

**Military Technical College
Kobry El-Kobbah,
Cairo, Egypt.**



**15th International Conference
on Applied Mechanics and
Mechanical Engineering.**

A COMPUTATIONAL CODE FOR THE DESIGN OF THREE-DIMENSIONAL COMPOSITE STRUCTURES

A. Ramsaroop¹ and K. Kanny²

ABSTRACT

Conventional design techniques for fibre reinforced composite laminates prove to be quite laborious. These methods involve numerous matrix calculations which make manual computations very tedious. To overcome this, a designer uses a finite set of fibre angles in a design; however the chosen angles may not be the optimum ones for the structure. Therefore MATLAB codes were developed to assist in the design process. These codes remove the guesswork experienced in the conventional design methods and perform all the necessary computations quickly and efficiently as well as being able to design three-dimensional structures. The developed codes were verified via finite element analyses. The development and operation of these codes are discussed further.

KEYWORDS

Composite Design, Hooke's Law, MATLAB coding, Finite Element Analysis.

-
- 1 Doctoral Student, Department of Mechanical Engineering, Durban University of Technology, Durban, South Africa, E-mail: avinashr@dut.ac.za.
 - 2 Professor, Department of Mechanical Engineering, Durban University of Technology, Durban, South Africa, Tel: +2731 373 2230; Fax: +2731 373 2139; E-mail: kannyk@dut.ac.za.

INTRODUCTION

Fibre reinforced composites are extremely versatile materials and have the ability to be tailor designed to suit a variety of applications. Designers strive to achieve the best design with the lowest weight and least cost [1-2]. Lower weight means easier installation and less labour, which consequently reduces costs. A further reduction in cost may be achieved by designing the structure with minimal material usage. However the conventional methods employed in the design of composite structures do not allow for this.

Conventional design techniques and finite element packages involve the use of Hooke's law for composite materials. Equations relating the stresses and strains in these materials have been developed and are available from various texts [3-6]. These equations are limited to flat two-dimensional unidirectional fibre reinforced composite laminates and are discussed in the next section. The use of the equations is quite laborious [7], and involves numerous matrix computations which make manual calculations tedious.

The equations based on Hooke's law require that the fibre angles, layer thicknesses and number of layers be known quantities but these are unclear at the onset of a design. For this reason as well as the complexity of manual calculations, designers usually have a finite set of fibre angles that they use in their designs [8]. However, choosing from a finite set of angles may reduce the difficulty of the computations, but the chosen angles may not be the optimum ones for the structure. The end effect may be failure or overdesign of the structure.

An overdesign leads to an increase in manufacturing costs as more material and labour would be required. A failed outcome would require redesigning and performing all the calculations again with modified angles, thicknesses or number of layers, or a combination of these. It would not be known which input values to change and by what quantity. This leads to a great deal of uncertainty and guesswork.

Therefore it was decided to develop computer algorithms that aided in the design process of composite structures. These algorithms were required to remove the guesswork involved and to perform all the necessary computations quickly and efficiently. MATLAB was the package chosen for the development of these algorithms as it is more than capable of handling the matrix algebra involved. Further, MATLAB is capable of interfacing with program codes developed in other languages such as C++ and FORTRAN [9-12].

Ramsaroop and Kanny [13] developed two MATLAB codes that were based on Hooke's law for two-dimensional composite laminates. The first code was aligned to the conventional design methods and was successfully tested with examples from the available texts. The main aspect in developing this code was to establish a basis for the other codes as well as a means of verifying future results via finite element analysis.

The second program code written by Ramsaroop and Kanny [13] was based on the proposed concept above. This code removed the guesswork experienced with the

conventional methods. The code required only the material properties and loading conditions as inputs, and it was able to design the fibre layup required to sustain the applied loads. The outputs of this code were the number of fibre layers required, fibre angle of each layer, and the thickness of each layer. The results from this code were verified using finite element models.

The code developed above could only be used to design two-dimensional structures as the governing equations were only valid for two-dimensional entities. As most structures are three-dimensional, the code was modified to accommodate the design of these structures. This paper focuses on the development of the MATLAB code that enabled the design of three-dimensional structures by analysing them as two-dimensional entities. The number of fibre layers, fibre angles of the layers, and layer thicknesses for each entity were the outputs of the new code.

The first step in the development of the above MATLAB code was to study the governing equations for composite laminates. Following this is a discussion on the previously developed program codes and thereafter the development of the new code is explained.

HOOKE'S LAW FOR TWO-DIMENSIONAL COMPOSITE LAMINATES

Before any equations can be discussed, a co-ordinate system needs to be established. Fig.1 shows the co-ordinate systems that the subsequent equations are aligned to. The first system is referred to as local co-ordinates (1-2 system). In this system the directions are based on the fibre axis. The 1-direction is associated with the longitudinal axis of the fibre while the 2-direction corresponds to the transverse of this axis.

Since fibres can be placed at various angles, one common co-ordinate system is required to serve as a reference. This system is referred to as global co-ordinates and is shown in Fig.1 as the x-y system. The local and global co-ordinate systems are related by the angle θ which corresponds to the fibre angle. All stresses and strains experienced by the laminate may be expressed in terms of the two co-ordinate systems.

The relationship between stress and strain in a fibre reinforced composite laminate is given by equation (1).

$$\begin{bmatrix} \sigma_1 \\ \sigma_2 \\ \tau_{12} \end{bmatrix} = \begin{bmatrix} Q_{11} & Q_{12} & 0 \\ Q_{12} & Q_{22} & 0 \\ 0 & 0 & Q_{66} \end{bmatrix} \begin{bmatrix} \varepsilon_1 \\ \varepsilon_2 \\ \gamma_{12} \end{bmatrix} \quad (1)$$

In the above equation σ_1 and σ_2 are the normal stresses in directions 1 and 2, respectively; τ_{12} is the shear stress in the 1-2 plane; ε_1 and ε_2 are the normal strains in directions 1 and 2, respectively; and γ_{12} is the shear strain in the 1-2 plane. The matrix [Q] is the reduced stiffness matrix, and its elements are determined using equation (2).

$$Q_{11} = \frac{E_1}{1 - \nu_{12}\nu_{21}}, \quad Q_{12} = \frac{\nu_{12}E_2}{1 - \nu_{12}\nu_{21}}, \quad Q_{22} = \frac{E_2}{1 - \nu_{12}\nu_{21}}, \quad Q_{66} = G_{12} \quad (2)$$

The formulae in equation (2) require that Young's modulus in directions 1 and 2 (E_1 and E_2), the shear modulus in the 1-2 plane (G_{12}), and Poisson's ratios in the 1-2 and 2-1 planes (ν_{12}, ν_{21}) be known. These quantities are the material constants.

Equation (1) is based on the local co-ordinate system and therefore needs to be translated to the global system. The stress-strain relationship in the global co-ordinate system is given by equation (3).

$$\begin{bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{bmatrix} = \begin{bmatrix} \bar{Q}_{11} & \bar{Q}_{12} & \bar{Q}_{16} \\ \bar{Q}_{12} & \bar{Q}_{22} & \bar{Q}_{26} \\ \bar{Q}_{16} & \bar{Q}_{26} & \bar{Q}_{66} \end{bmatrix} \begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{bmatrix} \quad (3)$$

In the above equation σ_x and σ_y are the normal stresses in directions x and y, respectively; τ_{xy} is the shear stress in the x-y plane; ε_x and ε_y are the normal strains in directions x and y, respectively; and γ_{xy} is the shear strain in the x-y plane. The matrix $[\bar{Q}]$ is the transformed reduced stiffness matrix. The local and global stresses and strains are related by equation (4).

$$\begin{bmatrix} \sigma_1 \\ \sigma_2 \\ \tau_{12} \end{bmatrix} = [T] \begin{bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{bmatrix} \quad \text{and} \quad \begin{bmatrix} \varepsilon_1 \\ \varepsilon_2 \\ \gamma_{12} \end{bmatrix} = [R][T][R]^{-1} \begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{bmatrix} \quad (4)$$

The matrix [T] in equation (4) is the transformation matrix while [R] is the Reuter matrix. These matrices are defined in equation (5).

$$T = \begin{bmatrix} \cos^2 \theta & \sin^2 \theta & 2 \sin \theta \cos \theta \\ \sin^2 \theta & \cos^2 \theta & -2 \sin \theta \cos \theta \\ -\sin \theta \cos \theta & \sin \theta \cos \theta & \cos^2 \theta - \sin^2 \theta \end{bmatrix} \quad \text{and} \quad R = \begin{bmatrix} 1 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & 2 \end{bmatrix} \quad (5)$$

The transformed reduced stiffness matrix $[\bar{Q}]$ is determined using equation (6).

$$[\bar{Q}] = [T]^{-1} [Q] [R] [T] [R]^{-1} \quad (6)$$

The above equations are only able to determine the stresses and strains in a single layer composite structure. However, as composites are multi-layered entities, the equations above need to be extended to cater for more layers. The resulting equation is shown below.

$$\begin{bmatrix} N_x \\ N_y \\ N_{xy} \\ M_x \\ M_y \\ M_{xy} \end{bmatrix} = \begin{bmatrix} A_{11} & A_{12} & A_{16} & B_{11} & B_{12} & B_{16} \\ A_{12} & A_{22} & A_{26} & B_{12} & B_{22} & B_{26} \\ A_{16} & A_{26} & A_{66} & B_{16} & B_{26} & B_{66} \\ B_{11} & B_{12} & B_{16} & D_{11} & D_{12} & D_{16} \\ B_{12} & B_{22} & B_{26} & D_{12} & D_{22} & D_{26} \\ B_{16} & B_{26} & B_{66} & D_{16} & D_{26} & D_{66} \end{bmatrix} \begin{bmatrix} \epsilon_x^0 \\ \epsilon_y^0 \\ \gamma_{xy}^0 \\ \kappa_x \\ \kappa_y \\ \kappa_{xy} \end{bmatrix} \quad (7)$$

In equation (7) N_x and N_y represent the normal forces; N_{xy} is the shear force; M_x and M_y represent the bending moments; M_{xy} is the twisting moment; ϵ^0 signifies the mid-plane strains; and κ corresponds to the mid-plane curvatures. The [A], [B], and [D] matrices are known as the extensional, coupling, and bending stiffness matrices, respectively. The elements of these matrices may be determined by the following three equations:

$$\begin{aligned}
 A_{ij} &= \sum_{k=1}^n \left[\overline{Q}_{ij} \right]_k (h_k - h_{k-1}) & i=1,2,3 \quad j=1,2,3 & (8) \\
 B_{ij} &= \frac{1}{2} \sum_{k=1}^n \left[\overline{Q}_{ij} \right]_k (h_k^2 - h_{k-1}^2) & i=1,2,3 \quad j=1,2,3 \\
 \dots (9) \quad D_{ij} &= \frac{1}{3} \sum_{k=1}^n \left[\overline{Q}_{ij} \right]_k (h_k^3 - h_{k-1}^3) & i=1,2,3 \quad j=1,2,3 \\
 & \dots (10)
 \end{aligned}$$

In equations (8), (9) and (10) n represents the number of fibre layers; and $\left[\overline{Q}_{ij} \right]_k$ corresponds to the i^{th} , j^{th} element of the $[\overline{Q}]$ matrix of the k^{th} layer. The value h_k is the distance of the top or bottom of the k^{th} layer from the mid-plane of the composite laminate. Fig.2 illustrates the positions of h_k from the mid-plane, and equation (11) may be used to determine the values of h_k .

$$h = \sum_{k=1}^n t_k \quad ; \quad h_0 = -\frac{h}{2} \quad ; \quad h_k = h_{k-1} + t_k \quad (11)$$

The mid-plane strains (ϵ^0) and mid-plane curvatures (κ) in equation (7) are related to the global co-ordinates (x-y system) via equation (12). In this equation the variable z represents the distance from the mid-plane.

$$\begin{bmatrix} \epsilon_x \\ \epsilon_y \\ \gamma_{xy} \end{bmatrix} = \begin{bmatrix} \epsilon_x^0 \\ \epsilon_y^0 \\ \gamma_{xy}^0 \end{bmatrix} + z \begin{bmatrix} \kappa_x \\ \kappa_y \\ \kappa_{xy} \end{bmatrix} \quad (12)$$

The above equations may be used to determine the stresses and strains experienced in a fibre reinforced composite laminate. However there is one more aspect of the design procedure that needs to be discussed and that is failure analysis. It is important to know whether the designed laminate succumbs to failure

or not. This may be determined by various failure theories, however, for this study, the Tsai-Wu failure criterion was chosen. This failure theory correlates with experimental data [4] and is given by equation (13).

$$H_1\sigma_1 + H_2\sigma_2 + H_6\tau_{12} + H_{11}\sigma_1^2 + H_{22}\sigma_2^2 + H_{66}\tau_{12}^2 + 2H_{12}\sigma_1\sigma_2 < 1 \quad (13)$$

In the above equation σ_1 and σ_2 are the local normal stresses in directions 1 and 2 on a particular layer; and τ_{12} is the local shear stress in the 1-2 plane on the same layer. The variables $H_1, H_2, H_6, H_{11}, H_{22}, H_{66}$ and H_{12} are the Tsai-Wu parameters and are determined via equation (14).

$$\begin{aligned} H_1 &= \frac{1}{(\sigma_1^T)_{ult}} - \frac{1}{(\sigma_1^C)_{ult}}; & H_2 &= \frac{1}{(\sigma_2^T)_{ult}} - \frac{1}{(\sigma_2^C)_{ult}}; & H_6 &= 0 \\ H_{11} &= \frac{1}{(\sigma_1^T)_{ult}(\sigma_1^C)_{ult}}; & H_{22} &= \frac{1}{(\sigma_2^T)_{ult}(\sigma_2^C)_{ult}}; & H_{66} &= \frac{1}{(\tau_{12})_{ult}^2} \\ H_{12} &= -\frac{1}{2} \sqrt{\frac{1}{(\sigma_1^T)_{ult}(\sigma_1^C)_{ult}(\sigma_2^T)_{ult}(\sigma_2^C)_{ult}}} \end{aligned} \quad (14)$$

In equation (14) the variable $(\sigma_{1,2}^T)_{ult}$ corresponds to the ultimate tensile stresses in directions 1 and 2; $(\sigma_{1,2}^C)_{ult}$ represents the ultimate compressive stresses in direction 1 and 2; and $(\tau_{12})_{ult}$ is the ultimate shear stress in the 1-2 plane. These values are the material limits.

In order to better facilitate the use of this failure theory, each stress component of equation (13) was multiplied by a variable [4] referred to as the strength ratio (SR). Its purpose is to directly determine by what ratio the local stresses must be increased or decreased to avoid failure. SR also directly relates to the applied loading. Combining SR in equation (13) results in equation (15).

$$(H_1\sigma_1 + H_2\sigma_2 + H_6\tau_{12})SR + (H_{11}\sigma_1^2 + H_{22}\sigma_2^2 + H_{66}\tau_{12}^2 + 2H_{12}\sigma_1\sigma_2)SR^2 < 1 \quad (15)$$

The criterion for SR is that it can only be positive. If SR is less than 1, then failure occurs because it means that the loading needs to decrease to avoid failure. A SR value of 1 implies that the composite laminate is perfectly suited for the applied loading conditions. If the value of SR is greater than 1, it implies that the laminate is capable of carrying more than the applied loading. For example, a SR value of 1.5 means that the loading may be increased up to 50% without failure occurring.

The above equations were used in the development of the various MATLAB codes. The first codes that are discussed are the ones developed by Ramsaroop and Kanny [13].

PREVIOUSLY DEVELOPED MATLAB PROGRAM CODES

As mentioned above, two codes were previously developed. The first code, named Conventional Method, was based on the above equations and followed the

conventional design process obtained from the literature [3-6]. Basically this code required six inputs, namely:

- the material constants (E_1 , E_2 , G_{12} , ν_{12} and ν_{21})
- the material limits ($(\sigma_1^T)_{ult}$, $(\sigma_1^C)_{ult}$, $(\sigma_2^T)_{ult}$, $(\sigma_2^C)_{ult}$, $(\tau_{12})_{ult}$)
- the number of layers (n)
- the fibre angle of each layer (θ)
- the thickness of each layer (t)
- applied loading conditions

The outputs of this code were the global and local stresses and strains experienced in a fibre reinforced composite laminate as well as the failure analysis via the Tsai-Wu theory. The code was tested against manually calculated examples from Kaw [4] and it was found that the MATLAB code results were exactly comparable to the values in the text. A finite element model was set up to provide further verification as well as providing a means for validation of future results from other codes.

Although the Conventional Method program code performed all the relevant calculations easily and efficiently, and significantly shortened the time of the design process, there was still much guesswork involved in the design of a composite laminate [13]. If failure occurred the laminate would have to be redesigned and there was uncertainty in which input values to change. If there was no failure then the structure may end up being overdesigned which lead to an increase in manufacturing costs. Therefore a second MATLAB code was developed to overcome these issues.

The second code was named Fibre-Angle-Output and its purpose was to remove the uncertainty experienced with the Conventional Method code. This code was still based on the above equations but instead of fixing the fibre angles and number of layers, it varied these parameters in order to achieve a composite structure that is perfectly suited to sustain the applied loading conditions. The number of inputs was reduced from six, for the conventional design method, to three. These three inputs were the:

- material constants
- material limits
- loading conditions

The Fibre-Angle-Output code periodically checked the integrity of the structure being designed via the Tsai-Wu failure theory and automatically adjusted the input parameters if failure occurred. The outputs of this code were the number of layers required, and the fibre angle of each layer. Finite element analyses with the model set up for the Conventional Method code verified the results of the Fibre-Angle-Output code. Although the code was able to remove uncertainty previously experienced and design a composite laminate optimally, it did not perform any thickness optimisation procedures. Thickness optimisation is a crucial step in designing a composite laminate in order to reduce material usage and thereby decrease manufacturing costs. Hence this factor is the first element explored in this paper.

MATLAB CODE FOR THICKNESS OPTIMISATION

The main function of this code was to optimise both the fibre angles and the thicknesses of the layers in a laminate and consequently the weight of the laminate. As a result the code was named Angle-Thick-Opt and it is an extension of the Fibre-Angle-Output code. The operation of the Angle-Thick-Opt program code can best be described by the flowchart in Fig.3.

First the material constants and the material limits have to be input (Block 1). With this data, the [Q] matrix (Block 2) and the Tsai-Wu parameters (Block 3) can be calculated immediately as these do not vary with fibre angle but rather with the material properties. The next step is to input the applied loading conditions (Block 4) and evaluate the resultant forces and moments (Block 5).

The program code now moves on to Block 6 where it has to determine the relevant matrices and the various stresses and strains. Block 6 is basically the Conventional Method program code. Hence the program code needs to know the number of layers, fibre angle of each layer, and the thickness of each layer in order to continue with the calculations. These were not part of the input parameters and therefore the code makes an initial assumption. This assumption (which is only done once) is that there is one layer with a fibre angle of 0° and a fixed thickness. Using this data, the h_k and z values as well as the [T], $[\bar{Q}]$, [A], [B] and [D] matrices are computed. The mid-plane strains (ϵ^0) and curvatures (κ), and, the global and local stresses and strains are then evaluated.

The Tsai-Wu failure theory is applied and a value for SR is obtained (Block 7). The code then analyses the SR values and verifies whether it is in a specified range (Block 8). The lower limit of this range is 1, as any value below this would mean failure, and the upper limit is 1.2 and this is to avoid overdesigning.

If the SR values for each layer are between 1 and 1.2, then the design of the composite laminate is complete and the program code moves to the outputs (Block 13). On the other hand, if the SR value for any of the layers is out of this range, the code varies that particular fibre angle to obtain a SR value between 1 and 1.2 (Blocks 9 and 10). However, due to the angle change all the matrix calculations have to be redone. New SR values are obtained for each layer and these are compared to the old values (Block 11). If the new SR value for a particular layer is higher than the old SR value, then the angle of that layer is changed to the new one. For example, after the initial calculation a layer with fibre angle of 30° has a SR value of 0.8. This is not acceptable and the program changes the angle to obtain a better SR value. If at 35° the SR value is 0.9, the program will then change the angle of the 30° layer to 35°. If at 35° the SR value was 0.7, then the program will retain the 30° angle.

The program continues to vary angles and compare old and new SR values, that is, it repeats Blocks 8 to 11, until the SR value can no longer be improved ("NO" option of Block 9). Thereafter the thickness optimisation function (Block 12) is then executed. The operation of this function is discussed later (Fig.4). The angle optimisation is repeated if necessary and the code determines whether a new layer is required (Block 13).

If a new layer is added, the program code returns to Block 6 where the various matrix computations are redone. The code follows the procedure as before until the SR values for each layer is between 1 and 1.2. Once this has been achieved, the code returns the outputs (Block 13). The outputs include the number of layers, fibre angle of each layer, and thickness of each layer.

The operation of the thickness optimisation function is represented by the flowchart in Fig.4 where the dashed rectangle represents Block 12 in Fig.3. The inputs of this block are the outputs of Block 9 in Fig.3 which include the number of layers, the fibre angle of each layer, and the thickness of each layer. The initial thickness was set as 1 mm.

The function first determines the minimum SR values of each layer (Block 12.1) and this is stored for later use. In the next step (Block 12.2), a pair of adjacent layers is chosen and the difference of their angles is determined. Block 12.3 verifies whether this difference is within a specified range. The criteria that was chosen was that the absolute value of the difference must be less than or equal to 10° . This value was determined after several runs of the code where it was found that, with a layer thickness of 1 mm, many of the fibre layers were within 10° of each other. This value may be changed if desired.

If Block 12.3 concludes that the difference is within the specified range, the code then combines (adds) the thicknesses of the layers under examination (Block 12.4). It uses the stored SR values to determine which layer has the better value (Block 12.5) and keeps the angle of the better layer. The other angle along with the layer it represents is discarded (Block 12.6), and as a result the number of layers decreases. The code then proceeds to Block 12.7.

If Block 12.3 finds that the difference is not in the specified range, the code proceeds directly to Block 12.7. Here the code checks whether all the layer pairs have been examined. A “false” output results in the code returning to Block 12.1 and repeating all the steps with the next layer pair until all the layer pairs have been analysed. Once Block 12.7 gives a “true” output, the code proceeds to Block 12.8 where the angles are optimised again with the new layer thicknesses.

Once the thickness optimisation function is completed, the program code returns to the original procedure (Block 13 in Fig.3) where it determines whether a new layer is required. For every new fibre layer that is added on, the thickness optimisation function is executed at the appropriate time.

The results of the Angle-Thick-Opt program code needed to be verified and was therefore tested against the example used in the verification of the Fibre-Angle-Output code. In this example a graphite/epoxy laminate had to be designed with the material constants and material limits given in Table 1. The applied loading conditions are also shown in this table and the thickness of each layer was 5mm, giving a total laminate thickness of 15mm. The resulting fibre angles and SR values for each layer from the Fibre-Angle-Output code is shown in Fig.5.

For the verification process of the Angle-Thick-Opt code graphite/epoxy was used with the material properties and loading conditions shown in Table 1. Here the initial

thickness of a layer was 1mm. These parameters were input into the program code and the results are shown in Fig.6.

It can be seen from Fig.6 that four layers are required with fibre angles of -71° , 31° , -70° and 32° . The thicknesses of the layers have also changed from being 5mm each to various values. These results were compared to those from the code where there was no thickness optimisation. Table 2 shows the comparison between:

- A. The Fibre-Angle-Output code (with fixed thickness)
- B. The Angle-Thick-Opt code (with variable thickness)

It can be seen from Table 2 that although Case B requires more layers than Case A, the total laminate thickness is just 6 mm. This is less than half the thickness required for Case A and this implies that less material will be used in Case B when compared to Case A. Less material usage means less material cost which leads to a saving in manufacturing costs.

In addition to the thickness optimisation, there was also a reduction in the laminate mass. Table 3 shows a comparison of the laminate masses for Cases A and B which were found via each laminate's density and volume. The density of a graphite/epoxy composite is 1620 kg/m^3 . The volume of each laminate was calculated using the overall thicknesses shown in Table 2 and assuming each laminate to be 300×300 mm. The laminate mass was determined using the density and the calculated volumes. The table shows that the mass for the laminate in Case B has been reduced by 1.3 kg when compared to Case A. The Angle-Thick-Opt program code reduced the laminate mass in Case B by 60% compared to Case A.

The results from Case B were analysed in the MSC PATRAN/NASTRAN finite element package. The model used for the analysis was similar to the one used for the Fibre-Angle-Output code. The finite element model included the material properties and loading conditions shown in Table 1. The fringe plots of the stress components from the finite element analysis are shown in Fig.7.

Figure 8 shows the local stresses experienced by the laminate in Case B above when it was designed in MATLAB. The stress values shown in Fig.7 correspond to the local stresses experienced by the laminate at the middle of each layer in Fig.8. This implies that the Angle-Thick-Opt program code is able to correctly optimise the fibre angles and layer thicknesses in a fibre reinforced composite laminate.

The Angle-Thick-Opt program code is capable of correctly designing a fibre reinforced composite laminate. The code determines the optimum number of fibre layers required and the fibre angle of each layer for a given set of loading conditions. Further it also optimises the thickness of each layer and thus the laminate as a whole. However the Angle-Thick-Opt code is based on equations that are valid for two dimensions. Therefore a new code is needed that is capable of designing and analysing three-dimensional structures. This code was developed and named 3D-Structure.

MATLAB CODE FOR THREE-DIMENSIONAL STRUCTURES

The 3D-Structure code takes advantage of a finite element package's ability to divide any structure into smaller sections called elements via meshing. In the case of fibre reinforced composite structures, meshing will essentially separate a three-dimensional structure into two-dimensional entities. Each entity or element may be analysed as a separate laminate provided the loading conditions for that element are known. If each element is treated as a separate then the Angle-Thick-Opt program code may be used to design the fibre layup for each element. The 3D-Structure code accommodates this elemental design method and uses the Angle-Thick-Opt code to determine the laminate properties for each element in a structure.

The flowchart shown in Fig.9 illustrates the procedure involved in the design of a three-dimensional structure. The first step (Block 1) in the above procedure requires the drawing of the structure that is being designed. This may be accomplished via a Computer Aided Design (CAD) package and then imported into the finite element package, or directly through the finite element package itself. Once this is completed, the surface(s) of the structure may then be divided into elements via meshing (Block 2). The recommended mesh shape is quadrilateral elements and the mesh size should be varied. In areas with little or no change in shape the mesh size should be large, while in regions with more details, such as bends, corners, holes, etc., the mesh size should be made finer.

Next each mesh element must be defined as a composite laminate. This is achieved by first creating a two-dimensional orthotropic material with the required material properties (Block 3). Following this a layered composite needs to be created using the material formed in the previous step and the built-in laminate modeller (Block 4). As there are no known values for the fibre angles and layer thicknesses at this time, any values will suffice. The MATLAB program code will rebuild the laminate one layer at a time without using these values. However, it is suggested that the laminate contain at least two layers as this will enable the model setup to be straightforward and it will allow for easy modification later on if validation of the MATLAB results are required.

The created composite laminate properties may now be assigned to each and every mesh element (Block 5), and thereafter the boundary and loading conditions are applied to the structure (Block 6). The structure is now ready for analysis (Block 7). Any number of outputs may be selected but the critical one is the elemental loading. The finite element results must be saved in binary format as an OP2 file. A consequence of the analysis is the distribution of the applied loading and boundary conditions through the structure.

Once the finite element analysis is completed, the saved results may be imported into MATLAB using the 3D-Structure program code (Block 8). This is achieved via a software package called IMAT which is available from ATA Engineering, based in San Diego, California, USA. This software package imports parameters from finite element packages such as MSC NASTRAN/PATRAN into MATLAB. The imported parameters include the number of elements, nodes contained in each element, node positions, node forces, geometry of the structure, applied loading and boundary conditions, and other entities.

The nodes contained in each element as well as the loading conditions of each node are now known. Therefore the 3D-Structure code is able to associate each element with its loading conditions (Block 9). Now each element can be treated as a separate laminate. The 3D-Structure code uses the Angle-Thick-Opt code to design the fibre layup of each element (Block 10-11). Once all elements have been analysed, the outputs are given (Block 12). These include the number of layers, fibre angle of each layer, and thickness of each layer for every element in the finite element mesh. Other outputs are the strength ratios, global and local stresses and strains, and loading vector for each element.

The 3D-Structure code was tested with the graphite/epoxy laminate used throughout this study. A finite element model representing the laminate was set up in MSC NASTRAN/PATRAN. This laminate was divided into four elements via quadrilateral meshing. A two-dimensional orthotropic material was created using the material properties shown in Table 1. Following this, a composite consisting of two layers, each with a fibre angle of 0° and thickness of 1mm was created. These properties were assigned to the four elements and the loading conditions shown in Table 1 were applied to the model. Fig.10 shows this finite element model with its four elements and the applied boundary and loading conditions. The analysis was executed and the results were saved as an OP2 file.

The fibre layup of the laminate was then designed by the 3D-Structure program code. The code first imported the finite element results into the MATLAB environment via the IMAT software package. Next each element was associated with its loading conditions and the fibre layup for each of the four elements was resolved. The number of layers, fibre angles, layer thicknesses and the loading vector determined for each element is shown in Fig.11.

The figure shows that each element has the same fibre layup and the thicknesses of each layer of each element are exactly the same. In fact these results are precisely equal to those obtained when testing the full Angle-Thick-Opt code above. This was anticipated as there was no change in shape of the structure, that is, there were no holes, bends, corners, etc. The contour of the shape remained flat and therefore the applied loading was evenly distributed throughout the structure. This implies that the 3D-Structure code is able to correctly design the fibre layup in a three-dimensional structure.

CONCLUSION

A MATLAB code based on Hooke's law for two-dimensional fibre reinforced composite laminates was developed to remove the uncertainty experienced with conventional design methods. This code designed a composite laminate based on the applied loading conditions, and outputted the number of layers required and the fibre angle of each layer. In addition to optimising the fibre angles and number of layers, the code also optimised the thickness of each layer. The code was tested and verified using examples from literature and finite element modelling.

The above code was only valid for two-dimensional entities. Hence it was enhanced to accommodate the design of three-dimensional structures. This new code was validated with previous examples and MATLAB.

REFERENCES

- [1] Park, C.H., Lee, W.I., Han, W.S., and Vautrin, A., "Weight minimization of composite laminated plates with multiple constraints", *Composite Science and Technology*, Vol. 63, No. 7, pp. 1015 – 1026, (2003).
- [2] Park, C.H., Saouab, A., Bréard, J., Han, W.S., Vautrin, A., and Lee, W.I., "An integrated optimisation for the weight, the structural performance and the cost of composite structures", *Composites Science and Technology*, Vol. 69, pp. 1101 – 1107, (2009).
- [3] Christensen, R. M., "Mechanics of Composite Materials," Dover Publications, New York, (2005).
- [4] Kaw, A.K., "Mechanics of composite materials", CRC Press LLC, Boca Raton, Florida, U.S.A., (1997).
- [5] Herakovich, C.T., "Mechanics of fibrous composites", John Wiley & Sons Inc., New York, U.S.A., (1998).
- [6] Hyer, M.W., "Stress analysis of fiber-reinforced composite materials", WCB/McGraw-Hill, U.S.A., (1998).
- [7] Mazumdar, S.K., "Composite Manufacturing: Materials, Product, and Process Engineering", CRC Press LLC, (2002).
- [8] Akbulut, M., and Sonmez, F.O., "Optimum design of composite laminates for minimum thickness", *Computers and Structures*, Vol. 86, No. 21–22, pp. 1974 – 1982, (2008).
- [9] Wikipedia, "MATLAB", Wikipedia, the free encyclopaedia, <http://en.wikipedia.org/wiki/MATLAB>
- [10] Dukkipati, R.V., "MATLAB for Mechanical Engineers", New Age International (P) Limited, New Delhi, (2008).
- [11] Gilat, A., "MATLAB: An Introduction with Applications", John Wiley & Sons Inc., USA, (2008).
- [12] Hahn, B.D., and Valentine, D.T., "Essential MATLAB for Engineers and Scientists", Butterworth-Heinemann, Italy, (2007).
- [13] Ramsaroop, A., and Kanny, K., "Using MATLAB to Design and Analyse Composite Laminates," *Engineering*, Vol. 2, 904–916, (2010).

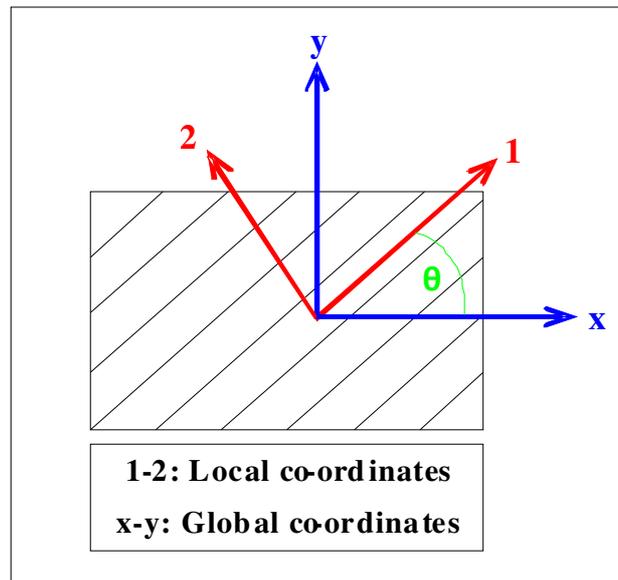


Fig.1. Co-ordinate systems associated with Hooke's law.

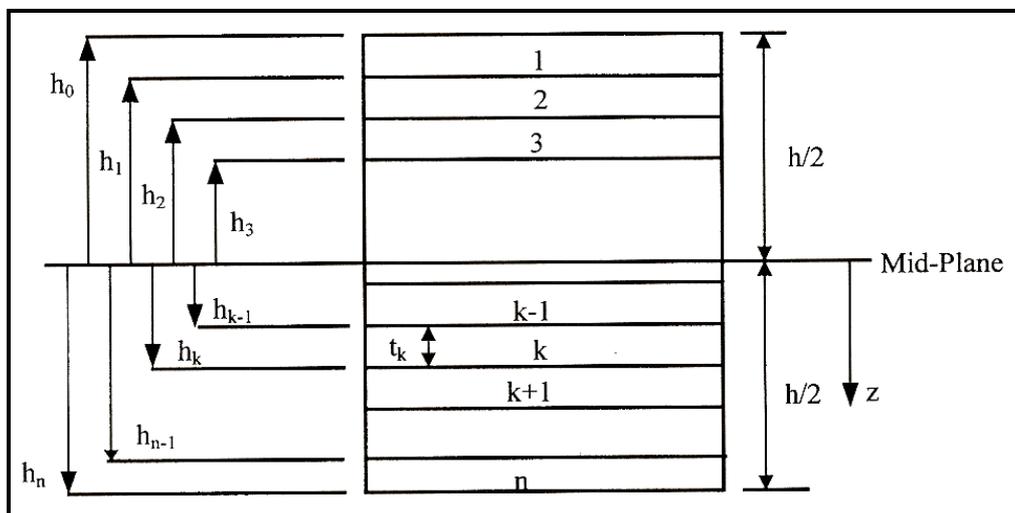


Fig.2. Locations of layers in a composite structure [4].

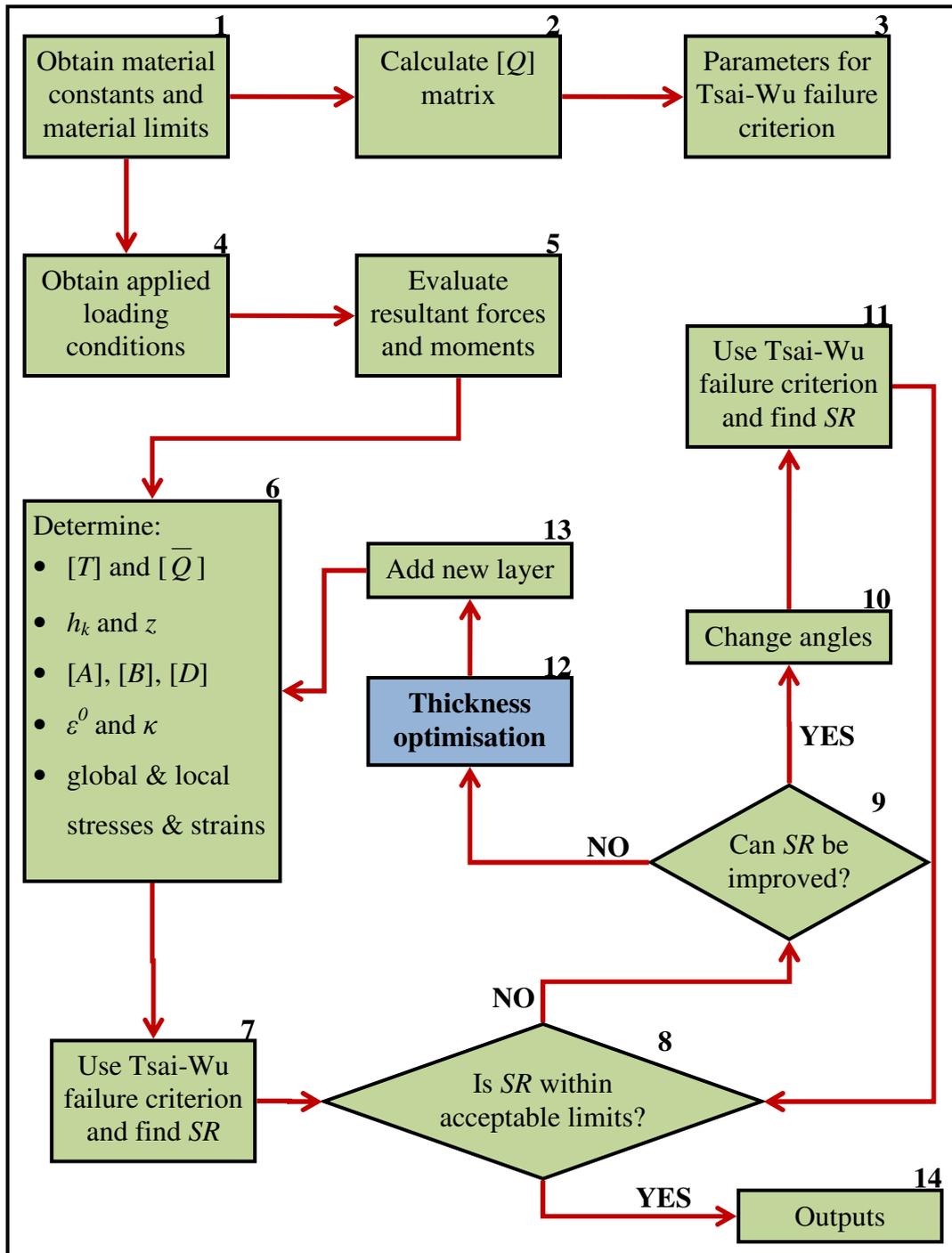


Fig. 3. Flowchart describing the operation of the Angle-Thick-Opt program code.

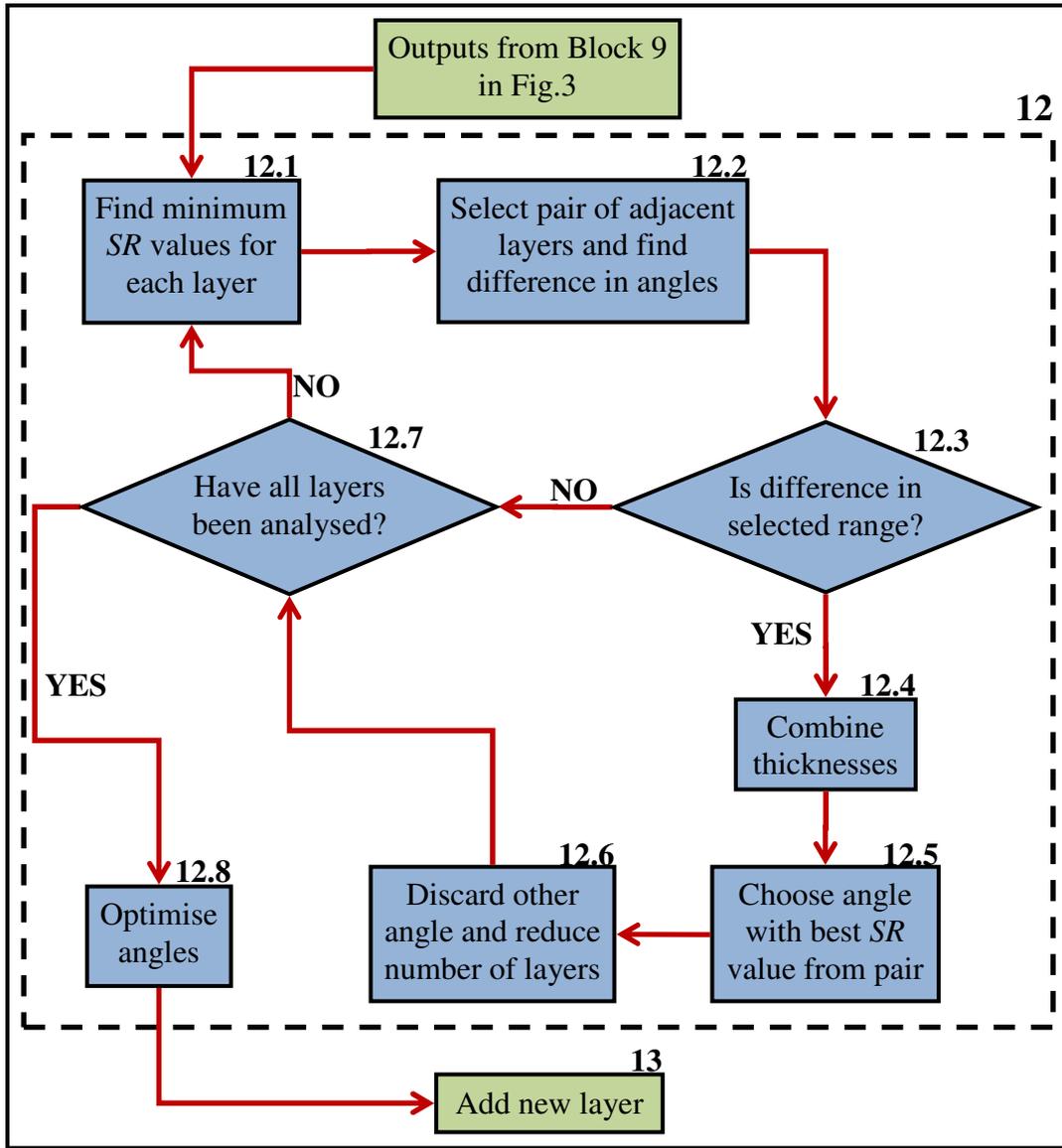


Fig. 4. Flowchart describing operation of the thickness optimisation function.

```

Command Window
Ply_Angle =

    49    -79    49

SR =

    1.0004    1.0004    1.0004
    1.0331    1.0331    1.0331
    1.0004    1.0004    1.0004

Complete
>>
  
```

Fig.5. Results of Fibre-Angle-Output code with Table 1 as inputs.

```

Command Window
New to MATLAB? Watch this Video, see Demos, or read Getting Started.

Fibre Angles
    -71    31   -70    32

Layer Thicknesses
     1     2     2     1

Complete
fx >>
    
```

Fig. 6. Results of the Angle-Thick-Opt program code.

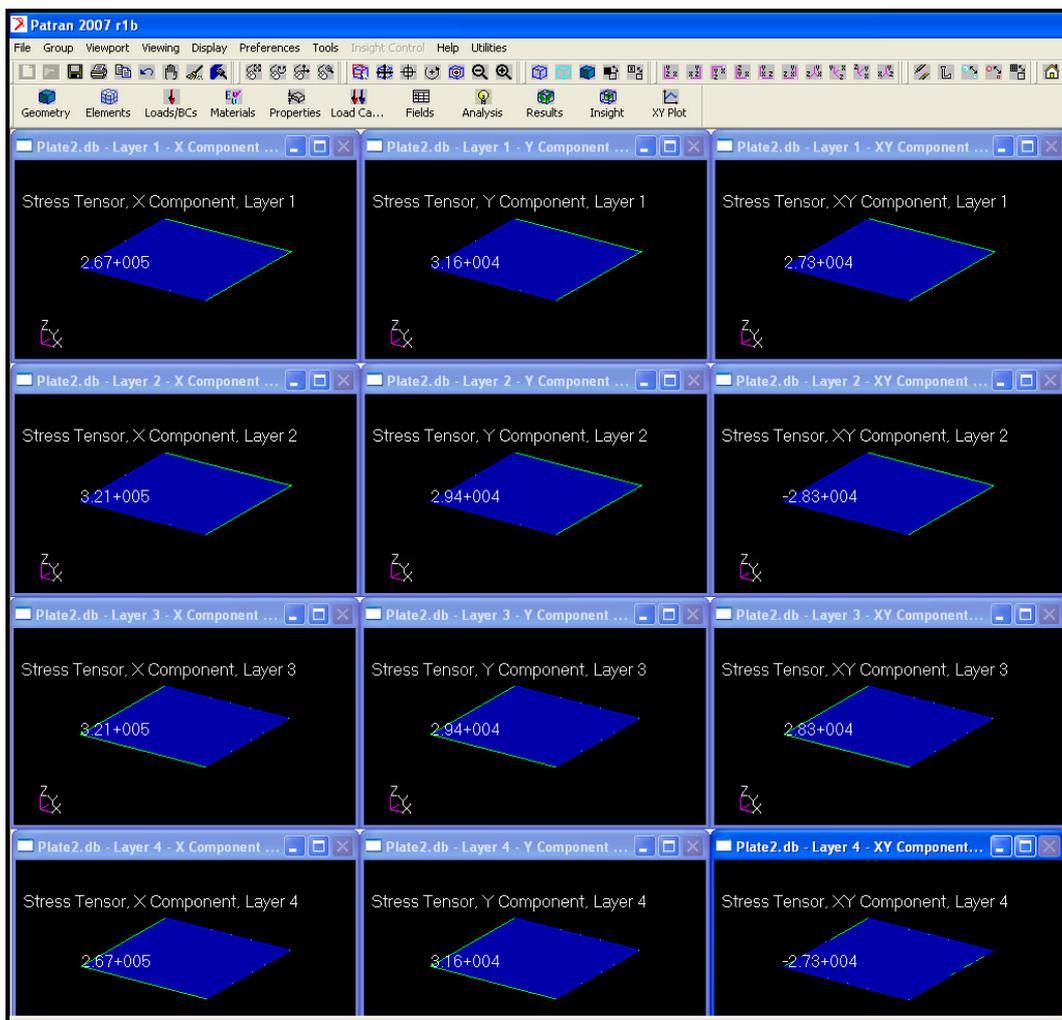


Fig.7. Fringe plots showing stress components for the laminate in Case B.

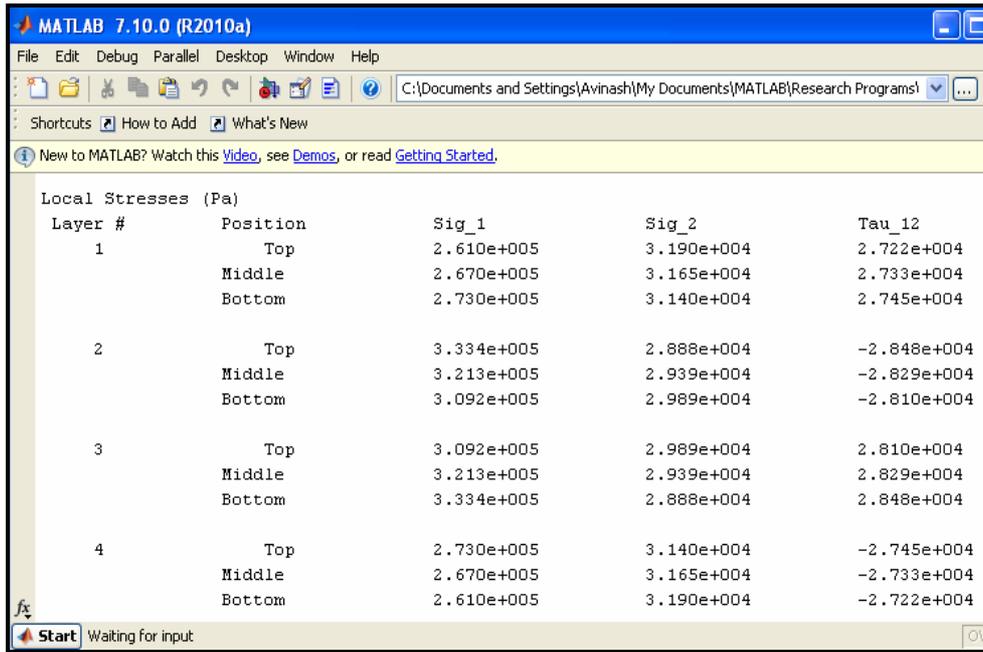


Fig. 8. Local stresses of laminate in Case B when analysed in MATLAB.

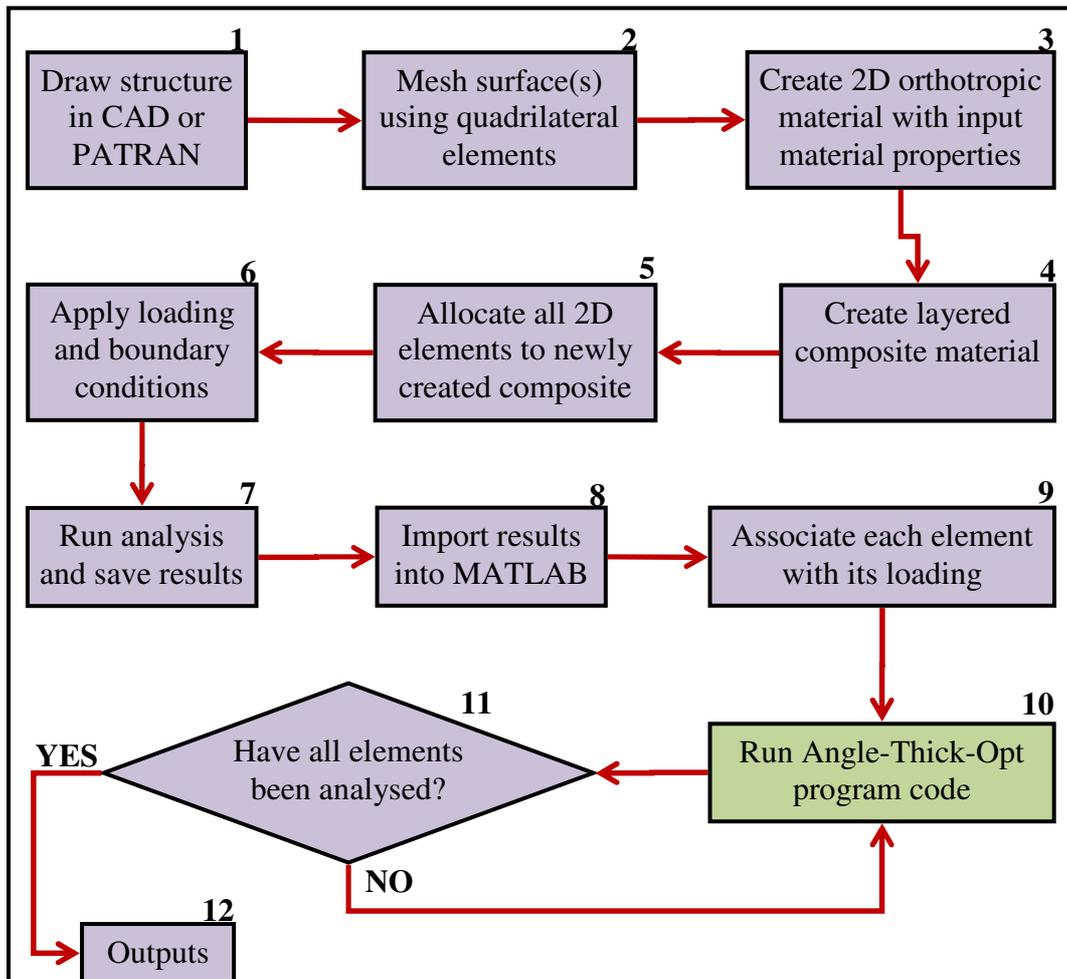


Fig.9. Flowchart illustrating the procedure for the 3D-Structure program code.

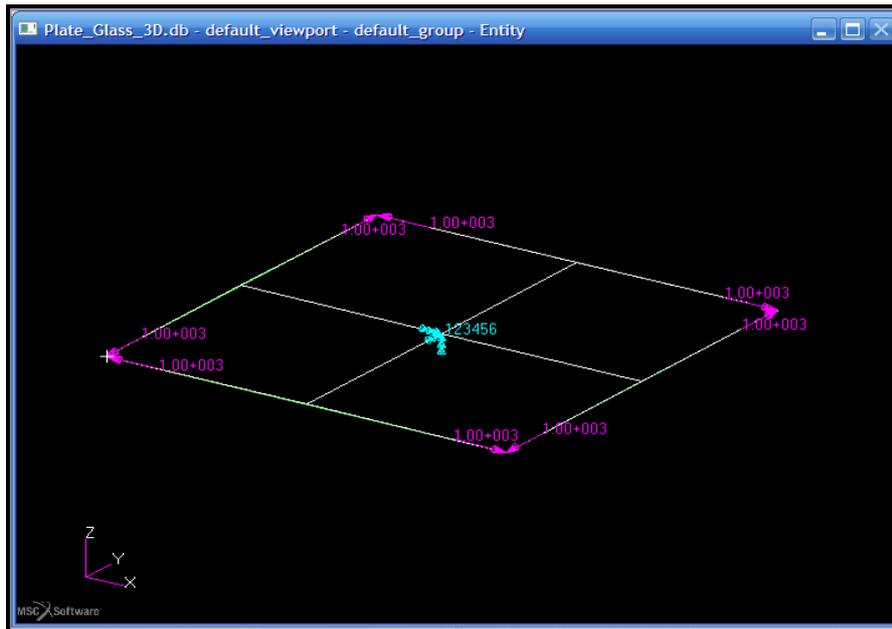


Fig.10. Finite element model showing elements, and applied boundary and loading conditions.

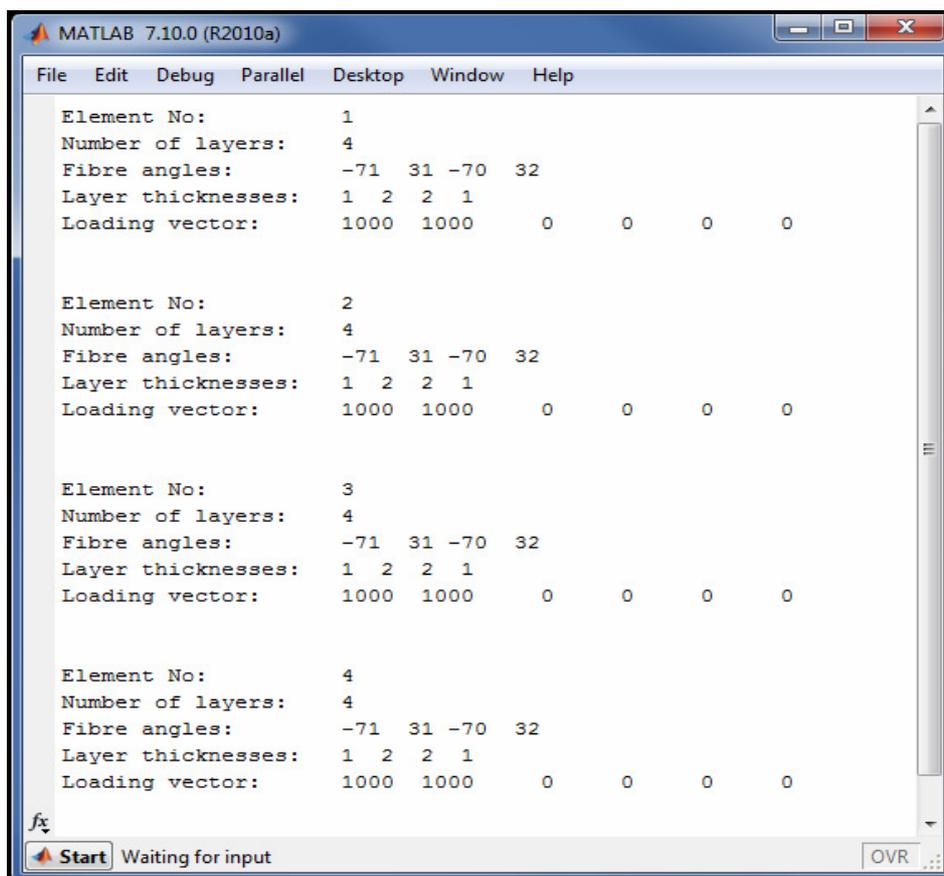


Fig.11. MATLAB results for each element of the graphite/epoxy laminate.

Table 1. Material constants, material limits and loading conditions for graphite/epoxy composite example.

Material Constants				Material Limits					Loading	
E_1	E_2	G_{12}	ν_{12}	$(\sigma_1^T)_{ult}$	$(\sigma_1^C)_{ult}$	$(\sigma_2^T)_{ult}$	$(\sigma_2^C)_{ult}$	$(\tau_{12})_{ult}$	Forces	Moments
(GPa)	(GPa)	(GPa)		(MPa)	(MPa)	(MPa)	(MPa)	(MPa)		
181	10.3	7.17	0.28	1500	1500	40	246	68	1000	0
									1000	0
									0	0

Table 2. Comparison of Angle-Thick-Opt code with Fibre-Angle-Output code.

	Case A: Fibre-Angle-Output code	Case B: Angle-Thick-Opt code
Number of layers	3	4
Fibre angles (°)	49 -79 49	-71 31 -70 32
Layer thickness (mm)	5 5 5	1 2 2 1
Total thickness (mm)	15	6

Table 3. Comparison of laminates masses for Cases A and B.

	Dimension (mm)	Volume (m ³)	Mass (kg)
Case A	300 X 300 X 15	0.00135	2.187
Case B	300 X 300 X 6	0.00054	0.875