27/28th International Conference
ENGINEERING MECHANICS 2022
Milovy, Czech Republic, May 9 –12, 2022
Paper **#91**, pp. 173–176, doi: 10.21495/51-2-173

APPLICATION OF THE FINITE ELEMENT ANALYSIS AS A NEW APPROACH IN PIPE EXPANSION JOINT ENGINEERING

Joszko K.*, Gzik-Zroska B., Wolański W.***, Gzik M.†, Tymkiewicz T.‡**

Abstract: *The research involved the performance of a strength analysis of a linear expansion joint of a large internal diameter, used in gas pipelines. Such devices aim to absorb and compensate multi-directional displacements caused by geological activity of the ground or climate conditions. The work involved the development of a numerical model of the expansion joint using the Ansys software programme. The model was verified by means of the digital image correlation system during a pressure test conducted in a testing station. The performed FEM analysis resulted in the obtainment of the states of stress and strain reduced in accordance with Huber–Mises–Hencky's hypothesis. The results of the analysis revealed that the structure of the linear expansion joint meets the safety requirements taking into consideration the service load.*

Keywords: Strength tests, Linear expansion joint, Digital image correlation, FEM analysis, Verification.

1. Introduction

Gas expansion joints are indispensable elements to the safety of gas pipelines operation. The main task of expansion joints is the absorption of axial, lateral and angular displacements as well as vibrations caused by geological activity of the ground or climate conditions. The selection of a proper expansion joint is particularly important in the case of pipelines located in geologically active areas, including the areas affected by mining. The designing of an expansion joint is a multi-phase process, where one of the stages involves a strength analysis (Abyani et al. 2021, Blyukher et al. 2005). To meet this end, numerical modeling is applied using specialist software based on the finite element method (FEM). It is a computational method used in solving engineering problems in disciplines such as biomechanics, materials testing, automotive industry, etc. (Gzik et al. 2018, Joszko et al. 2020). The application of modern techniques of designing and structure analysis makes it possible to quickly obtain significant results providing explanation of real phenomena, without the necessity of performing a series of expensive laboratory tests. The use of the CAD/CAE software programmes by engineers in various sectors of industry considerably shortens the time of launching a new product on the market and reduces the number of series of prototype products. Taking into account the above, this article attempted to generate a numerical model of a gas expansion joint of a large internal diameter of the following type: KLR dn1000 MOP84 for the purposes of the strength analysis. The strength analysis aimed to assess the effort of the expansion joint during service load. The developed numerical model was verified using the digital image correlation system (DIC) during a pressure test.

* Kamil Joszko,PhD.: Silesian University of Technology, Faculty of Biomedical Engineering, Roosevelta 40, 41-800 Zabrze, kjoszko@polsl.pl

** Bożena Gzik-Zroska, PhD Eng.: Department of Biomaterials and Medical Devices Engineering, Silesian University of Technology, Roosevelta 40, 41-800, Zabrze, Pl; bozena.gzikzroska@polsl.pl

*** Wojciech Wolański, Professor: Silesian University of Technology, Faculty of Biomedical Engineering, Roosevelta 40, 41-800 Zabrze, wojciech.wolański@polsl.pl

† Marek Gzik, Professor: Silesian University of Technology, Faculty of Biomedical Engineering, Roosevelta 40, 41-800 Zabrze, marek.gzik@polsl.pl

‡ Tomasz Tymkiewicz, Radiatym Sp. z o.o., Przewozowa 20, 44-100 Gliwice, biuro@radiatym.com.pl

2. Materials and Methods

The strength analysis of linear expansion joint KLR DN1000 MOP84 was conducted using the ANSYS software programme. ANSYS is one of the leading software programmes which uses the finite element method (FEM) in widely-understood strength analyses, including thermal, electromagnetic, acoustic and fluid mechanics analyses. Prior to numerical simulations, a thorough analysis of the submitted technical documentation (in the CAD system) was made. Next, the synthesis of the numerical models was performed. The evaluation of their complexity was conducted, which, in turn, enabled the introduction of structural simplifications. The applied structural simplifications did not affect the strength and rigidity of the whole structure; they only improved the quality of discretization (the mesh of solid-like finite elements). Then, the numerical strength analysis was conducted for the simplified FEM model of the linear expansion joint structure for three position / location variants (Fig. 1). The results of the analysis enabled the assessment of the distribution of stress and the rigidity of the expansion joint structure in the conditions of service load.

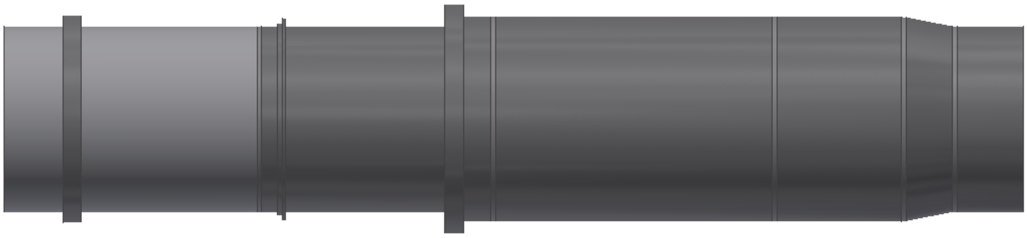


Fig. 1: The linear expansion joint in media position.

The model of the expansion joint structure has the following dimensions: length (distance between the ends in a medial position) L = 5240 mm, external diameter of the body ø = 1124 mm, internal pipe diameter ø = 1016 mm. Each element was provided with proper material properties, which are collated in Table 1.

Tab. 1: Mechanical properties of the analyzed materials

Material Name	Re [MPa]	Rm [MPa]	Young's Modulus [GPa]	Poisson's Ratio
L485ME	485	570	210	0.3
X65	448	531	210	0.3
Brass MO59	103.4	275	105	0.35
PAEK (Polyaryletherketone)	99.97	210	1.1	0.42
PAMos2	75	82.7	3.4	0.45
Rubber	21	27.6	0.003	0.49
Alloy Steel	250	400	205	0.3

Discretization of the major part of the model was carried out using hexahedral elements, which enabled the simulation of nonlinear phenomena during analysis. The finite elements mesh of welded parts, bottoms and conical reduction adapters was generated using tetrahedral elements. The FEM model of the expansion joint with the hexahedral and tetrahedral mesh is presented in Figure 2a. In the numerical model, the fixation was set on the lateral surface of the flange of the main body and pipe extension (Fig.2b). The loading in the form of operating pressure equal to 126 [bar] was set on the whole internal surface of the linear expansion joint and on the bottom surfaces (Fig.2c).

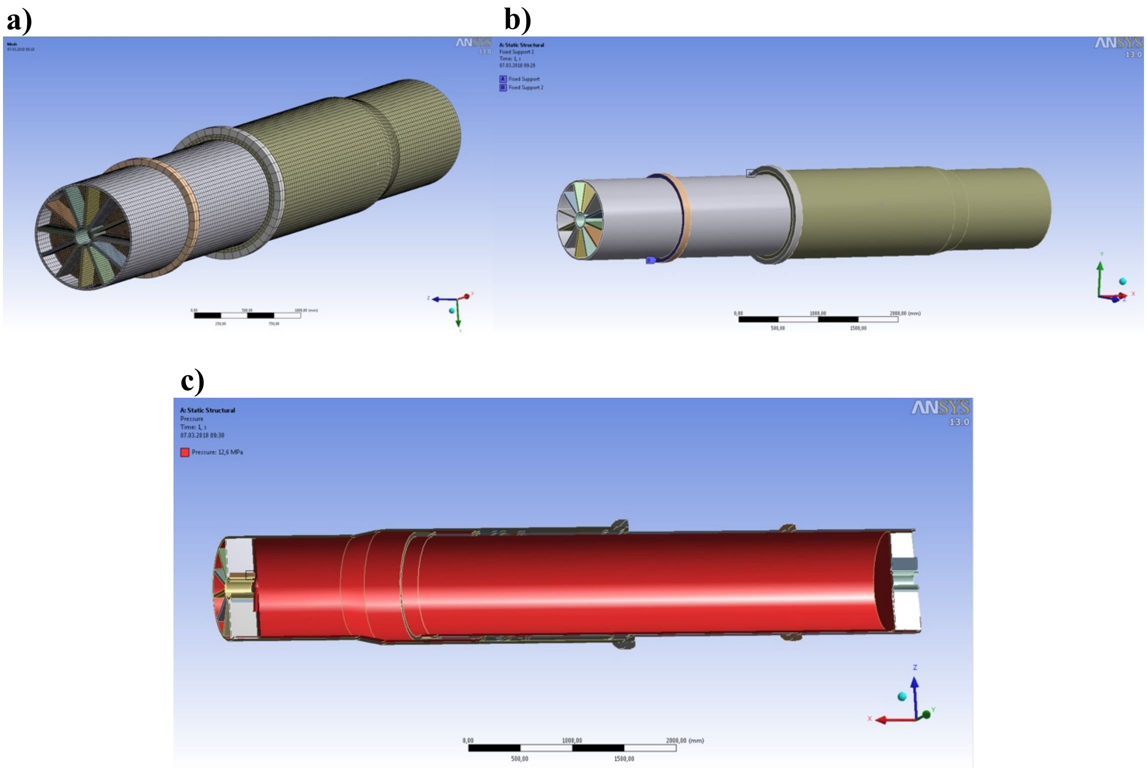


Fig. 2: Numerical model: a) FEM mesh, b) fixation / support sites, c) loading.

The verification of the model was carried out by means of the digital image correlation (DIC) system during a hydrostatic test (Fig. 3a). Then, researchers made a comparison of displacements maps obtained on the basis of the DIC and the numerical model (Figs. 3b and 3c). The total radial displacement obtained on the grounds of the numerical model amounted to 0.58 mm, whereas the resultant radial displacement recorded by the digital image correlation system in one direction equalled 0.28 mm. Therefore, the total radial displacement in both directions equalled 0.56 mm. Taking into account the above, it can be stated that the model was developed in a correct way.

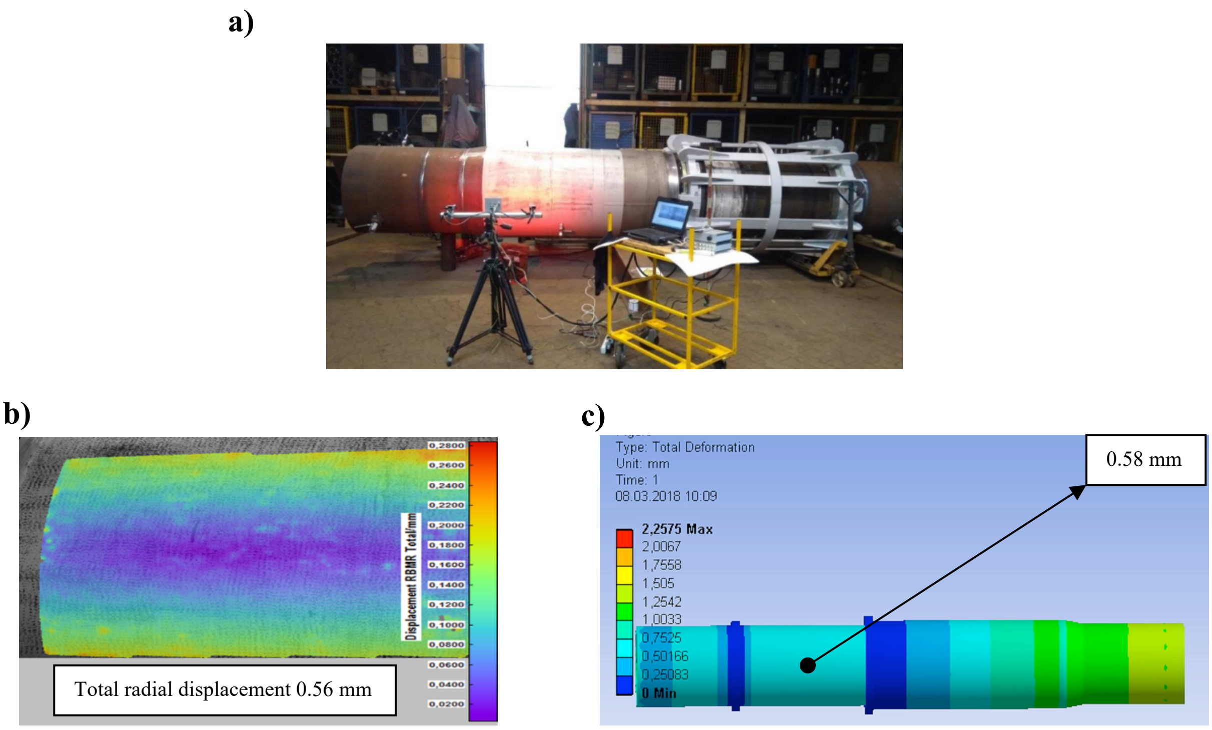
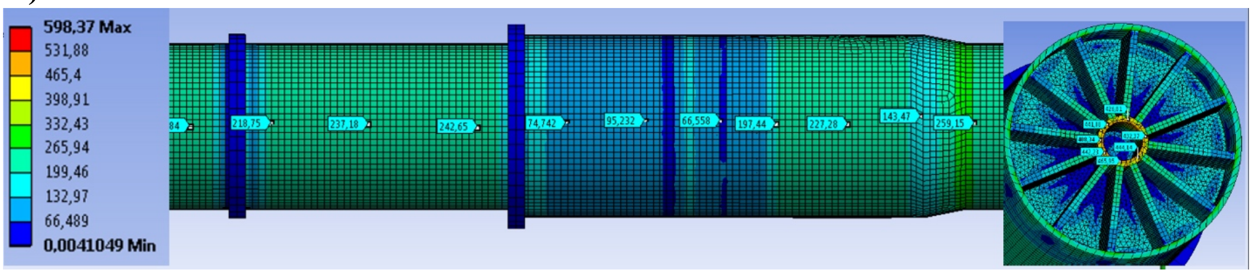


Fig. 3: Verification of the model: a) testing station using the DIC system, b) distribution of displacement on the basis of the DIC, c) distribution of displacement on the basis of numerical analysis.

3. Results

The numerical strength analysis made it possible to determine the states of strain and stress. The stresses reduced in compliance with Huber–Mises–Hencky's hypothesis (H-M-H) did not exceed the elasticity boundary of material L485ME. The mean value of the obtained stress equalled 195 MPa (Fig. 4a). The highest value of strain, reduced according to the H-M-H hypothesis, was recorded for the elements of bottoms, amounting to 0.0092 [mm/mm]. The highest value of the resultant displacement was recorded in the case of bottom elements, amounting to 2.26 mm (Fig. 4b).

a)



b)

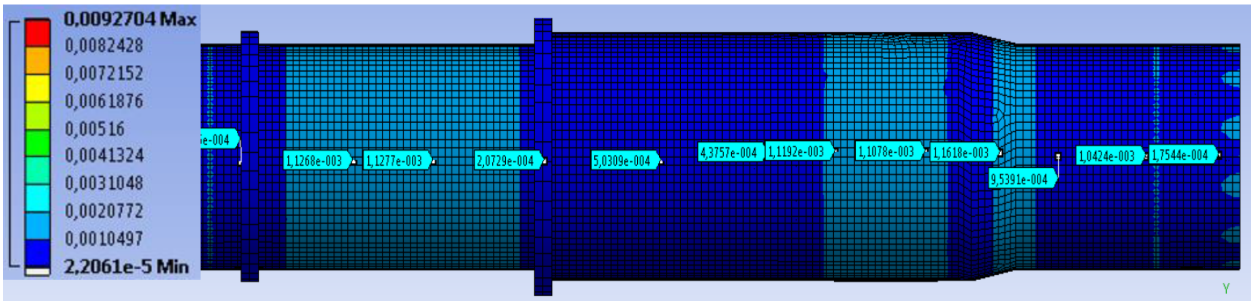


Fig. 4:a) Stresses reduced acc. to H-M-H hypothesis b) Strains reduced acc. to H-M-H hypothesis.

4. Conclusions

This work formulated a numerical model of gas expansion joint KLR DN1000 MOP84. The FEM strength analysis of the expansion joint was performed using the Ansys software programme. The developed model enabled the determination of the states of strains, stresses and displacements. The conducted computations confirmed that the structure of the linear expansion joint is safe and meets the regulations provided for in the norms and standards as well as satisfies the recommended safety coefficient for the analyzed pressure conditions of 126 [bar].

References

Abyani, M. and Bahaari, MR.(2021) A new approach for finite element based reliability evaluation of offshore corroded pipelines, *International Journal of Pressure Vessels and Piping*, 193, 4, pp. 1–13.

Blyukher, B., Niezgoda TM., Małachowski J. and Szymczyk W.(2005) Numerical and experimental research of pipelines safety including aspect of protection from terroristic threat or warfare conditions. In *Proceedings of the ASME Pressure Vessels and Piping Conference-2005*, American Society of Mechanical Engineers, Pressure Vessels and Piping Division PVP, 7, pp. 339–342.

Gzik, M., Wolanski, W., Gzik-Zroska, B., Joszko, K., Burkacki, M. and Suchon, S. (2018) Analysis of various factors impact on safety of armored vehicle crew during an IED explosion. In: Augustyniak P, Maniewski R, Tadeusiewicz R, eds. *Recent developments and achievements in biocybernetics and biomedical engineering. PCBBE 2017*. Advances in intelligent systems and computing. Cham: Springer, pp. 294–303, doi: 10.1007/978-3-319-66905-2_26

Joszko, K., Danecka, A., Burkacki, M. and Gzik, M. (2020) Impact of using passive safety systems on child trauma during a frontal collision, In: Fuis, V., ed., *Engineering Mechanics 2020*, BUT, Brno, pp. 250-253, doi: 10.21495/5896-3-250