27th ABCM International Congress of Mechanical Engineering
December 04-08, 2023, Florianópolis-SC, Brazil

COB-2023-0198
ANALYSIS OF MESH TYPES APPLIED TO THE NUMERICAL MODELING OF A COMPOSITE OVERWRAPPED PRESSURE VESSEL

Lucas L. Agne
Estevão L. Cunha
Grupo de Mecânica de Materiais e Estruturas (GMEC), Universidade federal de Santa Maria - Campus Cachoeira do Sul
lucas.l.agne@gmail.com
estevao.slc@gmail.com

Henrique R. Mença
Sandro C. Amico
Universidade federal do Rio Grande do Sul. Porto Alegre
henriquemenca@gmail.com
amico@ufrgs.br

Maikson L. P. Tonatto
Grupo de Mecânica de Materiais e Estruturas (GMEC), Universidade federal de Santa Maria - Campus Cachoeira do Sul
maikson.tonatto@ufsm.br

Abstract. *Composite overwrapped pressure vessels (COPV) are lightweight structures primarily constituted by a liner and composite overwrap. These COPV may be used for storing compressed natural gas (CNG) for onboard applications, typically working in up to 20 MPa of pressure. Improper designing of cylinders can lead to catastrophic accidents, and numerical simulation is used as a tool to minimize such issues and contribute to a better understanding of the mechanical response of these structures under internal pressure. This work investigates the burst pressure of COPVs using finite element analysis (FEA) for models built using four different mesh types: axisymmetric continuum, axisymmetric shell, 3D continuum e 3D shell with a quarter of symmetry. The Abaqus software was used to simulate these models using boundary conditions and loadings related to a CNG application. The WoundSim plug-in was used to build the composite overwrap. The analyses were performed for the operating pressure of the COPV using the Tsai-Wu, Tsai-Hill and maximum stress failure criteria. The obtained results are discussed in terms of simulation time and computational cost. Close values were obtained between the different models, and the axisymmetric model, that uses fewer elements, was validated, allowing a greater number of increments in the simulations.*

Keywords: *pressure vessel, composite overwrapped, numerical simulation, CNG application, mesh types.*

1. INTRODUCTION

Pressure vessels are designed to store pressurized gases, used in various engineering applications. Divided into categories according to their arrangement and materials used in the manufacture: Type I - manufactured exclusively in metallic material, usually steel due to the need for high mechanical strength; type II – addition of a composite material reinforcement in the cylindrical region of the tank; type III - the internal liner consists of aluminum and externally wound completely in composite; type IV has a non-metallic liner, polyethylene or polyamide polymer, and reinforced by composite like type III; and type V that has only the composite without liner, still with few researches developed (Regassa *et al.*, 2022; Azeem *et al.*, 2022).

The composite overwrapped pressure vessels (COPV), have outer layers of composite with reinforcement of continuous fibers and the matrix. The composite is arranged on the outer surface of the liner and wrapped around its geometry through the process of filament winding, in order to create a stacking sequence of layers of material with a certain fiber orientation for each layer (Peters, 2011). The composite material provides this equipment with some advantages, such as low weight and high strength, as well as the possibility of optimizing the structure, improving material economy and efficiency in its applications (Kaw, 2005). This device consists of some essential components for its operation, the boss is responsible for making the connection with the fluid passage control valve, the liner is essentially responsible for the tightness of the equipment, preserving against fluid leakage, while the composite layer comprising the external region, being dedicated exclusively to the mechanical strength of the equipment.

The global energy crisis and problems related to environmental pollution forced the development of new technologies that could meet the economic and ecological demand of today (Hu *et al.*, 2021a). Natural gas presents interesting characteristics that qualify it as an alternative to combat the problems of pollution, considerably fewer toxic emissions, ecological, economic and abundant. Used in its compressed form in pressure vessels, known as compressed natural gas (CNG), operating at pressures of 20 to 25 MPa to compensate for its low energy density. However, high-pressure storage is associated with several accidents related to bursts or leaks, so CNG cylinders for onboard vehicle applications only come into operation after passing a strict safety standard dictated by ISO 11439 (Sharma *et al.*, 2021).

In the standard there is a section that concerns the requirements for fully wrapped composite cylinders of type IV, focusing on the requirement of safety standards to be followed. The COPV project must carry out experimental tests described by it to prove the adequacy of the model to the safety requirements. One of the most important is the burst pressure (P_B), which refers to the maximum internal pressure supported by the pressure vessel until it bursts. According to the standard, burst pressure is correlated with work pressure (P_W) through a pressure ratio, which depends on the material chosen for the composite. For carbon fibers this value is equal to 47 MPa, while the test pressure used should be 1,5 times the work pressure, obtained 30 MPa.

Composites are orthotropic materials, making the design of type IV cylinders a task with greater complexity compared to other pressure vessels. Therefore, it is necessary to use numerical simulation to assist in the design of COPV minimizing sizing problems, and contributing to a better understanding of its mechanical behavior. The stacking sequence of the layers directly impacts aspects such as stress and deformation, as well as affecting their burst pressure.

Therefore, the objective of the study is to analyze the burst pressure obtained for COPV's using finite element analysis (FEA) for models built using four different mesh types: axisymmetric continuum, axisymmetric shell, 3D continuum and 3D shell with a quarter of symmetry. In this way, it is possible to identify the best mesh to be used for the generation of the layup in the design of the burst pressure of type IV pressure vessel in CNG application.

2. METHODOLOGY

One of the first steps in modeling the COPV model is defining the type of mesh that will be used. The WoundSim plug-in user manual presents the four main mesh alternatives: Axisymmetric continuum; 3D continuum; Axisymmetric shell; and 3D Shell. The four models will be used in the present work for the same working condition (S Vertical, 2023). The initial step of the simulation implementation process corresponds to the creation of the liner model in Abaqus, being created two models, one axisymmetric 2D and the other 3D with a quarter of symmetry.

The 2D liner model is used in conjunction with the axisymmetric continuum composite mesh. While the 3D liner model is used for 3D continuum. The shell meshes were not used in conjunction with the liner due to a convergence problem in the simulation due to the contact employed between the two parts. The liner material corresponds to polyamide 6, its properties were taken from the work developed by Hu *et al.* (2021b), the values can be observed in Table 1.

Table 1. Properties of material applied in liner (Hu *et al.*, 2021b)

Young's modulus, $\mathbf{E}$	1880 MPa
Poisson's ratio, ν	0,4
Yield stress, $\mathbf{Y_s}$	47 MPa

The finite elements mesh applied in the two liner models can be checked in Table. 2. The 2D dimension mesh uses quadrilateral linear type elements, while the 3D dimension uses linear hexahedral elements. Due to the higher number of elements in the 3D liner, it is expected that the computational cost associated with simulating the 3D continuum model will be higher.

Table 2. Mesh used for the liner

Dimension	Number of elements
2D	502 elements of type CAX4R
3D	23688 elements of type C3D8R

Composite material for COPV typically uses carbon fiber as a way to reach commercial implementations where a fiber with high strength, low cost and low elongation at break is preferred (Sharma *et al.*, 2021). The properties of the composite material used in its work correspond to carbon fiber with epoxy resin Table. 3. The values refer to the work done by Sharma *et al.* (2021).

The trajectory traveled by the filament can be defined in two ways, geodesic or non-geodesic. The geodesic path represents the shortest trajectory through the surface of the mandrel non-slipping, being a helical path of constant step, represented by the winding angle. The path that deviates from the geodesic path is called non-geodesic. Slippage on the non-geodesic path does not often occur due to the principle of friction. The Clairault condition Eq. 1 can by using to determine the smallest possible winding angle based on the geometry of the liner, according to (Sharma *et al.*, 2022) , the

Table 3. Composite material property for model creation (Sharma *et al.*, 2021)

Elastic modulus in 1 - direction, $\mathbf{E_1}$	132,95 GPa
Elastic modulus in 2 - direction, $\mathbf{E_2}$	8,4 GPa
Elastic modulus in 3 - direction, $\mathbf{E_3}$	8,4 GPa
Poisson's ratio in plane 12, $\nu_{\mathbf{12}}$	0,34
Poisson's ratio in plane 13, $\nu_{\mathbf{13}}$	0,34
Poisson's ratio in plane 23, $\nu_{\mathbf{23}}$	0,41
Shear modulus in plane 12, $\mathbf{G_{12}}$	3,85 GPa
Shear modulus in plane 13, $\mathbf{G_{13}}$	3,85 GPa
Shear modulus in plane 23, $\mathbf{G_{23}}$	2,97 GPa
Longitudinal tensile strength, $\mathbf{X_t}$	1860 MPa
Longitudinal compressive strength, $\mathbf{X_c}$	1470 MPa
Transverse tensile strength, $\mathbf{Y_t}$	76 MPa
Transverse compressive strength, $\mathbf{Y_c}$	85 MPa
Shear strength in plane 12, $\mathbf{S_{12}}$	98 MPa

equation is given by:

$$Rsin\left(\alpha\right) = constant \tag{1}$$

where R is the radius and α is the winding angle as a function of the radius so that there is no slippage, the above ratio is equal to a constant, therefore does not vary along the coordinates describing the contour of the mandrel. Considering the radii of the liner of the present work with 160 mm in the region of the cylinder, and 23 mm in the boss, the values obtained from the angle between these points are plotted in the graph of Figure 1. It is found that the smallest angle obtained for 23 mm is around 10^o, being considered approximately 12^o for safety.

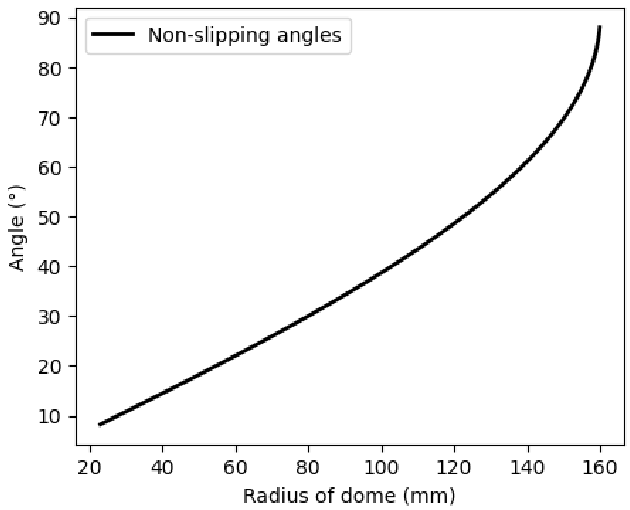


Figure 1. Angle calculate from Clairault equation

The design of COPV has several variables, so there is no general rule for the exact prediction of the stacking sequence that should be used, so Netting Analysis is used in many cases. Being a sizing method composed of equations to obtain an initial estimate of the thickness of the composite in the pressure vessel. The equations of this theory assume that the fibers provide all the stiffness and strength longitudinally, the assumption while conservative is a good basis for the fast design of pressure vessels in configurations where the liner does not share a significant load with the composite material. The analysis defines the required fiber thickness in the cylindrical region of the pressure vessel in the hoop, helical or polar directions, with this information the designer can define other important parameters for the design of the composite, such as number of layers or bandwidth (Peters, 2011).

The initial thickness estimate used for the hoop layers is calculated using the Eq. 2 given by:

$$t_{hoop} = \frac{PR\left(1 - \frac{tan(\alpha)^2}{2}\right)}{\sigma_H} \tag{2}$$

where P is the applied pressure, being considered in this case the burst pressure of 47 MPa, R is the radius of the cylinder, σ_H hoop tensile strength. Being considered equal to the longitudinal tensile strength of the composite, with a depreciation of 30% considering the detrimental effects of unexpected loads, adverse climatic and hydrothermal conditions, according to Sharma *et al.* (2021). The calculated value of the thickness is equal to 5,65 mm, rounding to a higher integer value is obtained 6 mm. This value divided by the layer thickness of 0,5 mm is obtained 12 hoop layers. Considering that the number of helical layers is equal to the number of hoop layers, a total composite thickness in the cylindrical region of 18 mm is obtained, since the helical layers have twice the hoop layer thickness. The layers used in the stacking sequence can be checked in the Table 4.

Table 4. Design layers of composite pressure vessel

Parameter	**Number of layers corresponding to the winding angle**									
Winding angle	12^o	18^o	24^o	30^o	36^o	42^o	48^o	54^o	60^o	89^o
Number of layers	4	1	1	1	1	1	1	1	1	12

The formulation of layer thickness as a function of the radius Eq. (3) used in WoundSim corresponds to the volume conservation method, given by the equation:

$$t\left(r\right) = \frac{t_{tl} \cdot cos\left(\theta_{tl}\right)}{cos\left(\theta_{r}\right)} \tag{3}$$

where t_{tl} is the thickness on the tangential line, θ_{tl} is the winding angle on the tangential line, and θ_r is the winding angle calculated for the defined radius. The modeling of the helical layers considered a geodetic path, being automatically calculated by the plug-in WoundSim the end position of the layer. While for the hoop layers the position had to be manually set, being defined as the final position of the cylindrical region.

Subsequently, the model of the generated stacking sequence is exported to Abaqus and incorporated into the liner through the tie contact for the continuum meshes. Shell meshes were not evaluated together with the liner. The finite element mesh refers to the composite material can be checked in the Tab. 5 for the four models. In shell models the number of elements is significantly lower than their counterparts of the same dimension, because there are less elements in the direction of thickness in this type of mesh.

Table 5. Mesh used for the composite layer

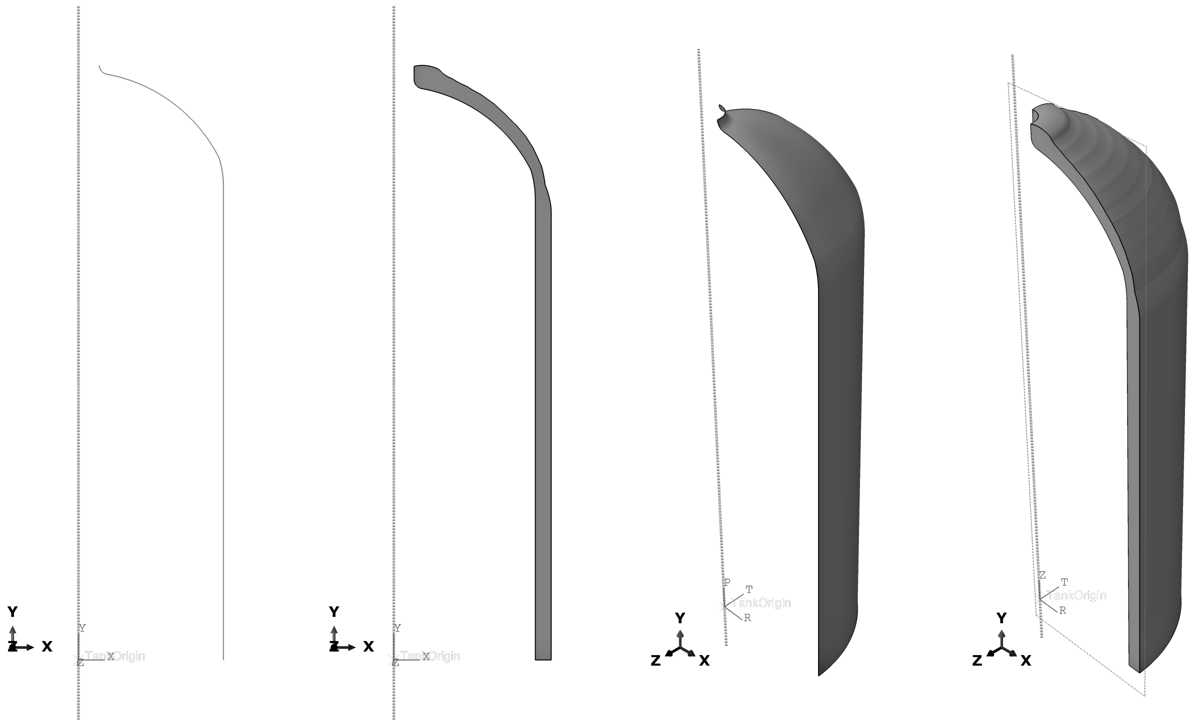
Mesh	**Number of elements**
Axisymmetric shell	150 elements of type SAX1
Axisymmetric continuum	8319 elements of type CAX8R and 53 of CAX6
3D shell	4500 elements of type S4R
3D continuum	444900 elements of type C3D20R and 3500 of C3D15

In Fig. 2a the stacking sequence geometry generated for the axisymmetric shell mesh is presented, and only one line in a 2D plane is observed. The axisymmetric continuum model is plotted in Fig. 2b and consists of elements in the direction of thickness in a 2D plane. In Fig. 2c the generated geometry corresponds to the 3D shell mesh with only one node in the direction of thickness in a 3D plane. Finally, the 3D continuum in Fig. 2d showing elements in the thickness direction and the 90^o revolution counterclockwise the Y-axis.

In 2D models the boundary condition employed corresponds only to the symmetry with respect to the Y-axis at the bottom edge of the meshes of Fig. 2a and Fig. 2b. While for the 3D models two more boundary conditions were added, of symmetry with X axis and with Z axis. The load corresponds to the pressure of 30 MPa, being the test pressure of the prototype. This pressure is applied on internal surface of liner for the continuum meshes. Taking into account that in type IV pressure vessels the liner does not share the load with the composite, an approach applying the load directly to the layup is feasible, and this method was adopted in the shell meshes, in order to solve the problem of convergence in this case. To calculate the burst pressure of the models, an extrapolation is performed based on the failure rate obtained at the test pressure.

3. RESULTS AND DISCUSSION

The first results refer to the burst pressure obtained for the meshes tested in the dome region. In Fig. 3 the failure values are observed using Tsai-Wu, Tsai-Hill and maximum stress failure criteria. Comparing the two models with continuum elements, it is observed that the largest error obtained is in the Tsai-Wu criterion with a 3% smaller burst in the 3D model. For the shell models, the greatest difference obtained between them was 44% using the maximum stress criterion, for the other criteria this difference is no more than 1%. Between the two 2D models the biggest difference is obtained with



(a) (b) (c) (d)

Figure 2. Geometry of mesh (a) axisymmetric shell, (b) axisymmetric continuum, (c) 3D shell and (d) 3D continuum.

Tsai-Wu with 37% smaller with the shell mesh. Comparing the 3D models the biggest difference is found in the maximum stress criterion with 36%.

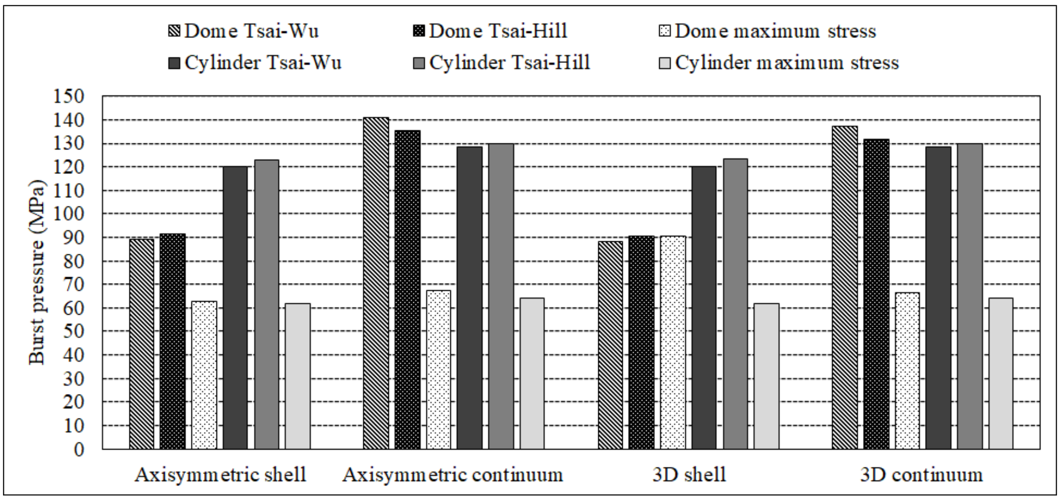


Figure 3. Burst pressure in dome and cylinder region for mesh types using the Tsai-Wu, Tsai-Hill and maximum stress failure criterion.

The figure shows the values obtained for burst pressure in the cylindrical region considering the three previous criteria. The results obtained in meshes of the same type of element with different dimensions are the same. Comparing the axisymmetric continuum and shell meshes, the biggest difference obtained is 7% found with the Tsai-Wu criterion. For the 3D meshes the largest calculated difference is 6% also obtained in Tsai-Wu.

The simulation time can be observed in Tab. 6, and it was verified that the simulation time is consistent with the number of elements used in the model, the equipment used for numerical modeling and simulation is a Workstation with 12 processing cores and 96 GB of RAM. With the exception of the axisymmetric continuum model, which has the shortest analysis time despite having a greater number of elements than the shell models. The continuum 3D model obtained the longest simulation time with 2 hours of analysis. A direct comparison between the obtained values and simulation time can be made between the two continuum meshes. The 3D model presents 2h of simulation and the 2D model a time of 20s, while the results obtained are approximately equal, taking into account the three failure criteria used.

Table 6. Simulation time with respect to the mesh used

Mesh	Simulatin time
Axisymmetric shell	00:01:05
Axisymmetric continuum	00:00:20
3D shell	00:01:21
3D continuum	02:01:23

4. CONCLUSION

The present work consisted of evaluating the influence of the type of mesh used in the layup modeling on the rupture pressure. For the tested meshes, what was already expected was found, the greater number of finite elements used in the model increase the simulation time. The axisymmetric continuum showed better results considering the simulation time and the amount of finite elements used.

The mesh influence on the burst pressure is more evident in the dome region, the shell meshes underestimate its value more. This behavior can be explained by the influence of the thickness in this region. Shell models are used when the thickness of the equipment is negligible in relation to its length. According to the formulation used in WoundSim, the foam in the dome region is greater than in the cylinder. In the cylinder region, the difference obtained for the burst pressure value between models is very small, and between meshes of the same type of elements it is negligible.

5. ACKNOWLEDGEMENTS

This work was funded by FAPERGS 02/2022- INOVA CLUSTERS TECNOLÓGICOS n^o 22/2551-0000839-9

6. REFERENCES

Azeem, M., Ya, H.H., Kumar, M., Stabla, P., Smolnicki, M., Gemi, L., Khan, R., Ahmed, T., Ma, Q., Sadique, M.R., Mokhtar, A.A. and Mustapha, M., 2022. "Application of filament winding technology in composite pressure vessels and challenges: A review". *Journal of Energy Storage*, Vol. 49. doi:10.1016/j.est.2021.103468.

Hu, Z., Chen, M.H., Zu, L., Jia, X., Shen, A., Yang, Q. and Xu, K., 2021a. "Investigation on failure behaviors of 70 mpa type iv carbon fiber overwound hydrogen storage vessels". *Composite Structures*, Vol. 259, pp. 1–11. doi: 10.1016/j.compstruct.2020.113387.

Hu, Z., Chen, M. and Pan, B., 2021b. "Simulation and burst validation of 70 mpa type iv hydrogen storage vessel with dome reinforcement". *International Journal of Hydrogen Energy*, Vol. 46, pp. 23779–23794. doi: 10.1016/j.ijhydene.2021.04.186.

Kaw, A., 2005. *Mechanics of Composite Materials*. CRC Press, New York.

Peters, S., 2011. *Composite Filament Winding*. ASM International, Ohio.

Regassa, Y., Gari, J. and Lemu, H.G., 2022. "Composite overwrapped pressure vessel design optimization using numerical method". *Journal of Composites Science*, Vol. 6. doi:10.3390/jcs6080229.

S Vertical, I.S.I., 2023. *WoundSim User Manual*. Paris Region. URL https://www.svertical.com/. Acesso em 06 jun. 2023.

Sharma, P., Chugh, P. and Neogi, S., 2021. "Study to methodize the design of a safe type-4 cng storage vessel using finite element analysis with experimental validation". *International Journal of Pressure Vessels and Piping*, Vol. 192, p. 104425. doi:10.1016/j.ijpvp.2021.104425.

Sharma, P., Sharma, S., Bera, T., Semwal, K., Badhe, R.M., Sharma, A., Kapur, G.S., Ramakumar, S.S. and Neogi, S., 2022. "Effects of dome shape on burst and weight performance of a type-3 composite pressure vessel for storage of compressed hydrogen". *Composite Structures*, Vol. 293, p. 115732. doi:10.1016/j.compstruct.2022.115732.

7. RESPONSIBILITY NOTICE

The authors are solely responsible for the printed material included in this paper.