Quickstart Guide

Composite Compressive Strength Modeller

A Windows™ based composite design tool for engineers

Version 2.0

2013

**M. P. F. Sutcliffe, X. J. Xin,
N. A. Fleck and P. T. Curtis**

**Engineering Department
Cambridge University
Cambridge, U.K.,
CB2 1PZ**

Composite Compressive Strength Modeller

Quickstart Guide

1. Introduction

Welcome to Composite Compressive Strength Modeller (CCSM) – a design tool for deformation analysis and failure prediction of composite materials. This quickstart guide gives an introductory guide to CCSM. It is intended to be followed at a computer running CCSM and details commands and data to type in. A more complete manual is also available. CCSM Version 2.0 is a simplified version of CCSM 1.4 to allow use of 64 bit Windows operating system.

Features of CCSM covered in this manual are:

1. Classical laminate theory for the prediction of laminate elastic properties;
2. Stress and strain analysis when in-plane forces and bending moments are applied;
3. Unnotched failure prediction by conventional failure criteria.

The program is a tool to predict laminate failure, once the loads on a section of the laminate are known. For simple geometries it may be clear what the loading is, while for more complicated geometries the program may be used as part of a larger calculation to check for failure at critical points in the structure. Nomenclature and sign conventions follow the CUED course 4C2 Designing with Composites, and are described in detail on the 4C2 Data Sheet.

Acknowledgements
The authors are grateful for helpful advice from Dr C Soutis and Mr I Turner, Ms V. Hawyes and P Schwarzel, additional programming from Mr A Curtis and for financial support from the Procurement Executive of the Ministry of Defence, contract 2029/267 and from the US Office of Naval Research grant 0014-91-J-1916.

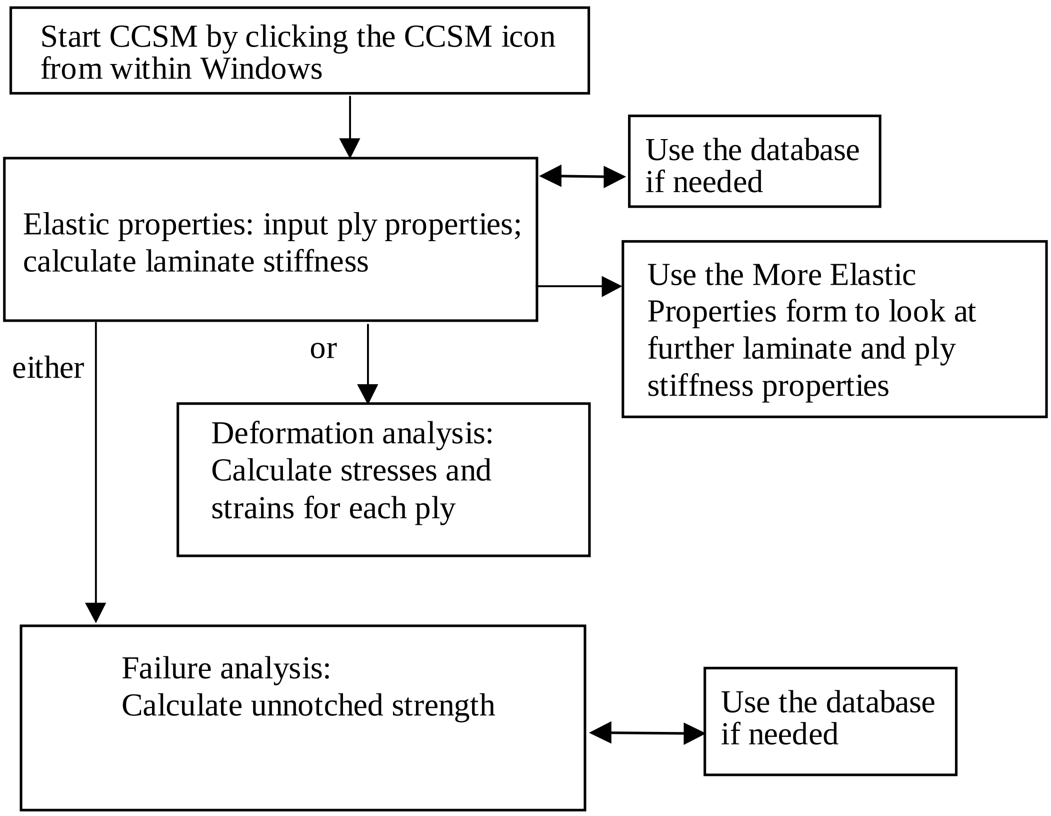
2 How to use CCSM

CCSM is written in Visual Basic (VB), a package designed to produce especially user-friendly graphical interfaces. The user should work through the various forms in CCSM by following the logic of a problem. For example, in order to calculate the stiffness of the laminate, sufficient information about each lamina should be provided first. Similarly, analysis of stress and strain would be meaningless without previously calculating the laminate properties and specifying the applied loads.

CCSM contains a number of forms for each stage of the analysis. Details at each stage are filled in using text boxes containing data, option buttons, and command buttons.

The following flow chart shows what to do in CCSM. In each form in CCSM corresponding to each box in the flow chart, there is an information box providing information about what to do next. Boxes in which to input data have a white background.

Further help can be obtained from the ? buttons and from the manual. Details of the program authors are included using the About menu.



A flow diagram showing the structure of CCSM

3 A quick start example

This section goes through a simple analysis to illustrate the essential features of CCSM.

The problem. Determine the stiffness and compliance matrices for a $[0^{o} / \pm 45^{o} / 0^{o}]_{s}$ symmetric laminate consisting of 0.1 mm thick unidirectional AS/3501 carbon fibre – epoxy laminae. Also find the stresses and strains for each lamina when the laminate is subjected to a single uniaxial force per unit length N_{x}=200 MN/mm. Use the Tsai-Wu failure criterion to decide the load level corresponding to first ply failure.

The following lamina stiffness and strength data are given: longitudinal modulus E_{11}=138 GPa, transverse modulus E_{22}=9 GPa, shear modulus G_{12}=6.9 GPa, Poisson's ratio ν_{12}=0.3, longitudinal tensile strength, denoted as SL(+)=1448 MPa, longitudinal compressive strength, SL(-)=1172 MPa, Transverse tensile strength, ST(+)=48.3 GPa, Transverse compressive strength, ST(-)=248 GPa, in-plane shear strength, SLT=62.1 GPa (taken from R. F. Gibson's Principles of Composite Material Mechanics, Table 2.2, P.48, 1994).

A step by step illustration is given below:

Step 1: Starting CCSM

After starting Windows, launch the Compressive Composite Strength Modeller program. The *Geometry and elastic analysis:* form is then loaded automatically and the cursor will be blinking in the C*omposite name* text box.

Step 2: Entering elastic properties data.

In this step laminate data and elastic properties are entered. All white text boxes are input boxes, and the light yellow boxes are output or information boxes. Now type in the following data, appropriate to the problem in hand:

Which input box	**What you type or do**	**Note**
Composite:	AS/3501	Optional
Comments:	Quickstart example	Optional
Total number of plies	8	
Ply No.	1	Type the current ply number into this box
Angle	0	Type the angle into this box (in degrees)
Thickness	0.1	**In mm**
E11	138	Lamina's Young's modulus in first (fibre) direction E_{11} (in local lamina co-ordinates), **in GPa**
E22	9	Lamina's Young's modulus in second (transverse to fibre) direction E_{22}
Nu12	0.3	Poisson's ratio ν_{12}
G12	6.9	In-plane shear modulus G_{12}
	(click Save Ply Data)	At this point, all necessary data for ply No. 1 have been input. Click the Save Ply Data button to save the input. The ply arrangement grid is filled for each ply where data has been saved. For ply Nos. 2 to No 4, the input procedures will be similar.
Ply No.	2	
Angle	45	
	(click Save Ply Data)	Notice that after inputting data for ply No. 1, the Lamina properties and thickness text boxes still hold the data for Ply No.1. Those data, because expressed in local lamina co-ordinate, are valid for other plies as long as they are for the same material. Different properties for each lamina are allowed in CCSM – you just type in the corresponding data for each lamina.
Ply No.	3	
Angle	-45	
	(click Save Ply Data)	
Ply No.	4	
Angle	0	
	(click Save Ply Data)	

Ply data for plies 5-8 have been automatically filled in, because the **laminate type option** is by default **symmetric**. The ply geometry and property data are symmetrical about the centre line.

Step 3. Calculating the stiffness of the laminate

At this point, data for all laminae have been entered. Now click the **Calculate** button to calculate the laminate stiffness. The first 5 components represent E_x, E_y, G_{xy}, ν_{xy} and ν_{yx}, for the laminate, using standard notation for an orthotropic laminate. The sixth component, E', is the appropriate elastic modulus for an orthotropic material such that $G = K^2 / E'$, where G is the elastic energy release rate and K is the mode I stress intensity factor. Lamina stiffness matrices can be viewed by clicking on the **More Elastic Properties** button.

Tips

Saving Ply Data

Remember to click on the **Save Ply Data** button for each ply and to click **Calculate** to update the laminate properties after finalising ply details.

Using the Database

Instead of typing the above data in each input text box, you can make use of the Database. Click on the **Database** button, then on the required material's name in the list of materials, 'AS/3501' in the present example. Finally click the **Take record as input** button.

Fast input

Once data for the first ply has been filled in, the lamina elastic properties and thickness become the default for subsequent plies. However the ply angle needs to be entered for each ply, and all the data needs to be saved. The **Ply Input** buttons give a fast method of inputting this data. The columns refer to the ply position, relative to the current ply. The row denotes the ply angle. Thus clicking on the **Next** column and the **90** row increases the number in the **Ply Number** data box by one, and puts 90 in the **Ply Angle** data box. Now the data can be saved by clicking the **Save Ply Data** button. The ply data can hence speedily be entered by repeatedly clicking on the appropriate row and column to get the right ply numbers and angles in the input boxes, saving ply data when required. The Enter Key defaults to the same action as the **Save Ply Data** button.

Ply Editing

Navigate through the plies in the ply arrangement box, using either the mouse or cursor keys. The saved material properties for the highlighted ply are displayed. Several plies can be selected for cutting or deleting by dragging with the mouse. Plies are pasted above the selected ply. If strength properties have already been defined for the laminate, these properties are associated with each ply and are cut and paste with the plies. The total number of plies is automatically updated when plies are cut or pasted. The number of plies can also be changed by entering a new number in the Total Number of Plies box.

Change All

The material properties and thickness of all plies can be changed at once, by entering the new data in the input data box, and then clicking on the appropriate button.

Step 4 Deformation analysis

In our example problem, we wish to find the stresses and strains in each ply. This is done, after **calculating** the laminate stiffness, using the Deformation Analysis, clicking on the appropriate button under the GoTo tool.

On the *Deformation analysis:* form, first input the applied load, in this case 0.2 in the *Force resultant* N_x text box (converting from MN/mm to MN/m). The other text boxes can be left empty as these components are zero.

The input text boxes should be:

Force resultants on laminate	N x	N y	N xy	M x	M y	M xy
	0.2					

This is the only input needed for the deformation analysis in this problem. Now click the **Calculate** button. The mid-plane strains and curvatures will be calculated, and shown as:

Mid-plane strains and curvatures	E x	E y	Gamma xy	K x	K y	K xy
	3075	1934	~0.	~0.	~0.	~0.

where ~0 is a very small value (of the order of rounding errors). The mid-plane shear strain γ_{xy} (Gamma xy) and all curvatures are zero, because the laminate is symmetric and there are no bending components.

Though the stresses for all plies have been calculated, only stresses in one ply will be shown in the *Stress State* grid (the first ply by default). Stresses at the top, middle and bottom of each selected ply are shown; this takes into account the possibility of linear variation in the stress in the presence of bending. At the present example, the stresses in each ply is constant. The bottom row shows the average stress through the full thickness of the laminate. The results are shown as:

Through thickness t of selected ply	Stress state for ply No. 1		
	Sigma_x	Sigma_y	Tau_xy
top	421.697	-9.161	~0.
middle	421.697	-9.161	~0.
bottom	421.697	-9.161	~0.
Laminate stresses	250	~0.	~0.

These stresses are in global coordinates (i.e. running along the x and y directions of the laminate). To see the ply stresses in local coordinates (i.e. running along and transverse to the fibre direction in each ply), click on the appropriate **Output Option**.

To display the ply stresses of other plies, navigate through the ply arrangement grid either clicking with a mouse or using cursor keys. The current ply is highlighted in this grid.

Notice that in the **Output Option** box, the "stress" option is set as the default. Clicking the "strain" option will display strain data giving:

Through thickness t of selected ply	Strain state for ply No. 1		
	Epsilon_x	Epsilon_y	Gamma_xy
top	3075	-1934	~0.
middle	3075	-1934	~0.
bottom	3075	-1934	~0.
Laminate strains			

Note that strains are output in microstrain – one microstrain = 10^{-6}. It can be seen that there are stress discontinuities at the ply interfaces, while the strains are continuous across the interface:- this reflects the basic assumption of the classical laminate theory.

Step 5: Failure analysis

To predict laminate failure, click on the **Failure Analysis** button, once the laminate properties have been entered and the laminate stiffness calculated.

There are five failure analysis criteria available in CCSM: the maximum stress, the maximum strain, the Tsai-Hill, the Tsai-Wu and the Budiansky-Fleck-Soutis (BFS) compressive failure criteria. All these criteria are **lamina** failure criteria. Select the "Tsai-Wu" option for the present problem. Now type in the following strength data for the material AS/3501:

Which input box:	What you type:	Note
Longitudinal tensile strength SL(+)	1448	In MPa.
Longitudinal compressive strength SL(-)	1172	
Transverse tensile strength ST(+)	48.3	
Transverse compressive strength ST(-)	248	
In-plane shear strength SLT	62.1	

Click on the **Save Data - All Plies** button, to store this data.

Instead of typing the above data in each input text box, you can make use of the Database menu. Click on the **Database** button. Click on the required material's name in the list of data records, which is 'AS/3501' in the present example, then click the **Take record as input** button. You will return to the *Failure analysis* form with the selected strength data displayed in the appropriate input text boxes. Now click on the **Save Data - All Plies** button as before.

Finally we need to input the Force pattern which is applied to the laminate. Enter the appropriate data in the force input boxes, e.g.

Force pattern applied to laminate	N x	N y	N xy	M x	M y	M xy
	0.2	0	0	0	0	0

Note that it is only the ratio of forces that is required here, so that a pattern

	N x	N y	N xy	M x	M y	M xy
Force pattern applied to laminate	1	0	0	0	0	0

would convey exactly the same information.

When the necessary data for the failure analysis have been input, click the **Calculate** button. The output boxes below the input force pattern show the failure loads,

Forces to give failure

N x	N y	N xy	M x	M y	M xy
0.327	0	0	0	0	0

telling us that the force to cause the first ply failure, when the Tsai-Wu failure criterion is applicable, will be N_x=0.327 MN/m with $N_y=N_{xy}=M_x=M_y=M_{xy}=0$. The magnitude of the input load N_x=0.2 MN/m will not affect the actual value of the failure load N_x. The relative magnitudes of the load components are determined by the input values of N_x: N_y: N_{xy}: M_x: M_y: M_{xy} ; the failure loads scales with this load pattern.

Observe also that in the ply arrangement grid on the left side, the fourth column marked "Fails first?" shows which plies have failed. For the present case, all the 45/-45 degree plies fail at the same time. Hence the grid appears as:

No.	Angle	Thick.	Fails first?
1	0	0.1	
2	45	0.1	Yes
3	-45	0.1	Yes
4	0	0.1	
5	0	0.1	
6	45	0.1	Yes
7	-45	0.1	Yes
8	0	0.1	

You have just successfully completed a deformation and failure analysis section using the standard features of the CCSM package.