FRI-ONLINE-1-MEMBT-02

SIMULATION DRIVEN DESIGN OF PLASTIC WATER TANK

Prof. Ivelin V. Ivanov, PhD
Department of Engineering Mechanics,
“Angel Kanchev” University of Ruse
Tel.: 082-888 472
E-mail: ivivanov@uni-ruse.bg

Assoc. Prof. Danail Gospodinov, PhD
Department of Material Science and Technology,
“Angel Kanchev” University of Ruse
Phone: 082-888 205
E-mail: dgospodinov@uni-ruse.bg

Assist. Prof. Dimitar Velchev, PhD
Department of Engineering Mechanics,
“Angel Kanchev” University of Ruse
Tel.: 082-888 224
E-mail: dvelchev@uni-ruse.bg

Abstract: *A plastic water tank of tamping machine is reported by many costumers as often failing part. The reason of that failure has been investigated by testing samples taken from the wall of the tank. Some of the samples include the joint between both parts welded by melting the edges at high temperature and pressing them each other. The weak connection between the parts is the main reason for the failure. A new design of the tank is suggested, which is based on plastic blowing technology and the tank body consists of only one part. The new design should be checked by finite element simulations for damages obtained after free falling on the ground and received under the vibrations of the machine. The filled by water tank has higher loading in free falling impact than empty one however fluid-structure interaction is a problem in the simulations. The shell element model of the tank has great leakage of the fluid when water is modelled by Lagrangian finite elements. The smoot particle hydrodynamic method gives quite better results in the simulations of the tank impact. The successful simulations show that the impact on the bottom and on the bottom vertex of the tank could cause sagging of the tank wall which remain permanent and this is unacceptable. Then another design of the tank is suggested, avoiding the projective shapes of some tank areas. The second version of tank design has no permanent deformation in the impact simulations and the stress level caused by the vibrations is low, so it is acceptable. The successful simulation driven design of plastic water tank is saving the expenses of manufacturing the prototypes and testing them in order to reveal unwanted features.*

Keywords: *Finite Element Simulations, Fluid-Structure Interaction, Smoot Particle Hydrodynamics*

INTRODUCTION

A plastic water tank of tamping machine is reported by many costumers as often failing part. The tank consists of two parts welded together under high temperature and pressure. The tank has no handle and in order to dismount it one should apply forces on the top of the tank (fig. 1). The tank is vulnerable because any occasional load applied on the tank can cause at least damages on the joint and eventually opening of a crack at the top of the tank. The aim of this work is to investigate the reason of the failure and to suggest a new design of the tank.

EXPOSITION

Testing joint of old design tank

Specimens of the material, cut out from the tank, are tested in tension. The base material HIFAX EP3080 (fig. 2) shows a good agreement to the Lyondell Basell Technical Data Sheet data:

yield stress $\sigma_y = 17$ MPa, ultimate stress at break $\sigma_u = 15$ MPa, elongation at yield $\epsilon_y = 6\%$, except the elongation at break $\epsilon_u = 500\%$, which is quite less – 150%. The failure is with "'whitening"' which can be observed and that shows less plasticity of the material than what is expected.

Fig. 1. Dismounting of failed tanks.

Two specimens including the joint between both parts are cut out and tested in tension. The tests show brittle failure at the joints (fig. 3). The failure is cohesive fracture of a part of the joint cross-section with an area ($b \times h$). The damage initiation of specimens is at force F_d, while the ultimate force at failure is denoted as F_u.

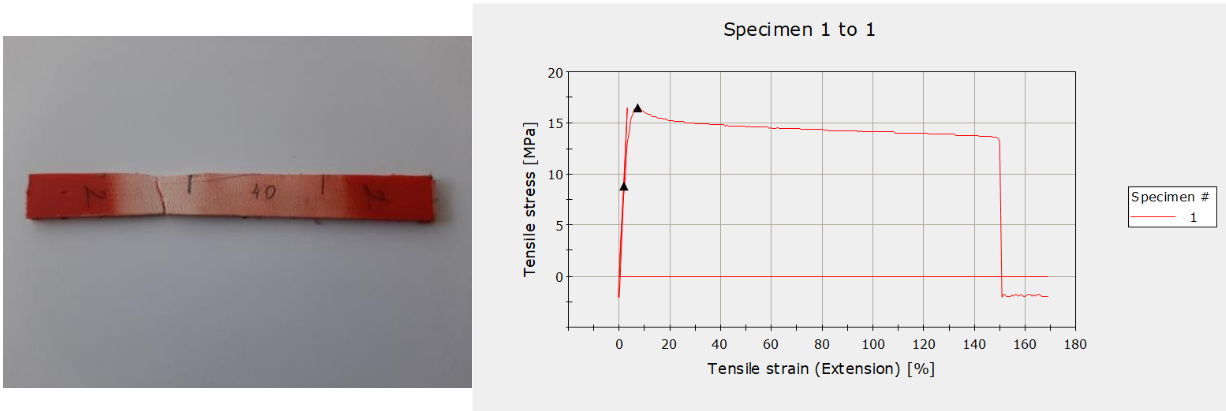


Fig. 2. Test results of the base material HIFAX EP3080

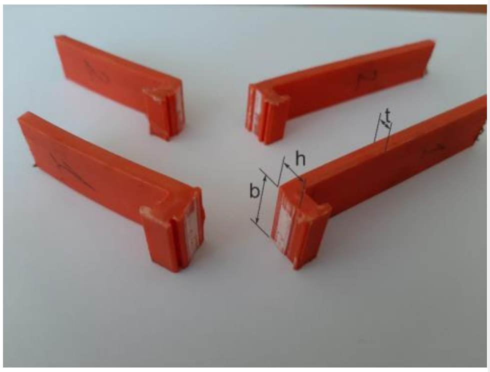
The tension in the joint cross-section is eccentric with eccentricity

$$e = \frac{h}{2} - \frac{t}{2} \tag{1}$$

The bending moment in the cross-section of the joint at the point of damage initiation is $M = F_d e$. The edge tensile stress in the joint cross-section at damage initiation is:

$$\sigma_d = \frac{F_d}{bh}\left(1 + \frac{6e}{h}\right) \tag{2}$$

 Copyrights© 2020 ISBN 978-954-712-826-2 (Print)



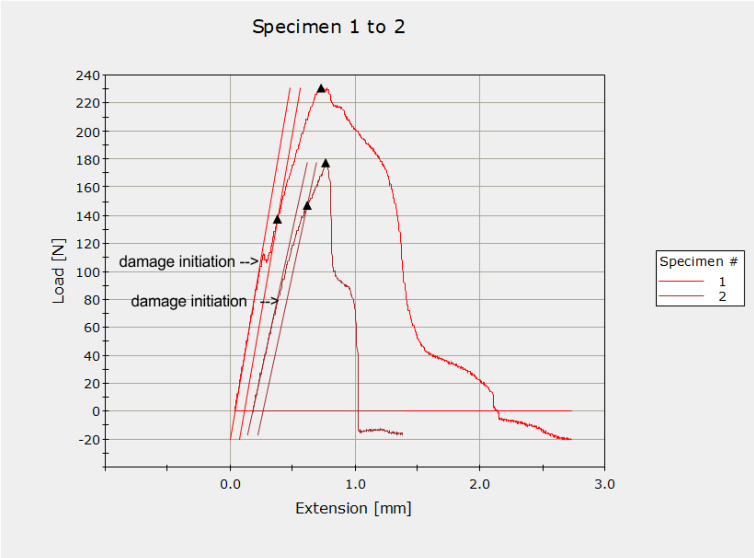


Fig. 3. Test results of the joint specimens

The far cross-sectional ultimate normal stress of a specimen is

$$\sigma_u = \frac{F_u}{bt} \tag{3}$$

Table 1. Results of measurements and stress calculations

specimen	b mm	h mm	t mm	F_d N	F_u N	σ_d MPa	σ_u MPa
# 1	16.3	9.0	3.3	130	250	2.57	4.65
# 2	14.3	8.3	3.4	100	197	2.33	4.05
average						2.5	4.4

The low values of the damage initiation stress and the far ultimate stress of the joint in comparison with the base material strength show that, it is the weakest element of the tank. Moreover, the cross-sectional area of the joint near the inlet neck of the tank is less than the others, because the pressure applied on the joint during the manufacturing is less near the holes since the higher compliance of the structure there. The bottom of the tank has double joint of the both parts of the tank strengthened by ribs and its stiffness and strength are very high. So, the weakest part of the joint is at the top of the tank.

It is necessary a new design of the tank to be suggested, avoiding welding of parts and having a handle. The new design of the tank should be examined on impact as a drop test and vibration loading. The examination should be done by simulations, because its more efficient than testing prototypes.

Impact simulations of new tank design

The simulations of the solid tank with water requires fluid modeling and fluid-structure interaction, which can be done by Eulerian elements (Souli, M. & Zolesio, J.P. ,2001) or Smooth Particle Hydrodynamic (SPH) elements (Sun, W.-K., Zhang, L.-W., & Liew, K.M. ,2020). The leakage is difficult to be avoided when solid is modeled by shell elements and the fluid by Eulerian elements (Hallquist J., 2006). This is why we preferred to model the water by SPH-elements.

The tank is modelled in Finite Element (FE) method (fig. 4) by shell elements with thickness of 3 mm. The flange shell elements have thickness of 13 mm. The rod is modelled by beam elements with inertia moments of area corresponding to 16 mm in diameter. The water is modelled by Smooth Particle Hydrodynamic (SPH) elements. The floor is modelled by rigid shell elements with thickness of 1 mm.

The material of the shell elements is elastic-plastic with kinematic hardening and corresponds to poly-propylene PP46 with 20% reduced yield stress: $\sigma_y = 18.4$ MPa, mass density $\rho = 0.9$ g/cm^3, Young's modulus $E = 1.3$ GPa, Poisson's ratio $\nu = 0.42$, and hardening tangential modulus of elasticity $\overline{E} = 4$ MPa. The material of the beam elements is the same.

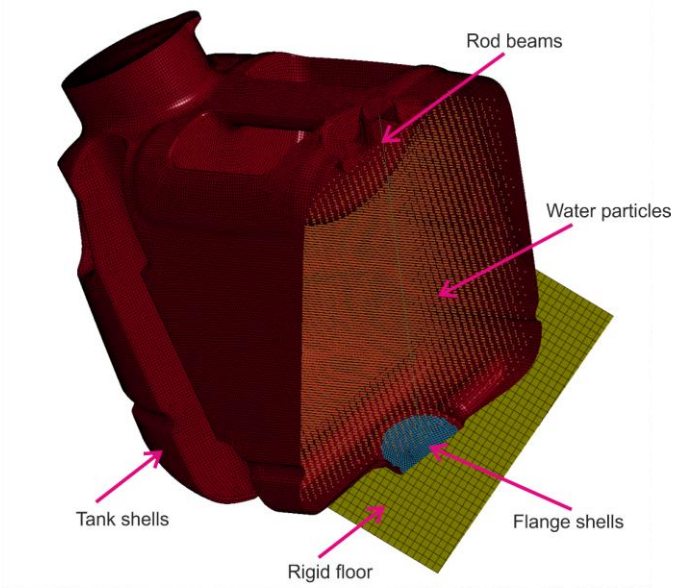


Fig. 4. FE model of the tank

The material of SPH elements corresponds to water with mass density of $\rho = 1$ g/cm^3 and cut off pressure $p_c = -1 \cdot 10^5$ MPa. The Murnaghan's Equation Of State (EOS) is used with parameters: $\gamma = 7$ and $k_0 = 0.357$.

Initial velocity of $v = 4.85$ m/s is prescribed to all nodes in negative y-direction. This velocity corresponds to free falling body from height $h = 1.2$ m ($v = \sqrt{2gh}$).

An impact in three directions was simulated (fig. 5):

1) Bottom impact – the first direction is the vertical direction of the tank position on the machine and the whole bottom of the tank is impacted.

2) Edge impact – the second direction is when the tank is tilted forward from the position on the machine at 30 degrees and the bottom front edge of the tank is the impacted area.

3) Vertex impact – the third direction is when the tank is tilted additionally to the left side at 40 degrees and the bottom front and left vertex is the impacted area of the tank.

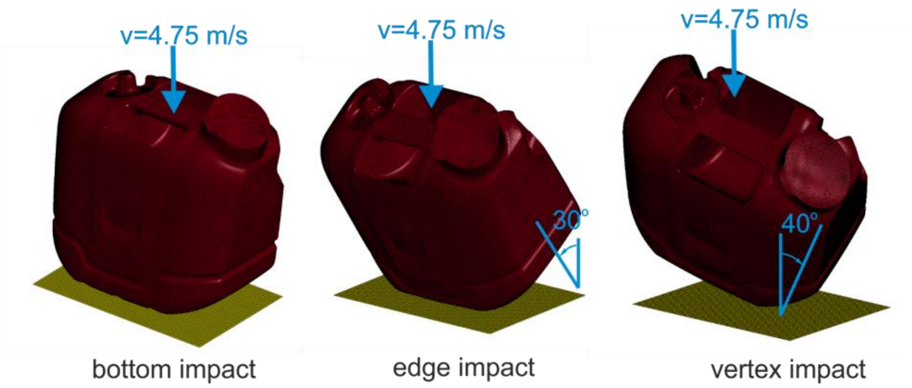


Fig. 5. The impact directions

Two simulations in each direction of the impact are carried out: one *with water* and the other

 Copyrights© 2020 ISBN 978-954-712-826-2 (Print)

without water (SPH elements). The contact force of the impact is always higher and longer acting when we have simulations with water in the tank than in the cases of empty tanks (fig. 6).

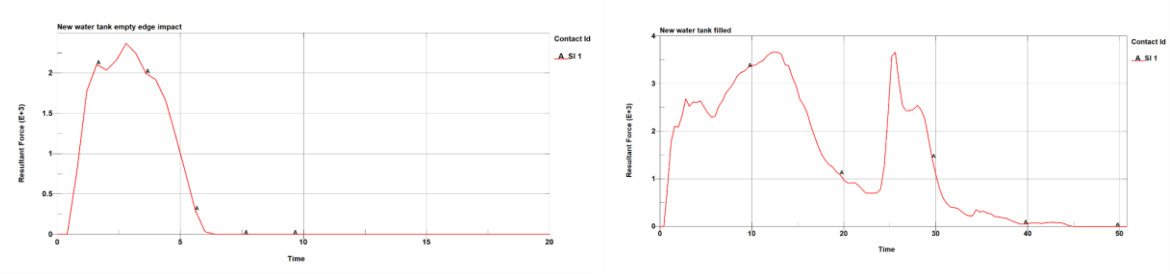


Fig. 6. Results from simulations: without and with water

The bottom impact causes the flange area to sag during the impact with high effective plastic strain (fig 7). When the contact is lost after the impact, the residual strain and stress is high and the sagging of the flange is visual and significant. The concavity of the flange is very high when the tank is filled by water. Spring back calculations are applied at the end of each simulation and the effect is minor. The other issue is that the rod for water dosing is in bending because of the flange rotation. The flange sagging could cause functional failure of the water dosing tap.

Fig. 7. A bottom impact and sagging of the flange

The edge impact cause minor effective plastic strain and the residual change of the shape of the tank could be neglected. The vertex impact, however, cause significant change of the shape because of sagging near the vertex area (fig. 8). The effective plastic strain is significant and the residual sag is not only visual but the tank could be damaged after repeating the impact.

Fig. 8. A vertex impact and sagging near the vertex area.

The effect of sagging was mitigated by higher thickness of the tank wall. The bottom impact and vertex impact simulations of full tank were repeated with shell thickness $t = 4$ mm. The results are shown in the table 2.

The maximum bending moment of the water dosing rod that can cause maximum stress very close to the yield stress of material PP46 is:

$$M_{\max} = \frac{\pi d^3 \sigma_y}{32} = \frac{\pi 16^3 \cdot 18.4}{32 \cdot 1000} = 7.4\ \mathrm{N.m} \tag{4}$$

Table 2. Results from simulations with different thicknesses

Case of impact	**empty tank** $\boldsymbol{t = 3}$ **mm**	**full tank** $\boldsymbol{t = 3}$ **mm**	**full tank** $\boldsymbol{t = 4}$ **mm**
flange concavity for bottom impact	4.69 mm	14.4 mm	11.0 mm
bending moment for bottom impact	4.87 N.m	5.81 N.m	5.62 N.m
vertex concavity for vertex impact	7.80 mm	16.0 mm	14.4 mm

Steady state vibrations

The vibrations of the frame of tamping machine are recorded by three-axial acceleration transducer and vibrational level meter. Three records have been done measuring at one point of the frame and the recorded signals have been proceeded by FFT (Fast Fourier Transformations) (fig. 9). The significant frequency amplitudes are taken only and averaged, so the excitation base motion of the tank has harmonics as follows in the table 3.

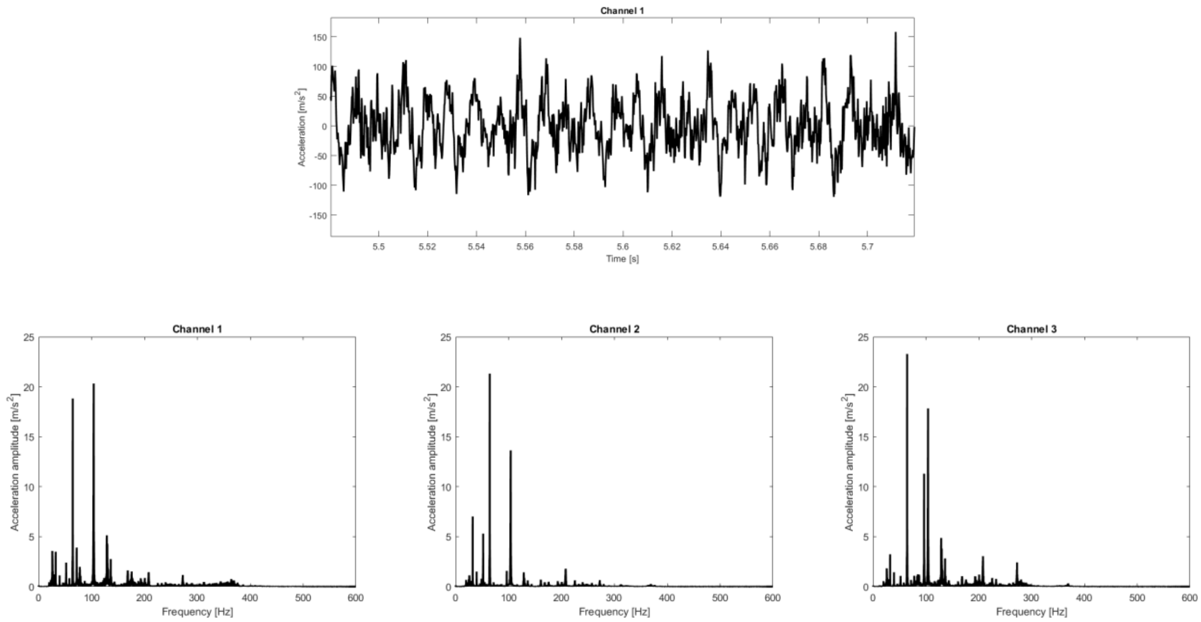


Fig. 9. A record of vibration and the fast Fourier transform (FFT) of the three records

The base motion nodes are chosen to be the contact nodes of the tank to the protective frame. Modal analysis of steady state vibrations is carried out at damping ratio of $\zeta = 0.017$. The calculated frequencies are 500 in the range 40-800 Hz. The inspection of the effective stress at all frequencies shows that the maximum is 8 MPa at 102.44 Hz. The place of the maximum, however, is a fake part of the model with stress concentration. The real maximum can be found by the fringe and the diagram of effective stress there shows that the maximum acceleration amplitude is 5.59 MPa. (fig. 10)

 Copyrights© 2020 ISBN 978-954-712-826-2 (Print)

Table 3. The harmonics of the excitation base motion of the tank

channel	1-x		2-y		3-z		
harmonics	1	2	1	2	1	2	3
frequency [Hz]	63.5	103	63.5	103	63.5	96	103
amplitude [m/s ^{2}]	15	16	16	12	19	11	14
phase angle [°]	0	0	90	90	180	180	180

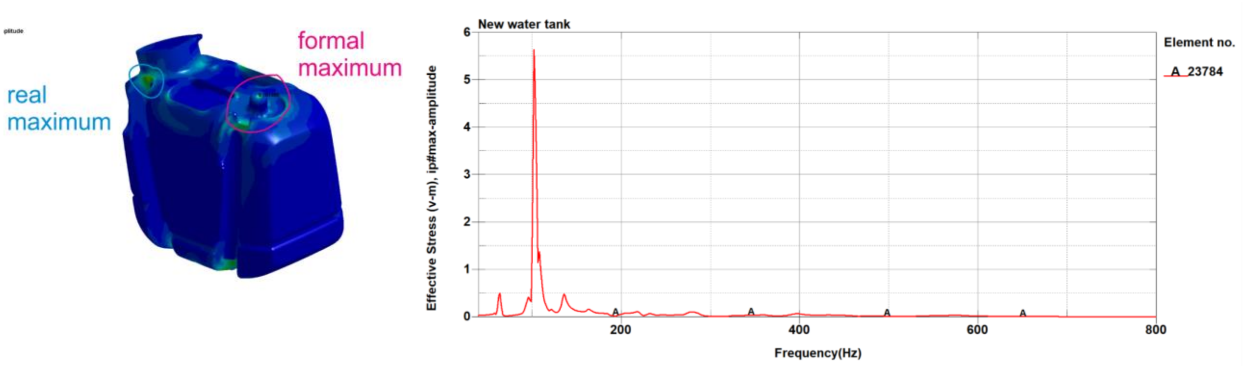


Fig. 10. Modal analysis of steady state vibrations at damping ratio of $\zeta = 0.017$

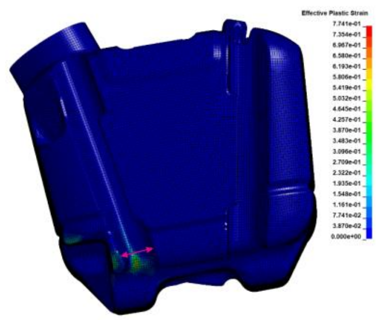
The steady state analysis of the vibrations shows that the highest amplitude of effective stress in the tank is below the half of ultimate strength of the material and therefore no fatigue and the design is reliable. However, the impact simulations show that the tank is vulnerable at the flange and the vertexes and the occasional impact could cause functional disability or residual change of shape. A new improved design is necessary in order to avoid the shortcomings.

Analysis of improved tank design

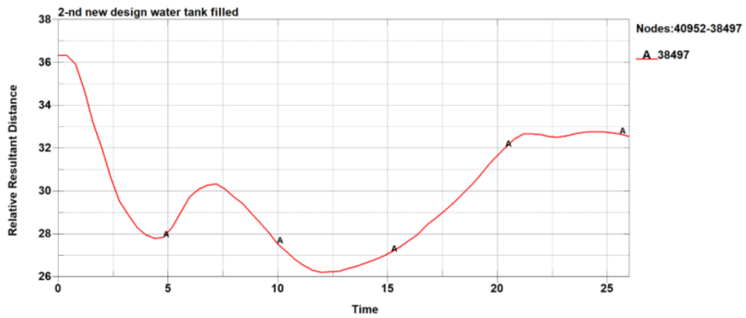
Impact simulations are carried out at the same conditions as the previous simulations of the last tank design. Only filled by water tanks are considered.

The maximum effective plastic strain after plane bottom impact is 18.9% and no residual shape change is observed. The bottom edge impact simulations of filled by water tank show high impact force as a result of water inertia. The maximum effective plastic strain is a little bit higher now – 24.1%, but no residual shape change is observed.

The bottom vertex impact simulations of filled by water tank impact show that the tank is the most vulnerable for bottom vertex impact. The maximum effective plastic strain is very high now – 77.4% (fig. 11a), and the distance between the channel walls is changed by 4 mm (fig. 11b).



a)



b)

Fig. 11. Results of effective a) plastic strain and b) distance change after vertex bottom impact

The steady state vibration simulations are carried out under the same conditions as the previous tank design simulations. By inspection, we found that the maximum bending moment in beam elements is 2090 N.mm (fig. 12a), and ignoring the fake stress maximum at the rod cap (fig. 12b), the maximum effective stress in shell elements is 2.99 MPa, both at 62.8 Hz frequency (fig.12c).

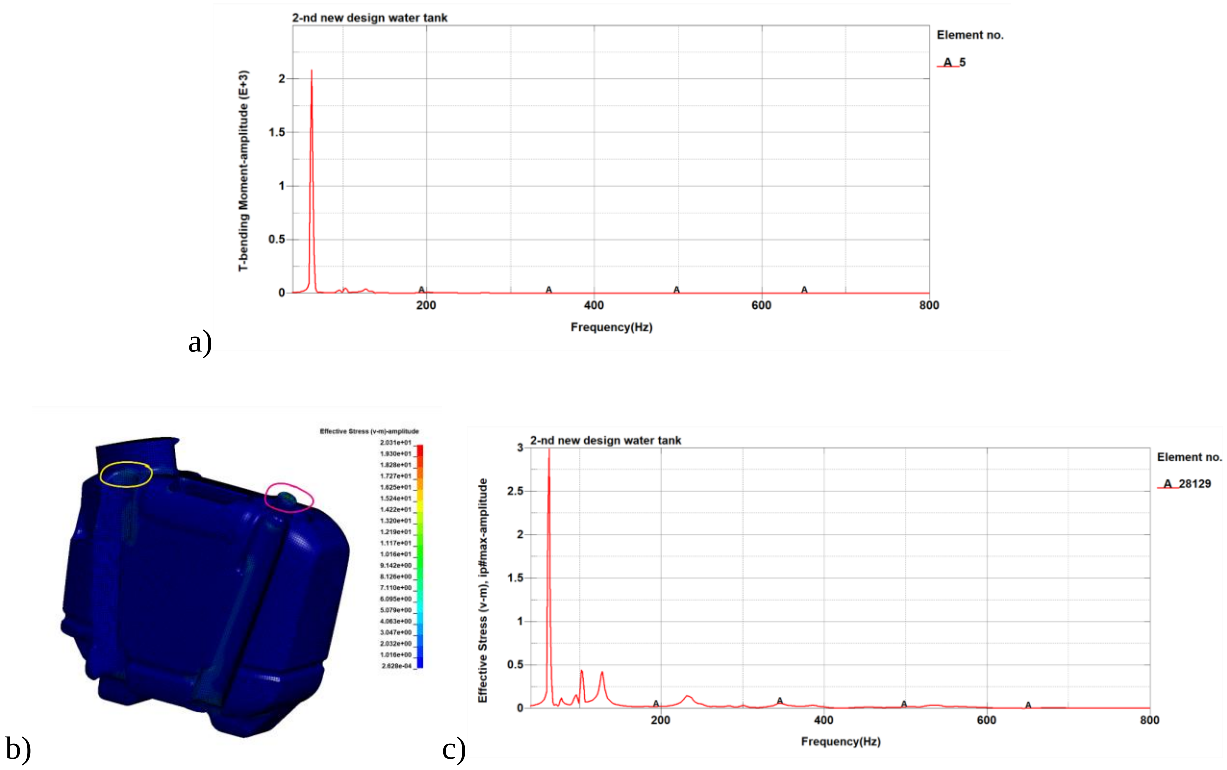


Fig. 12. Results from steady state vibration simulations

CONCLUSION

The simulation analysis of the first new design of plastic water tank shows that the tank is vulnerable to occasional impact and an improvement of the design was necessary. The simulations of impact and steady state vibrations of the improved tank design show that it has enough occasional impact survival ability and reliability under vibrational loading. The simulation analysis of two different designs has saved a lot of resources for prototype manufacturing and testing. The simulation driven design is very efficient method for the modern industry.

ACKNOWLEDGMENT

This paper presents results of the work on project No. 2020-MTF-02, funded by "Angel Kanchev" University of Ruse.

REFERENCES

Hallquist J. (2006). *LS-DINA Theory manual*. Livermore Software Technology Company

Souli, M. & Zolesio, J.P. (2001). Arbitrary Lagrangian-Eulerian and free surface methods in fluid mechanics. *Computer Methods in Applied Mechanics and Engineering* 191, 451-466.

Sun, W.-K., Zhang, L.-W., & Liew, K.M. (2020). A smoothed particle hydrodynamics–peridynamics coupling strategy for modeling fluid–structure interaction problems. *Computer Methods in Applied Mechanics and Engineering* 371, 113298.

 Copyrights© 2020 ISBN 978-954-712-826-2 (Print)