ГОДИШНИК НА УНИВЕРСИТЕТА ПО АРХИТЕКТУРА, СТРОИТЕЛСТВО И ГЕОДЕЗИЯ
СОФИЯ

Том Volume 52 | 2019 | Брой Issue 3

ANNUAL OF THE UNIVERSITY OF ARCHITECTURE, CIVIL ENGINEERING AND GEODESY
SOFIA

Received: 05.03.2019 г.
Accepted: 31.10.2019 г.
Published on line: 08.11.2019

NUMERICAL ANALYSIS OF VERTICAL STEEL STORAGE TANKS IN ANSYS WORKBENCH AND ANSYS MECHANICAL APDL

M. Pantusheva[1], L. Zdravkov[2]

Keywords: *steel storage tank, structure, liquid, interaction, finite elements, ANSYS*

ABSTRACT

Vertical cylindrical steel storage tanks are industrial facilities in which the solid steel shell and the contained deformable fluid are in contact and interact with each other. Modelling of these two media and their interaction requires the use of appropriate software, such as ANSYS. In an effort to meet the demands of new users, ANSYS has created the Workbench graphics interface. It is user-friendly and in that manner has an advantage over the classic Mechanical APDL appearance. Specific for Workbench is the differentiation of modules each specialized in solving tasks from a certain mechanic's field. However, the interaction between solids and fluids is an interdisciplinary problem that could be solved using different modules of the program. The crucial question here is whether those computing modules can be used for purposes they were not originally created for? And moreover, how reliable would the results be? In an effort to answer these questions the current paper presents a study on the behaviour of a water-filled steel storage tank analysed using static, modal and response spectrum analysis in ANSYS Workbench and in ANSYS Mechanical APDL. The obtained solution results are compared with analytical calculations.

[1] Mariya Pantusheva, Eng., Dept. "Steel, Timber and Plastic Structures", UACEG, 1 H. Smirnenski Blvd., Sofia 1046, e-mail: maria.pantusheva@gmail.com

[2] Lyubomir Zdravkov, Assoc. Prof. Dr. Eng., Dept. "Steel, Timber and Plastic Structures", UACEG, 1 H. Smirnenski Blvd., Sofia 1046, e-mail: zdravkov_fce@uacg.bg

1. Introduction

Steel cylindrical tanks are technological facilities in which two bodies far different by their characteristics (the relatively stiff steel shell and the deformable liquid) interact with each other. A study of their combined behaviour is possible through creation of experimental models in a research laboratory and/or through the use of specialized software for numerical analysis. Such software, widely used all over the world and applicable in all the fields of research activity, is ANSYS. The eponymous company was founded in 1970 in Pennsylvania, USA. In its classical appearance, ANSYS Parametric Design Language (APDL), the program gives the possibility for simulation of all kinds of complicated tasks and precise setting of the parameters in them. To be able to work in this environment, however, the user must know in detail the theoretical background behind the modelled processes and often has to use text commands. Unfortunately, this way of information input is obsolete and discourages new users, who prefer intuitive, easy to use graphical interface and less thinking. Aiming to respond to the new reality ANSYS creates the graphical interface Workbench that is far easier to use by young users. It rarely requires the usage of text commands.

Characteristic for Workbench is the differentiation of modules, each specialised in solving tasks from a particular science field. Simulating the interaction between solids and fluids is an interdisciplinary task that can be solved using different modules of the program. In each of them, the user can easily model the storage tank filled with liquid and obtain some results for their behaviour. And the question here is - how reliable would those results be? To answer it, a special case study was carried out. A steel tank for water storage was modelled and analysed in different modules of ANSYS Workbench v.19 using static /hydrostatic/, modal and dynamic /spectral/ analysis. Initially, only the solution options provided by the respective modules were used. However, due to the complexity of the problem under consideration, these basic capabilities proved to be insufficient to produce reliable results. Therefore, a second phase of the study was conducted in which some of the capabilities of the modules were expanded through user intervention and input of text commands. As a third step, the obtained results were compared with a solution in the classic version of ANSYS Mechanical APDL v.19 and with an analytical solution.

2. Modelling

For the purpose of the current study, a steel tank for water storage described in [10] is modelled in ANSYS Workbench. The facility has a diameter D = 5 m, height of the cylindrical shell H = 8 m, and a constant course and bottom thickness t = 6 mm. It is filled with the water to a level H_t = 6,5 m. The tank is investigated in two modifications – without upper stiffening ring (top angle) and with one - placed on the top of the shell, see Fig. 1.

Material properties are defined as follows:

a) Steel [1]

- *Young*'s modulus – E = 210 000 MPa;
- *Poisson'*s ratio – ν = 0,3;
- density – ρ_s = 7 850 kg/m^3.

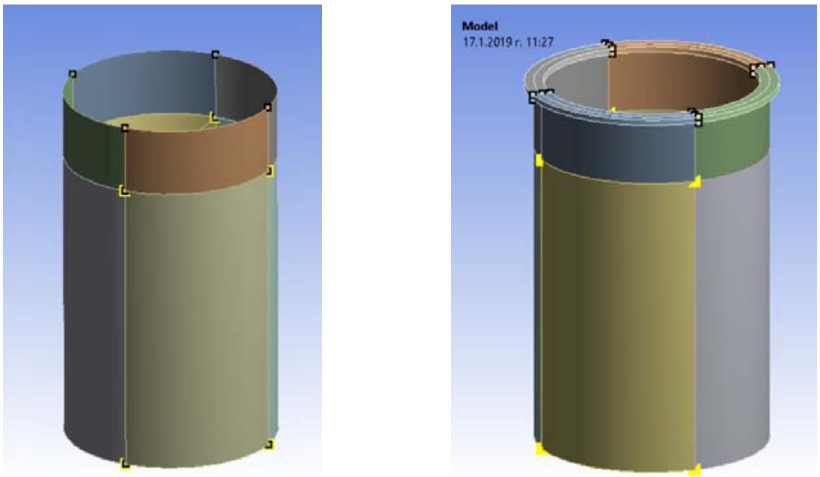
b)Water – in all Workbench modules, excluding Modal Acoustics and Static Acoustics:

- bulk modulus – B = 2,1.10^9 Pa [10];
- *Poisson'*s ratio – ν = 0,4999995. The adopted value of the *Poisson's ratio* in this study is different from the one indicated in [4] ν = 0,49999999, because using the latter, the solution in LS-Dyna could not converge;

- density – ρ_w = 1 000 kg/m^3;

Water – in modules Modal Acoustics and Static Acoustics in the second stage of the research and in the classical version ANSYS MAPDL:

- speed of sound in water – v_w = 1 482,1 m/s;
- density – ρ_w = 1 000 kg/m^3.



a) Tank with free upper end *b) Tank with upper stiffening ring*

Fig. 1. Variants of the geometry of the investigated tank

2.1. First stage of the research in ANSYS Workbench

The specific features in setting up the numerical models are:

a) The steel body of the tank is modelled using shell elements;

b) The water body is created using:

- solid elements for modules Static Structural, Explicit Dynamics, LS-Dyna, Modal and Response Spectrum;

- fluid elements for modules Modal Acoustics and Static Acoustics;

c) Workbench creates an automatic connection between the two materials, which can have different characteristics. In the current study is accepted that the connection type is “No separation” – the two materials can slide freely against each other but without separation normal to the interface between them. Mutual penetration is limited to 0,1 mm;

d) The liquid volume is divided into four parts, see Fig. 1. The idea is to improve the mesh of finite elements;

e) The tank part consists of two separate bodies – shell and bottom:

- the bottom is divided into four sections. Their form, dimensions and position correspond to the division of the liquid;

- the shell is divided into eight sections, see Fig. 1, so that against every edge of the water volume body exists edge of the shell body;

f) The maximum dimension of the finite elements is limited to 250 mm;

g) Low order finite elements are used. They have nodes only in the corners i.e. the middle nodes are deactivated. The reason for this decision is to reduce the size of the problem and the duration of the solution;

h) The tank is fixed at the base;

i) Gravity acceleration g = 9,807 m/s^2 is applied where possible;

j) Damping effects are not included in the analysis;

k) The modules used in ANSYS Workbench are Static Structural, Modal, Response Spectrum, Explicit Dynamics, LS-Dyna, Modal Acoustics and Static Acoustics, see Fig. 2.

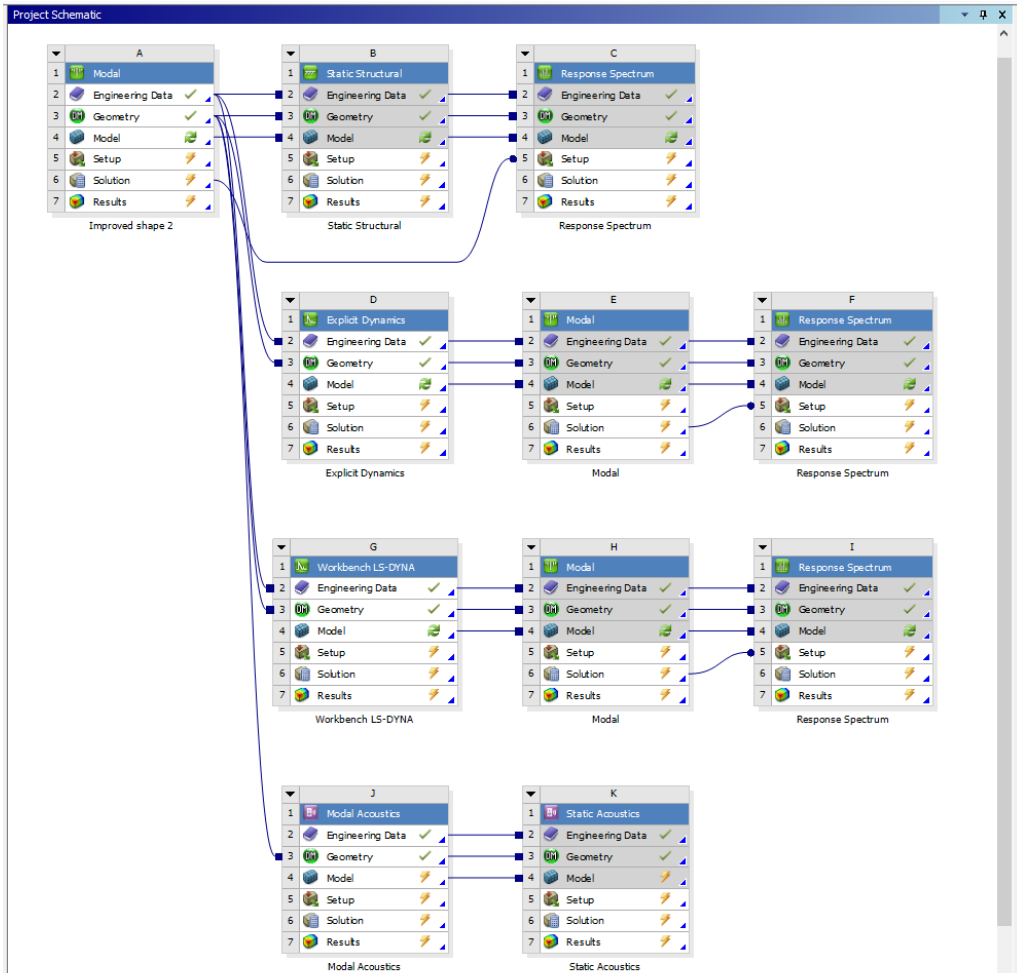


Fig. 2. ANSYS Workbench modules - Sequence of usage

The liquid is presented as Eulerian reference frame in modules Explicit Dynamics and LS-Dyna. In this way, the deformability of the fluid's mesh that interacts with the tank is ensured. Such an approach for modelling of a water-filled reservoir under rapidly changing seismic excitation is used in [6] and [7].

2.2. Second stage of the research in ANSYS Workbench

Only the differences to the prerequisites listed in section 2.1, will be described here:

a) Only Static Structural, Modal and Response Spectrum Workbench modules are used at this stage but with modifications;

b) The water body is created by overwriting the default solid elements (SOLID185) and their replacement by fluid elements. In this study, FLUID30 element is used, which is a 3D acoustic fluid with eight nodes situated only at the corners [3]. A low-order element is intentionally chosen to achieve a higher speed of solution. The purpose of the change of the solid element with a fluid one is to better capture the real mechanics. Unlike solid elements, fluid ones have an additional degree of freedom – pressure. In the case of fluid-solid interaction, the matrices describing the system's motion are non-symmetric.

The equation of motion in solid mechanics is described, as follows [3]:

$$[M].\{\ddot{u}(t)\}+[C].\{\dot{u}(t)\}+\{F^i(t)\}=\{F^a(t)\}, \quad (1)$$

where $[M]$ is structural mass matrix;

$[C]$ – structural damping matrix;

$\{\ddot{u}(t)\}$ – nodal acceleration vector;

$\{\dot{u}(t)\}$ – nodal velocity vector;

$\{F^i(t)\}$ – load vector of internal forces;

$\{F^a(t)\}$ – load vector of applied forces.

In the presence of fluid and structure in one common system, the two elements interact and the equations of motion for the individual subsystems change. Taking into account this interaction and the sloshing of the free surface of the liquid in the tank, the equation of motion transforms into the following [3]:

$$\begin{bmatrix} [M_S] & 0 \\ \overline{\rho_0}.[R]^T & [M_F]+[S_F] \end{bmatrix} . \begin{Bmatrix} \{\ddot{u}_e\} \\ \{\ddot{p}_e\} \end{Bmatrix} + \begin{bmatrix} [C_S] & 0 \\ 0 & [C_F] \end{bmatrix} . \begin{Bmatrix} \{\dot{u}_e\} \\ \{\dot{p}_e\} \end{Bmatrix} + \begin{bmatrix} [K_S] & -[R] \\ 0 & [K_F] \end{bmatrix} . \begin{Bmatrix} \{u_e\} \\ \{p_e\} \end{Bmatrix} = \begin{Bmatrix} f_s \\ f_F \end{Bmatrix}, \quad (2)$$

where S – is an index that refers to the structure;

F – is an index that refers to the fluid;

$[K]$ – stiffness matrix;

$[R]$ – boundary matrix;

$[S_F]$ – acoustic sloshing mass matrix;

$\{u_e\},\{\dot{u}_e\},\{\ddot{u}_e\}$ – nodal displacement, velocity and acceleration vectors for the solid;

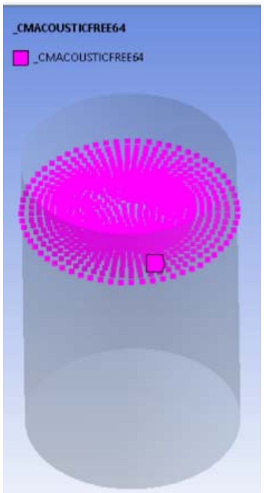
$\{p_e\},\{\dot{p}_e\},\{\ddot{p}_e\}$ – acoustic nodal pressure vector and its derivatives;

$\overline{\rho_0}$ – acoustic fluid mass density constant;

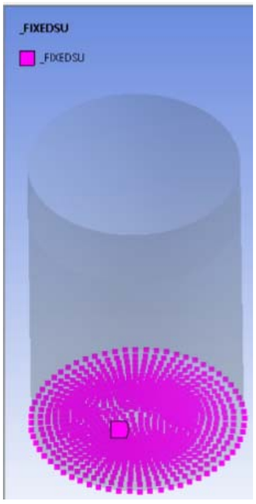
$\{f\}$ – external loading.

c) The interaction between the storage tank and liquid inside is defined by assigning a Fluid-Structure Interaction (FSI) Flag to the fluid elements, which are in contact with the tank, see Fig. 3;

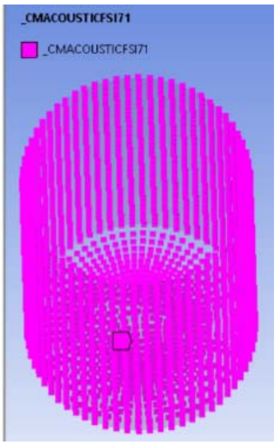
d) According to [3], when the liquid and the tank are defined as different parts in ANSYS WB and in the presence of FSI, the only possible option for the contact between them is type "bonded" defined with Multi-Point constraint (MPC) formulation. This type of connection adds additional constraint equations which bond the contact surfaces together.



a) Component for free surface definition



b) Component for definition of boundary conditions in the base



c) Component for fluid-structure interaction definition (FSI flags)

```
/Prep7
/com,*********** Elements for Body 2 "Water" ***********
et,matid,30
r,matid,2.e-005,101325.

/com,*********** Fixed Supports ***********
cmsel,s,_FIXEDSU
d,all,all
ddele,all,pres ! release the pressure DOF for acoustics analysis
nsel,all

/com,*********** Create Acceleration ***********
ACEL,,,9.8066

/com,*********** Create Fluid Solid Interface ***********
cmsel,s,_CMACOUSTICFSI171
esel,r,ename,,30
sf,all,fsi
allsel,all

/com,*********** Create Free Surface ***********
cmsel,s,_CMACOUSTICFREE64
sf,_CMACOUSTICFREE64,free
allsel,all

/SOLU
antype,2                    ! modal analysis
modopt,unsym,500,1.e-002,,
```

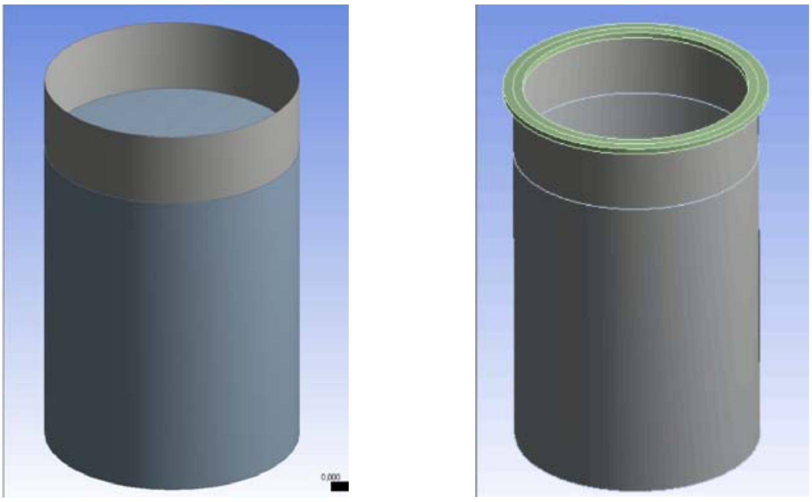
d) User's input commands for extending the capabilities of Workbench modules - definition of the element type "fluid", definition of boundary conditions in the base, free surface and fluid-structure interaction definition

Fig. 3. Definition of boundary conditions in ANSYS Workbench

In contact problems, the interacting elements are divided into two groups - contact elements and target elements. For asymmetric contact formulation penetration of contact elements through the target ones is prohibited. The proper way to model fluid-structure interaction is to define the fluid as the contact element and the solid – as the target element;

e) Manual geometry division is not necessary, see Fig. 4;

f) Gravity acceleration g = 9,8067 m/s^2 is applied, which is a necessary precondition for the presence of free surface of the liquid;



a) Tank with a free upper end *b) Tank with upper stiffening ring*

Fig. 4. Geometry of the tank in Stage 2 of the case study in ANSYS Workbench

g) "Free surface flag" is assigned to the free surface of the liquid to account for the sloshing effects. The latter is done by definition of Named Selections (components) and input of text commands in Workbench, see Fig. 3;

h) All degrees of freedom of the bottom of the tank are fixed, except for the pressure DOF of the fluid. This is achieved by input of text commands, see Fig. 3;

i) The non-symmetric matrices of the equation of motion require the usage of Unsymmetric Solver for the calculations.

2.3. Research in ANSYS Mechanical APDL

The definition of the elements and their interaction in ANSYS MAPDL is very similar to the above described in point 2.2. As the main difference, it can be pointed the way of building of the part "tank shell", which is in contact with the liquid. Here the option to merge the two bodies ("Form New Part") available in Workbench is missing, so it is necessary to use another approach for obtaining a compatible mesh in the zone of interaction between the tank and its containment. It can be done in the following way - instead of drawing the physical geometry of the shell it is generated automatically after the definition of the mesh of liquid – exactly matching the finite elements of the fluid. In this way, the discretization of the two elements is completely compatible. Elements of the shell, type SHELL181, are generated using the command ESURF [3].

2.4. Parameters of the research

With every analysis of the tank in a specific module of Workbench or in ANSYS APDL we are aiming to find:

a) Maximum radial stress σ_θ in the shell, as a result of the stored liquid in the tank, MPa;

b) Maximum radial deflection Δ_θ as a result of the stored liquid, mm;

c) Minimum f_{min} and maximum f_{max} value of natural frequencies, obtained in the following conditions:

- the model is solved for the 500 natural modes of vibration;
- the minimum sought frequency is 0,1 Hz.

The number of modes at the first stage of the research is limited to 500, because even if a larger number is defined, the program cannot determine their values and the calculation is stopped. In the second stage of modelling in Workbench 500 modes are sufficient for activation of 90% of the mass and therefore the obtained results should be considered reliable. In ANSYS MAPDL the necessary number of modes analysed exceeds 500 and for that reason 1000 modes are analysed;

d) Overturning moment M_y and shearing force H_x in the base obtained by response spectrum analysis, in the conditions of:

- peak ground horizontal acceleration $a_g = 2{,}55$ m/s^2;
- the soil beneath the tank is type „C", according to EN 1998-1 [8].

3. Results

3.1. Results obtained in the first stage of research in ANSYS Workbench

In the first stage of the research, the contact between the two materials (water and steel) is defined in two different ways. In the first group of models, the shell and the liquid interact as follows:

- the connection between the liquid and the elements of the bottom is defined as "bonded contacts" – the elements of the two bodies are bonded, i.e. sliding and separation are not possible;
- the connection between the liquid and the tank's shell elements is defined as "no separation contacts" – the contacting surfaces can slide but cannot separate.

The obtained results are summarized in Table 1.

Table 1. Results for contacts type "bonded" and "no separation"

Type of the analysis (module):		**Static Structural**		**Modal**		**Response Spectrum**	
Initial module	**Top stiffening ring**	**Stress σ_0, MPa**	**Displacement Δ_0, mm**	**Frequency f, Hz**		**Base Reactions**	
				f_{min}	f_{max}	M_y, kN.m	H_x, kN
Static structural	without	30,072*	0,35	0,176	1,118	46,41	8,962
	with top angle	30,078*	0,36	0,176	1,122	46,47	8,977
Explicit Dynamics	without	47,24	3,08	0,16	1,095	39,65	8,523
	with top angle	34,12	2,7	0,16	1,091	39,66	8,523
LS-Dyna	without	35,597	0,36	0,135	1,184	40,65	9,194
	with top angle	12,52	0,5	0,135	1,101	40,48	9,023

Type of the analysis (module):		Static Structural		Modal		Response Spectrum	
Initial module	Top stiffening ring	Stress σ_θ, MPa	Displacement Δ_θ, mm	Frequency f, Hz		Base Reactions	
				f_{min}	f_{max}	M_y, kN.m	H_x, kN
Modal Acoustic	without	0,0395	0	0,428	64,40		
	with top angle	0,441	0	0,428	36,79		

***Note**: The values presented in the table are calculated using the normal stresses for the mid-plane of the shell with activation of the option averaging at the nodes. When "elemental mean" or "membrane stress" option is activated, the values for σ_θ become as follows: 27,916 MPa for a tank without upper stiffening ring and 26,407 MPa for a tank with upper stiffening ring, see fig. 1*b*.

In the second group of models, the steel shell and the liquid interact only by "bonded contacts", i.e. sliding and separation are not possible.

The results from the analysis conducted only with "bonded contacts" are summarized in Table 2.

Table 2. Result when the defined contacts are "bonded" on all surfaces

Type of the analysis (module):		Static Structural		Modal		Response Spectrum	
Initial module	Top stiffening ring	Stresses σ_θ, MPa	Displacement Δ_θ, mm	Frequency f, Hz		Base Reactions	
				f_{min}	f_{max}	M_y,kN.m	H_x, kN
Static structural	without	29,85*	0,287	0,337	1,23	51,78	10,81
	with top angle	29,85*	0,29	0,338	1,24	51,1	10,19
Explicit Dynamics	without	51,76	2	0,337	1,23	54,11	11,07
	with top angle	53,77	3,56	0,338	1,242	51,22	10,22
LS-Dyna	without	"bonded contact" is not supported					
	with top angle	"bonded contact" is not supported					
Modal Acoustic	without	0,0395	0	0,428	64,40		
	with top angle	0,441	0	0,428	36,79		

***Note**: The values presented in the table are calculated using the normal stresses for the middle section of the shell with activation of the option averaging at the nodes. When "elemental mean" or "membrane stress" option is activated, the values for σ_θ become as follows: 25,825 MPa for a tank without upper stiffening ring and 26,196 MPa for a tank with upper stiffening ring, see fig. 1*b*.

3.2. Results obtained in the second stage of research in ANSYS Workbench

The results of the analysis carried out in ANSYS Workbench, where the capabilities of the respective modules are extended by user's input commands, are summarized in Table 3.

Table 3. Results obtained in the second stage of the research in ANSYS Workbench

Type of analysis (module):	**Static Structural**		**Modal**		**Response Spectrum**	
Stiffening ring at the top of the tank shell	**Stresses σ_θ, MPa**	**Displacement Δ_θ, mm**	**Frequency *f*, Hz**		**Base Reactions**	
			f_{min}	f_{max}	M_y, kN.m	H_x, kN
without	30,206*	0,362	0,4283	63,97	1810,90	503,97
with top angle	30,213*	0,362	0,4283	71,79	1898,50	517,11

***Note**: The values presented in the table are calculated using the normal stresses for the mid-plane of the shell with activation of the option averaging at the nodes. When "elemental mean" or "membrane stress" option is activated, the values for σ_θ are relatively 26,481 MPa for the tanks without upper stiffening ring, see Fig. 3*a*, and 26,483 MPa for tanks with upper stiffening ring, see Fig. 3*b*.

3.3. Results calculated in ANSYS Mechanical APDL

Table 4. Results calculated in the classical version ANSYS Mechanical APDL

Type of the analysis :	**Static**		**Modal**		**Response Spectrum**	
Stiffening ring at the top of the tank shell	**Stresses σ_θ, MPa**	**Displacement Δ_θ, mm**	**Frequency *f*, Hz**		**Base Reactions**	
			f_{min}	f_{max}	M_y, kN.m	H_x, kN
without	30,896*	0,362	0,4282	3,685	1827,22	493,36
with top angle	30,904*	0,362	0,4282	3,812	1884,22	497,50

***Note**: The values presented in the table are calculated using the normal stresses for the middle section of the shell with activation of the option averaging at the nodes (option: averaged).

3.4. Results calculated by analytical calculations

3.4.1. Static analysis

The maximum circumferential (hoop) stress in the cylindrical shell can be calculated by the well-known equation of *Laplace* (1740 ÷ 1827):

$$\frac{\sigma_x}{R_x} + \frac{\sigma_\theta}{R_\theta} = \frac{p}{t}, \tag{3}$$

where σ_x is meridional (axial) stress;

σ_θ – radial (circumferential) stress;

R_x – radius of curvature of the meridional section;

R_θ – radius of curvature of the circumferential section;

p – value of the normal pressure acting on the shell;
t – thickness of the shell.

In this particular case, where $R_x = \infty$, normal stresses in circumferential (radial) direction σ_θ can be expressed by the equation:

$$\sigma_\theta = \frac{p}{t}.R_\theta = \frac{6{,}5.9{,}8066}{0{,}006}.2{,}5 = 26559{,}5 \text{ kPa} \approx 26{,}56 \text{ MPa} \cdot$$

Comparing the result from Equation (3) and the values indicated in Tables 1 ÷ 4, it is clear that the results obtained in the module Static structural (with option "elemental mean" or "membrane stress" activated) are closest to the theoretical ones, and thus the most accurate.

3.4.2. Modal analysis

In order to validate the FEA results, the first two natural frequencies of the liquid are calculated by the equation [2]:

$$f_i = \frac{1}{2\pi}\sqrt{\alpha_i \frac{g}{R} \operatorname{th}\left(\alpha_i \frac{h}{R}\right)}, \tag{4}$$

where g = 9,8067 m/s^2 is the gravity acceleration;
R = 2,5 m – radius of the cylindrical body of the tank;
$h = H_t$ = 6,5 m – maximum level of water in the tank;
α_i – coefficient that has the following values:
= 0,586 π – for the first mode of vibration;
= 1,697 π – for the second mode;
= 2,717 π – for the third mode.

The first sloshing frequencies of the fluid are as follows:
- for the first sloshing mode – f_1 = 0,4277 Hz;
- for the second sloshing mode – f_2 = 0,7278 Hz.

For an additional benchmark, the first natural frequency of the analysed tank subjected to seismic action is calculated by the prescriptions of the other well-known scientists in this field and also according to the standards API 650 [5] and EN 1998-4 [9]. The results are presented in the Table. 5.

Table 5. Values for the first natural frequency f_1 of splashing

Source	***Уманский*** [2]	***Haroun*** [11]	***Housner*** [12]	**API 650** [5]	**EN 1998-4** [9]
f_1, [Hz]	0,4277	0,4276	0,4272	0,4298	0,4273

The values for the natural frequencies shown in Table 5 are very similar to the ones calculated using ANSYS MAPDL and modules Modal Acoustic and Modal in Workbench stage 2, where the liquid is modelled with element FLUID30. The results obtained in the other modules, where the water is modelled as solid element, are considerably different from the theoretically defined.

The theoretical shape of the free surface of the liquid for the first two natural sloshing modes is presented in Fig 5.

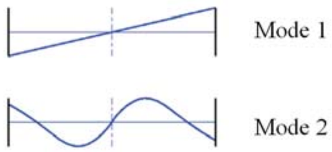
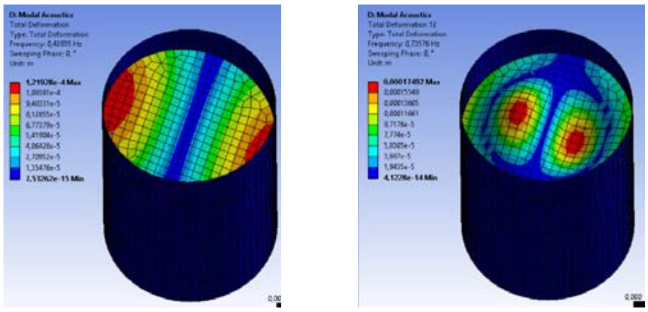
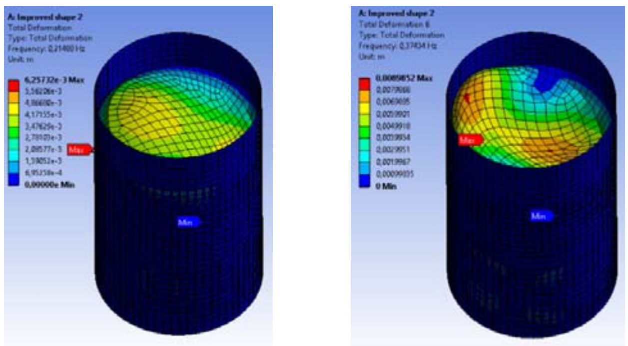


Fig. 5 Theoretical shape of the free surface of the liquid for the first two sloshing modes

The shapes of the free surface for the first two sloshing modes presented on Fig.5 have a very good coincidence with the models in which the liquid is simulated by element type "fluid". When the liquid body is type "solid" and the connection between the two media is "bonded", the shape of the free surface is different and not realistic, see Fig.6.



a) Sloshing modes 1 and 2, element type "fluid"



b) Sloshing modes 1 and 2, element type "solid", "bonded" connection

Fig. 6 Shape of the free surface of the liquid for the first two sloshing modes

3.4.3. Response spectrum analysis

A response spectrum analysis is performed for a seismic event, determined by horizontal ground acceleration a_g = 2,55 m/s^2. The soil is type „C", according to the classification of EN 1998-1-1 [5]. The shear force H_x and the overturning moment M_y in the base calculated using response spectrum analysis and the prescriptions of EN 1998-4 [9], are shown in Fig.7.

The data, presented in the tables 1÷4 indicate that reliable results for the generalized base reactions due to seismic action can be calculated accurately only by using user's input commands in ANSYS Workbench and in ANSYS Mechanical APDL. The values obtained by the latter differ from the analytical calculations done according to EN 1998-4. The base overturning moment is 4% - 5% lower and the base shear force differs between 19% - 21%

(compared to results for a tank with upper stiffening ring, because this formulation is closer to the one used in the analytical solution). Such differences are expected since the procedure, proposed in the regulatory documents is for approximate calculations. It is based on the assumption that the tank wall is rigid and non-deformable, which explains why the forces, calculated using numerical models where its deformability is taken into account, are lower.

Caclulation of base reactions obtained by response spectrum analysis acc. to EN 1998-4

s =	0.006	m	thickness of tank shell	собствени периоди на:		
R =	2.5	m	radius of tank shell	T_{imp} =	**0.061**	s impulsive component
H =	8	m	shell height	T_{conv} =	**2.340**	s convective component
H_L =	6.5	m	liquid height measured from the bottom of the tank			
H/R =	2.6000					
ρ =	1000	kg/m^3	liquid density			
m =	127.6	t	liquid mass	h_i/H =	0.451	
m_i/m =	0.810			h_i =	2.9	m for CoM of the impulsive mass
m_i =	103.4	t	impulsive mass	h_c/H =	0.795	
m_c =	24.2	t	convective mass	h_c =	5.2	m for CoM of the convective mass
m_w =	5.9	t	tank shell (wall) mass	h_w =	4.0	m
m_r =	0.0	t	tank roof mass	h_r =	0.0	m
$S_e(T_{imp})$ =	**5.197**	m/s^2	inlpulsive spectral acceleration, obtained from an elastic response spectrum			
$S_e(T_{conv})$ =	**2.272**	m/s^2	convective spectral acceleration, obtained from an elastic response spectrum			
Q_{rw} =	**623.278**	kN	$Q=(m_i+m_w+m_r)S_e(T_{imp})=m_cS_c(T_{con})$	A.37, EN 1998-4		
M_{rw} =	**1983.107**	kNm	$M=(m_ih_i+m_wh_w+m_rh_r)S_e(T_{imp})+m_ch_cS_e(T_{con})$	A.38, EN 1998-4		

Fig. 7. Calculation of the tank's base reactions obtained by response spectrum analysis according to the European standard EN 1998-4 [9]

4. Conclusions

The conclusions that could be obtained from the current case study can be summarised as follows:

a) For reliable calculation of the natural frequencies of a tank filled with liquid in ANSYS Workbench, module Modal Acoustics has to be used. Unfortunately, subsequent response spectrum analysis is not possible, because Response Spectrum module does not recognize the results obtained by Modal Acoustics;

b) Reliable calculation of natural frequencies of a tank filled with liquid in module Modal in Workbench is only possible when the model properties are modified using user's input text commands. In this case subsequent response spectrum analysis of the system can be performed;

c) Using modules Explicit Dynamics, LS-Dyna, Modal Acoustics and Static Acoustics normal stresses in the tank shell caused by hydrostatic pressure cannot be calculated correctly. The same applies to the natural oscillation frequencies;

d) The first natural frequency of the liquid is not influenced by the presence of stiffening ring on the top of the tank's shell;

e) In modules Modal Acoustics and Modal modified by input commands in Workbench, the presence of top stiffening ring changes considerably the frequencies of the high oscillation modes;

f) The calculated overturning moment M_y and shear force H_x in the base of the tank, using modified Workbench and ANSYS MAPDL, are similar to the theoretically obtained. This would allow their usage in practice – in research and design projects.

Acknowledgements

The current scientific research is financially supported according to contract Д-107/2018 by the Centre for Scientific Research and Design in the University of Architecture Civil Engineering and Geodesy, Sofia, Bulgaria. The authors express their gratitude for the provided funding.

The authors would also like to express their acknowledgement to the Centre of Mathematical Modelling and Computer Simulation and to the University of Chemical Technology and Metallurgy for the provided possibility to use the ANSYS v19 software.

LITERATURE

1. БДС EN 1993-1-1:2005. Еврокод 3: Проектиране на стоманени конструкции. Част 1-1: Общи правила и правила за сгради. БИС, юни 2007.

2. Справочник проектировщика промишленных, жилых и общественных зданий и сооружений. Под редакцией проф. д-р А. А. Уманского, Издание второе. Москва, 1973.

3. ANSYS® v.19 Documentation, ANSYS, Inc.

4. ANSYS. Crush of Filled Soda Can. Introduction to ANSYS Autodyn part II. ANSYS Inc., November 2012.

5. API Standard 650: Welded Tanks for Oil Storage, Twelfth Edition, 2013.

6. *Celik, A., Akgul, T.* Stress Analysis of Cylindrical Steel Storage Liquid Tanks During the Instantaneous Loads, Proceedings of the 3rd World Congress on Civil, Structural, and Environmental Engineering. Budapest, Hungary, April 8 – 10, 2018.

7. *Celik, A., Akgul, T., Apay, A.* Plastic deformation of cylindrical steel tank both under the kocaeli and El-Centro Earthquake. International Journal of Advance Engineering and Research Development, Vol. 5, Issue 09, September 2018.

8. EN 1998-1:2004. Eurocode 8: Design of structures for earthquake resistance. Part 1: General rules, seismic actions and rules for buildings. European Committee for Standardization, Brussels, 2004.

9. EN 1998-4:2006. Eurocode 8: Design of structures for earthquake resistance. Part 4: Silos, tanks and pipelines. European Committee for Standardization, Brussels, 2006.

10. *Goudarzi, M., Sabbagh-Yazdi, S.* Numerical Investigation on Accuracy of Mass Spring Models for Cylindrical Tanks under Seismic Excitation. International Journal of Civil Engineering, Vol. 7, No. 3, September 2009.

11. *Haroun, M. A.* Vibration studies and test of liquid storage tanks. Earthquake Engineering and Structural Dynamics, vol. 11, pp. 179–206, 1983.

12. *Housner, G. W.* The dynamic behavior of water tanks. Bulletin of Seismological Society of America, 53, 196