Modelling of polymeric fibre-composites and finite element simulation of mechanical properties

Robert A. Shanks
CRC for Polymers, Applied Sciences, RMIT University
GPO Box 2476V Melbourne Victoria
Australia
robert.shanks@rmit.edu.au

Summary

Polymer composites are formed as random or anisotropic fibre dispersions, sheets with long fibre felted or woven fabrics and laminates of such sheets. A thermoset polymer in liquid form is mixed with the fibres and then cured. Thermoplastics must be intermingled with the fabric and the matrix phase united by melting of the thermoplastic and compaction of the composition. The composite sheets are thermoformable by virtue of the thermoplastic matrix phase. These composites present many compositional, structural and processing variables that contribute to properties. These composites are less uniform than typical thermoset resin composites where fibres are wet with a liquid resin that is then solidified by chain extension and crosslinking. Typical characterisation and mechanical performance tests are available to investigate and optimise the composites. Finite element analysis (FEA) enables a theoretical approach to understanding of the structure–property relationships and confirmation of interpretation of measured properties. A displacement field is suited to identifying and quantifying stress intensities in local regions of the composite to determine parameters critical to the performance of the composites. This chapter reviews the application of FEA to various composite types, stress situations and failure mechanisms. The FEA model design and simulation method are evaluated and compared.

1. Introduction

1.1 Types of composite

Composites based upon thermoplastics such as polypropylenes, polyethylenes, poly(ethylene terephthalate), poly(butylene terephtahate), various polyamides, polystyene and its copolymers with butadiene and acrylonitrile, poly(vinyl chloride) thermoplastic polyurethanes, thermoplastic elastomers and biopolyesters such as poly(lactic acid), poly(hydroxybutyrate) and its copolymers with hydroxyvaleric acid. Thermosetting polymers include epoxy resins, unsaturated polyesters, vinyl esters, epoxy-acrylates, polyurethanes, polyisocyanurates, polybismaleimides, polysiloxanes, formaldehyde based

resins such as phenolic, melamine and urea, and many synthetic elastomers. Epoxy resins are the most commonly studied polymers using finite element methods and they are frequently cited as examples in this chapter.

Fibrous reinforcements include cellulosic fibres such as flax, hemp and others, glass fibres, carbon fibres, mineral fibres, synthetic fibres related to the matrix polymer. Table 1 shows Common fibre reinforcements with brief comments. Short fibres are chosen for direct addition to a polymer in extrusion or injection moulding. Common short fibres are chopped glass and carbon fibres, wood flour and other plant derived cellulosic fibres. Long fibres may be used in pultrusion, woven mats or felted (non-woven such as prepared by needle punching) mats. Mats require inclusion of polymer so that composites can be prepared by thermoforming since the structure of the mat must be maintained. Nano-fibres are subject to much recent investigation and they are of increasing commercial importance, such as carbon nano-tubes, microcrystalline cellulose or nano-cellulose, boron nitride and alumina whiskers. The composites discussed mainly include fibres, however other fillers such a glass spheres and particulate or platelet shapes are studied using FEA since they can be represented in two- or three-dimensional models of polymer composites. Polymer can be included as another fibre or as a powder, that are subsequently melted to form a uniform matrix between and around fibres, as a plastisol (polymer dispersed in plasticiser) that is thermally gelled to form a solid matrix, or as a chain-extendable or cross-linking pre-polymer. Sometimes a third component is included as an interphase surrounding the dispersed phase or when the layers of a laminate have different properties. A more complex model may include a density gradient within the matrix, often to distinguish surface layers from the interior.

Fibre	Composition	Description
Glass fibre	E-glass or S-glass	Chemically resistant, with size coating, need coupling agent
Carbon fibre	Graphite (formed by polymer pyrolysis)	High modulus and strength
Carbon nanotubes	Graphite cylinders	High modulus, strength, conductivity
Kevlar	Poly(phenylene terephthalamide)	High modulus / high strength types
Bast fibre (hemp, flax, rami)	Cellulose	Native or textile crystals, moisture sensitive, long fibres can be woven
Wood fibre or flour	Cellulose	As above, short fibres
Nano-cellulose	Partially hydrolysed cellulose	Crystalline perfection, high properties
Polymer fibres	Polypropylene, Polyamides, poly(ethylene terephthalate)	Moderate modulus and strength, used in special purpose composites
Mineral fibres	Rock wool, boron nitride, alumina	High performance

Table 1. Fibres for reinforcement of polymer composites

The thermoplastic composites under consideration contain long fibres that are structured by felting, typically using a needle-punch method, or woven. The composites are planar, with application as sheets or panelling, such as in automotive liner use or in building panels. The composite sheets are thermo-formable because the matrix is a thermoplastic. In automotive application they are formed into the many complex shapes required by the interior shape and contours. Fibre weave and/or entanglement will increase tensile strength across the plane of the composite. Fibres crossing the plane in either felted or woven composites can increase the interlayer strength thus preventing peeling of layers from sheets. The composite sheets can be laminates with one or two decorative or protective layers. In automotive use the lamination is typically an upholstered, leather-like or wood-like finish. The thermosetting composites may be readily formed into any shape, however the shape cannot be changed after formation, in contrast to thermoplastic composites.

1.2 Composite properties

Modulus that is orientation dependent and calculated as a volume fraction weighted mean of the matrix (polymer) and filler (fibre) in series or parallel (Equ. 1, where V = volume fraction, E = modulus, c = composite, f = fibre, m = matrix). An efficiency factor (g) is usually included to account for interfacial interaction and variables such as voids, inefficient dispersion of fibre bundles and variations in fibre alignments, etc (Equ. 2). The modulus can be measured using a universal test instrument in tensile mode (parallel with fibre orientation) that emphasises the properties of the fibres. The modulus measured in shear mode emphasises the matrix-fibre interface, while in flexure mode a combination of matrix and fibre modulus is measured, with the upper bended surface in tension and the lower surface in compression (Fig. 1). The Von Mises stress contour diagram (Fig. 1, lower image) shows a stress maximum at the fixture end and a compressive stress maximum in the lower centre surface. The same beam in three-point bend mode emphasises the stress in the load region with compressive stress concentrated near the upper surface and tensile stress concentrated near the lower surface (Fig. 2). An important observation for composite design is that the centre of the beam cross-section is relatively stress free.

$$E_c = V_f E_f + V_m E_m = V_f E_f + (1 - V_f) E_m \qquad (1)$$

$$E_c = g V_f E_f + V_m E_m = g V_f E_f + (1 - V_f) E_m \qquad (2)$$

Where V_f can be calculated from the length (l) and diameter (d) of fibres and their logitudinal (L) and lateral (S) spacing in the composite.

$$V_f = \frac{\pi l d^2}{4 L S^2} \qquad (3)$$

The Halpin, Tsai and, Kardos composite model (Equ. 4 and Equ. 5) is a refinement that contains a geometric fitting parameter, A, where A = 2(l/d) for tensile configuration with E as the tensile modulus, and the aspect ratio (l/d) of length (l) and diameter (d). A numerical solution is obtained for Equ. 4 and Equ. 5 to model a composite modulus.

$$E_c = \frac{E_m(1 + ABV_f}{1 - BV_f}$$

(4)

Where:

$$B = \frac{\dfrac{E_f}{E_m} - 1}{\dfrac{E_f}{E_m} + A}$$

(5)

Fibre composites do not display a yield strength, unlike the matrix thermoplastic, since fibre pull-out and fibre fracture occur intermittently until fracture. Progressive fragmentation of a composite is not well simulated by FEA since the composite structure must be changing to represent structural rupture. Simulation of modulus is made while the composite is still coherent. Measurement of composite mechanical properties provides an overall modulus without information on the contribution of components and variables within the structure. Three assumptions are made: (a). The fibre-matrix interface has perfect adhesion; (b). The fibre and matrix exhibit an elastic response to stress; (c). No axial load is transmitted through fibre ends.
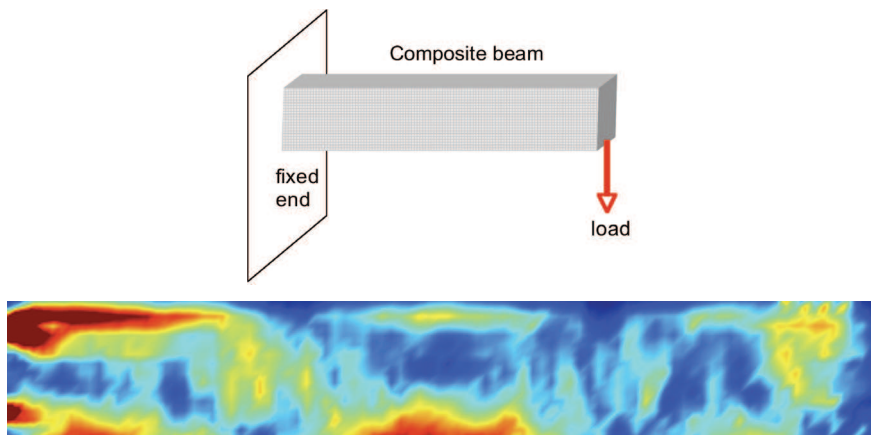

Fig. 1. Supported composite beam with deforming end load (upper) and a Von Mises stress contour plot showing the stress distribution (lower).
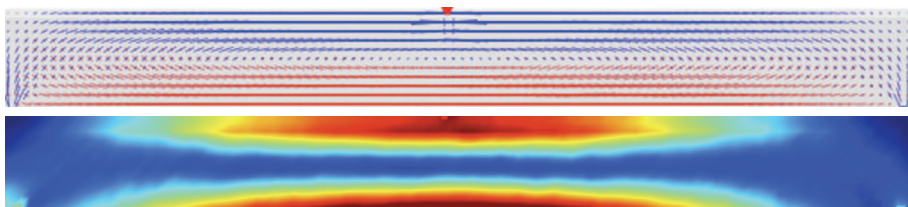

Fig. 2. Beam under three-point bend stress showing compressive stress near upper surface and compressive stress near lower surface, stress distribution (upper image) and Von Mises stress contours (lower image).

FEA requires a proposed model of the composite including the location of fibres within the matrix. The model can be validated by comparison with cross-sectional observation of the composite using optical or scanning electron microscopy. The model should correspond to the actual composite. Inclusion of interfacial interactions within the model is a hypothesis since the interfacial properties cannot be observed. A third phase of strongly adsorbed and immobilised polymer on the surface of fibres can be included to account for interactions (Fig. 3, upper image). The modulus and volume or thickness of this interphase must be estimated as between that of the matrix and fibres. The interphase will be diffused into the continuum matrix. A gradient of modulus can be used instead of a discrete interphase (Chen & Liu, 2001). The nature of the gradient could be linear or non-linear (such as exponential) from the fibre modulus to the matrix modulus. Complex geometry models can be formulated to include various specimen shapes, fibre orientations, woven bundles, and even random voids. The complexity of the model will determine the lowest scale that can be analysed while being representative of the whole composite (Vozkova, 2009). An example of a composite with an enlarged interphase is shown in Fig. 3 (lower image), where the simulation was an application of three-point bend stress. Complex stress fields are associated with the dispersed phase. This type of model is best studied by choice of a single particle or fibre when stresses associated with the interphase are to be considered (see later examples). The concept has been applied to the visco-plastic matrix behaviour of metal composites that reveal similar mechanics to polymer based composites (Shati, Esat, Bahai, 2001).
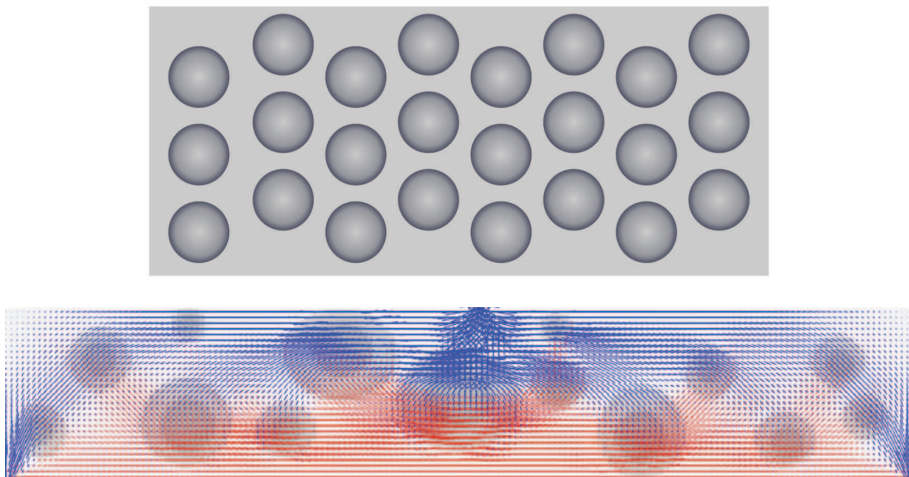

Fig. 3. Fibre composite cross-section schematic showing interphase (darkened) around each fibre (upper) and the lower image shows stress distribution in similar model under three-point bend stress.

The model is then tested by performing a simulation, that is disturbing the model with stress and monitoring the evolution of the strain over time, or applying a strain and monitoring the stress over time. The evolution of the model with time is computed at each node of the finite elements. Application of stress or strain requires that part of the model be fixed. Typically for a beam one end will be fixed and the external stress or strain applied at the other end, for tensile or flexural

(single cantilever bend) testing. Alternatively two ends can be fixed and the stress or strain applied in the centre, equivalent to a dual-cantilever bend. A three-point or four-point bend requires that the ends allow lateral movement or slippage, which is more complex to simulate in comparison to a real situation.

The hypothesis is that the model will behave the same as the real composite. If the hypothesis is correct then the behaviour of the composite will be well understood. If the hypothesis is incorrect them the model needs to be refined and further simulated until the data for the model and real composite converge. Refinement of the model may require inclusion of more variables to better represent the real situation. Unlike a model the real composite is likely to deviate from an ideal structure. Imperfections can be included as discrete entities such as voids, or as a general decrease in the model parameters analogous to an efficiency factor.

An analogy to an anisotropic long fibre composite is reflected in the model of a group of storage silos. Clusters of circular cylindrical structures are found in silos, chimneys and water towers. They are subjected to flexural stress by wind induced oscillations. The cylinders deform in similar fashion to a continuous fibre composite. Five rows of eight silos were modelled using ANSYS with the modulus, Poisson ratio and density as inputs using a harmonic two-node element. The model was validated from data measured in a high wind from ten accelerometers attached to the silo walls. The data revealed harmonic frequencies of the silos relative to the wind velocity (Dooms et al, 2006). The model is structurally similar to a polymer–fibre composite that can display a dimension dependent resonant frequency under modulated force mechanical testing.

1.3 Composite morphological structures investigated using FEA

Important considerations for the design of composites that are assessed using FEA are: The bulk mechanical properties of the composite are determined by the modulus of each phase and their respective volume fractions. Consolidation of the composite is complete and the composite density is a volume fraction average of the component densities. Where consolidation is incomplete this can be represented by a density gradient across the thickness of the composite sheet. Fibre concentration and fibre diameter determine the distribution of fibres since a fixed volume fraction of fibres may be due to few of large diameter or many with small diameter, with a varying total fibre-matrix surface area. The fibre–matrix interface may be implicitly defined such that dewetting or voiding can occur, or be omitted so that the interface will remain intact. Fibre orientation can be included in the model but this will require a larger mesh size to generate sufficient distribution. Fibres are generally included in parallel orientation in models constructed since a random distribution is difficult to model unless the model is large (Fig. 4). The surface versus bulk properties often vary in composites: there may be matrix rich or fibre rich surfaces, with density gradients from the surface to the bulk
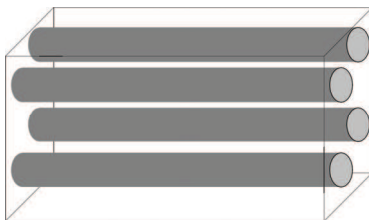

Fig. 4. 3D model of a oriented long fibre composite

A 2D model similar to the fibre composite of Fig. 4 is shown in Fig. 5. The fibres are shown from an aligned view so in the 2D perspective they could be particles. Other similar composite models are shown later in this chapter where the fibres are aligned with the longitudinal axis of the beam. This view of fibre ends is best for observing the stress field throughout the composite and because stress transfer between matrix is better tested laterally for long fibres. In this case the stress is in three-point bend configuration pulling downwards from the right side, while the left side is fixed. The lower section of the beam is in compression, while the upper section is in tension. Some stress concentrations are visible between fibre ends. The regions of stress concentration are between fibres rather than in surface regions; compare Fig. 5 with included fibres with Figs 1 and 2 without fibres, where the stress concentrations emanate from along the surfaces. Stress concentrations between the fibres mean that fracture is likely to initiate at fibre–matrix surfaces or within the matrix between fibres. Strong interfacial bonding and a strengthened interphase will contribute most to a composite in this circumstance. The maximum stress concentrations, both tensile and compressive are associated with the fixed end of the beam, away from the location of application of the stress. Stresses in the compression zone along the lower part of the beam have concentrated where fibres ends are juxta positioned downwards to the left, even though the model was constructed with the aim of randomising the fibre–end positions and diameters. The Von Mises stress plot is not shown because the colour shading obscured the circular fibres end.
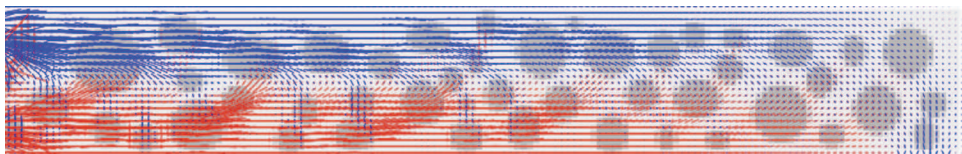

Fig. 5. Composite beam viewed lateral to fibres, fixed on left with bending stress acting down from the right.
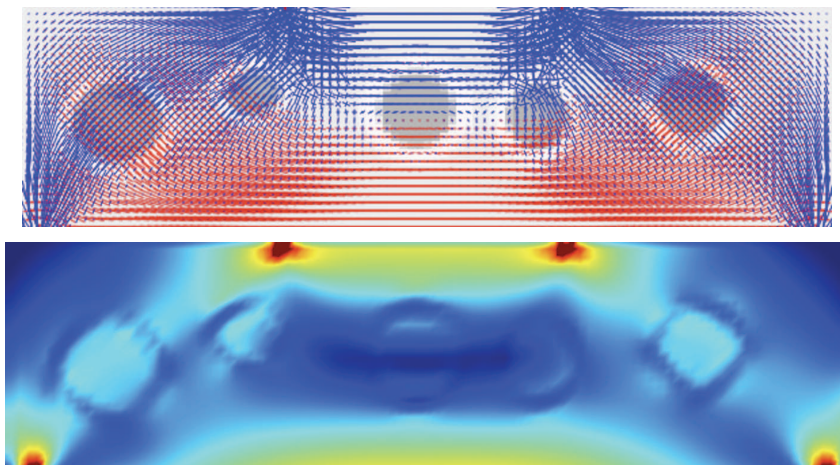

Fig. 6. A simplified model of a polymer with a soft interphase surrounding each hard fibre or particle under four-point stress, stress distribution diagram (upper) and Von Mises contour plot (lower).

A simplified model based on Fig. 5, with fewer hard inclusions surrounded by soft interphase, is shown in Fig. 6. The upper stress distribution diagram shows the compressive stresses passing from the application point through to the supported lower ends. The stress is not concentrated in the vicinity of the fibres, where the lighter shaded areas around each fibre is the soft phase. The lower central section of the beam is under a tensile stress situation. The lower Von Mises contour plot shows the stress reduced zones as darker areas around each fibre. A central stress free zones results from the four-point bend mode radiating stress away from the centre of the beam. Fig. 7 shows the same model as Fig. 6 with both ends fixed to represent a dual cantilever beam instead of a four-point bend configuration. The stress distribution in Fig. 7 is similar though less intense than that of Fig. 6 because some of the stress is redistributed as tensile stress emanating from the top corner fixtures. FEA depends upon the construction of a model to represent a material, such as a composite, and configuring the forces and constraints to best represent either a use situation or to reveal critical zones that are likely to cause performance problems.
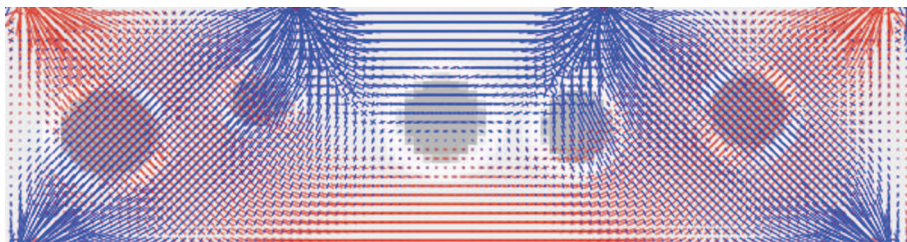

Fig. 7. A four-point stressed dual cantilever simulation of a polymer with a soft interphase surrounding each hard fibre or particle shown as a stress distribution diagram

The contribution of an interphase has been evaluated using a variant of FEA called the advanced boundary element method to model fibre-reinforced composites with consideration of varying thickness boundary layers (Chen and Liu, 2001). Fisher and Brinson (Fisher and Brinson, 2001) have used the Mori-Tanaka model and its extension by Benveniste to study a three-phase composite with either separately dispersed fibre and soft interphase material, that may include voids, or where the fibres were enveloped by the soft interphase material. The Mori-Tanaka model was more effective in predicting the matrix-dominated moduli of the composite. Physical aging was studied by using time and frequency shift factors for these thermorheologically complex materials, with frequency data being preferred. The 2D FEA results demonstrated the importance of the interphase in determining the overall shift rates of the composite. FEA was performed in 2D using a hexagonal array of inclusions with transverse hydrostatic and transverse shear superposition, to obtain the transverse Young's modulus and transverse shear complex moduli.

A composite interphase model was constructed including glass beads, an interphase and polycarbonate matrix, with perfect bonding at each interface and the beads symmetry packed in a cubic array. The Young modulus, stress concentration and stress distribution were simulated. The interphase increased fracture toughness at the expense of elastic modulus, with these observations becoming larger with increase in interphase thickness. A

suitable selection of filler content, interphase stiffness, thickness and Poisson ratio can reduce stress concentration with retention of composite modulus (Tsui et al, 2001).

Stress has been imparted in two orthogonal directions simultaneously on a cruciform shaped specimen (Lamkanfi et al, 2010). The cruciform is applicable to real systems such as rotor blades. Strain was concentrated in the conjunction of the two stresses and concentrated in the corners rather than the centroid of the specimen. The numerical model was validated by experiments by means of a digital image correlation technique. Two- and three-dimensional models were evaluated. An orthogonally stressed cruciform has been simulated independently (Fig. 8) with a fixed point at the centre. The stress distribution (Fig. 8, upper) shows the stresses concentrated in the central region near the corner. This is better depicted in the Von Mises contour plot where the darker shading passes from the arms around the centroid.
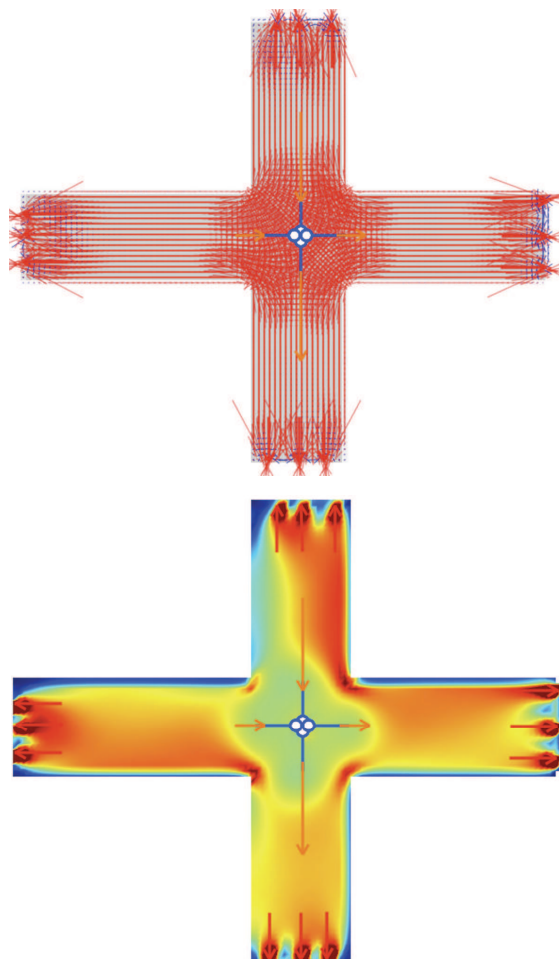

Fig. 8. Cruciform with orthogonal stress, stress distribution (upper), Von Mises contour (lower).

The planar shear bond test has been modelled with consideration of the moduli, bond layer thickness and loading conditions (Dehoff et al, 1995). The model consisted of cylindrical specimens bonded together, and asymmetric elements were used with harmonic stress. Large stress concentrations were confirmed at the bonded interface. The study is directed to assessing whether the shear bond test is useful for understanding the stress states that cause failure and if these stress states exist in clinical dental situations.

2. Computational methods

The finite element method (FEM) is used to solve partial differential equations using approximations that are iterated towards an optimised solution by numerical integration. The process of using FEM to solve practical problems is called FEA. Dedicated software provide the elements for most FEA computations. Examples shown in this chapter were prepared using the ForcePad (version 2.4.2, Division of Structural Mechanics, Lund University, Sweden) two-dimensional FEA program. ForcePad consists of three modes: a sketch mode where the model is prepared, a physics mode where forces and constraints are defined, and an action mode where the FEA calculations are made and the results visualised. The calculations shown in this chapter were performed with a mesh step setting of 6 (fine), vector constraint stiffness scale and force magnitude of 1-10 kN, weight of 1 kN, hard phase elastic modulus of 2 GN, stiffness scale factor of 1000, Young's modulus of 0.35 GN, relative thickness 0.1 and an element threshold of 0.02. Some of the pixelation in figures in this chapter arise from the mesh size used and not the resolution of the images. FEA software often consists of two components, model preparation, such as FEMAP, and the computational engine, such as Nastran.

Commercial software provides a visual and computational environment for application of FEA to problems ranging from simple components to complete complex systems. Typical commercial FEA software are: FEMAP and Nastran, Abaqus, ACTRAN, ANSYS, AcuSolve, ADINA, SLFFEA, LISA, ALGOR, Strand7, AutoFEA, LUSAS, MIDAS, FEAP, JANFEA, CADRE, FEMM, FesaWin, CALCULIX, COMSOL, Visual FEA, FEM-Design, DUNE, FEBio, ForcePAD, JFEM, OOFEM, TOCHNOG, MARC, OpenFEM, SJ MEPLA, FEAMAC, Opera 2D/3D, and many more specialised adaptions of the FEM. Some of the software rely on optional modules for particular and specialised applications. Details of the development and capability of each software package is available from suppliers and any comparison or discussion of performance is a personal choice and beyond the scope of this chapter.

General purpose software such as Mathematica, Matlab, MathCad, Igor, Maple can perform the necessary functions, and some ancillary plug-in programs are available to extend the basic programs. Users design modelling and computational applications within the environment of the software package with the aid of pre-written tools. Many custom written programs are used as described in the literature, which are written from basic principles using languages such as C++ and higher level object tools.

FEA experimental design: Description of the actual material, design of the model, validation of the model preliminary calculations, evolution of the simulation with stress, strain and time, analysis of the data. That is the response of each element to the displacement field associated with the environment of the material. Design of the model: finite element or mesh density and pattern, geometry of modelled composite: selected shape and region or finished article, geometry for application of stress or strain, parameters to be modelled (Fig. 9).

Boundary conditions need to be defined together with anchor or fixture points or surfaces. An element mesh needs to be designed and this mesh established a set of nodes in two-dimensions or three-dimensions. The elements can be triangular, tetrahedral, pyramidal, quadrilateral, hexahedral or other depending on the model and the type of problem. Choice of mesh resolution and geometry is important for each model and most advanced software packages provide a range of options and assistance to the modeller to capture smoothness, computational quality, simulation time, curvature and proximity features. Elastic and visco-plastic properties must be defined for the components. The links between elements are the mathematical expressions and their implementation that describe the theoretical aspects of the problem. The component material assembly can be designed as isotropic, orthotropic, anisotropic or laminate structure. The elements can be linear, such as spring, cable, truss, bar or beam. Surface elements include plane, plate, shear or membrane. Solid elements comprise various shapes that may be pre-stressed or face-stressed. The individual elements are allocated structural stiffness, that is a modulus. The modulus matrix is inverted and multiplied with the stress vector to simulate displacement vectors. Recent software includes wizards to assist the analyst in each step of the FEA process.: model design, validation, optimisation and simulation.
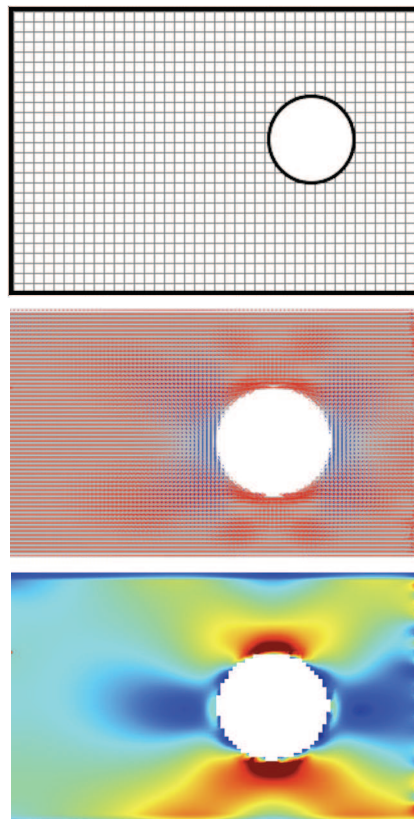

Fig. 9. FEM 2D model of a plaque with a hole for tensile or compression stressing

Validation of the model: Confirmation that the proposed model does portray reality. The model defined in this research used a 3D array of parallel fibres surrounded by matrix. The model did not consider looped entanglements of fibres, though the tensile strength in such cases must be equal to that of a fibre since a fibre must break to allow the matrix to extend. Inefficient wetting and voids are present in real composites, however they are not included in the models because the concentration, shape or distribution of voids is not known. Discrete voids can be replaced by a density gradient in composites that were partially consolidated.

Evolution of the simulation: Input of the material parameters. Digital incrementing of the independent variables. Computation of the dependent variables for each node, within each element, at each increment of stress or strain. The geometry of fixing the position of part/parts of the composite and the location of the displacement field are significant for the evolution of the model.

Analysis of the data. Stress-strain-time curves, analyses of variables between nodes, analysis of variables between composite models. Comparison with continuum composite models.

3. Examples of specific FEA composite simulations

3.1 Laminated composite structures

Damping of laminated carbon and glass fibre plastics is measured as the ratio of energy dissipated to the maximum strain energy stored per strain cycle. A finite damped element model including transverse shear has been used to measure and predict specific damping properties, mode shapes and natural frequencies of the composites (Lin et al, 1984).

Analysis of laminated composites has been performed where the failure mode was delamination of the layers. When a weaker ply fails first, stress will be distributed to the remaining plies and the process will continue giving a progressive failure of the laminate. The modulus of the layers was anisotropic so the strength of the composite depended on the relative orientation of each layer within the overall laminate. Symmetric and anti-symmetric ply laminates with different numbers of layers were formed (Pal & Ray, 2002).

Mechanical properties of a composite depends upon the geometry and aspect ratio of the fibres, while woven fibre composites are distinct from typical unidirectional long fibre composites. A woven fibre composite is usually prepared from multiple layers where the overall composite properties are the sum of the contribution layers. Flexural behaviour is a suitable way to characterise woven or unidirectional composites, since tensile force will be resisted by the modulus of only the continuous fibres. The layers of a woven composite can be divided into unit cells that are the smallest area in which the weave pattern is repeated (Fig. 10). A composite model can be formed by adding units cells laterally and in the thickness direction to form a structure as required for the model. This can be computationally performed by setting periodic boundary conditions. Computations for the property simulation used Abaqus. The FE model used curve beam elements to model warp and weft yarns of the unit cell. This simplified the model yet captured the actual morphology of the woven fibre composite, and hence allowed prediction of flexural modulus. Other weave types and replicating structures are suited to this efficient unit cell model with periodic boundary conditions (Soykasap, 2010).

A model of membrane layers of a laminated fibrous composite (analogous to Fig. 11) addressed the particular issue of interlaminar shear stresses concentrating along an edge region.
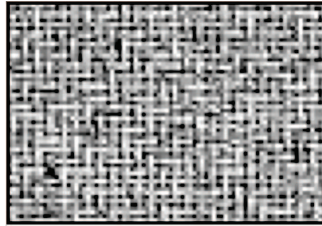

Fig. 10. Standard cross-ply woven fibre mat

The FEA simulation was limited to plane stress, where bending and warping of the laminate have not been considered. The simulated data were favourably compared with corresponding analytical data, with the FE technique presented capable of application in a range of laminate situations involving interlaminar shear stress (Isakson & Levy, 1971).

Through-width delamination of a laminate composite under a buckling compressive stress has been treated using a FEA parametric model. The stress leads to an instability related delamination. Comparison with measured lateral deflections showed that the analysis reflected actual specimen behaviour. Stress transfer was complex and not a simple function of applied stress or lateral deflection, with steep stress gradients at the delamination front suggesting a stress singularity. Hence strain energy release rates were much less responsive than the calculated stresses to mesh refinement. Correlation of the calculated strain energy release rates for mode I and II crack extension with actual delamination growth rates showed that delamination growth was dominated by mode I, even though mode II strain release rate was numerically larger (Whitcomb, 1981).
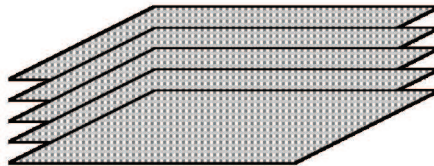

Fig. 11. A multi-layer woven fibre laminate composite

Orthogonal woven composites were studied using a 3D unit cell model using a custom approach followed by Strand6 FEA software. The woven fibre composite was for aerospace application, typically an epoxy-graphite fibre structure. The structural variables included geometric parameters, yarn volume fractions and engineering elastic constants. The in-plane modulus predicted using the unit cell model and a laminate model corresponded with experimental data form the literature (Tan, et al, 1998). Two-dimensional models were presented for the elastic analysis of a plain weave glass or carbon fabric epoxy resin laminate, with the aim of developing a simple though generalised model that will enable interpretation of the two-dimensional extent of the fabric, and the contribution of various fabric geometry parameters. This was intended for prediction of in-plane elastic modulus and selection of fabric geometry for any specific application. The fibre volume fraction

depended on weave geometry for a constant global fibre volume fraction. The elastic modulus predicted increased with fibre undulation, but reduced for 2D prediction and remained constant for a 1D parallel model with lamina thickness increase (Naik & Shembekar, 1992).

An example of a 2D to 3D global/local FEA of a laminated composite plate with a hole is presented. 2D models have shown limitations that can be addressed by using 3D models, though in many cases a 3D model is computationally inefficient for a particular task. An alternative is to perform a detailed 3D analysis on a local region of interest, followed by a more extensive, in geometry, stresses or time, simulation using a 2D model. In the example presented the local region of interest is the hole, around which the interlaminar stresses need to be examined in detail. There appeared to be a critical hole size in the laminate where the interlaminar stresses were maximised. A reduction in hole size for 3D laminates under compression suggested that if the hole contained a fastener, failure could result as the laminate fibres would be crushed into the rigid fastener. The degree of thickening of laminates under compression, around the hole, varied with lay-up, hole size and position about the hole (Muheim & Griffen, 1990).

Progressive failure of plain weave composites subjected to in-plane extension has been simulated using 3D FEA. Stress was parallel to the tow orientation and tow waviness was considered. The predicted strength decreased with tow waviness as progressive delamination occurred (Hyung, et al, 1991). The fibre shaping process in textile composites was evaluated from biaxial tests on cross shaped specimens. The influence of undulation variations in the weave and of interactions between the warp and weft were variables considered. The total energy was calculated as the sum of energies of each elementary cell that can repeat to form the entire composite. The weave undulations and interactions contributed to decline of the mechanical behaviour of the fabric composites (Boisse, et al, 1997).
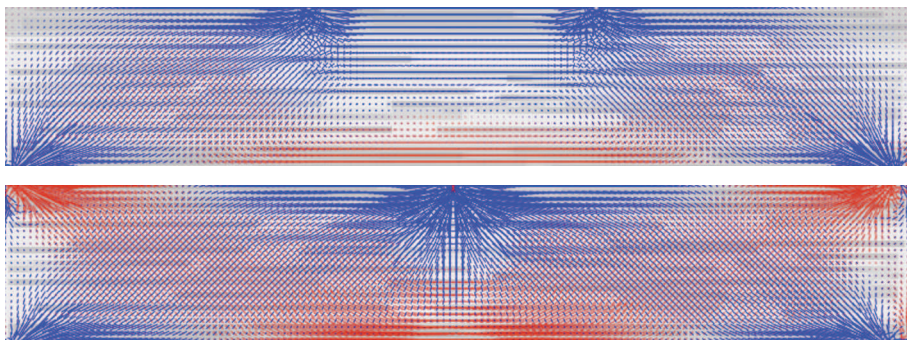

Fig. 12. A model of a laminated fibre-reinforced composite with stiff surface layers (darker shading) and a central softer layer under four-point bend (upper) and three-point dual cantilever (lower).

The laminated composite model shown in Fig. 12 has stiffer surface layer where resistance to stress is most important and a softer central layer where stresses from deformation modes is least. In practice the central layer could be a foam or material with limited consolidation. Each of the layers is reinforced with the same type of fibres. The simulated was performed

under four-point bending (upper image). A compressive stress concentration radiates from the application points to the two supported lower corners. A tensile stress is situated along the higher stiffness layer at the bottom. Stress transfer to fibres is visible throughout the laminated composite, but mainly as expected from regions of higher stress. A three-point dual cantilever simulation of the laminated composite is shown in Fig. 12 (lower). The stress concentrations are more intense with tensile stress regions emanating from the upper corner fixtures and and in two localised regions along the bottom layer. Stresses are greater in the stiffer matrix top and bottom layers and stresses are transferred to fibres throughout, though with increased intensity in the higher stress regions of the upper and lower layers of the laminate.

3.2 Anisotropy of thermoplastic composites

Orientations occur during processing operations such as extrusion and injection moulding. Fibrous and platelet fillers are orientated along the extrusion direction or along the more complex flow lines in injection moulds. This directional geometry of the fibres is normally welcome as maximum modulus and strength are required in the machine direction. These orientations simplify the preparation of finite element models and the simulation process. An anisotropic fibre composite is shown in Fig. 13 (left) with the stress applied parallel to the fibres. The stress concentrations are low and concentration in regions where the fibres were made shorter. When the stress was applied transverse to the fibres (right) the stress concentrations were much greater throughout since the lower modulus matrix carried more of the stress than the higher modulus fibres.
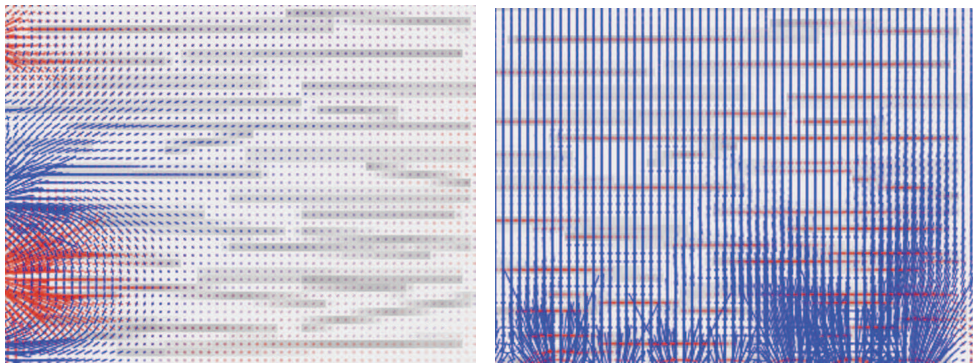

Fig. 13 Anisotropic fibre composite showing stress distribution when stress was applied parallel to the fibres (left) and transverse to the fibres (right).

A more complex consideration is anisotropy of the molecular or supramolecular structure within the matrix. Shear flow causes preferential orientation of molecules, crystallites and crystallite assemblies along the flow contours. Thus a polymer matrix will posses anisotropy of properties such as modulus, Poisson ratio, density fluctuations, residual stress, strength and ultimate properties such as toughness, break stress and strain. The anisotropy arising from these molecular considerations is difficult to accommodate in a model and subsequent simulation of the application of stress or strain. They are difficult to consider when interpreting actual performance properties of polymer composites since any detailed

molecular structure or morphological structure is difficult to measure and usually unknown to the tester and materials designer. The microstructure of the composite is known to be the determinant factor for the properties of fibre reinforced composites (Maligno, Warrior, Long, 2008).
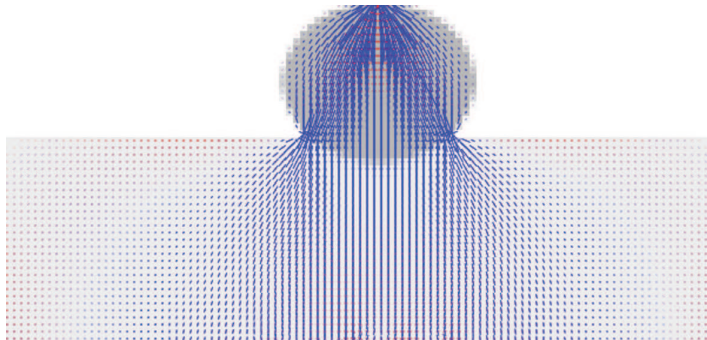

Fig. 14. Stress distribution diagram for indentation of a soft material by a hard round object.

A three-dimensional finite element model was used to evaluate indentation of polypropylene–cellulose fibre composite to determine the elastic modulus and hardness. FEA was conducted using a rigid flat cylindrical disk with a radius of 1 µm with Abaqus. Indentation to 50 or 100 µm depth provided no difference in the unloading values. Analysis of the interphase region showed a 1 µm wide property transition zone, though this zone could not be isolated from the contributions of the adjacent polypropylene and cellulose fibre properties (Lee, et al, 2007). An analogous model is shown in Fig. 14 in 2D and the simulation was performed with stress applied to the hard round object while the soft material was supported along its base. The stress diffuses from the application point with concentrations at the edge where the round object meets the soft material. Indentation by such a blunt object is not as critical as when a crack has been induced and the crack tip provides a focal point for stress concentration.

3.3 Composites with the same material for fibre and matrix

All-polypropylene composites have been prepared from polypropylene fibres with a lower melting temperature polypropylene matrix that may consist of a random polypropylene copolymer, different polypropylene tacticity or a different crystalline allotrope of polypropylene. The different melting temperatures of the polypropylene fibres and matrix are required so that the matrix can melt and form a continuous phase around the fibres without changing the oriented crystalline morphology of the fibres (Houshyar & Shanks, 2003). All-polypropylene composites can be prepared from polypropylene film-fibre layers, felted interwoven fibres of different melting polypropylenes, fibres impregnated with lower melting temperature polypropylene powder, fibres impregnated with polypropylene solution (Houshyar et al, 2005). Adhesion between polypropylene matrix and polypropylene fibres was characterized using a micro-bond test inspired by a fibre pull-out technique. The results showed that adhesion was appreciably increased when polypropylene fibres were used instead of glass fibres in the matrix (Houshyar & Shanks, 2010).

Finite element analysis showed that fibre volume fraction and diameter controlled stress distribution in all-polypropylene composites. The stress concentration at the fibre-matrix interface was greater with decreased fibre volume fraction. Changes in fibre composition were present in high stress regions. Stress concentration at fibres decreased and interfacial shear stress were more severe when higher modulus fibres were included. The relation between matrix and fibre modulus was significant, along with the interfacial stress in decreasing premature interfacial failure and enhancing mechanical properties. The simulations revealed that low fibre volume fraction provided insufficient fibres to disperse the applied stress. Under such conditions the matrix yielded when the applied stress reached the matrix yield stress, resulting in increased fibre orthotropic stress. With high fibre volume fraction there was matrix depletion and stress transfer had diminished capacity (Houshyar et al, 2009).

3.4 Biomaterial composites

A compromise in the design of biomaterials is the need for soft tissue bonding materials in contrast to hard wear resistant materials particularly in restorative dental composites. An approach is to use low modulus adhesive linings to release stress due to contraction during cure. Alternatively polymer without filler or lightly filled can be used as a low modulus relatively thick bonding layer to reduce the gradient to a high modulus restorative material. The FEA study aimed to predict the adhesive lining thickness and required flexibility. The model used a tooth shape connected by spring elements, constructed with 3D CAD using digitised images of a scaled tooth plaster model, and exported to Pro-Engineer, while FEA used ANSYS. The FEA determined optimum adhesive layer thickness for maximum stress release, though the models showed that a thin flexible adhesive layer had the same efficiency as a thicker higher modulus adhesive layer Ausiello, et al, 2002).

Polymer glass and carbon fibre reinforced dental posts were modelled to compare properties with those of gold alloy cast posts, using a natural tooth restoration as a reference. The models contain continuous, unidirectional fibres that were uniformly packed. The models were formed with reference to the actual geometry of an upper central incisor. The model meshes were created using Mentat software package and they were simulated using MARC solving code. Mechanical data was obtained by three-point bend testing and compared with the FE model results, under various loading conditions. The glass fibre exhibited lowest peak stresses because its modulus was similar to that of dentin, and thus similar to the natural tooth (Pegoretti, et al 2002).

The stress contributions and stress rates within posterior metal-free dental crowns made from new composite materials were investigated. An FEM of a first molar was constructed, and placed under load simulating maximum bite force and mastication force using FEA. The maximum stress concentrated about the loading point. When a load was applied horizontally it was found to be a critical factor determining failure (Nakamura, et al, 2001).

The fatigue and fracture characteristics of human bone subjected to long term dynamic loading may be due to gradual reduction of elastic modulus resulting from various strain environments. The FEM applied to these materials will establish information for the design of joint replacements. Femoral neck fractures were chosen as a situation in which to explore FEA using continuum damage mechanics applied to the fatigue behaviour. Minimal modulus degradation or accumulation of permanent strain was observed until near the end of their fatigue life, and the models were unable to predict the rapid deterioration in the last

Thank You for previewing this eBook

You can read the full version of this eBook in different formats:

- ➢ HTML (Free /Available to everyone)

- ➢ PDF / TXT (Available to V.I.P. members. Free Standard members can access up to 5 PDF/TXT eBooks per month each month)

- ➢ Epub & Mobipocket (Exclusive to V.I.P. members)

To download this full book, simply select the format you desire below