shift from its original position and this is not realistic. For example, initially the origin of the model was not fixed, thus the simulated graphic showed that the vessel did not only expand but it also shifted from its origin, which was found to be illogical for a pressure vessel to shift its position a lot during operation. In the actual situation, the pressure vessel could expand due to the increase of internal pressure but could not move distinctively from its original position because it had saddle leg to support and fixed the structure. For this analysis, the saddle leg was not included in the FE model. Nevertheless, its function was replaced by a prescribed boundary condition to fix the vessel at its origin.

It can be observed from Table IV that the current results (simulation) are very close to the analytical results. For the pressure applied between 3 – 4 N/mm^2, the error was found to be less than 1% for various values of applied pressure. This proves that the current FE model is accurate and therefore used for further analysis.

B. Case Study 2: Deformation and Stress Analysis of a Pressure Vessel with Expansion Joint

The sample outputs from the finite element software, ANSYS R16.0 are depicted in Fig. 6 and Fig. 7. Fig. 6 shows the coloured contour for the stress (Von Mises) distribution, while Fig. 7 shows the coloured contour for displacements distribution contour on the pressure vessel with expansion joint due to loading condition as specified in Table III.

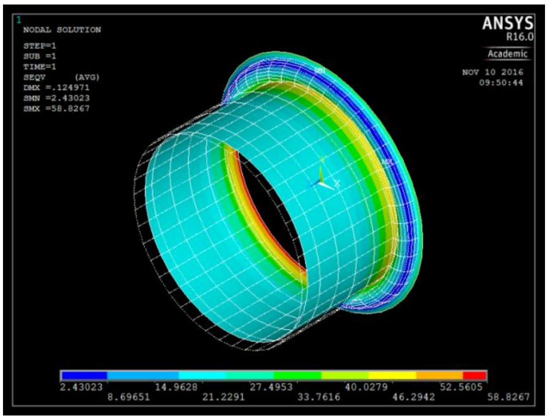

Fig. 6. Von mises stress contour (Max = 58.8MPa).

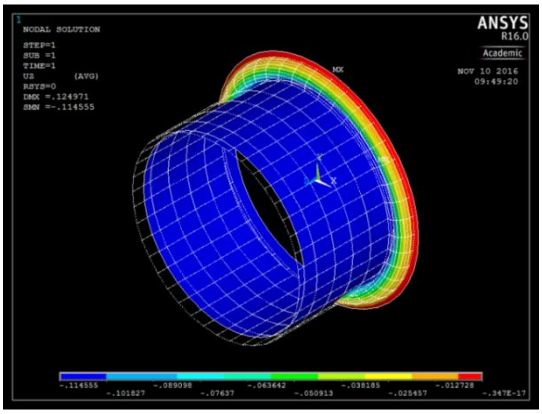

Fig. 7. Displacement contour (Max = 0.12 m).

For case study 2, an additional boundary condition was applied. Due to this, in terms of deformation analysis, the vessel found to expand logically. The use of linear elastic 4 noded 3D shell elements, which having bending and membrane capabilities has generated results portraying the vessel elongation and expansion. The stress contour in Fig. 6 is used to analyse the stress (Von Mises) distribution of the pressure vessel with expansion joint under the specified loading condition. The maximum stress was found to be 58.8 MPa.

The displacement contour in Fig. 7 is used to analyse the deformation distribution of the same vessel, and the maximum displacement is found to be 0.125 mm. The results are close to the results obtained by previous researchers [17], who analysed the original pressure vessel with the expansion joint. This could show that the FE model in this current study and simulated results obtained is accurate and reliable.

IV. Conclusion

This study has demonstrated the usefulness using general finite element software in conducting finite element simulation and analysis for the design of a pressure vessel with expansion joint, which usually these kind of tasks are performed using customised software. The validated results as shown in case study 1 have proven that the accurate model of a cylindrical pressure vessel could be achieved. Case study 2 demonstrates that accurate information and outputs, such as coloured contour for the displacement and stress distribution, could be correctly generated. This information is vital in the design process, where the values and results obtained from the analysis, could be used in fulfilling design criteria according to the selected design code and manual. Further work and analysis of the pressure vessel with expansion joint will be carried out to explore greater potential of the current approach. Nevertheless, it could be concluded that the current study has contributed in enhancing the knowledge about finite element simulation and analysis for the design of a pressure vessel with expansion joint.

Acknowledgment

This research is partially sponsored by the Ministry of Higher Education (MOHE) Malaysia and Universiti Teknologi MARA. The work was conducted at the Faculty of Mechanical Engineering, Universiti Teknologi MARA Shah Alam.

References

[1] D. Moss, *Pressure Vessel Design Manual*, 4th Edition, Elsevier, 2012.

[2] V. A. Carucci, "Overview of pressure vessel design," *Participant Guide ASME International*, 1999.

[3] E. A. Merrick, W. O'Toole and M. Malkmus, Repair of Bellows Expansion Joints," *J. Pressure Vessel Technology*, vol. 105, no. 1, pp. 80-84, 1983.

[4] J. Mackerle, "Finite elements in the analysis of pressure vessels and piping, an addendum: A bibliography (2001–2004)," *International Journal of Pressure Vessels and Piping*, vol. 82, no. 7, pp. 571-592, 2005.

[5] B. Praneeth and T. B. S. Rao, "Finite element analysis of pressure vessel and piping design," *International Journal of Engineering Trends and Technology*, vol. 3, no. 5, pp. 567-570, 2012.

[6] M. A. Porter and D. H. Martens, "A comparison of the stress results from several commercial finite element codes with ASME section VIII, division 2 requirements," vol. 336, pp. 341-346, 1996.

[7] M. A. Porter, D. H. Martens, and C. S. Hsieh, "A comparison of finite element codes and recommended investigation methodology," *ASME*, vol. 359, pp. 241-246, 1997.

[8] M. Amin and S. Ahmed, "Finite element analysis of pressure vessel with flat metal ribbon wound construction under the effect of changing helical winding angle," *Journal of Space Technology*, vol. 1, no. 1, pp 34-39, 2011.

[9] A. K. Sohaimi, A. P. P. A. Majeed, and J. Mahmud, "Preliminary numerical analysis of a platform structure," *Applied Mechanics and Materials*, vol. 680, pp. 280-283, 2014.

[10] M. I. Z Abidin, J. Mahmud, M. J. A. Latif, and A. Jumahat, "Experimental and numerical investigation of SUP12 steel coil spring," *Procedia Engineering*, vol. 68, pp. 251-257, 2013.

[11] M. R. M. Sahri, C. Muhamad, M. J. A. Latif, and J. Mahmud, "Parametric study of low velocity impact using finite element analysis," *Applied Mechanics and Materials*, vol. 393, pp. 397-402, 2013.

[12] M. J. A. Latif, N. H. Hashim, R. Ramlan, J. Mahmud, A. Jumahat, and M. R. A. Kadir, "The effects of surface curvature on cartilage behaviour in indentation test: A finite element study," *Procedia Engineering*, Elsevier, vol. 68, pp. 109-115, 2013.

[13] N. F. A. Manan, M. H. M. Ramli, M. A. Ab Patar, C. Holt, S. Evans, M. Chizari, and J. Mahmud, "Determining hyperelastic parameters of human skin using 2D finite element modelling and simulation," in *Proc. Symposium on Humanities, Science and Engineering*, 2012, pp. 805-809.

[14] T. Tang *et al.*, "Analysis of pressure vessel hydrostatic test," *New York Science Journal*, vol. 7, no. 1, pp. 129-132, 2014.

[15] N. Rahimi, M. Musa, A. K. Hussain, and J. Mahmud, "Finite element implementations to predict the failure of composite laminates under uniaxial tension," *Advanced Materials Research*, vol. 499, pp. 20-24, 2012.

[16] M. R. Mahbob, J. Mahmud, M. A. M. Azlan, and A. Jumahat, "Finite element analysis of telecommunication mini mast structure," *Procedia Engineering*, vol. 68, pp. 258-264, 2013.

[17] J. Mahmud and W. Kuntjoro, "Finite element analysis of expansion joint (HE 101)," Technical Report for MMC Tepat Teknik Sdn Bhd, January 2011.

Rafiq Rustam is currently an engineer at Panasonic VSC Network, Johor, Malaysia. He had joined the company for more than five years, after his graduation with a bachelor's degree (Hons.) in mechanical engineering from Universiti Teknologi MARA (UiTM), in 2011. During his study, he had attached at MMC Tepat Teknik Sdn Bhd, Shah Alam and embarked into a project related to the design and testing of pressure vessels. Apart from this area, his other research areas of interests include robotic, software analysis and programming.

Ahmad Faiz Ahmad Nazri received his B. Eng (Hons.) in mechanical engineering from Universiti Teknologi MARA, Malaysia in 2014. At present he is about to complete his masters of science degree under the supervision of Assoc Prof Ir. Dr. Jamaluddin Mahmud at the same university. His MSc study is funded by a grant scheme under the Ministry of Higher Education (MOHE) Malaysia. He is serving as a graduate research assistant for more than 2 years at the Faculty of Mechanical Engineering, Universiti Teknologi MARA, he involves in several research and gained valuable research experience in the area of engineering mechanics, particularly related to biomechanics, motion capture, the finite element method and engineering design.

Md Rizal Md Tasliman currently is a technical manager at MMC Tepat Teknik Sdn Bhd., Shah Alam. He joined the company ten years ago which developed his expertise in the design and testing of pressure vessels. He had completed numerous design projects and the structures had been successfully installed. Prior to that, he had served as an engineer at a few Malaysian companies. He received his degree in mechanical engineering (B.Eng) from Universiti Teknologi Malaysia (UTM) in 1998. His research areas of interest include design of pressure vessels and stress analysis.

Jamaluddin Mahmud obtained his PhD degree in (biomechanical) engineering from Cardiff University UK; MSc (manufacturing) engineering degree from International Islamic University Malaysia (IIUM); and B.Eng. (Hons.) mechanical engineering degree from Universiti Teknologi MARA (UiTM). He joined the Faculty of Mechanical Engineering UiTM Shah Alam, Malaysia, as a lecturer in 2001 and currently he is an associate professor. He has served UiTM management team at various levels such as a deputy dean, head of academic programmes and the head of Industry Linkage at the Chancellery's Office of Industry, Community and alumni Network (ICAN). He has more than two years of industrial experience, working as a service executive at UMW Equipment Sdn. Bhd. His competency in mechanical engineering supported by his past experience in industry and current involvement with industries has enabled him to register as a professional engineer (Ir) with the Board of Engineers Malaysia (BEM). Assoc Prof Ir Dr J Mahmud has been lecturing on Biomechanics, Composite Materials, Finite Element Method and Engineering Design, which happens to be his areas of research interest and has published more than 100 technical papers in journals and conference proceedings locally and internationally. Due to his expertise and experience, he has been appointed as a judge in design competitions, external examiner, keynote speaker, panel evaluator, visiting researcher and a member in various technical committees at national and international level.