The influence of different projection operators in the Virtual Element Method applied to bi-dimensional elastic problem

Paulo Akira Figuti Enabe[1], Rodrigo Provasi[1]

[1]*Dept. of Structural and Geotechnical Engineering, Polytechnic School, Universisty of São Paulo*
Avenida Professor de Almeida Prado tv. 2 n.83, 05508070, São Paulo, Brasil
paulo.enabe@usp.br, provasi@usp.br

Abstract. The Virtual Element Method (VEM) generalized the Finite Element Method (FEM) in terms of mesh discretization since any simple polygon can be used as elements in the mesh. By working with more general polygons, the shape functions are not always low-order polynomials, meaning that the VEM must compute these functions implicitly by using projection operators, which responsible to map functions from the virtual element space to polynomial spaces. The choice of different projection operators requires some adaptation of the VEM formulation but the general implementation pipeline still the same. In this way, this work shows those adjustments in the formulation and proposes to investigate how the choice of the operators influences the method convergence. For that, two numerical models are presented: a patch test in a unitary square and a static analysis regarding a zeta-profile (commonly used in pressure armors of risers). All the formulation is restricted to the Virtual Element Method linear two-dimensional case taking in consideration the Theory of Elasticity.

Keywords: Virtual Element Method, Finite Element Method, Pressure Armor, Projection Operators

1 Introduction

The Virtual Element Method (VEM) aims to generalize the classical Finite Element Method regarding mesh discretization. With the VEM, the mesh elements can be any simple polygon, i.e., convex or not polygon (a more precise definition is given in the next section). More generally, in the three-dimensional case it is possible to work with any polyhedral. As consequence of working with a variety of elements, the shape functions are not necessarily low order polynomials. In this way, these functions are computed implicitly by using a projection operator that maps the virtual element space (an analogous object to the finite element space) to a polynomial space.

The method was originally introduced in the works of da Veiga et al. [1] and da Veiga et al. [2]. The first paper presents the formulation for the Poisson Equation highlighting the mathematical aspects and without any engineering application. In the second work, the method is applied to the linear elasticity theory by focusing on the mathematical features on the formulation. An implementation framework for the Virtual Element Method applied to the Poisson Equation is presented on da Veiga et al. [3]. A implementation of the method for the Poisson case is presented on Sutton [4], where the author proposes a MATLAB implementation using few lines of code.

For the differential equations associated to the linear elasticity of solids, the work of Gain et al. [5] presents a matrix formulation and an implementation guide for the linear case to three-dimensional geometries. This work is particularized to the two-dimensional case in Ortiz-Bernardin et al. [6], where a implementation in C++ of the method is presented and the results on patch tests are discussed. In Artioli et al. [7], a more general matrix formulation of the Virtual Element Method regarding linear elasticity is presented. Thus, the model is not restricted to the linear case and higher order VEM is formulated. A engineering perspective of the method is presented on Mengolini et al. [8]. In this work, a matrix framework is proposed and the results of the VEM are compared with Finite Element Method (FEM).

The present work aims to present the formulation of the Virtual Element Method for two different types of projection operators. The first operator is used in Artioli et al. [7] and it project functions from the virtual element space to the strain space. The second operator maps functions from the deformation space to the rigid body motion and constant strain modes spaces as proposed by Gain et al. [5] and Ortiz-Bernardin et al. [6]. In this way, a comparison between these two approaches are presented considering the Theory of Elasticity in two dimensions.

2 The Virtual Element Method

In this section a general formulation pipeline for the Virtual Element Method applied to the context of bi-dimensional elasticity is describe. The formulation presented is mostly inspired on the work of da Veiga et al. [2], Gain et al. [5], Artioli et al. [7] and Ortiz-Bernardin et al. [6]. On those works, the formulation is restricted to the linear case, i.e., the polynomial degree is equal to one. The method was originally formulated to the Poisson Equation as can be verified on da Veiga et al. [1] and da Veiga et al. [3]. The general pipeline for the Virtual Element Method is described as follows:

- **Introduction of the weak formulation:** as in the Finite Element Method, the starting point to apply the Virtual Element Method is the weak formulation, i.e., the integral form of the differential equation. To achieve the weak form, a bilinear form and a linear functional (load terms) must be defined. Also, the test functions are introduced and the function space is the closure of C_c^∞ in the Sobolev space denoted by H_0^1. It is important to mention that the virtual element space is contained in the closure. In the linear elasticity context, the test functions are called virtual displacement field and the weak form is called Principle of Virtual Works.
- **Discretization of the weak form:** the discretization is made by choosing a decomposition τ_h of simple polygons. Before determining the discrete bilinear form and the discrete load terms, it is required to prove that the discrete form has solution and define the convergence criteria. Also, stability and consistency hypothesis are required to be satisfied.
- **Construction of the virtual element space:** as mentioned before, the virtual element space is a subspace of the closure. In this sense, it is possible to define local virtual element spaces and from then construct the global space. These spaces are constructed upon the definition of the degrees of freedom. The canonical choice for the degrees of freedom are the values of the functions in the vertices of each polygon, the values in each edge of each polygon and the values in the internal points (moments). In the linear case ($k = 1$), only the values on the vertices are required. It is important to mention that the choice of degrees of freedom is the same for both the Poisson Equation and for the linear elasticity context. Also, to construct the the local virtual element space, a linear space $\mathcal{E}_K$ is necessary. This space is associated to the linear polynomials and to the behavior of functions in the edges of each polygon.
- **Introduction of the projection operator:** the projection operator is responsible to compute the functions implicitly. The proposition is to project the shape functions from the local virtual element space (which are unknown) to a subspace of a polynomial space. In the case of the Poisson Equation the projections are made directly into the polynomial spaces.
- **Construction of the bilinear form:** the bilinear form must be constructed in order to satisfy both the consistency and stability criteria. To ensure stability, a symmetric bilinear term $S_{h,K}$ is introduced. There are different ways to define $S_{h,K}$ that can be verified in da Veiga et al. [1], Gain et al. [5], Wriggers et al. [9], da Veiga et al. [10] and Artioli et al. [7]. The stability term is responsible to handle the residue of the projection. Finally, the bilinear term can be computed directly from the degrees of freedom.
- **Construction of the load term:** analogous to the bilinear form, a projection operator is defined and the load terms are computed directly from the degrees of freedom.
- **Construction of the stiffness matrix and load vectors:** to build the matrix form of the bilinear form, one should choose an adequate basis for the virtual element space (e.g. Lagrange polynomials) and an adequate basis for the polynomial spaces and its subspaces. After the choice is made, by applying the definition of the bilinear term and the load terms within the basis, one should obtain, respectively, the stiffness matrix and the load vectors. Then, it is possible to assembly the global problem and solve it as in FEM.

In the case of linear elasticity context, there are different approaches for defining the projection operator. In da Veiga et al. [2] and Mengolini et al. [8] the projections are done similarly to the Poisson Equation. But in Artioli et al. [7] the projections are made directly into the strain field. In Gain et al. [5] and Ortiz-Bernardin et al. [6] the projection operator is divided in three parts, the first term is related to the rigid body motions, the second term is related to the constant strain modes and the third term is responsible to extract the polynomial elements. It is worth mentioning the rigid body motions and the constant strain modes are defined as subspaces of polynomial spaces. Here in this work, the implementation of two of those projection operators are used for the numerical simulations of the zeta-profile of a pressure armor.

3 Numerical results

In this section two different applications of the Virtual Element Method are presented. The first one consists of a patch test similar to the one showed in Artioli et al. [8]. The goal is to investigate based on a problem with analytical solution how the different projectors might influence in the final results. The second application refers to a zeta-profile, commonly used in pressure armors of risers. For further information about pressure armors and risers, one can refer to Neto et al. [11], Rahmati et al. [12], de Sousa et al. [13], Santos et al. [14] and Lu et al. [15]. A simplified model is presented and the results obtained are compered with an overkill model developed in the commercial software *Ansys*. Again, it is investigated how the projection operator influences on the convergence of the method.

To simplify notation, the model *A* denotes the method implementation using the projection operator proposed by Artioli et al. [7] and model *B* refers to the implementation using the projection operator proposed by Ortiz-Bernardin et al. [6].

3.1 Patch test

For this patch test an unitary square was considered with the movement restricted in two edges and a distributed load in a third one. Figure 1 shows the used geometry. The used elasticity module is $E = 1MPa$, the Poisson coefficient is $\nu = 0.3$ and the distributed load is $t = 1kN/m$.

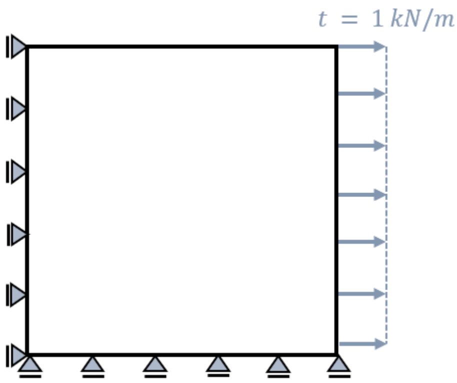

Figure 1. Unitary square with distributed load.

The analytical solution for this problem can be found in Artioli et al. [7] and the error is calculated using the L^2 norm in the difference between the analytical and numerical displacement values. A mesh using 25 quadrilateral elements is used within the VEM. Table 1 shows the errors for different element sizes.

Table 1. Errors in the patch test for different projectors.

Element sizes	Error A	Error B
0.08	2,9771E-14	0,10055574
0.04	1,8400E-14	0,10040522
0.03	4,1141E-14	0,10073508
0.02	2,0932E-13	0,10093803
0.01	1,4028E-14	0,10027786

It is interest to observe that the model *A* has a much lower associated error when compared to model *B*. This might have occur due to the less approximations in model *A* for the projection operator construction, since in this case the operator maps the virtual element space to the polynomial space.

3.2 Zeta pressure armor model

In this work, a simplified model of a zeta-profile inspired on the work referred in Mendonça [16] was considered. The aim here is to analyze the influence of the projection operator choice on the convergence of the VEM. For more information about pressure armors one shall refer to Mendonça [16], Vantadori et al. [17] and Pang et al. [18].A two dimensional model of the profile is constructed with the movement restricted on the left and right edges. A distributed external traction g is applied on the top of the geometry simulating the crushing load related to the installation process. This load occurs due to the action of the shoes that provides sustaining during installation. It is important to mention that it is considered the plane strain state hypothesis for this model. Figure 2 shows the considered geometry.

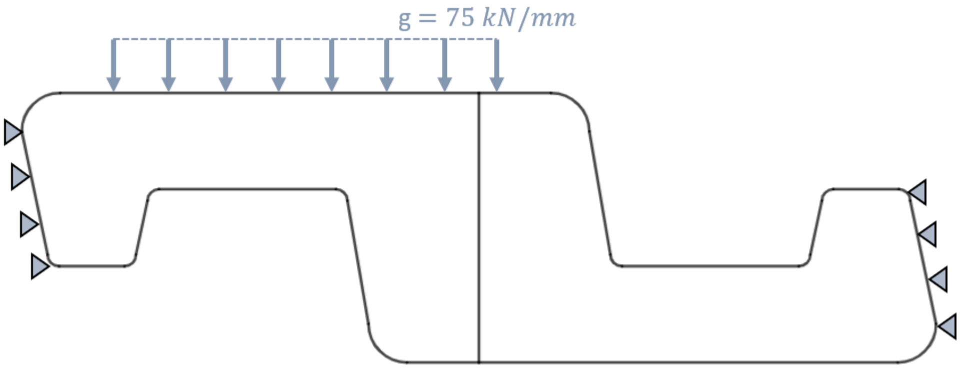

Figure 2. Adopted bi-dimensional model for the zeta-profile adapted from Mendonça [16].

Regarding the material parameters used, the elastic module is equal to $E = 207GPa$ and the Poisson coefficient is equal to $\nu = 0.3$. The mesh used with the virtual element method considers only quadrilateral elements and was generated using *Gmsh*, an open-source software for mesh generation (see Geuzaine and Remacle [19]). Figure 3 shows the geometry modeled in *Gmsh* and the quadrilateral mesh.

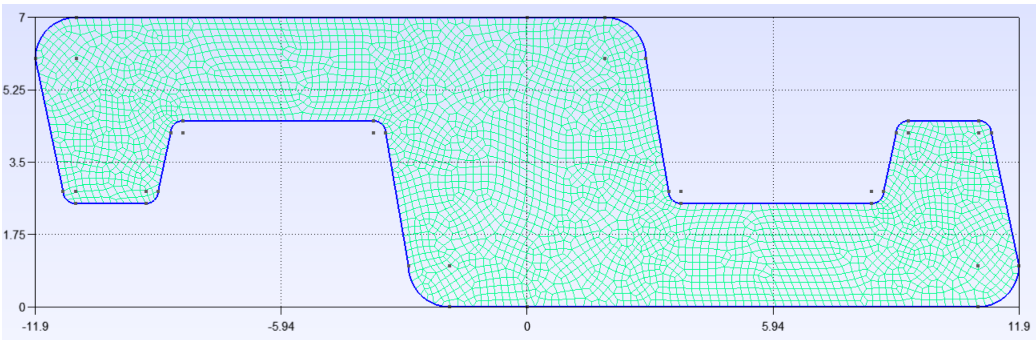

Figure 3. Zeta profile geometry and quadrilateral mesh in Gmsh.

The same problem was modeled using *Ansys*. Since the comparisons here are about the results and not performance, one must ensure that convergence was achieved and, thus, the a very refined mesh was used. This model is considered the reference solution and is compared with the Virtual Element Results. The two dimensional model in *Ansys* used a mesh with square elements and 120227 nodes. The maximum values of the horizontal displacement u_h and the vertical displacement v_h are chosen to compare the models.

Table 2 shows the deviation between the reference solution and the VEM implementations for different number of nodes. Figures 4 and 5 show the information in form of graphics. Using almost one sixth of the number of the nodes in reference model, the model *A* presented more consistent results than model *B*. Even model *B* presenting a deviation of almost 1% for u_h, it present a deviation near to 18% for v_h. On the other hand, model *A* presented a bigger deviation for u_h of almost 8%, it presented almost the half of the deviation for v_h when compared to model *B*. This results are significant once the Virtual Element Models are using near to one sixth of the number of nodes used in the *Ansys* model.

It is possible to observe that the convergence rate is not exactly the one expected for the linear formulation of VEM. The most probable explanation is the choice of the stabilization term. Since this terms handles the non-polynomial components, an imprecise choice of the stabilization term shall lead to a large associated residue generated by the projection of these components. Also, to improve the results, a different choice for the basis to the polynomial space shall present significant results once it can lead to a more precise representation of the real structural behavior.

Table 2. Deviation regarding zeta-shaped profile for Ansys reference solution and Virtual Element Method implementations.

Number of Nodes	**A-**$\max(\lvert u_h\rvert)$	**A-**$\max(\lvert v_h\rvert)$	**B-**$\max(\lvert u_h\rvert)$	**B-**$\max(\lvert v_h\rvert)$
235	32,6557	42,6249	16,1498	30,0267
475	14,0828	28,5505	7,9258	23,8121
897	3,7372	20,4186	5,3701	21,5844
1321	0,9612	16,2893	3,8917	20,2701
2365	3,5909	14,0465	2,5397	19,0502
4237	5,5328	12,4622	2,0789	18,6624
15577	7,6550	10,5314	1,2719	17,8758
19314	7,9270	10,2809	1,1831	17,7887

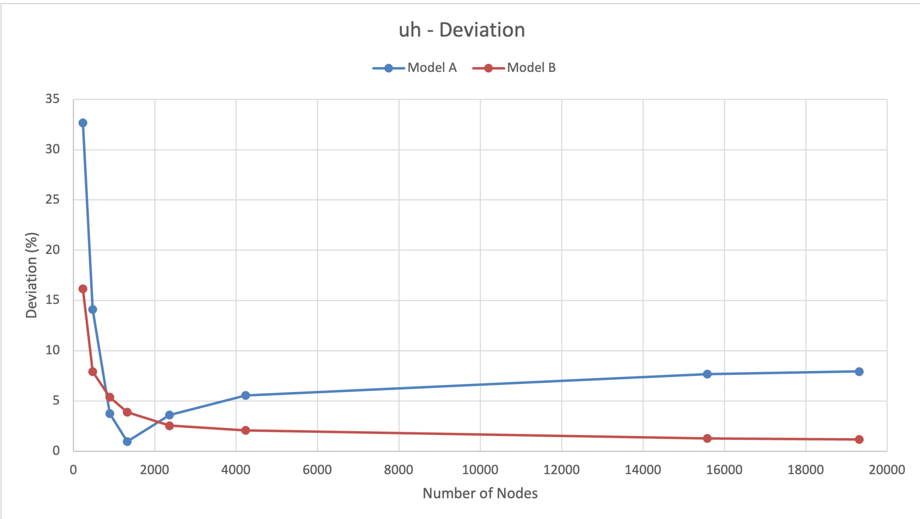

Figure 4. Deviation between the reference model and the Virtual Element Method regarding u_h.

4 Conclusion

The idea in this work is to investigate how the definition of different projection operators impacts in the convergence of the Virtual Element Method. For that an overview of the method constructions and its main characteristics are discussed. The VEM proposal is to generalize the classical Finite Element Method in terms of meshing. Any simple polygon can be used in the mesh and, as consequence, the shape functions are not always low order polynomials. In this sense, the VEM computes those functions implicitly using the projection operators.

It becames clear the methodology to formulate the Virtual Element Method. The first part consists of defining the virtual element space take in consideration polynomial and Sobolev spaces. Then, to build the discrete bilinear

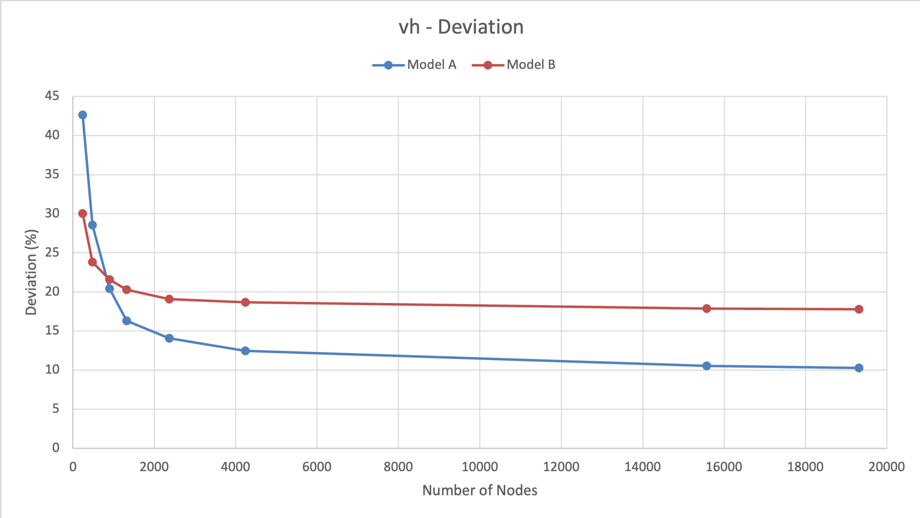

Figure 5. Deviation between the reference model and the Virtual Element Method regarding v_h.

form, the projection operators are introduced. These operators are responsible to project functions from the virtual element space to a known space (e.g., polynomial). In the bilinear form it is important to consider that it must satisfy both consistency and stability simultaneously. The last part is related to build the load vector using a projection operator defined in L^2 Lesbegue space.

Two different projectors operators are considered to formulate the Virtual Element Method. The first one refers to the work of Artioli et al. [7] that maps the virtual element space directly to the strain field. The second one concerns to the work of Ortiz-Bernardin et al. [6] and divide the projection operator in three related to the space of rigid body motion, space of constant strain modes and polynomial space. The first formulation is denoted here by model *A* and the second one by model *B*.

The first numerical application regards to a patch test in an unitary square. Here, different element sizes were tested for both models. It is possible to observe that model *A* presented a much lower error when compared to model *B*. This might be associated to the number of approximations that are made in the definition of the projection operators.

Using a quadrilateral mesh and considering a stress plane state problem, the second application of the VEM refers to a zeta-profile commonly used in pressure armors of risers. The implementations are compared with a reference solution using *Ansys*. It is possible to observe that the formulations presented different results. Model *A* presented a more consistent set of results than model *B*. The reason behind that should be the more approximations that is done when dividing the projection operator in three as in model *B*. And, in terms of convergence it is possible to conclude that the choice of stabilization term presented influence.

Acknowledgments. The authors acknowledge the Department of Structural and Geotechnical Engineering of the Escola Politécnica (University of São Paulo) for the research grant of the first author and Tiago Fernandes Moherdaui for all support during this work.

Authorship statement. The authors hereby confirm that they are the sole liable persons responsible for the authorship of this work, and that all material that has been herein included as part of the present paper is either the property (and authorship) of the authors, or has the permission of the owners to be included here.

References

[1] da L. B. Veiga, F. Brezzi, A. Cangiani, G. Manzini, L. D. Marini, and A. Russo. Basic principles of virtual element methods. *Mathematical Models and Methods in Applied Sciences*, vol. 23, n. 1, pp. 199–214, 2013a.

[2] da L. B. Veiga, F. Brezzi, and L. D. Marini. Virtual elements for linear elasticity problems. *SIAM Journal on Numerical Analysis*, vol. 51, n. 2, pp. 794–812, 2013b.

[3] da L. B. Veiga, F. Brezzi, L. D. Marini, and A. Russo. The hitchhiker's guide to the virtual element method. *Mathematical Models and Methods in Applied Sciences*, vol. 2, n. 8, pp. 1541–1573, 2014.

[4] O. Sutton. The virtual element method in 50 lines of matlab. *Numerical Algorithms*, 2016.

[5] A. L. Gain, C. Talischi, and G. H. Paulino. On the virtual element method for three-dimensional linear elasticity problems on arbitrary polyhedral meshes. *Computer Methods in Applied Mechanics and Engineering*, vol. 282, pp. 132–160, 2014.

[6] A. Ortiz-Bernardin, C. Alvarez, N. Hitschfeld-Kahler, A. Russo, R. Silva-Valenzuela, and E. Olate-Sanzana. Veamy: an extensible object-oriented c++ library for the virtual element method. *Numerical Algorithms*, 2019.

[7] E. Artioli, L. Beirão da Veiga, C. Lovadina, and E. Sacco. Arbitrary order 2D virtual elements for polygonal meshes: part I, elastic problem. *Computational Mechanics*, vol. 60, n. 3, pp. 355–377, 2017.

[8] M. Mengolini, M. F. Benedetto, and A. M. Aragón. An engineering perspective to the virtual element method and its interplay with the standard finite element method. *Computer Methods in Applied Mechanics and Engineering*, vol. 350, pp. 995 – 1023, 2019.

[9] P. Wriggers, W. T. Rust, and B. D. Reddy. A virtual element method for contact. *Computational Mechanics*, vol. 58, n. 6, pp. 1039–1050, 2016.

[10] da L. B. Veiga, C. Lovadina, and A. Russo. Stability analysis for the virtual element method. *Mathematical Models and Methods in Applied Sciences*, vol. 27, n. 13, pp. 2557–2594, 2017.

[11] A. G. Neto and C. d. A. Martins. Flexible pipes: Influence of the pressure armor in the wet collapse resistance. *Journal of Offshore Mechanics and Arctic Engineering*, vol. 136, n. 2014, 2014.

[12] M. T. Rahmati, S. Norouzi, H. Bahai, and G. Alfano. Experimental and numerical study of structural behavior of a flexible riser model. *Applied Ocean Research*, vol. 67, pp. 162–168, 2017.

[13] de J. R. M. Sousa, C. Magluta, N. Roitman, and G. C. Campello. On the extensional-torsional response of a flexible pipe with damaged tensile armor wires. *Ocean Engineering*, vol. 161, n. April, pp. 350–383, 2018.

[14] C. C. Santos and C. P. Pesce. An alternative strategy for offshore flexible pipes finite element analysis. *Marine Structures*, vol. 65, n. January, pp. 376–416, 2019.

[15] H. Lu, M. A. Vaz, and M. Caire. Alternative analytical and finite element models for unbonded flexible pipes under axisymmetric loads. *Ocean Engineering*, vol. 225, n. November 2020, pp. 108766, 2021.

[16] H. G. Mendonça. Análise paramétrica do efeito de cargas de esmagamento e estrangulamento sobre um tubo flexível na operação de lançamento, 2016.

[17] S. Vantadori, A. Carpinteri, and I. Iturrioz. Fretting failure of a pressure armour in an unbonded flexible riser. *International Journal of Fatigue*, vol. 128, n. June, pp. 105203, 2019.

[18] G. Pang, C. Chen, Y. Shen, and F. Liu. Comparison Between Different Finite Element Analyses of Unbonded Flexible Pipe via Different Modeling Patterns. *Journal of Shanghai Jiaotong University (Science)*, vol. 24, n. 3, pp. 357–363, 2019.

[19] C. Geuzaine and J.-F. Remacle. Gmsh: A 3-d finite element mesh generator with built-in pre- and post-processing facilities. *International Journal for Numerical Methods in Engineering*, vol. 79, pp. 1309 – 1331, 2009.