**PEMesh**: a Graphical Framework for the Analysis of the Interplay Between Geometry and PEM Solvers

Daniela Cabiddu\textsuperscript{a,\ast},\textsuperscript{1}, Giuseppe Patanè\textsuperscript{a,\ast}\textsuperscript{2} and Michela Spagnuolo\textsuperscript{a,\ast}\textsuperscript{3}

\textsuperscript{a}CNR-IMATI, Genova, Italy

**ARTICLE INFO**

Keywords: Computer-Aided Geometric Design, Graphical User Interface, Polytonal meshes, Partial Differential Equations, Polytopal Element Methods (PEM)

**ABSTRACT**

Partial differential equations can be solved on general polygonal and polyhedral meshes, through Polytopal Element Methods (PEMs). Unfortunately, the relation between geometry and analysis is still unknown and subject to ongoing research in order to identify weaker shape-regularity criteria under which PEMs can reliably work. We propose PEMesh, a graphical framework to support the analysis of the relation between the geometric properties of polygonal meshes and the numerical performances of PEM solvers. PEMesh allows the design of polygonal meshes that increasingly stress some geometric properties, by exploiting any external PEM solver, and supports the study of the correlation between the performances of such a solver and geometric properties of the input mesh. Furthermore, it is highly modular, customisable, easy to use, and provides the possibility to export analysis results both as numerical values and graphical plots. PEMesh has a potential practical impact on ongoing and future research activities related to PEM methods, polygonal mesh generation and processing.

**1. Introduction**

Over the last fifty years, computer-based simulation has dramatically increased its impact on research, design, and production, and is now an indispensable tool for development and innovation in science and technology. In particular, Partial Differential Equations (PDEs) offer a broad and flexible framework for modeling and analyzing a number of phenomena arising in fields as diverse as physics, engineering, biology, and medicine. Computer-based simulation of PDEs also relies on a suitable description of geometrical entities, such as the computational domain and its properties. The representation of geometric entities has been studied mainly in the field of geometric modeling, and often the requirements of shape design are different from those of numerical simulation.

In this context, Polytopal Element Methods (PEMs) allow solving differential equations on general polygonal and polyhedral meshes, thus offering a great freedom in the definition of mesh generation algorithms. Similarly to Finite Elements Methods (FEMs), the performance of PEMs (i.e., accuracy, stability, effectiveness of preconditioning) depends on the quality of the underlying mesh. Differently from FEMs, where the relation between the geometric properties of the mesh and the performances of the solver are well known [22, 6, 3], the definition of the quality of polytopal elements is still subject to ongoing research [11, 14, 2].

The proposed graphical framework is intended to support the analysis of the relation between the geometric properties of the mesh and the numerical performances of the solver, in terms of basis degree, conditioning number of the stiffness matrix, etc. To this end, our work covers several aspects, such as the design and generation of meshes to increasingly stress geometric properties, the study of the performances of PEM solvers, and the correlation between such performances and main geometric properties of the input meshes. Each step is performed by exploiting existing tools mainly coming from two related but independent research areas: geometric design and numerical methods for PEMs. Indeed, these tools rely on different representations of the same domain, and researchers are often required to be skilled programmers and expert tool users to allow such tools to be part of the same experimental pipeline.

**Figure 1**: Main window of PEMesh.

We introduce PEMesh (Fig. 1), as an open-source software tool designed to help researchers to perform experiments on the analysis and design of polygonal meshes for PEM solvers. PEMesh is an advanced graphical tool that seamlessly integrates geometric design pipelines and PEM simulations. It supports the design and generation of complex input polygonal
meshes by stressing geometric properties, while providing the possibility to solve PEMs on the generated meshes. Furthermore, PEMesh allows the user to correlate one or more geometric properties of the input polytopal mesh with the performances of PEM solvers, and to visualize the results though customisable and interactive plots.

PEMesh is highly modular and customisable. It allows researchers to simulate any PEM solver, by simply calling the PEM solver executable from an internal command line and providing possibly additional input parameters other than the geometric data set. To the best of our knowledge, PEMesh is the first graphical tool to generate complex discrete polytopal meshes and to support a study of the correlation between their geometric properties and numerical PEMs solvers. Indeed, it has a potential practical impact on research activities on this subject.

The paper is organised as follows. We briefly review previous work on PEMs and existing tools both to perform PEM simulations and to design geometric data sets (Sect. 2). Then, we describe the structure of PEMesh and its capabilities (Sect. 3), with technical details about its implementation (Sect. 4). Finally, we discuss some directions of future research (Sect. 5).

2. Background and related work

We briefly review previous work on numerical PEM solvers and libraries (Sect. 2.1) and meshing tools (Sect. 2.2).

2.1. Numerical PEM solvers and libraries

Main PEMs include Mimetic Finite Differences [27, 4], Discontinuous Galerkin-Finite Element Method (DG-FEM) [1, 5], Hybridisable and Hybrid High-Order Methods [7, 8], Weak Galerkin Method [28], BEM-based FEM [19], Poly-Spline FEM [20], and Polygonal FEM [23]. Main existing tools for the numerical solution of PDEs include (i) VEMLab [17], which is an open source MATLAB library for the virtual element method and (ii) Veamy, which is a free and open source C++ library that implements the virtual element method (C++ version of [17]). The current release of this library allows the solution of 2D linear elasto-static problems and the 2D Poisson problem [18]. Other libraries are (iii) the 50-lines MATLAB implementation of the lowest order virtual element method for the two-dimensional Poisson problem on general polygonal meshes [24], and (iv) the MATLAB implementation of the lowest order Virtual Element Method (VEM) [13].

As a matter of example, we demonstrate how a PEM solver can be integrated in PEMesh. Our use case exploits the Virtual Element Method (VEM) [26], which can be considered as an extension to FEM for handling general polytopal meshes.

2.2. Mesh generation tools

Nowadays, meshes are commonplace in a number of applications ranging from engineering to bio-medicine and geology. Depending on the application field, automatic mesh generation may be a difficult task, due to specific geometric requirements to be satisfied.

With reference to simulation with FEMs, the principle behind meshing algorithms in commercial FEM solvers are described in [15], and an open-source tool is provided. Free-FEM [10] is a popular 2D and 3D partial differential equations (PDE) solver used by thousands of researchers across the world, including its own mesh generation module. Although it provides plenty of functionalities, it is based on its own language and it has no graphical interface. Similarly, the MATLAB® suite provides its own mesh generator [16]. Both solutions focus on FEM requirements, and enable the possibility to generate triangle meshes, but they do not allow the generation of generic polygon meshes.

Concerning polygonal meshes, available Voronoi-based meshing tools (e.g. [9, 25]) are not suited for our study, because they produce convex elements that are not challenging enough to stress PEM solvers. For the best of our knowledge, the benchmark proposed in [2] is the only one providing a polygonal mesh generation approach specifically designed for PDE solvers. Unfortunately, the proposed approach is not easily customisable, and allows the generation of polygon meshes having a single non-triangle element.

PEMesh provides an advanced mesh generation module which enables the creation of generic polygonal meshes specifically designed for PDE solvers.

3. Proposed framework

PEMesh is aimed at evaluating the dependence of the performances of a PEM solver on geometrical properties of the input polygonal mesh, which is either generated by using PEMesh itself or provided as an external resource. PEMesh is mainly composed by four modules, each of them is provided as a specialized window.

Polygon mesh generation & loading allows the user to load one or more existing meshes or to generate a new ones from scratch by either exploiting a set of provided polytopal elements or providing an external one. The generation of new meshes is highly customisable, and the user is allowed to play with a large set of options and parameters (Sect. 3.1).

Geometric analysis allows the user to perform a deep analysis of geometric properties of the input polygonal meshes and to correlate each of them with the others. Results of such an analysis are shown through advanced plots (Sect. 3.2).

PEM solver allows the user to run a PEM solver and to analyse its performances on input polygonal meshes. Any PEM solver may be exploited, as long as it can be run from command line and provides its output according to a specific textual format. Both the solution and the ground-through of the PEM is shown directly on the meshes, while the performances of the solver are visualized through linear plots (Sect. 3.3).
**Correlation visualization** supports the analysis of the correlation between geometric properties of the polygonal meshes and numerical performances of the selected PEM solver. Results are made available in the form of customizable scatter plots (Sect. 3.4).

PEMesh provides the possibility to show the results of each analysis step on the display, to customize visualization aspects of plots (i.e., color, font sizes, etc.) and to interactively analyze them by clicking on visualized points and lines to show data values in the selected point. Furthermore, such results can be saved on disk as images and as textual files, to be possibly re-used by other applications.

### 3.1. Polygonal mesh generation & loading

This module is the main core of PEMesh and it is started as soon as the application is run. It provides the possibility to load an existing polygonal mesh or to generate a new one from scratch. In PEMesh, the mesh generation method takes a cue from the approach described in [2], where the domain is supposed to be a squared canvas, and the area of the domain which is not covered by a polygon is filled with triangles according to [21].

Differently from [2], PEMesh supports the generation of meshes with more than one non-triangle polygon, whose position, scale and rotation is chosen by the user before applying the triangulation of the external domain (Fig. 2). Furthermore, triangulation parameters are set according to the user needs (e.g., fixing the area of the triangles or the minimum angle) and the mirroring approach proposed in [2] can be eventually applied after the triangulation.

An additional feature is the aggregation of the generated triangles to create generic polygons (Fig. 3). This feature allows the generation of generic polygonal meshes where the number of triangles is reduced almost to zero and some geometric properties are stressed all over the discretised domain. The aggregation criterion guarantees that the diameter of the polygons generated by aggregation is at most equal to the diameter of the smallest user-selected polygon.

### 3.2. Geometric analysis

When one or more polygonal meshes are available, either generated from scratch or loaded from disk, PEMesh allows a deep geometric analysis and provides a visual summary of geometric properties. Specifically, our approach considers a set of polygonal metrics (Table 1, Fig. 4), also considering their minimum, maximum and average values.

Our polygonal metrics are classified into 6 main classes:

- **edges**: number of edges (nE) of the input polygon, shortest edge (SE);
- **angles**: ratio MA/mA, with MA, mA minimum, maximum inner angle of the polygon, respectively;
- **areas**: area (AR) of the polygon, kernel area (KE), kernel-area ratio (i.e., ratio between the area of the kernel of the polygon and its whole area), area-perimeter ratio (APR);
- **radii**: inscribed circle radius (IC), circumscribed circle radius (CC), circle ratio (CR:=IC/CC);
- **distances**: minimum point to point distance (MPD), normalized point distance (NPD) (i.e., normalized version of MPD);
- **shape regularity** (SRG), as ratio between the radius of the circle to the circumscribed to the polygon and the radius of the circle inscribed in the kernel of the element.

**PEMesh** provides three different visualizations of the geometric analysis, thus enabling the possibility to either analyze...
Table 1
Proposed polygonal metrics. For scale invariant measures, the fourth column indicates whether optimal values are at the top (1) or bottom (1) of the definition range. Polygon measures: inscribed circle (IC), circumscribed circle (CC), polygon area (AR), kernel area (KE), minimum angle (MA), shortest edge length (SE), and minimum point to point distance (MPD).

| Metric                  | Abbr. | Range           | Scale inv. |
|-------------------------|-------|-----------------|------------|
| Edges                   | nE    | (1, +∞)         | −          |
| Inscribed radius        | IC    | (0, ∞)          | −          |
| Circumscr. radius       | CC    | (0, ∞)          | −          |
| Circle ratio            | CR    | [0, 1]          | ↑          |
| Area                    | AR    | [0, ∞)          | −          |
| Kernel-area             | KE    | [0, ∞)          | −          |
| Kernal-area ratio       | KAR   | [0, 1]          | ↑          |
| Perimeter-area ratio    | PAR   | (0, ∞)          | ↑          |
| Min. angle              | MA    | (0, π)          | ↑          |
| Max. angle              | mA    | (0, π)          | ↑          |
| Shortest edge           | SE    | (0, ∞)          | −          |
| Edge ratio              | ER    | (0, 1)          | ↑          |
| Min p2p distance        | MPD   | (0, ∞)          | −          |
| Normal. point dist.     | NPD   | (0, 1)          | ↑          |
| Shape regularity        | SRG   | (0, 1)          | ↑          |

Figure 5: Screenshot of the window visualizing PEM solver results. On the top, both solver output and ground truth are color-mapped on the input polygon meshes, while on the bottom a set of linear plots show how solver performances vary in the data set.

D. Cabiddu, G. Patanè, M. Spagnuolo: Preprint submitted to Elsevier
PEMesh: a Graphical Framework for the Analysis of the Interplay Between Geometry and PEM Solvers

Figure 6: Scatter plot of the correlation between geometric properties and performance metrics of the PEM solver: correlation between (x-axis) (a) the area-perimeter ratio and (b) the minimum angle with (y-axis) the conditioning number of the stiffness matrix of the PEM solver.

of input polygon meshes, where $G$ is a geometric metric and $G'$ may either be a geometric metric or a PEM performance evaluation, PEMesh visualizes their correlation through a specialized scatter plot, with $G$-values on the x-axis and $G'$-values on the y-axis. Similarly to any other plot in the application, these scatter plots are highly customizable in terms of point color, size, labels, and can be exported as images. An example is shown in Fig. 6.

4. Implementation

PEMesh is a standalone multi-platform desktop application implemented in C++, and exploits Qt libraries for the design and implementation of the graphical user interface (Fig. 1) and Cinolib [12] to generate and visualize meshes. We now discuss the supported data formats (Sect. 4.1), the family of parametric polygons available for the generation of polygonal meshes (Sect. 4.1), and a description (Sect. 4.3) of the PEM solver used for the experiments on the interplay between geometry and analysis.

4.1. Supported data formats

PEMesh provides the possibility to either load an existing data set or to generate a new one from scratch. In both cases, PEMesh supports the most widely used file formats for the exchange of polygonal meshes, namely OBJ, OFF and STL. Furthermore, an additional output format is provided and produces .node and .ele files encoding vertices and polygons respectively. This latter mesh format is provided to support a large amount of PEM solvers requiring this kind of input.

4.2. Input polygons

To generate new polygon meshes from scratch, the user is asked to select one or more polygons to be added to the domain. PEMesh provides a list of available polygons of two types: parametric and random [2]. When a set of random polygon is selected, the generated data set is made of a single mesh; if at least one parametric polygon is chosen, then a family of meshes $D = \{ M(0), \ldots, M(1) \}$, is generated, where $M(0)$ contains all the parametric polygons at its initial phase (e.g., they do not present critical geometric features (Fig. 7(a)) and they are progressively made worse by a deformation, controlled by the parameter $t \in [0, 1]$ (Fig. 7(b)). In the latter case, the number of generated meshes is user-defined.

Other than the polygons in PEMesh’s list (Fig. 8), the user is allowed to load polygons from file. Such a polygon is automatically scaled and translated to be placed inside the domain, and additional editing can be applied by the user itself. PEMesh allows us to save the polygon configuration into a CSV file to be possibly reloaded during any following experimental session. The CSV stores, for each polygon, any data necessary to rebuild the configuration (i.e., position, scale, rotation).

4.3. PEM solver

As aforementioned, PEMesh does not include PEM solvers, but is intended to be a support for the analysis of external tools. Specifically, PEMesh requires the solver outputs to be saved according to a very simple textual file format. Specifically, both the numerical solution computed by VEM solver and the ground-truth solution (if any) must be as a list of their values at the mesh vertices. Finally, these two arrays are saved in a .txt file whose name is composed by the input filename and an additional ending to indicate which output it encodes (i.e. either “-solution” or “-ground-truth” respectively). For each performance evaluation, an additional .txt file is generated and its name is composed of the input filename and an additional string to indicate the performance.
name. The single value representing the solver performance must be written in the file.

Use case As a matter of example, we exploit the Virtual Element Method (VEM) [26] to demonstrate how the integration between the two tools works. The MATLAB® code of the method computes the PDE solution, provides the solution ground truth and also computes some evaluations of PEM solver quality, such as (Fig. 9)

- \( \mathcal{L}_1 \) condition number \( \kappa_1(S) = \|S\|_1\|S^{-1}\|_1 \) of the PEM stiffness matrix \( S \);
- relative error \( e_S := \|u - u_h\|_S / \|u\|_S \), with weighted norm \( \|v\|_S^2 = v^T S v \);
- relative \( \mathcal{L}_\infty \)-error \( e_\infty := \|u - u_h\|_\infty / \|u\|_\infty \), between the ground-truth \( u \) and the computed \( u_h \) solutions.

To enable PEMesh to visualize the results and the correlation between PEM solver performances and geometric properties, we wrapper the code into a MATLAB function to be called from a command line and we redirect the output of the Virtual Element Method to file, according to the file format described in Sec. 4.3. The MATLAB function requires both the input mesh and the output directory as parameters. These two simple operations are sufficient to make PEMesh and the Virtual Element Method communicate.

5. Discussion and conclusions

We presented PEMesh, a novel tool helping researchers to perform experimental design and analysis of polygonal meshes for PEM solvers. Through its easy-to-use graphical interface, it simplifies the execution of experimental pipelines, from the design of polygon meshes stressing specific geometric properties to the analysis of performances of user-provided PEM solvers. It also allows correlating geometric properties with PEM solver performances, and provides advanced visualization modalities of the results such an analysis. PEMesh is available as an open-source project and we expect it can be employed in several research activities.

Current limitations and future works There are several directions in which PEMesh can be improved. First, the current implementation allows the definition of polygon meshes by loading already exiting polygons, possibly designed by exploiting external tools. Since PEMesh is intended to support activities in several research fields, additional features, such as the possibility to draw polygons freehand, would simplify the mesh generation process by users coming from fields other than mesh design. A deeper analysis, including user studies, can support the development of an improved version of the tool according to user needs.

Also, PEMesh supports 2D meshes, but the entire architecture is agnostic to the dimension of the geometric input. It is almost trivial to extend the graphical user interface to support 3D meshes, but future investigations are necessary on 3D mesh generation approaches and definition of geometric properties that would be likely to be of interest for the research community.

Finally, PEMesh is designed as a desktop application exploiting RAM memory to both generate meshes and run PEM solvers. This architecture is sufficient to run preliminary test on sufficiently small meshes, but it is not robust enough to support arbitrary large inputs. Future activities will be addressed to improve the underlying architecture and guarantee efficiency independently from the input size.

Acknowledgments

This work is supported by the H2020 ERC Advanced Grant CHANGE, grant agreement N. 694515. Special thanks
are given to CNR-IMATI colleagues involved in the CHANGE Project for helpful discussions and comments on polygonal mesh generation and Polyhedral Elements Methods.

References

[1] Antonietti, P.F., Cangiani, A., Collis, J., Dong, Z., Georgoulis, E.H., Giani, S., Houston, P., 2016. Review of discontinuous Galerkin finite element methods for partial differential equations on complicated domains, in: Connections and Challenges in Modern Approaches to Numerical Partial Differential Equations. Springer, pp. 279–308.

[2] Attene, M., Biasotti, S., Cabiddu, D., Livesu, M., Patané, G., Pennacchio, M., Prada, D., Spagnuolo, M., 2019. Benchmark of polygon quality metrics for polytopal element methods. CNR-IMATI Technical Report 2019.

[3] Brandts, J., Korotov, S., Křížek, M., 2008. On the equivalence of regularity criteria for triangular and tetrahedral finite element partitions. Computers & Mathematics with Applications 55, 2227–2233.

[4] Brezzi, F., Lipnikov, K., Shashkov, M., 2005. Convergence of mimetic finite difference methods for diffusion problems on polyhedral meshes. SIAM J. Numer. Anal. 43, 1872–1896.

[5] Cangiani, A., Georgoulis, E.H., Houston, P., 2014. hp-version discontinuous Galerkin methods on polygonal and polyhedral meshes. Math. Mod. Meth. Appl. Sci. 24, 2009–2011.

[6] Carlet, P., 2002. The Finite Element Method for Elliptic Problems. Society for Industrial and Applied Mathematics.

[7] Cockburn, B., Dong, B., Guzmán, J., 2008. A superconvergent LDG-hybridizable Galerkin method for second-order elliptic problems. Math. Comp. 77, 1887–1916.

[8] Di Pietro, D.A., Ern, A., 2015. Hybrid high-order methods for variable-diffusion problems on general meshes. Comptes Rendus Mathématique 353, 31–34.

[9] Du, Q., Faber, V., Gunzburger, M., 1999. Centroidal voronoi tessellations: Applications and algorithms. SIAM review 41, 637–676.

[10] Hecht, F., 2012. New development in freefem++: A new version of Numerical Mathematics 20, 251–265. URL: https://freefem.org/.

[11] Lipnikov, K., 2013. On shape-regularity of polyhedral meshes for solving pdes, in: Research Notes, 22nd International Meshing Roundtable.

[12] Livesu, M., 2017. cinolib: a generic programming header only c++ library for processing polygonal and polyhedral meshes. https://github.com/mlivesu/cinolib/.

[13] Mascotto, L., 2018. Ill-conditioning in the virtual element method: Stabilizations and bases. Numerical Methods for Partial Differential Equations 34, 1258–1281.

[14] Mu, L., Wang, X., Wang, Y., 2015. Shape regularity conditions for polygonal/polyhedral meshes, exemplified in a discontinuous galerkin discretization. Numerical Methods for Partial Differential Equations 31, 308–325.

[15] Okereke, M., Keates, S., 2018. Finite element Mesh Generation. Springer International Publishing, Cham. pp. 165–180.

[16] Orolu, K., 2020. Finite element triangular mesh generator, matlab central file exchange. URL: https://www.mathworks.com/matlabcentral/fileexchange/32875-finite-element-triangular-mesh-generator.

[17] Ortiz-Bernardin, A., Alvarez, C., Hitschfeld-Kahler, N., Russo, A., Silva-Velazquez, R., Olate-Sanzana, E., 2016. VEMLab: a MATLAB library for the virtual element method URL: https://github.com/cemcen/veml.

[18] Ortiz-Bernardin, A., Alvarez, C., Hitschfeld-Kahler, N., Russo, A., Silva-Velazquez, R., Olate-Sanzana, E., 2019. Veamy: an extensible object-oriented c++ library for the virtual element method. Numerical Algorithms 82, 1189–1220.

[19] Rjasanow, D.S., Weißer, S., 2013. Higher order BEM-based FEM on polygonal meshes. SIAM J. Numer. Anal. 241, 103–115.

[20] Schneider, T., Dumas, J., Gas, X., Botsch, M., Panozzo, D., Zorin, D., 2019. Poly-spline finite element method. ACM Transactions on Graphics.

[21] Shewchuk, J.R., 1996. Triangle: Engineering a 2d quality mesh generator and delaunay triangulator, in: Applied computational geometry towards geometric engineering. Springer, pp. 203–222.

[22] Shewchuk, J.R., 2002. What is a good linear finite element? interpolation, conditioning, anisotropy, and quality measures (preprint). University of California at Berkeley 73, 137.

[23] Sukumar, N., Tabarraei, A., 2004. Conforming polygonal finite elements... International Journal for Numerical Methods in Engineering 61, 2045–2066.

[24] Sutton, O.J., 2017. The Virtual Element Method in 50 Lines of MATLAB. Numer. Algorithms 75, 1141–1159.

[25] Talischi, C., Paulino, G., Pereira, A., Menezes, I., 2012. Polymesher: A general-purpose mesh generator for polygonal elements written in matlab. Structural and Multidisciplinary Optimization 45, 309–328.

[26] Beirão da Veiga, L., Brezzi, F., Cangiani, A., Manzini, G., Marini, L.D., Russo, A., 2013. Basic principles of virtual element methods. Math. Mod. Meth. Appl. Sci. 23, 199–214.

[27] Beirão da Veiga, L., Lipnikov, K., Manzini, G., 2014. The Mimetic Finite Difference Method for Elliptic Problems. Number 11 in MS&A, Springer.

[28] Wang, J., Ye, X., 2013. A weak Galerkin finite element method for second-order elliptic problems. Journal of Computational and Applied Mathematics 241, 103–115.

D. Cabiddu, G. Patanè, M. Spagnuolo: Preprint submitted to Elsevier Page 7 of 7