Computer aided modelling of variable angle tow composites manufactured by continuous tow shearing

Byung Chul Kim *, Paul M. Weaver, Kevin Potter

ACCIS (Advanced Composite Centre for Innovation and Science), University of Bristol, Queen’s Building, University Walk, Bristol BS8 1TR, UK

Article history:
Available online 16 April 2015

Keywords:
Automated fibre placement (AFP)
Variable angle tow (VAT)
Computer-aided design (CAD)
Finite element modelling (FEM)

Abstract

A novel fibre placement method ‘continuous tow shearing (CTS)’ has recently been developed in order to minimise process-induced defects in a tow steering process for manufacturing variable angle tow (VAT) composites. As CTS utilises shear deformation of tow materials, it distinguishes itself from conventional automated fibre placement (AFP) processes that use in-plane bending deformation. In doing so, it produces distinct distributions of fibre angle and thickness in a tow steered panel even if the same reference tow trajectories are applied. In this work, a computer-aided modelling tool has been developed, which can create accurate ABAQUS finite element models reflecting the nonlinear fibre trajectories and thickness variations of VAT composites manufactured using the CTS by defining fibre paths with geometric features in a CAD software.

© 2015 The Authors. Published by Elsevier Ltd. This is an open access article under the CC BY license (http://creativecommons.org/licenses/by/4.0/).

1. Introduction

As composite aircraft structures are reaching their limits of structural efficiency, studies on variable angle tow (VAT) composites with enhanced efficiency are gaining increasing attention, despite the original concept being first developed in 1970s [1]. It is well known that aligning fibres along desired structural load paths can significantly improve the performance of a composite structure without increasing weight [2–6].

The current key technology for manufacturing VAT composites is the automated fibre placement (AFP) technique, which was developed at several places in the early 1980s [7]. However, its process-induced defects such as local fibre wrinkling, tow gaps and fibre discontinuities remain important problems limiting AFP application for VAT composites [8]. In the AFP process, fibres within a tow are aligned parallel to the reference tow path due to the head rotation even when the steered tow paths are simply shifted along a specific direction (shifting direction), which inevitably produces tow gaps or overlaps [8], as shown in Fig. 1(a).

The resin rich area or local thickening induced by tow gaps or overlaps causes difficulty in finite element modelling. When tow gaps exist, corresponding elements need to be replaced with pure resin elements. On the contrary, tow overlaps lead to irregular thickness change within the laminate although the bottom side mated with the tool surface is even. As local thickening increases the local stiffness, the number of plies at each element location as well as the fibre orientation needs to be calculated to build a realistic model. The tow drop technique allowing the tow gaps is much more common because it makes the modelling more convenient by assuming the ply thickness to be constant. However, depending on the coverage parameter from 0% to 100%, localised tow overlaps can occur as well as the tow gaps [9]. Even when 0% gap-coverage is applied, the staggering technique is still required to avoid the co-location of the gaps and produce a more uniform distribution of the defects, but the surface dimples and the resin rich areas near the tow drop regions [9,10]. (The actual surface geometry of the tow drop region is well presented in Ref. [10].)

Tatting and Gürdal have developed a pre-processor for the finite element solver STAGS, referred to as LDT (Laminate Definition Tool), which can create shell models for elastically tailored composite laminates with linear angle variations [11]. Subsequently, they have enhanced the numerical efficiency by using the Fortran user-subroutine of the STAGS solver [12]. In these approaches, the exact shape of the resin pockets at the end of tows cut perpendicular to the fibre direction could not be modelled properly since the triangular shape of the resin pockets was approximated using large quadrilateral elements. Blom et al. have modelled tow-drop areas more accurately by modelling the geometric details of each tow strip with 1.5 x 1.5 mm elements in a commercial code ABAQUS [13]. It was found that material failure was mainly initiated at tow-drop locations through progressive failure analyses using UMAT user-subroutine of ABAQUS. Their approach has been applied to a composite cylinder with tailored stiffness, and the
implementation using UGENS user-subroutine of ABAQUS for structural analysis was described [14]. Fayazbakhsh et al. introduced a method, called “defect layer method”, to reduce the burden of FE analysis of a variable stiffness composite laminate with embedded gaps and/or overlaps [15]. They included the portion of gap or overlap within each element so as to assign a reduced in-plane modulus or an increased thickness to the corresponding shell element in a commercial code ANSYS. The reduced computation time for calculating local properties considering the defects enabled them to apply this method to an optimisation process for design of a variable stiffness panel [16]. Falcó et al. built a more accurate finite element model using 3D hexahedral and wedge elements in ABAQUS in order to investigate the effect of triangle-shaped resin pockets produced by tow drops in the AFP process on the first-ply failure prediction [17]. No significant differences have been found between the regular and the structured meshes. However, the structured mesh was more suitable for studying the interlaminar damage progression.

The limitation of the modelling approaches described above is that the angle variation has to be represented by a numerical equation such as a linear angle variation. This constraint causes difficulty in modelling when it is not possible to find simple numerical expressions for the designed tow paths or when multiple different tow paths are defined within a ply [18]. Schueler et al. have developed a GUI (Graphical User Interface)-based pre-processor generating a finite element model for a commercial code NASTRAN by considering geometric features of each tow path [19]. However, the function creating geometries was limited in defining tow paths with a sufficiently high degree of freedom, and it was only usable for conventional AFP processes. Michael and Samih have developed a software tool doing the opposite, which can define fibre trajectories over a finite element mesh by means of a modified fast marching method [20]. Although it is a flexible tool to define fibre angles on a complex mesh surface by inputting actual fibre paths rather than equations for angle distributions, it can be only used for the parallel method of conventional AFP processes where all fibre paths are offset by the tow width. Thus, it cannot simulate the shifting method that generally creates multiple tow gaps, overlaps and thickness variation. Even if the tow gaps and overlaps are modelled correctly, it is impossible for the models to capture the out-of-plane and in-plane fibre waviness due to the bending deformation of the tows.

A novel fibre placement technique, continuous tow shearing (CTS), was recently developed to minimise process-induced defects of the VAT composites [8]. The key idea was to eliminate tow gaps or overlaps by utilising the shear deformation of partially impregnated tow materials. Due to this unique manufacturing characteristic, the modelling of VAT composites can be much simpler because complex geometric calculations are not necessary to determine the location of tow gaps or overlaps in most cases, as.
shown in Fig. 1(a). Instead, a new modelling method reflecting the new manufacturing characteristics of the CTS, which is the thickness change of the tow subjected to shear deformation, is required.

In this work, a CAD (Computer Aided Design) based pre-processor was developed, which can create ABAQUS FE models of VAT composites reflecting the manufacturing characteristics of the CTS. It was programmed with AutoLISP language to enable users to define fibre paths simply by creating geometric features in AutoCAD software. For more generalised purposes, a transformation function converting a 2D shell model to a 3D continuum or solid model was implemented as well as a mapping function to assign a variable angle distribution to an irregular shell mesh that was already created in the ABAQUS/CAE module. As an added benefit, that links design and manufacturing, tow paths defined in AutoCAD can be exported in DXF format to CAM (Computer Aided Manufacturing) software, which can be directly used in the automated CTS manufacturing process.

![Continuous tow shearing head laying up a single tow](image)

**Fig. 2.** Continuous tow shearing head laying up a single tow [19].

2. Manufacturing characteristics of the continuous tow shearing (CTS)

As shown in Fig. 1(a), the key feature of CTS is that the fibre placement head applies in-plane shear deformation to the tow material supplied continuously by fixing the head rotation while a single strip of tow material is laid [8]. This process can eliminate tow gaps or overlaps, as shown in Fig. 1(b), because all fibres within a tow can be aligned to have a constant angle along the shifting direction. This unique feature allows VAT composites to be manufactured with a smooth thickness variation without tow gaps or overlaps. It also makes the finite element analysis much simpler to model. Fig. 2 shows the developed CTS prototype head laying up a single tow. Within the CTS process, it is the thickness variation, as shown in Fig. 3 schematically, that needs to be considered. As the tow material impregnated with a constant amount of resin is sheared, keeping fixed boundaries, the tow thickness increases. The thickness can be easily calculated as

\[ t = t_0 / \cos \theta \]  \hspace{1cm} (1)

where \( t_0 \) and \( \theta \) are the original tow thickness and the shear angle, respectively [21].

Fig. 4 shows a manufactured VAT laminate and its thickness profile compared to the calculation. The X- and Y-axes are the reference and shifting directions, respectively. As shown in Fig. 4(c), Eq. (1) predicts the thickness variation reasonably well [21].

3. Computer-aided modelling method of the VAT laminate manufactured by the CTS

AutoLISP is a programming language designed for extending and customising the functionality of AutoCAD (Autodesk Inc., USA), which was firstly introduced as an application programming interface (API) of the AutoCAD in the 1950s [22]. Various built-in functions for complex geometric calculation such as finding

![Thickness change of the tow element due to the shear deformation](image)

**Fig. 3.** Thickness change of the tow element due to the shear deformation.
intersection points of curves and calculating the first and second derivatives and the curvature at a point on a curve can be readily utilised without need for complex mathematical equations. Geometries required for the calculations can be drawn by using the basic GUI of the AutoCAD or importing a DXF (Drawing Exchange Format) file.

3.1. Methodology overview

In order to create an accurate finite element model of a VAT composite laminate reflecting their nonlinear fibre trajectories and thickness variations, a plug-in program VATMESH which can be run in AutoCAD (Autodesk Inc., USA) was developed using the AutoLISP language. Fig. 5 shows the internal workflow as well as how VATMESH interacts with a commercial code ABAQUS. The biggest advantage is that any type of geometry such as lines, polylines, arcs, and splines can be used to define the fibre paths. Even the complex NURBS (Non-uniform rational B-spline) curves can be used after converting them into polylines or splines since AutoCAD only supports splines which is a particular type of NURBS curves currently. The input file can be generated by following the steps below.

- **Step 1 (Defining plies and reference angles of each ply):** The user needs to define the ply number and its reference angle for each ply. The reference angle means the rotation angle \( \beta \) of the reference direction with respect to the X-axis on the drawing canvas as shown in Fig. 6, which is the perpendicular direction to the shifting direction. Those can be assigned by creating a layer with the name comprising the ply number and its reference angle. (e.g. The layer name “1 90” means the first ply with the reference angle of 90°.)

- **Step 2 (Drawing a specimen area and fibre paths):** The laminate size needs to be input by creating a rectangle on the “0” layer. On each layer created in the previous step, one or multiple reference paths need to be drawn by creating any geometry. If there is no geometry on the “0” layer, the software initiates a mapping function explained in Section 4.1.

- **Step 3 (Loading and running the VATMESH in AutoCAD):** The user needs to upload VATMESH (.lsp or .vlx file) on the memory by executing ‘(load “vatmesh”)’ in the command window or using the ‘load application’ menu.

- **Step 4 (Inputting parameters and creating the FE model):** The number of elements along x and y direction, default tow thickness \( t_0 \) and width \( w_0 \) need to be input. One of three thickness calculation options also can be chosen. If necessary, an equation constraint can be assigned to constrain some degrees of freedom on the boundary edges. VATMESH will calculate the fibre angle and thickness of each element based on the input.

- **Step 5 (Running ABAQUS analysis with the created file):** The created FE model (.inp file) can be imported in ABAQUS/CAE. Alternatively, it can be included in another input file by using the “Include card.”
3.2. Fibre angle calculation method

Fig. 6(a) shows how VATMESH calculates the fibre angles of each element. If multiple fibre paths are defined within a ply, it considers each path as a tow edge. It calculates element centroids based on the number of elements that the user inputs. Then, it detects two adjacent fibre paths and calculates two intersection points among the two paths and a virtual line crossing the element centroid along the perpendicular direction to the reference axis. The reference axis is the zero-shear angle direction, perpendicular direction of the shifting direction, as well as the orientation of the CTS head during the layup. In Fig. 6, \( \beta \) shows the rotation angle of the reference direction with respect to the X-axis. At the same time, the distance \( L \) between the two intersection points is calculated to compensate the width adjustment. If it is different from the default width \( w_0 \), it means the assigned fibre paths are not simply shifted and the width adjustment needs to be considered to calculate the thickness variation, which is described in Section 3.4. The average fibre angle at the element centroid \( (\alpha) \) is calculated by linearly interpolating the two tangent angles at these points as

\[
\alpha = \alpha_1 + \frac{d}{L} (\alpha_2 - \alpha_1)
\]

where \( \alpha_1, \alpha_2, \) and \( d \) represent the tangent angles at the two intersection points and the distance from the element centroid to one of the intersection points, respectively.

If only one fibre path is defined within a ply, it will be treated as a simple shifting method where the fibre angle varies along the reference direction but is constant along the shifting direction without tow width variation. After the calculation process, all element fibre angles on each layer are displayed on automatically created layers, and the user can check the fibre angle distribution of each ply visually. Fig. 7 shows the fibre angle distributions of both cases when a single and multiple fibre paths are defined on a ply. In Fig. 7(b), in order to define the fibre angles of a non-prismatic VAT lamina, it has 3 tow edges defined. In this case, although more than 2 tows should be defined to cover the entire plate in reality, the user can input a larger value for the tow width, which corresponds to the number of tows required to cover the widest part, without adding all interpolated lines to represent every individual tow edge.

3.3. Thickness calculation methods

To consider the case when fibre paths are not simply shifted, a generalised approach that calculates the tow thickness based on the normal distance between two tow paths has been proposed [23]. However, the process-induced defects of AFP processes were not taken into account.

VATMESH provides three options for thickness calculation. The first option ignores the thickness variation, which can be used for verification of analytical solutions. The second option can be used if the fibre paths are identical and simply shifted along a shifting direction. As shown in Fig. 3, if the fibre volume fraction of the
tow is constant within a ply, the thickness of the sheared tow element is only dependent on the shear angle $\theta$, and can be calculated using the Eq. (1) [21]. The third option can be used if multiple fibre paths are defined. As shown in Fig. 8, if the original tow width is adjusted to $W'$, the thickness increases more to keep the cross-sectional area constant, and can be calculated as

$$t' = \frac{t_0 W}{W'} = \frac{1}{\cos \theta} \frac{W_0}{W} t_0$$

where $t_0$, $t$, and $t'$ represent the original tow thickness before shearing, after shearing, and after width adjustment followed by shearing, respectively.

In VATMESH, the above calculations are performed based on defined fibre paths and a reference axis of each ply. Based on Fig. 6, thickness variations can be calculated as

$$t = t_0 \text{ (Op1: Constant thickness)}$$

$$t = \frac{t_0}{\cos(\alpha - \beta)} \text{ (Op2: Based only on shear angle)}$$

$$t = \left(\frac{W_0}{L}\right) t_0 \cos(\alpha - \beta) \text{ (Op3: Based on shear angle and tow width change)}$$

Fig. 9(b) shows the thickness calculation results of a ply with multiple fibre paths shown in Fig. 9(a) according to different thickness calculation options.
3.4. Transformation method of a 2D shell model to a 3D model

In order to accurately consider inter-laminar shear stress or the effects of through-thickness coefficient of thermal expansion such as spring-in, 3D elements are required. The VATMESH provides an additional function to transform a generated 2D shell model into a 3D model. Fig. 10 shows how it creates the nodes for the upper surface of a ply. From the discrete thickness variation of the 2D shell model, a smooth upper surface is created for node connectivity. In the original 2D shell model, the VATMESH detects the adjacent shell elements sharing the same node (e.g. 4 elements for a node within the laminate, 2 elements for a node on the edge), and calculates the average nodal thickness. The node is duplicated and shifted by the calculated thickness along the thickness direction. For the next ply, the thickness distribution of the previous ply is used as a new reference.

A mixed mesh with triangular and rectangular elements can also be processed in the same manner. Although a single element layer is created per ply, the number of elements through the thickness can be adjusted by duplicating layers with the same fibre path in the shell modelling stage. Fig. 11(a) and (b) shows the examples of transformation from a 2D shell layer to a 3D element layer. Fig. 11(c) shows the transformation result of the shell model with a thickness distribution calculated using the third option that is

![Figure 10. Construction of the 3D elements based on discrete thickness distribution of the 2D shell elements sharing a node.](image)

![Figure 11. Transformation of a 2D shell model with multiple fibre paths into a 3D element model: (a) defined fibre paths in VATMESH, (b) converted 3D laminas for each ply plotted in ABAQUS CAE, (c) a 3D lamina converted from the example in Fig. 9 using the 3rd thickness calculation option.](image)
shown in Fig. 9. 3D solid or continuum shell elements can be chosen for the transformation.

3.5. Composing an input file for ABAQUS analysis

Basically, the VATMESH generates a 2D shell model using S4 or S4R elements based on the calculated fibre angle and thickness distribution. The ‘Shell General Section property card is used with the composite option. With this option, the element material orientations and element thicknesses are input as lists by using the ‘Distribution and ‘Distribution Table cards. In order to consider the flat bottom surface and uneven top surface of the VAT plate, the ‘offset option is set to be SNEG which means that the bottom surface of the shell is the reference surface. But if necessary, users can change this option in the created input file manually.

For the transformed 3D model, the ‘Shell Section property and ‘Solid Section property cards are used to create 3D continuum shell and solid models, respectively. Those cards are used together with the ‘Orientation and ‘Distribution cards to assign the rotation angles of the material property coordinates of each element.

When the created input file is imported into the ABAQUS/CAE, all distribution data will be transferred as well. The user can manipulate boundary and loading conditions as well as constraints. By using the ‘Include card, the composed input file can be included in another input file which defines the material properties, boundary conditions, analysis steps and equation constraints if necessary.

4. Extended functions

4.1. Mapping

VATMESH provides some extended functions for the case when an irregular mesh is required or when the model shape is not a simple rectangular plate. One of them is the mapping function that can enable the user to assign the fibre angle and thickness distributions on a shell model created in the ABAQUS/CAE by using the GUI of AutoCAD. The process is shown in Fig. 12. The user needs to create an input file of a 2D shell mesh in the ABAQUS/CAE first to

![Shell mesh created in ABAQUS/CAE](image1)

![Imported mesh and calculated fibre angle distribution in VATMESH](image2)

![Ply1](image3)

![Ply2](image4)

Fig. 12. Mapping function of VATMESH: (a) importing a mesh and calculation of fibre angle and thickness variations, (b) created mesh visualised in ABAQUS CAE.
import it into VATMESH. If there is no geometry on the "0" layer in the AutoCAD, VATMESH automatically requests the ABAQUS input file containing a shell model. After importing the input file, it creates the mesh geometry on a separate layer by reading the nodal coordinates and element connectivity that are used to calculate the centroids of the irregular elements as shown in Fig. 6(b). Subsequently, it starts the calculation of fibre angles and thicknesses based on the fibre paths defined on different AutoCAD layers. The shell model created by VATMESH can also be transformed into a 3D model using the same principle described in Section 3.5.
This mapping function can also be useful for modelling cracks because even the duplicated nodes and the node or element sets assigned in the original input file can be transferred to the result file of VATMESH. In the 2D-3D transformation, the elements around a crack seam element can also be connected smoothly by considering the nodal coordinates rather than node IDs of each element.

Mapping on a 3D curved surface is also available. For example, if a curved 3D shell is created in ABAQUS/CAE and imported to the VATMESH, fibre angle and thickness distributions are calculated on a 2D planar mesh, which is projected to the X–Y plane, based on the fibre paths defined in AutoCAD. Fig. 13 shows the example. However, the 3D transformation function cannot be supported in this case due to its geometric complexity.

However, the 3D mapping function is very limited in that fibre angle and thickness distributions cannot be mapped onto a double curvature surface or a non-prismatic single curvature surface. AutoLISP does not currently support various functions that can calculate geometric features of a 3D complex surface, and has limitations to define paths directly onto it. Without handling an exact 3D model of a complex surface, it is difficult to calculate the location of the next path from a reference one on its discretised surface model. If the exact surface is provided with a path on it, the calculation may be relatively easy for the conventional AFP processes since the surface normal and tangent directions at a point on the path can readily determine the tow width direction that is assumed to be perpendicular to both directions. However, for the CTS process, it is much more difficult because the direction of shift-widths may not be perpendicular to both directions, which is related to 3D head control algorithm that is still under development.

4.2. Ply stacking

The other extended function is for stacking 3D plies with thickness variations, as shown in Fig. 14. For delamination analyses, every node on the ply interface needs to be duplicated and located closely. Fig. 14 shows the case when two layers with different
matrix cracks are stacked together. Firstly, the shell element layers created in ABAQUS/CAE need to be converted into 3D laminas with single-element layers. If the ply stacking function is called, VATMESH requests several input files including these single-element layers. This process produces a 3D solid model with ply interfaces including duplicated nodes where each upper and bottom interfaces are set as independent element surface and node sets automatically. It is useful to connect the interfaces of laminas with thickness variations using cohesive elements in order to do delamination analyses [24].

4.3. Manufacturability

Manufacturing limitations of the CTS process can also be evaluated in VATMESH. At the end of input procedure, it can be asked to provide three manufacturing constraints for maximum shear angle, minimum steering radius and minimum tow width, respectively. Fig. 15 shows an example of a single VAT ply (100 mm × 100 mm) with three different reference paths at the reference angle of zero degree. Since shear angle variation along a path can be different depending on its shifting direction, the shear angles need to be checked as well as the steering radius. When VATMESH displays the shear angle of each element at the end of the process, the elements with shear angles larger than the maximum shear angle given by the user (40° in this example) are displayed in bold, as shown in Fig. 15(a). So the user can easily identify the regions where the defects might occur due to the shear bending induced by the shear locking behaviour of the tow [21]. Especially when multiple reference paths are defined, VATMESH can generate the interpolated lines between two adjacent reference paths. The maximum distance between the two paths, which is measured along the shifting direction, is divided by the default tow width, and the result is rounded up in order to calculate the required number of tows in integer value. After generating the interpolated paths, the regions with the local radii smaller than the minimum steering radius set by the user (50 mm in this example) are displayed with bold lines on a separate layer, as shown in Fig. 15(b). For each path, a point with the minimum local radius, if it is smaller than the minimum steering radius, is indicated as well. In the case that only a single reference path exists for a certain region, the path is simply shifted by the default tow width multiple times along the shifting direction towards the plate edge. When the tow width can be controlled in the CTS, the limitation to reduce the tow width can be considered. On a separate layer, the regions with the tow widths smaller than the minimum tow width set by the user (3 mm in this example, which is 50% of the default tow width 6 mm) are displayed in bold, as shown in Fig. 15(c), so that the user can easily check which regions cannot be manufactured without overlaps. Since the data for the three types of manufacturing limitations is automatically stored on separate layers in the AutoCAD, the user can check those individually by turning on and off each layer.

5. Conclusion

In this work, a computer-aided modelling tool, VATMESH, for variable angle tow composite laminates manufactured by the continuous tow shearing (CTS) technique was developed using the AutoLISP programming language for AutoCAD. By combining various geometry calculation functions of AutoLISP with geometry creation functions of AutoCAD, it provides a convenient way to create a finite element ABAQUS model of a composite structure with non-linear fibre angle variation that cannot be formulated mathematically. It also provides some extended functions such as 2D–3D transformation, mapping and stacking functions to help the composite designers to create more complex models easily for fracture, damage tolerance, and delamination analyses. Furthermore, the manufacturing limitations such as maximum shear angle, minimum steering radius and minimum tow width can be visualised so that the regions where the defects might occur can be identified before manufacturing. The VATMESH is an efficient tool for development of VAT composite products in that it can generate both inputs for manufacturing and analysis in a single software platform. However, further development is required for its applicability to 3D complex surfaces in conjunction with the development of a head control algorithm specialised for the CTS process.

Acknowledgements

This work was carried out with the support of the EPSRC (Engineering and Physical Sciences Research Council, UK), Airbus, and GKN under the project ABBSTRACT2 (EP/H025898/1).

References