A Methodology for FEA over Tangled Meshes

Josh Danczyk and Krishnan Suresh

University of Wisconsin, Madison, WI 53705, USA

Abstract. A finite element mesh is said to be '*tangled*' if two or more its elements overlap. This can occur, for example during mesh optimization and mesh morphing. Modern finite element theory and commercial FEA packages are not designed to handle tangled meshes, i.e., they can lead to erroneous results.
In this paper, a new mathematical framework for FEA is proposed for handling tangled simplicial meshes. Specifically, by defining a cell-decomposition of a tangled mesh, and an associated set of cell shape functions as an oriented linear combination of the classic element shape functions, it is shown that one can successfully carry out accurate FEA over tangled meshes. Numerical examples illustrate the correctness of the proposed framework.

1 Introduction

In modern finite element analysis (FEA), the underlying mesh is required: (1) to be connected, (2) to conform to the boundary, (3) to be of 'good quality', and (4) *not contain overlapping elements* [1,2]. Figure 1 illustrates an unacceptable mesh with overlapping elements; such a mesh is said to be tangled.

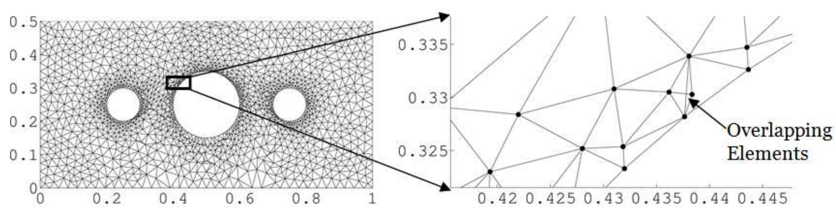

Figure 1: A tangled mesh and a pair of over-lapping elements.

Modern finite element theory and commercial FEA packages are not designed to handle tangled meshes, i.e., they can lead to erroneous results if such meshes are used. This is confirmed later in this paper through a simple experiment using ANSYS [15]. Further, tangling of finite element meshes can occur during:

- ***Mesh generation**:* Mesh generators are generally adept at constructing high-quality non-tangled meshes. Yet, for example, during all-hex mesh generation, tangling can inadvertently occur, resulting in a mesh-failure [4].
- ***Mesh optimization*:** Similarly, mesh optimizers, in an attempt to improve mesh quality, can inadvertently introduce tangling [5–8]. Further, classic notions of mesh quality are inappropriate in the presence of tangles [5,8–10]. Therefore, the mesh is first untangled [10], and then optimized [5]. More recently, re-

search efforts have focused on improving the quality of meshes while untangling [7,11,12].

- ***Mesh Morphing:*** When the underlying geometry is modified, it may be advantageous to morph, i.e., stretch and transform, an existing mesh rather than remesh [13]. Various mesh morphing techniques exist today [13]. Unfortunately, all mesh morphing methods, especially the simple and efficient ones, can lead to tangled meshes [13].
- ***Mesh deformation*:** Last, but not the least, in large-scale deformation, excessive node movement can result in tangling [14]. Currently, the only acceptable strategy is to remesh. However, remeshing can result in a significant loss in accuracy.

Researchers and practitioners today unanimously recommend untangling the mesh prior to analysis. For example, to quote [11]: "*Because tangled meshes generate physically invalid solutions, it is imperative that such meshes [be] untangled*." Unfortunately, untangling is as difficult as mesh generation and optimization [12]. Therefore, a new extension to FEA is proposed in this paper; this extension provides the necessary framework for handling tangled meshes.

2 Proposed Framework

2.1 Problem Identification

Consider the finite element mesh in Figure 2; specifically, consider an interior node within the mesh, and all elements attached to it. Now, in classic FEA, over each element, shape functions are defined such that: (1) each function takes a value of 1 at the node, (2) it goes to zero at all other nodes of that element, and (3) the functions are continuous across element boundaries. These shape functions are then 'stitched' together into single hat function ϕ that is continuous, and serves as the basis for seeking approximate solutions in FEA. In other words, one seeks solutions of the form:

$$u = \sum_i \hat{u}^{(i)} \phi^{(i)} \quad (1)$$

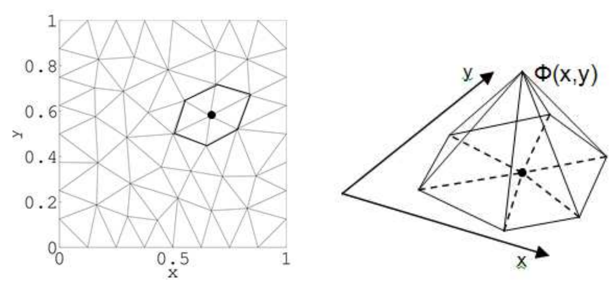

Figure 2: Finite element mesh and the 'hat-function'.

Now consider a tangled mesh in Figure 3. Specifically, consider the subset of the mesh associated with the node of interest. When a mesh gets tangled, the hat-function ϕ is ill-defined since there are points that belong to multiple elements (Figure 3), and therefore multiple element shape functions are defined at such points.

The ambiguity of ϕ can be resolved in multiple ways. For example, ϕ at an overlapping point can be defined as: (1) the maximum of all element shape function values at that point, or (2) the sum/average of these values, and so on. However, in addition to resolving the ambiguity, the continuity condition must also be satisfied. Thus, an arbitrary method of resolving the ambiguity will not do.

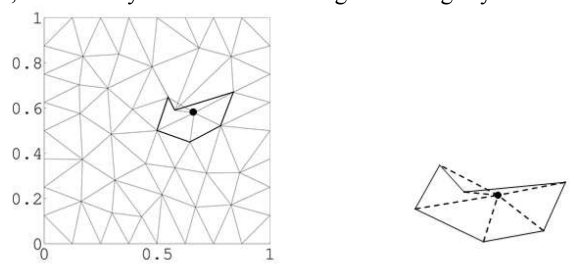

Figure 3: A tangled mesh surrounding a given node.

2.2 Proposed Theory

First observe that in a tangled simplicial mesh, some elements are positively oriented, while others are negatively oriented; this can be determined through a simple Jacobian test.

Definition 1: Let the orientation of a simplicial element be denoted by Θ_k, where $\Theta_k = \pm 1$. □

Next, since a point within a tangled mesh can belong to multiple elements, we define here an *index* associated with each point as:

$$I(p) = \{k \mid p \in E_k\} \tag{2}$$

i.e., it is the set of all elements (integers) that contain that point. The notion of an index naturally leads to the definition of a cell where we group together all points with identical index.

Definition 2: A *cell* is the set of all points with identical index I. □

Figure 4 illustrates the indices associated with the cell decomposition after tangling. Thus, there are points that belong to elements 1 and 6, for example. Further, in the cell {1,6}, element-1 is positively oriented, while element-6 is negatively oriented. Observe that a cell need not be convex, or even connected.

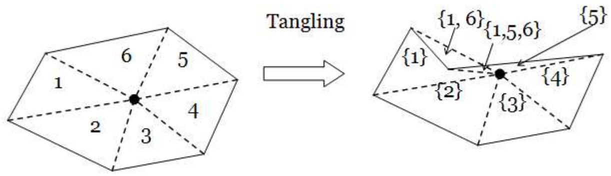

Figure 4: Cells indices.

Recall that, in classic FEA, over each element, a shape function N_k is defined. We can now combine these shape functions into a *cell shape function* S_α defined as follows:

$$S_\alpha(\cdot) = \sum_{k \in I_\alpha} \Theta_k N_k(\cdot) \tag{3}$$

where $\Theta_k = \pm 1$ is the orientation of the simplicial element E_k. In other words, the cell shape function is weightage average of the element shape functions associated with that cell, where the weighting captures of the orientation of that element. Finally, the most critical theorem in establishing a finite element framework for tangled meshes is stated next.

Theorem: *The cell shape functions defined via Equation* (3) *over the cell decomposition satisfy the following properties: (1) they are continuous across cell-boundaries, and (2) they vanish on the boundary of the corresponding cell complex*.

Proof: Will appear in a future publication; a preliminary proof appears in the recent PhD thesis [16].

□

Essentially, the theorem guarantees that the cell shape functions S_α can be used effectively as a basis for finite element analysis over a tangled mesh.

2.3 Implementation

The previous section introduced the concept of cells and cell shape functions for establishing certain theoretical properties. However, cells are unnecessary in a practical implementation of FEA over a tangled mesh. In other words, there is no need to explicitly compute the cell decomposition; the underlying reason is as follows.

Upon stitching the cell shape functions S_α together, one can show that an equivalent definition for the hat function is simply given by:

$$\phi(p) = \sum_{k \mid p \in E_k} \Theta_k N_k(p) \tag{4}$$

In other words, the hat function at any point is simply the oriented linear combination of all element shape functions of elements that contain that point. Observe that when the elements are not tangled this recovers the classic definition. Now, one can substitute this into the variational equation, for say, the Poisson equation resulting in the following linear system:

$$\left(K_{classic} + K_{overlapping}\right)\hat{\mathbf{u}} = f_{oriented} \tag{5}$$

where (the summation is interpreted in the usual sense of finite element assembly):

$$\begin{aligned} K_{classic} &= \sum_j \int_{E_j} \nabla\mathbb{N}_j^T \bullet \nabla\mathbb{N}_j \, d\Omega \\ K_{overlapping} &= \sum_j \sum_{k \neq j} \int_{E_j \cap E_k} \Theta_j \Theta_k \nabla\mathbb{N}_j^T \bullet \nabla\mathbb{N}_k \, d\Omega \\ f_{oriented} &= \sum_j \int_{E_j} \Theta_j \mathbb{N}_j^T f d\Omega \end{aligned} \tag{6}$$

and $\mathbb{N}_j$ is the vector of shape functions associated with an element, due to all nodes attached to it.

Performing FEA over a tangled mesh therefore requires the following steps:

1. The stiffness matrix $K_{classic}$ is computed exactly as in classic FEA [1].
2. Then, to compute $K_{overlapping}$ the overlapping regions must be computed (see Figure 5a). Then, in order to perform the integration, the overlapping region is subsequently triangulated, see Figure 5b.
3. Finally, $f_{oriented}$ is computed exactly as in classic FEA with one modification: the orientation of the element must be accounted for. This is accomplished by multiplying the local components of $f_{oriented}$ for each element by its orientation (± 1) before adding to the global $f_{oriented}$ term.

Further details will appear in a future publication; see [16].

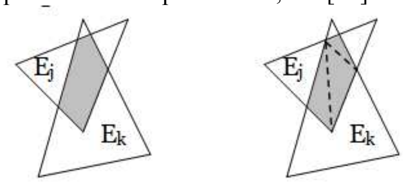

Figure 5: (a) Overlapping region of elements E_j and E_k, and (b) the triangulation of the overlapping region.

3 Numerical Results

The theory presented is now illustrated through numeric examples. In particular, the first few examples are 'validity tests'; this is analogous to patch tests in classic FEA [1,17,18]. These tests are designed in this paper to identify incorrect theory/implementation. In particular, classic FEA, including ANSYS 13 [15], a commercial FEA system, is compared against the proposed methodology. After these tests, the focus turns to a specific application of tangling, namely mesh morphing.

3.1 Single Tangle in 2-D

Consider a thermal conduction problem over a unit square with a thermal conductivity of 1. The left-edge is set to a temperature of 0, a thermal flux of 1 is applied on the right-edge, and the top and bottom edges are insulated. The exact solution to this problem is $T(x,y)=x$.

The problem is solved over two meshes with linear element shape functions: (1) a regular valid mesh of Figure 6a, and (2) a tangled mesh of Figure 6b that is constructed by flipping the x-location of the internal nodes in Figure 6a about the $x=0.5$ line. Since the solution falls within the finite element space, the exact solution should be recovered to within machine precision (even though the triangles are of poor quality).

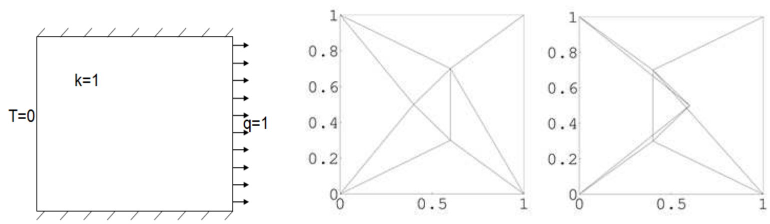

Figure 6: (a) Valid mesh and (b) tangled mesh.

Table 1 illustrates the solution for the two meshes at the location $(1,0)$, as produced by ANSYS and the proposed methodology. As expected, with a non-tangled mesh, the exact solution of 1.0 is recovered via ANSYS 13 [15] and the proposed methodology (to within machine precision). However, when the mesh is tangled, ANSYS results in a 1.2% error. Increasing the size of the inverted element leads to errors as large as 10%. On the other hand, the proposed theory yields the exact solution even in the case of the tangled mesh.

Table 1: Temperature at x=1, y=0

	ANSYS 13	Proposed Theory
Valid Mesh	1.0000	1.0000
Tangled Mesh	0.9875	1.0000

3.2 Multiple Tangles in 2-D

Next consider triangulating a unit square, see Figure 7a, and then randomizing the location of the interior nodes to produce the tangled mesh shown in Figure 7b. A (random) linear field $T(x,y)=0.323x-0.651y+0.998$ was chosen as the exact solution to a Poisson problem; Dirichlet conditions, Neumann conditions, and body forces were computed from this field, and applied on the tangled mesh as follows: Neumann conditions are applied at the bottom, right, and top boundaries, and Dirichlet conditions are applied to the left boundary. The objective is to recover the exact solution using linear elements. Similarly a random quadratic field was chosen as the exact solution, and the objective was to recover the exact solution using quadratic elements.

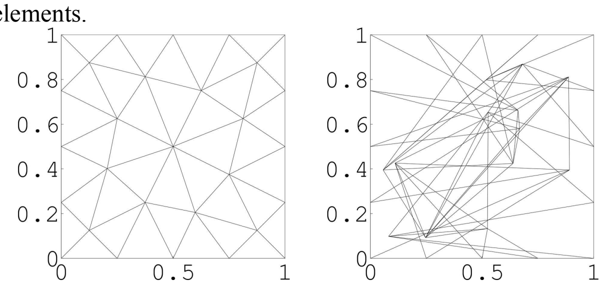

Figure 7: Initial untangled mesh (a) and the actual tangled mesh (b) used for validity tests.

The results are summarized in Table 2 where normalized errors over all nodal values (defined as $abs(\| u - u_{Exact} \|)/\| u_{Exact} \|$) from classic FEA and the proposed theory are summarized. Observe that, once again, the proposed theory recovers the exact solution, whereas classic FEA fails the test.

Table 2: Normalized errors for 2-D Poisson problem validity test

	Classic FEA	Proposed theory
Linear	1.8438	4.9145e-13
Quadratic	7.5844	4.0306e-12

Similar results were obtained for 2-D elasticity [16].

3.3 Mesh Morphing in 3-D

In the final example, a 3-D linear elasticity problem is considered. In the bearing block shown in Figure 8a two of the mounting holes are fixed, while the third mounting hole has a force in the y and z direction. The quantity of interest is the maximum displacement. The initial mesh configuration is shown in Figure 8b. The finite element problem is now solved over the mesh in Figure 8b using classic FEA. The quantity of interest is the maximum displacement.

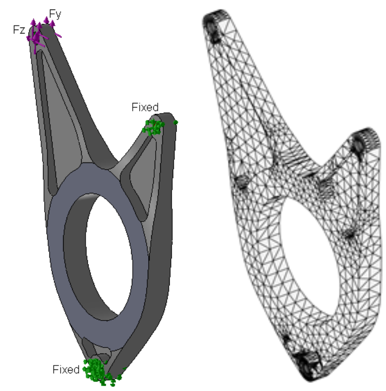

Figure 8: (a) Initial mesh that is subsequently morphed and tangled. (b) Overlapping regions (dark portions) of the tangled mesh.

Next, the diameter of the bearing surface (center hole) is increased from 63.5 mm to 72 mm as shown in Figure 9a. Instead of remeshing, the mesh is morphed using the simplex method described in [13]. After morphing, the mesh gets tangled, and the overlapping regions of the tangled mesh are illustrated in Figure 9b. The finite element problem is now solved over the tangled mesh in Figure 9b using the methodology described in this paper. Further, the new geometry is also remeshed and solved using classic FEA, to facilitate numerical comparison.

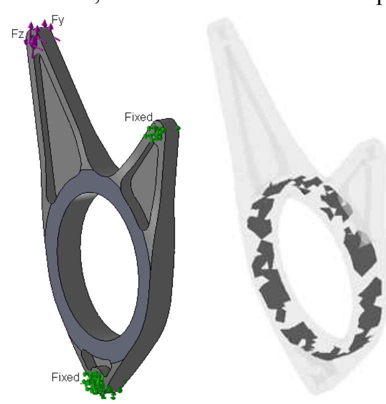

Figure 9: (a) Initial mesh that is subsequently morphed and tangled. (b) Overlapping regions (dark portions) of the tangled mesh.

Table 3 lists the maximum deflection for the initial configuration, the tangled configuration, and a remeshed configuration (all quantities are in mm). As one can observe, the displacement of the tangled mesh closely matches that of the remeshed solution. On the other hand, commercial FEA resulted in significant errors for the tangled mesh problem.

Table 3: Comparison of maximum total displacement

	Initial Configuration	Final Configuration	
		Morphed	Remesh
Maximum Total Displacement	2.1230e-2	2.5392e-2	2.5371e-2

4 Conclusions

In this paper an extension to the underlying mathematics of the classic finite element formulation is proposed. This extension allows FEA to be used in conjunction with tangled meshes that were previously considered unacceptable. In addition, it is shown that the proposed methodology can be easily incorporated into classic FEA with minor modifications. Numeric experiments illustrate the correctness of the proposed methodology; this is in contrast to commercial implementations of FEA.

While only simplicial elements were considered, the extension to non-simplicial elements, e.g. quadrilateral and hexahedral elements, is currently being investigated. From a tangling perspective, the most critical difference between simplicial and non-simplicial elements is that non-simplicial elements can also suffer from *implicit tangling* in that an element can overlap with itself. It would also be appropriate to reevaluate mesh generators and mesh optimizers. Similarly, while the theory extends to non-linear problems, further investigation and experiments are required.

References

[1] Zienkiewicz O. C., 2005, The Finite Element Method: Its Basis and Fundamentals, Elsevier Butterworth Heinemann.

[2] Lo S. H., 2002, “Finite element mesh generation and adaptive meshing,” Prog. Struct. Engng Mater., **4**, pp. 381–399.

[3] 2012, ANSYS 13, ANSYS; www.ansys.com.

[4] Knupp P. M., 2003, “A method for hexahedral mesh shape optimization,” Int. J. Numer. Meth. Engng., **58**(2), pp. 319–332.

[5] Freitag L. A., and Plassmann P., 2000, “Local optimization-based simplicial mesh untangling and improvement,” Int. J. Numer. Meth. Engng., **49**(1-2), pp. 109–125.

[6] Knupp P. M., 2000, “Achieving finite element mesh quality via optimization of the Jacobian matrix norm and associated quantities. Part II—A framework for volume mesh optimization and the condition number of the Jacobian matrix,” Int. J. Numer. Meth. Engng., **48**(8), pp. 1165–1185.

[7] Knupp P. M., 2000, “Achieving finite element mesh quality via optimization of the Jacobian matrix norm and associated quantities. Part I—a framework for surface mesh optimization,” Int. J. Numer. Meth. Engng., **48**(3), pp. 401–420.

[8] Escobar J. M., Rodriguez E., Montenegro R., Montero G., and González-Yuste J. M., 2003, “Simultaneous untangling and smoothing of tetrahedral meshes,” Computer Methods in Applied Mechanics and Engineering, **192**(25), pp. 2775–2787.

[9] Freitag L. A., and Knupp P. M., 2002, "Tetrahedral mesh improvement via optimization of the element condition number.," International Journal for Numerical Methods in Engineering, **53**, pp. 1377–1391.
[10] Xiang-yang L., and Freitag L. A., 1999, Optimization-Based Quadrilateral and Hexahedral Mesh Untangling and Smoothing Techniques.
[11] Knupp P. M., 2001, "Hexahedral and tetrahedral mesh untangling," Engineering with Computers, **17**, pp. 261–268.
[12] Bhowmick S., and Shontz S. M., 2010, "Towards high-quality, untangled meshes via a force-directed graph embedding approach," Procedia Computer Science, **1**(1), pp. 357–366.
[13] López E. J., Nigro N. M., and Storti M. A., 2008, "Simultaneous untangling and smoothing of moving grids," Int. J. Numer. Meth. Engng., **76**(7), pp. 994–1019.
[14] Staten M. L., and et. al., 2011, "A Comparison of Mesh Morphing Methods for 3D Shape Optimization," Proc. of the 2011 International Meshing Roundtable.
[15] Vachal P., Garimella R. V., and Shashkov M. J., 2004, "Untangling of 2D meshes in ALE simulations," Journal of Computational Physics, **196**(2), pp. 627–644.
[16] Danczyk J., 2012, "Finite Element Analysis over Tangled Meshes: Theory and Applications," University of Wisconsin, Madison.
[17] Strang G., 1973, An Analysis of the Finite Element Method, Prentice Hall, Englewoods Cliff, NJ.
[18] Cook R. D., 1989, Concepts and Applications of Finite Element Analysis, John Wiley, New York.