International Research Journal of Innovations in Engineering and Technology (IRJIET)

ISSN (online): 2581-3048
Volume 8, Issue 5, pp 135-141, May-2024
https://doi.org/10.47001/IRJIET/2024.805021

Deformation Analysis of Desalinated Water Storage Tank Using Finite Element Method

[1]Achmad Widodo, [2]*Toni Prahasto, [3]Ojo Kurdi, [4]Yan Calvin Hanggara

[1,2,3,4]Mechanical Engineering, Diponegoro University, Semarang, Indonesia
Prof. Soedarto, SH Street, Diponegoro University Tembalang, Semarang, Indonesia 50275
*Corresponding Author's E-mail: toniprahasto@lecturer.undip.ac.id

***Abstract* - Deformation in a structure is something that is very likely to occur when the object is subjected to a loading that leads to a pressure on the object. The approach taken to determine the deformation of the cylindrical tank was carried out using the finite element method using Ansys 2020 R2 software with axial loads due to the roof and circumferential loads on the cylinder from wind forces. The cylindrical tank that was the object observed in the practical work field was found to be deformed due to the very high loading from the roof and also the result of re-welding due to the repair of defective welds after being examined by ultrasonic testing. This was investigated using the finite element method to determine if the loading experienced by the cylinder was the main cause of the deformation and it was found that the maximum deformation was so small that it was difficult to say that the main cause of the deformation was the loading. It is known that re-welding a metal will change the mechanical properties of the material to become softer, which is the main reason why the deformation of the tank occurred in the practical field.**

Keywords: Deformation, Finite Element Method, Cylindrical Tank.

I. INTRODUCTION

Smelting plants require many resources, among which is clean fresh water. Desalination is the right alternative to meet the needs of a lot of water in a place that is near the beach or coast. The process of making desalination requires many components to support the success of effective desalination construction. One of the important components in a desalination plant is the desalinated water storage tank. The existence of the tank becomes an important and mandatory, therefore the construction and design of a sufficient and adequate tank is vital in desalination construction. It is also known that the actual process of construction and erection of the water storage is crucial to be analyzed and considered when deformation occurs. This might include the forming of steel plates until the testing or repairing of the water storage tank.

The development of science and technology that is becoming more advanced makes the construction and design industry more precise with the analysis using the Finite Element Method (FEM). The author's research in this report will discuss and explore the buckling behavior of the desalination water storage tank in a smelter desalination project. This simulation will aim to find out the locations of the tank that are prone to buckling so that in the future the design of the tank can be strengthened at points where buckling is prone to occur.

The shape of the deformation changes according to the load applied to the y-axis of the steel cylinder specified in figure 1 examined. The simulated contours shown in figure 1 indicates the total amount of deformation experienced by the indicated steel structure from the load region applied to the fixed support region [1]. Taskaya investigated the deformation occurring in a cylindrical geometry with the boundary condition that the bottom of the cylinder has a fixed support and the top circular surface of the cylinder is applied a force in the -y direction. The results of deformation calculations using Ansys have relatively small total deformation values.

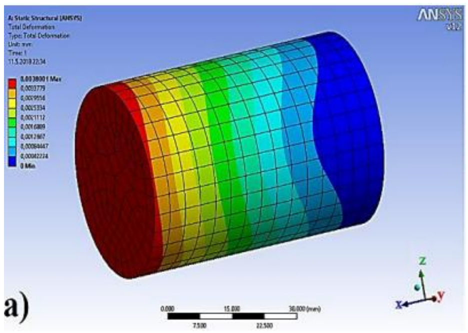


Figure 1: Total Deformation Contour of a Cylindrical Geometri [1]

The research conducted by Rathinam uses a cylinder that has a stiffener as reinforcement for the cylinder structure. The pressure application given is an axial load which the results of this calculation have an increased critical buckling load value [2]. According to the research carried out by Rathinam et al. the determination of the collapse load of this combination of loads becomes insurmountable. It is important to note that

International Research Journal of Innovations in Engineering and Technology (IRJIET)

ISSN (online): 2581-3048

Volume 8, Issue 5, pp 135-141, May-2024

https://doi.org/10.47001/IRJIET/2024.805021

numerically generated computer calculations can have imprecise results, which can be caused by poor convergence or the characteristics of the elements used in the modeling simulation [3]. Further simulations and studies are held within the scope of cylindrical shapes with eigen value buckling analysis to find the critical buckling load of each simulation [4][5].

1.1 Method

This study completed with finite element analysis software which is Ansys Workbench 2020 that used for static and eigenvalue buckling simulation. In this simulation, the type of simulation that is used is Linear Static analysis to obtain the thermal stress, total deformation. The vertical cylindrical tank that is the object is a simplified varying in its thickness of API 650 tank into a uniform thickness cylinder with 7 millimeters of thickness as shown in Figure 2. The thickness that is obtained in this case (7 millimeters) is obtained from the total thickness of the designed tank divided by the total of shell plate levels. The simplification of the model is done in order to comply with the limitations of meshes due to differences in thickness in Ansys 2020 R2.

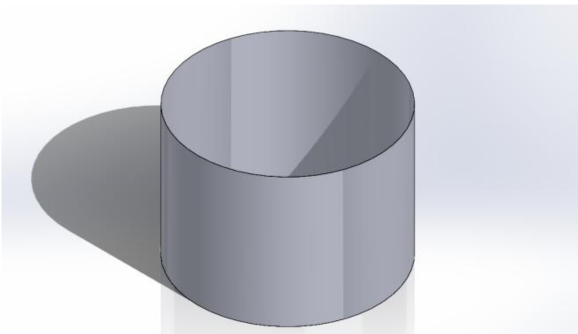


Figure 2: CAD Model of Vertical Cylindrical Tank

1.2 Theoretics

Buckling analysis, in general, can be divided into bifurcation and load deflection. Buckling bifurcation occurs when the maximum compressive axial maximum becomes equal to the buckling stress. The bifurcation analysis was performed using ABAQUS, a commercial finite element analysis program. The bifurcation analysis equation is expressed in the following form:

$$([K] + \lambda[S])\{\Psi\} = 0$$

Where [K] is the stiffness matrix of the system, λ is the loading factor or eigenvalue determining buckling load, and {Ψ} is the inherent vector determining the buckling mode [6].

Storage tanks are divided into spherical vessels and cylindrical vessels [7]. Vertical cylindrical vessels have roof that causes axial loading that needs to be calculated. Simulations were carried out to analyze the structural conditions and deformations after experiencing loading from the water storage tank structure when experiencing loading from the roof (axial stress) and also air (hoop stress). Hoop stress or radial stress is stresses that exist within the radius of the cylinder [8]. Figure 3 would carry out a better visualization before the actual simulation is held. Simulations are carried out with the help of ANSYS software. There are several stages that need to be done in the simulation process, namely the pre-processing stage, the processing stage, the post-processing stage.

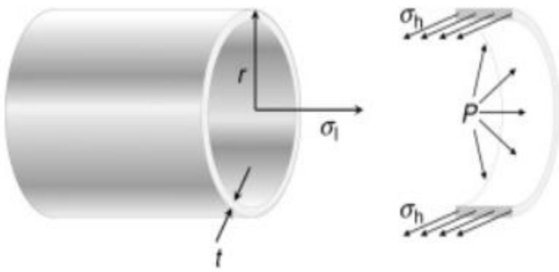


Figure 3: Hoop Stress Visualization on a Cylinder [9]

In making this mesh, a global size of 300 mm is used and face sizing on the top surface of the tank is 40 mm. The following Figure 4 and Figure 5 are the mesh images. Each of these images is an explanatory tool regarding the appearance of the mesh generated for simulation. The mesh quality is in the range of good as the average quality is with 0.5-0.8 on its skewness quality [10].

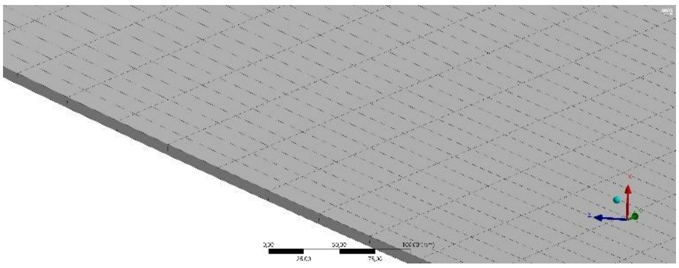


Figure 4: Meshing Results on the Transition Section of the Top Side and Shell Plate

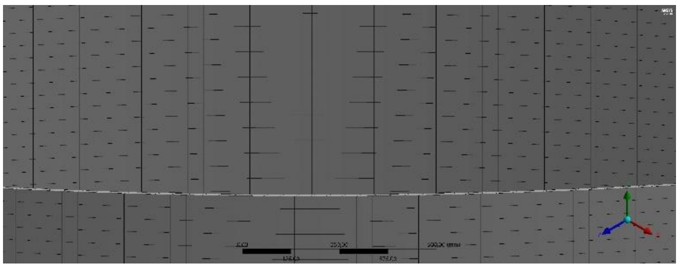


Figure 5: Meshing Results on the Bottom of Cylinder

International Research Journal of Innovations in Engineering and Technology (IRJIET)

ISSN (online): 2581-3048

Volume 8, Issue 5, pp 135-141, May-2024

https://doi.org/10.47001/IRJIET/2024.805021

II. LOADING AND BOUNDARY CONDITIONS

After going through the meshing process, boundary conditions can be applied for the simulation process. The boundary conditions applied consist of remote force, fixed support, force, and remote displacement which represent the condition of the cylinder undergoing loading and its boundary conditions. Fixed support is applied to the bottom side (floor) of the cylinder tank so that it does not move and deform in that part. Remote force and force are forces that represent loading from the roof and wind forces according to design specifications as the following analytical calculations.

$$W_{roof} = Roof\ mass \times Gravitational\ Constant$$

Where the acceleration of gravity = 9.81 m/s^2 and the value of the roof mass = 20268.97 kg, through these equations and the weight of roof becomes 198838.5957 N. As for the wind pressure could be counted by the analytical calculation of the following equation.

$$F_{wind} = Dynamic\ Pressure \times Surface\ Area$$

From the equation above using the acquired dynamic pressure of 1149 and surface area of 735 m^2 leading to a value of wind force of 846107. The stress value obtained from this calculation is then used in finding the total deformation and equivalent stress of the vertical cylindrical tank. The loading condition for this simulation can be seen in Figure 6 for further details for the loading and boundary condition for this simulation.

Figure 6: Boundary Conditions and Loading of Cylindrical Tank Using the Analytical Equation of Wind and Roof

Before advancing into the next step, grid validation is performed with numbers of elements varying from 115404, 150474, 161460, 319248, 409760, 619125, dan 1188922. The details of the results of error in total deformation can be seen in Figure 6 where the error reaches nearly 0% in the range of 619125 elements until 1188922.

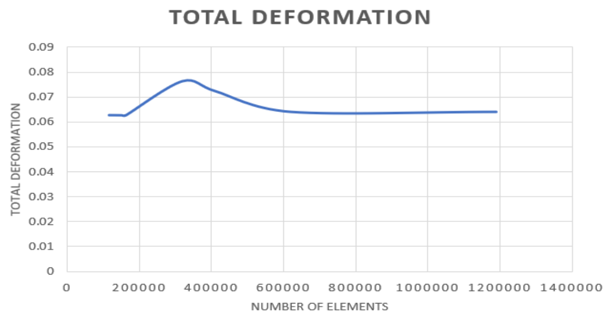


Figure 7: Grid Analysis of the Cylindrical Tank Geometry Using Total Deformation

After the meshing process is complete and all boundary conditions have been applied to the model of the cylindrical tank structure storing desalinated water, the simulation can begin. The further simulation that is performed is an eigenvalue buckling analysis of the cylindrical tank.

Validation of grid independence with total deformation values revealed that the mesh size was adequate and sufficient to continue the simulation for further analysis. Therefore, buckling analysis with Ansys through eigenvalue buckling was performed for the analysis. The boundary condition for eigenvalue buckling analysis can be seen in Figure 7 where the cylinder is subjected to compression pressure at the top and bottom of the cylinder tank. The boundary conditions used are fixed support at the bottom of the cylinder tank, remote displacement on the top tank side where the top side of the tank can only move in the axial direction of compression freely and a compression force of 1 N/m on the top side as shown in Figure 8. This is made as much as possible so that the boundary condition modeling in this simulation resembles the modeling done by Elso.

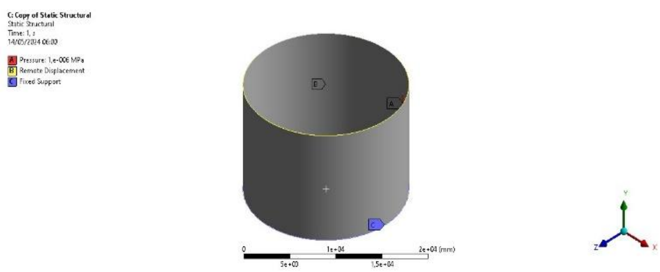


Figure 8: Boundary Conditions of Eigenvalue Buckling Analysis of Water Tank

Boundary conditions that are given at the top of the cylindrical tank are made according to the actual incident when the deformation of the water tank happened. The remote displacement made the geometry free to displace in the y direction as well as x, y, z axis rotationally. The type of fixed constraint where the displacement and rotation on all axes are zero is applied at the bottom side of the water tank. This

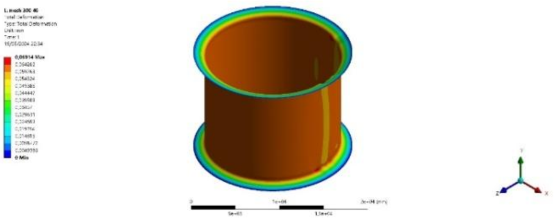
ISSN (online): 2581-3048
Volume 8, Issue 5, pp 135-141, May-2024
https://doi.org/10.47001/IRJIET/2024.805021

boundary condition represents the contact of the bottom side of the tank as it is welded to the ground. The pressure applied to the top side is a representation of the compression pressure due to loading by the roof. The value of the pressure will not have much effect on the critical load. The value of the critical load will be known in the results without entering the original number of loads. This was done to fulfill the simulation boundary conditions in the Ansys application.

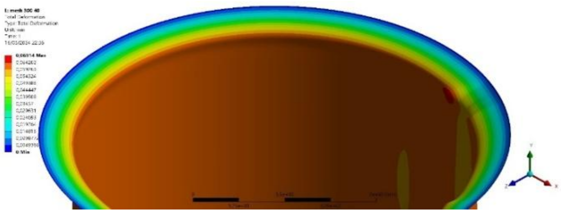
III. RESULTS AND DISCUSSIONS

3.1 Static Structural Analysis with Wind Loading and Roof Loading

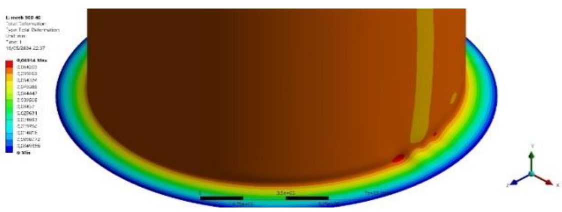
Based on the research simulations carried out, the results obtained in the form of total deformation in 2 hours storage tanks. The following in Figure 9 and Figure 10 are the results of tank simulations that are given loading and produce total deformation and equivalent Von-Mises stress. The simulation results can be seen in Figure 10. This is sought to determine where the stress concentration of the cylinder tank is located when experiencing loading and the location of deformation. Figure 9c is a picture of the stress on the cylinder as a whole, followed by Figure which shows where the highest stress concentration is located in the cylindrical tank.



(a)

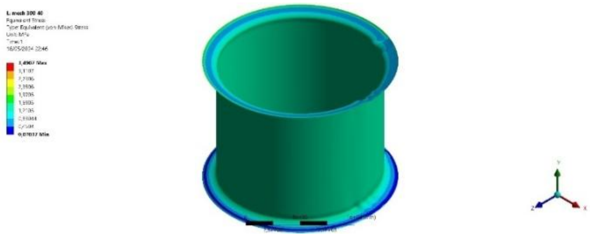


(b)

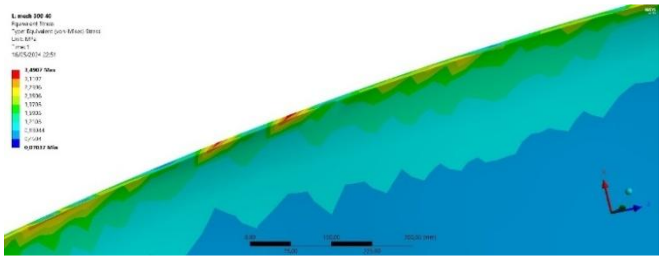


(c)

Figure 9: Water Storage Tank Simulation Results Total Deformation Contour (a) Isometric Overview (b) Top Side of Cylindrical Water Tank (c) Bottom Side of Cylindrical Water Tank



(a)



(b)

Figure 10: Water Storage Tank Simulation Results Equivalent Stress Contour (a) General View, (b) Maximum Equivalent Stress Location

From the simulation results on the cylindrical tank, the total deformation and equivalent stress values are obtained in all parts of the cylindrical tank. In Figure 8 in the form of total deformation results at the top of the tank, it can be seen that at the top, the largest value is located at the center of the cylinder to the bottom of the cylinder until the tank leg bends again with a total deformation of 0.06914 mm. In Figure 9, the equivalent stress on the entire tank can be seen. The largest equivalent stress value in the tank structure is located on the end surface of the X hours operation water storage tank which is directly related to the roof of the tank which is the main axial loading source with an equivalent stress of 3.4907 MPa. The same applies to the bottom (floor) of the tank which experiences maximum stress.

3.2 Eigenvalue Buckling Analysis of Water Tank

Based on the analysis conducted previously, the total deformation value appears insufficient to say that buckling occurs due to loading in accordance with the design specifications. Therefore, further analysis and simulation were made by making eigenvalue buckling analysis. Buckling analysis is discussed through the Euler method for adjacent equilibrium conditions. This method consists of a linearized stability analysis of an undeformed equilibrium configuration, whose critical state is defined by a proportionally scaled load combined with a geometric stiffness or initial stress constructed from an exact nonlinear strain-displacement relationship.

Buckling simulations were carried out up to 30 buckling modes and Figure 10 is an image of the buckling eigenvalue

International Research Journal of Innovations in Engineering and Technology (IRJIET)

ISSN (online): 2581-3048

Volume 8, Issue 5, pp 135-141, May-2024

https://doi.org/10.47001/IRJIET/2024.805021

analysis at mode 28 and Figure 11 obtained the smallest load multiplier value with a deformation of 1.0346 mm in mode 1 and a load multiplier of 3.598e+007 or 35980000. The value of the buckling load can be seen in the following equation:

$$1\frac{N}{m2} \times 35980000 = 35980000\ N/m2$$

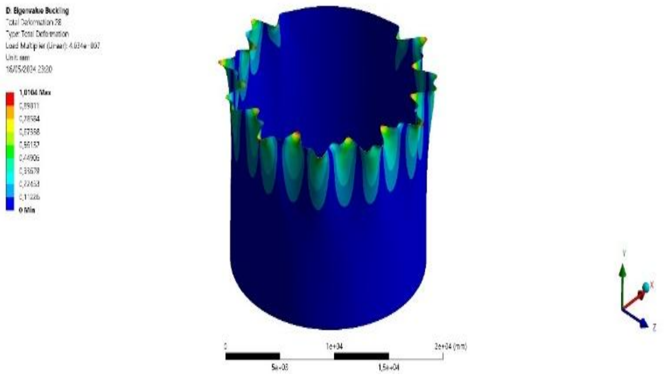


Figure 11: Cylindrical Water Tank Eigenvalue Buckling at Mode 28

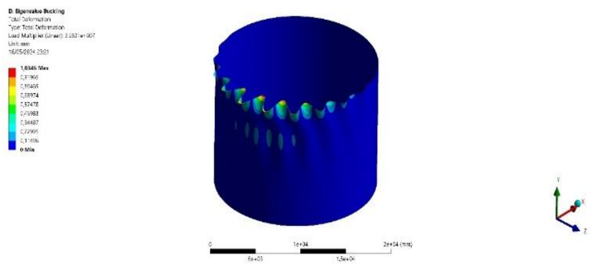


Figure 12: Load Multiplier at Mode 2

The critical buckling load that is considered as the accepted value for this simulation is at the mode 2. This is due to the fact that the critical buckling load at this mode is the lowest compared to the other buckling modes. The lowest critical buckling load was chosen for the means of safety purposes of the water tank. Further details of the load multiplier in the eigenvalue buckling analysis could be seen in Table 1 where all the load multipliers from mode 1 until 30 are shown.

Table 1: Eigenvalue Buckling Mode Load Multiplier of Mode 1 to 30

Buckling Mode	Load Multiplier
1	3.58E+11
2	3.58E+11
3	3.60E+10
4	3.60E+10
5	3.62E+11
6	3.62E+11
7	3.63E+11
8	3.63E+11
9	3.65E+11
10	3.65E+11
11	3.67E+11
12	3.67E+11
13	3.70E+11
14	3.70E+11
15	3.73E+11
16	3.73E+11
17	3.77E+11
18	3.77E+11
19	3.81E+11
20	3.81E+11
21	3.86E+11
22	3.86E+11
23	3.91E+11
24	3.91E+11
25	3.97E+11
26	3.97E+11
27	4.03E+11
28	4.03E+10
29	4.09E+11
30	4.09E+11

3.3 Overall Deformation Analysis Desalinated Water Tank

The construction of a vertical cylindrical tank for storing desalinated water is made using the jacking method. This jacking method uses a hydraulic jack that will raise the curved plate. This method constructs the tank from the top shell plate first and normally is done with the roof plate but in this project the roof is constructed at the end. In the process of lifting the plate and before it is connected to the other plate below, a strongback is used to so that the position of the plate does not move, then welding is carried out.

The initial construction material is a thin flat carbon steel plate that is roller bent to form a circle. Forming by this method, according to local workers, the results are not perfectly curved or there are parts that are not curved. Then, they are transferred to the relevant location by specialized vehicles to transport the rolled sheets and the installation operation is performed using cranes and manpower. From this step, many imperfections caused by various implementation factors are created; some of these are partially modified and some remain in the tank even after the implementation is completed.

The constructed desalinated water storage tank was found to have a defect in the form of porosity as shown in Figure 12 (not the original radiograph from the practical work field) in the weld after being radiographically tested, so repairs were required. The standard tolerable porosity in the construction itself is that there is a cluster (1 or more pores) within a length of 100 mm and its diameter does not exceed 2.5 mm. The repair involved re-grinding of the weld and re-welding. Repeated welding will reduce the strength of the metal used,

ISSN (online): 2581-3048
Volume 8, Issue 5, pp 135-141, May-2024
https://doi.org/10.47001/IRJIET/2024.805021

making it more prone t o deformation. During the re-welding process, a strongback was used just like the initial construction process to ensure that the plate did not move before re-welding. When the re-welding process is completed, the strongback is removed, but due to the reduction in the strength of the material used, the plate under the roof will experience loading and as a result of the weakening of the material previously mentioned, a deformation of the water storage tank occurs.

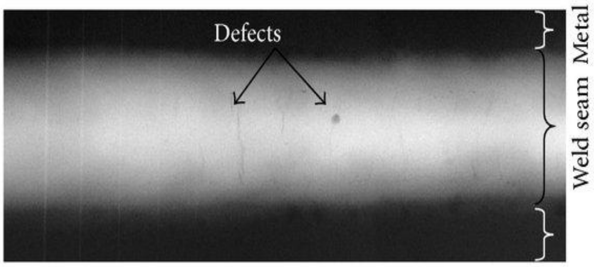


Figure 13: Example of Porosity in a Weld [12]

IV. CONCLUSION

Based on the results of the research that has been done, some conclusions can be drawn as follows:

1) The cylindrical tank structure has been simulated by the finite element method and the total deformation of 0.06914 mm and equivalent stress of 3.4907 MPa with wind loading and tank roof were obtained. Also obtained is the critical buckling load value which is 35980000 Pa or 35.98 MPa.
2) Based on the results of the Fatigue simulation, it can be concluded that variations in the value of different loads will result in different number of cycles as well. The higher the loading value, the lower the number of cycles obtained, such as the number of cycles obtained with the largest thermal stress of 38.92 MPa, which is 294,000 cycles. While the number of cycles obtained with the smallest thermal stress of 28.79 MPa is 14,570,000 cycles.
3) Deformation of the tank according to the simulation is very unlikely to occur due to design errors. The damage occurred due to repeated welding which changed the nature of the shell plate to become softer and deformed after the repair process.

REFERENCES

[1] S. Taşkaya, S. Taşkaya, S. Taskaya, and S. Taskaya, "Investigation of Static Structure Effect According to Axial Coordinates by Using Finite Element Method in Ansys Workbench Software of AISI 310 Austenitic Stainless Cylindrical Mode Investigation of Static Structure Effect According to Axial Coordinates by Using Finite Element Method in Ansys Workbench Software of AISI 310 Austenitic Stainless Cylindrical Model Steel," Article in International Journal of Scientific and Engineering Research, vol. 2, no. 11, pp. 65–70, 2018, doi: 10.5281/zenodo.2538535.

[2] N. Rathinam, B. Prabu, and N. Anbazhaghan, "Buckling analysis of ring stiffened thin cylindrical shell under external pressure," Journal of Ocean Engineering and Science, vol. 6, no. 4, pp. 360–366, Dec. 2021, doi: 10.1016/j.joes.2021.03.002.

[3] D. Kowalski, "Stresses In The Tank Shell With Shape Deformation,"International Conference on Design, Inspection, Maintenance and Operation Of Cylindrical Steel Tanks and Pipelines. 2003.

[4] M. I. Elso, "Finite Element Method studies on the stability behavior of cylindrical shells under axial and radial uniform and non-uniform loads made in Department of Mechanical and Process Engineering Hochschule Niederrhein presented by Summary of the Final Project Work Project Title: Finite Element Method studies on the stability behavior of cylindrical shells under axial and radial uniform and non-uniform loads," 2012.

[5] M. Rastgar and H. Showkati, "Buckling behavior of cylindrical steel tanks with concavity of vertical weld line imperfection," J Constr Steel Res, vol. 145, pp. 289–299, Jun. 2018, doi: 10.1016/j.jcsr.2018.02.028.

[6] S.-E. Kim and C.-S. Kim, "Buckling strength of the cylindrical shell and tank subjected to axially compressive loads," 2002. [Online]. Available: www.elsevier.com/locate/tws

[7] F. P. (Ferdinand P. Beer, Mechanics of materials. McGraw-Hill, 2011.

[8] L. Skinner, "Introduction," in Hydraulic Rig Technology and Operations, Elsevier, 2019, p. xxi. doi: 10.1016/b978-0-12-817352-7.09989-9.

[9] L. W. Mckeen et al., "FATIGUE AND TRIBOLOGICAL PROPERTIES OF PLASTICS AND ELASTOMERS Third Edition."

[10] N. Fatchurrohman and S. T. Chia, "Performance of hybrid nano-micro reinforced mg metal matrix composites brake calliper: Simulation approach," in IOP Conference Series: Materials Science and Engineering, Institute of Physics Publishing, Nov. 2017. doi: 10.1088/1757-899X/257/1/012060.

[11] S. Tunde Azeez and P. Madindwa Mashinini, "Radiography examination of friction stir welds of dissimilar aluminum alloys," Mater Today Proc, vol. 62, pp. 3070–3075, Jan. 2022, doi: 10.1016/j.matpr.2022.03.225.

[12] M. Ben Gharsallah and E. Ben Braiek, "Weld

ISSN (online): 2581-3048
Volume 8, Issue 5, pp 135-141, May-2024
https://doi.org/10.47001/IRJIET/2024.805021

inspection based on radiography image segmentation with level set active contour guided off-center saliency map," Advances in Materials Science and Engineering, vol. 2015, 2015, doi: 10.1155/2015/871602.

Citation of this Article:

Achmad Widodo, Toni Prahasto, Ojo Kurdi, Yan Calvin Hanggara, "Deformation Analysis of Desalinated Water Storage Tank Using Finite Element Method", Published in *International Research Journal of Innovations in Engineering and Technology - IRJIET*, Volume 8, Issue 5, pp 135-141, May 2024. Article DOI https://doi.org/10.47001/IRJIET/2024.805021

****

