Modelling complex geometry using solid finite element meshes with correct composite material orientations

Jiye Chen^{a,*}, Stephen Hallett^b, Michael R. Wisnom^b

^a Faculty of Technology, University of Portsmouth, Portsmouth PO1 3AH, UK

^b Advanced Composites Centre for Innovation and Science, University of Bristol, Bristol BS8 1TR, UK

ARTICLE INFO

Article history:

Received 10 August 2008

Accepted 2 February 2010

Available online 23 February 2010

Keywords:

Doubly curved laminates

Fibre orientation

Extrusion

FE modelling

ABSTRACT

This article addresses the requirement to generate solid hexahedral finite element meshes for modelling composite components with complex geometric shapes. Standard commercial finite element pre-processors are not capable of unambiguously defining the material orientations in such situations. This has necessitated the development of a purpose written pre-processor, which extrudes solid elements with element orientations that accurately match complex doubly curved geometries. An efficient methodology for the creation of the mesh has been outlined through the combined use of the dedicated software and MSC PATRAN. An example presented shows that this technology has been successfully used to model a representative vane from a gas turbine engine.

Crown Copyright © 2010 Published by Elsevier Ltd. All rights reserved.

1. Introduction

1.1. The need for three dimensional meshes and finite element analysis

It is well known that fibre based composite materials have good in-plane strength but weak through-thickness properties [1–8]. Many failures such as delamination [12–14] in fibre composite laminates are caused by high interlaminar stresses. A number of particular investigations can be seen in Refs. [4,11,15,16], which shows that interlaminar stresses caused delamination or debonding in curved components, the main failure mode in humpback specimens [4], implanted femoral stems [11] and T-piece specimens [15,16]. Prediction of interlaminar stress is therefore crucial in the design of such curved composite components. It is essential that finite element modelling for design of complex composite structures is capable of accurate prediction of through-thickness stresses. To meet this requirement, 3D finite element meshes are needed in the analysis since shell meshes can only be used for predicting in-plane strength.

1.2. Current capabilities of pre-processors

Generally, there are three coordinate systems available for the definition of material orientation in finite element codes. Rectangular coordinate systems can easily define isotropic material orientation in any solid body. For solids with simple curvature,

cylindrical coordinate systems can define their orthotropic material axes. For solid bodies with doubly curved shapes the above two coordinate systems cannot accurately define orthotropic material orientations. Since the material properties in the fibre direction are significantly different to transverse or through-thickness properties, it is necessary to accurately define the material orientations of any solid mesh of a composite component. For some components with complex geometric shapes this is additionally complicated in the case of doubly curved surfaces since a single global coordinate system will not be sufficient. Material orientations should therefore be relative to the local element orientations. Using a number of local coordinate systems will increase accuracy but if curvature is significant it will be necessary in the limiting case to have as many coordinate definitions as there are elements. Therefore using element coordinate systems to define material orientation in curved bodies is the best approach, thus necessitating careful control of the mesh to be used in any analysis.

Most commercial finite element solvers have elemental coordinate systems for defining material orientations. For example, MSC NASTRAN and ABAQUS have element coordinate systems in 2D and 3D meshes [9,10]. The use of local element coordinate systems in shell meshes is well established and very effective. It is however, significantly more difficult to define material orientation in 3D meshes by using local coordinate systems since this requires very accurate control of the meshing procedure. This is particularly true when the domain is a multiple curved body. The accuracy of stress predictions will be directly affected by the accuracy of material orientations. The level of inaccuracy can be quite significant due to the highly anisotropic nature of material properties, particularly

* Corresponding author. Tel.: +44 02392842427.

E-mail address: jiye.chen@port.ac.uk (J. Chen).

between in-plane and through-thickness directions. To overcome the difficulties in creation of solid element meshes with accurate material orientations a purpose written pre-processor for curved composite components has been developed. This pre-processor uses shell meshes with correct element orientations on the surface of the body as the starting point as these can currently be easily and accurately defined. The solid element mesh is then extruded from this based on the shell orientations.

1.3. The extrusion of solid element meshes

The principal idea behind the extrusion of the solid mesh is based on existing shell meshes created by any commercial package. MSC PATRAN has been used in this case. In a shell element mesh each grid or node has an average normal calculated from those elements connected to it. New grid points are created for the solid elements by extruding along such a normal for a given distance, thus defining the new coordinate positions. Fig. 1 shows a shell element with four nodes, c1 to c4. $\vec{a}$ is the vector from c1 to c3, and $\vec{b}$ is the vector from c2 to c4. The normal of this shell $\vec{N}$ can be calculated as the cross product of these two vectors. Positive normal, $\vec{N} = \vec{a} \times \vec{b}$ and negative normal, $\vec{N} = \vec{b} \times \vec{a}$ as shown in Fig. 1.

Fig. 2 shows a mesh with four shell elements and their respective normals. The averaged normal $\vec{N}$ at node j can be obtained by

$$\vec{N}_j = \sum_{i=1}^m \vec{N}_i / m \quad (1)$$

where, m is the number of elements connected to node j , $\vec{N}$ is normal of element i connected to node j . The extruding distance vector, $\vec{D}_j$ at node j can be formed as below.

$$\vec{D}_j = T_j \cdot \vec{N}_j \quad (2)$$

where, T_j is the extruding distance at node j .

Using the above principle and giving a constant extruding distance at each node of the mesh shown in Fig. 2, a solid mesh can be extruded (Fig. 3). This is the methodology which has been used to create well controlled solid element meshes with correct local element orientations in the analysis of composite components with complex shapes.

2. Application to doubly curved bodies

2.1. Simple shapes to highlight problems

Fig. 4a shows a curved body with three sections a, b and c. The material orientation in sections a and c can be determined by using local cylindrical coordinate systems individually since one rotated coordinate can express the location of such singly curved bodies. Section b however, has a double curvature. More generally, this

Fig. 1. A shell element with normal.

Fig. 2. Geometry and orientation of a mesh with four shell elements.

Fig. 3. Geometry and orientation of extruded solid elements together with original shell elements.

Fig. 4a. A curved body.

section can be freely curved and is not necessarily determined by a spherical coordinate system. In the absence of being able to use accurate local element coordinate systems, for numerical estimation, this section can be divided into several subsections. Each subsection can be given a local orthogonal coordinate system at certain positions, for example at the centre as shown in Fig. 4a. In the finite element analysis the material orientation of all elements within one subsection will be determined by one local coordinate system. The accuracy of estimation increases as the number of subsections increases. The best case is when the number of subsections equals the number of elements in the doubly curved domain. This leads to the use of individual element coordinate systems for the entire mesh. Fig. 4b shows an entire mesh with element coordinate systems, which has the material orientation following the varied geometry. Extruding vector at each node in this generated mesh was principally given in the Section 1.3 and graphically shown in Figs. 1 and 2.

Fig. 4b. Element coordinate systems in the mesh.

2.2. Pre-processor

Fig. 5 shows a more realistic example, a T-piece specimen similar to the shapes one might encounter in actual component design. This can be broken down into three significant regions; the aerofoil section, the radius region and the platform region. For each section a bespoke solution is required to generate an accurate finite element mesh of the whole component. The height of T-piece specimen is 170 mm, the thickness 9 mm and 6.7 mm are given at the

leading and trailing edge of aerofoil section, respectively, the most thickness at the middle of aerofoil section is taken as 20 mm, the length of platform is 200 mm, the width and thickness of platform are assumed as 150 mm and 20 mm, respectively. Each section presented in Fig. 5 is doubly curved except the platform of T-piece.

2.2.1. Extruding approach for aerofoil section

An aerofoil is a special case of a doubly curved body, varying in curvature and thickness both in the chordwise and spanwise directions. Fig. 6 shows a theoretical cross-section, developed here for demonstration of the pre-processor capabilities only. Since the upper and lower surfaces will have different chord lengths, if an equal number of elements are placed on each surface and extruded inwards, normal to the outer surfaces, a congruent mesh will not be formed on the centre-line. It is therefore necessary to start from the centre-line and to extrude outwards to the aerodynamic surfaces. This is done from a shell mesh on the mid-surface of the vane. Each point on this surface will have a different extrusion distance extracted from the actual vane geometry. Extrusion vector is also varied at each point on this surface, which is determined by the way given in the Section 1.3. In the example presented in Fig. 5, the spanwise section is constant for simplicity, but this need not necessarily be the case. In practice, both ends of aerofoil section in Fig. 6 will be cut off because there are no fibres in the sharp area. This treatment was used for the T-piece specimen in Fig. 5 and it does not affect the mechanical analysis of fibre composites actually. By this way, extrusion with hex elements can go through-thickness between the upper and lower surfaces of aerofoil to meet above extruding purpose.

Fig. 5. A T-piece specimen.

Fig. 6. An aerofoil section.

Fig. 7. Radius region and deltoid.

Fig. 8. Base of aerofoil section and radius region.

2.2.2. Extruding approach at the base of aerofoil section

The aerofoil section divides into two parts at the base. This forms doubly curved sections on both the convex side and concave side. The solid between two radius regions is called the deltoid (shown in Fig. 7) and is typically filled with uni-directional material. In the radius region, extrusion is necessarily from the outer geometric surface through a distance defined by the thickness at the end of the aerofoil section, since the concept of a “mid-plane” no longer exists (see Fig. 8). This process creates solid elements for the radius regions where the accuracy of definition of fibre direction is paramount in finite element analysis since this is a region which sees high through-thickness stresses and is a likely area for failure to initiate.

2.2.3. Platform region

The material from the aerofoil section which is carried through the radius regions continues into the platform. The generation of this part of the mesh is done in a similar way as for the aerofoil

section, starting at the outer surface and extruding through a known distance. The remainder of the platform mesh is formed by extruding from the base of this existing mesh to the lower free surface as shown in Fig. 9. This is necessary to complete the platform to uniform thickness since the material from the vane extensions will still have a variable cross-section in the chordwise direction.

Fig. 10 gives a flow chart to demonstrate the process of producing solid meshes of such typical components. The output, the completed mesh is shown in Fig. 11 with the deltoid mesh for the deltoid shown in Fig. 7 being completed in MSC PATRAN, which principally uses some standard extruding or sweeping techniques [17–19]. It should be noted that the hexahedral solid elements in the deltoid have the correct orientations, which follow the curved body at the base of the vane. However, the element orientations of the wedge solid elements at the apexes of the deltoid region, which are unavoidable as worst elements extruded by MSC PATRAN, do not comply with this rule. This is likely to be a resin rich region without reinforcing fibres and therefore can be given isotropic properties.

The quality of entire extruded meshes shown in Fig. 11 was assessed using MSC PATRAN which gives a range of element aspect ratio, between 1.68 and 13.4, of extruded solid elements. Geometrically, ratio 1 is perfect, and it becomes worse when ratio increases. There is no mesh distortion found in this investigation if ratio is less than 15 which could be accepted as a limit of the quality measure in this case. This assessment in average shows a good quality of generated meshes for this particular geometry.

3. Application

3.1. Meshing representative geometry using commercial package only

In order to investigate the effects of material orientation, a second mesh of the representative geometry has been created entirely by the commercial package MSC PATRAN in this case. Since it is not possible to completely and unambiguously define and control the elemental coordinate systems, a series of local coordinate systems was required to be used instead. It is thus necessary to divide the relevant domain into several sub-regions or sub-domains, which reference different coordinate systems as shown in Fig. 12. Three rectangular local coordinate systems are created to define material orientations in each of the aerofoil section, deltoid and radius regions. One local coordinate system is used in the platform region. Fig. 12 also presents the radius region on the convex side with three rectangular local coordinate systems to estimate material orientation. A similar treatment is used on the concave side. It should be noted that a cylindrical coordinate system is not appropriate in the definition of the above geometry as the shape has fully free curved characteristics. An entire mesh of this second model is shown in Fig. 13, which has the same mesh density as that of model 1 shown in Fig. 11. It should be noted that the local coordinate z shown in Fig. 12 indicates the uniform direction for deltoid region.

Fig. 9. Schematic view of platform.

Fig. 10. Flow chart of pre-processor.

Fig. 11. Extruded solid mesh of entire vane showing local element orientation and solid mesh of deltoid.

Fig. 12. Geometry with local coordinate systems.

Fig. 13. Mesh of second model.

3.2. Modelling analysis and comparisons

To investigate the effects of variations in material orientation, analyses based on the models shown in Fig. 11 (with correct elemental coordinates) and Fig. 13 (with domains each referencing a series of local coordinate systems) are created as model 1 and model 2, respectively. In both cases they are loaded by a prescribed vertical displacement of 0.01 mm upwards at the top of aerofoil section. A fully fixed restraint was applied to both ends of the platform. The majority of the elements are eight noded hexahedral elements with some six noded wedge elements being necessary at the edges of the deltoid region. Table 1 gives material properties used which, since these analyses were for demonstration purposes only, were taken as uni-directional prepreg from Ref. [12].

The deformed shape and maximum values of displacement appears of both models are very similar results because two models

were given a same displacement. However, the normal and shear stresses in the through-thickness direction are different, which are most relevant for the prediction of failure. Fig. 14 gives the through-thickness normal stress component σ_{zz} in the aerofoil section and radius region. From this we can see significant differences not only in maximum values but also in the location of stress concentration. Model 1 gives the location of high values of stresses in the middle of the radius region, which is one of initial failure areas according to experimental tests of similar components [15,16].

More detail of the σ_{zz} distribution at the mid-chord section of the vanes can be seen in Fig. 15. Model 1 gives a smooth transition between the regions, but model 2 produces a high stress concentration on the top surface at the transition between the radius and platform. This can be understood as in model 2 there is a mismatch in the fibre direction as the radius region connects to the platform because there is only one local coordinate system used in the radius region. The two local coordinate systems used are shown in Fig. 15, one in the radius region, and the other in the platform.

Fig. 16 presents the through-thickness shear stress component, σ_{xz} . The difference between the two models regarding maximum value and location of stress concentrations can also be seen to be significant from this figure, with model 2 giving a peak stress more than double that of model 1. The stress concentration of both σ_{zz} and σ_{xz} produced by model 2 is located at the same transition between the radius and platform. This is a direct effect from the limited number of rectangular local coordinate systems used in the radius region. The stress is very high on the surface of model 2, whereas it should go to zero since it is a free surface. This effect should be reduced if more local coordinate systems are used, particularly in the two curved sections of the radius region. Model 2a with 27 sub-regions in the radius region was created to study the convergence of stress. The division of sub-regions can be seen in Fig. 17 which requires 27 local coordinate systems. Fig. 17 also shows σ_{zz} in the radius region which has a maximum value closer to that obtained from model 1 than model 2. The location of the stress concentration has changed from the radius-platform interface as in model 2 to a similar location to model 1. This improved solution indicates that the effect of a limited number of local coordinate systems reduces when the number of local coordinate systems increases in the radius region. The value of maximum stress predicted is, however, still significantly different although converging on the values for model 1 as can be seen in Table 2.

Table 1
Material properties.

E_{11}	E_{22}	E_{33}	G_{12}	G_{23}	G_{31}	ν_{12}	ν_{23}	ν_{31}
139 GPa	9.5 GPa	9.5 GPa	5.4 GPa	3.7 GPa	5.4 GPa	0.32	0.5	0.02

Fig. 14. (a) σ_{zz} (MPa) in model 1. (b) Max σ_{zz} (MPa) in model 2.

Fig. 15. (a) σ_{zz} (MPa) in model 1. (b) σ_{zz} (MPa) in model 2.

Fig. 16. (a) σ_{xz} (MPa) in model 1. (b) σ_{xz} (MPa) in model 2.

Fig. 17. σ_{zz} (MPa) in model 2a.

Table 2
Selected results.

Model type	Max displacement (mm) on the top of aerofoil section	Max σ_{zz} (MPa)	Max σ_{xz} (MPa)
Model 1	0.01	28.2	25.6
Model 2	0.01	42.3	57.2
Model 2a	0.01	31.5	54.0

4. Summary

This paper has shown the significant effects of insufficiently accurate material orientations in the finite element analysis of composite components with complex geometry. A simplified representation of actual components has been used for demonstration purposes. Significant differences were found between the accurate model 1 using the new extrusion approach and the standard model 2 because of the limited number of local coordinate systems used. The improved model 2a using an increased number of local coordinate systems gave results much closer to those given by model 1. When the number of local coordinate systems equals the number of elements, model 2a will have the same solution as model 1. This shows that it is necessary to use element coordinate systems to define material orientation in the analysis of geometrically complex composite components. The purpose written pre-processor for composite materials introduced in this paper successfully achieves the correct material orientation in the analysis of such composite blade bodies with doubly curved shapes. It should be noticed that

this technique has not been tested in more complicated extrusion problems. At the end, it should be noticed that the aero engines industry has benefited huge time saved (roughly the time for creating model 1 is less than 1% of the time for creating model 2, and the best model 2 needs much longer time actually) in their practice using the technique presented in this paper.

Acknowledgement

The authors would like to thank Rolls-Royce Plc for funding this research.

References

- [1] Wisnom MR, Jones MI. Size effects in interlaminar tensile and shear strength of unidirectional glass fibre/epoxy. *J Reinf Plast Comp* 1996;15:2–15.
- [2] Wisnom MR, Jones MI, Hill GFJ. Interlaminar tensile strength of carbon fibre-epoxy – specimen size layup and manufacturing effects. *Adv Compos Lett* 2001;10(4):171–7.
- [3] Jackson WC, Martin RH. An interlaminar tensile strength specimen. *Compos Mater* 1992;11:333–54.
- [4] Wisnom MR, Petrossian ZJ, Jones MI. Interlaminar failure of unidirectional glass/epoxy due to combined through thickness shear and tension. *Compos Part A* 1996;27A:921–9.
- [5] Cui GY, Ruiz C. Through thickness failure of laminated C/epoxy composites under combined stresses. *Compos Sci Technol* 1995;53:253–8.
- [6] O'Brien TK, Chawan AD, Krueger R, Paris IL. Transverse tension fatigue life characterization through flexure testing of composite materials. *Fatigue* 2002;24:127–45.
- [7] Wisnom MR, Jones MI. Through thickness fatigue failure of fibre-reinforced composites. *Aeronaut J Roy Aeronaut Soc* 1998;102:83–8.
- [8] Wisnom MR. 3-D element analysis of curved beams in bending. *Compos Mater* 1996;30(11):1178–90.
- [9] MSC Nastran user manual, MSC. Software Corporation, 2 MacArthur Place, Santa Ana, California 92707, USA.
- [10] ABAQUS User Manual, Hibbitt, Karlsson & Sorensen Inc., 1080 Main Street, Pawtucket, RI 02860-4847, USA.
- [11] Chen J, Browne M, Taylor M, Gregson PJ. Application of an interface failure model to predict fatigue crack growth in an implanted metallic femoral stem. *Comput Meth Prog Biomed* 2004;73(3):249–56.
- [12] Chen J, New A. Application of decohesive model with mixed damage scale in fracture analysis of composite materials. *Fatigue Fract Eng Mater Struct* 2001;24(11):761–9.
- [13] Chen J. Predicting progressive delamination of stiffened fibre-composite panel and repaired sandwich panel by decohesion model. *Thermoplastic Compos Mater* 2002;15(5):429–42.
- [14] Chen J, Crisfield M, Kinloch AJ, Busso EP, Matthews FL. Predicting progressive delamination of composite material specimens via interface elements. *Mech Compos Mater Struct* 1999;6(4):301–17.
- [15] Chen J, Ravey E, Hallett S, Wisnom M, Grassi M. Prediction of delamination in braided composite T-piece specimens. *Compos Sci Tech* 2009;69:2363–7.
- [16] Chen J. Simulation of multiple-crack growth in braided composite T-piece specimens using cohesive models. *Fatigue Fract Eng Mater Struct* [accepted for publication].
- [17] Mingwu L, Benzley SE, White DR. Automated hexahedral mesh generation by generalized multiple source to multiple target sweeping. *Int J Numer Meth Eng* 2000;49:261–75.
- [18] Roca X, Sarrate J, Huerta A. Mesh projection between parametric surfaces communications in numerical. *Meth Eng* 2006;22(6):591–603.
- [19] White DR, Saigal S, Owen SJ. CCSweep: automatic decomposition of multi-sweep volumes. *Eng Comput* 2004;20:222–36.