12
The use of draping simulation in composite design

J W KLINTWORTH, MSC Software Ltd, UK and
A C LONG, University of Nottingham, UK

12.1 Introduction

Composites forming is an important part of the overall development process for components and structures made of composites materials. Although composite materials are used extensively for applications that are not strength-critical, this review will concentrate on structures needing high mechanical performance.

The military aerospace market has used high-performance composites for many years, but civil applications are now growing in importance with a huge increase in the use of CFRP (carbon fibre reinforced plastics) for full-sized civil aircraft.

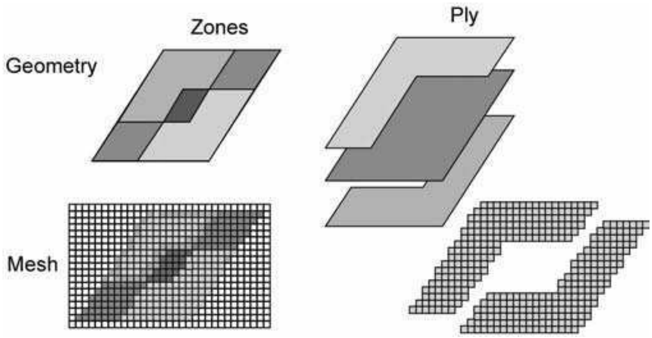
While large composite structures are now manufactured using tape or tow placement, the majority of structures are laid up of plies of sheet materials that have to be applied to 3D mould surfaces. The application methods remain predominantly manual, assisted by laser projection systems. Robotic application systems are being tested to improve the quality of layup and these are being introduced. Automated forming processes are also growing in popularity, including diaphragm forming for thermosets (Chapter 10) and stamping for thermoplastics (Chapter 11).

In this chapter the use of draping and forming simulations within the composites design and development process will be discussed. This will focus on the use of modelling techniques to inform component design and to assist with component manufacture. In particular the use of forming simulations within the composites design and optimization environment will be described.

12.2 Zone and ply descriptions

During the design and analysis tasks of the composites development process, engineers use both zone and ply descriptions of a laminate as shown in Fig. 12.1.

Analysts have traditionally used finite element analysis techniques to determine the performance of composite structures using a mesh-based approximation of the model. From the perspective of the finite element analysis code, the



12.1 Alternate descriptions of the laminated composites model.

manufacturing requirements are irrelevant and only the local properties of the laminate on each element are significant. This has lead to the use of a 'Zone' description of the laminate, where a 'laminate material' consisting of a stack of layers of homogeneous material is defined and assigned to the underlying mesh. This description is quick to generate for simple structures, but impossible to use on complex structures where the fibre orientation varies continuously.

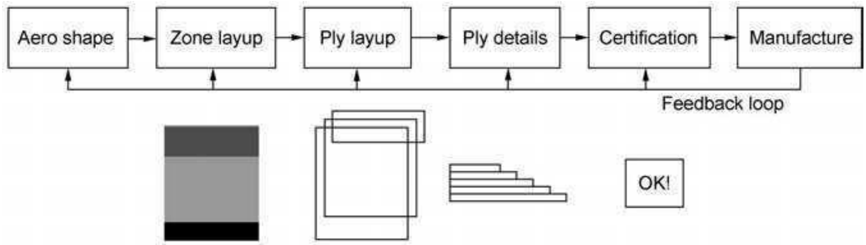
Designers have traditionally defined the layup using curves defining ply boundaries on an underlying surface, i.e. a 'Ply' description. This description mirrors the final manufactured configuration closely and allows the designer to draw in details such as ply drop-offs and insert placement. However, drawing this detailed description is relatively time-consuming.

These traditional descriptions are not compatible and the interaction between analyst and designer is severely impeded by the different descriptions. To improve the efficiency of the composites development process, analysts now often use a ply description on the mesh, as pioneered in Laminate Modeler from MSC.Software (Klintworth and MacMillan, 1992). This allows rapid definition and modification of the composites structure and ready incorporation of the results of draping and forming simulations in the finite element model.

Recently, as more work has become centralised in PLM (product lifecycle management) systems, the zone description used in preliminary work is now supported by mainstream design-centric systems. The most sophisticated systems offer some capabilities for transferring zone descriptions to ply descriptions in a semi-automated fashion.

12.3 Composites development process

The composites development process spans a number of tasks and functions as shown in Fig. 12.2. In the majority of aerospace or automotive applications, the basic geometrical shape is fixed by aerodynamic and packaging constraints defined earlier in the project cycle. This data is typically generated and stored using an enterprise-wide CAD system such as CATIA.



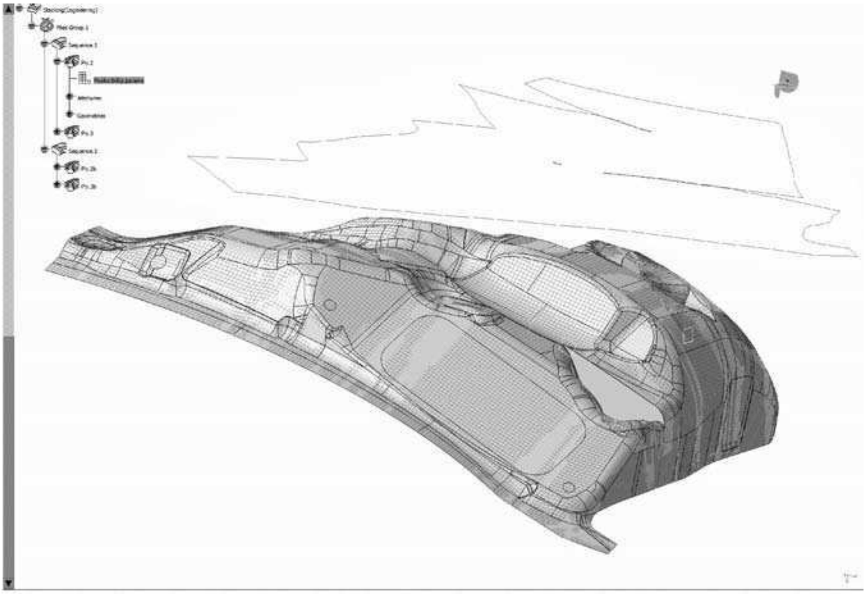
12.2 The composites development process.

After the initial shape has been determined, preliminary structural analysis is undertaken in order to quantify the thickness and orientation of reinforcing materials in the structure required to carry the imposed loads. The finite element analysis technique is typically used to quantify the mechanical and thermal performance of the structure. In the aerospace industry, this work is typically undertaken using software packages such as MSC.Patran and MSC.Nastran®. At this stage, the composite model is defined using laminate materials (e.g., MSC.Nastran PCOMP cards) or ply descriptions based on the finite element model.

Once preliminary sizing is completed, the composites model is transferred to the CAD system where the manufacturing details of the model are added. In particular, ply drop-off and splicing details are specified to ensure that no ambiguity is possible during the manufacture of the structure. These processes are typically undertaken using a composites design tool like CATIA Composites Design, which defines all composite data in terms of geometrical entities. Increasingly advanced simulation tools are becoming available for simulating fibre orientations, even on complex surface geometries as shown in Fig. 12.3. These simulations have brought techniques previously restricted to specialists into the domain of the design engineer. Accounting for fabric behaviour earlier leads to fewer problems in the downstream process.

For composite structures, it is likely that the detailed design process will introduce a number of changes in the structure. Therefore, it is wise to re-evaluate the final performance of the structure just before manufacturing. Certification analysis has been greatly improved recently by the introduction of ply tracking in the commercial finite element analysis codes to allow identification and interpretation of laminate results. For example, the new PCOMPG card in MSC.Nastran 2004 will automatically sort layer results on the basis of global plies as illustrated in Fig. 12.4.

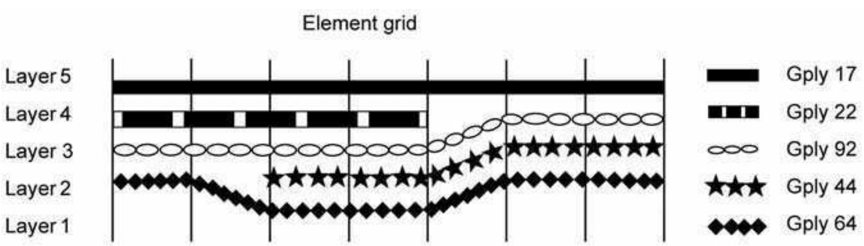
Once design requirements have been satisfied, the manufacturing information must be generated in the form required by the manufacturing process. A feature of the past few years has been the advance in techniques for simulating composites manufacture. For example, nonlinear FEA solvers such as MSC.Marc and ESI PAM-FORM can now undertake detailed forming analysis



12.3 Advanced fibre simulation in the design environment.

of fabrics and composites to predict regions liable to wrinkling. These codes now also have the capability of simulating curing, involving the coupled analysis of mechanical, thermal and resin curing problems. These problems are particularly important for thick open sections typically used on spars or frames. The use of FEA for forming simulation is discussed in more detail in Chapter 3, whilst curing simulation and implications for component distortion are described in Chapter 7.

Clearly, the composites development process is multidisciplinary in nature and makes use of a wide range of tools and skills. Most fundamentally, the way in which composites data are defined by analysts and designers is fundamentally different from traditional materials. This has introduced substantial inefficiencies in the development process and delayed the use of composite materials in cost-sensitive industries.

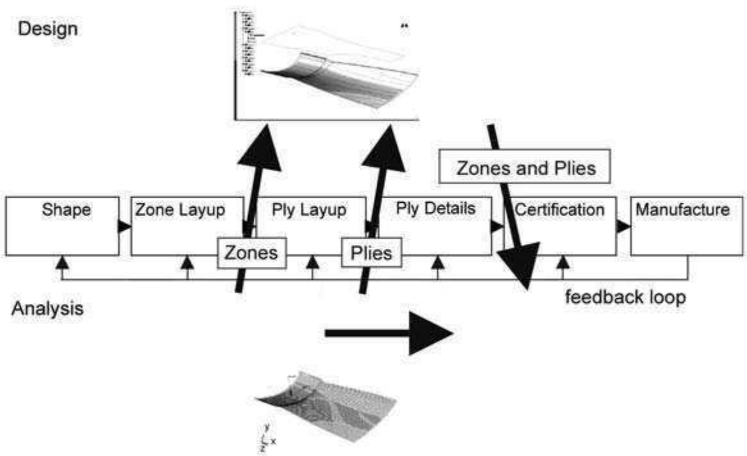


12.4 Global ply modelling in FEA codes.

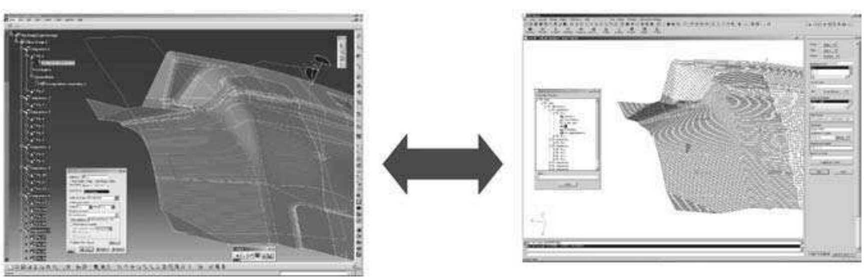
12.4 Composites data exchange

Design environments are becoming increasingly sophisticated and permit baseline mechanical analysis using finite element methods. However, the design environment remains unsuitable for advanced analysis, particularly when abstraction needs to be changed, as the design environment usually assumes that the finite element mesh is tied to the geometry. This approach cannot allow for typical analysis techniques like the use of multi-point constraints to model hinges. Therefore, specialised analysis environments retain their usefulness and a key requirement during the development process is to transfer models between the design and analysis environments as shown in Fig. 12.5.

Such data transfer was traditionally effected using file transfer. Unfortunately, file formats are not rich enough to transfer full data incorporating both geometry and associated data. Happily, modern design systems incorporate rich interfaces so it is possible to transfer all required data in a seamless operation. Figure 12.6 shows the transfer of a complete ply model from design to analysis environments, complete with layup tree and associated data.



12.5 Data transfer between design and analysis environments.



12.6 Example of data transfer.

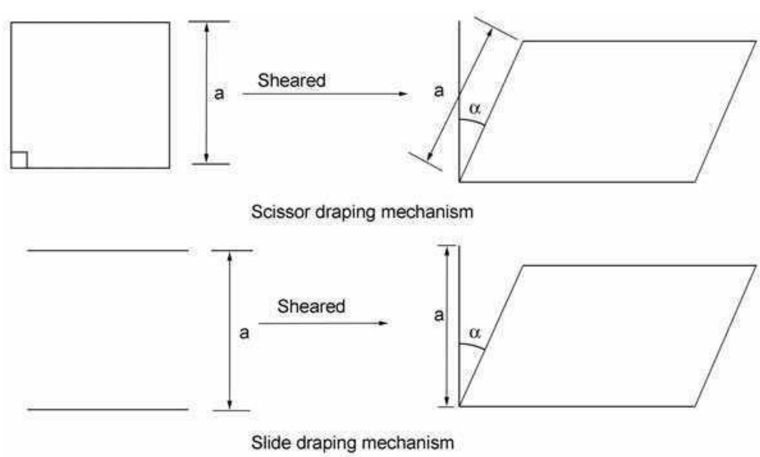
12.5 Draping and forming simulation

During the composites development process, it is important to simulate the draping or forming of the plies to predict problems, estimate material requirements, generate manufacturing data and provide accurate fibre angles for subsequent finite element analysis. Most tools in commercial use implement kinematic draping algorithms that use a geometrical approach to provide accurate simulations for hand-layup processes. Where friction is significant, such as if matched die tooling or diaphragm forming techniques are used, full finite element techniques can be utilized.

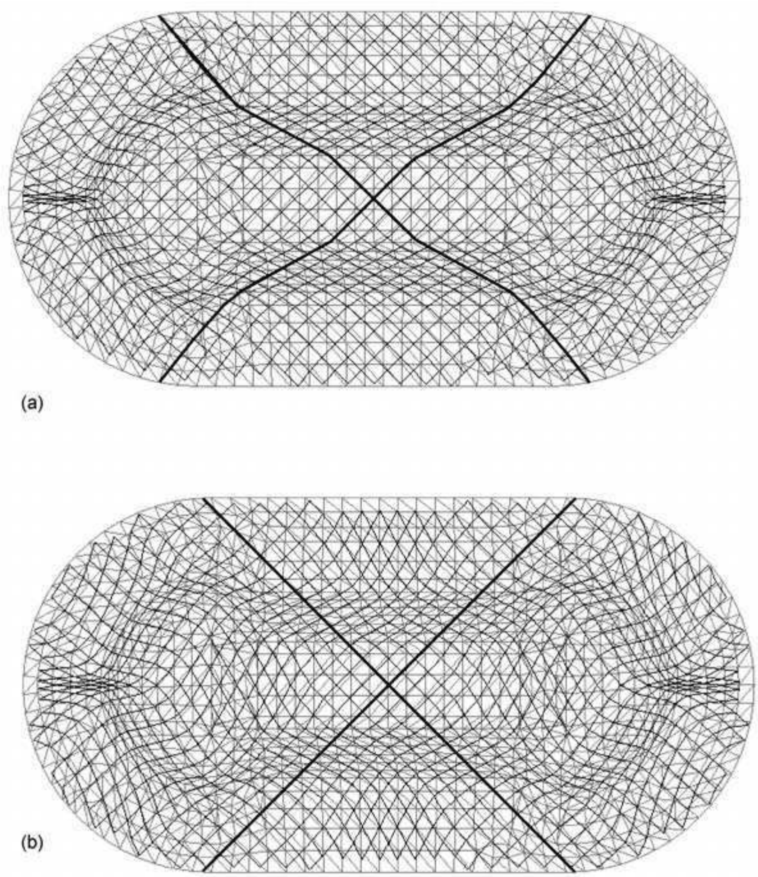
12.5.1 Kinematic draping simulation

To form flat material to a 3D surface, the material must shear. Kinematic draping simulation begins with an assumption of the material shearing behaviour which depends on the material structure. Typical material models are for scissor and slide deformation, related to biaxial woven fabric and unidirectional material respectively (Fig. 12.7). Based on this assumption, the incremental draping over a surface can be calculated. (i.e. the local draping problem). Results can be used directly to provide fibre orientation data for structural analysis, to determine ply net-shapes for fabric cutting, and to approximate manufacturing difficulties such as wrinkling when shear deformation exceeds a user-defined limit (‘locking angle’).

To define the overall fibre pattern, the way in which the fabric extends from an initial point must be assumed. A unique draped pattern can be obtained by specifying two intersecting fibre paths (generators) on the surface of the component or forming tool. The remaining fibres are positioned using a



12.7 Scissor and slide draping mechanisms for woven and unidirectional materials.



12.8 Kinematic drape simulations for a double dome geometry, based on geodesic (top) and projected (bottom) generators.

mapping approach, which involves solving geometric equations to determine the intersection of the surface with possible crossover points for the fibre segments. Several strategies are available for specifying the generator paths, and their correct specification is critical as this will determine the positions of all remaining fibres. This is illustrated in Fig. 12.8, which compares predicted fibre patterns obtained using a scissor model with geodesic and projected generator paths. A number of different approaches have been developed, tested and incorporated into commercial codes. For example, the MSC.Patran Laminate Modeler allows the user to use geodesic, minimum energy, or minimum shear extension methods.

The user may also be able to apply constraints to the simulation by defining initial warp or bias directions using lines. These lines in turn can be drawn on 2D shapes and projected onto the 3D mould surface to allow very accurate fibre placement on the shop floor. The most advanced kinematic simulations allow the user to define the order in which the draping proceeds. This concept is very familiar in the metal forming industry where multi-stage stamping is common, and provides excellent accuracy for typical manufacturing situations.

12.5.2 Finite element forming simulation

Finite element techniques are used in situations where friction effects are important, or where material behaviour cannot be approximated using one of the simple deformation modes outlined above (e.g., for non-isothermal stamping of thermoplastic composites). Such techniques also allow ply wrinkling to be predicted directly via determination of compressive loads within plies. In general, because detailed material characterization is required, extensive testing is necessary before simulation begins (see Chapter 1). Nevertheless, finite element techniques can be useful for extending and checking kinematic simulations for specific simulations. The basis of such techniques is described in Chapter 3.

12.5.3 Comparison of kinematic and finite element forming simulations

Kinematic simulations are well proven and are usually completed in less than one second. By comparison, finite element techniques take between about a minute for an implicit single-ply simulation to hours or days for a multi-ply explicit simulation. Therefore, finite element techniques are currently primarily a research topic for all except the most specialised applications. However, the use of such techniques may become more widespread as automated forming processes are adopted.

12.6 Linking forming simulation to component design analysis

12.6.1 Initial zone sizing

During the initial sizing phase of composites product development, ubiquitous optimization techniques can be applied to define the initial layup. The general optimization problem can be written as:

$$
\begin{array}{llll}
\text{minimize:} & F(X) & & \text{objective function} \\
\text{subject to:} & g_j(X) \leq 0 & j = 1, 2, \ldots n_g & \text{inequality constraints} \\
 & h_k(X) = 0 & k = 1, 2, n_h & \text{equality constraints} \\
 & x_i^l \leq x_i \leq x_i^u & i = 1, 2, \ldots n & \text{side constraints} \\
\text{where:} & X = \{x_1, x_2, \ldots x_n\} & & \text{design variables}
\end{array}
$$

Many different methods have been developed to solve this mathematical problem, and the numerical methods required are relatively mature.

12.6.2 Optimization in FEA codes

The numerical optimization techniques can be used with finite element analysis using the latter as a 'black box' for calculating results for a given set of design variables. However, the usefulness of numerical optimization has been increased enormously by embedding numerical optimization algorithms into the finite element analysis codes themselves (Klintworth, 2001).

By doing this, multiple constraints can be applied to a model simultaneously. For example, users can limit stresses, displacements or even derive results like failure indices simultaneously. Furthermore, constraints can be imposed for multiple analysis types at the same time. For example, users can specify a minimum natural frequency from a normal modes analysis together with a maximum stress from a linear static solution.

Optimization techniques based on gradient techniques require the calculation of the sensitivities of the design constraints to changes in the design variables. This was traditionally calculated via finite difference techniques, but can be calculated analytically within the FEA code. Together with the use of double precision variables, this improves the speed and numerical accuracy of the optimization algorithm by orders of magnitude.

In addition to gradient techniques, other numerical methods such as topology optimization algorithms and fully stressed design techniques can be integrated into a FEA solver with commensurate speed and accuracy gains (Klintworth, 2005).

12.6.3 Zone description

The zone description in a FEA code consists of a laminate record, which is referenced by a set of elements, typically shells. For example, the MSC.Nastran PCOMP card has the format shown in Table 12.1.

The LAM option allows explicit specification of individual layers of the laminate, as well as new smeared laminate options that are of particular relevance to optimization problems. By initially neglecting the stacking

Table 12.1 Format of the MSC.Nastran PCOMPG card

1	2	3	4	5	6	7	8	9	10
PCOMP	PID	Z0	NSM	SB	FT	TREF	GE	LAM	
	MID1	T1	THETA1	SOUT1	MID2	T2	THETA2	SOUT2	
	MID3	T3	THETA3	SOUT3	etc.				

Table 12.2 Effect of laminate options in MSC.Nastran

LAM option	New 2001	Membrane [A]	Bending [B]	Coupling [D]	Ply results	Comments
BLANK		Y	Y	Y	Y	Default
SYM		Y	Y		Y	
MEM	Y	Y			Y	Wing skins
BEND	Y		Y		Y	
SMEAR	Y	Y	Y			Smeared
SMCORE	Y	Y (core N)	Y			Smeared with core

sequence of the laminate, extraneous modes resulting from membrane-bending coupling are suppressed and the algorithm converges more rapidly. The laminate options consider the effects shown in Table 12.2.

The SMEAR and SMCORE options model 'percentage ply laminates', where the overall percentage of fibres in various directions are known, but the stacking sequence is not. This description is exactly what is required at the beginning of the design process.

12.6.4 Design variables for thickness and orientation

Within MSC.Nastran, the design variables are defined using the DESVAR card. These can be linked with any property variable (e.g., PCOMP) using the DVPREL1 and DVPREL2. This allows enormous flexibility in setting up the composites optimization model.

In theory, the user can treat every thickness and every orientation on every PCOMP card as a design variable. However, the use of orientation as a design variable is limited by two factors. First, the principal stiffness of a layer of material rotated through a small angle Θ varies approximately proportional to $\cos^2\Theta$. This means that small rotations can have very little effect on the constraints, which in turn means that it is numerically difficult to find an optimum solution and convergence is slow. In addition, it is also very easy to achieve a solution where layers in adjacent zones are at different angles, which cannot be achieved sensibly during manufacture.

It is therefore common practice to define design variables as the thickness of layers that have a fixed orientation with respect to some basis, e.g. a projection of a vector onto the shell surface. For the case of unidirectional plies, the nominal orientations of $-45°/0°/45°/90°$ are usually used, with the core oriented at zero degrees. This gives a total of five design variables of thickness per PCOMP card. For a typical automotive composite model having 100 PCOMP regions, this yields 500 design variables, which can be processed satisfactorily using desktop computers.

12.6.5 Discrete optimization

Materials supplied in sheet form and used to manufacture composite structures are supplied in particular thicknesses reflecting the chosen reinforcement. For example, carbon fabric-based prepreg materials typically have a thickness of 0.25 mm.

Therefore, it is useful to constrain the design variables to some multiple of this thickness in order to achieve a result that can be manufactured. A new feature with MSC.Nastran 2001 allows users to define allowable values in a table. For the optimization calculation, continuous variables will be used for reasons of numerical stability. Then, after a user-defined design cycle (the default is the final cycle), the variables will be forced to the most appropriate discrete value allowing for the model constraints.

12.6.6 Zone orientation

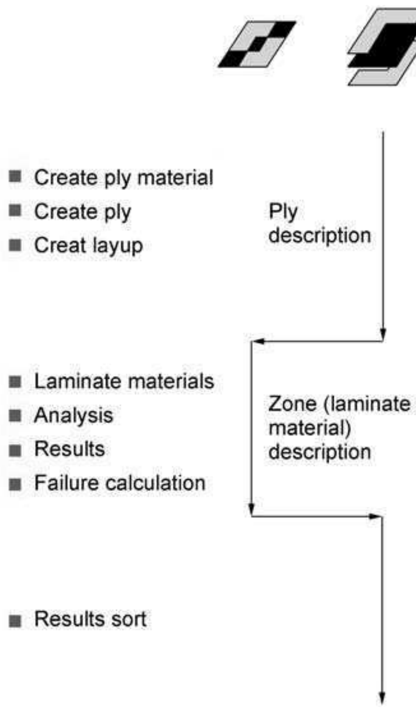
It was noted above that one common technique of orienting the zone laminate orientation is through the use of a projected coordinate system. While this is simple for a relatively flat component like a wing skin, problems arise with curved shapes. In these cases, a kinematic draping algorithm can be used to simulate the drape of a fabric ply over the surface. From this, the orientations of the warp fibre on every element can be calculated and stored in a field within MSC.Patran, which can then be used to orient the zones. This provides a much more realistic orientation of the zones in a way that follows the orientation of manufactured plies. By accounting for manufacturing at this early stage of the development process, the user can be sure that plies indicated by the zone optimization process are actually producible.

12.6.7 Ply-based analysis

Once the initial sizing is completed, the analyst can define the initial layup using plies defined on the shell element mesh. This technique allows rapid modification of the model by simply adding and removing plies, or changing their coverage. At the same time, possible manufacturing problems can be identified using the draping simulation found in suitable tools.

Once a ply layup has been defined, the model needs to be translated to the zone-based laminate material description supported by commercial finite element codes. Then after completion of the analysis run, the results (on a zone layer basis) need to be sorted in the format of the layup plies to allow for effective interpretation. This translation between ply layup and zone layer descriptions is shown in Fig. 12.9.

The translation from ply layup to zone description is generally specific to the analysis code used. First, the orientation system supported by the analysis code is chosen by the user. For example, MSC.Nastran users can orientate laminate



12.9 Relationship between ply and zone modelling.

materials using a coordinate system or an angle from the first edge of each element. Second, the laminate materials on each element can be formulated based on the draped directions and the orientation system. Finally, sorting procedures are used to minimise the number of laminate materials generated within a user defined tolerance. Typical values are 5º of layer orientation and 5% of layer thickness.

Following the finite element calculations, the results can be sorted back on the basis of plies. Historically, this sorting was done as a separate postprocessing operation by the users. However, modern FEA analysis codes can track the global ply identifiers so they can automatically sort results themselves. This capability (MSC, 2005) makes the use of ply descriptions seamless in the analysis environment. This in turn allows designers and analysts to communicate easily.

12.6.8 Accounting for material shear

When fabric shears during the forming process, it changes thickness and the fibre orientations change. For a typical woven fabric with warp and weft fibres initially at right angles, this changes the mechanical properties markedly. For dry fabrics to be impregnated by resin transfer moulding, this also changes the permeability and hence the resulting resin flow pattern (Long *et al.*, 1998).

Calculating the change is dependent on the material system used. Predictive techniques based on micromechanics to predict local elastic constants and layered shell elements to represent the fibres within each ply can be used for simple materials such as non-crimp fabrics or UD prepreg (Crookston *et al.*, 2002). However for woven materials the accuracy of such an approach is questionable. Hence commercial laminate modelling systems allow the user to create material property sets for particular shear states (MSC, 2003). These sets can be defined theoretically or experimentally as required. Then the system will use the appropriate property set for the particular shear state on an individual element. This clearly increases the number of laminate materials required for the analysis.

12.6.9 Failure analysis

Failure analysis is used on an everyday basis for failure calculation in FEA codes. The most common failure analysis is a simple empirical criterion within the plane of each layer (Hill, Hoffman, Tsai-Wu or Maximum Strain are common), with a simple maximum stress criterion for out-of-plane calculations. This allows analysts to quickly identify lowest Margins of Safety on whole models, elements within models, and even plies on each element.

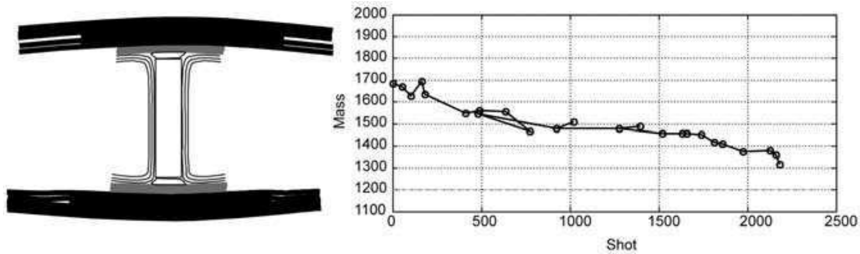
Specialised postprocessors are available to examine multiple failure criteria over multiple loadcases (Anaglyph, 2003). This capability is especially important where a structure has many potentially critical results cases. A particular instance of this occurs when structures are weakly nonlinear so loadcases cannot be generated by linear superposition.

For strongly nonlinear problems where loading may continue after failure, it is necessary to implement failure criteria as user subroutines within nonlinear implicit and explicit analysis codes. This allows users to track progressive failure in components under ultimate loading. This is especially important for crushing and crash analysis. This level of detailed analysis is seldom completed on a routine basis due to its complexity.

12.6.10 Stochastic analysis

Conventional gradient optimization techniques can be very useful for initial sizing, but their usefulness is limited further down the design process. The main problem is numerical instability, but the zone composites description also excludes manufacturing constraints. Therefore the resulting optimized model cannot be manufactured.

However, if the user generates a ply layup model, design improvement reflecting manufacturing constraints can be effected naturally as the plies are already manufacturable. Consider the wing section where a rib intersects the skin as shown in Fig. 12.10. Here, it is essential that plies lie in the wing skin, on



12.10 Stochastic design improvement (SDI) of a section.

pads between rib and skin, in the rib and on joins between rib and skin. Using a ply layup, it is possible to set up a model that reflects the manufacturing requirements.

This ply model can then be improved using different techniques such as stochastic methods (Monte Carlo) or Genetic Algorithm (GA) approaches. An example of a stochastic design improvement of an aerofoil is shown in Fig. 12.10. This required several thousand runs of MSC.Nastran taking several days to run. However, with the increasing power of computers, this sort of improvement will soon be all in a day's work.

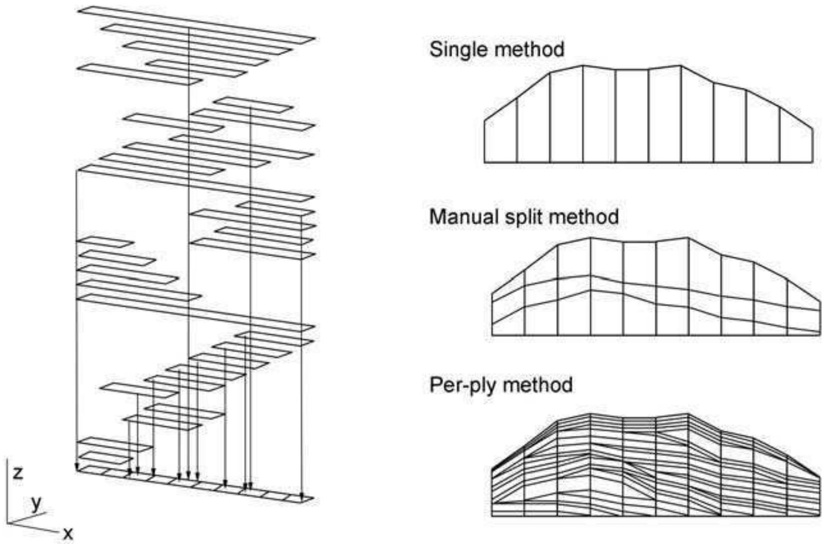
12.6.11 Solid element generation

Composite structures have traditionally been analysed using shell elements. For the linear regime, these elements typically use classical laminate theory to calculate effective in-plane and bending stiffnesses in the shell. Equilibrium methods are typically used to calculate approximate out-of-plane intralaminar stresses (i.e. bond stresses between plies). These methods continue to serve the industry successfully for relatively thin structures.

However, this approach does not yield sufficiently detailed recovery of out-of-plane shear and normal stresses. Consequently, specialized finite elements such as P/COMPOSITE from PDA Engineering were developed in the 1980s. These incorporated high-order shape functions and allowed the definition of plies at an arbitrary orientation in space at the Gauss points. These powerful elements, however, needed detailed models that were not economic to build, and use of these very advanced technologies has tapered off.

Over the past decade, general-purpose solid laminated composite elements have been incorporated in many mainstream finite element codes. These elements generally work under the following assumptions:

- The laminate material referenced by the element is sandwiched between opposite faces of the element. For example, in an 8-noded hexahedral element, the layers are parallel to the 1,2,3,4 and 5,6,7,8 faces of the element.
- If the sum of the layer thicknesses does not match the physical spacing of the



12.11 Solid generation from ply layup.

appropriate faces, the laminate material is effectively scaled to fill the gap or compress the excess material.

- The layers are assumed to cover the entire area of the appropriate faces of the elements, i.e. ply drop-off within the element is not supported.

These assumptions, particularly the requirement that the element faces are approximately parallel to the laminate layers, restrict the extent to which solid composites elements can be used effectively. However, this means that extrusion of the elements from a shell model is a highly appropriate method of model generation.

Recently, composites modelling tools have been extended to automatically generate solid composite elements by extruding shell models through an appropriate thickness. Sophisticated controls are provided, including the use of 'split' plies, to generate the required solid mesh as shown in Fig. 12.11.

12.7 Conclusions

The composites development process is complex and multidisciplinary, and can be facilitated by analysis tools at several key stages. Draping or forming simulation tools allow fibre orientations to be predicted across the component, and also allow determination of possible manufacturing problems. Coupled with modern techniques for definition of zones and plies, this information can be linked directly to the lay-up procedure. This data also informs structural analysis by ensuring that the correct material orientations and hence mechanical properties are specified in the different zones. Given increases in processing speeds, these techniques can be incorporated within design iteration and optimization schemes, ultimately leading to more efficient and effective composites design.

12.8 References

Anaglyph (2003) Laminate Tools 2.0 User Guide, Anaglyph Limited, London, UK.

Crookston J, Long A C and Jones I A (2002) Modelling effects of reinforcement deformation during manufacturing on elastic properties of textile composites, *Plastics Rubber and Composites* 31(2) 58–65.

Klintworth J W (2001) Multidisciplinary Optimization of Composites Structures, *International SAMPE Technical Conference, Seattle*, November 2001.

Klintworth J W (2005) Detailed Simulation of Primary Composite Structures, *SAMPE Long Beach*, May 2005.

Klintworth J W and MacMillan S (1992) Effective Analysis of Laminated Composite Structures, *Benchmark*, December, 20–22.

Long A C, Blanchard P J, Rudd C D and Smith P (1998) The development of an integrated process model for liquid composite moulding, *Composites Part A* 29(7) 847–854.

MSC (2003) MSC.Patran 2003 Release Guide, MSC.Software Corporation, Los Angeles, California, USA.

MSC (2005) MSC.Patran 2005 Release Guide, MSC.Software Corporation, Los Angeles, California, USA.