A stitching method for the generation of unstructured meshes for use with co-volume solution techniques

Igor Sazonov *, Desheng Wang, Oubay Hassan, Kenneth Morgan, Nigel Weatherill

Civil and Computational Engineering Centre, School of Engineering, University of Wales, Swansea SA2 8PP, Wales, UK

Received 1 December 2004; received in revised form 22 February 2005; accepted 26 May 2005

Abstract

The successful implementation of co-volume time-domain solution techniques requires the use of high quality, smooth dual meshes. The generation of Delaunay–Voronoi diagrams with these properties, for two-dimensional domains of complicated geometrical shape, is considered. In the adopted approach, near-boundary regions are discretised according to prescribed criteria and the remainder of the computational domain is discretised using an ideal mesh. The two meshes are stitched together to provide a consistent mesh for the complete domain. After smoothing, the resulting mesh is found to be of a quality which exceeds that of meshes produced by standard, automatic, unstructured mesh generation methods. Examples, involving electromagnetic wave scattering, are included to demonstrate the computational performance that can be achieved with a co-volume time domain solution algorithm on these meshes. © 2005 Elsevier B.V. All rights reserved.

Keywords: Delaunay triangulation; Voronoi tessellation; Unstructured co-volume mesh generation; Co-volume time domain solution technique; Electromagnetic wave propagation

1. Introduction

The Yee scheme [1–3], for the solution of the Maxwell equations, and the MAC algorithm, for the solution of the Navier–Stokes equations [4], are co-volume solution techniques that exhibit a high degree of computationally efficiency, in terms of both CPU and memory requirements. However, despite the fact that
real progress has been achieved in unstructured mesh generation methods over the last two decades, such schemes have not generally proved to be effective for simulations involving domains of complex shape. This is due to the difficulties encountered when attempting to generate high quality dual meshes, that are sufficiently smooth, for such problems. In this paper, we will address this difficulty and describe a procedure for generating unstructured dual meshes, suitable for use with co-volume solution techniques, for general two-dimensional domains.

To illustrate the practical usefulness of the procedure, the simulation of two-dimensional electromagnetic wave scattering problems is considered. The Yee scheme is the classical finite difference co-volume time domain solution technique for this class of problems. A basic requirement for the successful implementation of a co-volume scheme is the existence of two high quality, mutually orthogonal, meshes for the problem under consideration. We begin by describing the implementation of a co-volume solution technique in the unstructured mesh context, where the obvious dual mesh choice is the Delaunay–Voronoï diagram. The requirements that must be placed on the dual meshes are derived and the concept of an ideal mesh for the co-volume solution technique is introduced. Standard mesh generation methods are designed to create high quality Delaunay triangulations, but do not attempt to provide a high quality dual Voronoï mesh. For this reason, we split the problem of triangulation of a domain of complicated shape into a set of relatively simple problems of local triangulation. Each local mesh is constructed with properties which are close to those of the ideal mesh and the local triangulations are combined, to form a consistent mesh, by using a stitching algorithm. The quality of the stitched mesh is improved by the use of standard mesh quality enhancement methods. The usefulness of the proposed procedure is investigated by undertaking electromagnetic wave scattering simulations with both the co-volume technique and an explicit finite element procedure, on meshes produced by different mesh generation methods. It is demonstrated that the co-volume algorithm, implemented on meshes generated by the stitching method, is a significantly more effective computational procedure.

2. Co-volume solution techniques on unstructured meshes

2.1. A co-volume scheme for electromagnetics

We demonstrate the characteristics required of a mesh by constructing initially a co-volume solution technique for electromagnetics. Consider the simulation of two-dimensional problems involving the propagation of transverse electric waves through a medium of permittivity $\varepsilon$ and permeability $\mu$. With respect to a Cartesian $(x, y, z)$ co-ordinate system, we assume that the electric field $E$ lies in the $(x, y)$ plane, with the magnetic field $H$ in the $z$ direction. A co-volume scheme requires two mutually orthogonal meshes and we choose to employ the Delaunay–Voronoï dual diagram.

The governing equations are considered in the integral, time domain, form of the laws of Ampère and Faraday [5]. A suitable starting point for the development of the solution algorithm is then to express the laws in terms of integrals over the edges of the Delaunay and Voronoï cells. To illustrate the process, consider a triangular element $m$ of the Delaunay mesh. This element will share an edge with $N_m$ elements, with numbers $m_i$, $1 \leq i \leq N_m$, where $N_m = 3$, unless the element has an edge representing the boundary of the domain. Suppose the Delaunay edge $mm_i$ is the common edge between elements $m$ and $m_i$ and let the length of this edge be denoted by $\ell_{mm_i}$. Similarly, suppose that the Voronoï edge $mm_i$ is the line segment connecting the circumcentres of element $m$ and element $m_i$. The length of this Voronoï edge will be denoted by $h_{mm_i}$. As basic unknowns in the solution algorithm, we consider the value of the $z$-component of the magnetic field at the Voronoï vertices, and denote this by $H_{mm_i}$, and the projection of the electric field at the midpoint of the Delaunay edge $mm_i$ in the direction of the edge, and denote this by $E_{mm_i}$. In this case, the laws of Ampère and Faraday can be approximated, using central differencing, as
with the superscript $n$ denoting an evaluation at time $t = t_n$ and $t_{n+1} = t_n + \Delta t$, where $\Delta t$ is the time step and $S_m$ is the area of element $m$. This is a staggered explicit scheme, where the time step size for a stable implementation may be determined from the requirement \[ \Delta t < \sqrt{\mu \min \{\ell_{\min}, h_{\min}\}} \] Here $\ell_{\min}$ and $h_{\min}$ are the minimum Delaunay and Voronoï edge lengths respectively. It is apparent, from this equation, that a practical implementation requires the use of meshes which do not include either very short Delaunay, or very short Voronoï, edges. However, Voronoï edge lengths may vanish completely, on a general unstructured mesh, when two adjacent triangles have a common circumcentre. When this happens, the simple remedy is to merge these two triangles to form a single quadrilateral element. The discrete formulae of Eqs. (1) and (2) may be applied directly to this quadrilateral, with appropriate redefinition of $N_m$. Moreover, the same merging procedure can be adopted when more than two triangles share a common circumcentre and Eqs. (1) and (2) applied again to the polygonal cell that is created by merging the triangles in this manner. This merging process is illustrated in Fig. 1.

2.2. Mesh requirements

Meshes employed for wave propagation problems are generally constructed to have a uniform element size, $\delta$, that is related to the characteristic wavelength $\lambda$. Usually, a value of $\delta$ in the range $\lambda/30$ to $\lambda/15$ is employed with the Yee scheme \[5\]. If the boundaries of the computational domain are ignored, the ideal mesh of triangles for the co-volume scheme is a structured mesh of equilateral triangular elements, with Delaunay edge length $\delta$ and Voronoï edge $\delta_V = \delta/\sqrt{3}$. The solution scheme of Eqs. (1) and (2) will be second order accurate on such a mesh, as all edge integrals are approximated by values at the edge midpoints and the integrals over triangles are approximated in terms of values at the barycentres. For general unstruc-
tured meshes, the point of intersection of the Voronoï and Delaunay edges will not be the midpoint of the Voronoï edge and the circumcentre will not coincide with the barycentre. This will result in the loss of the second order accuracy of the scheme [6]. If a circumcentre lies outside the corresponding element, the situation is worse, as the integral over the triangle is then approximated in terms of a value located at a point outside the element. A similar conclusion may be reached if the Delaunay and Voronoï edges do not intersect. These situations will occur whenever a triangular element in the mesh is obtuse.

Based upon these considerations, it is apparent that the mesh required for the successful implementation of the co-volume solution technique should be such that:

1. it forms a Delaunay–Voronoï dual diagram, but may contain non-triangular elements if merging has been performed because a Voronoï edge length vanishes on the original mesh;
2. all Voronoï and Delaunay edge lengths are bounded from below;
3. every circumcentre lies inside the corresponding element, so that each triangular element is acute;
4. the Delaunay mesh spacing is uniform, in general, with a prescribed discretisation length scale \( \delta \);
5. Delaunay and Voronoï edge lengths are bounded from above by a value that is not significantly greater than \( \delta \);
6. any deviation in the location of the midpoint of a Voronoï edge from the point of intersection of the Voronoï and Delaunay edges should be minimised;
7. any deviation of circumcentres from the barycentres should be minimised.

In this list of requirements, the first three entries are the most important in securing a practical, stable, implementation of the co-volume scheme.

2.3. Mesh quality measures appropriate to co-volume solution techniques

A global mesh quality criterion of the form

\[
q = \min \left\{ \frac{e_{\min}}{\delta}, \frac{h_{\min}}{\delta_y} \right\}
\]

may be used to determine the suitability of a generated mesh for use with a co-volume solution technique. Note that the traditional criterion of maximizing the minimum angle for an individual element is not essential for present purposes. For example, Fig. 2 illustrates that a mesh of stretched isosceles triangles is better

![Fig. 2. Examples of the Delaunay–Voronoï dual diagram in the case of (a) stretched triangles and (b) triangles with maximum angles approaching 90°.](image)
than a mesh of well-shaped triangles, having maximum angles approaching 90°, where certain Voronoï edges are very short.

In general, the quality of an isotropic triangular element is greater when its circumcentre is at a maximum distance from each of its edges. For a triangular element, let \( d_i \), \( 1 \leq i \leq 3 \), denote signed distances from the circumcentre to the edges, with the distances being taken as positive if the circumcentre is inside the triangle and negative otherwise. Normalizing with the mean square of the edge lengths, we can introduce the parameter

\[
q_d = \frac{6 \min(d_1, d_2, d_3)}{\sqrt{d_1^2 + d_2^2 + d_3^2}} = \begin{cases} 
1, & \text{for an equilateral triangle}, \\
0, & \text{for a right angled triangle} 
\end{cases}
\]  

(5)

and employ this as a measure of the quality of an individual triangular element. If \( \alpha_{\text{max}} \) denotes the maximum angle of a triangle, an equivalent quality parameter

\[
q_a = 3 - \frac{6}{\pi} \alpha_{\text{max}} = \begin{cases} 
1, & \text{for an equilateral triangle}, \\
0, & \text{for a right angled triangle} 
\end{cases}
\]  

(6)

can also be used. It is possible to show that, for all triangles, \( q_a \approx q_d \), with the maximum discrepancy of 0.0338 occurring for the isosceles triangle with the angle 29°. In addition, \( q_a \leq q_d \) for all triangles, and both parameters are negative for obtuse triangles. Based upon these observations, we prefer to employ the parameter \( q_a \) as a local measure of the quality of the triangulation.

2.4. Traditional unstructured mesh generation methods

Traditional unstructured mesh generation methods, such as the advancing front method [7] and the Delaunay triangulation [8], are not designed to guarantee the creation of a mesh meeting the requirements set out above. These methods generate meshes in which the element edge length is acceptable, but they do not guarantee the regularity of the edge lengths of the dual mesh. An alternative approach is the construction of the centroidal Voronoï tessellation (CVT) and its dual. The CVT employs generating points that are the mass centroids of the corresponding Voronoï regions with respect to a given density function [9].

The quality of the meshes generated by the advancing front method and by the Delaunay triangulation has been analysed. For both procedures, mesh enhancement techniques, based on swapping and reconnection [10] and Laplace smoothing, are used to obtain an optimal position of the generated nodes. In general,
the meshes do not meet the mesh quality criteria set out above, even for domains of simple geometrical shape. For example, typical meshes generated for a circular domain are shown in Fig. 3 (a) and (b). The quality of the meshes generated by the CVT method is essentially better and this is certainly apparent for the case of the circular domain, as shown in Fig. 3(c). The values of the various quality measures for each of the meshes shown in Fig. 3 are displayed in Table 1. The ratio of the minimum and maximum Voronoi and Delaunay edges to the required mesh spacing, together with the minimum and maximum angle in any triangle, are shown for each case and the underlined number denotes the value of \(q\), defined in Eq. (4). Also shown is the value of the quality parameter \(q_a\) and the number of obtuse triangles in each mesh. Using the quality measure of Eq. (6), it is apparent that these meshes do not possess the characteristics required for a successful implementation of the co-volume solution scheme of Eqs. (1) and (2).

3. A stitching method of mesh generation

We have seen that the ideal mesh for a co-volume solution scheme, in the absence of boundaries, is a structured mesh of equilateral triangles with element edge length \(\delta\). Since this mesh has the maximum quality, with \(q = 1\), it possesses the maximum compliance in the sense that, in comparison to other meshes, it can be distorted to a greater extent before its quality drops below a critical level. For problems with boundaries, this suggests that a possible mesh generation approach could be to cover the computational domain with the ideal mesh, move the nearest nodes to the boundaries, delete all elements lying outside the computational domain and apply smoothing to the resulting mesh. However, with this approach, the quality of the distorted elements which appear cannot, in general, be significantly improved, because of the boundary constraints.

The alternative approach, which will be followed here, is to stitch together, in a consistent fashion, meshes that have been separately generated to meet the desired criteria. The ideal equilateral triangular mesh is employed, away from boundaries, in the major portion of the domain and a high quality, body fitted mesh is used in the vicinity of boundaries. The mesh in the near-boundary regions is generated by using a modified form of the advancing front method. The outer layer of the near-boundary elements is then stitched to a region of ideal mesh. The space allowed initially between these two mesh regions should be sufficient to enable mesh improvement techniques to be employed to enhance the quality of any distorted elements which may result from the stitching process.

3.1. Discretisation of boundary curves

Cubic spline interpolation is used to represent the boundary curves and boundary nodes are generated to comply with the mesh spacing requirement [7]. It may also be desirable to place additional nodes in high curvature regions to ensure that the boundary shape is adequately resolved, as illustrated in Fig. 4.

For the stitching process to be successful, it is important that the edges employed in the process of merging with the ideal mesh are of a length that is close to \(\delta\). However, if the distance between the boundary nodes is set exactly equal to \(\delta\) in areas of high curvature then, after the generation of a number of layers,
the elements will have edges with lengths that are respectively longer/shorter than $\delta$ in the case of convex/concave boundaries. This implies that it is appropriate to space the boundary nodes at a distance which is a function of both the curvature and the number of near-boundary layers that will be generated. It can be shown that the local edge length, $\delta_c$, at the boundary that will produce an edge length exactly equal to $\delta$ at the interface with the ideal mesh, after the generation of one layer, can be determined approximately as

$$\delta_c = \frac{\delta}{\sqrt{1 + \sqrt{3}C}}. \quad (7)$$

Here, $\overline{C}$ denotes a mean boundary curvature, which is computed by taking the angle $\Delta \theta$ between the tangents at the two end points of the local boundary edge divided by the arc length between the points. This arc length is approximated by the length of the edge itself. To account for the generation of more than one near-boundary layer, the modified formula

$$\delta_c = \frac{\delta}{\sqrt{1 + \sqrt{3}K_c \Delta \theta}} \quad (8)$$

may be adopted, where $K_c$ is a user-specified parameter. Normally, the value $K_c = 1$ is employed, but small improvements in mesh quality can often be produced, for boundaries with small curvature, by setting the value of $K_c$ to be equal to the number of layers of near-boundary elements to be generated.

As the element size will control the allowable time step for the co-volume solution scheme, it is important to restrict the minimum boundary edge size to a user specified value. The value chosen normally lies in the range $\delta/4$ to $\delta/2$. In addition, to maintain the highest possible element quality measure, it has been found that the ratio of the lengths of two adjacent boundary edges should not be allowed to exceed 0.8.

### 3.2. Triangulation of near-boundary regions

Near-boundary elements are generated by a modified form of the advancing front method, in which the placement of the nodes is designed to allow for the construction of a complete row of elements, each of which is close to optimal. In the standard advancing front process [7], a new node is located at a distance $\delta$ from two existing nodes, labelled $n$ and $n + 1$, that form an edge in the front. In this case, obtuse triangles are generated if $\delta_{n,n+1} > \sqrt{2}\delta$. In the modified technique, the new node is located by taking into account the lengths of the adjacent edges, $\delta_{n-1,n}$ and $\delta_{n+1,n+2}$, in the front and the angles $\alpha_n$ and $\alpha_{n+1}$ between these edges. The process of obtaining the number, and the location, of nodes to be placed is:

1. Determine the optimal number of nodes, $N_n$, to be generated around node $n$ in the front. This number will depend, mainly, on the angle $\alpha_n$ between the two boundary edges containing the node and, to a lesser degree, on the ratio between the lengths of these two edges. When $\alpha_n = P_n \pi/3$, where $P_n$ is an integer, the optimal number of added triangles is equal to $P_n$ and $N_n = P_n - 1$. If $P_n$ is not an integer, an extra node will be added only if the quality of the resulting elements is improved. For example, for
a convex boundary, it can be shown that if two adjacent edges are of equal length, the parameter, \( q_{ins} \), for the elements generated with the addition of an extra node, such as point \( A \) in Fig. 5, and the parameter, \( q_a \), for the elements generated without the addition of an extra node, may be computed as

\[
q_a = \frac{7 - 6}{\pi} \alpha_n \quad q_{ins}^a = -1 + \frac{3}{2\pi} \alpha_n. \tag{9}
\]

Comparing these formulae, it is apparent that \( q_{ins}^a > q_a \) if \( \alpha_n > \frac{16\pi}{15} \). Using this information, and the constraint of 0.8 on the allowable ratio between adjacent edge lengths, the number of points to be placed is determined, following a complicated analysis procedure, as

\[
\begin{align*}
\text{if } \alpha_n &> 5.672 & N_n = 5, \\
\text{else if } \alpha_n &> 4.616 & N_n = 4, \\
\text{else if } \alpha_n &> 3.555 & N_n = 3, \\
\text{else if } \alpha_n &> 2.483 & N_n = 2, \\
\text{else if } r_n \sin(\alpha_n) + \sin(4\alpha_n) &< 0 & N_n = 1, \\
\text{else} & & N_n = 0.
\end{align*}
\tag{10}
\]

Here, \( r_n \), defined as

\[
r_n = \frac{\min\{\ell_{n-1,n}, \ell_{n,n+1}\}}{\max\{\ell_{n-1,n}, \ell_{n,n+1}\}} \tag{11}
\]

is the ratio of the lengths of the shortest to the longest edges containing node \( n \).

(2) With the number of points to be generated determined, potential points are placed on the perpendicular bisectors of the two edges adjacent to the node. The distance \( b_{n,n+1} \) of the potential point from the edge joining node \( n \) to node \( n + 1 \) is determined, using the slope of the tangent to the boundary at these two nodes and the lengths of the neighboring edges, from the expression

\[
b_{n,n+1} = \frac{1}{2} \left( \ell_{n-1,n}^2 \ell_{n,n+1}^2 \ell_{n+1,n+2} \right)^{1/4} \tan \left( \frac{\pi}{3} + \frac{\theta_{n+1} - \theta_n}{3} \right). \tag{12}
\]

Here, \( \theta_n \) and \( \theta_{n+1} \) are the slopes of the tangent to the boundary at node \( n \) and \( n + 1 \), respectively. The use of the lengths of the adjacent edges is designed to guarantee a smooth transition.

(3) When the edge lengths are varying, the perpendicular bisector is not the optimal direction to use to ensure a high quality element measure. In this case, the base point is shifted toward the shortest adjacent edge, as shown in Fig. 6. The shift, \( \xi_{n,n+1} \), of the base point from the midpoint of the edge joining the points \( n \) and \( n + 1 \) depends upon the ratio of the two adjacent edge lengths and is determined as

\[
\xi_{n,n+1} = \frac{\ell_{n,n+1}}{2} \left[ 1 - \sqrt{r_n} \right]. \tag{13}
\]

Fig. 5. Selecting the optimal number of new points to be generated for a node in the front.
(4) After the placement of the two nodes associated with the adjacent edges, the additional nodes to be generated are placed by dividing the remaining gap into equal sectors. Along the boundary of each sector, a point is placed at a distance which guarantees a uniform variation of edge lengths around the node.

The robustness of this procedure is improved if regions of high curvature and points of slope discontinuity in the boundary geometry definition are considered first. The practical implementation, therefore, includes a preliminary automatic shape analysis which locates high curvature portions of boundaries, corners and tight gaps, where the distance between two boundaries, or different parts of the same boundary, is less than $2\delta$. Meshes generated following this approach are found to be of better quality than meshes which are obtained by always starting from the smallest edge in the front. However, edge length is the next criterion used in the process of edge selection. Fig. 7 illustrates this process of triangulation applied to the region inside a NACA0012 aerofoil and it can be seen how the trailing edge singularity and the region of high curvature at the nose are treated first. The same features are apparent in Fig. 8(a), which shows the initial stages of the triangulation of the region surrounding a two component aerofoil configuration. Note also how, in regions where only boundary points need to be triangulated, e.g., in the vicinity of small gaps with a thickness less than $\delta$ as shown in Fig. 8(b), boundary nodes may be repositioned so that the generated triangles can be merged to form quadrilateral elements.

3.3. Stitching to an ideal mesh

To connect the near-boundary mesh to a region of ideal mesh, an additional temporary layer of near-boundary elements is generated. The new nodes of this extra layer are marked as potential nodes for connection, as shown in Fig. 9(a). For each of these potential nodes, the closest node in the ideal mesh is identified, see Fig. 9(b). This set of nodes is denoted by $P_{id,cl}$ and is ordered according to the order of
potential nodes. A closed polygon, shown as a dashed line in Fig. 9(c), or set of polygons is formed by joining consecutive nodes from the set \( P_{id,cl} \). A possible approach would be to connect the near-boundary mesh and the ideal mesh at this stage by triangulating the domain bounded by the polygons and the outer region of the near-boundary mesh. However, a better quality mesh can be obtained if we locate seam nodes at the points lying midway between potential nodes and the closest ideal mesh nodes, as illustrated in Fig. 9(d). Delaunay triangulation is now used to connect the domain bounded by the boundary of the near-boundary elements and the ideal mesh nodes located one layer away from the polygons. In this triangulation process, the seam nodes are added to ensure the generation of elements of the correct size and aspect ratio, as in Fig. 9(e). Standard mesh enhancement procedures, such as element edge swapping based [10] and Laplace smoothing, can then be applied to the triangulated region to improve the quality of the generated elements and produce the final form of the mesh shown in Fig. 9(f).

The complete algorithm for mesh generation may be now described in terms of the following major algorithmic steps:

1. Generate the near-boundary mesh of high quality elements and let \( P_{nb,f} \) denote the set of nodes forming the front following the generation of the final layer;
2. Generate an ideal mesh with side length \( \delta \) covering all the remaining parts of the domain;
3. Locate the potential nodes, by an extension of the near-boundary triangulation, numbering them sequentially along the front;
4. For each potential, determine the closest node in the ideal mesh and enter it into the set \( P_{id,cl} \), maintaining the order of the potential nodes;
5. Form the separating polygons by connecting the nodes \( P_{id,cl} \) in order;
6. Locate a seam node at the point lying midway between each potential node and the closest ideal mesh node and denote the set of seam nodes by \( P_{mp} \);
7. Determine the set \( P_{id,f} \) of ideal mesh nodes which lie one layer away from the polygons;
8. Form a Delaunay triangulation of the point sets \( P_{nb,f}, P_{id,f} \) and \( P_{mp} \);
9. Use mesh enhancement procedures to improve the quality of the connecting elements.

To demonstrate the quality of the meshes that can be produced by following this procedure, consider again the problem of discretisation of the region inside a circle. For this geometry, the mesh produced by the stitching method, with \( K_c = 2 \) and two layers of elements in the near-boundary mesh, is shown in Fig. 10 and the corresponding mesh quality parameters are displayed in Table 2. The underlined number is the value of \( q \) for the mesh, as defined in Eq. (4). From this figure and table, the quality improvement that
Fig. 9. Stages of the stitching method. (a) Near-boundary elements (solid lines), ideal mesh (dotted lines), potential nodes (dots). (b) The identification of the nearest ideal mesh node for each potential node. (c) Separating polygon, small open circles indicate selected ideal mesh nodes. (d) Near-boundary front nodes (dots), seam nodes (squares), ideal mesh nodes with index less than 6 (circles). (e) Delaunay connection of these nodes (dotted lines). (f) Final mesh after the application of quality enhancement procedures.

Fig. 10. Triangulation of the interior of a circle by the stitching method, with two layers of near-boundary elements.
has been achieved compared with the meshes displayed in Fig. 3, produced by the standard methods, is apparent. Fig. 11 shows meshes generated by the stitching method, with $K_c = 1$ and two layers of near-boundary elements, for the region outside a two component aerofoil configuration and the region outside a three component aerofoil configuration, while Table 3 details the quality measures of meshes generated for different geometries. In this table, the value of $q$ for the mesh is underlined and $L$ denotes the length of the main component of the aerofoil. It can be seen that excellent element quality is also achieved in each of these cases.

A similar approach to the problem of mesh generation, using the concept of an ideal mesh, has been proposed by Muylle et al. [11]. However, in their case, the near-boundary triangulation is performed according to totally different criteria and, as a result, their meshes do not satisfy the requirements that are necessary in the current context.

The effect of employing different values for the parameter $K_c$ in the mesh generation process is illustrated in Fig. 12, which shows details of meshes generated by the stitching method for a domain exterior to a NACA0012 aerofoil geometry. Parameters for the meshes shown in Fig. 12, and for a similar mesh produced by the CVT method, are listed in the Table 4. Again, the underlined number denotes the value of $q$ for the mesh, as defined in Eq. (4).

![Fig. 11. Meshes generated, with $K_c = 1$ and two layers of near-boundary elements, for (a) a two component and (b) a three component aerofoil configuration.](image-url)
3.4. Mesh generation for electromagnetic scattering simulations

Electromagnetic scattering simulations involve the modelling of the interaction between waves, generated by a source located in the far field, and a general geometry. The infinite problem domain must be truncated to produce a domain appropriate for use in the numerical simulation. A practical approach to mesh generation for such problems is to start with a regular triangulation of a structured cartesian mesh and remove elements to construct a rectangular hole, or a number of such holes, around the prescribed geometry. Each hole may then be meshed using the stitching method outlined above. A typical example of a mesh constructed in this fashion is shown in Fig. 13(a). The co-volume electromagnetic solver will automatically merge pairs of triangular elements in the structured mesh region to produce a hybrid mesh of squares and triangles, as illustrated in Fig. 13(b). This process reduces the number of elements and edges in the mesh and leads to an enhancement in the efficiency of the resulting solution scheme. The numerical simulation of the truncated far field boundary condition is achieved by the addition of a rectangular absorbing perfectly matched layer (PML) at the far field boundary [12].
4. Numerical examples

4.1. Accuracy of the co-volume solution technique

The example of scattering of a plane single frequency electromagnetic wave by a perfectly conducting infinite prism of square cross-section is considered. The objective is to use this example to illustrate the order of accuracy that can be achieved with a co-volume solution technique on unstructured meshes. The simple geometry means that the computational domain may be discretised using a structured mesh of square elements and, in this case, the co-volume scheme of Eqs. (1) and (2) reduces to the classical Yee scheme. This method is known to be second order accurate on uniform structured grids [5], so the computational error should decrease as $O(\delta^2)$. The distribution of the radar cross-section per unit length (RCS) obtained using the Yee scheme on a fine cartesian grid, with 512 elements per wavelength, is taken as the benchmark solution. The problem is solved on a series of cartesian and unstructured meshes, with mesh spacings ranging from 8 to 256 elements per wavelength. A typical cartesian mesh, with 16 elements per wavelength, is shown in Fig. 14(a) and a corresponding unstructured mesh is shown in Fig. 14(b). With the stitching method in this case, three layers of near-boundary elements are generated in the region adjacent to the interior boundary, while two layers are generated adjacent to the external boundary. On the unstructured meshes, the problem is also solved using an explicit nodal finite element time domain (FETD) algorithm [13]. In each case, the error in the solution is determined as the maximum difference between the computed and benchmark RCS values at the same viewing angle. The variation of this computed error with the number of elements per wavelength, $N = \lambda/\delta$ is shown in Fig. 15. It can be observed that a similar convergence rate, of around $O(\delta^{1.6})$, is obtained with the co-volume scheme on both the structured and the unstructured meshes. It is probable that the presence of field singularities near the sharp corners is the reason that the theoretical rate of convergence is not achieved in this case. The results computed with the FETD method also indicate convergence, but the error at each stage is larger than that of the co-volume scheme.

4.2. Computational performance of the co-volume solution technique

Electromagnetic wave scattering by three different geometrical configurations is considered. For the first two examples, meshes are generated by both the stitching method and the CVT approach and the simulations are performed, on both meshes, with the co-volume scheme of Eqs. (1) and (2) and the FETD method.

Fig. 14. Meshes generated for the simulation of scattering by a PEC prism of square cross-section using 16 elements per wavelength: (a) a structured cartesian mesh and (b) a mesh constructed by the stitching method.
As the number of time steps required by the co-volume scheme is controlled by the smallest edge length in either the Voronoï or the Delaunay diagram, it is essential to eliminate small Voronoï edges in the meshes generated by the CVT method. For this reason, two adjacent triangles on a CVT mesh are merged to form a quadrilateral, if the distance between their respective Voronoï vertices is less than $M$ times the minimum edge length in the mesh. Different values for the parameter $M$ are used, up to a maximum value of 0.3. In this case, the point midway between the circumcentres of the merged triangles is interpreted as the circumcentre of the quadrilateral. The third example is simulated using the co-volume scheme only. The accuracy achieved and the time required is presented in each case. Accuracy is evaluated by comparing with the analytical solution, when available, or with the solution computed on a significantly finer mesh.

4.2.1. Scattering by a perfectly conducting cylinder of circular cross-section

Scattering of a plane single frequency wave by a perfectly conducting circular cylinder of diameter $15\lambda$ is considered. The meshes employed are generated to meet a specified requirement of 15 elements per wavelength. For the stitching method, six layers of near-boundary elements are generated in the region adjacent to the cylinder with the parameter $K_c = 6$. The PML is located at a minimum distance of $\lambda$ from the cylinder. The solution is advanced for 20 cycles of the incident wave and the computed RCS distributions are compared to the exact RCS distribution in Fig. 16. Good agreement with the exact solution is observed using both meshes. Three values of $M$ are used to reduce the number of time steps required per cycle (spc) when the co-volume scheme is implemented on the mesh generated using the CVT method. Table 5 displays information on the meshes and the results of the simulations. For the co-volume scheme on the CVT mesh, it can be observed that increasing the value of $M$ from 0.1 to 0.3 results in a three fold reduction in the required number of time steps per cycle and an insignificant increase in the magnitude of the measured error. However, larger values of $M$ are found to result in a less significant reduction in the number of time steps required and a more significant increase in the magnitude of the measured error. The cpu time required to complete the simulation with the co-volume scheme is nearly 34 times less than the time required by the FETD scheme for this case. It is likely that the error in the FETD results is slightly less because the approach adopted for the evaluation of the RCS integral requires an interpolation, in the co-volume scheme, to obtain all the field components at one location.
4.2.2. Scattering by a perfectly conducting NACA0012 aerofoil

The second example involves the simulation of scattering of a plane single frequency wave by a perfectly conducting NACA0012 aerofoil of length \(8\lambda\). The meshes are generated to meet the user-specified requirement of 15 elements per wavelength, with an automatic reduction in mesh spacing on the boundary in the vicinity of the high curvature region at the leading edge. For the stitching method, six layers of near-boundary elements are generated in the region adjacent to the aerofoil with the parameter \(K_c = 6\). The same boundary discretisation is used to generate an unstructured mesh using the CVT method. The unstructured meshes are generated to a distance of \(\lambda\) from the aerofoil and a detail of the mesh generated by the stitching method is given in Fig. 17. The RCS distributions computed with the co-volume method and the FETD scheme are seen to be in excellent agreement in Fig. 18. Table 6 compares the number of time steps required and the cpu time taken to compute the solution. For this problem, the reduction in the spacing at the leading edge results in an automatic increase in the number of time steps required per cycle.

\[
\begin{align*}
\text{Table 5} & \quad \text{Scattering by a } 15\lambda \text{ cylinder} \\
\text{Method} & \quad M & q & \text{spec} & \text{cpu time, s} & \text{Error, dB} \\
\text{Co-volume scheme} & \quad & & & & \\
\text{Stitching} & 0.0 & 0.69 & 55 & 9.9 & 0.70 \\
\text{CVT} & 0.1 & 0.13 & 295 & 63.9 & 0.98 \\
\text{CVT} & 0.2 & 0.25 & 147 & 31.3 & 1.12 \\
\text{CVT} & 0.3 & 0.38 & 99 & 21.6 & 1.25 \\
\text{FETD scheme} & \quad & & & & \\
\text{Stitching} & – & – & 59 & 334 & 0.35 \\
\text{CVT} & – & – & 59 & 398 & 0.68 \\
\end{align*}
\]

Fig. 16. Comparison of the exact RCS distribution for scattering by a 15\(\lambda\) perfectly conducting cylinder (continuous line) with distributions computed by the FETD scheme (dotted line) and the co-volume scheme. The co-volume results were produced on a mesh generated by the stitching method (dashed line) and on a mesh generated by the CVT approach (dot-dash line) with \(M = 0.1\).
Fig. 17. Detail of the mesh generated by the stitching method, with curvature correction, for a NACA0012 aerofoil of length $8\lambda$ using 15 elements per $\lambda$.

Table 6

<table>
<thead>
<tr>
<th>Method</th>
<th>$M$</th>
<th>spec</th>
<th>cpu time, s</th>
<th>Error, dB</th>
</tr>
</thead>
<tbody>
<tr>
<td>Co-volume scheme</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Stitching</td>
<td>0.0</td>
<td>74</td>
<td>2.063</td>
<td>–</td>
</tr>
<tr>
<td>CVT</td>
<td>0.0</td>
<td>23,607</td>
<td></td>
<td>***</td>
</tr>
<tr>
<td>CVT</td>
<td>0.1</td>
<td>239</td>
<td>5.765</td>
<td>0.25</td>
</tr>
<tr>
<td>CVT</td>
<td>0.2</td>
<td>123</td>
<td>3.234</td>
<td>0.28</td>
</tr>
<tr>
<td>CVT</td>
<td>0.3</td>
<td>82</td>
<td>2.328</td>
<td>0.41</td>
</tr>
<tr>
<td>FETD scheme</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Stitching</td>
<td>–</td>
<td>77</td>
<td>79.86</td>
<td>0.25</td>
</tr>
<tr>
<td>CVT</td>
<td>–</td>
<td>59</td>
<td>66.00</td>
<td>0.35</td>
</tr>
</tbody>
</table>

Fig. 18. RCS distributions for scattering by an $8\lambda$ NACA0012 aerofoil computed by the co-volume scheme (continuous line) and the FEDT scheme (dashed line).
Fig. 19. Scattering by a PEC cavity showing (a) the mesh generated by the stitching method and (b) the computed total field in the extended domain after 150 cycles.

Fig. 20. RCS distributions computed for the PEC cavity after 150 cycles by an FDFE method (bold), the FETD method (thin) and the co-volume scheme (dashed).
4.2.3. Scattering by a perfectly conducting cavity

The final example considers the simulation of scattering of a plane single frequency wave by a perfectly conducting U-shaped cavity. The thickness of the cavity walls is equal to 0.4λ, the internal cavity width is 2λ and the internal cavity length is 8λ. In the simulation, the wave is incident upon the open end of the cavity and propagates in a direction which lies at an angle \( \theta = 30^\circ \) to the main axis of the cavity. The stitching method is used to generate an unstructured mesh, with typical edge length \( \lambda/15 \), in the region lying within a distance of \( \lambda \) from the scatterer, as shown in Fig. 19(a). For this example, \( K_c = 1 \) and two layers of near-boundary elements are generated in the region adjacent to the cavity. The simulations are advanced for 150 cycles and the typical distribution of the computed total magnetic field in the extended domain, excluding the PML, is shown in Fig. 19(b). A comparison of the computed RCS distributions is given in Fig. 20. Also shown in this figure is the RCS distribution computed using a high-order finite element frequency domain (FEFD) simulation [14]. The number of steps per cycle is 57 for the co-volume scheme and 59 for the FETD method. For this example, the co-volume scheme requires 31 s of cpu time, while the FETD method requires 1980 s. This represents a speed-up by a factor of 65.

5. Conclusions

A method for the generation of a smooth Delaunay–Voronoï dual meshes has been proposed. The method is fast, with the number of operations required being proportional to the length of the boundary rather than the area of the total domain. This means that, as the discretisation size \( \delta \) is reduced, the number of operations required varies as \( O(\delta^{-1}) \) rather than the \( O(\delta^{-2}) \) achieved with other standard unstructured mesh generation methods for two-dimensional domains. This is because most of the domain is filled with an ideal mesh with known connectivities.

It has been demonstrated that the method gives high quality meshes that can be employed for effective implementation of co-volume integration algorithms. In particular, the efficiency of a co-volume staggered time domain scheme for electromagnetic wave scattering simulations has been illustrated in comparison with a standard finite element time domain method.

The mesh generation method can be generalised for triangulation of surfaces, with locally high quality fragments of the triangulation being stitched to each other in a similar manner.

Some preliminary investigations have also indicated how the method can be extended to three-dimensional domains. Space filling with almost regular Delaunay tetrahedra [15] can be employed to form the three-dimensional analogue of the ideal mesh. Such a tetrahedral mesh provides the best Voronoï/Delaunay edge length ratio and, hence, provides the highest compliance. It also provides guidance in the construction of the optimal near-boundary mesh for co-volume schemes and work in this area is already underway.

Acknowledgements

The authors thank the UK Engineering and Physical Sciences Research Council for the financial support provided under Research Grant GR/R38972/01.

References


