








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

Daniele Landi¹ , Alessio Vita² , Samir Borriello³ , Martina Scafà⁴  and Michele Germani⁵ 

¹Ancona Università politecnica delle Marche, d.landi@univpm.it

²Ancona Università politecnica delle Marche, alessio.vita@staff.univpm.it

³Ancona Università politecnica delle Marche, s1079613@studenti.univpm.it

⁴Ancona Università politecnica delle Marche, m.scafa@pm.univpm.it

⁵Ancona Università politecnica delle Marche, m.germani@univpm.it

Corresponding author: Daniele Landi, d.landi@univpm.it

Abstract. In this paper, an original approach for the virtual prototyping of composite pressure tanks is proposed. The main tests to be conducted for the homologation of the vehicle tank is the burst pressure, which is a quasi-static test. This method aims to reduce the finite element model development time by the integration between the computational software MATLAB and the FEA tool Abaqus. Since the dome shape has fundamental influence on the mechanical performances of the composite pressure vessel, the presented procedure allows the designer to quickly import the suitable dome geometry into Abaqus, without the need of going through CAD software. The first step of the method here reported is the definition of all the geometric and operational parameters necessary to the construction of the dome meridian profile. The second step is to enter those parameters in a MATLAB script, which is able to integrate the dome profile differential equation, to generate the whole tank profile and to import this profile into Abaqus. Once the geometry has been imported, a FE model of the high-pressure vessel can be built and virtual simulations can be performed. This approach could be implemented in a dome optimization process to find which dome meridian profile gives the best tank performances.

Keywords: Composite Vessels, Virtual Prototyping, Filament Winding.

DOI: <https://doi.org/10.14733/cadaps.2020.1229-1240>

1 INTRODUCTION

The millennium development goals that were established in 2000 under the United Nations cover only narrowly defined notions of social development goals in areas such as education and health. In an effort to include more fundamental factors such as the environment, a new set of sustainable development goals were defined in 2015 through key international forums and summits. In

addition to the SDGs, efforts are being made through the United Nations Framework Convention on Climate Change to reduce the emissions of greenhouse gases (GHGs). Currently, many nations are pursuing efforts to reduce emissions from the transportation sector [3].

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 [18] and it is also already adopted by modern hydrogen commercial cars [7].

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 [4]. Except for type V tanks, type III and type IV pressure vessels are currently considered the best solutions to the hydrogen storage issue due to their lightness and fatigue resistance [13][1]. These tanks are manufactured by the filament winding process. The liner is installed on the filament winding machine where it 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 and geometry 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. [8] or Hua et al. [6], 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. [9][10], 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. [13] and Xu et al. [16], have developed accurate finite element models for the prediction of composite tanks burst pressure, followed by the comparison between numerical and experimental results. However, some aspects of the design process of composite pressure vessels are omitted in those papers. One of these aspects is the phase of tank geometry definition and geometry importation into FEA environment, which is rarely discussed in literature.

In the case of pressure vessels, the domes shape has fundamental influence on the mechanical performances of the tank. For isotropic material (e.g. steel) the typical dome geometry is the hemispherical or the semi-elliptical one [11], while for non-isotropic materials (such as CFRP) the adequate geometry becomes more complicated. The analytical models suggested in literature that describe the optimal dome meridian profile for composite pressure vessels are based on a differential equation [2][9][10][12].

So, the first step of geometry definition is to numerically integrate the equation to obtain the profile as a point-sequence curve. Then, in order to develop the FEA model, the curve must be imported in the FEA software. A traditional approach could be to import the profile in a CAD software [15], save it in neutral file format (like STEP or IGES) and then open the generated file with the FEA software. It is a simple but not automatic and time-consuming method.

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.

2 METHODOLOGY

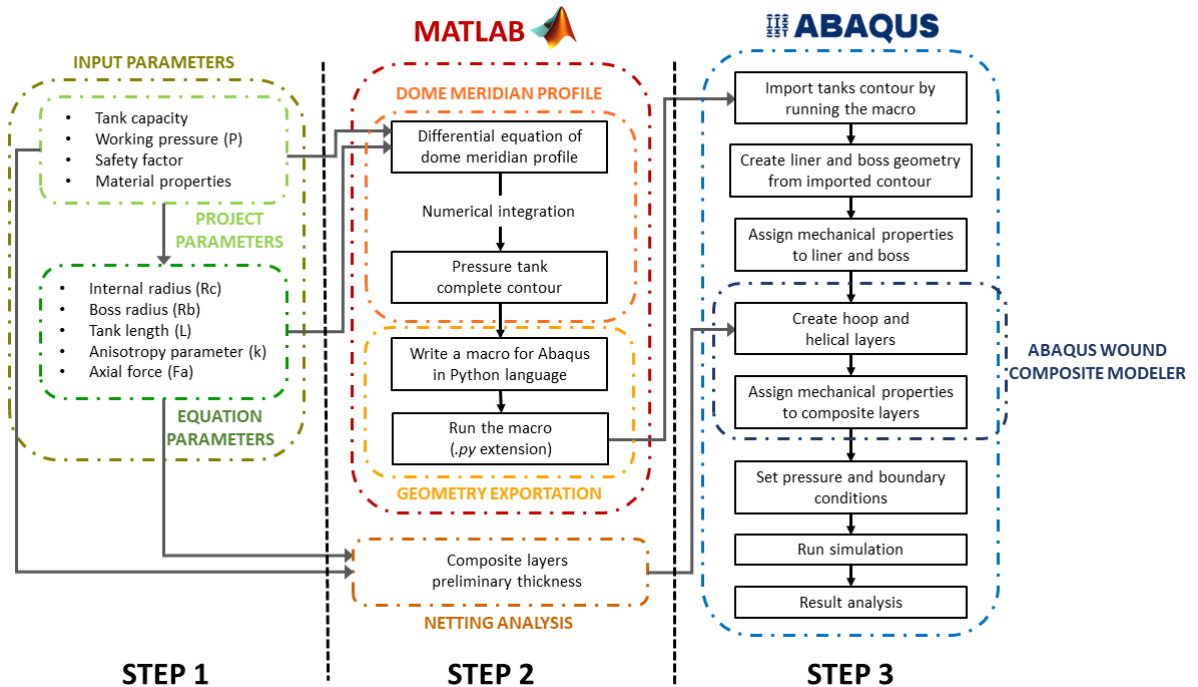


Figure 1: Workflow of the composite pressure vessel design process.

The design method presented in this paper is shown in Figure. The design 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 derive other parameters which are necessary for the tank geometry definition (Step 1), such as: radius of the cylindrical part, tank length, boss radius, the anisotropy parameter of the composite material and the axial force at the tank end.

Once these parameters have been defined, the second step is to numerically integrate a differential equation to define the optimal dome meridian profile and to export the geometry to the FEA software. As described in the following paragraph (Paragraph 2.2), the computational software Matlab has been used both to solve the equation and to generate the profile directly on the FEA platform exploiting a python macro. Before the FE model is created, a preliminary thickness of the tank can be obtained by simple hand calculations (Netting analysis).

The final step is to develop the FE model (Step 3). Since in the Netting Analysis the mechanical contribution of the resin is neglected, the preliminary thickness calculated in Step 2 is greater than necessary [12]. Therefore, the FE analysis should be performed to obtain more accurate results about the tank stress condition and then to reduce the layers thickness. In this case, the Wound Composite Modeler plug-in for Abaqus has been selected to perform FE simulations. This is a

sophisticated and powerful tool for the FE modeling of parts manufactured by the filament winding process [6][13].

The core of this research is represented by the Step 2 (Figure 1), however, for the sake of completeness, other complementary phases of composite pressure vessels design are described.

2.1 The Dome Meridian Profile

The domes shape of composite pressure vessels has fundamental influence on tank mechanical performance. 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 first optimal shape can be obtained by using the classical shell membrane theory. Even if this theory is not highly accurate for thick-walled pressure vessels, it is considered a good starting point in the designing of those tanks [10]. A commonly used dome geometry is the isotensoidal one, based on geodesic roving trajectories [11]. This profile is obtained by imposing the shear stress at layer level equal to zero [9][10][12]. There are also other analytical models for dome profile in literature, like the maximum tensile strength-based profile and the stress criterion minimizing profile [10]. However, the dome shapes obtained with these models are very similar to the one obtained with the zero in-plane shear stress condition [10].

The parameters and the coordinates necessary for the dome meridian profile definition are shown in Figure 2.

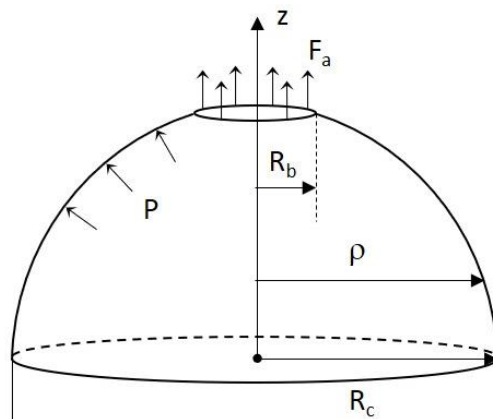


Figure 2: Dome schematic representation.

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

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

- $Y = \rho / R_b$ 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 (Equation (2.1)) is then [6,9,10]:

$$Z'(Y) = \pm \frac{Y(Y^2 + rY_{eq}^2)}{\sqrt{\left(\frac{k+Y^2-1}{k+Y_{eq}^2-1}\right)^{k+1} (1+r)^2 Y_{eq}^6 - Y^2(Y^2 + rY_{eq}^2)^2}} \quad (2.1)$$

where k is the anisotropy parameter. The expression of k which gives the zero in-plane shear stress condition at fibers [10][12] is considered in this paper (Equation (2.2)).

$$k = k_e = \frac{E_2(1 + \nu_{12})}{E_1(1 + \nu_{21})} \quad (2.2)$$

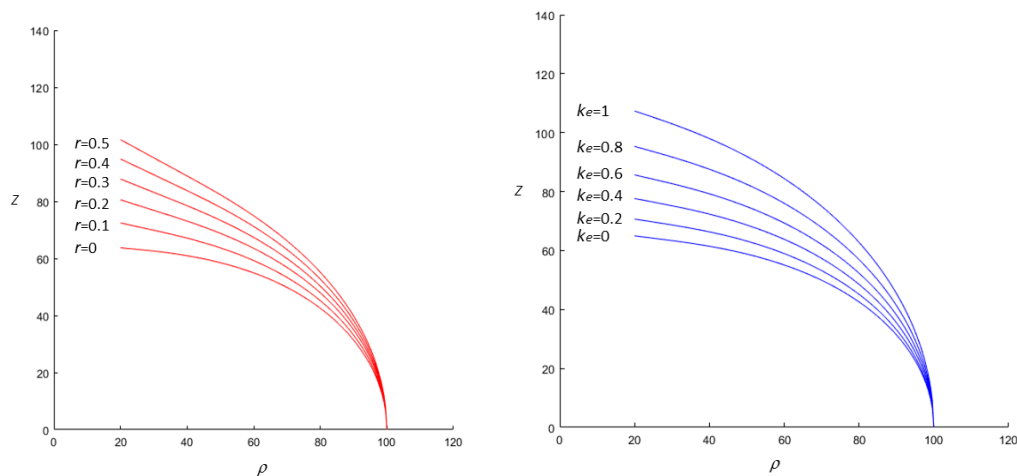


Figure 3: Influence of r and k on the meridian profile geometry.

As shown in Figure , both r and k have high influence on the meridian profile geometry. When the anisotropic parameter is equal to 1, the material is isotropic and the dome profile is almost hemispherical. When k is equal to 0, the transversal mechanical properties of the material are neglected (e.g. resin does not contribute on the mechanical properties of the CFRP).

The profile Equation (2.1) has no analytical solution, so it must be numerically integrated by a dedicated software.

2.2 Geometry Exportation

Once the desired geometry has been calculated, the next step is to import the dome meridian profile in a FEA platform.

A simple method could be to build a profile curve using a CAD tool and then to save it in a file format that can be opened with the FEA software [15]. Unfortunately, this is a manual method that cannot be used in automatic processes, such as optimization routines. As a matter of fact, each time the designer has the need to define a different dome shape, he/she must integrate the Equation (2.1), import the points coordinates (ρ, z) on the CAD software, generate the point-sequence curve, save it in a neutral file format and finally open the file with the FEA software. It is evident how this is a time-consuming process which requires a CAD tool as well as skilled designers.

Another method could be to use dedicated FEA plug-in to directly generate the profile in the FEA software, without the need of integrate the profile equation or use CAD software. One of these plug-ins is for example the Abaqus Wound Composite Modeler (WCM) plug-in. WCM 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) or the axial load (defined as the dimensionless force r). Because of this, it is not possible to create a tank model with an arbitrary dome shape by using only this plug-in.

In order to reduce the time for the tank geometry importation in the FEA tool and to have the possibility to set all the parameters for the profile calculation, a suitable MATLAB script has been developed. First, this program allows the user to define all the input data and, thus, not only R_b and R_c , but also k and r . Next, it numerically integrates the Equation (2.1) and generates the whole tank profile as a point-sequence curve. Finally, it writes a macro with all the instructions for the profile drawing in Abaqus. This macro is a command lines sequence written using Python programming language, which is the graphical user interface language of Abaqus. The macro can be run in Abaqus by using the Macro Manger tool, or directly in Matlab. Once the macro has been run, the tank profile is generated automatically in Abaqus and the finite element model can be built on the imported geometry. In few seconds, if necessary, the designer can change the input parameters and draw a new curve by running the Matlab script again. This procedure is summarized in Figure 1 (Step 2).

The comparison between the traditional approach based on the manual process and the proposed method is shown in Figure.

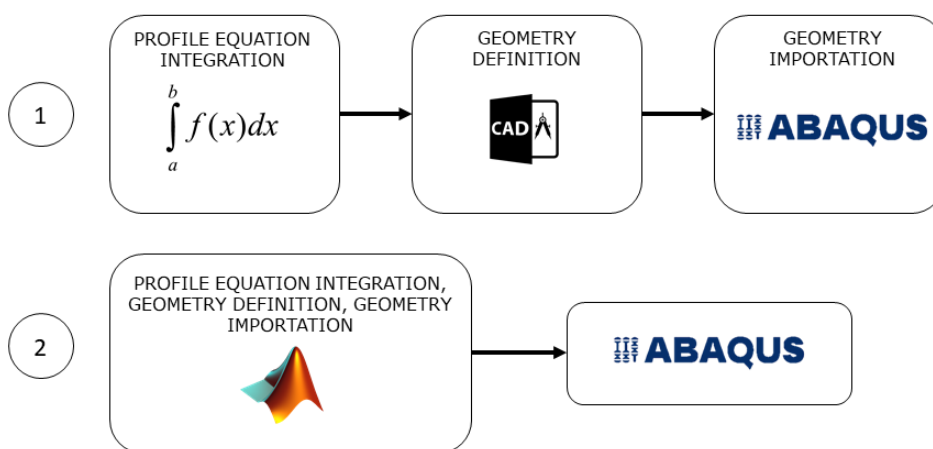


Figure 4: The traditional manual method workflow (1) compared to the one here proposed (2).

2.3 The Netting Analysis

Before the development of a FEA model, the composite material preliminary thickness should be calculated.

Netting analysis allows to estimate the wall thickness of helical and hoop CFRP layers by assuming that the fibers carry all the load, without any resin mechanical contribute [12]. The analytical expressions of the total thicknesses for hoop layers (t_{90}) and helical layers (t_{α}) are:

$$t_{90} = \frac{pR_c(2 - \tan^2\alpha)}{2VF\sigma_{f,90}} \quad (2.3)$$

$$t_{\alpha} = \frac{pR_c}{2VF\sigma_{f,\alpha}\cos^2\alpha} \quad (2.4)$$

where R_c is the cylinder radius, p is the internal pressure, α is the fiber winding angle in the cylindrical part for the helical layers, $\sigma_{f,90}$ is the design allowable fiber tensile strength for hoop layers, σ_α is the design allowable fiber tensile strength for helical layers and VF is the fiber volume fraction in the composite [6].

After the calculation of the preliminary composite thickness and the import of tank profile into Abaqus, the FE model can be developed (Step 3 in Figure). The boss geometry and the liner geometry can be built directly using the Abaqus drawing tools by using the tank profile as a reference geometry. Next, the CFRP layers can be modeled by the same tank profile using the WCM plug-in, considering a nominal thickness equal to the one calculated with the netting analysis.

3 CASE STUDY

The proposed design method has been applied during the design of a hydrogen 700 bar, type IV pressure vessel. Hydrogen storage is an important component for the new vehicle concept, but badly-designed tanks can leak or explode, so good tank design is vital. One of the main tests to be conducted for the homologation of the tank is the burst pressure, which is a quasi-static test. For this reason, we have taken into account only the static condition. All the three steps defined in Figure 1 are described in the next paragraphs.

3.1 Step 1: Input Parameters

The design parameters are reported in Table 1 while the materials properties are reported in Table 2. The CFRP is made out of T700 unidirectional fibers and epoxy resin. The mechanical properties of the composite material have been found in literature [11][6].

R_c	R_b	L	P	k_e	r
100 mm	20 mm	300 mm	70 MPa	0.088	0.1

Table 1: Case study input parameters.

	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	-	-	-	-	-

Table 2: Materials properties for FEA.

3.2 Step 2: Geometry Definition and Exportation

The tank profile has been obtained integrating the Equation (2.1) using the Matlab script (Figure 5(a)). Then, it has been imported into Abaqus by running the Python macro. As shown in Figure 5(b) the dome profile here proposed is clearly different from the one generated by the WCM plug-in. This, as mentioned above, is due to the adoption of an analytical expression for the dome profile that takes into account also the parameters k and r .

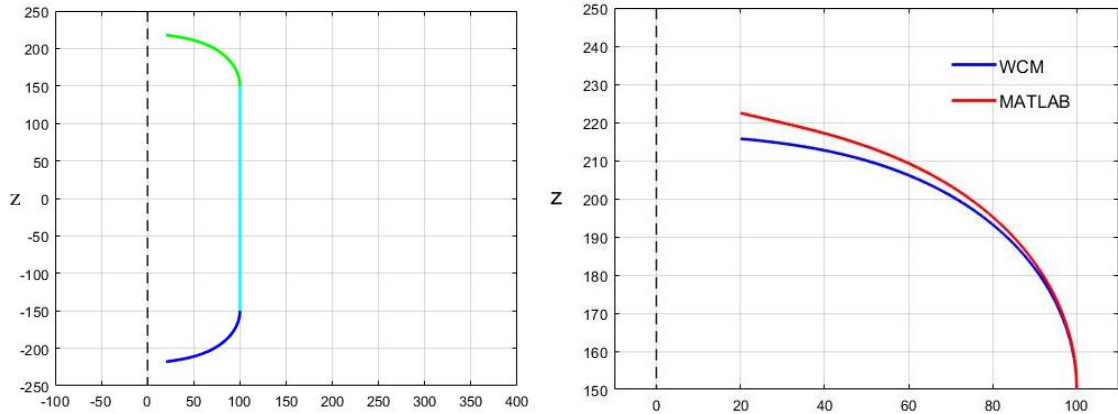


Figure 5: Tank overall profile generated by the Matlab script (a), comparison between Matlab dome profile and WCM dome profile (b).

Then, the netting analysis has been performed considering a failure stress along fiber directions of 2550 MPa for CFRP and a safety factor of 2.25, a value typically suggested by the standards. A total thickness of 9.5 mm for composite material has been obtained: 6 mm for the hoop layers and 3.5 mm for helical layers. The HDPE liner thickness has been set to 4 mm. A FE model of the tank has been developed using these preliminary thicknesses and the imported geometry.

3.3 Step 3: The FE model

Axisymmetric elements have been used in order to simplify the analysis. The 3D geometry of the tank and the model mesh are illustrated in Figure 6(a) and Figure 6(b). The elements of the composite layers are depicted with different colors as they have different fiber orientation (Figure 6(b)).

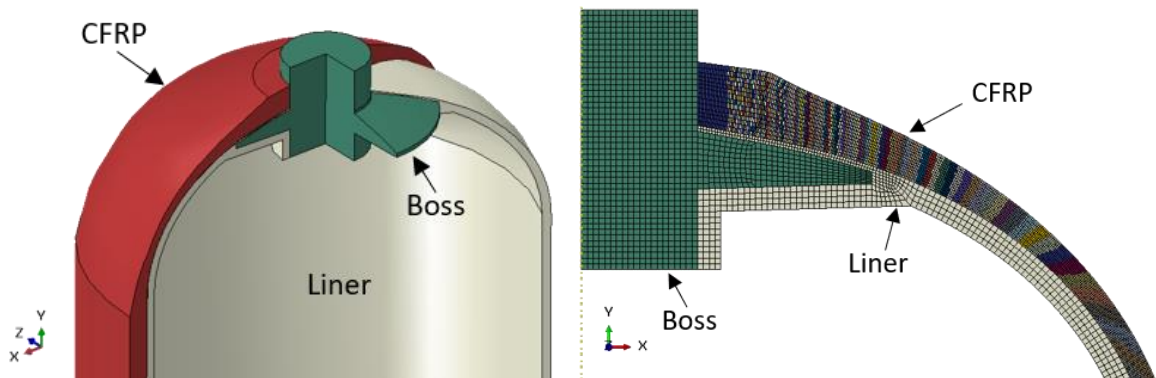


Figure 6: Tank 3D geometry (a); part view of the axisymmetric mesh of the FE model (b).

Figure 7 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. A complete design process for composite high-pressure vessels also requires experimental validation to be performed, in order to assess the FE model accuracy level.

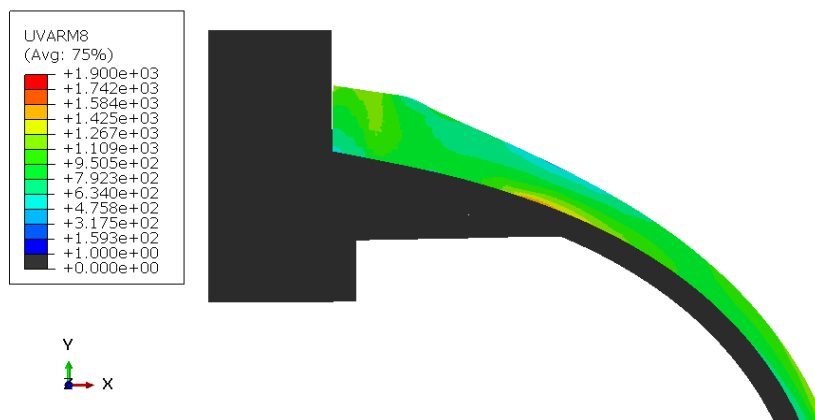


Figure 7: Normal stress in composite material along fiber direction (stress values in MPa).

4 DISCUSSION

A comparison between the time consumption of the traditional method and the time consumption of the method here proposed has been performed. In Figure 4(1) the workflow of a possible manual method to import the tank profile in Abaqus is illustrated, however, there are many software that can be used to solve the task.

For the analysis here presented a potential traditional method has been defined, in order to evaluate the corresponding time consumption for the design process. This approach is based on the computational software Microsoft Excel and the CAD software CATIA. First, the profile differential equation (Equation (2.1)) is integrated in MS Excel to obtain the points coordinates of the dome profile. Next, the point coordinate list is imported in another pre-defined MS Excel file, which is installed by default in the same CATIA program folder. This tool is used to automatically draw the tank profile in CATIA as a spline curve. Finally, the profile is saved as an IGES file that can be opened by Abaqus.

All the steps of the two processes are summarized and described in Figure. The time necessary to complete the two processes has been recorded. All the phases of the two method have been executed by the same person without making any mistakes. For the sake of completeness, the time spend to write the Excel file for the numerical integration and the time spend to write the whole Matlab script for the automatic method have also been evaluated. The results are summarized in Table 3.

	<i>Manual method</i>	<i>Automatic method</i>
Time required to write the Matlab script (automatic method)	-	2h
Time required to write the Excel file (traditional method)	1h	-
Time required to complete the steps	8-10 min	1 min

Table 3: Time consumption for the two methods.

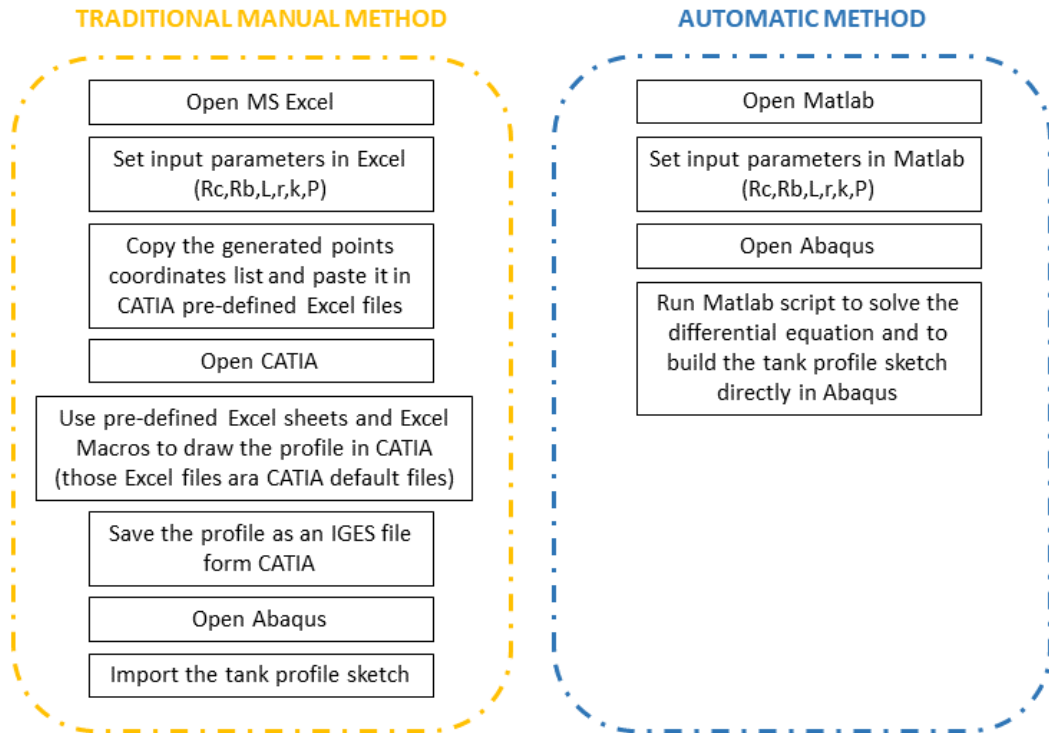


Figure 8: Steps to follow to draw the tank profile in Abaqus: (a) an example of traditional method, (b) the automatic method here proposed.

Every time the designer needs to import a different tank profile in Abaqus to develop the FE model, it is possible to save approximately 7-9 minutes by using the method here proposed instead of the traditional one. In addition, the developed method can be easily interfaced with optimization tools (such as modeFrontier or iSight) able to perform iterative optimization processes automatically. Then, this approach, once the script has been built, can be used effortlessly also by not skilled designers. Finally, considering the less time needed for the construction of the FE model, designers whose perform the geometry optimization manually are also encouraged to test more profiles with different parameters, thus investigating a larger portion of the optimization problem domain.

5 CONCLUSION

This paper presents an original approach to composite pressure vessel geometry building and importation in FEA software. 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.

However, the main strong suit of the proposed approach is the possibility to integrate it in optimization routines. For example, it could be implemented in a dome optimization process to find which values of k and r give to the composite layers an optimal stress condition. In this case it would not be possible to use the manual method, since this kind of routine requires too many FE analysis to be performed in sequence.

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] Castorani, V.; Landi, D.; Mandolini, M.; Germani, G.: Design optimization of customizable centrifugal industrial blowers for gas turbine power plants, *Computer-Aided Design and Applications*, 16(6), 2019, 1098-1111. <https://doi.org/10.14733/cadaps.2019.1098-1111>
- [3] Cicconi, P.; Landi, D.; Germani, M.: A virtual modelling of a hybrid road tractor for freight delivery, *ASME International Mechanical Engineering Congress and Exposition, Proceedings (IMECE)*, 2016.
- [4] 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.
- [5] Hong, J.-H.; Han, M.-G.; Chang, S.-H.: Safety evaluation of 70 MPa-capacity type III hydrogen pressure vessel considering material degradation of composites due to temperature rise, *Composite Structures*, 113, 2014, 127-133. <https://doi.org/10.1016/j.compstruct.2014.03.008>
- [6] 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>
- [7] Johnosn, K.; Veenstra, M.; Gotthold, D.; Simmons, K.: Advancements and opportunities for on-board 700 bar bompresed hydrogen tanks in the progression towards the commercialization of fuel cell vehicles, *SAE International Journal of Alternative Powertrains*, 6(2), 2017, 201-218. <https://doi.org/10.4271/2017-01-1183>
- [8] Kerviel, A., Pesyridis, A., Chalet, D.; Boosting system options for high efficiency fuel cell electric vehicles, *4th Biennal International Conference on Powertrain Modelling and Control Testing, Mapping and Calibration*, 2018.
- [9] Koussios, S.: Design of cylindrical composite pressure vessels: integral optimization, *17th International Conference on Composite Materials*, Edinburgh, UK, 2009.
- [10] 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 04(11)*, 2017, 617-625.
- [11] Parmar, K.-R.; Khan, A.-A.; Khiraiya, K.-B.: Comparative FE analysis of pressure vessel of hemispherical, ellipsoidal and torospherical end connection, *International Journal of Engineering Development and Research*, 3, 2014, 46-50.
- [12] Peters, S.-T.: *Composite Filament Winding*, ASM International, 2011.
- [13] 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(38), 2015, 13183-13192. <https://doi.org/10.1016/j.ijhydene.2015.05.126>
- [14] Roh, H.-S.; Hua, T.-Q.; Ahluwalia, R.-K.: Optimization of carbon fiber usage in Type 4 hydrogen storage tanks for fuel cell automobiles, *International Journal of Hydrogen Energy*, 38(29), 2013, 12795-12802. <https://doi.org/10.1016/j.ijhydene.2013.07.016>
- [15] Villalonga, S.; Thomas, C.; Nony, F.; Thiebaud, F.; Geli, M.; Lucas, A.; Kremer-Knobloch, K.; Maugy, C.: Application of full thermoplastic composite for type IV 70MPa high pressure vessel, *18th International Conference on Composite Materials*, August 2011.

- [16] Xu, P.; Zheng, J.-Y.; Liu, P.-F.: Finite element analysis of composite hydrogen storage vessel, *Materials and Design*, 31(7), 2009, 2295–301. <https://doi.org/10.1016/j.matdes.2009.03.006>
- [17] Yamashita, A.; Kondo, M.; Goto, S.; Ogami, N.: Development of high-pressure hydrogen storage system for the Toyota “Mirai”, *SAE Technical Paper*, 01, 2015, 1169-1181. <https://doi.org/10.4271/2015-01-1169>
- [18] 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(1), 2012, 1048-1057. <https://doi.org/10.1016/j.ijhydene.2011.02.125>