

Title:

A Methodological Approach for the Design of Composite Tanks Produced by Filament Winding

Authors:

Alessio Vita, alessio.vita@univpm.it, Università Politecnica delle Marche
 Samir Borriello, s1079613@studenti.univpm.it, Università Politecnica delle Marche
 Daniele Landi, d.landi@univpm.it, Università Politecnica delle Marche
 Martina Scafà, m.scafa@pm.univpm.it, Università Politecnica delle Marche
 Michele Germani, m.germani@univpm.it, Università Politecnica delle Marche

Keywords:

Composite Vessels, Virtual Prototyping, Filament Winding

DOI: 10.14733/cadconfP.2019.288-292

Introduction:

Nowadays, hydrogen is one of the most promising alternatives to fossil fuels. The development of a lightweight, economical and safe hydrogen storage system is the main goal to be reached for the large-scale deployment of fuel cell vehicles. Storing gaseous hydrogen in high pressure vessels is the most mature and popular technology [10] and it is also already adopted by modern hydrogen commercial cars.

There are different types of high pressure vessels: type I (completely metallic vessel), type II (metallic vessel reinforced by composite hoop wrap), type III (metallic liner, fully wrapped with composite material) and type IV (polymeric liner, fully wrapped with composite material). Type V tanks (linerless structure, completely made of composite material) are also under development [2]. Except for type V tanks, type III and type IV pressure vessels are currently considered the best answers to the hydrogen storage issue due to their lightness and fatigue resistance [8,1]. These tanks are manufactured by the filament winding process. The liner is installed on the filament winding machine where is used as mandrel. Then, it is fully wrapped with carbon fibers, in both helical and hoop directions, impregnated with epoxy resin [1]. The nominal working pressure of composite pressure vessels for hydrogen storage is ordinarily 35-70 MPa [1].

Various aspects must be considered during the design process of composite overwrapped pressure vessels. Firstly, the materials choice and geometry definition have to be chosen. Secondly, through the netting analysis, the designer can calculate a starting thickness for hoop and helical layers. The number and the thickness of composite layers can then be refined by finite element analysis. All these phases are widely discussed in literature. Some authors, such as Johnson *et al.* [4] or Hua *et al.* [3], have carried out the performance assessment of hydrogen 70 MPa pressure vessels considering alternative carbon fibers with different mechanical properties or alternative epoxy resins. Others, like Koussios *et al.* [5,6], have extensively worked on the analytical model that describes the optimal dome shape for cylindrical tanks, considering both geodesic and non-geodesic fiber trajectories. Still others, like Ramirez *et al.* [8] and Xu *et al.* [9], have developed accurate finite element models for the prediction of composite tanks burst pressure, followed by the comparison between numerical and experimental results. However, the whole design process of composite pressure vessels is rarely discussed in literature.

In this paper, an original approach for the virtual prototyping of composite pressure tanks is proposed. This method aims to reduce the finite element model development time by the integration between the computational software MATLAB and the FEA tool (Abaqus). The presented procedure allows the designer to quickly import the suitable dome geometry into Abaqus, without the need of

going through CAD software. First, an analytical model for the optimal dome shape of composite pressure vessel is presented, followed by an overview on the meaning of the different equation parameters. Then, the working principle of the MATLAB script is described in depth with a focus on the phase in which the dome profile is imported into Abaqus. Finally, a simple case study of the application of this method is proposed.

Main Idea:

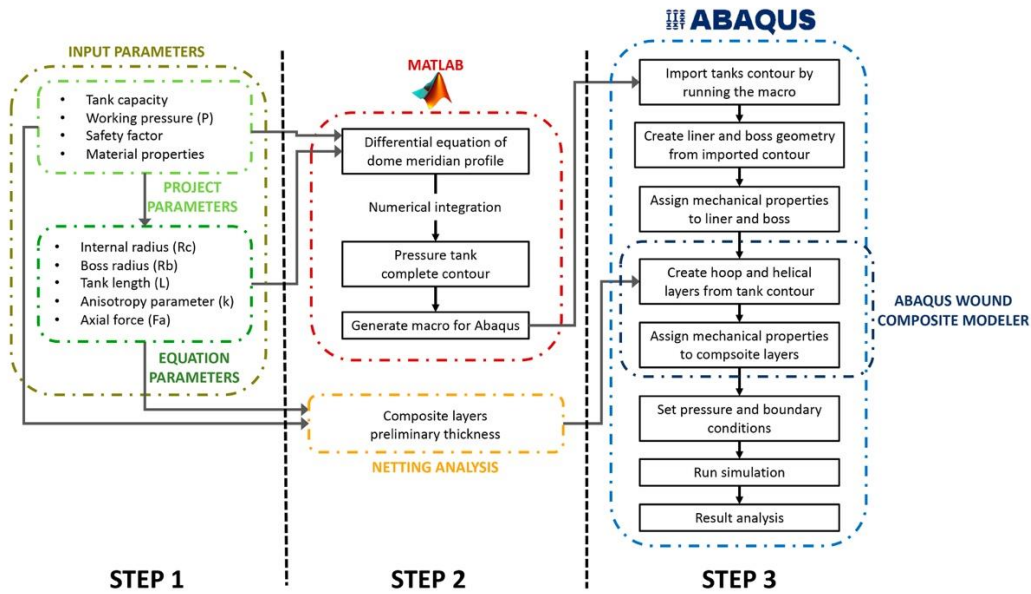


Fig. 1: Workflow of the composite pressure vessel design process.

The design method presented in this paper is shown in Fig. 1. The procedure starts with the definition of the project parameters, which are typically the tank capacity, the working pressure, the material properties and the safety factor for burst pressure. From these data, the designer can calculate other parameters which are necessary for the tank geometry definition (Step 1). These parameters are then used for the dome shape design. The dome shape of composite pressure vessels has fundamental influence on tank mechanical performances. The use of an anisotropic material such as carbon fiber reinforced polymer (CFRP) requires the dome contour to be accurately modeled to ensure adequate stress conditions. A commonly used dome geometry is the isotensoidal one, based on geodesic roving trajectories [7]. The parameters necessary for the profile generation are shown in Fig. 2.

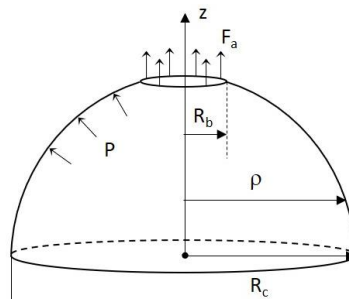


Fig. 2: Dome schematic representation.

R_b is the boss radius, R_c is the cylinder radius, ρ is the radial coordinate, z is the axial coordinate, P is the tank pressure, and F_a is the axial load.

Starting from these parameters and coordinates, it is possible to introduce the dimensionless ones:

- $Y = \rho / R_b$, with $Y \geq 1$
- $Y_{eq} = R_c / R_b$,
- $Z = z / R_b$, with $Z > 0$
- $r = F_a / \pi P R_c^2$

The analytical expression of the dome meridian profile (Eqn. (1)) is then [1-3]:

$$Z'(Y) = \pm \frac{Y(Y^2 + rY_{eq}^2)}{\sqrt{\left(\frac{k + Y^2 - 1}{k + Y_{eq}^2 - 1}\right)^{k+1} (1 + r)^2 Y_{eq}^6 - Y^2(Y^2 + rY_{eq}^2)^2}} \quad (1)$$

where k is the anisotropy parameter. The expression of k which gives the zero in-plane shear stress condition at fibers [2-3] is considered in this paper (Eqn. (2)).

$$k = k_e = \frac{E_2(1 + \nu_{12})}{E_1(1 + \nu_{21})} \quad (2)$$

The Abaqus Wound Composite Model (WCM) plug-in is a tool that simplifies the development of finite element model for filament winding manufactured parts. This plug-in allows to automatically create the geodesic-based dome geometry by setting the boss radius (R_b) and the cylinder radius (R_c). However, it does not allow the user to set the anisotropy parameter (k_e) or the axial load (defined as the dimensionless force r). Because of this, it is not possible to create a tank model with the optimal dome shape by using the plug-in alone. In this case the drawback has been solved by creating a suitable MATLAB script. This program receives as input, in addition to R_b and R_c , also k_e and r . Next, it numerically integrates the Eqn. (1) and generates the whole tank profile. Finally, it writes a new macro with all the instructions for the profile drawing in Abaqus (the macro is run directly in Abaqus). Once that macro has been run, the tank profile is generated in Abaqus and the finite element model can be built. This procedure is summarized in Fig. 1 (Step 2).

Netting analysis allows to estimate the wall thickness of CFRP layers by assuming that the fibers carry all the load, without any resin mechanical contribute [7]. Netting analysis can be performed before the FE analysis to define composite preliminary thickness for the FE model (Fig. 1, Step 2). Once the geometric information has been imported in Abaqus, the FE model can be built. The liner and the boss are modeled from the imported tank profile (generated with MATLAB) using Abaqus drawing tools. The CFRP layers are modeled by the same imported tank profile using the WCM plug-in, considering the thicknesses calculated with the netting analysis (Step 3).

Case Study

The parameters assumed in the presented case study are: R_c 100 mm, R_b 20 mm, L 300 mm, P 70 MPa, k_e 0.088, r 0.1. In Fig. 3a, the comparison between the dome profile generated by the WCM plug-in and the one generated by the MATLAB script is reported. The different is due to the fact that in the MATLAB script, the parameters k_e and r take into consideration the material properties and operational conditions while in the plug-in they are assumed as constant. Using the latter method, a non-perfect isotenoidal geometry could be obtained. Thus, using the script, the tank profile generated with those parameters is illustrated in Fig. 3b.

A FE model of a 70 MPa pressure vessel has been developed. Netting analysis has been performed considering a failure stress along fiber directions of 2550 MPa and a safety factor of 2.25. The materials properties are reported Tab. 1.

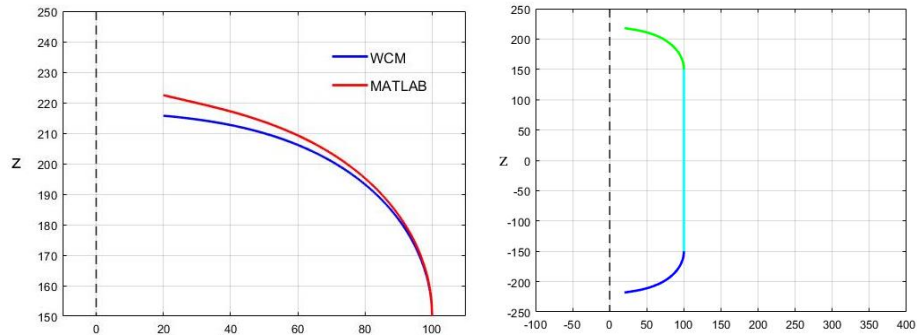


Fig. 3: WCM and MATLAB dome profile (a), tank overall profile generated by MATLAB script (b).

The thicknesses of the liner is 4 mm while that of the CFRP is 9.5 mm (6 mm the layers in hoop direction and 3.5 mm in the helical).

	E_1	E_2	E_3	ν_{12}	ν_{13}	ν_{23}	G_{12}	G_{13}	G_{23}
CFRP	135 GPa	9.6 GPa	9.6 GPa	0.25	0.25	0.4	6 GPa	6 GPa	3.5 GPa
HDPE (liner)	1.6 GPa	-	-	0.3	-	-	-	-	-
Aluminum (boss)	70 GPa	-	-	0.3	-	-	-	-	-

Tab. 1: Materials properties for FEA.

Axisymmetric elements have been used in order to simplify the analysis. The geometry of the tank and the model mesh are illustrated in Fig. 4(a). Mesh elements of composite layers are depicted with different colors as they have different fiber orientation (Fig. 4(b)).

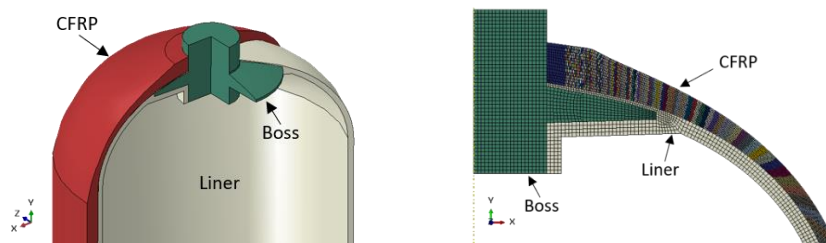


Fig. 4: Tank 3D geometry (a); part view of the axisymmetric mesh of the FE model (b).

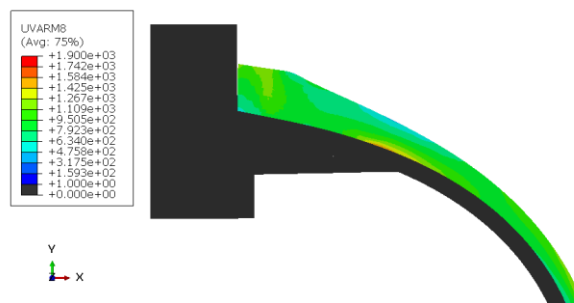


Fig. 5: Normal stress in composite material along fiber direction.

Fig. 5 shows the results of FE analysis for the model presented as case study. In particular the normal stress along fiber directions at top dome region is illustrated. Those results refer to a first attempt tank model with non-optimized parameters. However, the dome meridian profile could be improved by an optimization process in order to find which dome shape gives the best stress condition. Experimental validation should also be performed to show the FE model accuracy level.

Conclusions and Future Development

This paper presents an original approach to composite pressure vessel geometry importation in FEA software. This method allows the designer to have control on all parameters which have influence on dome shape. Indeed, the Abaqus WCM plug-in does not permit to specify the anisotropy parameter and the axial load. The MATLAB script is able not only to integrate the dome meridian profile equation (Eq n. (1)), but also to write a macro to import quickly the profile in Abaqus, without using a CAD software. A comparison between the time consumption of the traditional method and the time consumption of the method here proposed has been performed. This method allows the designer to have control on all parameters which have influence on dome shape. Indeed, commercial tools, such as the Abaqus WCM plug-in, do not permit to specify the anisotropy parameter and the axial load. Moreover, this method allows designers to save about 7-9 minutes every time they need to import a tank profile in a FE software, compared to a traditional manual method.

The proposed design method is limited to tank geometry definition, geometry importation in FEA platform and FE model creation. However, this approach could be implemented in a dome optimization process to find which values of k_e and r gives to the composite layers an optimal stress condition. Future work will be focused on the testing of real composite tanks in order to validate the FE model and on the correlation between the input parameters and the obtained mechanical performances of the composite vessel.

References:

- [1] Barthelemy, H.; Weber, M.; Barbier, F.: Hydrogen storage: Recent improvements and industrial perspectives, *International Journal of Hydrogen Energy*, 42, 2017, 7254-7262. <https://doi.org/10.1016/j.ijhydene.2016.03.178>.
- [2] Cronin, J.; Mallick, K.; Lake, M.; Warner, M.; Munshi, N.: Damage and leakage barrier in all-composite pressure vessels and storage tanks, U.S. Patent 8,074,826 B2, December 13, 2011.
- [3] Hua, T. Q.; Roh, H.-S.; Ahluwalia, R.-K.: Performance assessment of 700-bar compressed hydrogen storage for light duty fuel cell vehicles, *International Journal of Hydrogen Energy*, 42, 2017, 25121-25129. <https://doi.org/10.1016/j.ijhydene.2017.08.123>.
- [4] Johnosn, K.; Veenstra, M.; Gotthold, D.; Simmons, K.: Advancements and opportunities for on-board 700 bar compressed hydrogen tanks in the progression towards the commercialization of fuel cell vehicles, *SAE International Journal of Alternative Powertrains*, 6, 2017, 201-218. <https://doi.org/10.4271/2017-01-1183>.
- [5] Koussios, S.: Design of cylindrical composite pressure vessels: integral optimization, 17th International Conference on Composite Materials, Edinburgh, UK, July 2009.
- [6] Koussios, S.; Lei Zu; Tapeinos, I. G.; Beukers, A.; Sippel, M.; Kopp, A.: Some observations on the influence of the classic failure criteria on the optimal design of pressure vessels, *Proceedings of the American Society for Composites - Twenty-eighth technical conference*, State College, PA, September 2013.
- [7] Peters, S.-T.: *Composite Filament Winding*, ASM International, 2011
- [8] Ramirez, J. P.-B.; Halm, D.; Grandidier, J.-C.; Villalonga, S.; Nony, F.: 700 bar type IV high pressure hydrogen storage vessel burst, simulation and experimental validation, *International Journal of Hydrogen Energy*, 40, 2015, 13183-13192. <https://doi.org/10.1016/j.ijhydene.2015.05.126>.
- [9] Xu, P.; Zheng, J.-Y.; Liu, P.-F.: Finite element analysis of composite hydrogen storage vessel, *Materials and Design*, 31, 2009, 2295-2301. <https://doi.org/10.1016/j.matdes.2009.03.006>.
- [10] Zheng, J.; Liu, X.; Xu, P.; Liu, P.; Zhao, Y.; Yang, J.: Development of high pressure gaseous hydrogen storage technologies, *International Journal of Hydrogen Energy*, 37, 2012, 1048-1057. <https://doi.org/10.1016/j.ijhydene.2011.02.125>.