16 Knowledge-Based Engineering, Computer-Aided Design, and Finite Element Analysis

Computer-aided engineering (CAE) and knowledge-based engineering (KBE) are tools that are making a considerable impact on the design, manufacture, and through-life support of composite structures. Improvements in both the integrity of data and the accuracy of analysis have enabled engineering project teams to adopt CAE as a primary tool and reduce reliance on the manufacture of prototypes and experimental testing to prove design concepts. In composites manufacture, the applications of CAE include analysis and design using finite element analysis (FEA), design drafting, virtual prototyping, and the control of manufacturing processes, including control of robots and curing processes. KBE refers to processes that capture the intellectual property and core knowledge of the company. KBE applications include design rules that ensure compliance with best practice and tools to automate repetitive tasks.

16.1 Knowledge-Based Design Systems

In the context of this text, KBE is an engineering process in which knowledge about the product (e.g., the techniques used to design, analyze, and manufacture a product, and provide through-life support) is stored in the design system. The system captures the intellectual property of the company, including analysis experience, design rules, best practice, and advantages and limitations that define the best process to manufacture different components. It also provides tools to monitor the performance of the product so that operational experience can influence future design. The basis of the knowledge system is a database including a library of features, a list of rules, and checks on which the design can be based. These rules and checks are built into the design software and monitor the decision processes as the design evolves. By this means, the design process is controlled, and departures from recommended practice are immediately flagged. Applications in composite structures include the choice of manufacturing processes for a specific application, selection of materials, and preferred ply lay-ups. The design applications are also integrated into management software to support the decision-making process through accurate cost monitoring, support for marketing, and the evaluation of risk.

KBE can be applied at all stages of a project. The design parameters for the design of a riveted connection between a skin panel and supporting structure are shown in Figure 16.1. The dependence of the strength of the joint on fastener spacing and the distance to the edge of the panel is shown in Figure 9.24. If company policy selects bearing failure as the preferred failure mode, the minimum fastener spacing and fastener pitch can be defined. In addition, design practice may call for the edge distance to be based on the next larger fastener size. This allows for tolerance in drilling processes and allows the hole to be reamed and a new fastener inserted to execute a repair. This "best practice" can be implemented as rules and checks that are presented interactively to the design engineer using the system. Alternatively, they can be implemented in automated design software that is made available as a tool executed from the drafting system. Once data about the lay-up of the virgin panel and loads to be transferred by the joint are entered to the system, selection of the fastener to be used, fastener pitch, and edge distance can be automated.

KBE is not confined to structural details. Ribs (Fig. 16.2) are structural components that are repeated at regular intervals span-wise in an aircraft wing.

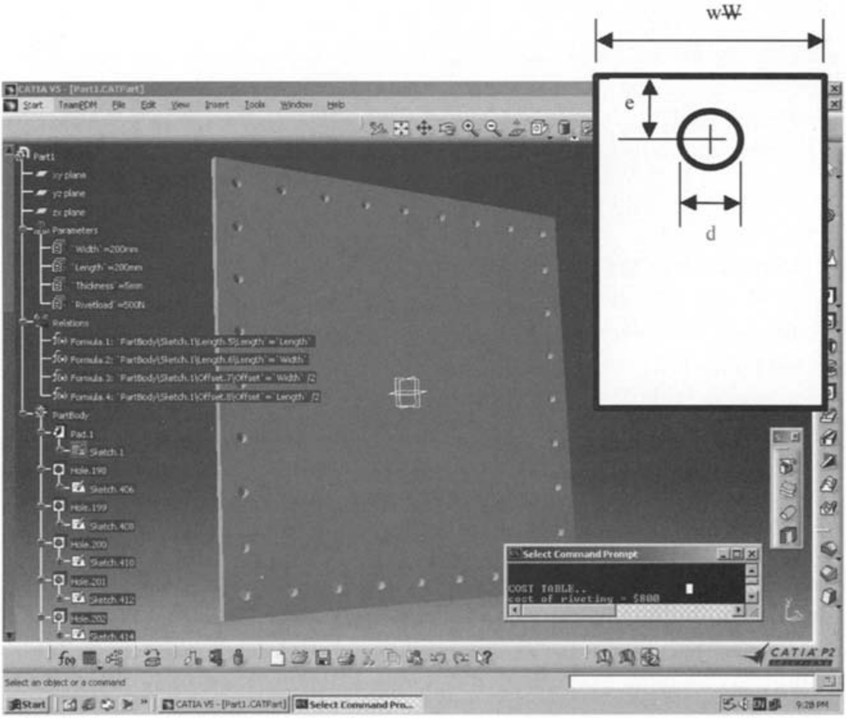


Fig. 16.1 Design detail in a riveted connection. Ratios e/d and w/d affect manufacture, repairability, and durability. Screen shows application of CATIA V5 (Ref. 1).

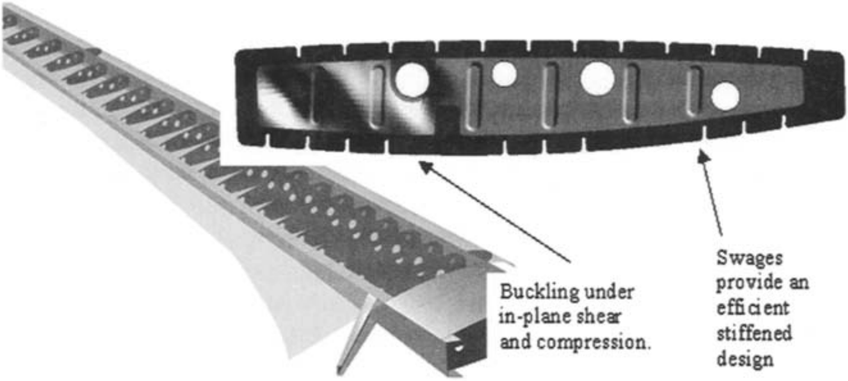


Fig. 16.2 Design of a structural component.

If the wing tapers, each rib will have different dimensions, defined by the profile of the wing and spars, as well as varying loads. The design of these structures can be automated using a KBE system so that each rib is individually optimized. The materials, structural details, manufacturing methods, and parts list can all be defined from a knowledge database. This database could include knowledge about the company's manufacturing facilities, a preferred design strategy including goals of minimum cost or weight, and experience with design detail, for example, the design of swages to prevent panel buckling or lay-ups to improve damage tolerance. Issues related to tooling and definition of the manufacturing process can be embedded in the software and can incorporate the company intellectual property (IP) and specialized capabilities that define the position of the company in the market place.

Commercial Modelling Systems[1,2] provide a familiar framework for the design process. They can be used for the geometric modelling of structure, can provide virtual prototypes for assessing and marketing the design, and can be used to check operational functionality such as the ability to gain access for inspection and maintenance. An interface with a finite element program provides the geometry for meshing with finite elements, and an interface with a manufacturing system provides the tool paths for manufacture of molds and fittings. Cost and weight estimates are immediately available. These systems can also contain the knowledge base discussed in the preceding text. The knowledge may include a library of pre-designed and producible components allowing repeated items such as cleats, swages, and joint details to be inserted by a "paste" operation. Scaling to local geometry will ensure structural compatibility.

The aim of these systems is to reduce the time for a new product to be developed and to enter service. By capturing best practice about design and manufacture of composite products, the occurrence of mistakes and design errors is minimized.

16.2 Finite Element Modelling of Composite Structures

The validation of designs requires analysis of the structural performance under service loads. The analytical procedures described in Chapter 6 can be applied to simple components such as a plate or a beam. Once an effective modulus is defined, these classical analytical techniques can be implemented, and deflections and stresses can be predicted from standard formulae.[3] However, when the structure is more complex, the classical techniques become more difficult to apply and analysts resort to numerical methods to achieve the answers they need.

The finite element method is the computational tool most widely used to validate the performance of structures. It provides the analysis required for the design process. Typically it can be used to define the displacements, stresses, vibration, and buckling characteristics of a structure composed of metal or composite materials under a defined set of loads and displacement boundary conditions. It can either be applied at the macroscopic level to analyze the stiffness and strength of the complete structure, or it can be applied at the microscopic level to study the interface between fiber and resin. Finite element analysis procedures are either embedded inside the geometric modelling systems that carry the geometry database or are stand-alone packages capable of special analysis, such as post-buckling behavior or response to shock loading. The finite element analysis can be applied to assess structural performance, to form the geometric model to which an optimization algorithm is applied, or to provide simulations of molding processes and manufacturing strategies.

Here we will concentrate on the use of finite element analysis (FEA) to assess structural performance. A simple description of the finite element method in structural analysis[4] is that it involves dividing the structure into discrete elements as shown in Figure 16.3. Each panel becomes an assembly of non-overlapping plate or brick elements that are connected at discrete points called nodes. The behavior of each element is defined by a relation between force and displacement at the nodes that are usually located on the boundaries of the element. The elements are then assembled into a structure by satisfying the equilibrium of the forces. This process is equivalent to pinning the elements together at the nodes. Constraint of the deformation on the element ensures an appropriate level of continuity of displacement and slope is maintained on the element boundary. The result is a set of linear algebraic equations that are solved to determine the displacements of the nodes.

A more sophisticated description of the finite element method[4] regards it as a piecewise polynomial approximation defined in terms of nodal displacements. The best values at the nodes are defined by minimizing a physically meaningful global quantity such as the total potential energy. This minimization generates the set of linear equations that are solved for the displacements. This formal mathematical approach provides the vehicle for applying the method to an ever-increasing variety of problems in applied mechanics. The extension to heat transfer and flow modelling is based on these generalized theories.

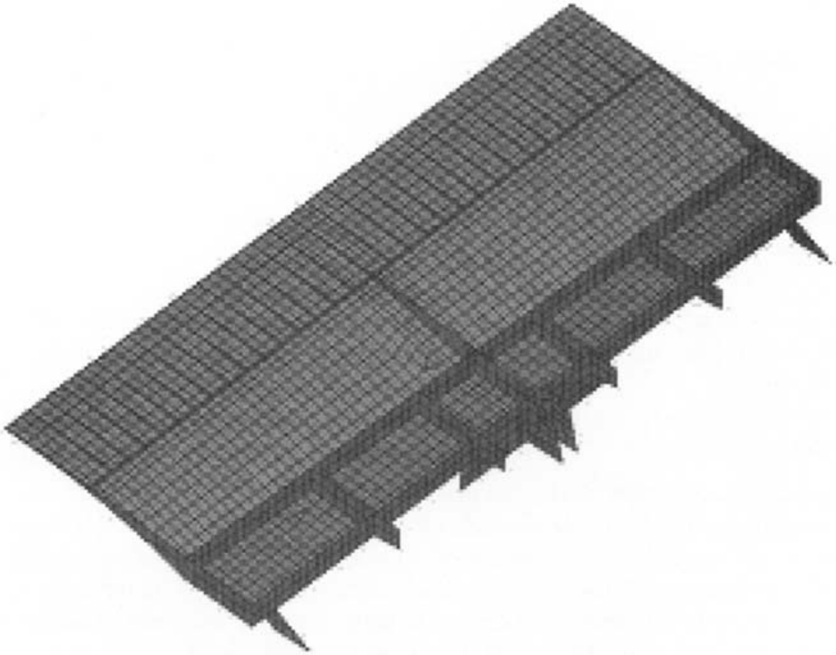


Fig. 16.3 Finite element model for a wing spoiler—top skin removed.

The finite element model, however, does not provide an exact match to experimental results. The theoretical processes are based on a numerical approximation related to the element size, the type of element used, the underlying theory, and the type of analysis performed. The modelling process involves approximations for geometry and may not reflect the true detail such as the change of the orientation of fibers during lay-up and cure. The stiffness of joints and imperfections such as the straightness of beams and flatness of panels can have considerable influence on the performance of actual structures.[5] The achievement of relevant and useful results relies on an understanding of the characteristics of the solution process and care when developing the numerical models.

16.3 Finite Element Solution Process

The finite element method provides the design team with information regarding the stiffness and strength of the structure. What confidence should the design team have in the results when analyzing composite structures? Unfortunately, the answer depends, in part, on how well the method is implemented. To indicate some of the features of the finite element method, the

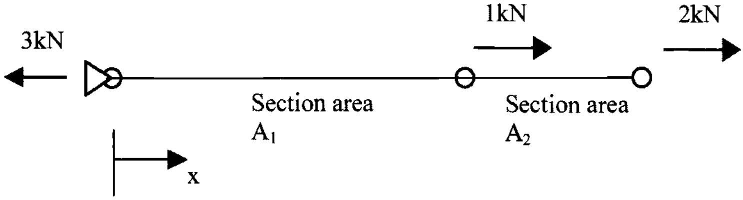


Fig. 16.4 A 1-D example to introduce FEA. A hollow tube of composite or metal is supported at its left-hand end and carries the loads shown.

one-dimensional problem shown in Figure 16.4 will be considered. The emphasis will be on explaining how the method works and how is it applied to composite materials.[6]

In all applications, as in this example, the development of the finite element model and completion of the analysis involves six basic steps:

Step 1. Select the type of analysis that will be executed. The selection of a full three-dimensional non-linear analysis is the most general but can lead to a model with a very large number of elements if the structure consists of thin panels. Finite elements need to be of moderate aspect ratio. The size of a three-dimensional brick element will therefore be governed by the minimum dimension (usually the thickness). As a result, engineers have developed beam, plate, and shell approximations to structural behavior. These approximations reduce the dimension of the problem. For example, in the classical laminate plate theory described in Chapter 6, the in-plane strains are assumed to be constant through the thickness in planar two-dimensional analysis, and linear through-the-thickness when the response includes bending. A finite element analysis based on plate elements will use these approximations to eliminate the thickness dimension. The finite elements are planar, and the size of each element is governed by the in-plane dimensions. The implications of assuming the behavior is linear will be discussed later.

The geometry considered here is that relevant to a hollow tube. The most detailed analysis could include accurate modelling of the tube wall in three dimensions with a layer of elements for each ply, modelling of the fixed support, modelling of the change in section, and how the load is applied. A simpler analysis would model the wall using plate or shell elements with the element lying in the plane of the wall, limiting the variation of strain through the thickness to a linear distribution and eliminating the modelling of more complex behavior.

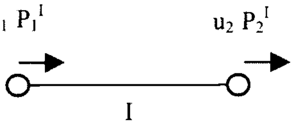


Fig. 16.5 The rod finite element used in the analysis.

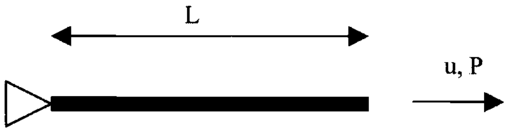


Fig. 16.6 Load on an isolated element.

Here, the simplest rod element shown in Figure 16.5 will be chosen. The stress and strain will be assumed constant over the cross-section of the tube, and the behavior will be assumed linear. The change in cross-sectional area of the tube where the 1-kN load is applied will occur at the boundary between elements. The detail of how the cross-section changes, that is, how the number of plies is reduced, will not be included in the rod element model.

Step 2. Define the relation between force and displacement at the nodes of the rod element.

For the geometry shown in Figure 16.6 and small axial strain: $\sigma = E\varepsilon$.

For the simple tube: $\sigma = P/A$ and $\varepsilon = u/L$. Therefore

$$\frac{P}{A} = E\frac{u}{L} \quad \text{or} \quad \frac{AE}{L}u = P. \tag{16.1}$$

A value is required for the longitudinal stiffness $(AE)_{\text{effective}}$ for the composite tube. The effective modulus can be defined using equation (*6.19*) using the lay-up defined for the walls of the tube. The true crossectional area of the walls of the tube is the required A.

Step 3. Use the element load/displacement relations to define the element matrix equations between nodal force and nodal displacement.

On element I shown in Figure 16.7, we can assume a relation between forces and displacements at the nodes in matrix form:

$$\begin{bmatrix} k_{11} & k_{12} \\ k_{21} & k_{22} \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \begin{Bmatrix} P_1^I \\ P_2^I \end{Bmatrix}$$

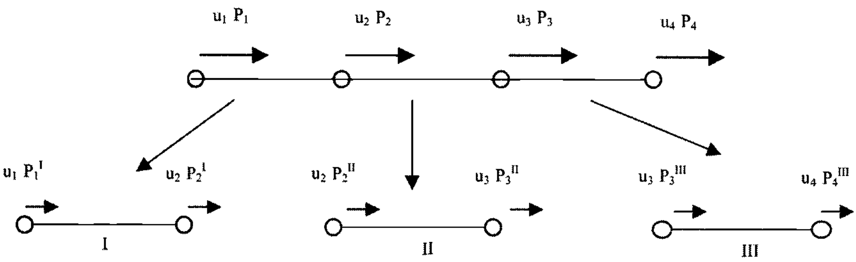


Fig. 16.7 Discretization into elements.

Set u_2 equal to 1 and u_1 equal to zero. Then, from equation (*16.1*),

$$P_1^I = -\frac{A_I E_I}{L_I} \quad \text{and} \quad P_2^I = \frac{A_I E_I}{L_I}$$

Substituting in the matrix equation:

$$\begin{bmatrix} k_{11} & k_{12} \\ k_{21} & k_{22} \end{bmatrix} \begin{Bmatrix} 0 \\ 1 \end{Bmatrix} = \begin{Bmatrix} -\dfrac{A_I E_I}{L_I} \\ \dfrac{A_I E_I}{L_I} \end{Bmatrix}$$

giving

$$k_{12} = -\frac{A_I E_I}{L_I} \quad \text{and} \quad k_{22} = \frac{A_I E_I}{L_I}$$

Defining a similar problem with u_1 equal to 1 and u_2 equal to zero identifies:

$$k_{11} = \frac{A_I E_I}{L_I} \quad \text{and} \quad k_{21} = -\frac{A_I E_I}{L_I}$$

Combining these results gives the matrix relation for element I.

$$\begin{bmatrix} \dfrac{A_I E_I}{L_I} & -\dfrac{A_I E_I}{L_I} \\ -\dfrac{A_I E_I}{L_I} & \dfrac{A_I E_I}{L_I} \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \begin{Bmatrix} P_1^I \\ P_2^I \end{Bmatrix} \tag{16.2}$$

Step 4. Define the element mesh. In this step, the number of degrees of freedom in the model is set.

The elements are connected at nodes at which there are displacement degrees of freedom. The displacements on the element, and hence the stresses and strains, are uniquely defined by the displacements at the nodes on the element. The number of degrees of freedom in the model, and hence the accuracy of the approximation, is therefore linked directly to the number of elements.

Two factors that affect the accuracy of the finite element model have therefore been defined—the decision to base the model on a simple rod element and the design of the mesh. The size of the analysis model may be restricted by the memory available in the computer and may also be limited by the time a designer is prepared to wait for the solution once an analysis request is submitted.

Here a hand calculation is to be executed. Therefore the number of elements in the model will be restricted to, for example, three. The model shown in Figure 16.7 has four nodes with only four degrees of freedom. The degrees of freedom are the axial displacements u_I at each node.

The matrix relation in equation (*16.2*) is defined for each element.

On element I
$$\begin{bmatrix} \frac{A_I E_I}{L_I} & -\frac{A_I E_I}{L_I} \\ -\frac{A_I E_I}{L_I} & \frac{A_I E_I}{L_I} \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \begin{Bmatrix} P_1^I \\ P_2^I \end{Bmatrix}$$

On element II
$$\begin{bmatrix} \frac{A_{II} E_{II}}{L_{II}} & -\frac{A_{II} E_{II}}{L_{II}} \\ -\frac{A_{II} E_{II}}{L_{II}} & \frac{A_{II} E_{II}}{L_{II}} \end{bmatrix} \begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} P_2^{II} \\ P_3^{II} \end{Bmatrix}$$

On element III
$$\begin{bmatrix} \frac{A_{III} E_{III}}{L_{III}} & -\frac{A_{III} E_{III}}{L_{III}} \\ -\frac{A_{III} E_{III}}{L_{III}} & \frac{A_{III} E_{III}}{L_{III}} \end{bmatrix} \begin{Bmatrix} u_3 \\ u_4 \end{Bmatrix} = \begin{Bmatrix} P_3^{III} \\ P_4^{III} \end{Bmatrix}$$

where L_i, A_i and E_i are the length, cross-sectional area, and effective modulus of the ith element.

Step 5. Assemble the global equations by applying equilibrium of the forces at each node.

In this step, the global geometry is assembled from the smaller "finite" elements. Note that the fundamental principle of equilibrium is used. A second physical concept—that the structure must remain connected under load—is also enforced by the assumption that there is only one displacement at each node and that displacement is shared by both neighboring elements. Note the overlap of the displacements between the matrix relations defined in Step 4.

The forces on the nodes are identified in Figure 16.8. Note that the element forces defined in Figures 16.5 and 16.7 are forces applied to the element. The forces are reversed in Figure 16.8 because they are forces applied by the elements onto the nodes.

Applying equilibrium at node 1,

$$-P_1^I + P_1 = 0 \text{ or substituting for } P_1^I \text{ from the matrix relation}$$

$$\left(\frac{A_I E_I}{L_I} u_1 - \frac{A_I E_I}{L_I} u_2\right) = P_1$$

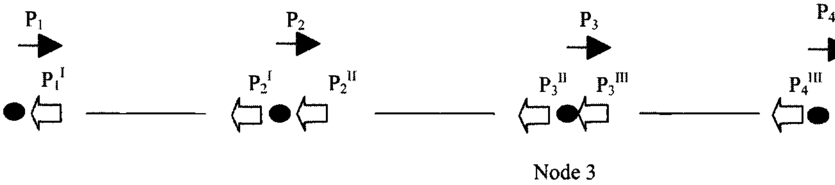


Fig. 16.8 Applying equilibrium at the nodes.

At node 2,

$$-P_2^I - P_2^{II} + P_2 = 0 \text{ or substituting from the matrix relation}$$

$$\left(-\frac{A_I E_I}{L_I} u_1 + \left(\frac{A_I E_I}{L_I} + \frac{A_{II} E_{II}}{L_{II}}\right) u_2 - \frac{A_{II} E_{II}}{L_{II}} u_3\right) = P_2$$

At node 3,

$$-P_3^{II} - P_3^{III} + P_3 = 0 \text{ or substituting from the matrix relation}$$

$$\left(-\frac{A_{II} E_{II}}{L_{II}} u_2 + \left(\frac{A_{II} E_{II}}{L_{II}} + \frac{A_{III} E_{II}}{L_{III}}\right) u_3 - \frac{A_{III} E_{III}}{L_{III}} u_4\right) = P_3$$

At node 4,

$$-P_4^{III} + P_4 = 0 \text{ or substituting from the matrix relation}$$

$$\left(-\frac{A_{III} E_{III}}{L_{III}} u_3 + \frac{A_{III} E_{III}}{L_{III}} u_4\right) = P_4$$

Assembling these equations into a matrix gives:

$$\begin{bmatrix} \frac{A_I E_I}{L_I} & -\frac{A_I E_I}{L_I} & 0 & 0 \\ -\frac{A_I E_I}{L_I} & \frac{A_I E_I}{L_I} + \frac{A_{II} E_{II}}{L_{II}} & -\frac{A_{II} E_{II}}{L_{II}} & 0 \\ 0 & -\frac{A_{II} E_{II}}{L_{II}} & \frac{A_{II} E_{II}}{L_{II}} + \frac{A_{III} E_{III}}{L_{III}} & -\frac{A_{III} E_{III}}{L_{III}} \\ 0 & 0 & -\frac{A_{III} E_{III}}{L_{III}} & \frac{A_{III} E_{III}}{L_{III}} \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{Bmatrix} = \begin{Bmatrix} P_1 \\ P_2 \\ P_3 \\ P_4 \end{Bmatrix}$$

$$\text{or} \quad Ku = P \tag{16.3}$$

Step 6. Obtain the solution for displacements and stresses.

The loads and boundary conditions are now defined. For a unique solution, either the displacement or the load must be defined at each node, but not both. Application of this rule ensures that the number of unknowns in the matrix relation is equal to the number of equations. At fixed nodes a reaction will exist, but that reaction is determined after the solution for displacement. For the problem defined in Figure 16.4, the displacement u_1 is set to zero and the corresponding load becomes the reaction R_1. P_2 is set to zero because there is no external load at node 2. Solution for the unknown degrees of freedom u_2, u_3, u_4, and the unknown reaction R_1 proceeds by a standard matrix equation algorithm such as Gaus elimination.

The displacements at the nodes have now been defined. The stresses are determined by application of the element relations used in Step 2. A further subtle point arises. We note that the relations used in Step 2 depend on the deformation of the elements. This means that the set of equations contains no information about the absolute displacements—only the relative displacements. We must, therefore, ensure that the boundary conditions fix all rigid body motion, otherwise the solution process will fail. This failure manifests itself as a singular matrix K. In mathematical terms, this means the set of equations has no unique solution because the reference points relating deformation of the structure to absolute displacements have not been defined.

The material used in manufacturing the rods only enters the analysis in Step 2, where the relation between stress and strain is defined. Whether the rod is of circular, square, or more complicated cross-section is of no consequence to the following analysis. Only the cross-sectional area and the effective longitudinal modulus of the material are required. If the tube is manufactured from plies, the laminate analysis developed in Chapter 6 can be used directly. Most commercial finite element codes have laminate modellers included for this purpose. These modellers produce the coefficients A_{ij}, B_{ij}, and D_{ij} in equation (*6.27*) when defining the element matrices required for plate and shell analysis.

In addition, if different materials are mixed in the structure, this presents no difficulty to the analysis. One element could be made from carbon fiber and a neighboring element from glass fiber or metal. If materials are mixed within the element, the accuracy of the analysis will depend on the accuracy of the effective modulus defined in Step 2.

Often it is possible to model the material behavior as linear, especially when the response is dominated by stiff fibers. In linear analysis, the material properties are assumed constant, and deformation of the geometry under load is assumed to be small. Solution then follows by solving the matrix equation $Ku = P$ for the displacements. If however, the material properties depend on the strain in the material as indicated in Figure 16.9, or if deformations are large enough to

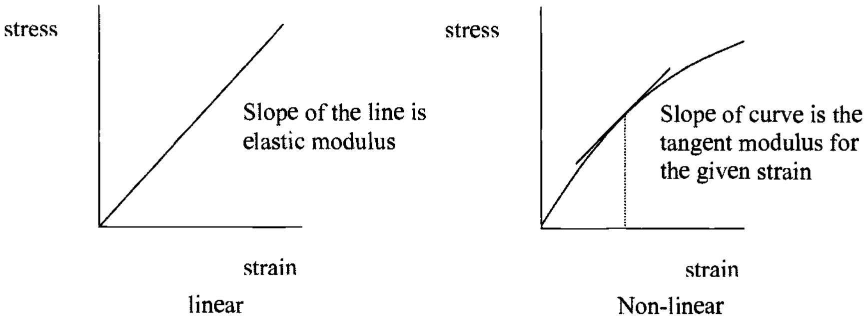


Fig. 16.9 Linear and non-linear material behavior.

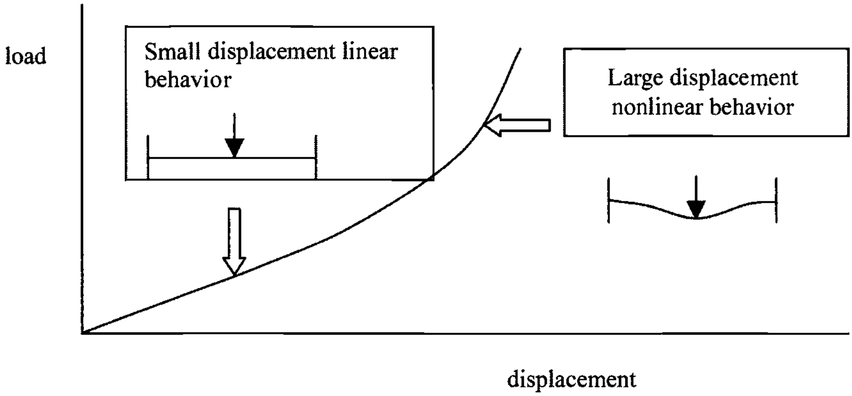


Fig. 16.10 Linear and non-linear structural behavior. Note: an additional load path is introduced by the deformation of the structure.

cause geometry changes that modify the equilibrium relations as indicated in Figure 16.10, then a non-linear solution must be executed.

These non-linear solutions require iteration because the strain levels and deformation are not known in advance. Algorithms such as Newton-Raphson are used to execute the analysis. Typically, the load application is divided into a number of small steps and properties are defined from the current state of stress and strain in the structure (initially zero). After completion of an analysis, defined in Steps 4 and 5, an initial estimate of the stress and strain at the end of that load step is obtained. These stresses and strains allow the departure from the true behavior to be assessed, and the analysis is repeated until the updated properties converge.[4]

Commercial finite element systems can execute these more complicated analyses. However, the analyst must define the non-linear material properties and specify appropriate load levels and boundary conditions. All solutions are to some extent non-linear, but the iteration process can be expensive, and the definition of data to drive a non-linear analysis, such as a complete stress/strain curve for the material, can be difficult. Part of the skill required to execute an engineering analysis is in knowing when the linear approximation is adequate.

It is important to realize that the finite element solution is approximate. Equilibrium is satisfied at the nodes, but it is usually not satisfied on the elements or, in the two-dimensional and three-dimensional elements, across the boundary between elements. We expect the solution to converge as the element size is reduced. However, the solution will only converge to the mathematical solution of the theory implemented in the finite element formulation. For example, several assumptions are listed for the classical laminate theory described in Chapter 6 for linear analysis. Plate elements defined using this theory will converge to the solution afforded by laminate theory.

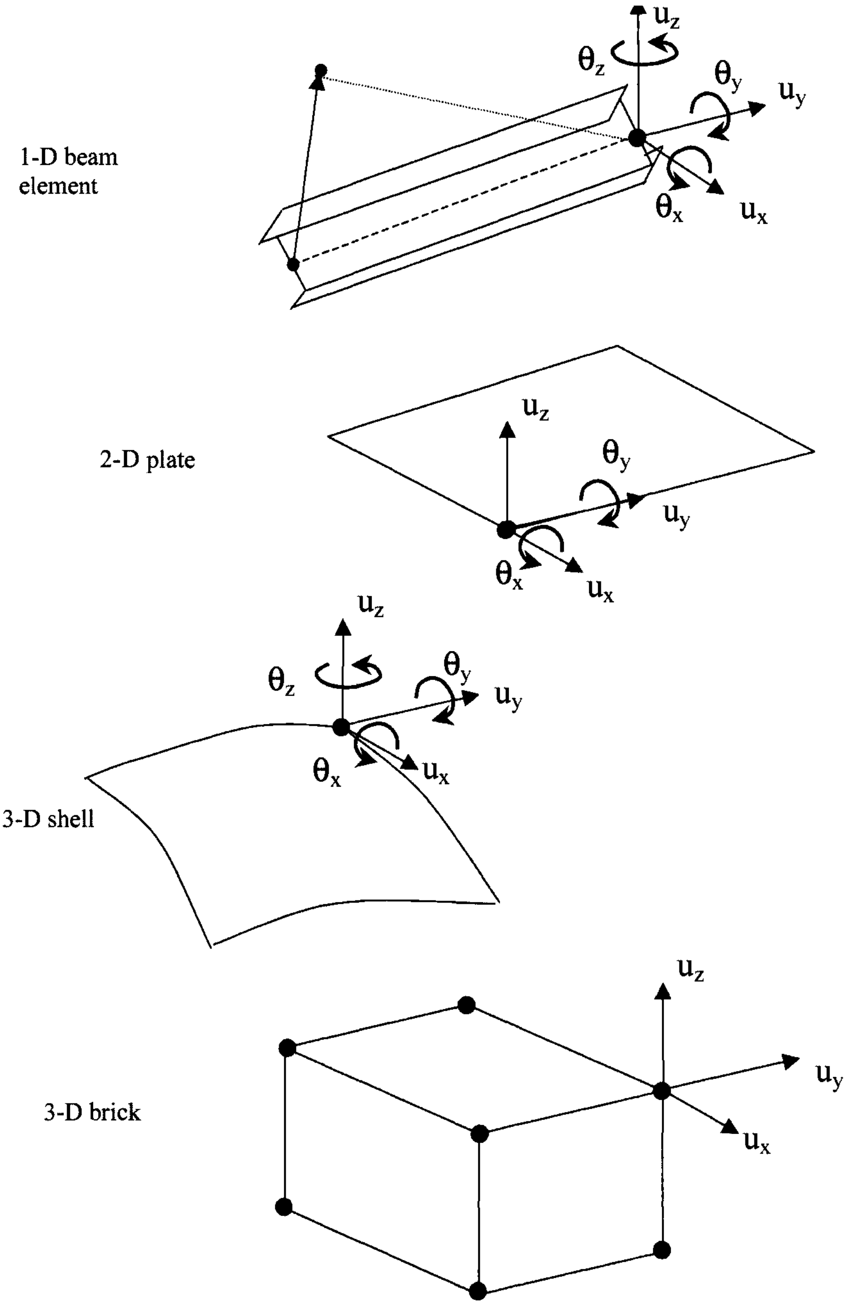


Fig. 16.11 Element types in the element library of a commercial FEA system.

The geometric model is also not exact. If the finite elements are based on an assumption that the fibers remain in their design orientation and are not realigned in the draping process, then the solution will converge to the exact solution for this configuration. The analysis described above using rod elements will also give no information about the detailed stresses at the point where the tube cross-section changes. A planar axi-symmetric or full three-dimensional analysis, which includes the geometry detail where the section changes, is required to generate these stresses.

At the end of Step 6, the analyst has information available to help make decisions about the accuracy of the modelling process. If a linear analysis was executed, the stress levels can be checked to determine whether the value of the stiffness modulus used in Step 3 was appropriate and the displacements can be checked to determine whether the predicted deformation of the structure is sufficient to introduce non-linear effects due to geometry change. If the answer to either question is "yes" then a non-linear analysis is required and the analysis returns to Step 3 with appropriate algorithms to update the solution and iterate towards the correct result.

16.4 Element Types

A finite element system contains a library of finite elements,[6] as indicated in Figure 16.11. The most general three-dimensional representation is given by brick elements. In a micro-mechanical model, these elements can be used to model individual fibers and resin to investigate, for example, stresses caused by thermal shrinkage of the resin relative to the fiber in the failure theories considered in Chapter 6.

For problems involving the behavior of structural components, such as the spoiler in Figure 16.3 models built with bricks can involve an unmanageable number of elements. A feature of digital computers is that the storage of numbers and performance of algebraic operations is not exact but is determined by the number of digits used to represent each number in the computer memory. If a brick element becomes long and thin, its stiffness in the shorter dimensions becomes much higher than its stiffness in the longer dimensions [see the dependence on $1/L$ in equation (*16.1*)]. Bending stiffness can vary with the cube of the length, and therefore only a moderate aspect ratio can be achieved before the terms in equation (*16.3*) will have vastly different magnitude and numerical instability becomes a problem.[4] The rule is that brick elements should be simple cubes, or at least should be of moderate aspect ratio. Therefore, if the cross-section of a structural beam or a thin plate was to be modelled with bricks, an extraordinary number of cube-like elements would have to be used.

To overcome these problems, finite element systems include elements that are based on beam, plate, and shell theory. These theories have been developed by engineers to analyze special structural configurations. The beam element is used for structural members for which the required stiffness and load paths can be represented by one-dimensional straight and curved elements. The calculations in the plane of the section are completed mathematically when the element is formed. The finite element model is then used to determine the response dependent on the axial stiffness, EA; the bending stiffness, EI; and the torsional stiffness, GJ. The difference in stiffness in the plane of the section compared with the longitudinal axis of the beam is then removed from the analysis.

The same is true for plates. The plan dimensions of a brick representing a plate are limited by the through-thickness dimension. Therefore, a large number of bricks are required to model a relatively simple plate. A brick model of the assembled set of plates that comprise a structure, such as that presented in Figure 16.3, is usually not feasible. However, if the limitations defined at the beginning of Chapter 6 are acceptable, the assumption that the strain varies linearly through the thickness can be used to integrate the through-thickness behavior as the elements are assembled, leaving the deformation defined by mid-plane strains and curvatures.

Plate elements are therefore two-dimensional and lie on the mid-plane of the surface they represent. The size of the elements is not linked to the thickness of the plate and the mesh requirement is that the elements be 'near-square' in plan view and small enough to capture the variation in stresses and strains caused by geometry features in the plane of the plate. A shell is similar to a plate, but the surface can be curved.

16.5 Finite Element Modelling of Composite Structures

Structural laminates can be modelled using the two-dimensional planar plate elements. Beams of various sections can also be assembled from plate elements if plate elements are used to represent the webs and flanges of the beam. In this approximation, the stresses acting through the thickness are not modelled. A pressure load applied to the plate can be approximated as varying linearly from the value equal to the applied pressure at the surface on which the pressure is applied, to zero, on the unloaded surface. Shear stresses with through-thickness components are defined in a post-processing process that applies plate theory.

If accurate through-thickness stresses are required, the model must be based on three-dimensional elements with, at least, one brick element in the through-thickness direction for each stack of plies with the same orientation.

The simplification of the model to one-dimensional rod and beam elements involves significant approximations and is only implemented when the beams are slender and simple axial deformation and bending theory is a good approximation to the structural response.

The steps taken in Section 16.3 are the same whether the structure is made from metal or composite. In the derivation of all types of finite elements, the modification required to allow the analysis of composite rather than a metal structure lies only in the definition of the material properties—the relationship between stress and strain—in Step 2.

16.5.1 Plate and Shell Elements

Plate and shell elements can be used to model structures composed of laminates such as the wing spoiler in Figure 16.3.

16.5.1.1 Plate and Shell Elements for Laminates Constructed from Orthotropic Ply Material. For laminates constructed from orthotropic plies, the laminate theory of Chapter 6 is used to define the properties of the plate elements. Pre-processors in finite element systems allow the lay-up to be defined. First, a set of orthotropic properties is defined for the unidirectional or fabric plies to be used in the laminate. A tabular input is then used to define the orientation, thickness, and material for each ply. The properties required for the plate are then automatically calculated. As an added feature, the coefficients A_{ij}, B_{ij}, and D_{ij} defined in the laminate theory can be listed together with the effective orthotropic elastic constants E_{11}, E_{22}, etc.

16.5.1.2 Woven and Knitted Material (Unit Cell Approach). Defining the elastic constants for a woven material is more complex, and a unit cell approach is often adopted.[7] In this approach, equivalent elastic constants are derived using a detailed finite element model of a representative unit cell. Actual fiber architectures are modelled using brick elements for the fiber tow and resin. Displacements are then applied to the cells to isolate the effective longitudinal, bending, and torsional components of deformation required to define the elastic matrices describing the laminate behavior. These constants are then available for subsequent analyses.

16.5.1.3 Brick Elements. Brick elements can be used to model individual layers in a laminate, or a single element can be used to model several layers. When used to model layers, an "average" property for the combination of resin and fibers has to be defined. The stress-strain relationship for a general anisotropic material involves all 21 coefficients in the symmetric matrix $[G]$ defined below. The 21 coefficients are defined in terms of 12 elastic constants.

The three-dimensional stress-to-strain relationship for an orthotropic material representing a layer of fabric or unidirectional tape is simpler and is defined by:

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{Bmatrix} = [G]\left[\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{Bmatrix} - (T - T_{REF})\begin{Bmatrix} \alpha_x \\ \alpha_y \\ \alpha_z \\ \alpha_4 \\ \alpha_5 \\ \alpha_6 \end{Bmatrix}\right]$$

where for an orthotropic material

$$[G] = \begin{bmatrix} G_{11} & & & & & \\ G_{12} & G_{22} & & \text{symmetric} & & \\ G_{13} & G_{23} & G_{33} & & & \\ 0 & 0 & 0 & G_{xy} & & \\ 0 & 0 & 0 & 0 & G_{yz} & \\ 0 & 0 & 0 & 0 & 0 & G_{zx} \end{bmatrix}$$

and

$$G_{11} = \frac{1 - \nu_{yz}\nu_{zy}}{E_y E_z \Delta} \quad G_{12} = \frac{\nu_{yz} + \nu_{zx}\nu_{yz}}{E_y E_z \Delta} \quad G_{13} = \frac{\nu_{zx} + \nu_{yx}\nu_{zy}}{E_y E_z \Delta}$$

$$G_{22} = \frac{1 - \nu_{xz}\nu_{zx}}{E_x E_z \Delta} \quad G_{23} = \frac{\nu_{zy} + \nu_{xy}\nu_{zx}}{E_x E_z \Delta} \quad G_{33} = \frac{1 - \nu_{xy}\nu_{yx}}{E_x E_y \Delta}$$

where ν_{ij} = Poisson ratios; E_x, E_y, E_z = Young's modulus in the x, y, and z directions; G_{xy}, G_{yz}, G_{zx} = Shear moduli; and

$$\Delta = \frac{1 - \nu_{xy}\nu_{yx} - \nu_{yz}\nu_{zy} - \nu_{xz}\nu_{zx} - 2\nu_{yx}\nu_{zy}\nu_{xz}}{E_x E_y E_z}$$

16.5.1.4 Rod and Beam Elements. Rod and beam elements are seldom used in the analysis of composite components because the laminate theory lends itself to the definition of plate elements, and plate elements can usually be used effectively to model the webs and flanges of the standard beam sections. In Figure 16.12, plate and brick element models are defined for an I-beam. The plate model shown in Figure 16.12 can accurately represent the bending and torsional stiffness of the beam including (for more general sections) the

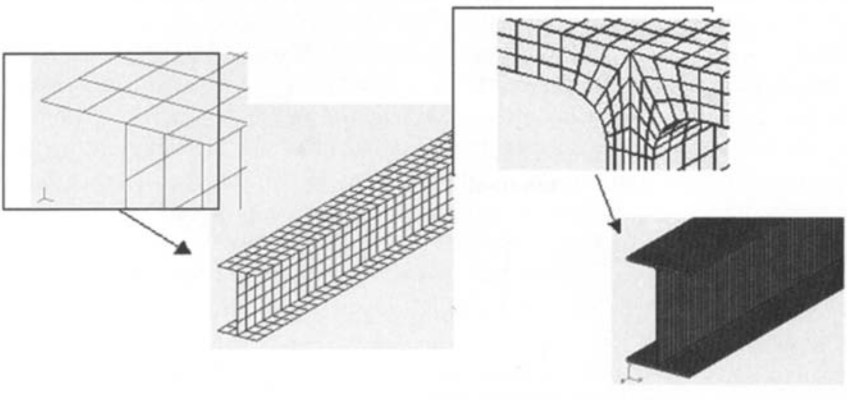


Fig. 16.12 Plate and brick models for an I-beam.

interaction between bending and torsion when the shear center is offset from the centroid of the section. In addition, local buckling modes involving the buckling of the flanges and web of the beam will be predicted, in addition to the global modes of Euler beam buckling when a buckling analysis is executed. However, the model cannot predict the detailed stress and strain distribution at the intersection of the web and flanges. If details of stress and strain are required at this intersection, a three-dimensional brick analysis is required, as indicated in the Figure.

If the structure is comprised of beams which are slender and transfer load by axial, bending and torsional components, then rod and beam elements can be used. The success of the analysis, summarized in Section 16.3 for the rod element, depends on the definition of the effective stiffness components.

16.6 Implementation

Once the details of how the composite material will be modelled have been resolved, the full power of the finite element analysis becomes available. The finite element method can be applied to structures of arbitrary shape and can include advanced applications including:

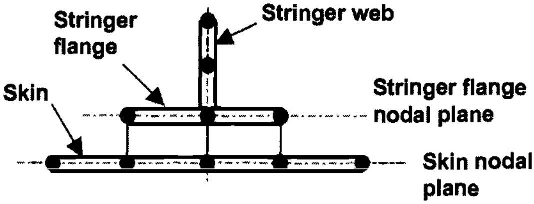
- Non-linear analysis for post-buckling behavior modes of failure, and ultimate collapse
- Static or dynamic solutions, including crash simulation
- Optimization.

16.6.1 Post-Buckling Performance and Stiffener Separation

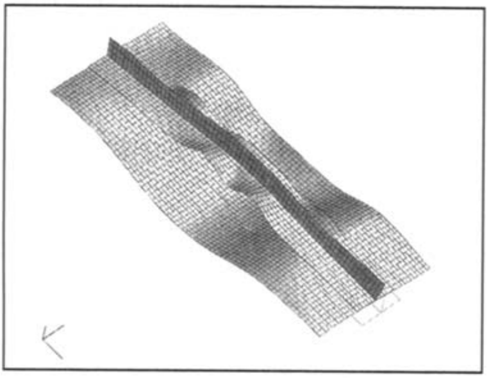
Local failure of composite components has been discussed in Chapter 6. However, failure of structural assemblies was not considered. The analysis of these structures and the prediction of failure rely on finite element analysis.

For example, composite structures are often composed of assemblies of stiffened plates. Local buckling can be allowed to occur below the ultimate failure load in these stiffened structures so long as the residual strength of the structure is sufficient to carry the applied loads. Analysis is required to predict the post-buckling behavior as well as ensuring that buckling does not trigger separation of the stiffeners that would lead to a significant loss of strength. A geometric non-linear analysis is required to predict the post-buckling behavior of the panel shown in Figure 16.13b, and a fracture mechanics approach is used to predict growth of a disbond between the panel and the stiffener.[8] The initial disbond weakens the panel and is introduced into the analysis to prove its damage tolerance.

The analysis[8] is based on plate elements in a commercial finite element system.[9] The panel and stiffener have been modelled separately and connected by



a) Modelling using plates and rigid elements



b) Post-buckling deformation

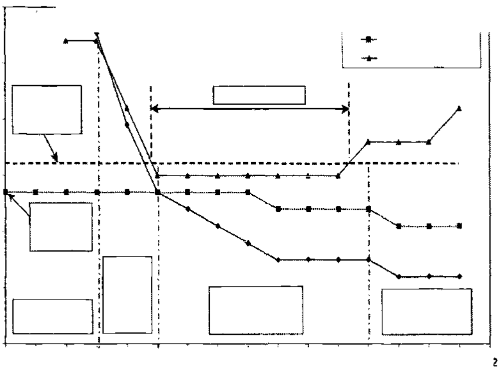


Fig. 16.13 Finite element analysis of stiffener separation.

rigid links. The rigid links are removed to model the area affected by the disbond. A virtual crack extension procedure is used to determine the energy release rate in modes I and II crack extension and predict growth of the disbond. Studies were completed to determine the buckling loads and the post-buckling behavior of a damaged panel in an empennage structure, the critical length of the disbond, and the residual strength of the damaged panel.

16.7 Design Optimization

The combination of knowledge-based systems and optimization algorithms allow an automated design process to include both performance and cost objectives in the design analysis. For example, the structure in Figure 16.3 was subject to an optimization process searching for a design that combined low weight and low manufacturing cost.[10] The structural performance under pressure loads is controlled by ply orientation in the laminate, by the thickness of the panels, and by the geometry, thickness, location, and shape of the stiffening members. All can be varied in a finite element model to satisfy strength, stiffness, and buckling requirements.

Procedures available in commercial finite element systems[11] include a topology optimization system,[12] to identify the principle structural members, and a parametric design algorithm that allows variables in the optimization algorithm to include dimensions such as the location of the stiffening members. Successful applications of these algorithms[10] do not necessarily require an automated process; rather, the algorithms can be used to guide the evolution of the design and help the design team to evaluate a number of different design options in a concurrent engineering approach. As the design evolves, so does understanding of the load paths in the structure and the redundancy that provides damage tolerance, often a primary design requirement.

16.7.1 Cost Estimates for Design Optimization

The design of weight-effective aerospace structures can no longer guarantee the success of the product. Manufacturers need to verify that their products are an optimal tradeoff between minimum weight, cost, and risk, all within a time frame that meets time-to-market needs. Costs here are taken to mean whole-of-life costs including manufacturing, operation, and maintenance costs. Optimization of these objectives must be carried out in the conceptual design phase because, once the design concept is fixed, up to 80% of the whole-of-life costs will also be fixed.[13] Further detail design alterations may not produce substantial reductions in manufacturing cost or reduce the requirements for through life support.

Implementation of CAE gives the designer the opportunity to enhance the understanding of the functionality of the design and to determine the primary drivers for cost and weight. Preliminary cost estimates for setting cost and weight targets can be based on \$/kg and $\$/\text{m}^2$ parameters, validated using data from completed projects. These estimates can be enhanced by identifying cost drivers that represent the complexity of the design, ensuring the cost estimate is not simply proportional to weight or size.[14]

As the design evolves, these cost estimates cannot reflect the effect of detailed design changes on cost. In the final stages of the design, the cost will rise as weight is removed from the structure. One approach to estimating cost is based on the Process Costing Analysis Database (1992–1997).[15] This approach separates a manufacturing process into discrete steps and identifies the parameters from which manufacturing time can be estimated. For example, the time required for a non-destructive inspection can be related to the length of a joint that is to be inspected for defects. Cost estimates follow by applying an appropriate hourly rate for the cost of labor.

Design and analysis of efficient composite structures is a complex process. Successful exploitation of these materials will depend on the efficient use of the latest CAE and KBE tools.

References

[1] CATIA Proprietary Software, Dassault Systems, www.catia.com.

[2] ICAD Proprietary Software, Knowledge Technologies International. www.ktiworld.com

[3] Young, W. C., *Roark's Formulas for Stress and Strain*, 7th ed., McGraw-Hill, 2000.

[4] Cook, R. D., *Finite Element Modeling for Stress Analysis*, Wiley, 1995.

[5] Adams, V., and Askenazi, A., *Building Better Products with Finite Element Analysis*, Onward Press, 1999.

[6] Matthews, F. L., Davies, G. A. O., Hitchings, D., and Soutis, C., *Finite Element Modeling of Composite Materials and Structures*, Woodhead Publishing, 2000.

[7] Tan, P., Tong, L., and Steven, G., "Micromechanics Models for the Elastic Constants and Failure Strengths of Plain Weave Composites," *Composite Structures*, Vol. 47, 1999, pp. 797–804.

[8] Yap, J., Scott, M., Thomson, R., and Hachenberg, D., *The Analysis of Skin-to-Stiffener Debonding in Composite Aerospace Structures, ICCS-11*, Monash Univ., Melbourne, Australia, Nov. 2001.

[9] MSC.NASTRAN Proprietary software of the MacNeal Schwendler Corporation, www.msc.com.

[10] Raju, J., "A Conceptual Design and Cost Optimization Methodology," *Proceedings of the 44th AIAA/ASME/ASCE/AHS Structures, Structural Dynamics and Materials Conference*, April 2003.

[11] ANSYS Proprietary software of Swanson Analysis Inc, www.ansys.com.

[12] Bendsoe, M., Diaz, A., and Kikuchi, N., "Topology and Generalised Layout Optimization of Elastic Structures" *Topology Design of Structures*, edited by M. P. Bendsoe and C. A. Mota Soares, Kluwer Academic Publishers, Netherlands, pp. 159–205.

[13] Wang, K., Kelly, D., Dutton, S., "Multi-objective Optimisation of Composite Aerospace Structures," *Composite Structures*, 2002, pp. 141–148.

[14] Hinrichsen, J., "A380—Flagship Aircraft for the New Century," *SAMPE Journal*, Vol. 38, No. 3, 2002, pp. 8–12.

[15] Gutowski, T. G., Neoh, E. T., and Polgar, K. C., *Adaptive Framework for Estimating Fabrication Time of Advanced Composite Manufacturing Processes*, Technical Report, Laboratory for Manufacturing and Productivity, Massachusetts Institute of Technology, Cambridge, MA, 1995.