Application of submodeling in strength analysis of horizontal tank for storage of petroleum products

Pejo Konjatić[1], Sara Radojičić[2] Dorian Turk[3], Matej Pudić[4]
University of Slavonski Brod, Croatia[1]
pkonjatic@unisb.hr
University of Slavonski Brod, Croatia[2]
dturk@unisb.hr
University of Slavonski Brod, Croatia[3]
sradojicic@unisb.hr
TD Komunalac d.o.o., Croatia[4]
matejpudic.sfsb@gmail.com

Abstract: *This paper shows the approach of submodeling in finite element analysis in process of designing the horizontal tank for storing petroleum products. Those kinds of tanks are made at exact standards or slightly altered by customers' demands. Before the numerical analysis of the tank, the tank standards are described as well as numerical analysis. In the paper, the methods which will be used in obtaining solutions by finite element method were described. 3D finite elements have been used for numerical analysis. After getting the results of full tank model meshed with 3D tetrahedral final elements, the technique of submodeling has been used to achieve more accurate results in critical locations of the tank. In the final aspect, result analysis was conducted, where the results were confirmed so that the tank complies with the standard and meets all the criteria given to the designer.*

Keywords*: FINITE ELEMENT METHOD, SUBMODELING, HORIZONTAL FUEL TANK, LIFTING LUG*

1. Introduction

Fuel tanks are used for the storage of various types of liquid fuels, fuel oils and similar liquids with a maximum density of up to 1,9 kg/l, which assigns fuel tanks to Class A according to the relevant standard which is used in this analysis EN 12285-1:2018. [1]. They are manufactured as single-wall and double-wall tanks. Single-wall tanks are used when the tank is located in a location that protects the environment from possible fuel leaks. With double-wall tanks, there are no restrictions on the location of the tanks. Double-walled tanks allow continuous leakage control (negative or positive pressure) and provide long-term safety in use [2].

Fuel tanks are considered pressure vessels subjected to tensile forces within the walls of the container. The normal stress in the walls of the container is proportional to the pressure and radius of the vessel and inversely proportional to the thickness of the walls. [3]. Pressure vessels and tanks are different in both design and construction: tanks, unlike pressure vessels, are limited to atmospheric pressure; and pressure vessels often have internals pressure while most tanks do not have that [4].

It is imperative for an engineer to design and analyze the pressure vessel that will provide safety, durability and serviceability to the end user. Accomplishing this task will require a very good knowledge of design parameters, the most important being, geometry of pressure vessel that must be analyzed to comply design standards [5]. For this reason, many studies have been carried out to explain the design and finite element analysis of horizontal tanks on saddle supports [6-9].

The standards also include lifting lugs for transporting and lifting tanks. Each tank must be equipped with lugs for lifting the tank. The number of these lifting lugs shall be at least one for a tank up to 20 m^3 and not less than two above 20 m^3. They shall be placed so that the tank can be lifted in a horizontal position. The lugs welded to the tank shall be of such size and number that the empty tank can be lifted. The minimum diameter of the hole on the tank lifting lug must be 60 mm [1].

This paper presents the results of the numerical analysis of the strength of the horizontal tank used for storage of petroleum products. The ANSYS 18.0 [10] software package was used to obtain the results of the stresses and strain in the tank. Before using the ANSYS software package, it was necessary to create a 3D model of the tank to be used in the software. The tank 3D model was created in SOLIDWORKS 2021 [11].

2. Problem description

This paper aims to verify the structure of the horizontal tank for storing oil derivates and determine whether design changes are needed. The verification will be performed by determining the equivalent von Mises stresses and the maximum total displacement of the fuel tank. The results are acquired by comparing the stresses obtained by FEM analysis from global model and for each submodel.

Fuel tanks are normally mounted on two stands. The stand can be welded to the tank or can be movable in relation to the tank. The baseplate has holes for fixing the plinth to the base plate and the ribs have holes for the floor connection. The plinths are primed and given a final coat of paint, i.e. bitumen board or Recitol if the plinth is used for the installation of tanks in the excavation pit [12]. An overview of the tank model developed in the SOLIDWORKS 2021 software can be found in Figure 1.

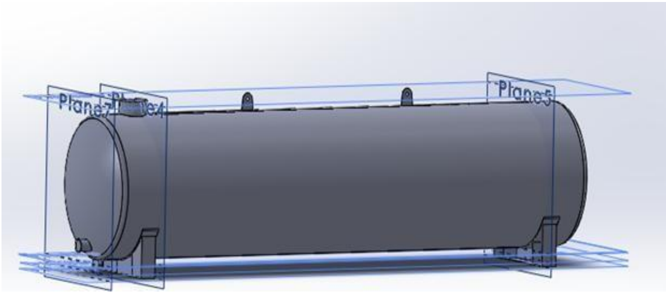

Fig. 1 *3D model of horizontal tank for storage of petroleum products*

The tank is used for the storage of petroleum products (liquid fuels up to 1,1 kg/l) with wall thickness of 7 mm and is supported by two saddles, which are also made according to the standards for the tank manufacturing. Investigated tank is manufactured according to standard mentioned earlier in the paper. Its dimensions and other data can be seen in Figure 2. As this is a 50 m^3 tank, a lot of computational power is needed to create a high-quality mesh for the meshing of a geometric model from appropriately selected finite elements.

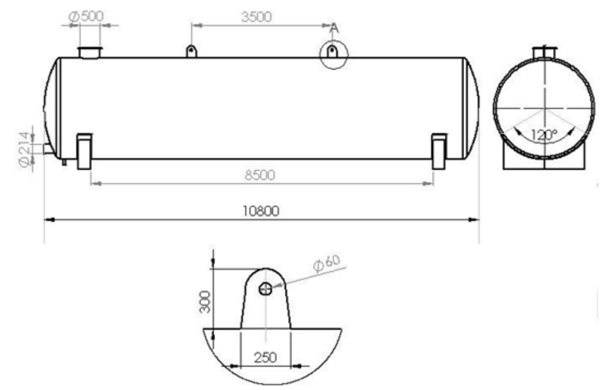

***Fig. 2** Dimensions of the 50 m^3 horizontal tank*

3. Finite element analysis of the global model

In order to obtain the required results, material properties of steel S235JR are assigned to the model. The Young's modulus of elasticity E = 210 GPa, yield strength $R_{p0,2}$ = 235 MPa and the value of Poisson's ratio v = 0,3 [13, 14].

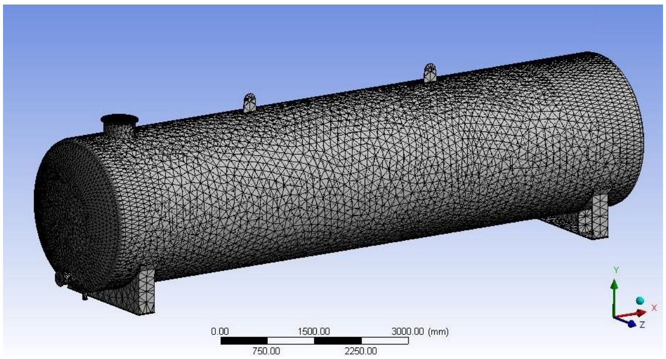

***Fig. 3** Display of a 3D finite elements mesh*

Global model with a coarse mesh is made with tetrahedral finite element mesh size of 100 mm and boundary conditions are applied (Figure 3 and Figure 4). Total number of finite elements in global model is 54271 with 106702 nodes. As a boundary condition, fixed support is used as well as displacement support with restrained y-direction and a test pressure of 0,2 MPa is set.

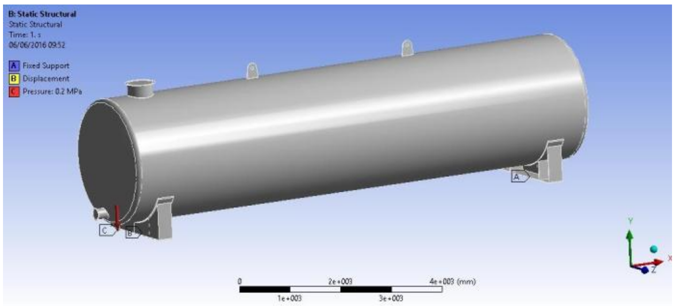

***Fig. 4** Boundary conditions of the 3D model*

The representation of the equivalent von Mises stresses in the 3D model is shown in Figure 5. With the 3D model, the highest stresses are at the junction between the cylindrical and the bottom part of the tank. The magnitude of the maximum equivalent stresses is 191,76 MPa.

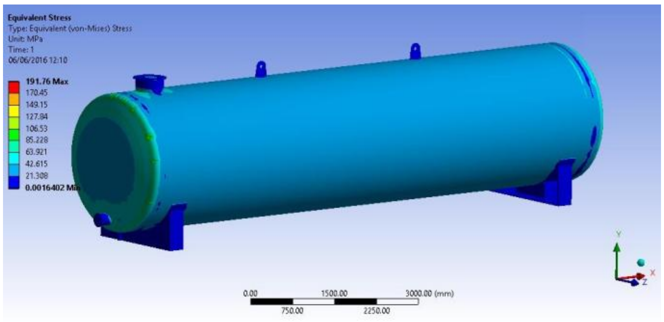

***Fig. 5** Distribution of the equivalent von Mises stress*

It can be seen that for largest part of the tank, the stress intensity does not exceed 50 MPa. The distribution of the total displacements can be seen in Figure 6. The top opening is analyzed in more detail as submodel in order to obtain more precise stress distribution. The displacements in the largest part of the tank are very small and the maximum displacement is 2,2057 mm at the opening of the tank.

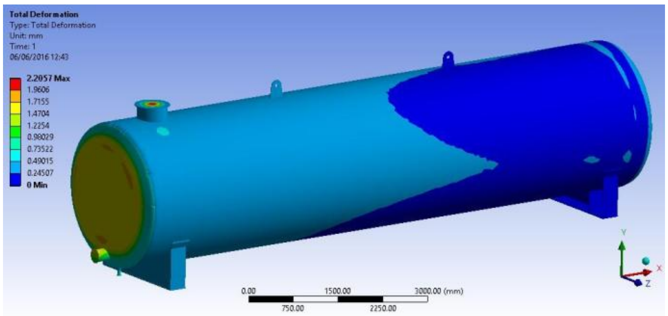

***Fig. 6** Global model distribution of the displacement*

4. Finite element analysis of submodels

All material and test pressure data given previously were used to obtain results for the tank opening submodel. Displacement results obtained from the global model with coarse finite element mesh are imported into the submodel of the opening, and mesh refinement of submodel is done to obtain more accurate results.

The finite element mesh convergence of the 3D submodel is performed to determine adequate mesh size. Table 1 shows the data of the number of elements, nodes and equivalent von Mises stress in the submodel of the top opening.

***Table 1** Number of elements, nodes and equivalent von Mises stress in the top opening submodel*

Element size, mm	25	15	13	11
Submodel node numbers	51912	127338	129423	177553
Submodel element numbers	25780	64423	65320	89918
Maximum equivalent von Mises stress, MPa	194,34	184,74	183,17	185,31

The convergence of the von Mises stress solutions is shown in Diagram 1. It can be seen that the deviations are within 5% between the 15, 13 and the 11 mm finite elements, which means that the finite element mesh size of 11 mm is sufficient enough for acquiring accurate results.

Diagram 1 *The convergence of the von Mises stress solutions*

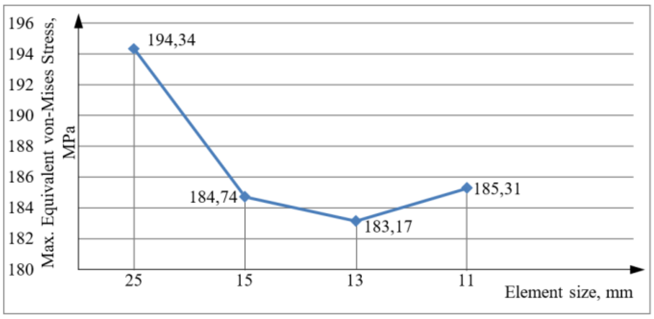

Figure 7 shows a finite element submodel obtained using 11 mm 3D finite elements thus the mesh consists of 177553 nodes and 89912 elements. The mesh in the submodel is significantly denser, showing more accurate results around the tank opening. The submodel of the top opening was created due to the increased displacement and stresses around this area of the tank.

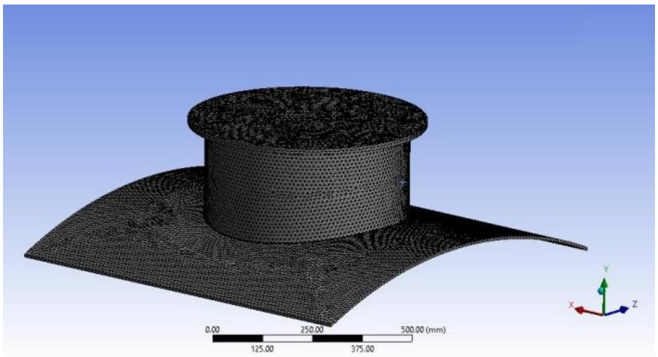

Fig. 7 *Submodel with fine mesh of the top opening*

An overview of the total displacements at the tank opening can be seen in Figure 8. The maximum displacement of 2,8447 mm has appeared in the area of the top of the opening.

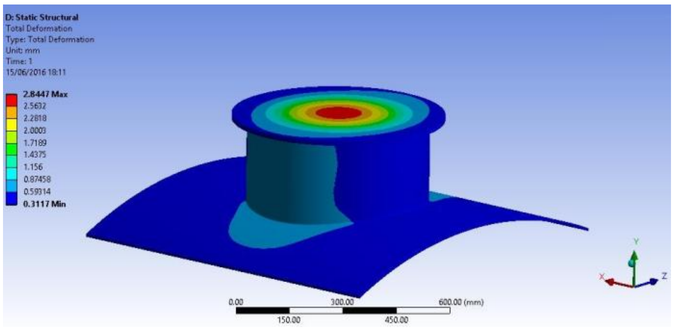

Fig. 8 *Distribution of the displacement of top opening submodel*

An overview of the equivalent stresses in the top opening model can be seen in Figure 9. The submodel of the top opening shows increased stresses. By increasing the number of elements in the area of the tank opening, the stresses are approaching more accurate values. The largest stresses occur in the inner part where the top opening and the cylindrical part of the tank connect. These stresses do not affect the safety of the structure and they occur at the edge between the cylindrical part of the tank and the tank opening with maximum value of 185,31 MPa.

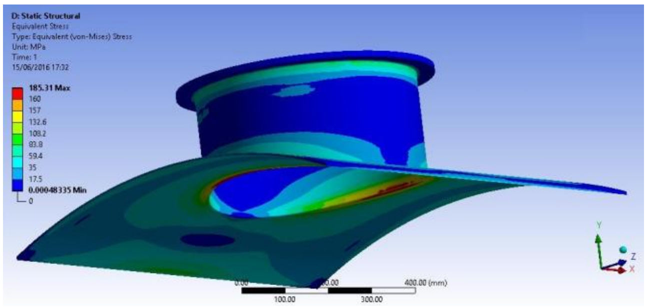

Fig. 9 *Distribution of the equivalent von Mises stress of top opening submodel*

For the calculation of the submodel for lifting the tank as well as for the submodel for opening the tank, it is necessary to use the results of displacement of the global 3D tank model. As a boundary condition for lifting lug submodel, a calculated force of 28318,23 N needed for lifting horizontal tank is applied. Figure 10 shows the distribution of the displacement of the entire model when lifting the tank, and Figure 11 shows the distribution of the equivalent stress according to von Mises. The lifting lug submodel is made in order to get more accurate displacement and stress distribution. During the lifting of the horizontal tank, maximum displacement is 2,6935 mm on the lifting lug.

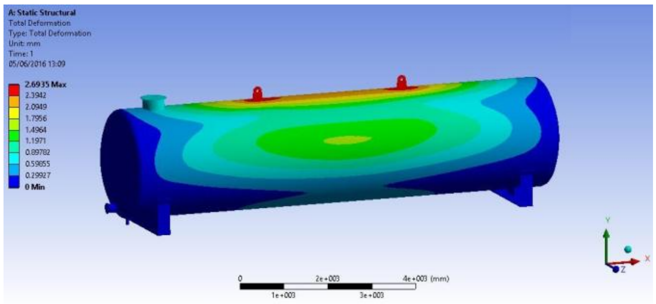

Fig. 10 *Distribution of the displacement of global model during lifting of horizontal tank*

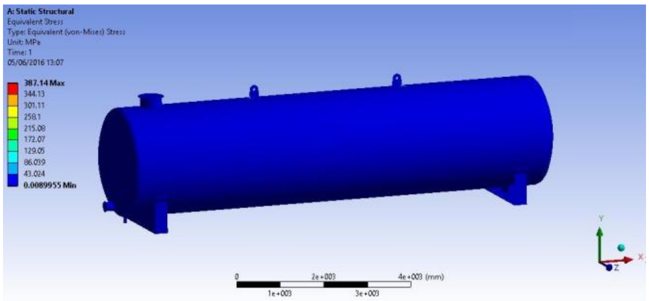

Fig. 11 *Distribution of the equivalent von Mises stress of global model during lifting of horizontal tank*

A detail of the equivalent von Mises stress distribution around the lifting lug is shown in Figure 12. The maximum stress according to von Mises is 387,14 MPa.

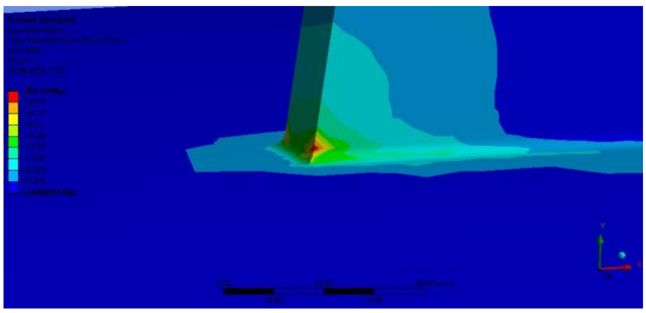

Fig. 12 *Distribution of the equivalent von Mises stress of lifting lug submodel*

Figure 13 shows a mesh of a lifting lug submodel consisting of 696362 nodes and 423237 finite elements.

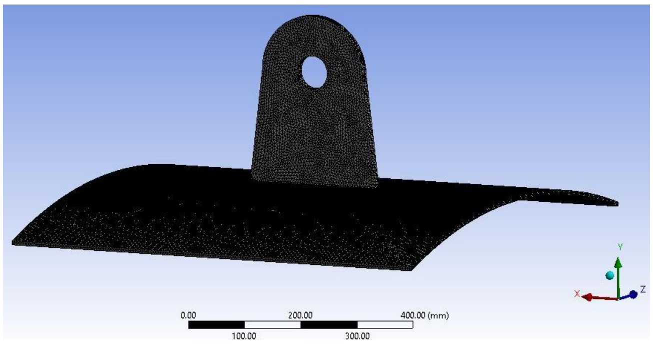

Fig. 13 *Finite element mesh of lifting lug submodel*

After meshing, lifting force was applied as a boundary condition in the tank lifting lug submodel. An overview of the total displacement distribution of the tank lifting lug submodel is shown on Figure 14. The maximum total displacement is 2,9195 mm. The equivalent von Mises stresses in the lug model are presented in Figure 15.

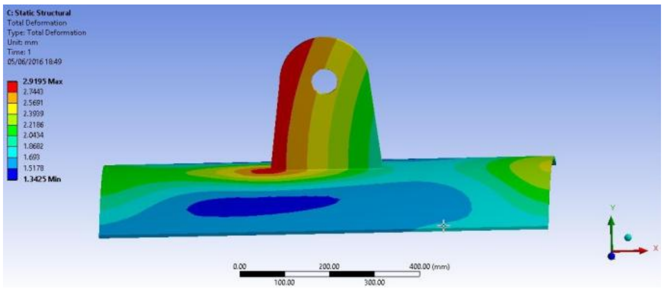

Fig. 14 *Distribution of the displacement of lifting lug submodel*

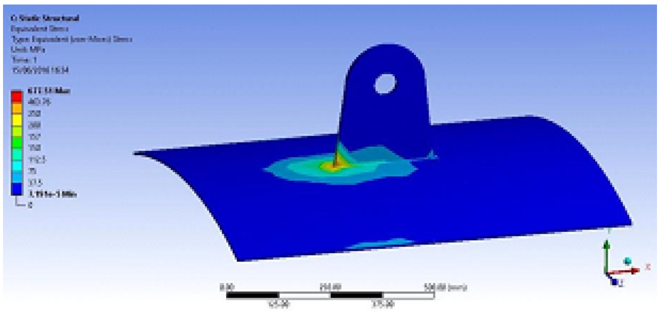

Fig. 15 *Distribution of the equivalent von Mises stress of lifting lug submodel*

Figures 14 and 15 show a high stress of 677,51 MPa at the joint between the tank and the lifting lug in the area where the lifting lug is not fully welded, due to stress concentration caused by geometry. Significant displacements are not visible in this part. The maximum total displacement is at the very top of the lifting lug and along the edge of the lug. The displacement for this part is 2,9195 mm.

4. Conclusions

Tanks for the storage of petroleum products, like all other tanks, whether pressurized or not, are manufactured according to precisely defined standards. The standards determine everything about the tank, from the materials, dimensions, welds, wall thickness, tank supports, etc. Once the tank volume has been determined and selected, it is necessary to follow the procedure specified by the standards to manufacture the tank. The standard used in this paper is EN 12285-1:2018. Since only some dimensions are defined in the standard, all dimensions must be harmonized to obtain a tank with the desired volume.

In this paper, the CAD / FEM has contributed a lot to confirm the required stresses and strains of the tank. In the paper, the tank is tested as specified in the standard. The test is carried out at a precisely specified pressure, which depends on the application of the tank. To achieve greater accuracy of the results, it is necessary to make the mesh as dense as possible at the location of the stress concentration. Increasing the number of elements and nodes would give more accurate results. Since the design is large and lacks computational power, it was useful to create submodels to describe the displacements and stresses at the tanks locations that are of interest.

At the very end, it can be stated that the tank fulfils all the conditions set out in the paper. The tank complies with the standard and meets all the criteria given to the designer in the manufacture of such tanks.

5. References

1. EN 12285-1:2018, *Workshop fabricated steel tanks - Part 1: Horizontal cylindrical single skin and double skin tanks for the underground storage of flammable and nonflammable water polluting liquids other than for heating and cooling of buildings*, 122, (2018.)
2. Pireko d.o.o., Oroslavje, Hrvatska, Technical Catalogue. Fuel tanks.
3. Ibrahim, A., Ryu, Y. and Saidpour, M. *Stress Analysis of Thin-Walled Pressure Vessels.* Modern Mechanical Engineering, 5, 1-9., (2015)
4. Anandhu, P. D., Avis, A. *Design and Analysis of Horizontal Pressure Vessel and Thickness optimization*. International Journal of Innovative Research in Science, Engineering and Technology (IJIRSET), 7, 8171-8177., (2017)
5. Zore, A. Y., Qaimi, M. G. *Design and Optimization of Saddle for Horizontal Pressure Vessel*, International Engineering Research Journal (IERJ) Special Issue 2, 4, 4201-4204., (2015)
6. Birhane, A.H., Hari, Y. *Finite Element Analysis and Design of a Horizontal Tank on Saddle Supports.* Proceedings of the ASME 2002 Pressure Vessels and Piping Conference. Vancouver, BC, Canada. August 5–9, 7, 275-281. (2002)
7. Agale, C.M., Awachat, P.N. *Finite Element Analysis of Horizontal Pressure Vessels Saddle*, International Journal of Innovations in Engineering and Science, Vol. 2, No.2, 7, 7-13. (2017)
8. Patil, Jaydip & Kumbhar, Samir & More, Nilesh & Patil, Ranjit. *Finite Element Analysis of Saddle Support for Stack Heat Exchanger*. International Journal of Engineering Applied Sciences and Technology. 5, 78-85. (2020)
9. Patil R. B., Kulkarni V. V. *Finite Element Analysis of Support Designs for Road Tankers*, International Research Journal of Engineering and Technology (IRJET), Vol. 5, No.2, 8, 836-843. (2018)
10. Ansys, ANSYS 18.0, Ansys Inc., 2017.
11. SolidWorks Premium 2021, Dassault Systems, 2021.
12. W. B. Greyer, *Handbook of storage tank systems: Codes, Regulations, and Designs*, 360, (2000.)
13. EN 10028-2:2017, *Flat products made of steels for pressure purposes – Part 2: Non-alloy and alloy steels with specified elevated temperature properties*, 31, (2008.
14. EN 1993-1-7:2008/NA, *Eurocode 3: Design of steel structures -- Part 1-7: Plated structures subject to out of plane loading*, 5, (2008.)