Sirte University Scientific Journal(Applied Sciences) *Vol.9(1) ,June 2019*

Numerical Study On The Hydrostatic Test of Penstock

Mahdi Algool[1], Nasar Abddlssalam[2] and E.M.Elmabrouk[3]
e-mail: algmahdi103@gmail.com

[1,2]Mechanical Engineering Department, Faculty of Engineering, Sirte University
[3] Chemical Engineering Department, Faculty of Engineering, Sirte University

Abstract

The hydrostatic test of a full-scale model of penstock has been modeled in an ABAQUS, a finite element software package, to simulate the behavior of the finite element model with inner pressure. the von Misses stress distribution will be investigated in two steps; the first load-unload (FL-UNL) and the second load-unload (SL-UNL), and focus on where the yielding initiates and spreads [1]. This model has been sketched as the experimental model of the penstock, Figure 1, except that the third segment of cylindrical mantle of the experimental model has been left out, because the effect of size of the geometrical model on the run time of finite element analysis. On the other hand, this part of the experimental model is not important and could be negligible.

Keywords: *Weld Joint, Initial Plastic Deformation, Residual Stresses, Penstock.*

1. Introduction

In the end of seventies, a penstock was built-in in a hydropower plant. This penstock was design and examined with all required experimental tests to verify its efficiency for the purpose that was designed for [2]. Further details for the experimental tests, kind of steel was used, and production process are provided in previous studies [1, 3]. Moreover, the geometry model of penstock was modeled as illustrated in Figure 2. A finite element model (FEM) of penstock is built in to simulate the experimental obtained results.

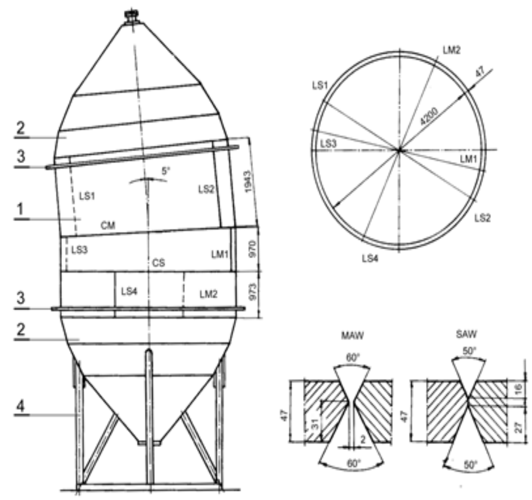

Figure .1 Design of penstock segment full-scale model: 1- mantle; 2- lid; 3- stiffener; 4- supports, L - Longitudinal, C- Circular; MAW– shielded manual arc welding (M); SAW- submerged arc welding [2].

2. Methodology and Mechanical Properties of FE Model

The geometry model of the penstock that has been modeled using ABAQUS/CEA software is shown in Figure 2. This model has been drawn as exactly as the one used in the experimental model [3], except that the third segment of cylindrical mantle of the experimental model has been left out, because the effect of size of the geometrical model on the run time of finite element analysis. On the other hand, this part of the experimental geometry is not considered to be important. Therefore, it can be neglected. When ABAQUS software is used, the elastic and plastic properties are required to carry out the elastic-plastic analysis. For elastic properties, Young‘s modulus and Poisson's ratio were defined in ABAQUS sheet of elasticity. Furthermore, in order to improve the plastic range in the used software, the yield strength and the plastic strain corresponds to each increment of stresses should be defined in its ABAQUS sheet. All the mechanical properties, that have been used for the model simulation was taken from the experimental tests results.

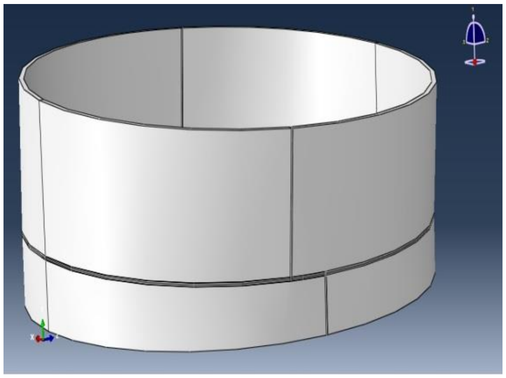

Figure .2 The finite element model of the penstock as drawn in ABAQUS/CEA

3. Methodology and Mechanical Properties of FE Model

In finite element FEA analysis, the accuracy that can be obtained is an important issue. This accuracy is directly related to the finite element mesh that is going to be used, where a poor mesh could give unrealistic results. Whereas, high density mesh will lead directly to high level of complexity [4]. A mesh density that was used in the study was performed to achieve a fine mesh of the model, as demonstrated in Figure 3. For the finite element to simulate this experiment, the two ends of FE model are fixed from displacing or rotating them in the three directions X, Y and Z. The boundary conditions applied in this simulation are showed in Figure 4. In the finite element model of the penstock, the inner surface was subjected to inner pressure as in the real surface in the pressure vessel.

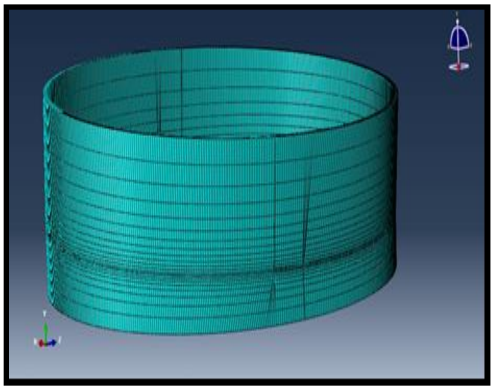

Figure .3 The Finite Element model mesh

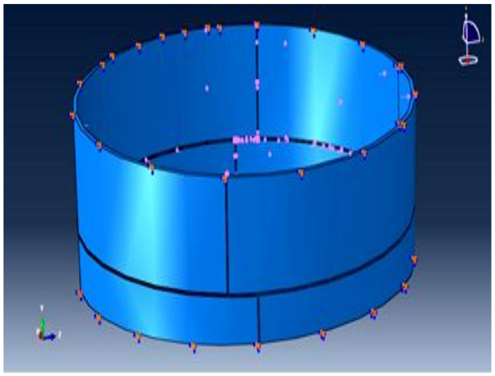

Figure .4 Boundary condition and applied inner pressure.

3.1 Initial residual stresses (RS) for the first loading (FL)

To simulate the effect of residual stresses on the behavior of weld joints of the first load, 40% of yield strength was added to each weld joint as a predefined field, and the six values of von Misses stresses were defined in its ABAQUS sheets.

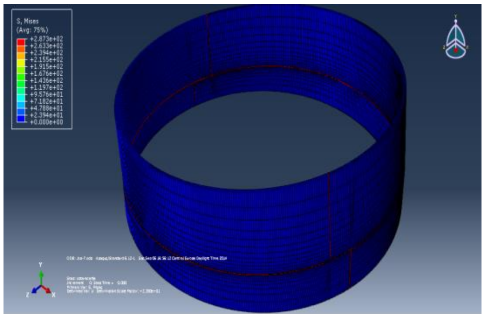

Figure .5 The initial residual stresses for FL of FE model

3.2 Initial Residual Stresses (RS) for Second Load (SL)

As it will be shown later, the resulted residual stress after first load-unload is much lower than the initial residual stresses, and to simulate the effect of initial residual stresses on weld joints for the second load, we assumed that the value of initial residual stresses for the second load is equal or a little bit higher than that one used for the first load.

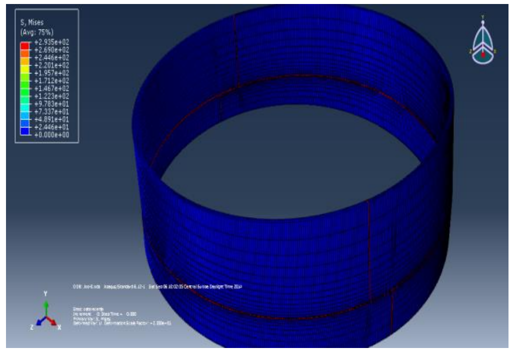

Figure .6 Initial residual stresses for SL of FE model

4. Results and Discussion

4.1 Plastic Strain, (FL-UNL without RS)

As indicated in Figure 7, the plastic strain is only initiated in the weld joint (LS1 Sub Merged Arc (SAW). This behavior is due to the lower yield strength of the joint and its location in the stress concentration region.

4.2 Plastic Strain (FL-UNL, with RS)

Figure 8 shows the initiation of plasticity after first load of FE model, where the plasticity initiated is in the weld joint LS1 SAW. This behavior is due to the lower yield point and its location in this joint.

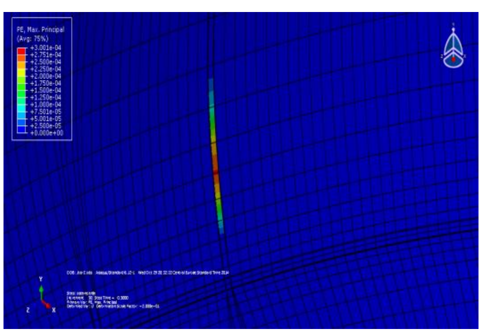

Figure .7 Plastic deformation of FE model without RS (FL-UNL, P=14.5MPa)[3]

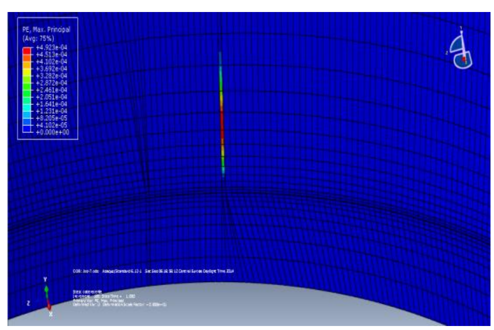

Figure .8 The plastic strain of WM LS1 SAW after FL with RS (P=11.2

4.3 Plastic Strain (SL-UNL without RS)

As illustrated in Figure 9, the levels of von Misses stresses have exceeded the yielding of the base metal and weld joints at that side of the stress concentration region and the plasticity initiated and spreads in base metal and weld joints in this area.

4.4 The Plastic Strain (SL-UNL, with RS)

As the inner pressure increased for second load, the plastic strain initiated in the other weld joints, it was at the shorter side CMAW, LS3 SEW, (Fig. 10).

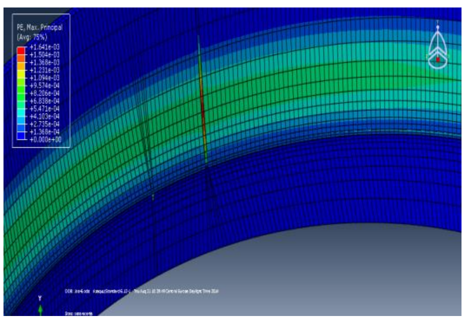

Figure .9 plastic deformation of FE model without RS (SL-unL, P=18.5MPa)[3]

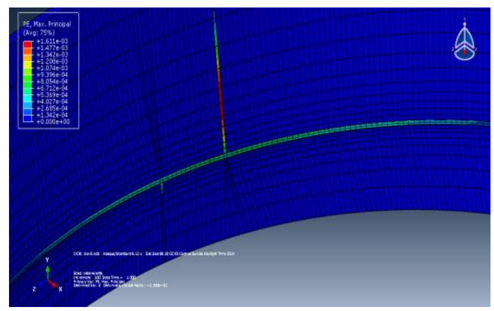

Figure .10 The plastic strain of WM LS1 SAW with RS after SL (P=14.4MPa)[3]

4.5 Von Misses Stresses- Strain Relation

Figure 11 shows the relationships von Misses stress – strain for weld joint LS1 obtained using strain gauge readings, finite element calculations without taking into account residual stresses and geometrical imperfections, and finite element calculations using residual stresses into account. The position of weld joint LS1 is on the shorter side of the upper cylinder with a slope of 5° and it is logical to expect high tensile stress and strains because under the inner pressure, the cylinder is trying to reach the ideal cylinder.

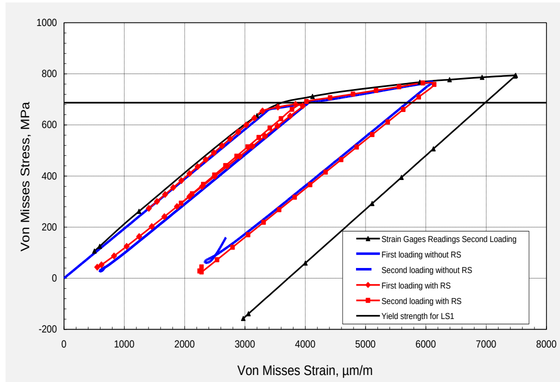

Figure .11 Von Misses Stress – Strain relationships for SMAW weld joint LS1

5. Conclusions

- The von Misses stress distributions of the finite element model of penstock showed that, the highest stress level has been on the shorter side of the model (at a 5º angle), which represents the stress concentration region. This behavior is affected by the geometrical shape, which exerted more compression on that side in an axial direction.
- The upper segment of penstock on the shorter side represents the critical part of the structure. The experiment model and the finite element model have been shown the critical point of the penstock at that part (weld metal joint LS1 SAW), where the plasticity started earlier than the other joint weld metal which has the same properties (yield strength).

References

[1] S. Sedmak and A. Sedmak, "An Experimental Investigation Into The Operational Safety Of A Welded Penstock By A Fracture Mechanics Approach," Fatigue & Fracture of Engineering Materials & Structures, vol. 18, pp. 527-538, 1995.

[2] A. S. Sedmak, Simon & Milović, Ljubica, "Pressure equipment integrity assessment by elastic-plastic fracture mechanics methods," 2011.

[3] M. M. A. Algool, "Initial Plastic Deformation And Residual Stress Influencing The Welding Joint Behavior In The Presence Of Cracks," Doctoral Dissertation, Faculty Of Mechanical Engineering, University Of BELGRADE, Belgrade, 2015.

[4] Y. Liu, Effects of Mesh Density on Finite Element Analysis vol. 2, 2013.