DINAMIČKA ANALIZA KONSTRUKCIJE MIKSERA ZA MIJEŠANJE TEČNOSTI UNUTAR REZERVOARA

DYNAMIC ANALYSIS OF THE MIXER CONSTRUCTION FOR MIXING LIQUIDS INSIDE THE TANK

Elma Ekinović [1]
Amel Karić [1]
Adnan Barlov [1]
Kenan Šabanović [1]

[1] *University of Zenica, Faculty of Mechanical Engineering*

Ključne riječi:
dinamika, frekvencija, mikser, oscilacije, rezervoar

Keywords:
dynamics, frequency, mixer, oscillations, tank

Paper received:
19. 02. 2024.
Paper accepted:
15. 04. 2024.

Stručni rad

REZIME

Cilj ovog rada jeste provesti dinamičku analizu velikog industrijskog miksera koji se koristi za miješanje tečnosti unutar vertikalnog rezervoara. Analiza je provedena numerički koristeći softverske alate SolidWorks *i* Ansys *za analize na bazi metode konačnih elemenata. Komparativna analiza rezultata dobijenih iz oba softvera je prikazana. Iako postoje odstupanja u rezultatima usljed razlika u softverima, ukupni rezultati su zadovoljavajući. Najveće odstupanje je primijećeno kod maksimalnog ekvivalentnog napona na krovu rezervoara. Konstrukcija pokazuje elastično ponašanje pod oscilatornim opterećenjem, održavajući svoj integritet. Dok razlike u 3D modeliranju utječu na raspodjelu napona, vlastite frekvencije u softverima se poklapaju s analitičkim proračunom. Nakon prestanka djelovanja opterećenja, struktura se smiruje prema očekivanjima, ukazujući na dobro postavljenu dinamičku analizu. Unatoč izazovima u smislu lokalizacije napona, studija potvrđuje validnost provedene analize, uzimajući u obzir kompleksnost strukture.*

Professional paper

SUMMARY

The aim of this paper is to perform a dynamic analysis of a large industrial mixer used for mixing liquids inside a vertical tank. The analysis is performed numerically using software tools SolidWorks and Ansys for finite element method analysis. A comparative analysis of results obtained from both software platforms is presented. Although there are discrepancies in the results due to software differences, the overall results are satisfactory. The largest discrepancy was noticed in the maximum equivalent stress on the tank roof. The structure exhibits elastic behavior under oscillating loading, keeping its integrity. While deficiencies in 3D modeling affect stress distribution, natural frequencies in software coincide with analytical calculations. Following loading, the structure settles as expected, indicating well-established dynamic analysis. Despite challenges in stress localization, the study confirms the validity of the conducted analysis, considering the complexity of the structure.

1. INTRODUCTION

The task is to conduct a dynamic analysis of the structure of a large industrial mixer used for liquid mixing within a vertical tank. This analysis is performed numerically using commercial software tools SolidWorks and Ansys, based on the finite element method. Since the analysis is conducted by using two software platforms, a comparison of the obtained results will be made at the end of the study, followed by conclusions. The schematic diagram of the tank structure with the integrated mixer is shown on Figure 1. The figure shows that in the case of a vertical tank, the mixer, consisting of a drive motor, shaft, and blades, is connected to the tank's roof. In order to optimize the finite element mesh, and assuming the tank shell and bottom being irrelevant for dynamic analysis, the construction is represented as the model shown on Figure 2. Additionally, the drive motor is not of interest and is therefore excluded from the analysis. Thus, the focus of this analysis is to determine the dynamic state of the tank roof, mixer shaft and blades, as well as the junction of the mixer with the roof, which contains bolted connections.

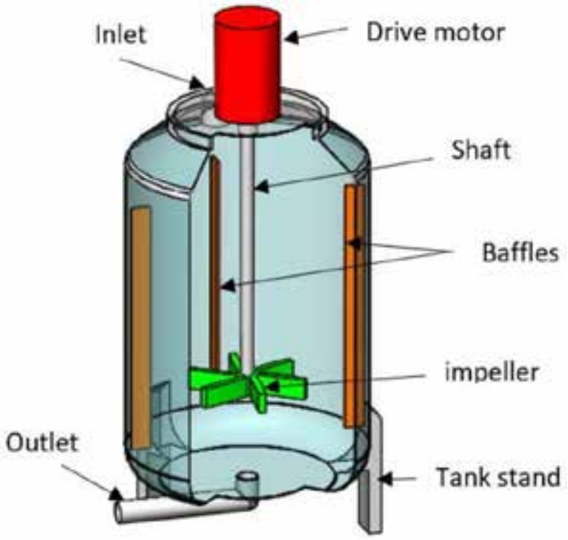


***Figure 1** Principle design of a tank with a built-in mixer [1]*

The following image provides an illustration of the construction used for analysis in this study.

***Figure 2** Construction for dynamic analysis*

The Figure 2 shows the stiffeners welded to the bottom surface of the roof are, also, included in the analysis. They ensure there are no large deflections of the roof due to self-weight of the mixer connected to it. Under operating conditions of the tank, the mixer works continuously and rotates at 89 rpm, which is not a high angular velocity. Actually, the higher angular velocity is not necessary as it is only important that it constantly rotates, so the liquid (or mixture of several liquids) in the tank retains its characteristics and there is no possible sedimentation at the inside bottom of the tank.

2. FREQUENCY ANALYSIS OF THE MIXER SHAFT

Given the numerical simulations showing the dynamic behaviour of a structure as extremely demanding, especially with complex structures such as the one being analysed, it is necessary to first determine the validity of the concept of conducting those simulations and the way of setting boundary conditions. For this purpose, in this work, a frequency analysis of a simplified construction is first carried out, i.e. only the mixer shaft as the part which is assumed to be most exposed to its own oscillations due to its dimensions and position. For simple constructions there are already derived analytical expressions for natural frequencies, and they are used as a check of the validity of the numerical simulations. Therefore, an analytical calculation was done first, which gave the expected results, and then numerical simulations were used to confirm these results.

2.1. Analytical calculation

The shaft itself can be seen as a console, so on the upper side there is a fixed support and on the lower side it is free, i.e. there is no support whatsoever. The shaft is made of stainless steel 1.4404 (X2CrNiMo17-12-2) and the cross-section is full circular and of constant diameter with the exception of a small part at the top where the pin groove is located. The natural circular frequencies and the natural frequencies of the first three tones of the shaft oscillation are of interest for analytical calculation. The natural circular frequencies of the first three tones of the unloaded cantilever oscillation are according to [2]:

$$\omega_1 = 1{,}875^2 \sqrt{\frac{EI}{\rho A L^4}}\ [rad/s] \tag{1}$$

$$\omega_2 = 4{,}694^2 \sqrt{\frac{EI}{\rho A L^4}}\ [rad/s] \qquad (2)$$

$$\omega_3 = 7{,}855^2 \sqrt{\frac{EI}{\rho A L^4}}\ [rad/s] \qquad (3)$$

Values used in these equations are:

$E = 200\ GPa$ - modulus of elasticity of steel from which the shaft is made,

I - axial moment of inertia of the shaft section,

$d = 50\ mm$ - shaft diameter,

$\rho = 8000\ kg/m^3$ - the density of the steel from which the shaft is made,

A - cross-sectional area of the shaft,

$L = 2950\ mm$ - shaft length.

Based on the above equations, the natural circular frequencies of the first three tones are:

$\omega_1 = 25{,}249\ rad/s$

$\omega_2 = 158{,}242\ rad/s$

$\omega_3 = 443{,}127\ rad/s.$

The natural frequencies of the first three tones of oscillation are related to the circular natural frequencies in the following way:

$$f_1 = \frac{\omega_1}{2\pi}\ [Hz] \qquad (4)$$

$$f_2 = \frac{\omega_2}{2\pi}\ [Hz] \qquad (5)$$

$$f_3 = \frac{\omega_3}{2\pi}\ [Hz]. \qquad (6)$$

By entering the values, the natural frequencies of the first three tones of the oscillation are:

$f_1 = 4{,}018\ Hz$

$f_2 = 25{,}185\ Hz$

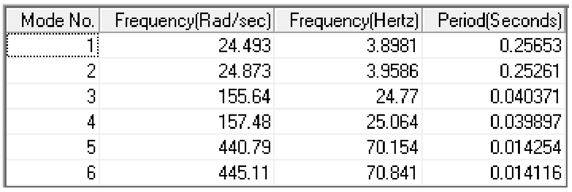
$f_3 = 70{,}526\ Hz.$

2.2. Numerical calculation

The numerical calculation was performed in SolidWorks and Ansys software platforms. Given the calculation settings are the same, here will be shown those common settings and then the individual results obtained in each software. To set up an analysis in SolidWorks, it is necessary to go to the Simulation module and select the Frequency Study type of analysis, while in Ansys, the Modal analysis type is selected. Previously mentioned material 1.4404 is then assigned to the shaft, and since it is only one component, it is not necessary to assign a mode of interaction between the components. Since it is a console, it is necessary to place a fixed support on the upper side of the shaft, which is in the place of the groove where the pin comes on the structure, according to the Figure 3.

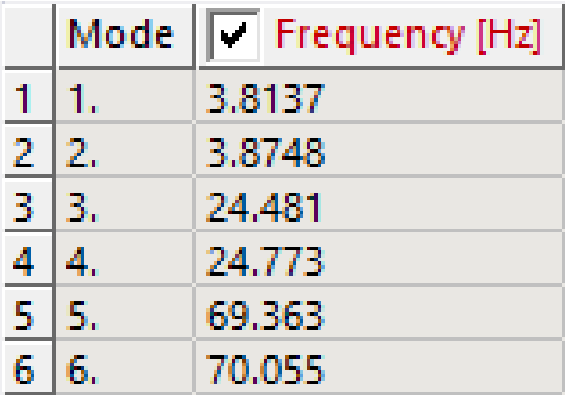
Figure 3 *Fixed support in shaft frequency analysis - SolidWorks (left) and Ansys (right)*

From the loads acting on the shaft, only the own weight of the shaft (Gravity in SolidWorks or Standard Earth Gravity in Ansys) was selected, in order to achieve better accuracy of the results. As for the finite element mesh, a global mesh size of 10 mm is chosen and the mesh element type is tetrahedral. Solvers are automatic, that is, the software itself chooses the best solution for the given problem. The obtained results are given in Figure 4 and Figure 5.



Mode No.	Frequency(Rad/sec)	Frequency(Hertz)	Period(Seconds)
1	24.493	3.8981	0.25653
2	24.873	3.9586	0.25261
3	155.64	24.77	0.040371
4	157.48	25.064	0.039897
5	440.79	70.154	0.014254
6	445.11	70.841	0.014116

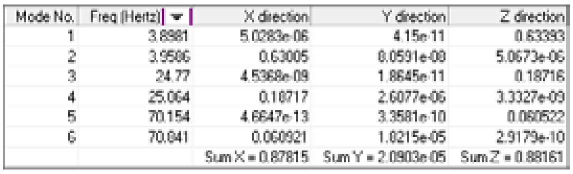
Figure 4 *Results of natural frequencies for the first three tones of shaft oscillation in SolidWorks*



	Mode	☑ Frequency [Hz]
1	1.	3.8137
2	2.	3.8748
3	3.	24.481
4	4.	24.773
5	5.	69.363
6	6.	70.055

Figure 5 *Results of natural frequencies for the first three tones of shaft oscillation in Ansys*

It is noted here that in cases where the body is symmetrical, the SolidWorks and Ansys software platforms give two results for each tone of oscillation, such as the observed shaft which is free from the bottom side. These two results actually represent bending oscillations in two orthogonal planes, which can also be seen in the Figure 6 following the mass distribution during oscillation. It can be seen that the distribution of masses in Mode 1 and Mode 2, then Mode 3 and Mode 4, and Mode 5 and Mode 6 is almost identical, only the values along the X and Z axes have been replaced.



Mode No.	Freq (Hertz)	X direction	Y direction	Z direction
1	3.8981	5.0283e-06	4.15e-11	0.63393
2	3.9586	0.63005	8.0591e-08	5.0673e-06
3	24.77	4.5368e-09	1.8645e-11	0.18716
4	25.064	0.18717	2.6077e-06	3.3327e-09
5	70.154	4.6647e-13	3.3581e-10	0.060522
6	70.841	0.060921	1.0215e-05	2.9179e-10
		Sum X = 0.87815	Sum Y = 2.0903e-05	Sum Z = 0.88161

Figure 6 *Mass distribution in the first three tones of oscillation*

2.3. Comparison of results

The comparison of values of natural frequencies of the first three tones of shaft oscillation are given in Table 1. It contains the combined presentation of the results for the first three natural frequencies obtained analytically and numerically. It is obvious the numerically obtained results largely agree with the analytical values due to very small deviations.

The largest deviation of the natural frequency is about 5% and it is the only deviation greater than 3%, which means that the numerical method of calculation is relevant. From here it can be concluded the desired goal of conducting the frequency analysis of the shaft itself has been achieved, i.e. the validity of the concept of conducting numerical simulations has been established, and the dynamic analysis of the entire structure can now be entered into.

It is evident from the comparison that the numerically obtained results closely align with the anticipated analytical values, exhibiting minimal disparities. Consequently, the successful completion of the frequency analysis of the shaft underscores the feasibility of conducting numerical simulations, thereby paving the way for the comprehensive dynamic analysis of the entire structure.

Table 1 Comparison of the results od the natural frequencies of the first three tones of shaft oscillation

Oscillation tone	Natural frequency, Hz			Deviation		
	Analytical calculation	SolidWorks calculation	Ansys calculation	SolidWorks/ Analytical	Ansys/ Analytical	Ansys/ SolidWorks
1	4,018	3,898	3,814	-2,987 %	-5,077 %	-2,155 %
2	25,185	24,770	24,481	-1,648 %	-2,795 %	-1,167 %
3	70,526	70,154	69,363	-0,527 %	-1,649 %	-1,128 %

3. DYNAMIC ANALYSIS OF MIXER CONSTRUCTION

Here, in addition to the previously analysed shaft, all other observed components of the structure are included in the analysis. Since it is a complex construction, it is not possible to carry out an analytical calculation to obtain tentative results, but only numerical calculations are done. In order to somehow determine the validity of the simulation settings, it is again necessary to carry out a frequency analysis of the entire structure in both software platforms, where its unloaded behaviour will be seen.

3.1. Frequency analysis

Given the calculation settings in both software are the same, those common settings will be displayed and then the results obtained in each of the software individually. To start the analysis in SolidWorks, it is necessary to go to the Simulation module and select the Frequency Study type of analysis, while in Ansys, the Modal analysis type is selected. On the construction, stiffener ends as well as outside roof surface are welded to the tank shell, so the surfaces on which a fixed support is set are shown on Figure 7 and Figure 8.

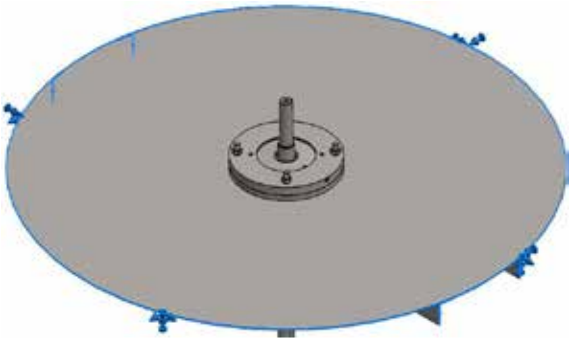


***Figure** 7 Fixed support in frequency analysis of construction – SolidWorks*

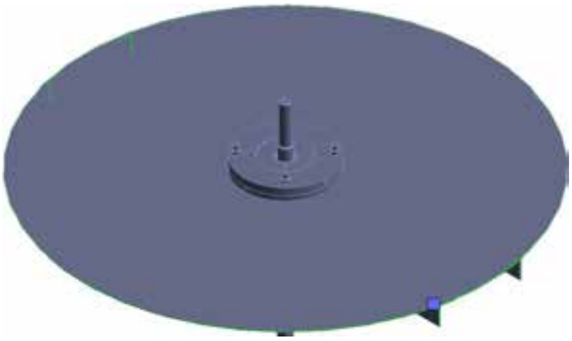
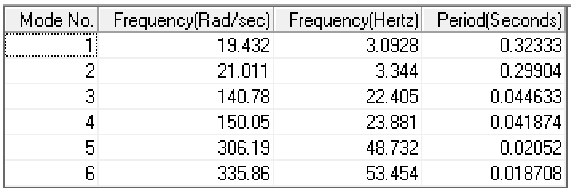


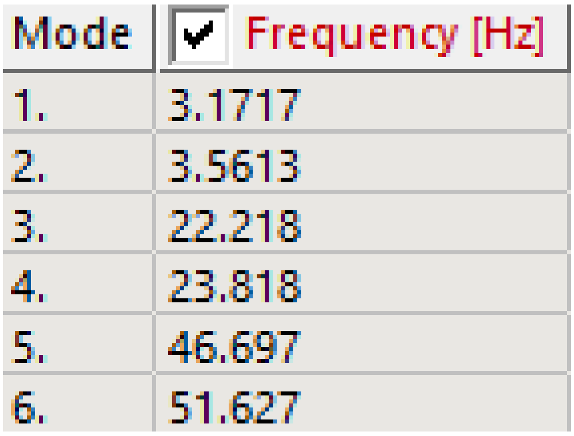
***Figure 8** Fixed support in frequency analysis of construction - Ansys*

From the loads acting on the shaft, only the own weight of the structure (Gravity in SolidWorks or Standard Earth Gravity in Ansys) is chosen, in order to achieve better accuracy of the results.



Mode No.	Frequency(Rad/sec)	Frequency(Hertz)	Period(Seconds)
1	19.432	3.0928	0.32333
2	21.011	3.344	0.29904
3	140.78	22.405	0.044633
4	150.05	23.881	0.041874
5	306.19	48.732	0.02052
6	335.86	53.454	0.018708

***Figure 9** Results of natural frequencies for first three tones of structure oscillation in SolidWorks*



Mode	☑ Frequency [Hz]
1.	3.1717
2.	3.5613
3.	22.218
4.	23.818
5.	46.697
6.	51.627

***Figure 10** Results of natural frequencies for first three tones of structure oscillation in Ansys*

The results of the analyses are given in Figure 9 and Figure 10. Again, due to the symmetry of the model, the results for Modes 1, 3 and 5 from the previous images are adopted for the first three oscillation tones, so the first oscillation tone represents Mode 1, the second oscillation tone represents Mode 3, and the third oscillation tone represents Mode 5. Within the figure it can be seen that compared to the shaft itself, slightly lower natural oscillation frequencies are obtained, which is expected because an additional mass in the form of blades is placed at the bottom of the shaft, increasing the inertia of structure. The obtained results in both software platforms match quite well, serving as a sufficient sign that the simulations are well set and one can enter the dynamic analysis.

3.2. Dynamic analysis setup

Dynamic analysis represents the analysis when the structure is submitted to load. In this case, the load is represented by the rotation of the shaft and the blades with number of revolutions of 89 rpm. To set up an analysis in SolidWorks, it is necessary to go to the Simulation module and select the Modal Time History analysis type, while in Ansys, the Transient Structural analysis type is selected.

Given the SolidWorks software, unlike Ansys, does not allow in this type of analysis to specify the rotation of the component, that rotation is converted into forces depending on the mass of components and the number of revolutions. For the credibility of the calculation, this way of assigning rotation is performed in both analysis software platforms. SolidWorks also does not allow the self-weight gravity load to be set in the dynamic analysis, so it is necessary to take into account the mass of the rotating parts when setting the forces. The mass of the shaft is 45,22 kg, and the mass of the blades is 9,57 kg, so the total mass of the rotating parts is m_1 = 54,79 kg. The default number of revolutions is n = 89 rpm, which, converted into a circular frequency, amounts to:

$$\omega = \frac{n\pi}{30} = \frac{89 \cdot \pi}{30} = 9{,}32\, rad/s \tag{7}$$

$$f = \frac{\omega}{2\pi} = \frac{9{,}32}{2 \cdot \pi} = 1{,}48\, Hz. \tag{8}$$

The radius of the shaft is actually the radius of revolution and it is r_1 = 25 mm, so the normal component of acceleration is:

$$a_{n1} = \omega^2 r_1 = 2{,}17\, m/s^2, \tag{9}$$

and the force is therefore:

$$F_1 = m_1 \cdot a_{n1} = 118{,}89\, N. \tag{10}$$

In order for this force to simulate a real circular motion, it is necessary to apply two forces of the same intensity and on the same surfaces in both software platforms, which simulate a sine and a cosine function. It is chosen that the force is given as if it acts during period of one revolution, that is, the load is adopted as during one revolution of the shaft and blades. For this purpose, it is necessary to determine the period of one revolution, which amounts to:

$$T = \frac{1}{f} = \frac{1}{1{,}48} = 0{,}67\, s. \tag{11}$$

If the simulation were to be started with the current settings, the real situation would not be obtained because on the 3D model everything is perfect and nothing would cause the oscillations that are always present, not even rotating of shafts and blades. Because of this, it is necessary to physically add some eccentricity, which means assuming the situation is not quite perfect. It is concluded it is the most favorable that eccentricity represents the imperfection of geometry and construction of the blades, as well as the possible adhesive mass on some of them during, for example, the operation of the mixer and mixing of the solution in the tank. It is chosen for analysis needs that the eccentricity represents a mass of 1 kg.

Again, it is necessary to determine the force to be set in analysis. The radius of revolution in this case is distance from axis shaft to middle of blades and it is r_2 = 260 mm, so the normal component of acceleration is in this case:

$$a_{n2} = \omega^2 r_1 = 22{,}58\, m/s^2, \tag{12}$$

and the force is therefore:

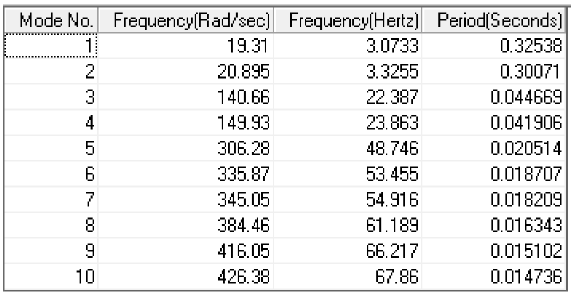
$$F_2 = m_2 \cdot a_{n2} = 22{,}58\, N. \tag{13}$$

This new force is assigned completely analogously to the previous one, with the only difference being the surface on which it is applied, where instead of the shaft, it acts on the surfaces of the blades at the connection with the shaft.

The analysis is set to last a total of three rotation periods, to see how the structure calms down after one rotation. Duration of the analysis is set to 2,01 s. Time increment is set to 0,05 which means a total of 41 analysis steps. After the dynamic analysis is completed, the obtained results are presented.

3.3. Dynamics analysis results

As far as the obtained results of the dynamic analysis are concerned, the structure displacements and the stress state in the structure are of greatest interest. These results will mainly be analysed in this section. Before that, it is necessary to see again the natural frequencies of the structure, but this time with added mass, as shown in the Figure 11.



Mode No.	Frequency(Rad/sec)	Frequency(Hertz)	Period(Seconds)
1	19.31	3.0733	0.32538
2	20.895	3.3255	0.30071
3	140.66	22.387	0.044669
4	149.93	23.863	0.041906
5	306.28	48.746	0.020514
6	335.87	53.455	0.018707
7	345.05	54.916	0.018209
8	384.46	61.189	0.016343
9	416.05	66.217	0.015102
10	426.38	67.86	0.014736

Figure 11 *Natural frequencies of structure with added mass*

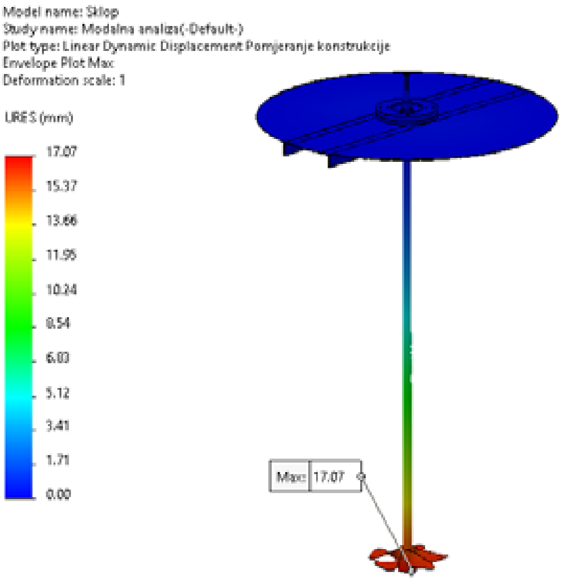


Figure 12 *Maximum displacement of the structure – SolidWorks*

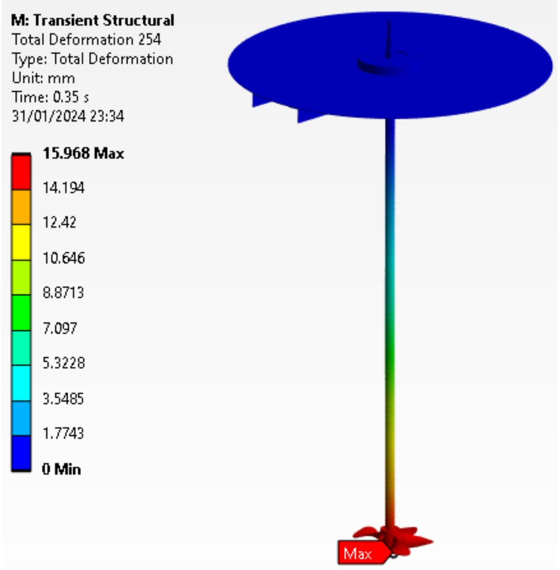


Figure 13 *Maximum displacement of the structure - Ansys*

Figure 12 and Figure 13 show the largest movements during the set duration of the analysis, i.e. the moment when the largest movement occurs. Given the bottom of the shaft, together with the blades, is the part of the structure farthest from any supports, it was logical to expect that this is where the greatest displacement will occur compared to the initial position without load. The values obtained in both software platforms are very close.

As for the stresses on the tank roof, it is found that the maximum equivalent stresses occur around the opening for the shaft and near the place where the opening is closest to the stiffeners, as expected, because these are the most loaded places of the roof. Other parts of the roof are less exposed to stresses. The resulting stresses are quite low (Figure 14 and Figure 15).

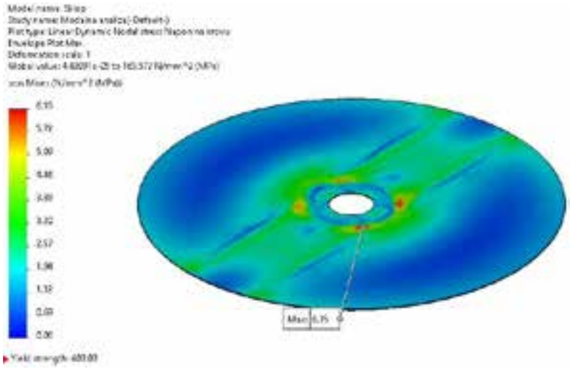


Figure 14 *Maximum equivalent roof stresses – SolidWorks*

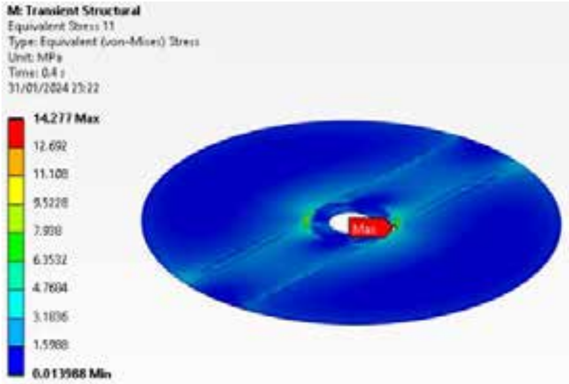


Figure 15 *Maximum equivalent roof stresses – Ansys*

The roof displacements, also, occur as expected. Figure 16 and Figure 17 clearly show how the displacement of the central part of the roof and on the outside cylindrical sufrace of the roof is close or equal to zero, because of stiffeners and fixed supports placed there. The largest movement occurs in the parts of the roof furthest from the supports.

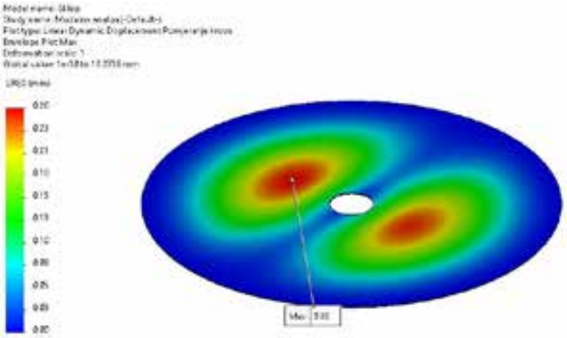


Figure 16 *Maximum displacement of the roof - SolidWorks*

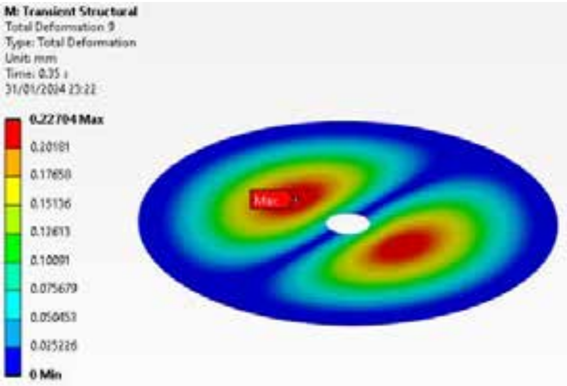


Figure 17 *Maximum displacement of the roof – Ansys*

On the blades, the maximum equivalent stresses in software platforms appear relatively similar in values, but in different places. According to SolidWorks, due to the construction of the blades themselves, the largest stress occurs at the rounding of the blades (Figure 18), while according to Ansys, a good part of the stress occurs at that place as well, but it is still the largest on the outer surface of the blade connection with the shaft (Figure 19). It is not necessary to show displacements of the blades in relation to the initial position, because they are already covered earlier.

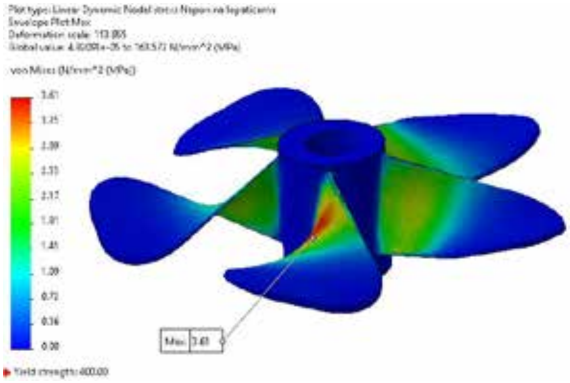


Figure 18 *Maximum equivalent stresses on blades - SolidWorks*

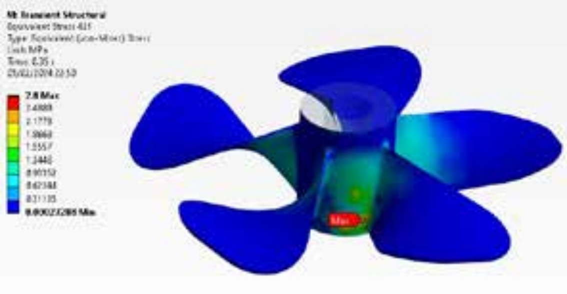


Figure 19 *Maximum equivalent stresses on blades – Ansys*

The stiffeners located under the roof are additionally deflected because of the roof, so slightly higher maximum stresses are expected for them compared to the roof (Figure 20 and Figure 21), while the displacements are expected to be slightly smaller due to the fact that their role is to reduce displacements. Exactly such results are obtained and shown on Figure 22 and Figure 23.

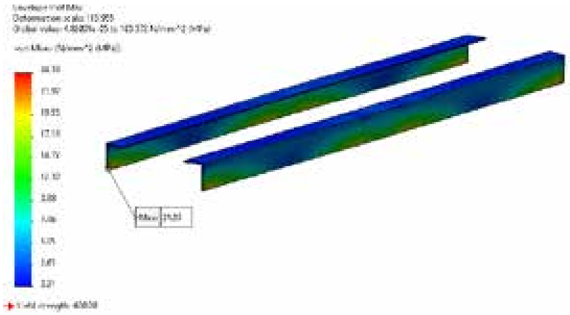


Figure 20 *Maximum equivalent stresses on stiffeners – SolidWorks*

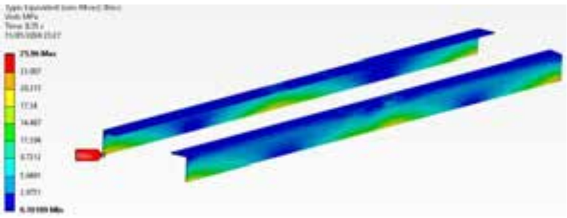


Figure 21 *Maximum equivalent stresses on stiffeners – Ansys*

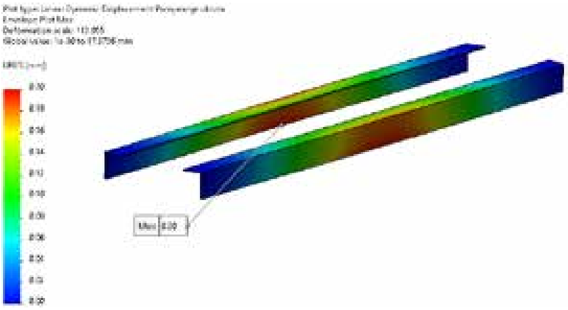


Figure 22 *Maximum displacement of stiffeners - SolidWorks*

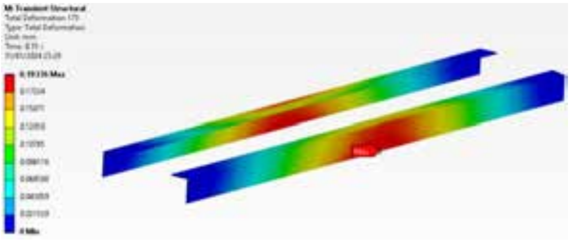


Figure 23 *Maximum displacement of stiffeners - Ansys*

The highest stresses on the ring that represents connection between the shaft and the roof of the tank occur on the lower outer cylindrical edge, what is logical, because that edge is directly connected to the roof of the tank (Figure 24 and Figure 25). The displacements are expected to be almost the same as in the case of stiffeners, and that exactly is obtained (Figure 26 and Figure 27).

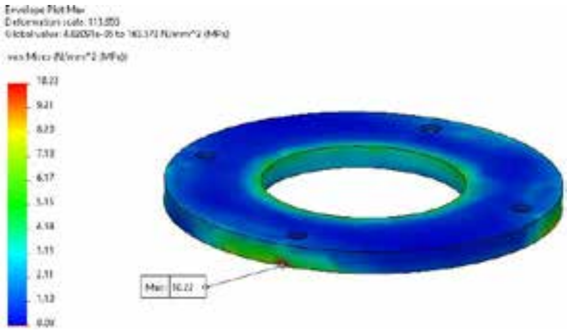


Figure 24 *Maximum equivalent stresses on connection ring – SolidWorks*

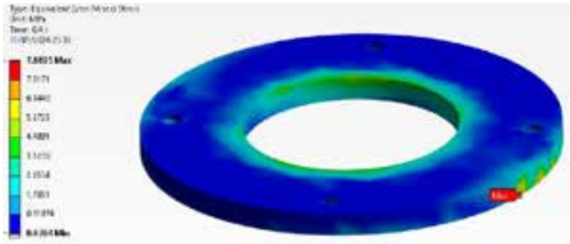


Figure 25 *Maximum equivalent stresses on connection ring – Ansys*

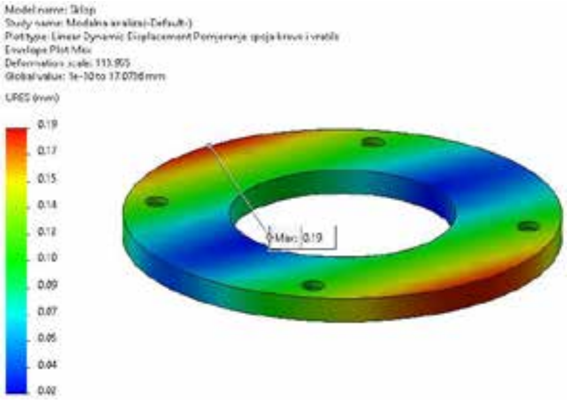


Figure 26 *Maximum displacement of connection ring – SolidWorks*

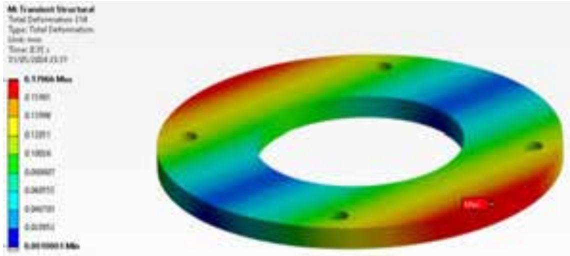


Figure 27 *Maximum displacement of connection ring – Ansys*

The following two figures, Figure 28 and Figure 29, show another interesting diagram extracted from both software platforms. It is a diagram of the change in the maximum displacement of the complete structure through individual steps of the analysis. One can see their fine matching. The moment in which the action of the load stops is the moment in which the settlement of the structure begins, but it can be seen that for the set time of the load acting the structure does not settle down.

Table 2 contains an overview and comparison of the obtained results of the observed quantities of this dynamic analysis in both SolidWorks and Ansys, i.e. the maximum values obtained.

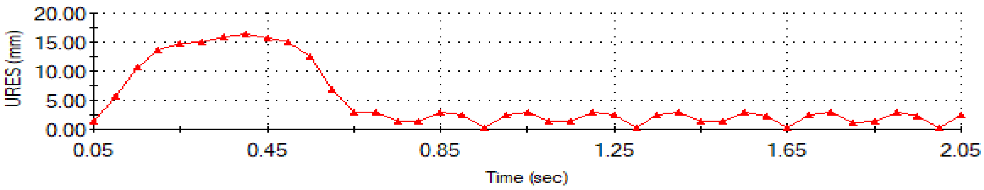


Figure 28 *Diagram of maximum displacement of structure through steps – SolidWorks*

Figure 29 *Diagram of maximum displacement of structure through steps – Ansys*

Table 2 Overview and comparison of the obtained results of dynamic analysis

Quantity	SolidWorks value	Ansys value	Deviation of Ansys to SolidWorks
Displacement of complete construction	17,07 mm	15,97 mm	-6,44 %
Equivalent stress on roof	6,35 MPa	14,28 MPa	124,88 %
Displacement of roof	0,26 mm	0,23 mm	-11,54 %
Equivalent stress of blades	3,61 MPa	2,80 MPa	-22,44 %
Equivalent stress of stiffeners	24,39 MPa	25,96 MPa	6,44 %
Displacement of stiffeners	0,20 mm	0,19 mm	-5 %
Equivalent stress of connections ring	10,22 MPa	7,89 MPa	-22,80 %
Displacement of connection ring	0,19 mm	0,18 mm	-5,26 %

4. CONCLUSION

The overall results are quite satisfactory. The values cannot ideally match due to the difference in the nature of operations of the two pieces of software.

There is only one large deviation in terms of the maximum equivalent stress on the roof of the tank. However, the values are small, the difference between the two values is only around 8 MPa, so this result can be taken as acceptable. No extreme values of stress and displacement appear anywhere, the structure remains deep in the area of elasticity as it should, because the load on the structure is not too great. It can be seen after the end of the load, the structure settles down, as it should, providing evidence to conclude that the observed dynamic analyses are well established.

The biggest problem regarding the dynamic analysis is the maximum equivalent stress of the complete structure, which appears in the wrong places and with the wrong values. The reason for this is the imperfection of the 3D model of the shaft. However, as the obtained values of natural frequencies of the structure match in both software platforms, but also with the expected values obtained by analytical calculation in the case of the observed shaft itself, the conclusion is that the performed analyses are valid taking into account the bulkiness of the structure.

The final conclusion is that the structure is well-constructed, having no weak spots sensible to vibrations which could damage its integrity in working conditions.

5. REFERENCES

[1] Torotwa, Ian and Changying Ji. "A Study of the Mixing Performance of Different Impeller Designs in Stirred Vessels Using Computational Fluid Dynamics." *MDPI Online Journal* (March 8, 2018), https://www.mdpi.com/2411-9660/2/1/10.

[2] Brčič, Vlatko. *Dinamika konstrukcija*. Beograd: Univerzitet u Beogradu, 1978.

[3] He, Jimin and Zhi-Fang Fu. *Modal Analysis*. Oxford, 2001.

[4] Patwary, Anayet U., Waleed F. Faris, K. M. Nurul Amin and S. K. Loh. "Dynamic Modal Analysis of Vertical Machining Centre Components." *Hindawi Online Journal* (February 2, 2010), https://www.hindawi.com/journals/aav/2009/508076/.

[5] Katsikadelis, John T. *Dynamic Analysis of Structures*. Athens: National Technical Univesity of Athens, 2020.

Corresponding author:
Adnan Barlov
University of Zenica, Faculty of Mechanical Engineering
Email: adnan.barlov.23@dl.unze.ba
Phone: + 387 61 704 690