LAP

User Manual

Laminate Analysis Program

L A P

User Manual

This document is the User Manual for the Laminate Analysis Program version 4.0 for Windows. It is also available on-screen through the program's Help menu command.

Document Revision 5

CONTENTS

INTRODUCTION 3

General Description 3
What is LAP? 3
What is the definition of a "laminate"? 3

Running LAP 3
What is needed to run LAP? 3
Memory 3

Installing the software 4

GETTING STARTED 5

General Usage 5
Data Definition 5
Solving 6
Printing 6
Saving 6
Design Tools 6

Menu Commands 7

Units 9

DATA DEFINITION 10

Material Definition 10
Non-Linear Material Property Definition 11
Design data for Materials 13

Lay-up Definition 13

Loading Definition 15
Design data for Loadings 17
Loading Tools 17

Preferences 18

RESULTS 21

General Results 21

Layer Stress and Strain 23

Y vs. X Results 23

Polar Results 24

Failure Indices 25

Displaced Shape 26

BFS Notched Strength 26

PRINTING PROCEDURES 28

General 28

Editing Print Reports 29

Print Report Items 30

DESIGN PROCEDURES 32

Laminate Design 32
- General 32
- Step 1 - Materials 33
- Step 2 - Loadings 33
- Step 3 - Angles 34
- Step 4 - Angle Options 34
- Step 5 - Angle Combinations 35
- Step 6 - Stacking Sequence Options 35
- Step 7 - Stacking Sequences 36
- Step 8 - Angles for final analysis 37
- Step 9 - Final Stacking Sequences 37

Batch Solution 38
- General 38
- Step 1 - Lay-ups 38
- Step 2 - Loadings 39
- Step 3 - Batch Solution Results 39

BACKGROUND THEORY 40

Solution Theory Basics 40
- Stiffness analysis 40
- Strength analysis 43
- The BFS failure criterion 45
- Prediction of Toughness by the Rule of Mixtures 47
- Non-linear solution procedures 47

Analysis Tips 49

Detailed Calculations 50

ADDITIONAL INFORMATION 56

Importing foreign data file formats 56
- Previous LAP versions 56
- *CoDA* 56
- Text files 56

Exporting to other programs 58
- NASTRAN 58
- LUSAS 58
- CoDAT (M/Vision) 59
- Text files 59
- Other 59

Using the Custom failure criterion 59

Product Information and Support 61
- Contacts 61
- Acknowledgements 61
- References 61

Introduction

General Description

What is LAP?

LAP (Laminate Analysis Program) is a software tool for the design and analysis of composite material laminates. The solution algorithms employed are based on the Classical Laminate Theory which is discussed in the *Theory* section (page 40) and in several textbooks on composite materials [1,2].

The LAP software is built around a central *Basic* module, which provides the User Interface and the simple analytical solution framework. The software further includes **optional** *Non-Linear* analysis, *Design* and *Additional Failure Criteria* modules. The present document covers all modules and therefore some sections may not be applicable to your specific licence. The program's startup screen, as well as the **About** window, show which modules are active.

The program requires the definition of basic material engineering properties at the layer level, definition of stacking sequences and definition of loadings or constraints. Material stiffness properties can be constant, or piecewise functions of stress or strain (*Non-Linear* module). Due to the efficient solver algorithm, solutions are obtained very fast and results are displayed interactively on multiple windows at the user's request. The *Design* module builds stacking sequences that satisfy design criteria of stiffness and strength, based on user-specified fibre angles. It also checks similar requirements by doing batch solutions..

The program is very easy to use via the familiar Microsoft Windows interface. Data such as material properties and laminate stacking sequences can be entered and saved in special-format binary files for later re-use. Each such file is a database in its own right.

Detailed instructions on program operation can be found in the appropriate reference sections.

What is the definition of a "laminate"?

A laminate is a thin plate of infinite length and width, made up of one or more orthotropic laminae bonded together, each of potentially different thickness and material, and each with the principal stiffness axis (or fibres) at a user-defined orientation.

LAP may be used to analyse such laminates subjected to in-plane and bending loads, deformations and hygrothermal effects. A designer can thus optimise a complex lay-up for use in a particular application. The properties of this lay-up can then be used with confidence in a more advanced type of analysis that will also take into account the structural shape and dimensions of interest.

If there are features that you would like to see supported please don't hesitate to contact the software developers.

Running LAP

What is needed to run LAP?

LAP is a Windows 32-bit application for the Intel processor platform. It runs on Windows 95/98/2000/NT, but not on earlier versions. The appearance of the User Interface may differ between Windows installations, subject to the user's display preferences (colour, borders, etc.).

Memory

There are no built-in limitations for the number of data items (materials, lay-ups, etc.) that can be held in a single data file. It is recommended, however, that users divide their data into several separate files to facilitate usage and to minimise the risk of loss through error or data corruption. Tools are provided for copying data between files.

Installing the software

To install, simply execute the **`SETUP.EXE`** program from the distribution medium. The setup procedure is straightforward: the necessary files are copied into the user-specified directory and a group is added to **Start** ▸ **Programs**.

To uninstall, follow the standard procedure through the Windows Control Panel and the Add/Remove Programs icon.

Getting started

General Usage

This topic summarises the LAP operational procedures and thus provides a brief tutorial for the program. Familiarity with the Windows user interface is assumed.

To start LAP, point to **Start** ▸ **Programs** ▸ **Laminate Analysis** ▸ **LAP 4.x** (this is the default location). It is recommended that the LAP top-level window is maximised once the program is started, to occupy the entire screen so that the multiple child windows (data and results) that need to be displayed are overlapping as little as possible. It is also recommended to use a high-resolution display configuration.

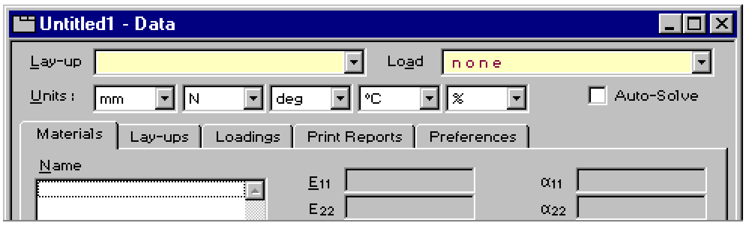
As with most Windows applications, any number of LAP data files may be open simultaneously. New (empty) files can be created by selecting **File | New**, whereas existing files can be opened by selecting **File | Open**. In addition, files of selected non-native formats can be opened through the **File | Import** menu.

General program operation is very simple. When working with a new file, the first step is to define one or more materials, then use these to define a lay-up, then apply a loading before obtaining a solution. This is briefly explained below.

NOTE: Context-sensitive help can be obtained at any time by hitting the F1 key.

Data Definition

When creating a new file, a **Data** window appears automatically. All data quantities for this file may be accessed via this window, whether to view, create or edit.



Additional **Data** windows may be opened by selecting the **Window | New Data Window** menu item, so that different data objects can be viewed side by side. When more than one data file is open at a time, the text in a window's titlebar is used to distinguish between data files: the name of the file appears on the title bar, followed by the type of window, e.g. **MyFile.ld - Data**.

Materials can be defined by first selecting the **Materials** tab in the **Data** window. Here, existing material objects are displayed, if any. There are several ways to create a new material: (a) select the menu item **Data | Add Object...**, or (b) click with the right mouse button on the window to display a floating context menu where **Add Object...** can be selected, or (c) click on the toolbar button, or (d) use the key combination Ctrl+INS. Then the name of the new material must be given, along with specifying whether the material stiffness properties are to be defined as constant or variable.

The new material name is then highlighted and its material properties can be edited. Property boxes that are left blank are assumed to be undefined, so that any results that depend on such properties will themselves be undefined. Material objects and their properties are fully covered in section *Material Definition*, page 10.

Adding or changing properties raises the question of units. To simplify this, LAP constantly displays on Data windows the currently active units (different for each file), which can be changed at any time by the user if required. Changing the current units causes all displayed data and results to re-scale accordingly. Any user input is interpreted in the currently active units.

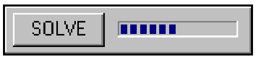
Lay-ups, or stacking sequences, can be defined by first selecting the **Lay-ups** tab in a **Data** window. The procedure is then similar to that for material creation, as discussed above. The creation of layers for the lay-up is then achieved in a similar fashion, the simplest method being to use the toolbar button . A lay-up may contain pre-defined materials only and it is not complete until both a material and a non-zero thickness have been assigned to each one of its layers (this is checked when a solution is requested). Lay-up objects and their properties are fully covered in section *Lay-up Definition*, page 13.

Loadings, in the form of loads per unit length, moments per unit length, strain or curvature constraints, operating temperature and moisture uptake can be defined in similar ways by selecting the **Loadings** tab in a **Data** window. Loadings are covered in detail in section *Loading Definition*, page 15.

Solving

Before a solution can be carried out, a Lay-up and a Loading must be selected in the drop-down boxes at the top of a **Data** window. The solution is always carried out for this Lay-up / Loading pair, irrespective of which Lay-up or Loading is highlighted in the tabs for Lay-up / Loading definition. The built-in Loading "n o n e" allows simple solutions where only Lay-up characteristics are obtained.

The timing of the solution depends on whether the **Auto-Solve** box is checked in the **Data** window. Checking it means that every time there is any change in the database that could affect the solution set, a new solution is carried out automatically. Not checking it means that the user must explicitly request a solution as needed, by using the **SOLVE** button in the toolbar, or by pressing F9.



During solution, an **ABORT** button replaces the **SOLVE** button, and can be used to stop a solution before completion. The solution can be aborted in between loading steps, but not during convergence calculations within *each* step. The progress bar shows the progress of the solution.

Solution results can then be inspected via the various options available through the **Results** menu. These include effective stiffness and hygrothermal coefficients, solution vectors and matrices, layer stresses and strains, indices based on various failure criteria, plots of various quantities and a representation of the laminate's displaced shape. A detailed discussion of these is available in section *Results*, page 21.

Printing

A hard copy of database entries and solution results can be obtained in a variety of ways. On one hand, the user can preview and print the contents of any window individually, on the fly; on the other hand, generic reports can be defined that will produce the same set of selected items for any solution, thus promoting consistency.

Printing can be achieved via the **File** menu, or the toolbar, or context menus (right-clicking on the desired window), or key shortcuts. In addition, the **Print Reports** tab in **Data** windows is used for generic reports. All output is generated in the currently selected units.

For a detailed discussion on using printers, please refer to section *Printing procedures*, page 28.

Saving

Finally, the data defined during any session should be saved on disk so that it can be re-used. Through the **File** menu item, select **Save** or **Save As...** to save the data to a file. This file can then be read back into LAP by selecting **File | Open...**

Design Tools

There are two design tools available: Laminate Design and Batch Solution.

Briefly, the **Laminate Design** (page 32) procedure consists of specifying a set of mechanical loadings, each with stiffness and strength constraints which must be satisfied by an optimum laminate to be designed. The materials and fibre angles to be considered by the software, as well as a number of other options, are specified by the user during the design process. LAP carries out a thorough investigation of the candidate stacking sequences and presents a finite set to the user, for further selection or optimisation.

With **Batch Solution** (page 38), a number of laminates are examined to confirm that they indeed satisfy the stiffness and strength requirements set by the user. Typically, this is used to confirm compliance after changes have been introduced to the laminates suggested by the software.

These two Design procedures are initiated via the relevant toolbar buttons, or through the **Results** menu. The data used for design originate from various places in the program, and these are examined in full detail in the appropriate reference sections.

Menu Commands

In this section the functions of the menu commands are explained. Where applicable, the relevant toolbar button and key shortcut are also displayed.

<u>The **File** menu contains the following items:</u>

New Ctrl+N	Creates a new (empty) LAP datafile.
Open... Ctrl+O	Displays the standard "File Open" window, where the user can search for the desired LAP datafile to be loaded. The datafile is opened and read without being locked, i.e. another instance of LAP can work on the same file concurrently. This is not recommended, especially in a networked environment, and should not be practised to avoid confusion and overwriting data from different parallel sessions.
Close	Closes the active file, prompting to save it if necessary. The file also closes when the user manually closes all windows open for that file.
Save Ctrl+S	Saves the active file to disk (**.ld** format). If no name is already defined, the **Save As...** option takes over, requesting a filename definition.
Save As...	Saves the active file to disk by requesting a filename.
Import ▸	See page 56.
Export ▸	See page 58.
Print... Ctrl+P	Prints the contents of the active window.
Print Preview Ctrl+R	Switches the active window to a Print Preview layout.
Print Setup...	Allows for the selection of the printing device, as well as for custom setup regarding paper size, orientation, etc.
Preferences...	Displays a window where a number of parameters can be defined for general program usage.
(Recent Files)	Displays a list of the most recently opened datafiles, for quick selection.
Exit	Terminates the current session. If there are unsaved changes in any datafile, the user is prompted for a last chance to save to disk.

<u>The **Edit** menu contains the following items:</u>

Undo . . . . Ctrl+Z	Cancels the last action applied to a Material, Lay-up, Loading, or Report object. A reminder of that action is given.
Redo Ctrl+Y	Reapplies whatever was just cancelled.
Copy (text) Ctrl+C	Places the contents of the active window on the Windows clipboard. This is done in text format even if the window is displaying a graph, and may not be an exact copy of the window contents. This function is intended to facilitate the quick transfer of meaningful results to spreadsheets or word-processors.

The **View** menu contains the following items:

Toolbar	Displays or hides the program's main toolbar.
Solve Bar	Displays or hides the program's SOLVE button toolbar.
Status Bar	Displays or hides the program's status bar.

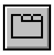
The **Data** menu contains the following items:

Add *Object...* Ctrl+INS	Creates a new Material, Lay-up, Loading, or Report object.
Delete *Object* Ctrl+DEL	Deletes the highlighted Material, Lay-up, Loading, or Report object.
Rename *Object...*	Renames the highlighted Material, Lay-up, Loading, or Report object.
Duplicate *Object...*	Duplicates the highlighted Material, Lay-up, Loading, or Report object.
Convert Material ▸	
To Non-Linear by Stress **To Non-Linear by Strain**	Converts the highlighted Material to one with variable stiffness properties. The existing stiffness and strength data are used as a reference in the conversion.
Layer(s) ▸	
Add below Shift+INS	Adds a layer below each highlighted layer. If no layers are highlighted, a layer is added at the bottom (or mid-plane) of the lay-up.
Add above	Adds a layer above each highlighted layer. If no layers are highlighted, a layer is added at the top of the lay-up.
Delete Shift+DEL	Deletes the highlighted layer(s).

The **Results** menu contains the following items:

General F3	Displays general results in tabular form, including effective stiffness and hygrothermal coefficients, effective strength, solution vectors and matrices.
Layer Stress / Strain F4	Displays stress or strain results layer by layer, in graphical or tabular form.
Y vs. X F5	Displays combinations of various result quantities, such as Load vs. Strain, in graphical or tabular form. Useful mainly for non-linear solutions or multi-order loadings.
Polar F6	Displays effective stiffness and hygrothermal coefficients at off-axis orientations, in graphical or tabular form.
Failure Indices F7	Displays the various failure indices (criteria) layer by layer, in graphical or tabular form.
Displaced Shape F8	Displays the displaced shape of a square laminate element, in 3D view.
BFS Notched Strength F11	Displays a window where the Notched Longitudinal Compressive Strength of various configurations can be obtained.
SOLVE : Lay-up+Loading SOLVE F9	Solves for the Lay-up, subjected to the Loading, as selected in the top controls of the Data window.
DESIGN : Laminates Ctrl+F9	Initiates the Laminate Design procedure, whereby a laminate is designed based on specific user requirements.
DESIGN : Batch Solution Shift+F9	Initiates the Batch Solution Design procedure, whereby a number of Lay-ups are checked for a number of Loadings, to ensure stiffness and strength conformance.

The **Window** menu contains the following items:

New Data Window  F2	Opens a Data window.
Cascade	Arranges windows in an overlapped fashion, from top-left to bottom-right.
Tile	Arranges windows in non-overlapped tiles.
Arrange Icons	Arranges icons of minimised windows.
(Window 1,2,...)	All open windows register their presence here, so that obscured windows can be brought to the top by selecting their name. This facility can also be used to move between windows in the absence of a mouse.

The **Help** menu contains the following items:

Help Topics	Brings up the on-line help at the Contents page.
About LAP...	Displays software version, owner registration and copyright information.

In addition, some of the above menu items also appear in context menus (obtained when using the right mouse button), where they serve similar functions.

Units

Changing the current units simply involves accessing a **Data** window and making selections in the five drop-down units boxes which are arranged in the order:

Length - Force - Angle - Temperature - Strain.

All numerical data and results are entered and displayed in the **current units**, as selected here. The current units can be changed at any stage during program operation, except when the program is busy or when displaying certain windows that prevent access to other windows, such as the non-linear material definition dialog. Changing the current units causes all the displayed numbers to re-scale accordingly. Print reports are also generated in the units current *at the time of requesting the print*.

Often, data and results are displayed by LAP without making reference to the relevant units of each item. For example, stiffness modulus numbers may be displayed as:

150.3

instead of: 150.3 N/mm^2

However, information on the dimensional units applicable to each such item can be obtained in the interactive help. For example, in the help topic for Materials it is shown that stiffness moduli are expressed in units of $Force/Length^2$.

Exceptionally, some quantities are entered or displayed in fixed units, such as the Moisture Content in the Loadings window which is always expressed as a percentage increase in weight, and the view angles for the Displaced Shape window which are always expressed in degrees.

There is one peculiarity with the Temperature units: scaling between Celsius and Fahrenheit is simply to multiply or divide by 1.8, i.e. the shift in origin of 32°F is not taken into account. Please ensure that all temperatures are entered in the same units.

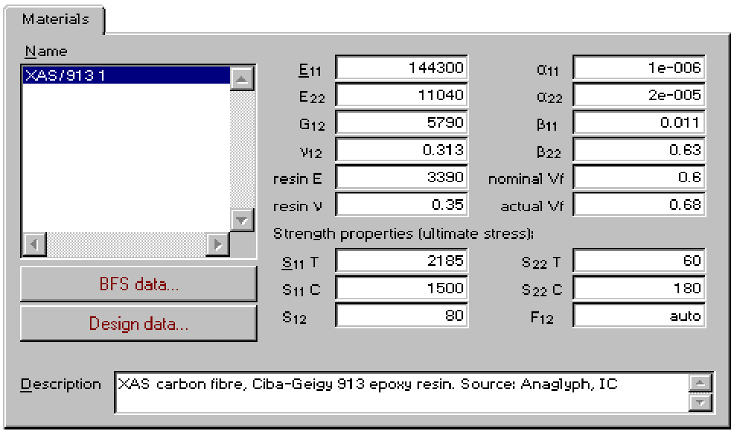
The Strain units affect strain values *only*. For example, stiffness modulus is defined as (stress / strain) and should strictly be affected by Force, Length and Strain units. However, the Strain unit is simply intended to make strain values easier to examine and therefore it is only used for strains.

There is also a Mass unit that is used by LAP, for the definition of material density. This is fixed in kilogrammes (kg) in the current version of the software. Similarly, there is a Currency unit, which is used to define material cost for design purposes. This is a user-specified name or symbol in the general design preferences and is used simply for comparison and reporting purposes.

Data definition

Material Definition

The **Materials** tab in a **Data** window is used to view and edit material properties.



New materials can be created via:

- The top-level menu item **Data | Add Material...**, or
- The window's context menu item **Add Material...** (click the right mouse button to display the menu), or
- The toolbar button, or
- The key combination Ctrl+INS.

A unique **Name** must be given to each material, for identification purposes. There is no real limit to the length of such a name but, for practical reasons, it should be kept short. The **Description** field can be used for a longer, more descriptive sentence.

When a new material is created, it must be classified as linear or non-linear (only if the *Non-Linear* module is active), referring to the stiffness properties variation with load ("linear" means that all properties are constant). If non-linear, the user must also select the way stiffness properties are to be defined: as functions of strain or of stress. The choice here depends on the available data, but also on material behaviour.

Materials that have been defined as linear can be converted to non-linear at any time, via the menu items **Data | Convert Material ▸ To Non-Linear by Stress** and **To Non-Linear by Strain**. These conversions use any existing stiffness and strength data to build property "functions" that are constant between the specified strength limits. These can then be used as a starting point for further user editing.

When working with a lay-up that contains only linear materials, LAP applies the selected loading to the selected lay-up in one step (see *Loading Definition*, page 15 for more information). If the resulting stresses cause failure (according to any one of the selected failure criteria, page 18), LAP simply reports the fact that some layers have failed. No iterative calculations are carried out to reduce the stiffness of the failed layers. On the other hand, if the selected lay-up contains at least one material with non-linear properties, then the loading is applied in the designated number of steps, so that stiffness properties can vary and failed layers be "eliminated" progressively. More details on stiffness degradation due to failure can be found on page 11.

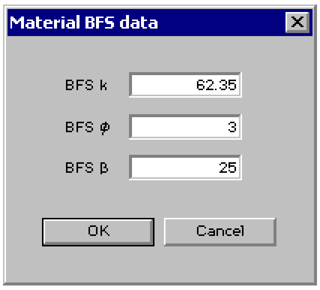
Additional operations on material objects are: **Delete**, **Rename** and **Duplicate**, available from the top-level **Data** menu, the local context menu, or the toolbar. These operations are applicable to the highlighted material.

Material objects can be copied between LAP files using the "drag and drop" method. Source and target datafiles must both be open. The material to be copied is first selected *while holding the Ctrl key down*, then dragged and dropped over the Materials tab of the target datafile. In this procedure, the cursor changes to the familiar arrow with a [+] sign attached, to denote a **copy** operation.

All operations on material objects (add, delete, copy, edit) can be undone via the **Edit | Undo** menu item. There are several undo (and redo) levels available which can trace all edit steps backwards (or forwards). Note, however, that undo actions are not specific to materials, they apply to edit operations for any type of object. To avoid confusion, the Undo menu item provides an indication of the next action to be undone. Also note that the individual property edit boxes also offer an Undo feature in their context menu, but this is limited to local typing activity.

Material properties are viewed, defined or edited for the highlighted material.

- The minimum requirement for definition includes the stiffness moduli $\mathbf{E_{11}}$, $\mathbf{E_{22}}$, $\mathbf{G_{12}}$ and the Poisson's ratio $\nu_{\mathbf{12}}$. Materials are automatically classified as orthotropic.
- Thermal expansion coefficients (α) are required only if temperature-related stresses and strains are expected.
- Similarly, moisture expansion coefficients (β) are required only if moisture-related stresses and strains are expected.
- **Resin** properties (E and ν) are required only if the nominal and actual fibre volume fractions (**Vf** ’s) are not identical. In this case, LAP can make small corrections to layer thickness and stiffness, to account for manufacturing imperfections. Because only small variations in Vf are allowed, the permissible ratio of Vf*nominal*/Vf*actual* must be in the range 0.75 to 1.25.
- **Strength** properties are required (for linear materials) only if the failure criteria calculations are needed. In particular, the $\mathbf{F_{12}}$ term for the Tsai-Wu and Hoffman equations can be explicitly defined, or left for the program to calculate a suitable value.
- The **BFS data...** button displays a window where the BFS failure criterion parameters are viewed and modified. These parameters are required (for linear materials) only if the calculation method for the BFS Unnotched longitudinal compressive strength is based on the Budiansky-Fleck model (see *Lay-up Definition*, page 13). The three BFS parameters are explained in the Theory section, page 40.



All material properties must be defined at the **nominal** fibre volume fraction.

Property boxes that are left blank are assumed to be undefined, so that any results that depend on such properties will themselves be undefined.

The window's context menu also offers **Print** and Print Preview operations, with which material properties can be printed. If a material name is highlighted in the list, printing occurs only for that material; otherwise printing occurs for all materials.

Relevant Units:

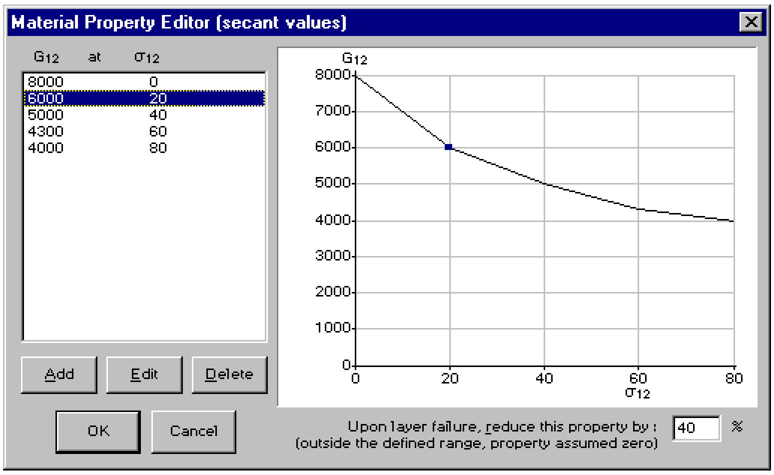
E, G, S, BFS k	Force / Length²
ν, β, Vf	No units
α	1 / Temperature
F_{12}	(Length² / Force) ²
BFS ϕ, β	Angle

Non-Linear Material Property Definition

In the case of non-linear materials, the stiffness property boxes are replaced by  buttons, through which the non-linear behaviour may be defined. Also, the material strength boxes are in this case disabled because strength is calculated automatically, based on the stress or strain range used to define each stiffness component.

E_{11}, E_{22}, G_{12} are defined in terms of their equivalent stress or strain, whereas the Poisson's ratio is defined based on the stress or strain along the fibre direction.

Clicking on a  button displays the **Material Property Editor** window.



Here, points of *[secant property value]* vs. *[stress or strain]* can be **added**, **edited** or **deleted** thus defining the variation of stiffness with load, as well as the limiting values for stress or strain. These limits (translated to stress if needed) are displayed in the Strength boxes of the **Materials** tab and are used by the various failure criteria. They cannot be edited, since they are deduced from the non-linear data.

The non-linear properties must *always* be defined as **secant**. The solution algorithms depend on this fact. Also, it makes sense to provide stiffness data from tests carried out at the mean operating temperature, since that is where the mechanical loads are assumed to apply.

Outside the defined ranges of stress or strain, properties are assumed to be **zero**. This can cause catastrophic failure behaviour and an appropriate message is displayed at solution time. On the other hand, the remaining stiffness in the laminate may be adequate in supporting the applied loads and the solution carries on. For more details please refer to the *Theory* section.

As points are added they are sorted by increasing stress or strain. Points cannot coincide in stress or strain values. A sudden change in property (indicating a partial failure perhaps) is not recommended when defining by stress, since a sudden change in stiffness at the same stress can cause the solver problems in balancing the internal stresses to the applied loads (the message *Maximum number of iterations exceeded* will appear during solution). On the other hand, two points can be specified at adjacent strain values, implying a drop in both stiffness and load due to partial failure. Hard loading (strain stepping) will pick up this drop, whereas soft loading (load stepping) will result in a jump in strain.

The **percentage reduction** for a property can be used to model degradation in stiffness upon layer failure. This is applicable when any one of the selected failure criteria locally exceeds 1 at the end of a load step. In this case, layer stiffness properties are reduced by the specified amounts before additional load is applied. Note, however, that if a local stress or strain component falls outside the range used to define a stiffness property, then that property is reduced to zero.

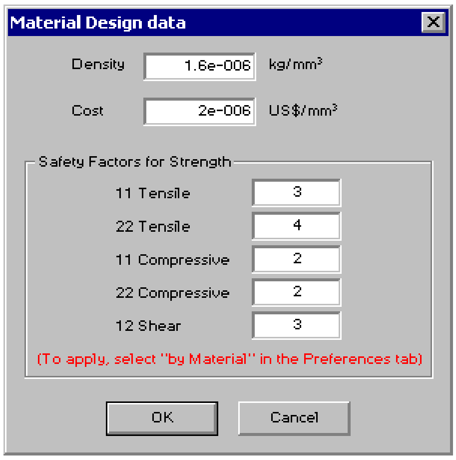
It must be stressed that the defined percentage reductions occur only the **first** time any one selected failure criterion exceeds 1 at that layer. Depending on the problem, the selected failure criteria can build up to exceed 1 again, despite the reduction in stiffness.

Relevant Units:

E, G, σ	Force / Length^2
ν	No units
ε	Strain

Moduli and shear stress or strain values cannot be negative.

Design data for Materials



The Design data window for Materials is accessible via the **Design data...** button on the **Materials** tab.

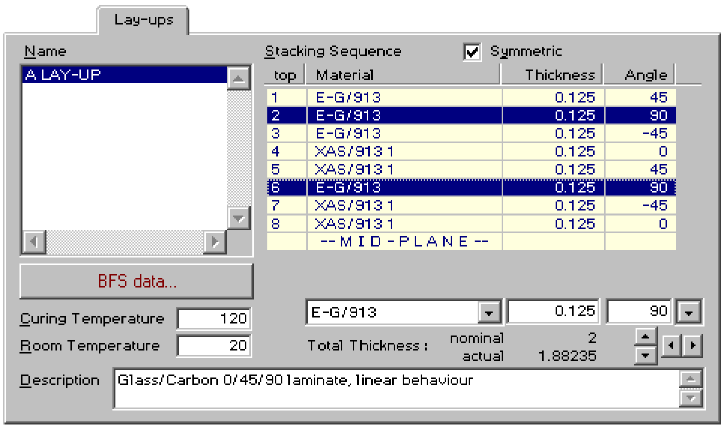
Here, a number of parameters can be defined, which are used by the *Design* module.

The **Density** and **Cost** are necessary if this particular material is to be a candidate material in the Laminate Design procedure (page 32). Note that the mass unit is fixed in kilogrammes. The currency is specified in the Design Options window (page 18), and is common to all materials.

The **Safety Factors** are necessary for the Design procedures, but only if the method of using safety factors is set to "**by Material**", an option found in the Design Options window. In design calculations for strength compliance, material strength properties are divided by the equivalent safety factors.

Lay-up Definition

The **Lay-ups** tab in a **Data** window is used to view and edit lay-up properties.



New lay-ups can be created via:

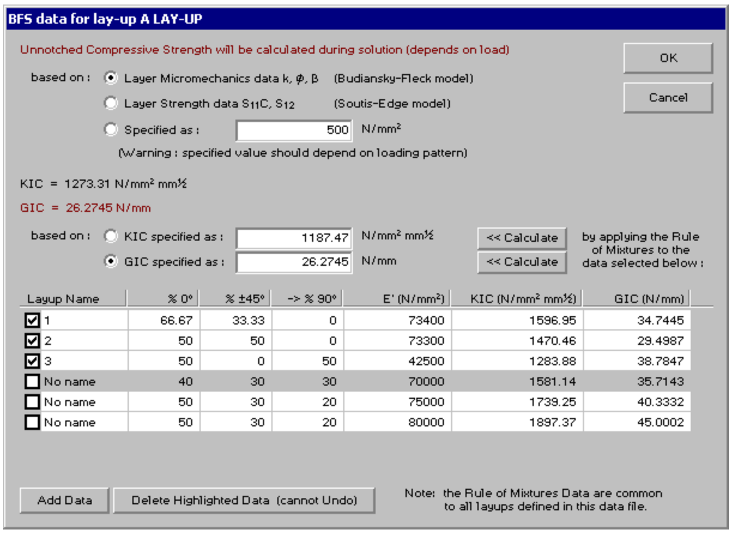
- The top-level menu item **Data | Add Lay-up...**, or
- The window's context menu item **Add Lay-up...** (click on the right mouse button to display the menu), or
- The toolbar button +, or
- The key combination Ctrl+INS.

In other words, the operations for creation of lay-ups are identical to those already seen for materials, as long as it is the Lay-ups tab that is selected. Similarly, as with materials, a short unique **Name** must be given to each lay-up for identification purposes, while a **Description** field can be used for a longer, more descriptive sentence. Furthermore, **Delete**, **Rename**, **Duplicate** and **Undo** functions are available for lay-ups in ways similar to what has already been discussed for material objects.

Drag-and-drop **copy** operations are also available for lay-up objects, but with one added complication: when a lay-up is copied from one datafile to another, the data for its constituent materials are also copied with it. If, however, a material already exists by the same name in the target datafile, then the source material object is not transferred. The copied lay-up object ends up using the pre-existing material by the same name, so it probably has different properties in the target datafile. LAP issues a warning for such situations.

Lay-up properties can be viewed, defined or edited for the highlighted lay-up:

- The **Symmetric** option specifies a lay-up that is symmetric about the mid-plane in all respects.
- The **Curing** and **Room temperatures** are used to calculate curing strains.
- The layer **stacking sequence** can be edited via the top-level **Data | Layer(s)** menu, the local context menu, the toolbar buttons, or the keystrokes Shift+INS and Shift+DEL, which allow for **insertion** or **deletion** of layers. To select more than one layer, use the standard Windows method of Ctrl+Click. The default thickness for new layers can be defined in the general Preferences (page 18).
- The three edit boxes just below the stacking sequence window (layer **material**, layer nominal **thickness**, layer 1-axis orientation **angle**) are used to define or change individual layer properties, for single or multiple layer selections. The drop-down box at the far right can be used to enter the frequently used values of 0°, 90° or ±45º for angles.
- The **BFS data...** button displays a window where two properties need to be defined: the Unnotched longitudinal Compressive Strength and the compressive Fracture Toughness:



The unnotched strength may be specified explicitly, or a model for its calculation can be selected (Budiansky-Fleck or Soutis-Edge). The toughness must be specified explicitly (K_{IC}) or in terms of the strain energy release rate G_{IC}.

As an aid in specifying the toughness, the Rule of Mixtures can be applied to any available experimental lay-up data, which may be kept in the table provided. The data required are: the percentages of the 0º and ±45º layers (the percentage of the 90º layers is then computed), the measured effective elastic modulus E' (explained in the *Theory* section), and the measured K_{IC} or G_{IC}. These last three parameters are related via K^2=GE'. The data may be input or modified by clicking on the appropriate cell, and are available to all Lay-up objects. Once the RoM data input is complete, the selection boxes must be checked for the lines contributing to the computation and one of the **<<Calculate** buttons pressed. As a result, the K_{IC} or G_{IC} is estimated for the current lay-up.

The **up-down** arrow buttons below the angle property box can be used to move any selected layers up or down the stacking sequence. This may be especially useful for quick modifications to lay-ups built by the *Design* module.

Similarly, the adjacent **left-right** arrow buttons can be used to instantly rotate an entire lay-up in steps of ±5º. Since automatic solutions are normally very quick, this feature can be used to inspect various aspects of laminate behaviour at off-axis positions (despite the fact that several polar results are available anyway).

The total lay-up thickness is displayed both *nominal* and *actual*. The total nominal thickness is simply the sum of the specified layer thicknesses, whereas the total actual thickness also takes into account any disparity between the nominal and actual fibre volume fractions for the material in each layer. The actual thickness for a layer is calculated from:

$$t_{actual} = t_{nominal} \frac{Vf_{nominal}}{Vf_{actual}}$$

All data are displayed and entered in the currently selected units. Angles are limited to the ±90º range.

There is a built-in material called " c o r e", which is used for the core layers in the sandwich laminate that is built by the Laminate Design procedure. This appears in the materials drop-down box so that it can be selected into layers, but it does not appear in the **Materials** tab because it is not editable.

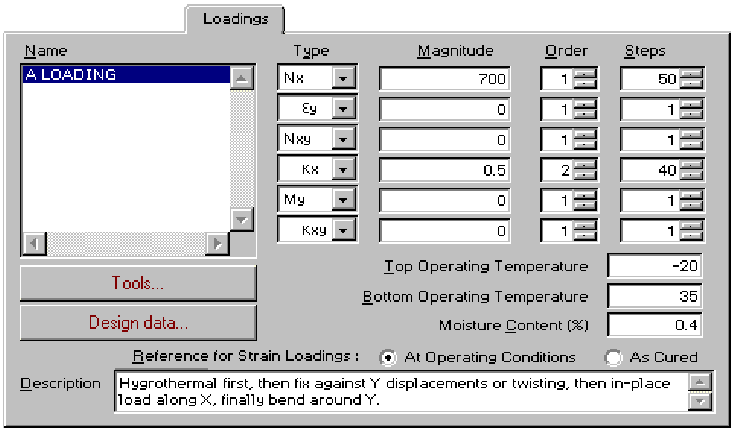
The window's context menu also offers **Print** and Print Preview operations, with which lay-up properties can be printed. If a lay-up name is highlighted in the list, printing occurs only for that lay-up; otherwise printing occurs for all lay-ups.

Relevant units:

temperatures	Temperature
thickness	Length
angle	Angle
unnotched Strength	Force/Length²
K_{IC}	Force/Length² Length$^{1/2}$
G_{IC}	Force/Length

Loading Definition

The **Loadings** tab in a **Data** window is used to view and edit loading properties.



The operations for creation, deletion, etc. of loadings are identical to those already seen for materials or lay-ups, as long as it is the Loadings tab that is selected. Furthermore, undo and drag-and-drop copy operations are equally available for loading objects.

Loading properties can be viewed, defined or edited for the highlighted loading:

- The **Type** boxes define whether a load component is a Force / Moment or a Strain / Curvature. A Force / Moment loading can be used to model soft loading in a non-linear analysis (load-controlled stepping) while a Strain / Curvature loading can be used to model hard loading (displacement-controlled stepping).
- The **Magnitude** of a component is simply its value, in relevant units.
- The **Order** field specifies an optional order in the application of the mechanical components. Mechanical loading is assumed to apply at the operating conditions, i.e. the overall loading order is Curing, then Moisture, then Thermal and finally Mechanical components. This feature can be important in certain cases where a combination of component loadings is not as detrimental as a subset of it. If there are gaps in the overall order specified (for example if Nxx has order=1 and all other components have order=3) then the gaps are ignored during a solution.
- The **Steps** field defines the number of equal incremental steps that will be used to carry out a non-linear solution (this field is ignored for lay-ups with linear materials only). If different steps are defined for

components in the same order, the *maximum* value is used.

The number of steps need not be very large if only the final state of stresses and strains is required. LAP has efficient methods of auto-correcting at the end of each load increment, thus giving very accurate final results even with a few steps. However, if the exact shape of non-linear Y vs. X curves, or if exact failure stresses and strains are required, then a high number of steps is necessary.

- The **Top** and **Bottom Operating Temperatures** are the operating temperatures at the top and bottom surfaces of the laminate respectively, in units of Temperature. Their relative difference from the lay-up's room temperature is used to calculate the thermal strains. A linear variation of temperature is assumed through the laminate thickness.
- The **Moisture Content (%)** is the laminate moisture uptake (by weight), and is assumed constant through the laminate thickness.

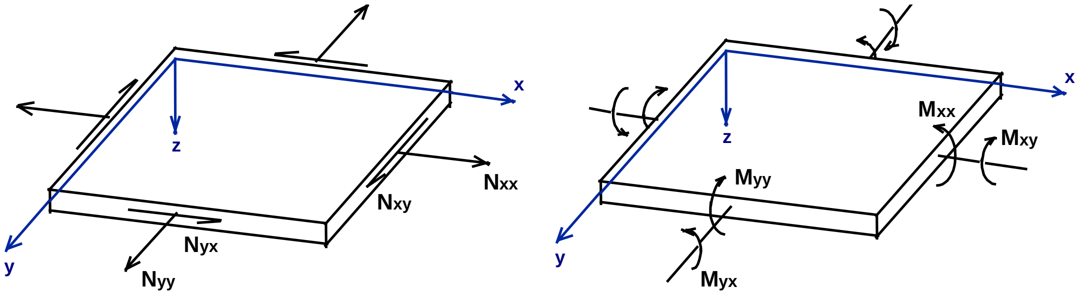
If strain or curvature constraints exist in the loading specification, a reference point must be given, this being either the **Operating Conditions** state (where curing and hygro-thermal strains are non-zero) or the **As Cured** state (where only curing strains exist).

An example may help clarify this last point. Let us assume that we have an arbitrary laminate. Once manufactured, this laminate exhibits deformations that make up a strain vector εc (this is the "As Cured" state). In LAP, this strain comes from the temperature change "Curing" to "Room". It is then exposed to moisture, which changes the strain vector by εh. It is then brought to the operating temperature, which further changes the vector by εt. In LAP, this strain comes from the temperature change "Room" to "Operating" temperatures. At this point, the total strain vector is εc + εh + εt (this is the "At Operating Conditions" state). We now want to "stretch" the laminate in the X direction by a certain known displacement, say 1% strain. Using LAP, we impose a mechanical strain εmx=1% using reference: "At Operating Conditions" and leave all other components "free", i.e. with load=0. The load vector in the results gives us the force (Fx) needed to achieve this additional extension of 1%. The TOTAL strain in the X direction will be (εcx + εhx + εtx + 0.01).

An alternative scenario could be as follows: the laminate is brought to service immediately after manufacture (without moisture and at room temperature), and it is "stretched" by the 1% strain, say because it must fit in place where it is bolted. The laminate then absorbs moisture and is brought to the operating temperature, but the X strain does not change because the laminate is bolted in place. To analyse this case with LAP, we must impose the mechanical strain with reference at the "As Cured" state, so that the TOTAL strain in the X direction is "kept" at (εcx + 0.01). This means that as the moisture uptake and operating temperature change, the mechanical stresses are adjusted so that the total strain is kept where it is required.

The window's context menu also offers **Print** and Print Preview operations, with which loading properties can be printed. If a loading name is highlighted in the list, printing occurs only for that loading; otherwise printing occurs for all loadings.

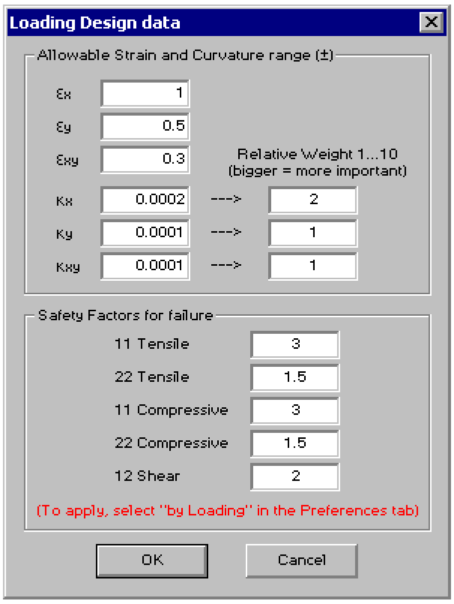
The axis system convention for loads is shown below:



Relevant Units:

forces	Force / Length
moments	Force
strains	Strain
curvatures	1 / Length
temperatures	Temperature
moisture	No units (but entered as a percentage)

Design data for Loadings



The Design data window for Loadings is accessible via the **Design data...** button on the **Loadings** tab.

Here, a number of parameters can be defined, which are used by the *Design* module.

The **Allowable Strain and Curvature range** parameters are necessary if this particular loading is to be included in the Laminate Design procedure (page 32). The allowable range parameters define the limits within which the optimum laminate can deform, and hence define its stiffness properties. By specifying unreasonably wide ranges, the design process can be forced to ignore laminate stiffness and concentrate on strength only. The **Relative Weight** parameters should be set to 1, unless emphasis can be placed on a certain component, thus helping the design algorithm optimise the laminate more efficiently.

The **Safety Factors** are necessary for the Design procedures, but only if the method of using safety factors is set to "**by Loading**", an option found in the Design Options window (page 18). The safety factors are used to effectively reduce the material strengths when these are compared to layer stresses during strength compliance calculations.

Relevant Units:

strains	Strain
curvatures	1 / Length

Loading Tools

The **Tools...** button on the **Loadings** tab is used to display the **Loading Tools** window.

In the top half of this window, a rotation can be applied to the currently selected loading. This may be useful if loading details are known at a certain angular offset from the fibre orientations of the lay-up that is to be used with this loading. For example, if the applied load is a single tensile force of 100N at +30° to the lay-up's x-axis, we can define a loading Nxx=100N, with the other components equal to zero, and then rotate this by +30° in order to obtain the correct Nxx, Nyy and Nxy components. The application of the rotation permanently modifies the loading values when the OK button is pressed.

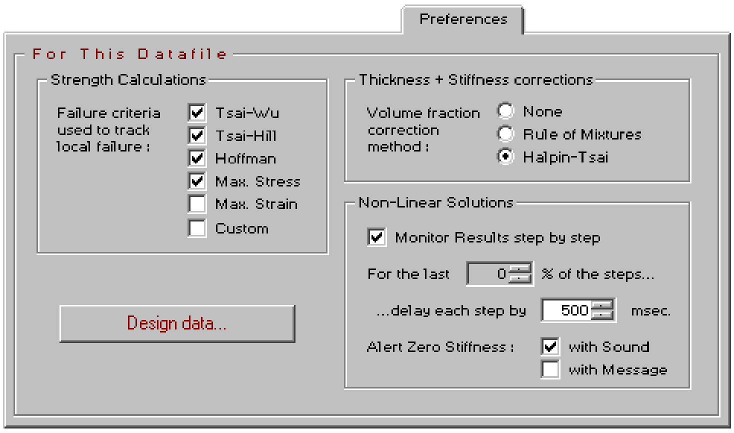
The orientations where shear force or twisting moment are zero are shown for the unrotated loading, rounded up to the nearest integer value for clarity. Note that in order to transform the loading to achieve either zero shear or zero twisting, an equal and opposite rotation must be applied to the one indicated.

Rotations are not available for loadings that include strain or curvature components.

In the bottom half of the Tools window, the relation between curvature and surface strain can be studied. This tool is simply to help users convert a required surface strain value to the appropriate curvature data. The values entered here are temporary and do not modify the selected loading parameters.

Preferences

A number of options applicable to each datafile separately can be set in the **Preferences** tab of a **Data** window.



The **failure criteria** to be monitored (for exceeding 1) during a solution can be selected individually. The criteria that are not selected are calculated nevertheless, but are ignored in strength assessment. The selection of the Custom failure criterion is possible only if the *Additional Failure Criteria* module is active.

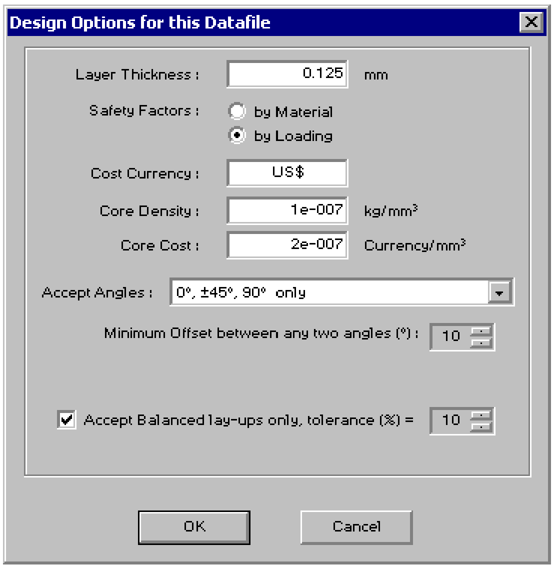
The method for **volume fraction** corrections may be selected here, including the choice of deactivating this feature completely.

The **Monitor Results step by step** option is used to enable or disable window updates upon completion of each load step in non-linear analyses. If monitoring is disabled, solutions complete much faster.

When monitoring results step by step, the option is also given to "slow down" the application of some of the last load steps, by specifying a **delay** time. This is useful when attempting to watch what is happening near failure zones, but the computer goes too fast for comfortable observation. In addition to this functionality, users can manually follow a solution step-by-step, by pressing the **ABORT** button at any point. Then, a dialog box will offer the options of aborting, or applying one further load step, or continuing.

For non-linear materials, properties are set to zero if the local layer stress or strain exceeds the given range of validity (see *Non-Linear Material Property Definition*, page 11). The exact step where this happens can be notified with a short beep or with a popup message. These options are set with the **Alert Zero Stiffness** boxes.

The **Design data...** button displays the **Design Options** window, where parameters for the *Design* module may be examined and modified. All options are applicable to the Laminate Design procedure only, apart from the Safety Factors option which also applies to the Batch Design solutions.



The **Layer Thickness** is the thickness used for a single layer in the laminate to be designed. It determines the "granularity" of the solution, as it defines the "step" for increasing laminate thickness as well as the proximity of fibres at different orientations.

The **Safety Factors** option determines whether the safety factors that are used in combination with material strength properties to calculate strength compliance during the design process, originate from each material or from each loading. The former choice modifies a material's strength properties consistently for any loading. The latter choice modifies the strength properties depending on the load.

The **Cost Currency** is simply a name for the currency used in specifying material cost. It is common to all materials so that cost comparisons can be made during design.

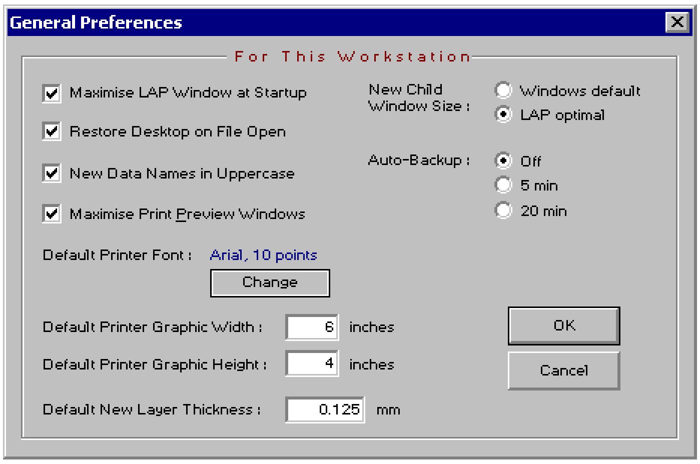
There is a built-in material called " c o r e", which is used for the core layers in the sandwich laminate that is built by the Laminate Design procedure. The **Core Density** and **Cost** are needed to calculate the weight and cost of these layers, simply for comparison with the non-sandwich laminates.

The **Accept Angles** drop-down box is used to define which fibre orientations are acceptable for Laminate Design. This is more of a recommendation than a restriction, since selecting angles that are not within this preference simply produces a warning during design.

The **Minimum Offset between any two angles** specifies that the group of fibre angles finally selected for Laminate Design cannot contain any pair whose difference is less than this value. The range allowed for this option is 1° to 90°, but must normally be kept below 45°.

By activating the last option, to **Accept Balanced lay-ups only**, stacking sequences that exhibit coupling between stretching and shearing will be disqualified by Laminate Design. A less severe restriction is introduced by setting a **tolerance** value for this rule, based on the relative magnitude of certain terms in the lay-up's stiffness matrix.

Furthermore, options that are applicable to <u>all datafiles</u> can be defined via the **File | Preferences...** menu item. The options specified here are in fact stored in the Windows Registry, so they are applicable to all datafiles that are accessed by running LAP on the same computer.



The **Maximise LAP Window at Startup** option ensures that every time that the LAP application is launched it will appear in a maximised window.

The **Restore Desktop on File Open** option makes the program remember the position, size and parameters for all Data and Results windows that were open at the time of the last file save operation. This can make users standardise on a favourite screen layout and thus save time in re-establishing it every time. The window information is stored with each datafile, so it is applicable to each datafile separately. The functionality of this option is independent of screen resolution, since scroll bars appear whenever necessary to access all the restored windows.

The **New Data Name in Uppercase** box offers the option of working with object names that are uppercase only, or a mixture of upper- and lowercase.

The **Maximise Print Preview Windows** option controls the appearance of windows when in print preview mode.

The **New Child Window Size** option controls whether new LAP windows will be created in a size that is decided by LAP or by Windows. The advantage of using the LAP option is that the windows will be optimally sized for viewing and will always have the same size, irrespective of screen resolution or of parent window size.

The **Auto-Backup** option can be activated to produce a backup of the current data in the Windows Temporary directory. The backup files are deleted whenever a LAP application is closed so, if needed, they must be used while LAP is active!

The **Default Printer Font** is the font that is used for new Print Report objects, but also for all print operations that are not done through the Print Report procedures. It is advisable to use True Type fonts only, and also to avoid fonts that may not appear on most computers.

The **Default Printer Graphic Width** and **Height** are the printed dimensions of all graphic objects in new Print Report objects, but also for all print operations that are not done through the Print Report procedures.

The **Default New Layer Thickness** is used to standardise on the thickness of newly created layers for lay-ups.

Results

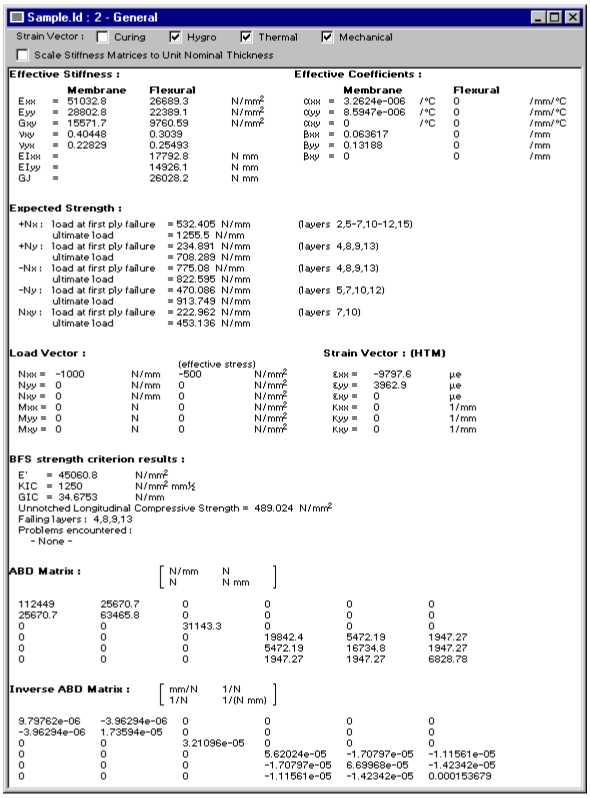
Multiple results windows can be opened concurrently to display the various analysis results. Windows can be moved and resized as required, to make optimum use of the screen. The menu item **Window | Tile** can also be useful in achieving a convenient window arrangement.

Multiple windows of the same type of results are allowed. For example, there can be several windows showing layer stresses, where each window is configured to display stress at a different orientation (xx, yy, 11, etc.).

Results windows can be launched from the **Results** menu, form the toolbar buttons , or using keys F3 to F8 and F11.

The displayed results are relevant to the last solution carried out. Furthermore, except for the laminate strength and the Y vs.X set of results, all other results (stiffness, stress, etc.) are reported at the last step that was successfully completed. Also, wherever there is the facility to examine Curing, Hygroscopic, Thermal and Mechanical components separately, it must be remembered that for non-linear analyses these components can influence one another and are strictly inseparable.

General Results



Selecting **Results | General** from the menu displays a window with a variety of numerical results.

Exx, Eyy, Gxy, νxy, νyx (membrane and flexural), **EIxx, EIyy** and **GJ** are the effective laminate stiffness properties and are reported at the last completed load step.
The flexural moduli are calculated from the equation $M = E\, t^3\, \kappa / 12$.
The effective bending stiffness EI is defined by $EI = M / \kappa$.
The effective twisting stiffness GJ is defined by $GJ = (2\, M_{xy}) / (½\, \kappa_{xy})$.

αxx, αyy, αxy (membrane and flexural) are the effective thermal expansion coefficients for the laminate, which are equal to the thermal strain vector for a uniform temperature difference $\Delta T = 1$.

βxx, βyy and **βxy** (membrane and flexural) are the effective moisture expansion coefficients, which are equal to the hygroscopic strain vector for a moisture uptake m = 1.

The laminate **expected strength** (load per unit width) is calculated for the 5 main independent in-plane load components: +Nx, +Ny, -Nx, -Ny and Nxy. Curing strains are taken into account in calculating the loads that cause failure, but thermal loading and moisture are ignored. The failure criteria considered for layer failure are those selected by the user (page 18, Custom criterion excluded), and thus first ply failure loads are easily computed. For the failing layers it is assumed that all stiffness components reduce to zero, except for E_{11}. The latter is in fact also reduced to zero if the local σ_{11} exceeds S_{11}. The calculations for the ultimate loads are therefore iterative procedures, progressively eliminating layers while monitoring maximum load capacity.

The resulting strength values are only indicative of the expected laminate strength and can be conservative, depending on lay-up configuration.

The Load vector is composed of **Nxx, Nyy, Nxy, Mxx, Myy, Mxy**. These are either the applied forces / moments, or the resulting reactions in the case of strain / curvature constraints.

The effective **stress** is also displayed for each load component. This is the stress that would exist in a single-material laminate of the same overall stiffness properties. The bending stress is the maximum value, at the outer surface.

The Strain vector is composed of **εxx, εyy, εxy, κxx, κyy, κxy**. These are either the resulting strains / curvatures due to environmental conditions / forces / moments, or the applied strain / curvature constraints. It is possible to select the components to be shown (Curing, Hygroscopic, Thermal, Mechanical).

The **BFS strength criterion results** section (requires the *Additional Failure Criteria* module) shows the unnotched longitudinal compressive strength (as an effective stress and as actual loads) as well as the corresponding failing layers. In addition, the effective elastic modulus E' and the fracture toughness values K_{IC} and G_{IC} are shown for completeness. Any errors or warnings are also reported. For example, the unnotched strength cannot be calculated if the loading includes bending or if it places the lay-up in longitudinal tension. As explained in the *Theory* section, the unnotched strength depends on the loading pattern. The reported value corresponds to the effective axial compressive stress that causes initial failure in the 0º layers, provided that the loading pattern is exactly that of the applied loading, i.e the relative proportions of the axial, transverse and shear mechanical loading components remain constant. Furthermore, because loading of type strain is allowed, the above loading pattern is taken to be the resulting Load Vector.

The **ABD** and Inverse ABD matrices are also shown here. They can be displayed at Unit nominal thickness as an option. This simply excludes the thickness term from the calculation and gives the equivalent matrices for a lay-up that has all layer thicknesses scaled to give a total nominal thickness of 1 in the currently selected Length units. This can be useful for:
(a) relating strain and curvature to effective stress instead of to N and M,
(b) calculating equivalent stiffness terms other than those reported by LAP, and
(c) using the ABD matrix as a stiffness matrix in other software.

Full derivation of the equations leading to the above quantities can be found in the *Theory* section, page 40. For laminates that include non-linear materials, all values are secant at the full currently applied load, using the strain at the Operating Conditions state (where mechanical load is not applied yet) as a reference point.

The window's context menu item **Copy (text)** (accessible via the right mouse button) makes it possible to copy the window contents to the Windows clipboard. This is intended for quick transfer of the underlying results to a spreadsheet, where a Paste operation neatly places numbers in individual cells. Equally, the results can be copied in this fashion to a word processing package for inclusion in a report.

Print and print preview operations are also possible via the window's context menu, but also via the top-level toolbar and menus, or via the standard key shortcuts. This printing functionality is intended only for obtaining an instant hardcopy of the results. It is not recommended for general use, as it includes no reference to the relevant data, datafile, units, date, etc. It is better to use Print Reports for consistent printing.

Relevant units: (as shown)

E, G, stress	Force / Length²
EI, GJ	Force Length
N	Force / Length
M	Force
α	1 / Temperature
β, ν	No Units
ε	Strain
κ	1 / Length
K_{IC}	Force/Length² Length^{½}
G_{IC}	Force/Length

ABD matrix (sub-matrices of 3x3):

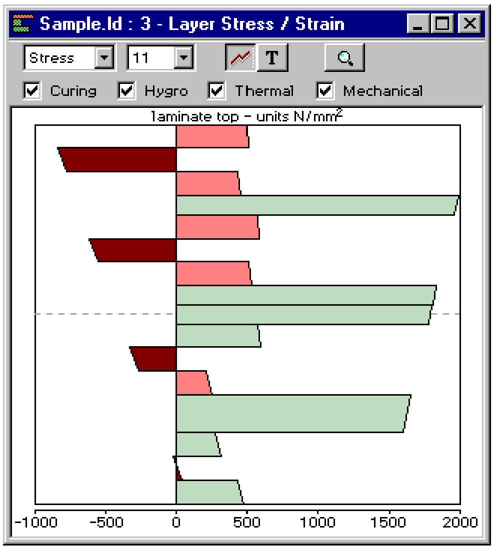
[	A	B	]	=	[	Force/Length	Force	]
[	B	D	]		[	Force	Force Length	]

Inverse ABD matrix (sub-matrices of 3x3):

[	Length/Force	1/Force	]
[	1/Force	1/(Force Length)	]

ABD matrix (for Unit thickness)	Force / Length^2
Inverse ABD matrix (for Unit thickness)	Length^2 / Force

Layer Stress and Strain



The Stress or Strain distributions through the laminate layers are plotted or tabulated by selecting **Results | Layer Stress / Strain** from the menu.

The stress or strain component to be displayed is chosen by the user, by selecting combinations of **Curing**, **Hygro**, **Thermal** and **Mechanical** components for one of the **XX**, **YY**, **XY**, **11**, **22** or **12** principal layer directions. The choice of tabular or graphical presentation is made by using the buttons T.

For graphics, LAP uses the maximum absolute value in the displayed results to scale the plot. However, custom axis limits can be defined through the button , thus making the relative comparison between the various stress or strain components much more meaningful, as well as the monitoring of load step progress better. The user-defined scale limits change automatically to reflect the current units.

Normally, layer stress or strain graphics are filled with a **light green** background. For those layers where at least one of the selected failure criteria has exceeded unity, this background colour changes to **light red**, thus indicating the presence of excessive stresses (and the use of reduced stiffness in the case of non-linear materials). Furthermore, for those layers where a non-linear stiffness component has been reduced to zero (because its corresponding stress or strain has exceeded the range of property definition), the background colour used is **dark red**. Finally, the background colour may occasionally be **white**. This is a warning that at least one of the selected failure criteria is undefined.

The window's context menu item **Copy (text)** (accessible via the right mouse button) makes it possible to copy the window contents to the Windows clipboard. This is intended for quick transfer of the underlying text results to a spreadsheet, where a Paste operation neatly places numbers in individual cells. **Print** and print preview operations are also possible via the context menu. If using a colour printer, the green and red colours will be printed as shown on the screen, otherwise they will be printed in shades of grey, as in the above graphic.

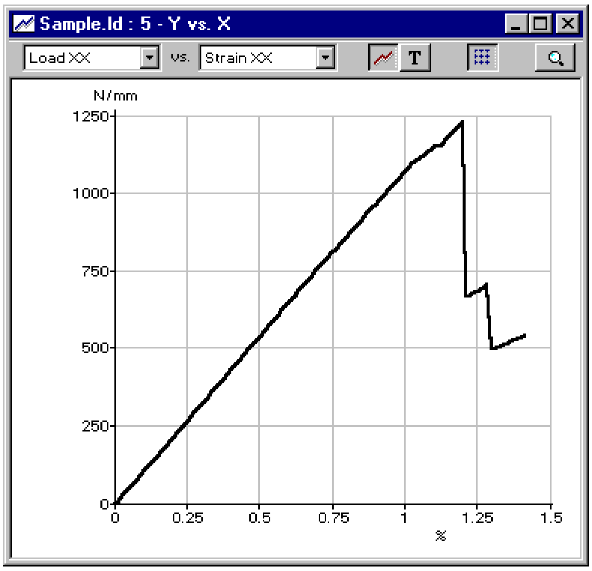
Relevant Units: (as shown)

Stress	Force / Length^2
Strain	Strain

Y vs. X Results

Pairs of various result quantities can be plotted or tabulated against one another by selecting **Results | Y vs.X** from the menu. This facility is intended for non-linear solutions.

Loads and **Moments** are based on the Load vector, so they include any possible reactions to imposed strains or curvatures.



Strains and **Curvatures** are based on the full strain vector (Curing + Hygro + Thermal + Mechanical), minus the strain vector at the Operating Conditions state (Curing + Hygro + Thermal *before* mechanical load is applied). They are therefore the strains and curvatures that would be measured during a test, carried out at the operating conditions.

All effective **stiffnesses** and hygro-thermal **coefficients** are secant at the full currently applied load, using the strain at the Operating Conditions state (where mechanical load is not applied yet) as a reference point. The brackets "(f)" stand for "flexural".

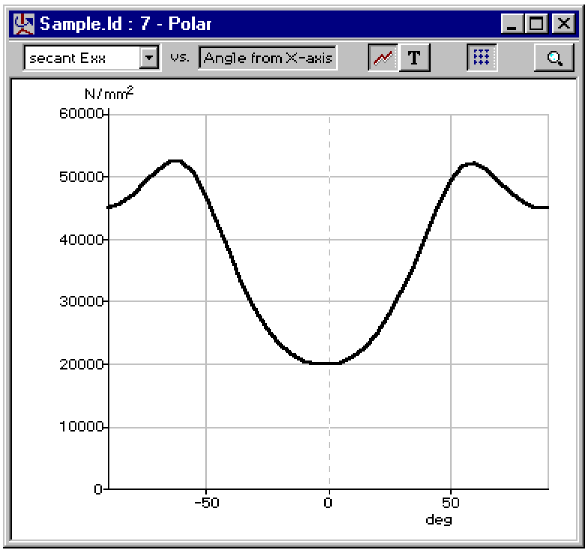
The choice of tabular or graphical presentation is made by using the buttons . A grid can also be applied to the graphics, using the button . For graphical presentations, LAP uses the extreme values in the data to scale the plot. However, custom axes limits can be defined through the button, thus making the relative comparison between different components much more meaningful, as well as the monitoring of load step progress better.

The window's context menu item **Copy (text)** (accessible via the right mouse button) makes it possible to copy the underlying text window contents to the Windows clipboard, in the usual manner. **Print** and print preview operations are also possible via the context menu.

Relevant Units: (as shown)

Loads	Force / Length
Moments	Force
Strains	Strain
Curvatures	1 / Length
E, G	Force / Length²
ν, β	No Units
EI, GJ	Force Length
α	1 / Temperature

Polar Results



Polar results of effective stiffness and hygrothermal coefficients can be plotted or tabulated against the offset angle from the laminate global x-axis, by selecting **Results** | **Polar** from the menu.

All effective stiffnesses and hygrothermal coefficients are secant at the full currently applied load, using the strain at the Operating Conditions state (where mechanical load is not applied yet) as a reference point. The brackets "(f)" stand for "flexural".

Polar results are calculated in 5° steps for an offset angle range of -90° to +90°, which in effect covers all 360°. The yy-component quantities are not included since their variation is readily deduced from the xx-components (at 90° phase difference).

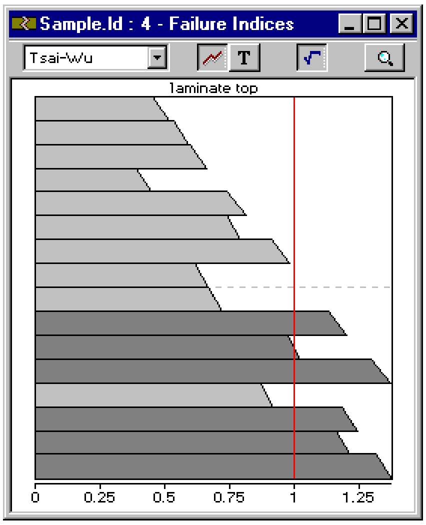
The choice of tabular or graphical presentation is made by using the buttons . A grid can also be applied to the graphics, using the button . For graphical presentations, LAP uses the extreme values in the data to scale the plot. However, custom axes limits can be defined through the button, thus making the relative comparison between different components much more meaningful, as well as the monitoring of load step progress better.

The window's context menu item **Copy (text)** copies the underlying text window contents to the Windows clipboard, in the usual manner. **Print** and print preview operations are also possible via the context menu.

Relevant Units: (as shown)

E, G	Force / $Length^2$
ν, β	No Units
α	1 / Temperature

Failure Indices



LAP calculates 6 failure criteria at the end of each load step: these are known as Tsai-Wu, Tsai-Hill, Hoffman, Maximum Stress, Maximum Strain and Custom. Details about the equations involved for each criterion can be found in the *Theory* section, page 40. All 5 of the built-in criteria are evaluated at the end of each load step, irrespective of whether they are selected for tracking local failure (page 18). The Custom one is calculated only if it is selected. Once calculated, only the selected ones are used in colour-coding layer stress/strain graphics, or in applying layer stiffness reduction and elimination (*Non-Linear* module).

Note that the Custom failure criterion is available only with the *Additional Failure Criteria* module, and requires some preparation on behalf of the user. Instructions on how to use this feature can be found on page 59.

The distribution of the failure indices, i.e. the results of the failure criteria equations, through the laminate layers can be plotted or tabulated by selecting **Results | Failure Indices** from the menu. These values are dimensionless. Each criterion can be plotted individually, or all the selected criteria can be plotted or listed together.

The choice between tabular or graphical presentation is made by using the buttons . For graphics, LAP uses optimum axis limits to scale the plot. However, custom limits can be defined through the button , thus allowing for more focused examination of the results.

Normally, layer graphics are filled with a **light grey** background. For those layers where the *displayed* failure criterion has exceeded unity, this background colour changes to a **dark grey**, thus indicating failure by that theory. The red line at the x=1 axis value is also useful in visualising this.

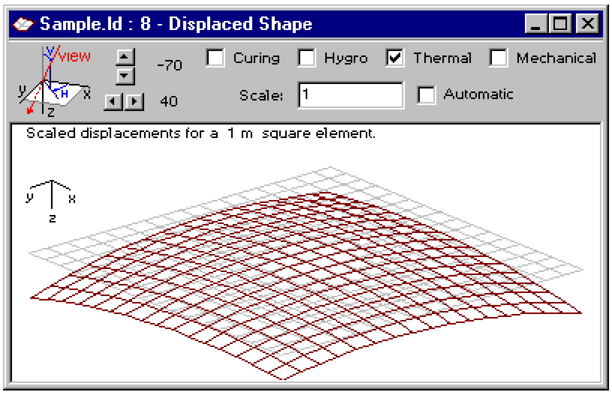
Colour is used to distinguish the various criteria when plotted together.

The button can be used to display the 3 energy based criteria (Tsai-Wu, Tsai-Hill and Hoffman) square-rooted. This helps when making comparisons with the Maximum Stress or Strain criteria, but also gives a more realistic, almost linear, comparison between applied loads and strength capacity. The display of the Custom criterion is not affected by this option.

In cases where some or all strength properties are missing, failure criteria that cannot be evaluated display a "?" instead of a value.

The window's context menu item **Copy (text)** copies the underlying text window contents to the Windows clipboard, in the usual manner. **Print** and print preview operations are also possible via the context menu. If using a colour printer, the various colours will be printed as shown on the screen, otherwise they will be printed in shades of grey. This may be confusing in the case of all the selected criteria plotted together and is not recommended.

Displaced Shape



A graphical representation of the laminate distortions in- and out-of-plane can be obtained by selecting **Results | Displaced Shape** from the menu.

The strain components used to calculate displacements are chosen by the user, by selecting combinations of **Curing**, **Hygro**, **Thermal** and **Mechanical** components.

The initial undisplaced shape is plotted in grey, whereas the displaced shape in dark red. LAP scales the displacements automatically to a magnitude suitable for viewing. However, a custom magnification factor can be specified, thus providing a feel for the actual displacements.

Displacements due to strains are calculated from:

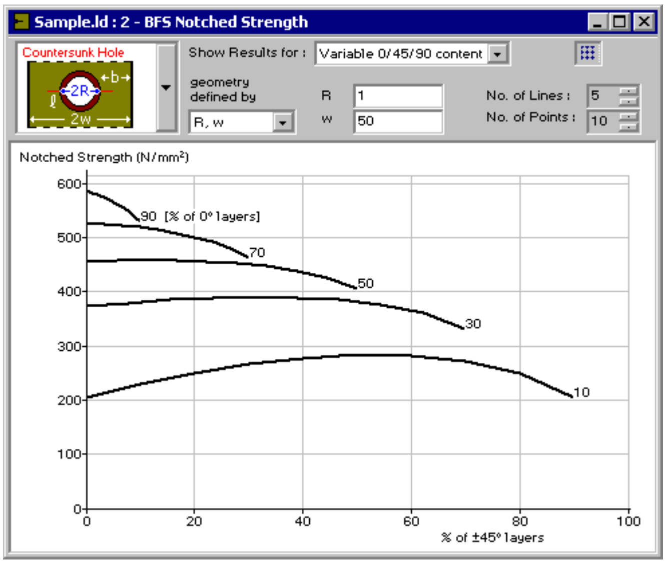
(distance from centre of square) x (strain) x (magnification factor)

For example, a +1% xx-strain with a magnification factor of 100 will show a displaced shape of twice the length of the original laminate. However, a -1% xx-strain with the same magnification factor will reduce the displaced length to zero! Increasing the magnification factor further will even make the displaced shape fold onto itself and display erroneously. Care should be taken therefore with negative strains and relatively high magnification factors.

Also note that the actual dimensions of the displayed panel are 1 length-unit square. This has a strong influence on bending displacements.

The laminate can be viewed from any direction, by suitable use of the four arrow buttons, or by clicking anywhere on the window and dragging the mouse left-right or up-down. The two view angles represent the latitude and longitude of the viewing position, as depicted in the graphic provided.

BFS Notched Strength



Calculations for the **Notched** longitudinal compressive strength for the selected lay-up and load combination can be carried out by selecting **Results | BFS Notched Strength** from the menu. The BFS (Budiansky-Fleck-Soutis) failure criterion is used in the calculations (see *Theory* section). Contrary to the presentation of other types of results that are calculated at the time of requesting a solution, the calculations for Notched strength are carried out interactively here and are particular to each open window of this type. The *Additional Failure Criteria* module is required for this feature.

Additional data are necessary to specify the notch geometry:

- The type of notch (centre notch, open

hole, etc.),
- The dimensions of the notch and laminate (notch size and laminate width, or other convenient combination),
- Other information as necessary.

The results are presented in one of three variations:
- **Single Solution**, where the notched strength is reported for a single geometry configuration.
- **Variable Geometry**, where a range is given for a chosen geometrical parameter and a series of notched strength values are computed. These are then presented in a graphical or tabular fashion.
- **Variable 0/45/90 content**, where a carpet plot is shown for the notched strength of a lay-up for which the relative proportion of 0º, ±45º and 90º layers is variable, while the geometry is constant. It is assumed that the composite is composed only of 0º, ±45º and 90º layers. The proportion of 0º layers varies between 10% and 90%, and the proportion of ±45º layers varies between 0% and (100%- 0º percentage). The remaining layers are assumed to be at 90º. The number of intermediate steps is specified by the number of Lines and Points, and the geometry is constant. The proportion of 0º layers is held constant for each line.

The notch strength results computed for carpet plots may be slightly different to those shown for the fixed (single) or variable geometry options. The reason is that hygrothermal stresses are ignored and corrections for deviations from the nominal fibre volume fraction are not applied in the case of carpet plots (to calculate the unnotched strength that is then used to calculate the notched strength). Also note that the fracture toughness is taken as constant for the production of carpet plots.

There are several restrictions placed on the notched strength analyses. The most important one is that the unnotched strength must be available, which may not be true for a variety of reasons: the lay-up must be in longitudinal compression without bending, the necessary material parameters must be defined, the lay-up must be symmetric, etc. Additional restrictions apply for carpet plots, for example hybrid lay-ups are not allowed.

Finally, warnings may be issued where necessary, e.g if there exist transverse or shear loadings, but these warnings are not shown when a graphical view is selected.

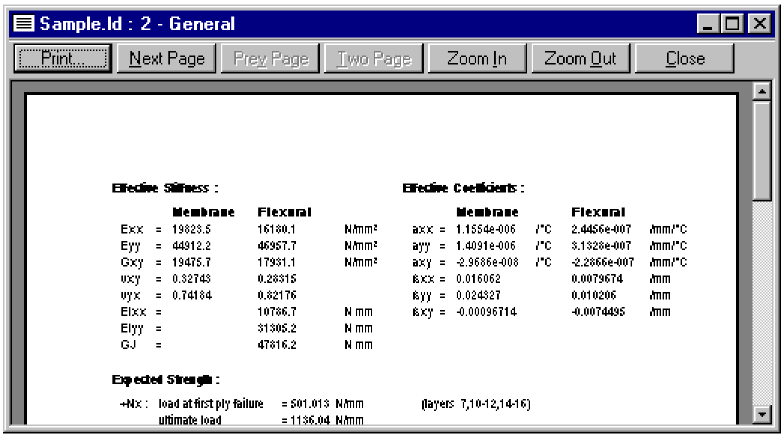
As with other Results windows, the choice of tabular or graphical presentation is made by using the buttons . A grid can also be applied to the graphics, using the button . For graphical presentations, LAP uses optimum values in the data to scale the plot. However, custom axes limits can be defined through the button.

The window's context menu item **Copy (text)** copies the underlying text window contents to the Windows clipboard, in the usual manner. **Print** and print preview operations are also possible via the context menu.

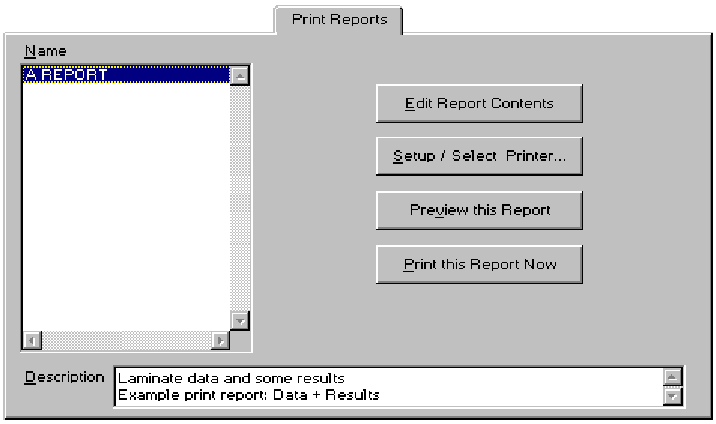
Printing procedures

General

The contents of any active window can be printed or previewed individually via the **Print** or **Print Preview** commands that appear in the **File** menu and in context menus. The same functionality is achieved with the toolbar buttons or the key shortcuts Ctrl+P and Ctrl+R. All output is generated in the currently selected units, while the default font and graphic dimensions can be defined in the general Preferences (page 18).



While this functionality is convenient and fast, for consistent printing operations it is recommended to use Print Reports instead. The **Print Reports** tab in **Data** windows is where generic print reports are defined, modified, previewed and printed. Reports print the same set of selected items for any solution, thus ensuring consistency. They also include references to the datafile name, units, failure criteria options, the current date, and can be made to print data as well as results in one operation. In this fashion, confusion as to the origin of individually printed results can be kept to a minimum.



The operations for creation, deletion, etc. of print reports are identical to those already seen for materials, lay-ups and loadings, as long as it is the Print Reports tab that is selected. Furthermore, undo and drag-and-drop copy operations are equally available for print report objects (see page 10).

The **Edit Report Contents** button displays the **Edit Print Report** window which is used to add items to the selected report, from a list of predefined items. These items are generic, in other words they do not operate on specific objects but instead they operate on the Lay-up / Loading pair selected for solution, at the time of actually requesting a print. For example, a report that includes the Loading properties can be used to print the properties of any Loading that is selected for solution. For more information, please refer to the next section.

The **Setup / Select Printer** button can be used to select a destination printer device and also to select options for it, such as paper size and orientation. The same functionality is achieved via the **File | Print Setup** menu command. This step is optional, and if omitted the default Windows printer device is used. Any selections or changes made here are applicable to all printing operations for the current program session.

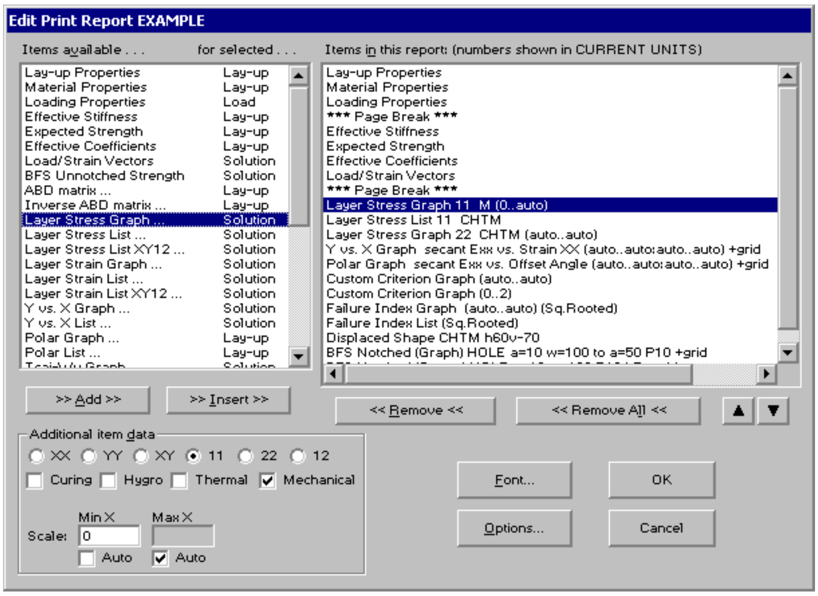
Clicking on the **Print this Report Now** button prepares a print job based on the highlighted report name and sends it to the default, or user-selected, printer. It is at this point that LAP examines the Lay-up, Loading and Units selections and uses them to fill the report items with data before printing. If results are not available, a solution is *not* carried out and all results-based items are skipped. The current units are used for all data and results, and any user-defined scales in the report are converted to these units.

The **Preview this Report** button does a similar job, but does not actually produce the hard copy. An image of the expected printed pages is prepared, for screen preview purposes only.

Editing Print Reports

The **Edit Print Report** window is where the definition of the report contents is done. The simplest way to get here is by double-clicking on the desired report name.

A number of predefined items are available which can be selected in any order, while custom options can be set for most of them.



The **>>Add>>** button adds the highlighted "available" item to the bottom of the list of items in the report being edited.

The **>>Insert>>** button inserts the highlighted "available" item immediately before an item that is already included in the report and which is highlighted.

The **<<Remove<<** button removes the highlighted item from the report.

The **<<Remove All<<** button removes all the items currently in the report.

The up-arrow and down-arrow buttons can be used to move the highlighted report item up or down in the printing order.

The **Font...** button allows for the selection of a font that will be used for all text when printing. For best results, it is recommended to work with True Type fonts only.

The **Options...** button allows for the definition of printing margins and for the specification of the size of any graphics to be printed.

The **OK** and **Cancel** buttons are used to accept or discard, respectively, all changes made to the report being edited.

All available items with names ending in ellipsis (...) may require **additional data** to be defined by the user. The necessary window controls to enter these data are visible only when such items are highlighted, and include stress / strain components, axes directions, axes scales, etc. These work in a similar way to what is available through screen-based results windows.

Where data entry is needed which is unit dependent, values should be entered in the current units. However, the hard copy will be in the units that are active at the time of requesting the print.

Print Report Items

The available items for inclusion in print reports, together with a brief explanation of each one's function, are as follows:

Lay-up Properties	Information for the lay-up selected for the solution.
Material Properties	Material data for all materials in the selected lay-up.
Loading Properties	Information for the loading selected for the solution.
Effective Stiffness	Effective stiffness properties for the selected lay-up.
Effective Strength	Effective (expected) strength properties for the selected lay-up.
Effective Coefficients	Effective hygrothermal coefficients for the selected lay-up.
Load/Strain Vectors	Load and Strain vectors. The strain vector is shown for the individual Curing, Hygro-Thermal and Mechanical components, as well as for the total.
BFS Unnotched Strength	Unnotched longitudinal compressive strength, based on the BFS (Budiansky-Fleck-Soutis) failure criterion.
ABD matrix	ABD matrix for the last solution.
Inverse ABD matrix	The Inverse of the ABD matrix for the last solution.
Layer Stress Graph	Graph of individual stress components, layer by layer.
Layer Stress List	List of stresses, layer by layer, of any of the Curing, Hygro-Thermal or Mechanical components in any one direction, as well as their sum (up to 5 columns). Right justified, fixed format.
Layer Stress List XY12	List of stresses, layer by layer, in the XX, YY, XY, 11, 22 and 12 directions (6 columns) for any combination of Curing, Hygro-Thermal or Mechanical components. Right justified, fixed format columns.
Layer Strain Graph	Graph of individual strain components, layer by layer.
Layer Strain List	List of strains, layer by layer, of any of the Curing, Hygro-Thermal or Mechanical components in any one direction, as well as their sum (up to 5 columns). Right justified, fixed format.
Layer Strains XY12	List of strains, layer by layer, in the XX, YY, XY, 11, 22 and 12 directions (6 columns) for any combination of Curing, Hygro-Thermal or Mechanical components. Right justified, fixed format columns.
Y vs. X Graph	Plot various quantities, such as Load, Moment, Strain, Curvature, Stiffness, etc. against each other. Intended for non-linear solutions.
Y vs. X List	Tabular list of above quantities.
Polar Graph	Plot of effective stiffness and hygrothermal coefficients against the offset angle, as measured from the laminate global x-axis.
Polar List	List of above, in 5º steps.
Tsai-Wu Graph	Graph of the Tsai-Wu failure criterion, layer by layer.
Tsai-Hill Graph	Graph of the Tsai-Hill failure criterion, layer by layer.
Hoffman Graph	Graph of the Hoffman failure criterion, layer by layer.
Max.Stress Graph	Graph of the Maximum Stress failure criterion, layer by layer.

Max. Strain Graph	Graph of the Maximum Strain failure criterion, layer by layer.
Custom Criterion Graph	Graph of the Custom failure criterion, layer by layer.
Failure Index Graph	Graph of all the selected failure criteria together. Use with colour printers preferably.
Failure Index List	List of all the selected failure criteria together. Right justified, fixed format columns.
Displaced Shape	Displaced 3D shape of the laminate.
BFS Notched (Single)	Notched longitudinal compressive strength, based on the BFS failure criterion. Notch geometry parameters are necessary.
BFS Notched (Graph)	Graphic variation of the notched longitudinal compressive strength for variable notch geometry.
BFS Notched (List)	Tabular presentation of the notched longitudinal compressive strength for variable notch geometry.
BFS Notched (Carpet)	Carpet plot of the notched longitudinal compressive strength for variable 0º/±45º/90º content and fixed notch geometry.
*** Page Break ***	Forces a page break, to ensure that the next item will start at the top of a new page.
All Materials	Material data for all the materials currently in the datafile.
All Lay-ups	Information for all the lay-ups currently in the datafile.
All Loadings	Information for all the loadings currently in the datafile.

The results are relevant to the last solution carried out. Furthermore, except for the laminate strength and the Y vs.X set of results, all other results (stiffness, stress, etc.) are reported at the last step that was successfully completed. Also, wherever there is the facility to examine Curing, Hygroscopic, Thermal and Mechanical components separately, it must be remembered that for non-linear analyses these components can influence one another and are strictly inseparable.

Design procedures

Laminate Design

General

During the LAP Laminate Design procedure, the software constructs symmetric laminates that satisfy a group of Loadings, as specified by the user. A number of options are user-controlled, including limiting deflections, materials that may be considered for the laminate, safety factors, fibre angles and their relative positions, etc.

The procedure is completed in several steps, during which the design process narrows down the choice of successful laminates as the user selects from candidate materials, fibre orientations and various options. The general strategy behind the design algorithm is:

1. build a symmetric laminate that satisfies in-plane (axial) loads, while keeping thickness to a minimum,
2. add layers to this laminate symmetrically, until out-of-plane (bending) loads are also satisfied.

The steps in the design procedure are briefly to:

1. set overall options, such as the individual layer thickness (see page 18),
2. select candidate materials,
3. select loadings, that include limiting deflections which in turn specify stiffness requirements,
4. select fibre orientations that will be used to build laminates which initially satisfy in-plane (axial) loads,
5. for one of the successful laminates, select fibre orientations that will be used to build thicker laminates which satisfy all loads (axial + bending).

Backward movement in this step-by-step procedure is possible. As long as data are not modified, the calculations are not repeated when moving forward again. The time necessary for completing all steps depends very much on the problem at hand. A very important parameter that governs the efficiency of the calculations is the individual layer thickness, as specified in Design Options (page 18). The software has a built-in limit of 64 layers for each half of the symmetric laminates being designed, in order to achieve a solution in finite time. Therefore, the default value of 0.125mm for layer thickness (typical of pre-preg materials) is suitable for designing laminates of up to 16mm total thickness. Another important parameter is the total number of angles to consider, as the possible ways to stack multiple instances of layers at different angles increases dramatically with the number of candidates.

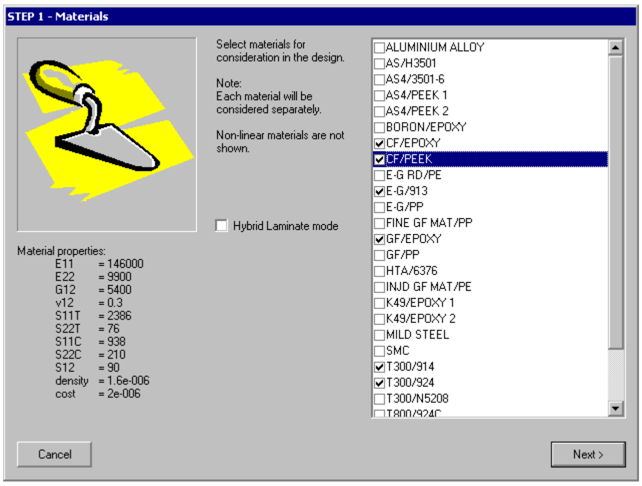
It is important to realise that the final results simply provide a good starting point for further manual modifications in order to arrive at the desired laminate. This is because a number of parameters which are not considered in the design process will undoubtedly influence the final choice. These parameters may include impact properties, consideration of edge effects, achieving non-symmetrical behaviour, etc.

In addition, in order to speed up the solution, the design algorithm ignores curing, thermal and moisture effects, fibre volume fraction corrections, and uses only the maximum stress strength criterion. The algorithm's final recommendations for the optimum laminates should be manually improved and then tested for compliance using the *Batch Solution* design procedure (page 38). Also note that for particularly large problems (many layers being used), the design algorithm attempts to reduce the solution set in a non-deterministic way, so that a solution is reached in finite time. The consequences are two-fold: firstly, the final results may differ slightly between any two runs, and secondly the absolute best stacking sequence may be missed, although a very close match is guaranteed. After all, the designer simply requires a good starting point for further manual improvement.

A useful tip for ensuring an efficient design solution is to name the loadings that are involved in the design procedure so that their order of importance is the same as their alphabetical order: the software checks candidate laminates against loading requirements by going through the list of loadings in an alphabetical order. As soon as non-compliance is found, the remaining loadings are not checked. It is therefore advantageous to place the loadings that are most important (eg. highest loads) at the top of the list, by naming them appropriately.

Prior to initiating Laminate Design, overall Design Options should be set, as described in *Preferences*,. To initiate Laminate Design, select **DESIGN: Laminates** from the **Results** menu, or press **Ctrl+F9**, or press the toolbar button . The steps involved in Laminate Design are now described in detail. Remember that context-sensitive help can be obtained at any stage by using the **F1** key.

Step 1 - Materials



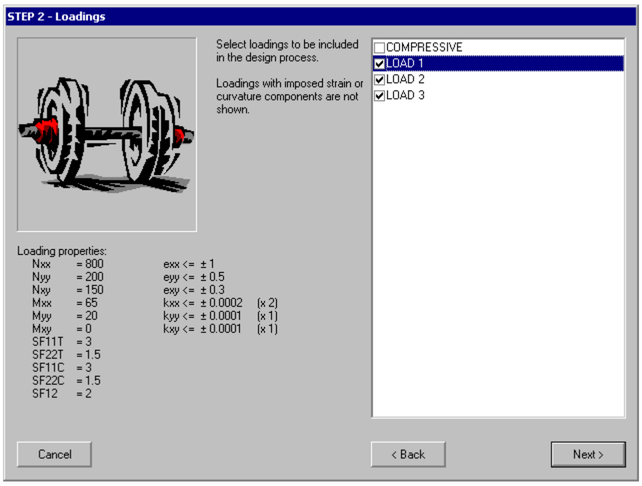
The first step is to select candidate materials for consideration in the design:

Non-linear materials are not shown here, as they cannot be used in design. To select a material, make sure that its check-box has a tick mark in it.

The relevant properties for the highlighted material are shown in the current units, for reference. These include safety factors only if the method of using safety factors is "by Material", as defined in Design Options (page 18). The safety factors will be used to effectively reduce the material strengths when these are compared to layer stresses during strength compliance calculations. If any of the displayed material properties are undefined, the material cannot be used in design.

The selected materials will be considered one at a time, forming laminates that will later be presented to the user for a second selection as to which one will be used to proceed with design. Alternatively, the **Hybrid Laminate mode** option can be selected in order to specify 2 materials that will be used together, forming hybrid laminates.

Step 2 - Loadings



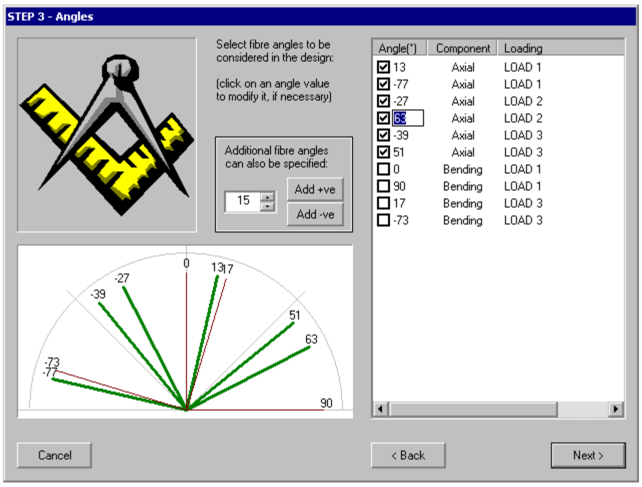
The second step consists of selecting the loadings that must each be satisfied by the final laminates:

Loadings that include strain or curvature components are not shown here, as they cannot be used in design.

The relevant properties for the highlighted loading are shown in the current units, for reference. These include safety factors only if the method of using safety factors is "by Loading", as defined in Design Options (page 18). The safety factors will be used to effectively reduce the material strengths when these are compared to layer stresses during strength compliance calculations. If any of the displayed loading properties are undefined, the loading cannot be used in design.

The allowable strain and curvature ranges effectively define the required laminate stiffness, as discussed on page 17. The design algorithm tries to build the thinnest laminate that satisfies the stiffness requirements for each loading separately, while ensuring that the resulting stresses do not cause failure.

Step 3 - Angles



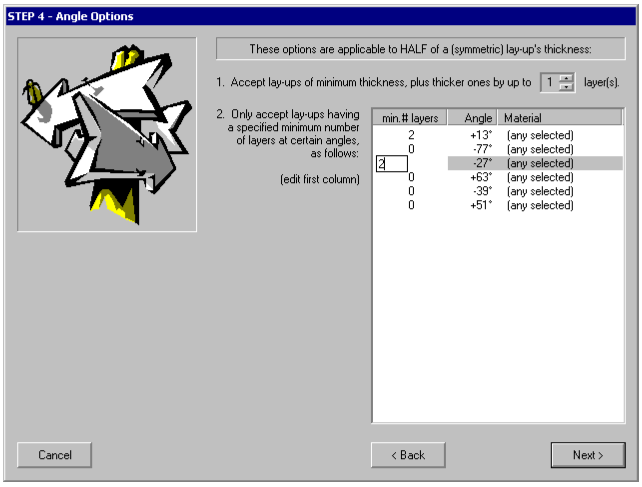
Next, the fibre angles that will be considered in the first stage of design (to satisfy axial loads) are specified:

Here, the software presents the angles that are closest to the loading paths, while account is also taken of the user-specified acceptable angle group (page 18). The angles originating from the axial components of the applied loadings are selected by default. The suggested angles may be unselected or modified, while additional angles can also be specified in the range -89° to +90°.

The interactive graphic helps visualise the relative position of the angles. This can be useful in highlighting cases where the selected angles form or approximate convenient patterns, such as 0°/90°/±45°, albeit rotated at an angle, for example +15°/-75°/+60°/-30°.

When designing in Hybrid Laminate mode, the fibre angles are specified separately for each one of the two selected materials, in two steps.

Step 4 - Angle Options



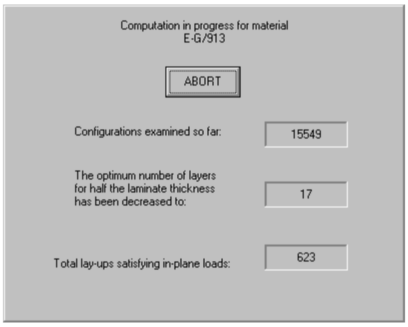
Next, certain options may be applied prior to initiating the first design calculation:

By default, the design algorithm attempts to identify suitable combinations of the selected angles that satisfy axial loadings, while the total number of layers is kept to a minimum in order to optimise weight and cost. This is done for each selected material separately, or once for the hybrid if in hybrid mode. Note that the exact stacking sequence of these angles, or layers, is irrelevant at this stage as only axial loadings are being considered.

The first option in this step instructs the software to also consider angle combinations that result in lay-ups that are thicker than the optimum ones. This is quantified by specifying the exact number of extra layers to consider (for each half of the symmetric lay-ups being built).

The second option further instructs the software to use a minimum number of layers for certain angles. For example, we may want to ensure that at least 1 layer will be used at 90° (in each half of the symmetric lay-ups being built). In hybrid mode this is specified for each selected material and angle pair.

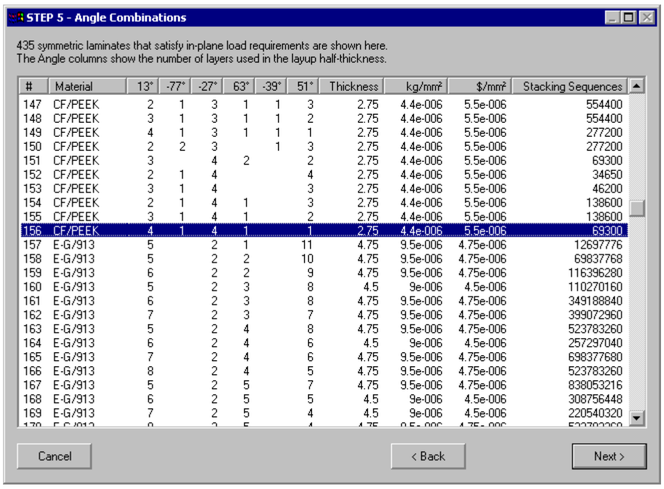
When the **Next** button is pressed, a calculation begins and the following window is displayed:



The counters are provided for convenience, so that the progress of the calculation can be monitored. If necessary, the calculation can be aborted early. Note that the maximum number of successful lay-ups is limited to 2000, so the process may be interrupted before all materials or angle combinations have been considered.

Step 5 - Angle Combinations

The successful combinations of angles are shown in step 5, for each candidate material or for the hybrid:



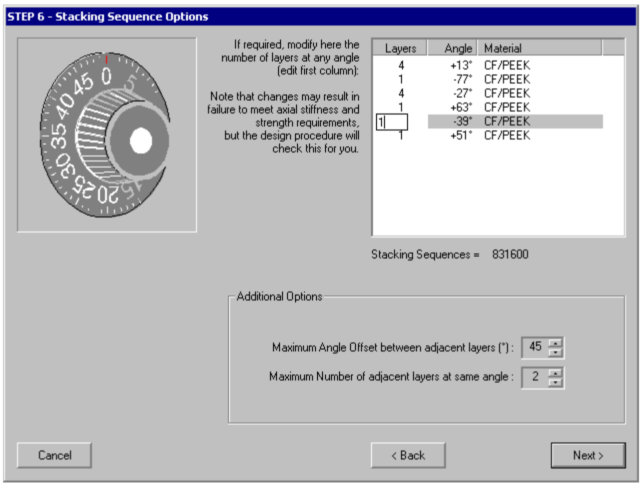
Note that this window may be resized or maximised for easier viewing. As depicted, the angle columns show the number of layers used in the lay-up half-thickness for that angle. The exact position of the layers, or stacking sequence, is irrelevant at this stage of considering axial loads only. For convenience, the total thickness, weight and cost are shown for each successful combination. The total number of possible stacking sequences for each combination of angles is also shown, so that the complexity of the next step can be assessed.

The results are not sorted in any particular way to start with. Clicking on a column header sorts the results according to that column, in alternating ascending or descending order.

One successful combination of angles must be selected in order to proceed.

Step 6 - Stacking Sequence Options

The next calculation step consists of building stacking sequences, based on the selection of angle combinations made in Step 5. These stacking sequences will be tested against bending load requirements in order to establish the ones that best satisfy those requirements. If necessary, additional steps will then follow, whereby additional layers will be added to the outside of one stacking sequence in order to fully satisfy all load requirements. Prior to analysing stacking sequences in detail, some final options may be applied:



The table in the above window presents the selection made in the previous step for angle combinations. It shows that there will be X layers at angle Y for material Z, and so on. Here, the user has a last chance to intervene so that the number of layers at any specific angle is increased or decreased, possibly based on symmetry requirements. The total number of mathematically possible stacking sequences is shown at the bottom of the table.

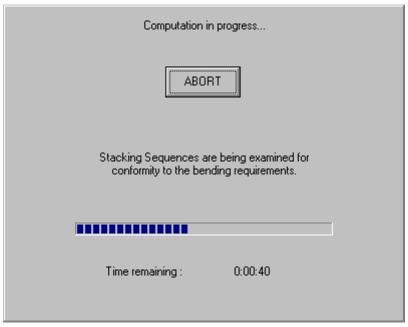
The **Additional Options** are as follows:

- The **Maximum Angle Offset between adjacent layers** ensures that the change in fibre orientation from any layer to its immediate neigbours cannot be more than the value given. This can be useful in keeping interlaminar stresses low. However, it may be physically impossible to satisfy this restriction. For example, if only 0° and 90° angles are used in building the stacking sequences, any value below 90° for this option will result in an unsuccesful search for suitable laminates, unless unidirectional material is sufficient to satisfy the applied load requirements. Although this example is a trivial case and could easily be identified by the algorithm as unacceptable, the general case is impossible to check for, and hence users must be careful when setting this option to any value below 90°.
- The **Maximum Number of adjacent layers at same angle** can be used to avoid clusters of layers at the same angle. For example, it would probably be unacceptable to place all the 0° layers on the outside of the laminate, even if that would result in good bending stiffness in that direction. The default value of 64

is equal to the maximum number of layers allowed in the laminate half-thickness and is therefore not restrictive. A small value, such as 1 or 2, would present the same potential problems as those discussed with the previous option: it may prove impossible to build a laminate that satisfies this restriction. Again, users must be careful when setting this option to a small value.

When the **Next** button is pressed, several checks are carried out. If the modified combination of angles results in failure to satisfy axial load requirements, an error message is displayed and the user is required to revise the modifications made to the number(s) of layers. Note that, if modifications are made to the number of layers at any angle, these will be lost when the **Back** button is pressed.

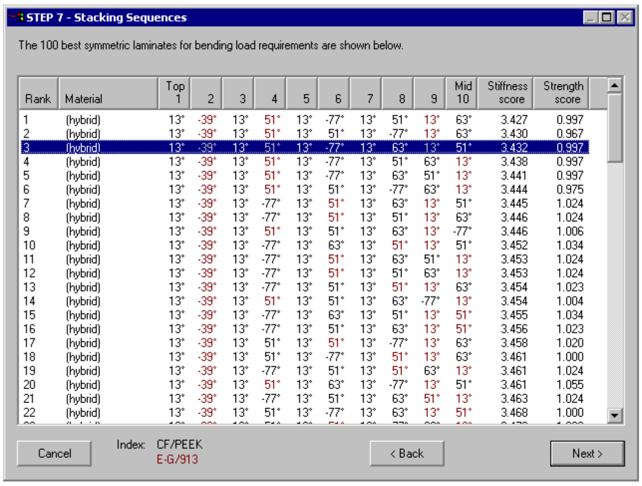
If all is well, a calculation begins when the **Next** button is pressed and the window shown opposite is displayed:



Depending on the particular problem and on the computer processor's speed, this calculation may take anything between a fraction of a second to several hours. The calculation can be aborted early, while keeping the results reached by that stage.

Step 7 - Stacking Sequences

Following a rigorous examination of the possible ways to stack the specified number of layers at different angles, the best stacking sequences are presented here:



Rank	Material	Top 1	2	3	4	5	6	7	8	9	Mid 10	Stiffness score	Strength score
1	(hybrid)	13°	-39°	13°	51°	13°	-77°	13°	51°	13°	63°	3.427	0.997
2	(hybrid)	13°	-39°	13°	51°	13°	51°	13°	-77°	13°	63°	3.430	0.967
3	(hybrid)	13°	-39°	13°	51°	13°	-77°	13°	63°	13°	51°	3.432	0.997
4	(hybrid)	13°	-39°	13°	51°	13°	-77°	13°	51°	63°	13°	3.438	0.997
5	(hybrid)	13°	-39°	13°	51°	13°	-77°	13°	63°	51°	13°	3.441	0.997
6	(hybrid)	13°	-39°	13°	51°	13°	51°	13°	-77°	63°	13°	3.444	0.975
7	(hybrid)	13°	-39°	13°	-77°	13°	51°	13°	63°	13°	51°	3.445	1.024
8	(hybrid)	13°	-39°	13°	-77°	13°	51°	13°	51°	13°	63°	3.446	1.024
9	(hybrid)	13°	-39°	13°	51°	13°	51°	13°	63°	13°	-77°	3.446	1.006
10	(hybrid)	13°	-39°	13°	-77°	13°	63°	13°	51°	13°	51°	3.452	1.034
11	(hybrid)	13°	-39°	13°	-77°	13°	51°	13°	63°	51°	13°	3.453	1.024
12	(hybrid)	13°	-39°	13°	-77°	13°	51°	13°	51°	63°	13°	3.453	1.024
13	(hybrid)	13°	-39°	13°	-77°	13°	51°	13°	51°	13°	63°	3.454	1.023
14	(hybrid)	13°	-39°	13°	51°	13°	51°	13°	63°	-77°	13°	3.454	1.004
15	(hybrid)	13°	-39°	13°	-77°	13°	63°	13°	51°	13°	51°	3.455	1.034
16	(hybrid)	13°	-39°	13°	-77°	13°	51°	13°	63°	13°	51°	3.456	1.023
17	(hybrid)	13°	-39°	13°	51°	13°	51°	13°	-77°	13°	63°	3.458	1.020
18	(hybrid)	13°	-39°	13°	51°	13°	-77°	13°	51°	13°	63°	3.461	1.000
19	(hybrid)	13°	-39°	13°	-77°	13°	51°	13°	51°	63°	13°	3.461	1.024
20	(hybrid)	13°	-39°	13°	51°	13°	63°	13°	-77°	13°	51°	3.461	1.055
21	(hybrid)	13°	-39°	13°	-77°	13°	51°	13°	63°	51°	13°	3.463	1.024
22	(hybrid)	13°	-39°	13°	51°	13°	-77°	13°	63°	13°	51°	3.468	1.000

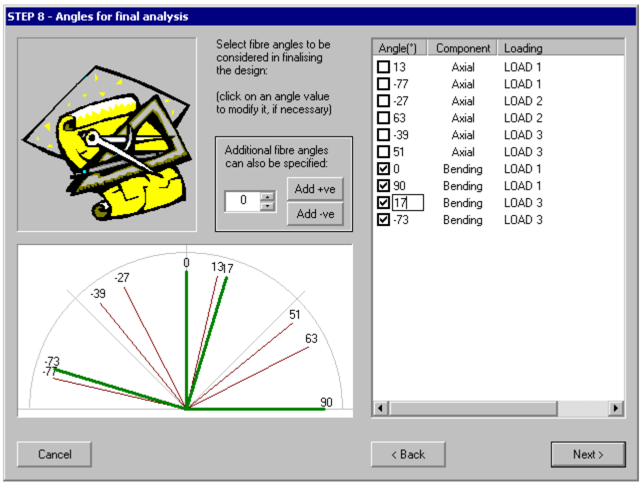
Note that this window may be resized or maximised for easier viewing. The total number of the displayed stacking sequences is limited to 100. In the case of hybrid laminates, colour is used to distinguish the usage of either material in any layer.

Two scores are calculated for each stacking sequence and shown in the above table: a stiffness score and a strength score. These reflect the suitability of a stacking sequence relative to the applied bending load requirements. The scores are formulated so that a value close to 1, but less than 1, represents the ideal solution. Values over 1 signify that a stacking sequence is not stiff or strong enough. It is possible that some, or all, of the stacking sequences built by this stage already satisfy all the load requirements. This will be reflected by both scores being lower than 1, and the design procedure will be complete.

The results are initially sorted by increasing stiffness score. Clicking on a column header (except those for layers) sorts the results according to that column, in alternating ascending or descending order.

If there are no stacking sequences with both scores below 1, it is necessary to use more layers so that stiffness and/or strength are increased. To proceed in this fashion, one stacking sequence must be selected and the **Next** button must be pressed.

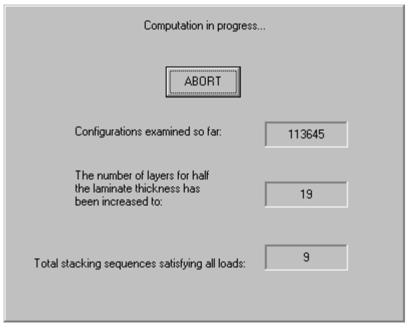
Step 8 - Angles for final analysis



The final step in the design process consists of adding layers to the outside of the stacking sequence built and selected in the previous step. As in Step 3, the fibre angles that will be used for this task are specified by the user and may be related to the main bending load paths.

As in Step 3, it is possible to unselect the angles suggested by the software, or to modify them, or to select others, or to add yet more angles for selection. Again, the interactive graphic helps visualise the characteristics of the fibre angle pattern being used.

When designing in Hybrid Laminate mode, the fibre angles are specified separately for each one of the two selected materials, in two steps.

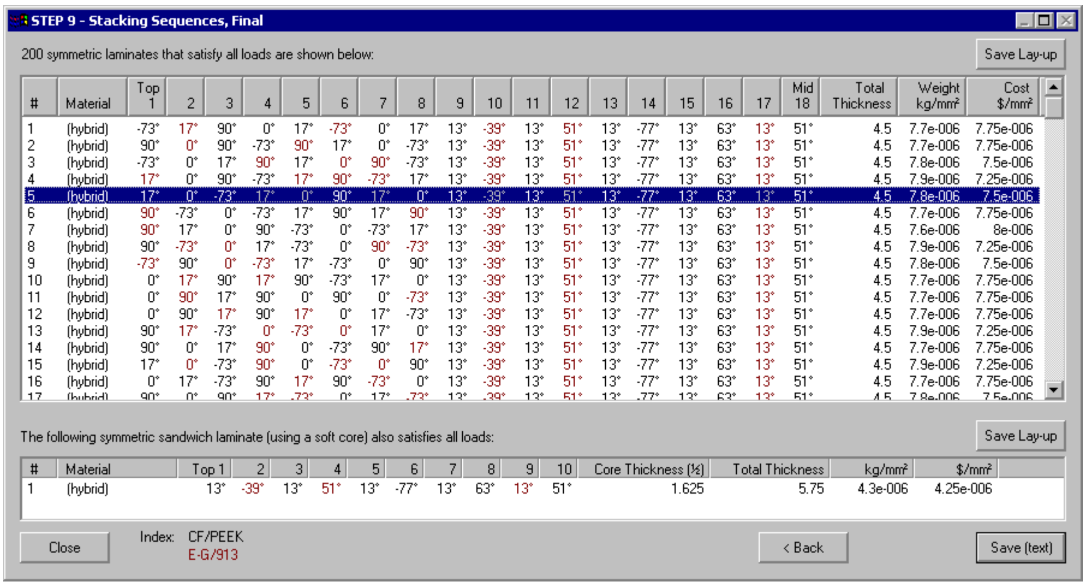


When the **Next** button is pressed, a calculation begins and the window shown opposite is displayed:

The counters are provided so that the progress of the calculation can be monitored. If necessary, the calculation can be aborted early while keeping the results calculated by that time. Note that the maximum number of successful laminates is limited to 200, so the process may be automatically interrupted before all possible solutions have been considered.

Step 9 - Final Stacking Sequences

Finally, a set of stacking sequences that satisfy all loading requirements is presented:



Note that this window may be resized or maximised for easier viewing. The total number of the displayed stacking sequences is limited to 200. In the case of hybrid laminates, colour is used to distinguish the usage of either material in any layer.

In addition to the laminates constructed by adding layers to the outside of the basic laminate selected in Step 7, the algorithm also calculates a sandwich laminate that is constructed by adding core material to the inside of that same basic laminate. The core, as used in the calculations, is a special material with essentially zero in-plane stiffness.

The total thickness, weight and cost are shown in this final table, for completeness. The results are not sorted in any particular way to start with. Clicking on a column header (except those for layers) sorts the results according to that column, in alternating ascending or descending order.

The top-right button **Save Lay-up** can be used to save the selected entry in the list as a LAP Lay-up object, for further manipulation and test. Similarly, the **Save Lay-up** button that is located just above the sandwich laminate box can be used to save the sandwich as a LAP Lay-up object. In this case, the built-in core material object in the lay-up may be replaced with a user-specified material of suitable stiffness properties.

The **Save (text)** button can be used to export the results to a tab-delimited text file, typically for safe keeping or for reading into a spreadsheet for further analysis.

Batch Solution

General

The Batch Solution design procedure is used to carry out automated stiffness and strength analyses for a number of Lay-ups, each subjected to a number of Loadings. The result of this procedure is simply a table of pass/fail flags, based on several user-defined requirements.

The purpose of the Batch Solution is to verify that one or more lay-ups, that have probably been constructed using the Laminate Design procedure (page 32), do indeed satisfy loading requirements. The Batch Solution is more rigorous than the Laminate Design algorithm, because it takes into account any curing, thermal or moisture effects, it makes corrections for any fibre volume fraction effects, it allows strain or curvature as well as force loadings, and it uses all the user-selected failure criteria.

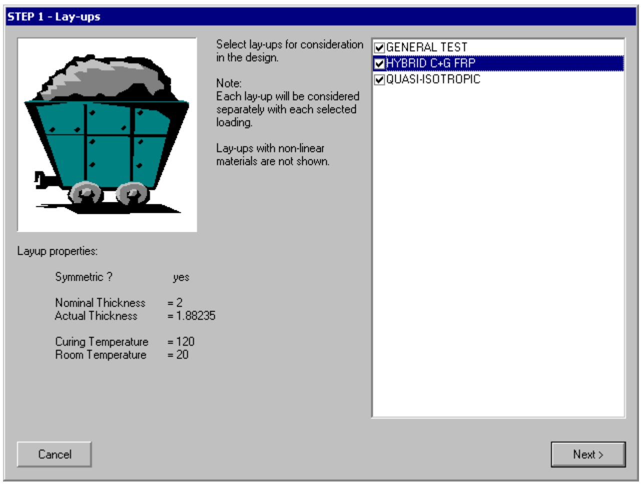
The only Design Option (page 18) applicable to the Batch Solution is the choice of using safety factors "by Material" or "by Loading".

Step 1 - Lay-ups

The first step in the Batch Solution procedure is to select the Lay-ups to be analysed:

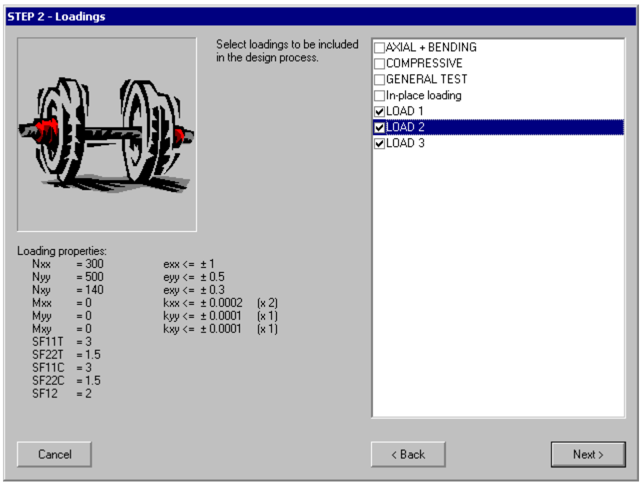
Lay-ups that include non-linear materials are not shown here, as they cannot be used in the batch analysis. To select a lay-up, make sure that its check-box has a tick mark in it.

Some properties for the highlighted lay-up are shown in the current units, for reference. When the **Next** button is pressed, a number of checks are carried out and error messages may be produced, as appropriate.



Step 2 - Loadings

The next step consists of selecting the Loadings that will be applied to each selected Lay-up:

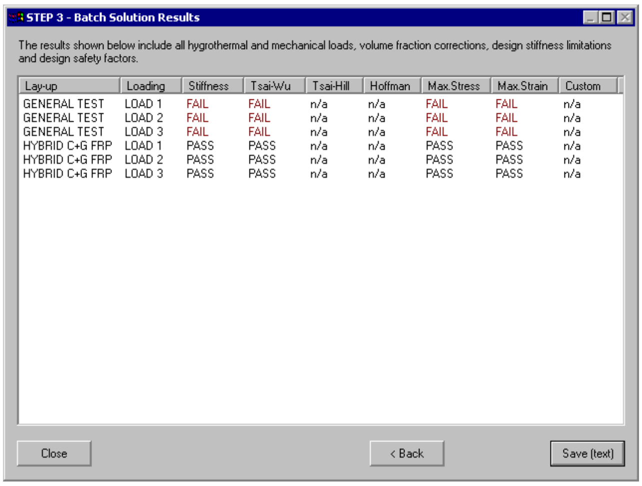


The relevant properties for the highlighted loading are shown in the current units, for reference. These include safety factors only if the method of using safety factors is "by Loading", as defined in Design Options (page 18). The safety factors will be used to effectively reduce the material strengths when these are compared to layer stresses in the pass/fail tests for strength. If any of the displayed loading properties are undefined, the loading cannot be used in design.

The allowable ranges for strains and curvatures define the pass/fail test for stiffness.

Step 3 - Batch Solution Results

Finally, the results from the multiple automated analyses are shown in a table, as in the example below:



The results for stiffness or strength compliance for each Lay-up + Loading combination are simply PASS or FAIL flags, or the text "n/a" for strength criteria that are not selected for monitoring (page 18). In addition, a question mark sign ("?") may appear where a calculation was not possible for some reason.

The **Save (text)** button can be used to export the results to a tab-delimited text file for safe keeping.

Background theory

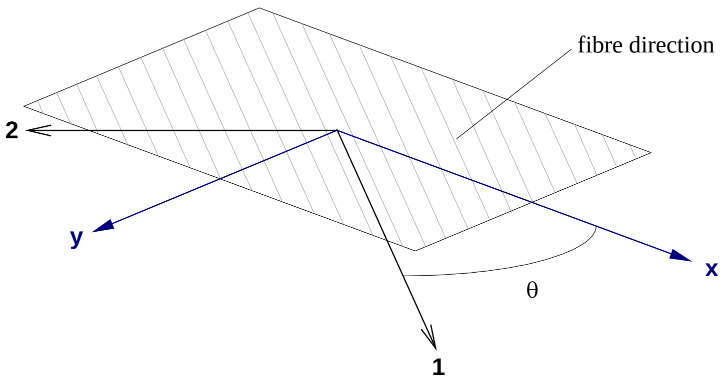
Solution Theory Basics

Stiffness analysis

The solution algorithms in LAP are based on the Classical Laminate Theory approach, briefly outlined below. This theory provides the solution for stiffness and internal stresses / strains, while strength prediction is based on several failure criteria, also discussed below.

In the simple Classical Laminate Theory approach, laminates are assumed to be of infinite length and width, i.e. edge effects are ignored. Also a plane stress state is assumed, thus ignoring certain stress components.

For each layer in the laminate, the following axis system is assumed:



where x and y are parallel to the global laminate dimensions and 1 is parallel to the fibre direction.

For each layer, the stress-strain relations in the axes defined by the fibre direction are:

$$\begin{bmatrix} \sigma_1 \\ \sigma_2 \\ \sigma_{12} \end{bmatrix} = \begin{bmatrix} Q_{11} & Q_{12} & 0 \\ Q_{12} & Q_{22} & 0 \\ 0 & 0 & Q_{33} \end{bmatrix} \begin{bmatrix} \varepsilon_1 - \alpha_1 \Delta T - \beta_1 m \\ \varepsilon_2 - \alpha_2 \Delta T - \beta_2 m \\ \varepsilon_{12} \end{bmatrix}$$

where:

$Q_{11} = E_{11}/(1-\nu_{12}\nu_{21})$

$Q_{12} = \nu_{12}E_{22}/(1-\nu_{12}\nu_{21})$

$Q_{22} = E_{22}/(1-\nu_{12}\nu_{21})$

$Q_{33} = G_{12}$

$\nu_{21} = \nu_{12}E_{22}/E_{11}$

?T is the applied temperature difference

m is the moisture content, by weight

In the axis system defined by the principal laminate directions, this becomes:

$$\begin{bmatrix} \sigma_x \\ \sigma_y \\ \sigma_{xy} \end{bmatrix} = \begin{bmatrix} \overline{Q}_{11} & \overline{Q}_{12} & \overline{Q}_{13} \\ \overline{Q}_{12} & \overline{Q}_{22} & \overline{Q}_{23} \\ \overline{Q}_{13} & \overline{Q}_{23} & \overline{Q}_{33} \end{bmatrix} \begin{bmatrix} \varepsilon_x - \alpha_x \Delta T - \beta_x m \\ \varepsilon_y - \alpha_y \Delta T - \beta_y m \\ \varepsilon_{xy} - \alpha_{xy} \Delta T - \beta_{xy} m \end{bmatrix}$$

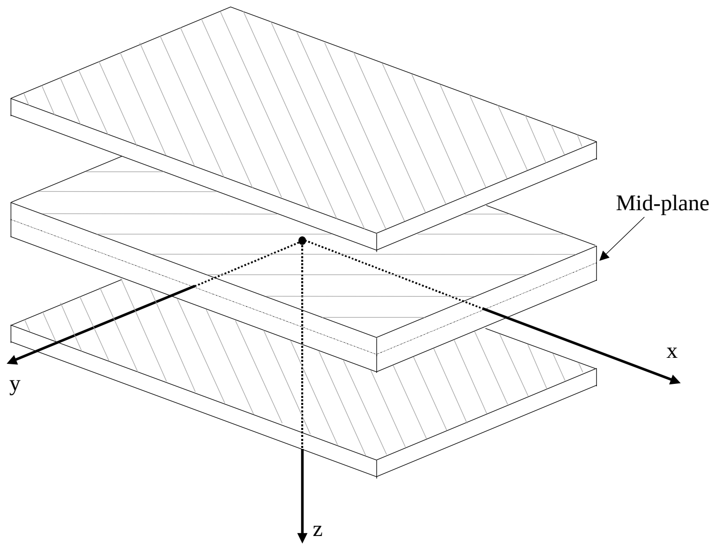
where:

$$\overline{Q}_{11} = Q_{11}\cos^4\theta + 2(Q_{12} + 2Q_{33})\sin^2\theta\cos^2\theta + Q_{22}\sin^4\theta$$
$$\overline{Q}_{12} = (Q_{11} + Q_{22} - 4Q_{33})\sin^2\theta\cos^2\theta + Q_{12}(\sin^4\theta + \cos^4\theta)$$
$$\overline{Q}_{13} = (Q_{11} - Q_{12} - 2Q_{33})\sin\theta\cos^3\theta + (Q_{12} - Q_{22} + 2Q_{33})\sin^3\theta\cos\theta$$
$$\overline{Q}_{22} = Q_{11}\sin^4\theta + 2(Q_{12} + 2Q_{33})\sin^2\theta\cos^2\theta + Q_{22}\cos^4\theta$$
$$\overline{Q}_{23} = (Q_{11} - Q_{12} - 2Q_{33})\sin^3\theta\cos\theta + (Q_{12} - Q_{22} + 2Q_{33})\sin\theta\cos^3\theta$$
$$\overline{Q}_{33} = (Q_{11} + Q_{22} - 2Q_{12} - 2Q_{33})\sin^2\theta\cos^2\theta + Q_{33}(\sin^4\theta + \cos^4\theta)$$
$$\alpha_x = \alpha_1\cos^2\theta + \alpha_2\sin^2\theta$$
$$\alpha_y = \alpha_1\sin^2\theta + \alpha_2\cos^2\theta$$
$$\alpha_{xy} = 2(\alpha_1 - \alpha_2)\sin\theta\cos\theta$$
$$\beta_x = \beta_1\cos^2\theta + \beta_2\sin^2\theta$$
$$\beta_y = \beta_1\sin^2\theta + \beta_2\cos^2\theta$$
$$\beta_{xy} = 2(\beta_1 - \beta_2)\sin\theta\cos\theta$$

and

$$\begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_{xy} \end{bmatrix} = \begin{bmatrix} \varepsilon_x^0 \\ \varepsilon_y^0 \\ \varepsilon_{xy}^0 \end{bmatrix} + z \begin{bmatrix} \kappa_x \\ \kappa_y \\ \kappa_{xy} \end{bmatrix}$$

where ε^o and κ are the mid-plane strains and curvatures respectively. z is measured from the mid-plane:



Exploded view of typical laminate

The resultant forces and moments in the laminate can be derived from:

$$N_i = \int_{-t/2}^{t/2} \sigma_i \, dz$$

$$M_i = \int_{-t/2}^{t/2} \sigma_i \, z \, dz$$

and, substituting for the stress terms, we get

$$\begin{bmatrix} N_x \\ N_y \\ N_{xy} \\ M_x \\ M_y \\ M_{xy} \end{bmatrix} = \begin{bmatrix} A_{11} & A_{12} & A_{13} & B_{11} & B_{12} & B_{13} \\ A_{12} & A_{22} & A_{23} & B_{12} & B_{22} & B_{23} \\ A_{13} & A_{23} & A_{33} & B_{13} & B_{23} & B_{33} \\ B_{11} & B_{12} & B_{13} & D_{11} & D_{12} & D_{13} \\ B_{12} & B_{22} & B_{23} & D_{12} & D_{22} & D_{23} \\ B_{13} & B_{23} & B_{33} & D_{13} & D_{23} & D_{33} \end{bmatrix} \begin{bmatrix} \varepsilon_x^0 \\ \varepsilon_y^0 \\ \varepsilon_{xy}^0 \\ \kappa_x \\ \kappa_y \\ \kappa_{xy} \end{bmatrix} - \begin{bmatrix} N_x^H \\ N_y^H \\ N_{xy}^H \\ M_x^H \\ M_y^H \\ M_{xy}^H \end{bmatrix} - \begin{bmatrix} N_x^T \\ N_y^T \\ N_{xy}^T \\ M_x^T \\ M_y^T \\ M_{xy}^T \end{bmatrix}$$

where

$$A_{ij} = \sum_{k=1}^{NL} (\overline{Q}_{ij})_k \, (z_k - z_{k-1})$$

$$B_{ij} = \frac{1}{2} \sum_{k=1}^{NL} (\overline{Q}_{ij})_k \, (z_k^2 - z_{k-1}^2)$$

$$D_{ij} = \frac{1}{3} \sum_{k=1}^{NL} (\overline{Q}_{ij})_k \, (z_k^3 - z_{k-1}^3)$$

NL = number of layers in laminate

z_k, z_{k-1} = z coordinates at layer boundaries, $z_k > z_{k-1}$

$$\begin{bmatrix} N_x^H \\ N_y^H \\ N_{xy}^H \end{bmatrix} = \int \begin{bmatrix} \overline{Q}_{11} & \overline{Q}_{12} & \overline{Q}_{13} \\ \overline{Q}_{12} & \overline{Q}_{22} & \overline{Q}_{23} \\ \overline{Q}_{13} & \overline{Q}_{23} & \overline{Q}_{33} \end{bmatrix}_k \begin{bmatrix} \beta_x \\ \beta_y \\ \beta_{xy} \end{bmatrix}_k m \, dz$$

$$\begin{bmatrix} M_x^H \\ M_y^H \\ M_{xy}^H \end{bmatrix} = \int \begin{bmatrix} \overline{Q}_{11} & \overline{Q}_{12} & \overline{Q}_{13} \\ \overline{Q}_{12} & \overline{Q}_{22} & \overline{Q}_{23} \\ \overline{Q}_{13} & \overline{Q}_{23} & \overline{Q}_{33} \end{bmatrix}_k \begin{bmatrix} \beta_x \\ \beta_y \\ \beta_{xy} \end{bmatrix}_k m \, z \, dz$$

$$\begin{bmatrix} N_x^T \\ N_y^T \\ N_{xy}^T \end{bmatrix} = \int \begin{bmatrix} \overline{Q}_{11} & \overline{Q}_{12} & \overline{Q}_{13} \\ \overline{Q}_{12} & \overline{Q}_{22} & \overline{Q}_{23} \\ \overline{Q}_{13} & \overline{Q}_{23} & \overline{Q}_{33} \end{bmatrix}_k \begin{bmatrix} \alpha_x \\ \alpha_y \\ \alpha_{xy} \end{bmatrix}_k \Delta T \, dz$$

$$\begin{bmatrix} M_x^T \\ M_y^T \\ M_{xy}^T \end{bmatrix} = \int \begin{bmatrix} \overline{Q}_{11} & \overline{Q}_{12} & \overline{Q}_{13} \\ \overline{Q}_{12} & \overline{Q}_{22} & \overline{Q}_{23} \\ \overline{Q}_{13} & \overline{Q}_{23} & \overline{Q}_{33} \end{bmatrix}_k \begin{bmatrix} \alpha_x \\ \alpha_y \\ \alpha_{xy} \end{bmatrix}_k \Delta T \, z \, dz$$

With LAP, it is assumed that the moisture uptake m is constant through the laminate and therefore not a function of z. The temperature distribution ΔT is assumed to be a linear function of z.

The Aij, Bij and Dij terms make up the so-called **ABD matrix**. If the strains and curvatures are known, then the overall laminate forces and moments can be obtained from the above equation. If, on the other hand, the forces and moments are known, then the strains and curvatures can be obtained by inversion of the ABD matrix. If a combination of forces and strains are known, the solution is obtained by partitioning and solving a set of two equations. Finally, having solved for all the mid-plane strains and curvatures, the layer stresses can be calculated.

During non-linear solution, stresses are obtained at the end of each load step using the stiffness values which are calculated based on the stresses / strains at the end of the previous step. Within each load step, an **iterative** process ensures that internal stresses are always in balance with the applied loads. If this balance cannot be achieved, it means that the current laminate stiffness is unable to withstand the desired applied loads and an appropriate message is produced.

Non-linear local layer stiffness is calculated by interpolation from the user-specified piecewise functions of stiffness (as defined in terms of stress or strain). In the case of definition by stress, the local layer stresses are calculated with contributions from all types of loading and are then used for the interpolation. In the case of definition by strain, the local layer mechanical strains are used for the interpolation.

The **Effective Stiffness** properties are calculated from the Inverse of the ABD matrix and are therefore the secant values in the global laminate axes at the latest stress state. For example, Exx = 1 / ($InvABD_{11}$) / (Nominal Laminate Thickness). The **Effective Coefficients** are calculated directly from the Curing and Moisture strain vectors, by assuming unit "load". The detailed example further below can help clarify these calculations.

Adjustments for **fibre volume fraction** are simple but important. It is assumed that, due to manufacturing imperfections, the amount of resin in the finished composite is not always as expected, but the amount of fibres is fixed. This leads to an actual layer thickness that is slightly different to the nominal one, hence the position of the fibres through the laminate is affected, thus influencing its bending properties. The actual layer thickness is calculated from:

$$t_{actual} = t_{nominal}\ Vf_{nominal} / Vf_{actual}$$

where Vf is the fibre volume fraction.

Actual layer stiffness is also affected somewhat, further contributing to the effect. The resin's Young's modulus and Poisson's ratio, as well as the nominal and actual fibre volume fractions, are used in the corrective calculations which are adequate for adjustments to small variations in the fibre volume fraction.

There are two methods available for corrections: the rule of mixtures method and the Halpin-Tsai method.

The rule of mixtures states that:

$$P_c = P_f\ Vf + P_m\ (1\text{-}Vf)$$

where subscripts c,f,m refer to the composite, fibre and matrix respectively, while P can be replaced by E_{11}, ν_{12}, ($1/E_{22}$), or ($1/G_{12}$).

Suitable manipulation of the above leads to an expression for the adjusted properties:

$$P'_c = P_c\ Vf_{act}/Vf_{nom} + P_m\ (1 - Vf_{act}/Vf_{nom})$$

The Halpin-Tsai equations for E_{11} and ν_{12} are identical to those by the rule of mixtures. For E_{22} and G_{12} however, they are:

$$E_{22c} = E_m\ (1 + 2\ \eta\ Vf) / (1 - \eta\ Vf) \qquad \text{where } \eta = (E_f/E_m - 1) / (E_f/E_m + 2)$$

$$G_{12c} = G_m\ (1 + \eta\ Vf) / (1 - \eta\ Vf) \qquad \text{where } \eta = (G_f/G_m - 1) / (G_f/G_m + 1)$$

Suitable manipulation of the above leads to the adjusted:

$$E'_{22c} = E_m\ (E_{22c} + 2\ E_m + 2\ (E_{22c} - E_m)\ Vf_{act}/Vf_{nom}) / (E_{22c} + 2\ E_m - (E_{22c} - E_m)\ Vf_{act}/Vf_{nom})$$

$$\text{and } G'_{12c} = G_m\ (G_{12c} + G_m + (G_{12c} - G_m)\ Vf_{act}/Vf_{nom}) / (G_{12c} + G_m - (G_{12c} - G_m)\ Vf_{act}/Vf_{nom})$$

Evidently, the above corrections work properly only if the supplied stiffness data correspond to the *nominal* fibre volume fraction.

All internal calculations are carried out for the actual thickness and stiffness properties. Stress results, however, are reported at nominal thickness, for convenience.

A solution **example** with detailed calculations is included on page 50, showing how most of the results are calculated by LAP. It should be consulted if a better understanding of the solution procedures is required.

Strength analysis

Provided the necessary unidirectional strength components are specified, LAP calculates the Tsai-Wu, Tsai-Hill, Hoffman, Max.Stress and Max.Strain failure criteria equations at the end of each load step (the Custom criterion is discussed on page 59). The equations are:

Tsai-Wu:

$$F_1\,\sigma_1 + F_2\,\sigma_2 + F_{11}\,\sigma_1^2 + F_{22}\,\sigma_2^2 + F_{66}\,\sigma_{12}^2 + 2\,F_{12}\,\sigma_1\,\sigma_2 = 1$$

where

$$F_1 = \frac{1}{S_{1T}} - \frac{1}{S_{1C}} \qquad F_{11} = \frac{1}{S_{1T} S_{1C}}$$

$$F_2 = \frac{1}{S_{2T}} - \frac{1}{S_{2C}} \qquad F_{22} = \frac{1}{S_{2T} S_{2C}}$$

$$F_{66} = \frac{1}{S_{12}^2} \qquad F_{12} = -\sqrt{F_{11}\, F_{22}}/2 \text{ , or user - defined}$$

Tsai-Hill:

$$\frac{\sigma_1^2 - \sigma_1 \sigma_2}{S_1^2} + \frac{\sigma_2^2}{S_2^2} + \frac{\sigma_{12}^2}{S_{12}^2} = 1$$

The strengths S1 and S2 above are either the tensile or the compressive values, depending on the sign of the respective stresses.

Hoffman:

$$F_1\, \sigma_1 \;+\; F_2\, \sigma_2 \;+\; F_{11}\, \sigma_1^2 \;+\; F_{22}\, \sigma_2^2 \;+\; F_{66}\, \sigma_{12}^2 \;-\; F_{11}\, \sigma_1\, \sigma_2 \;=\; 1$$

(where the F terms are as above)

Maximum Stress:

$$S_{1C} < \sigma_1 < S_{1T} \quad \text{and} \quad S_{2C} < \sigma_2 < S_{2T} \quad \text{and} \quad |\sigma_{12}| < S_{12}$$

(the maximum of the 5 stress/strength ratios is displayed by LAP)

Maximum Strain:

$$\varepsilon_{u1C} < \varepsilon_1 < \varepsilon_{u1T} \quad \text{and} \quad \varepsilon_{u2C} < \varepsilon_2 < \varepsilon_{u2T} \quad \text{and} \quad |\varepsilon_{12}| < \varepsilon_{u12}$$

(the maximum of the 5 strain/ult.strain ratios is displayed by LAP)

It must be noted that for the Tsai-Wu, Tsai-Hill and Hoffman results the **square root** of the above equations can be displayed as an option, so that a more realistic, almost linear, comparison can be made between applied loads and strength capacity.

During a solution all 5 failure criteria are calculated, but only the *selected* failure criteria are monitored for detecting failure (page 18), including the Custom one if selected. "Failure" by any criterion does not automatically stop the solution or produce an error message. When working with a lay-up that contains only linear materials, LAP applies the selected loading to the selected lay-up in one step. If the resulting stresses cause failure (according to any one of the selected failure criteria), LAP simply reports the fact that some layers have failed, by using a light red background colour in graphical stress/strain results. No iterative calculations are carried out to reduce the stiffness of the failed layers. On the other hand, if the selected lay-up contains at least one material with non-linear properties, then the loading is applied in the designated number of steps, so that stiffness properties can vary and failed layers be "eliminated" progressively. The stiffness reduction data for non-linear material properties are used to model degradation in stiffness upon layer failure. This is applicable when any one of the selected failure criteria locally exceeds 1 at the end of a load step. In this case, layer stiffness properties are reduced by the specified amounts before additional load is applied. Note, however, that if a local stress or strain component falls outside the range used to define a stiffness property, then that property is reduced to zero.

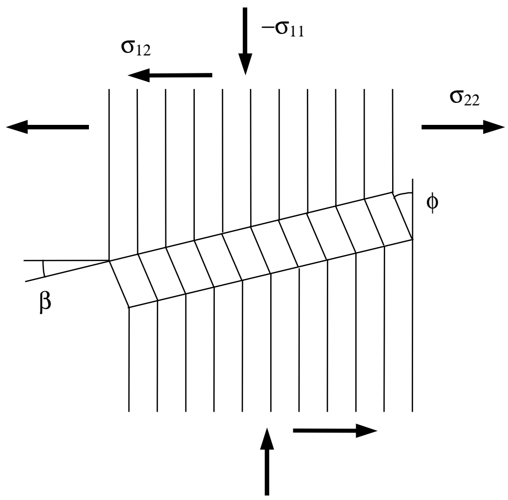
The **Expected Strength** is based on a quick iterative process, carried out before any application of loads. For non-linear materials, the stiffness at zero mechanical load is used, irrespective of other data. Curing strains are taken into account in calculating the loads that cause failure, while the failure criteria considered for layer failure are those selected by the user (page 18), but excluding the Custom criterion. First ply failure loads are easily computed. For the failing layers it is assumed that all stiffness components reduce to zero, except for E_{11}. The latter is in fact also reduced to zero if the local σ_{11} exceeds S_{11}. The calculations for the ultimate loads are therefore iterative procedures, progressively eliminating layers while monitoring maximum load capacity. The resulting strength values are only indicative of the expected laminate strength and can be conservative, depending on lay-up configuration.

The BFS failure criterion

The BFS (Budiansky-Fleck-Soutis) failure criterion is used to calculate the unnotched and notched longitudinal compressive strength of the selected lay-up, subjected to the selected loading. The present section discusses the theoretical background behind the calculations, but full details should be sought in the relevant literature [3-10]. The discussion does not apply to non-linear materials, for which the unnotched compressive strength is not calculated.

The **unnotched compressive strength** can be predicted based on either a micromechanics model or using strength data for each 0º layer.

The micromechanics model is based on the Budiansky-Fleck compressive failure analysis [7]. This criterion assumes that the unnotched compressive strength of a layer is governed by imperfection-sensitive plastic microbuckling with the imperfection in the form of fibre misalignment, as shown opposite:



For the case where the composite displays a rigid-perfectly plastic in-plane response, we have, at failure [6]:

$$|\sigma_{11}| = \frac{\alpha k - \sigma_{12} - \sigma_{22}\tan\beta}{\phi}$$

where:

k is the matrix shear strength,

ϕ is the fibre waviness (or imperfection),

β is the angle of propagation of the microbuckle,

α^2 is equal to $(1+R^2\tan^2\beta)$, and R is taken as 1.5 [9].

Assuming that the applied mechanical (M) and hygrothermal residual (R) stress components can be safely separated, we have:

$$-F\sigma_{M11} - \sigma_{R11} = \frac{\alpha k - F\sigma_{M12} - \sigma_{R12} - F\sigma_{M22}\tan\beta - \sigma_{R22}\tan\beta}{\phi}$$

where F is the factor by which the applied mechanical load (all components) must be scaled in order to just cause failure. This factor is calculated for all 0º layers in the lay-up and the minimum value is the one causing failure for the entire lay-up, assuming a conservative first ply failure mode.

The unnotched longitudinal compressive strength is then:

$$S_{1UC} = \frac{F\,|Nx|}{(\text{Layup Thickness})}$$

In practice, instead of Nx (the applied load in the X direction), the first element of the resulting Load Vector is used, because the applied load may be of type strain with LAP. Furthermore, corrections for deviations from the nominal fibre volume fraction are applied in the usual fashion when calculating the layer stresses, as discussed earlier.

Please note that the effect of transverse stresses has not been verified by experimental research, and this part of the model should be used with caution. Also note that modes other than plastic microbuckling may occur, such as elastic microbuckling, splitting, fibre crushing or matrix failure. A check on the elastic microbuckling limit is carried out by the software, but users should be careful in checking that these other modes do not occur.

In an alternative formulation, the unnotched strength can be calculated using strength data for each 0º layer. According to this model, as suggested by Soutis and Edge [6], failure occurs when:

$$\frac{-\sigma_{11}}{S_{1C}} + \frac{\sigma_{12}}{S_{12}} \geq 1$$

As above, the layer stress can be separated into mechanical and non-mechanical components:

$$\frac{-F\sigma_{M11} - \sigma_{R11}}{S_{1C}} + \frac{F\sigma_{M12} + \sigma_{R12}}{S_{12}} = 1$$

where F is again the factor by which the applied mechanical load must be scaled in order to just cause failure. The procedure for the calculation of the lay-up's unnotched strength is then as above.

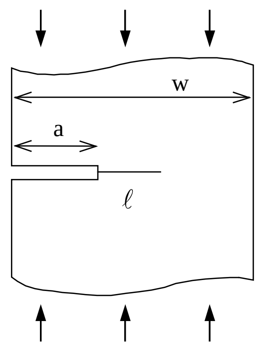
The **notched compressive strength** is calculated based on the Fleck-Soutis model, which assumes that a microbuckle and associated delamination damage grow from the edge of a sharp notch or hole. The resistance to damage can be modelled using the unnotched strength and a compressive "fracture toughness" of the laminate.

The solution method involves crack bridging models, the description of which is beyond the scope of the present documentation. Full details can be found in the associated literature [10,3], while some simple guidelines are given in this section.

The strategy is to concentrate all inelastic deformation associated with microbuckling, plasticity, cracking etc. within a crack and to assume some form of traction-displacement bridging law across the crack faces.

Considering the geometry shown below, the *net* stress intensity factor Km at the tip of the microbuckle of length ℓ, as well as the net displacement $u(x)$ along the microbuckle (half the relative displacement of the two faces) are given by:

$$Km = K\sigma + Kr \quad = 0$$
$$u(x) = u_{\sigma}(x) + u_{r}(x)$$



where:

$K\sigma$	is the stress intensity factor due to the distribution of normal compressive stresses $\sigma(x)$ along the microbuckle,
Kr	is the stress intensity factor due to the remote stress,
$u_{\sigma}(x)$	is the crack closing displacement due to $\sigma(x)$, and
$u_r(x)$	is the crack closing displacement due to the remote stress.

$K\sigma$ is given by:

$$K\sigma = \int_0^{\ell} \sigma(x)\ m(\ell, x)\ dx$$

where the weight function $m(\ell,x)$ can be conveniently found from the point load solution given by published results for a crack emanating from a central hole or for a single edge or centre notch, which are the three geometries covered by the present software.

The compressive stress $\sigma(x)$ is related to the crack closing displacement $u_{\sigma}(x)$ by a functional relationship, assumed to be linear:

$$\frac{\sigma}{S_{1UC}} = \begin{cases} 1 - \dfrac{u}{u_c} & \text{for } 0 \leq u \leq u_c \\ 0 & \text{for } u > u_c \end{cases}$$

where u_c is a critical microbuckle overlap displacement and is found from:

$$G_{IC} = S_{1UC} u_c$$

In addition, the fracture energy G_{IC} is related to K_{IC} by:

$$G_{IC} = K_{IC}^2 / E'$$

where E' is the orthotropic equivalent elastic modulus, defined by:

$$\frac{1}{E'} = \left(\frac{1}{2E_{xx}E_{yy}} \right)^{1/2} \left[\left(\frac{E_{yy}}{E_{xx}} \right)^{1/2} - \nu_{yx} + \frac{E_{yy}}{2G_{xy}} \right]^{1/2}$$

The weight function method is also used for Kr and $ur(x)$, but the stress along the microbuckle is then that along the crack line for a configuration that contains no crack.

Finally, a numerical method is used to solve for the remote stress. The cohesive zone is divided into N elements of equal length and the integrals are calculated numerically at the node points. Furthermore, the use of predetermined look-up tables simplifies the process and increases the solution speed.

There are several points to note:

- The weight functions used are for isotropic materials, but research has shown that errors in using them for orthotropic materials are not too large,
- The bridging analysis employed here does not strictly apply when transverse or shear loads are present, however an estimate of the failure load under these conditions can be made by using the unnotched strength obtained with the presence of the off-axis loads,
- For countersunk and filled holes, a simple knockdown or strengthening factor of 0.85 and 1.21 respectively is applied [6], an approximate approach that must be used with caution,
- The equivalent hole can be used to model post-impact compressive strength, as described by Soutis and Curtis [5].

Prediction of Toughness by the Rule of Mixtures

A Rule of Mixtures formulation can help predict the toughness of a lay-up, from existing experimental data on lay-ups that consist of the same material and layer orientations. The general approach is to calculate the contribution to the overall toughness from each layer orientation separately (best fit). These values are then used to sum up the layer contributions for the lay-up under consideration (rule of mixtures prediction).

Currently, lay-ups with layers in 3 orientations are supported by the software. These 3 orientations are shown as 0º, ±45º and 90º, but they could really be any 3 angles, provided they are consistent in all lay-ups involved. The experimental results used for the best fit must be unique and not linearly related. In the best fit stage, the contribution from each layer orientation is calculated, say as G_0, G_{45} and G_{90} if the rule is applied to G. Then, for the lay-up considered:

$$G = G_0 V_0 + G_{45} V_{45} + G_{90} V_{90}$$

where V_0, V_{45} and V_{90} are the volume fractions for the 3 layer orientations.

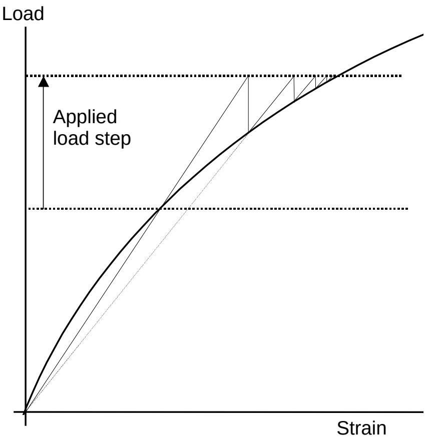
It may be worth noting that a rule of mixtures based on the strain energy release rate G might be expected to give more accurate predictions than one based on K, since we are summing up the energy release rate in a number of different plies.

Non-linear solution procedures

The procedure followed by LAP for a laminate that contains materials with non-linear stiffness properties is iterative. Load steps are applied using layer stiffness derived from the previously calculated local stresses or strains. Hygrothermal stresses are adjusted at each load step.

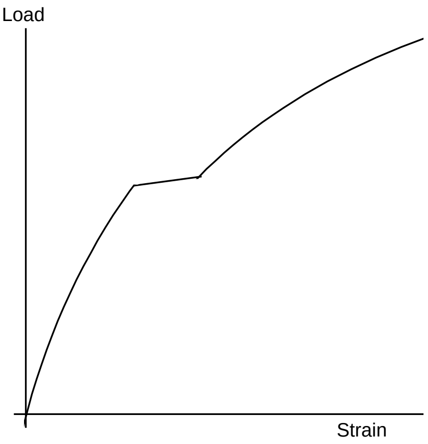
The iterative part of the solution can be illustrated with the figure on the right. Here, the curve is supposed to be the true response. Using the secant stiffness at the start of the load step, we obtain the first result for the final strain. Using the updated stiffness (based on this strain) we obtain a new result, and so on until the final strain converges to the required value.

It must be noted that the Load and Strain are not scalar values, they are vectors. This makes the mathematics considerably more complicated because convergence must occur for all elements of the Strain vector together.

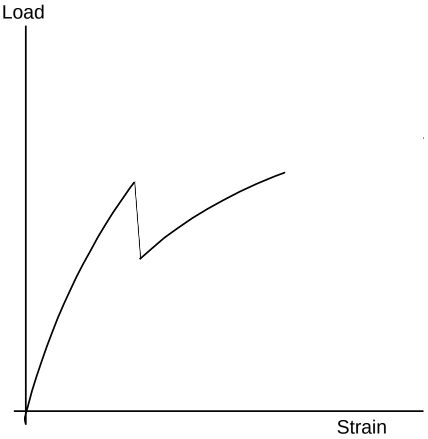


It is possible that the laminate cannot withstand the applied load step. Then the program will report a "catastrophic failure" and the maximum load will be the load at the last completed step.

It is also possible that some of the layers will fail during the iterative process within the load step, in which case a kink will appear in the Load-Strain curve, as shown opposite (for a more realistic, small, step).



If the loading is strain-based, layer failure will be shown by a sharp drop in the Load-Strain curve, as opposite.



There are some points to note regarding non-linear solutions:

- Let us assume that there is an imposed strain loading of ε_x=1%, applied at the "As Cured" state. The program will work out the curing and hygrothermal strains and apply a mechanical strain load ε_x = 0.01 - (hygrothermal strain xx), so that the final (hygro + thermal + mechanical) strain xx is equal to 1%. However, because the hygrothermal strains will be redistributed during the course of the loading due to stiffness changes, the final sum (H+T+M) will not be equal to 1%. Hence the ε_x value shown in the General Results window will be slightly different to what expected.
- For the same reason, the final ε_x shown in the Y vs.X list windows will not be exactly equal to what is expected. As already mentioned, this value is equal to the current (C+H+T+M) strain minus the "Initial" strain, which is (C+H+T) prior to the application of any mechanical loading.
- When a stiffness property is indeterminate because it falls outside its defined range, it is not actually set to zero but instead to a small number that is effectively zero. For this reason, failed layers display small stresses along failed directions, instead of zero.

- It is possible for the solution to diverge within a step. This will normally result in the message *Maximum number of iterations exceeded …*, but it can also result in layer failure as the stresses diverge away from their true values. This is due to a combination of factors that cannot always be handled by the solver. For example, let us consider a 0º/90º laminate in tension, with a material for which E_{11} increases with stress and E_{22} decreases with stress. Depending on the magnitude of the non-linearity, this can cause the solution to oscillate between two strain vectors that are not "close" enough for convergence. If you experience slow solutions, oscillating layer stresses, or results that are fairly dependent on load step size, then the problem is ill-conditioned. This is usually noticeable only close to failure stresses or at points of sudden change in stiffness properties.

Analysis Tips

This section discusses some useful tips for carrying out laminate analyses with LAP.

When specifying **loading conditions**, ensure that the strain / curvature constraints match the conditions seen by the laminate under investigation. For example. shear or twist may be suppressed in the real-life test or model because of supports, and this must be included in the analysis. Furthermore, the hard (displacement controlled) or soft (load controlled) nature of the real-life loading must also be modelled correctly by specifying strain or force data.

For non-linear solutions that involve **bending**, each physical layer can be modelled with more than one layers to achieve a better accuracy in the through-thickness stress and strain calculations if, for example, effects such as neutral axis shift are important. This can also help failure calculations because the failure criteria are applied to the mid-plane of each layer in order to determine failure.

Similarly, marginally different **material definitions** can be used to model the same material in a laminate but in different layers if, for example, it is felt that a material behaves differently when placed on the outside of a laminate (different moisture content, different fibre volume fraction, more initial cracks and therefore lower strength, etc.) or at different orientation (e.g. different strength properties when at 0º, 45º or 90º).

The loading order is always assumed to be Curing, then Hygroscopic, then Thermal, then Mechanical. Since mechanical loading normally produces the highest stresses, all **stiffness property data** should be specified at the operating environment conditions. During mechanical load stepping, the non-mechanical environmental strains and stresses are constantly re-calculated to account for the updated stiffness which can result in redistribution of the initial stresses. The results for stresses and strains displayed in the Results windows correspond to the latest load increment. Therefore, the various components that can be examined separately must be viewed with caution because in non-linear analyses each component can be influenced by the other ones and hence cannot be viewed on its own. For example, the curing stresses reported at the end of a non-linear solution that involves mechanical loading are not the same as the curing stresses at zero load. Similarly, the mechanical stresses at the end of that analysis would be different in the absence of the initial curing stresses.

Increasing the number of **load steps** to improve on accuracy will have a small effect on the final stiffness and strain results. This is because of the iterative procedure within each step that ensures balance between applied loads and internal stresses. On the other hand, accuracy will improve on the drawing of non-linear XY graphs and also on pin-pointing failure stresses and strains.

Results following a solution that was user-**aborted** or that produced the message *Catastrophic failure…* must be treated with caution, because the analysis was literally interrupted half-way and the desired loading does not match the reported stresses and strains.

When **printing results**, make sure that you also print the material, lay-up and loading data that produced these results, to avoid confusion when referring to the printout. Print reports can be created so as to produce the stress and strain components that are most important in your work.

If you plan to use the same material, lay-up or loading properties in more than one datafile, use the available drag-and-drop copy functionality, and try to assign one datafile that will be used as a **master copy** for changing such data and then updating all other files. This will ensure that data is duplicated consistently.

Detailed Calculations

As an example, let us consider a hypothetical unbalanced hybrid laminate under a combination of environmental and mechanical loading. This example is intended to cover a very general case that addresses all the calculations involved in a linear LAP solution. Most of the relevant equations have been described earlier.

Basic Units : mm N deg /ºC -

Material data

Material name	E-G/913	XAS/913
E_{11} (N/mm^2)	45600	144300
E_{22} (N/mm^2)	10730	11040
G_{12} (N/mm^2)	5140	5790
ν_{12}	0.274	0.313
α_{11} (/ºC)	7E-6	1E-6
α_{22} (/ºC)	30E-6	20E-6
β_{11}	0.011	0.011
β_{22}	0.63	0.63
E resin (N/mm^2)		3390
ν resin		0.3
Vf nominal	0.6	0.6
Vf actual	0.6	0.68
S_{11T} (N/mm^2)	1000	2185
S_{22T} (N/mm^2)	90	60
S_{11C} (N/mm^2)	1450	1500
S_{22C} (N/mm^2)	270	180
S_{12} (N/mm^2)	80	80
F_{12}	auto	auto

Lay-up configuration

Layer	material name	thickness (nominal, mm)	angle (deg)
	- - - top - - -		
1	E-G/913	0.125	0
2	XAS/913	0.250	45
	- - bottom - -		

Curing Temperature = 120 ºC
Room Temperature = 20 ºC

Loading configuration

				order	
N_{xx}	=	40	N/mm	1	
N_{yy}	=	15	N/mm	2	
N_{xy}	=	0	N/mm	1	
M_{xx}	=	0	N	1	
M_{yy}	=	0	N	1	
κ_{xy}	=	0	/mm	1	(ref: As Cured)
Top Temperature	=	50	ºC		
Bottom Temperature	=	0	ºC		
Moisture Content	=	1	% by weight		

Calculate actual laminate thickness

Nominal laminate thickness = 0.375
For each layer, $t_{actual} = t_{nominal}\, Vf_{nominal} / Vf_{actual}$

$\therefore$ actual laminate thickness = 0.125 + 0.25 • 0.6 / 0.68 = 0.3456 mm

Calculate Q's and layer effective coefficients

Layer 1 :

ν_{21} = 0.274 • 10730 / 45600 = 0.06447,

$(1 - \nu_{12}\,\nu_{21})$ = 1 - 0.274 • 0.06447 = 0.9823

Q_{11} = 45600 / 0.9823 = 46420

Q_{12} = 0.274 • 10730 / 0.9823 = 2992.9

Q_{22} = 10730 / 0.9823 = 10923

Q_{33} = 5140

$\sin\theta = 0$, $\cos\theta = 1$

$\bar{Q}_{11}$ = 46420 $\bar{Q}_{12}$ = 2992.9 $\bar{Q}_{13}$ = 0

$\bar{Q}_{22}$ = 10923 $\bar{Q}_{23}$ = 0

$\bar{Q}_{33}$ = 5140

α_x = 7E-6 α_y = 30E-6 α_{xy} = 0

β_x = 0.011 β_y = 0.63 β_{xy} = 0

Layer 2 :

Corrections for fibre volume fraction:

Using the rule of mixtures equation $E_{11} = E_{fibre}\,Vf + E_{resin}\,(1-Vf)$

we get (for the layer) $E'_{11} = E_{11}\,Vf_{act}/Vf_{nom} + E_{resin}\,(1 - Vf_{act}/Vf_{nom})$

Similarly, using $1 / E_{22} = Vf / E_{fibre} + (1-Vf) / E_{resin}$

we get $1 / E'_{22} = 1/E_{22}\,Vf_{act}/Vf_{nom} + 1/E_{resin}\,(1 - Vf_{act}/Vf_{nom})$

Finally, ν'_{12} behaves similarly to E'_{11} and G'_{12} similarly to E'_{22}.

Hence,

E'_{11} = 144300 • 0.68 / 0.6 + 3390 (1 - 0.68 / 0.6) = 163088

E'_{22} = 1 / [1/11040 • 0.68/0.6 + 1/3390 (1 - 0.68/0.6)] = 15791

G_{resin} = 3390 / 2 / (1 + 0.3) = 1303.8

G'_{12} = 1 / [1/5790 • 0.68/0.6 + 1/1303.8 (1 - 0.68/0.6)] = 10705

ν'_{12} = 0.313 • 0.68 / 0.6 + 0.3 (1 - 0.68 / 0.6) = 0.3147

Then,

ν'_{21} = 0.3147 • 15971 / 163088 = 0.03082,

$(1 - \nu'_{12}\,\nu'_{21})$ = 1 - 0.3147 • 0.03082 = 0.9903

Q_{11} = 163088 / 0.9903 = 164685

Q_{12} = 0.3147 • 15791 / 0.9903 = 5018

Q_{22} = 15791 / 0.9903 = 15946

Q_{33} = 10705

$\sin\theta = \cos\theta = 0.7071$

$\bar{Q}_{11}$ = 164685 • 0.25 + 2 (5018 + 2 • 10705) 0.25 + 15946 • 0.25 = 58372

$\bar{Q}_{12}$ = (164685 + 15946 - 4 • 10705) 0.25 + 5018 (0.25 + 0.25) = 36962

$\bar{Q}_{13}$ = (164685 - 5018 - 2 • 10705) 0.25 + (5018 - 15946 + 2 • 10705) 0.25 = 37185

$\bar{Q}_{22}$ = 164685 • 0.25 + 2 (5018 + 2 • 10705) 0.25 + 15946 • 0.25 = 58372

$\bar{Q}_{23}$ = (164685 - 5018 - 2 • 10705) 0.25 + (5018 - 15946 + 2 • 10705) 0.25 = 37185

$\bar{Q}_{33}$ = (164685 +15946 - 2 • 5018 - 2 • 10705) 0.25 + 10705 (0.25 + 0.25) = 42649

α_x = 1E-6 • 0.5 + 20E-6 • 0.5 = 10.5E-6

α_y = 1E-6 • 0.5 + 20E-6 • 0.5 = 10.5E-6

α_{xy} = 2 (1E-6 - 20E-6) 0.5 = -19E-6

β_x = 0.011 • 0.5 + 0.63 • 0.5 = 0.3205

β_y = 0.011 • 0.5 + 0.63 • 0.5 = 0.3205

β_{xy} = 2 (0.011 - 0.63) 0.5 = -0.619

Calculate ABD matrix terms

z_0 = -0.172795, z_1 = -0.047795, z_2 = 0.172795

A_{11} = 46420 ((-0.047795) - (-0.172795)) + 58372 (0.172795 - (-0.047795)) = 18679

$B_{11} = [46420\ ((-0.047795)^2 - (-0.172795)^2) + 58372\ (0.172795^2 - (-0.047795)^2)] / 2$
$= 164.8$
$D_{11} = [46420\ ((-0.047795)^3 - (-0.172795)^3) + 58372\ (0.172795^3 - (-0.047795)^3)] / 3$
$= 180.6$

etc., for the remaining members of the ABD matrix.

LAP reports ABD as:

18676	8528	8201.6	164.61	468.36	512.6
	14239	8201.6	468.36	654	512.6
		10049	512.6	512.6	517.1
			180.63	69.953	65.295
				120.88	65.295
					83.541

and the inverse as:

9.3591E-5	-2.132E-5	-5.6877E-5	2.2725E-4	1.3384E-5	-2.795E-4
	1.491E-4	-8.7902E-5	1.3384E-5	-3.8729E-4	5.2246E-5
		2.5591E-4	-2.795E-4	5.2246E-5	-5.1793E-4
			8.6761E-3	-2.185E-3	-4.82E-3
				1.6221E-2	-8.9993E-3
					2.7371E-2

Curing forces: $\Delta T = -100$

$N^C_x = [\ (46420 \cdot 7E\text{-}6 + 2992.9 \cdot 30E\text{-}6) \cdot 0.125 +$
$(58372 \cdot 10.5E\text{-}6 + 36962 \cdot 10.5E\text{-}6 - 37185 \cdot 19E\text{-}6) \cdot 0.22059\ ] \cdot (-100) = -11.68$
$M^C_x = [\ (46420 \cdot 7E\text{-}6 + 2992.9 \cdot 30E\text{-}6) \cdot ((-0.047795)^2 - (-0.172795)^2) / 2 +$
$(58372 \cdot 10.5E\text{-}6 + 36962 \cdot 10.5E\text{-}6 - 37185 \cdot 19E\text{-}6) \cdot$
$((0.172795)^2 - (-0.047795)^2) / 2] \cdot (-100) = 0.1658$

Hygrothermal forces: m = 0.01

$N^H_x = [\ (46420 \cdot 0.011 + 2992.9 \cdot 0.63) \cdot 0.125 +$
$(58372 \cdot 0.3205 + 36962 \cdot 0.3205 - 37185 \cdot 0.619) \cdot 0.22059\ ] \cdot 0.01 = 19.62$
$M^H_x = [\ (46420 \cdot 0.011 + 2992.9 \cdot 0.63) \cdot ((-0.047795)^2 - (-0.172795)^2) / 2 +$
$(58372 \cdot 0.3205 + 36962 \cdot 0.3205 - 37185 \cdot 0.619) \cdot$
$((0.172795)^2 - (-0.047795)^2) / 2\ ] \cdot 0.01 = 0.7088$

Thermal forces: $\Delta T = 5 - 144.68\ z$

$N^T_x = [\ (46420 \cdot 7E\text{-}6 + 2992.9 \cdot 30E\text{-}6) \cdot 0.125 +$
$(58372 \cdot 10.5E\text{-}6 + 36962 \cdot 10.5E\text{-}6 - 37185 \cdot 19E\text{-}6) \cdot 0.22059\ ] \cdot 5 +$
$[\ (46420 \cdot 7E\text{-}6 + 2992.9 \cdot 30E\text{-}6) \cdot ((-0.047795)^2 - (-0.172795)^2) / 2 +$
$(58372 \cdot 10.5E\text{-}6 + 36962 \cdot 10.5E\text{-}6 - 37185 \cdot 19E\text{-}6) \cdot$
$((0.172795)^2 - (-0.047795)^2) / 2] \cdot (-144.68) = 0.8239$
$M^T_x = [\ (46420 \cdot 7E\text{-}6 + 2992.9 \cdot 30E\text{-}6) \cdot ((-0.047795)^2 - (-0.172795)^2) / 2 +$
$(58372 \cdot 10.5E\text{-}6 + 36962 \cdot 10.5E\text{-}6 - 37185 \cdot 19E\text{-}6) \cdot$
$((0.172795)^2 - (-0.047795)^2) / 2] \cdot 5 +$
$[\ (46420 \cdot 7E\text{-}6 + 2992.9 \cdot 30E\text{-}6) \cdot ((-0.047795)^3 - (-0.172795)^3) / 3 +$
$(58372 \cdot 10.5E\text{-}6 + 36962 \cdot 10.5E\text{-}6 - 37185 \cdot 19E\text{-}6) \cdot$
$((0.172795)^3 - (-0.047795)^3) / 3] \cdot (-144.68) = -0.1841$

LAP calculates internally:

$\{N^C\ M^C\}$	=	{-11.68	-10.85	0.6493	0.1658	0.0747	0.0406}
$\{N^H\ M^H\}$	=	{19.62	25.27	-5.655	0.7087	0.0858	-0.3534}
$\{N^T\ M^T\}$	=	{0.8239	0.6507	0.0262	-0.1841	-0.1635	0.00545}

Calculate the non-mechanical strain vectors

$\{\varepsilon^{0C}\ \kappa^{C}\}$ $= (ABD)^{-1}\ \{N^{C}\ M^{C}\}$
$= \{9.3591E\text{-}5 \bullet (-11.68) - 2.132E\text{-}5 \bullet (-10.85) - 5.6877E\text{-}5 \bullet 0.6493 + 2.2725E\text{-}4 \bullet 0.1658 + 1.3384E\text{-}5 \bullet 0.0747 - 2.795E\text{-}4 \bullet 0.0406\ ,\ \ldots..\}$
$= \{-8.714E\text{-}4, \ldots..\}$

etc. for the other strain components.

LAP reports:

$\{\varepsilon^{0C}\ \kappa^{C}\}$ =	{-8.7131E-4	-1.4509E-3	1.7211E-3	-1.9013E-3	4.5651E-3	2.0007E-3}
$\{\varepsilon^{0H}\ \kappa^{H}\}$ =	{1.8802E-3	3.804E-3	-4.7947E-3	1.4042E-2	-6.7954E-3	-1.5097E-2}
$\{\varepsilon^{0T}\ \kappa^{T}\}$ =	{1.6188E-5	1.3828E-4	-5.7245E-5	-1.0779E-3	-2.5379E-3	2.2978E-3}

Calculate the mechanical strain vector

The load vector for loading "order" 1 is:

{N M} = {40 0 0 0 0 (κ_{xy}=0)}

but κ_{xy} is held at zero from the "As Cured" state, which means that the mechanical κ_{xy} must be equal to the hygro-thermal κ_{xy} ($=\kappa^{H}_{xy}+\kappa^{T}_{xy}$), but of opposite sign. Therefore, for the mechanical part,

{N M} = {40 0 0 0 0 (κ_{xy}=0.0128)}

We need to partition the ABD matrix as follows:

$$\begin{bmatrix} N_x \\ N_y \\ N_{xy} \\ M_x \\ M_y \end{bmatrix} = \begin{bmatrix} A_{11} & A_{12} & A_{13} & B_{11} & B_{12} \\ A_{12} & A_{22} & A_{23} & B_{12} & B_{22} \\ A_{13} & A_{23} & A_{33} & B_{13} & B_{23} \\ B_{11} & B_{12} & B_{13} & D_{11} & D_{12} \\ B_{12} & B_{22} & B_{23} & D_{12} & D_{22} \end{bmatrix} \begin{bmatrix} \varepsilon^0_x \\ \varepsilon^0_y \\ \varepsilon^0_{xy} \\ \kappa_x \\ \kappa_y \end{bmatrix} + \begin{bmatrix} B_{13} \\ B_{23} \\ B_{33} \\ D_{13} \\ D_{23} \end{bmatrix} \kappa_{xy}$$

or, $\mathbf{N_1} = \mathbf{A_1}\, \varepsilon_1 + \mathbf{A_2}\, \varepsilon_2$

and

$$\begin{bmatrix} M_{xy} \end{bmatrix} = \begin{bmatrix} B_{13} & B_{23} & B_{33} & D_{13} & D_{23} \end{bmatrix} \begin{bmatrix} \varepsilon^0_x \\ \varepsilon^0_y \\ \varepsilon^0_{xy} \\ \kappa_x \\ \kappa_y \end{bmatrix} + D_{33}\kappa_{xy}$$

or, $\mathbf{N_2} = \mathbf{A_3}\, \varepsilon_1 + \mathbf{A_4}\, \varepsilon_2$

The unknowns are ε_1 and $\mathbf{N_2}$

Hence, $\varepsilon_1 = (\mathbf{A_1})^{-1} (\mathbf{N_1} - \mathbf{A_2}\, \varepsilon_2)$ and $\mathbf{N_2} = \mathbf{A_3}\, (\mathbf{A_1})^{-1} (\mathbf{N_1} - \mathbf{A_2}\, \varepsilon_2) + \mathbf{A_4}\, \varepsilon_2$

Therefore:

$$\begin{bmatrix} \varepsilon^0_x \\ \varepsilon^0_y \\ \varepsilon^0_{xy} \\ \kappa_x \\ \kappa_y \end{bmatrix} = \begin{bmatrix} 18676 & 8528 & 82016 & 164.61 & 468.36 \\ & 14239 & 82016 & 468.36 & 654 \\ & & 10049 & 512.6 & 512.6 \\ & & & 180.63 & 69.953 \\ & & & & 120.88 \end{bmatrix}^{-1} \left(\begin{bmatrix} 40 \\ 0 \\ 0 \\ 0 \\ 0 \end{bmatrix} - \begin{bmatrix} 512.6 \\ 512.6 \\ 517.1 \\ 65.295 \\ 65.295 \end{bmatrix} 0.0128 \right) = \begin{bmatrix} 0.003499 \\ -0.00081 \\ -0.00273 \\ 0.004868 \\ -0.00735 \end{bmatrix}$$

and

$$\begin{bmatrix} M_{xy} \end{bmatrix} = \begin{bmatrix} 512.6 & 512.6 & 517.1 & 65.295 & 65.295 \end{bmatrix} \begin{bmatrix} 0.003499 \\ -0.00081 \\ -0.00273 \\ 0.004868 \\ -0.00735 \end{bmatrix} + \begin{bmatrix} 83.541 \end{bmatrix} \begin{bmatrix} 0.0128 \end{bmatrix} = 0.876$$

At the end of loading order 1, we have:

{N M} =	{40	0	0	0	0	0.876}
$\{\varepsilon^{0M}\ \kappa^{M}\}$ =	{0.003499	-0.000810	-0.002730	0.004868	-0.007350	0.0128}

The load vector for loading "order" 2 is:

{N M} = {0 15 0 0 0 (κ_{xy}=held)}

"κ_{xy}=held" means that for the second part of the mechanical loading κ_{xy} must be set to zero so that it remains unchanged. Using the above system of equations with ε_2=0, we get:

ε_1 = {-0.000310 0.002235 -0.001300 0.000339 -0.005550}

N$_2$ = {-0.02869}

And therefore at the end of loading order 2, we have (adding up):

{N M} = {40 15 0 0 0 0.8473}

{ε^{0M} κ^M} = {0.003189 0.001425 -0.004030 0.005207 -0.012900 0.0128}

LAP reports:

{N M} = {40 15 0 0 0 0.8474}

{ε^{0M} κ^M} = {0.003187 0.001428 -0.004033 0.005210 -0.012900 0.012799}

Calculate layer strains and stresses

We can now calculate strains and stresses at any point through the laminate thickness.

First, let us calculate the total strain and stress, resulting from all components (C+H+T+M). We have:

{ε^0 κ} = {0.004212 0.003919 -0.007163 0.01627 -0.01767 0.00200}

At the intersection between the two layers, z =-0.047795 and the total ΔT =-88.1. Then,

ε_x = 0.003434, ε_y = 0.004764, ε_{xy} = -0.007259

and for layer 1,

σ_x = 46420 [0.003434 - 7E-6 • (-88.1) - 0.011 • 0.01] +
2992.9 [0.004764 - 30E-6 • (-88.1) - 0.63 • 0.01] = 186.25

and similarly, σ_y = 23.88, σ_{xy} = -37.31

For layer 2,

σ_x = 58372 [0.003434 - 10.5E-6 • (-88.1) - 0.3205 • 0.01] +
36962 [0.004764 - 10.5E-6 • (-88.1) - 0.3205 • 0.01] +
37185 [-0.007259 - (-19E-6) • (-88.1) - (-0.619) • 0.01] = 57.20

and σ_y = 85.63, σ_{xy} = 18.30

However, the stresses for layer 2 have been calculated for the actual layer thickness and need to be scaled to the nominal thickness using $\sigma_{nominal} = \sigma_{actual}\, Vf_{nominal} / Vf_{actual}$, to give

σ_x = 50.47, σ_y = 75.56, σ_{xy} = 16.15

To transform strains and stresses to the 1-2 axis system, we use the following transformations:

$$\begin{bmatrix}\varepsilon_1\\ \varepsilon_2\\ \varepsilon_{12}\end{bmatrix}=\begin{bmatrix}\cos^2\theta & \sin^2\theta & \sin\theta\cos\theta\\ \sin^2\theta & \cos^2\theta & -\sin\theta\cos\theta\\ -2\sin\theta\cos\theta & 2\sin\theta\cos\theta & \cos^2\theta-\sin^2\theta\end{bmatrix}\begin{bmatrix}\varepsilon_x\\ \varepsilon_y\\ \varepsilon_{xy}\end{bmatrix}$$

$$\begin{bmatrix}\sigma_1\\ \sigma_2\\ \sigma_{12}\end{bmatrix}=\begin{bmatrix}\cos^2\theta & \sin^2\theta & 2\sin\theta\cos\theta\\ \sin^2\theta & \cos^2\theta & -2\sin\theta\cos\theta\\ -\sin\theta\cos\theta & \sin\theta\cos\theta & \cos^2\theta-\sin^2\theta\end{bmatrix}\begin{bmatrix}\sigma_x\\ \sigma_y\\ \sigma_{xy}\end{bmatrix}$$

where θ is measured positive from x-y to 1-2.

Hence, for layer 1,

ε_1 = 0.003434, ε_2 = 0.004764, ε_{12} = -0.007259

σ_1 = 186.25, σ_2 = 23.88, σ_{12} = -37.31

and for layer 2,

ε_1 = 0.000470, ε_2 = 0.007729, ε_{12} = 0.001329

σ_1 = 79.17, σ_2 = 46.87, σ_{12} = 12.55

Effective Stiffness

E_{xx} = 1 / ($InvABD_{11}$) / (nominal thickness) = 1 / 9.3591E-5 / 0.375 = 28493

E_{yy} = 1 / ($InvABD_{22}$) / (nominal thickness) = 1 / 1.491E-4 / 0.375 = 17885

G_{xy} = 1 / ($InvABD_{33}$) / (nominal thickness) = 1 / 2.5591E-4 / 0.375 = 10420

ν_{xy} = - ($InvABD_{12}$) / ($InvABD_{11}$) = 2.132E-5 / 9.3591E-5 = 0.2278

ν_{yx} = - ($InvABD_{21}$) / ($InvABD_{22}$) = 2.132E-5 / 1.491E-4 = 0.143

The effective bending stiffnesses EI are defined by: $EI = M / \kappa$
$EI_{xx} = 1 / (InvABD_{44}) = 1 / 8.6761E-3 = 115.3$
$EI_{yy} = 1 / (InvABD_{55}) = 1 / 1.6221E-2 = 61.65$
The effective twisting stiffness is calculated from: $GJ = (2 M_{xy}) / (½ \kappa_{xy})$
$GJ = 4 / (InvABD_{66}) = 4 / 2.7371E-2 = 146.1$
The flexural moduli are defined by: $M = E t^3_{nominal} \kappa / 12$
$E_{xx\,(flex)} = 12 / (InvABD_{44}) / 0.375^3 = 26228$
$E_{yy\,(flex)} = 12 / (InvABD_{55}) / 0.375^3 = 14028$
$G_{xy\,(flex)} = 12 / (InvABD_{66}) / 0.375^3 = 8313.7$
$\nu_{xy\,(flex)} = - (InvABD_{45}) / (InvABD_{44}) = 0.2518$
$\nu_{yx\,(flex)} = - (InvABD_{54}) / (InvABD_{55}) = 0.1347$

Effective Coefficients

The effective thermal expansion coefficients are equal to the thermal strain vector for a temperature difference of $\Delta T = 1$. Therefore, we can use the curing strains and divide by the curing ΔT of -100, to get:

$\alpha_{xx} = 8.7131E-6$ $\alpha_{yy} = 14.509E-6$ $\alpha_{xy} = -17.211E-6$
$\alpha_{xx(f)} = 19.013E-6$ $\alpha_{yy(f)} = -45.651E-6$ $\alpha_{xy(f)} = -20.007E-6$

The effective hygrothermal expansion coefficients are equal to the hygrothermal strain vector for a moisture uptake of m = 1. Therefore, we can use the moisture strains and divide by the moisture "loading" m of 0.01, to get:

$\beta_{xx} = 0.18802$ $\beta_{yy} = 0.3804$ $\beta_{xy} = -0.47947$
$\beta_{xx(f)} = 1.404$ $\beta_{yy(f)} = -0.67954$ $\beta_{xy(f)} = -1.5097$

Failure criteria

Let us calculate the Tsai-Wu failure criterion equation:

$$F_1 \sigma_1 + F_2 \sigma_2 + F_{11} \sigma_1^2 + F_{22} \sigma_2^2 + F_{66} \sigma_{12}^2 + 2 F_{12} \sigma_1 \sigma_2 = 1$$

where

$$F_1 = \frac{1}{S_{1T}} - \frac{1}{S_{1C}} \qquad F_{11} = \frac{1}{S_{1T} S_{1C}}$$

$$F_2 = \frac{1}{S_{2T}} - \frac{1}{S_{2C}} \qquad F_{22} = \frac{1}{S_{2T} S_{2C}}$$

$$F_{66} = \frac{1}{S_{12}^2} \qquad F_{12} = -\sqrt{F_{11} F_{22}}/2 \text{ , or user - defined}$$

All strengths S are expressed as positive numbers. The stresses σ must include contributions from all loadings (Curing, Hygro-Thermal and Mechanical).

The value for F_{12}, as calculated automatically by the program, is based on work by Tsai and Hahn and is suitable for highly anisotropic materials. It can be overridden by the user if an alternative equation must be used. Therefore, for layer 2 and using the **nominal** stresses calculated above:

$F_1 = -2.09E-4$, $F_{11} = 3.0511E-7$, $F_2 = 0.011111$,
$F_{22} = 9.2593E-5$, $F_{66} = 1.5625E-4$, $F_{12} = -2.6576E-6$

and the Tsai-Wu equation = 0.7144

LAP reports 0.714 (*not* square-rooted).

Although LAP reports failure criteria results at layer boundaries, it is the values at the middle of each layer that are used to determine if the layer has "failed" or not, and only for the subset of failure criteria selected to "track" failure.

For linear problems, once a layer "fails" it is shown in red in layer stress and layer strain graphics. No warning or error message is issued and no stiffness change takes place.

For non-linear problems, there can be a user-defined stiffness reduction for each stiffness property separately, resulting in "kinks" or "jumps" in X-Y graphics.

Additional information

Importing foreign data file formats

Previous LAP versions

The current version of the software is 4.x. Files prepared with LAP versions 3.xx are opened directly as normal, but once they are saved they cannot be read back into versions 3.xx.

The software can also read data files of the **.ldf** format that have been prepared with LAP versions 1.01, 2.0E,or 2.0F. To do so, simply select **File | Import ▸ LAP for Windows (.LDF) data file** from the menu. The converted file is not readable by previous LAP versions and should be saved under a new filename.

If you need to copy individual data file items from old data files by using the drag-and-drop method, the old file must be converted to the current file format first.

LAP for DOS data files can be imported by using the **File | Import ▸ LAP for DOS data file** menu option.

In LAP versions 2.0F and 1.01, non-linear materials that were defined by strain had one extra property: a reference point for the given strains, either at the Operating Conditions, or at the As Cured state. This property has been dropped as of version 3.x because it was found to cause confusion. For the record, the reference point is now always the Operating Conditions state.

In LAP version 1.01, effective stiffness, layer stresses and failure criteria were shown for the *actual* layer thickness, in cases where the nominal and actual fibre volume fractions were not identical for a material in a layer. In the current version, the above results are shown for the *nominal* layer thickness. It is believed that the current formulation is more suitable for engineering design.

CoDA

The NPL *CoDA* software (Component Design Analysis) was developed by Anaglyph for the National Physical Laboratory (NPL) in the UK. It is an analysis tool for composite structures, and shares many data structures with LAP.

The commands **File | Import ▸ CoDA 2.x data file** and **File | Import ▸ CoDA 3.x data file** can be used to import all relevant information. The converted file may not be readable by *CoDA* (depending on version) and should be saved under a new filename.

Data items that are meaningless to LAP are not imported. Hence *CoDA* fibres, resins and structures are ignored, leaving just the materials and lay-ups to be imported.

Drag-and-drop operations between LAP and *CoDA* 3.x are supported for common data types, such as materials.

Text files

Text data file import is intended for transfer of data from any application. Use the **File | Import ▸ LAP Text Data** menu item to read ASCII text files in the supported format. Furthermore, text files can be loaded from the command line by using the "-txt" argument, for example "LAP -txt mydata.txt".

The format for text files is fairly free. LAP itself can generate such files by means of the **File | Export ▸ All Data** and **File | Export ▸ Lay-up + Load Data** menu items, and users are invited to examine such generated files for their format.

Because LAP can generate these text files, they can be used for transfer of data between files, in a similar fashion that a drag-and-drop operation would function. The import command appends the imported data to the active file, or opens a new file when none is active.

There are 10 commands allowed in text input. Each command is issued on a new line and its data are separated by commas. Data values that are skipped (i.e. nothing between commas) are set to default where possible, otherwise an error message is issued. Whitespace characters are ignored, except where they are part of string data. The commands are as follows:

FORMAT

This must be the first command in the file since it states the version format of the data structures to follow. The data format for the commands described here requires the FORMAT statement to be issued as:

FORMAT,LAP,3

There can be only one FORMAT command. Text files of previous formats (1 or 2) can also be imported.

COMMENT

This command is used to embed a comment line. Anything can follow this keyword. Any number of comment commands can be issued, at any point.

UNITS

The UNITS command is used to set the units for all subsequent input data. The keyword is followed by 5 comma-separated standard names of units, in the sequence Length, Force, Angle, Temperature, Strain. The default units are mm, N, deg, °C, %. Other allowed unit names are: cm,m,in, kN,MN,lbf, rad, °F, -,µe.

The command can be issued any number of times and affects only the lines that follow it. Where unit names are skipped, the previously defined units remain. For example, the command **UNITS,m,kN,deg** can be issued to set the units for a few lines of input, then the command **UNITS,,N** can be issued to change only the Force units for further input.

CONTROL

The CONTROL command is used to import various options that map onto the data window controls and most user options. There are three variations to this command, depending on the keyword that follows it: GENERAL, NL or DESIGN. The data fields are numerous for each variation:

CONTROL, GENERAL, Layup for solution, Loading for solution, Auto-Solve, Vf Correction method, track Tsai-Wu, track Tsai-Hill, track Hoffman, track Max.Stress, track Max.Strain, track Custom.

CONTROL, NL, Monitor solution, Last Steps %, Delay msec, Sound alert, Message alert.

CONTROL, DESIGN, Design Layer Thickness, SF method, Currency, Core density, Core cost, Angles group, min Offset (°), Balanced only, Tolerance %.

The simple flag fields, such as Auto-Solve or track Tsai-Wu, are 0 for OFF, 1 for ON.

The option fields (Vf correction method, SF method, Angles group), are equal to 0,1,2... in the same order as shown in the LAP user interface.

Fields that are skipped are given default values. Example:

CONTROL,GENERAL,A LAYUP,A LOADING,1,2,1,1,1,1,0,0

MATERIAL

The MATERIAL command is used to import material data. The data fields are numerous: Name, Type, E_{11}, E_{22}, G_{12}, ν_{12}, α_{11}, α_{22}, β_{11}, β_{22}, Eresin, νresin, Vfnominal, Vfactual, S1T, S2T, S12, S1C, S2C, F12, BFS k, BFS ϕ, BFS β, E_{11} % drop, E_{22} % drop, G_{12} % drop, ν_{12} % drop.

The Type field is 0 for linear materials, 1 for non-linear defined by strain, 2 for non-linear defined by stress.

The E_{11}, E_{22}, G_{12}, ν_{12} fields are the actual data for linear materials, but for non-linear materials they can be the number of definition points to follow for each property (MATPOINT commands).

Fields that are skipped are considered undefined. Example:

MATERIAL,AS4/PEEK 1,0,134000,8900,5100,0.28,,,,,,,,,2130,80,160,1100,200

MATPOINT

The MATPOINT command is used to import a single point of (stiffness property at a strain or stress) for the definition of stiffness of non-linear materials. The command takes 3 data fields: property type (E11 or E22 or G12 or v12), property value, stress or strain value. MATPOINT commands refer to the last issued MATERIAL command in the file. Example:

MATPOINT,E11,165000,2185

LAYUP

The lay-up command is used to import lay-up information. The data fields are: Name, number of layers, symmetry, Curing temperature, Room temperature, Unnotched strength method, Unnotched strength specified, K_{IC} method, K_{IC} specified, G_{IC} specified.

The number of layers field is not used by LAP, but can be the total layers for non-symmetric lay-ups, or half of the total for symmetric ones. In this fashion, it is equal to the number of LAYER commands in subsequent input.
The symmetry field is 0 for non-symmetric, or 1 for symmetric lay-ups.
The Unnotched strength calculation method field is 0 for the Budiansky-Fleck method, 1 for the Soutis-Edge method and 2 for user-specified.
The K_{IC} method field is 0 for direct definition or 1 for definition through G_{IC}.
Default values are used for skipped fields.
Example:
`LAYUP,HYBRID C+G FRP,8,1,120,20,0,,0,1250`

LAYER

The LAYER command defines a single layer for the last issued LAYUP command, and inserts it at the "bottom" of that lay-up. It takes 3 data fields: material name, thickness, fibre angle.
The material name field must refer to a material that has already been defined.
Example:
`LAYER,E-G/913,0.125,-45`

LOADING

The LOADING command imports load information. There are numerous data fields: Name, Strain Reference, Nxx type, value, order, steps, Nyy type, value, order, steps, Nxy type, value, order, steps, Mxx type, value, order, steps, Myy type, value, order, steps, Mxy type, value, order, steps, Top temperature, Bottom temperature, Moisture %.
The Strain Reference is 0 for the "Operating Conditions" state, 1 for "As Cured".
The Type fields are 0 for force or moment, 1 for strain or curvature.
Default values are used for skipped fields.
Example:
`LOADING,AXIAL + BENDING,0,0,1225,1,30,1,0,,,,,,,,90,,,,,,,,,,,50,10,1.2`

KcRoMData

The KcRoMData command defines experimental data that can be used for Rule of Mixtures calculations for K_{IC} and/or G_{IC}. It takes 7 data fields: lay-up name, selection flag, % 0º layers, % ±45º layers, E', K_{IC} or G_{IC} definition flag, K_{IC} or G_{IC} value. One KcRoMData command must be issued for each set of data.
The selection flag is 0 for "not-selected", 1 for "selected".
The K_{IC} or G_{IC} definition flag is 0 if K_{IC} is given, 1 if G_{IC} is given.
Example:
`KcRoMData,Layup 1,1,25,50,70000,0,1500`

Exporting to other programs

NASTRAN

LAP can export Lay-up information to NASTRAN (Finite Element software by MSC) by means of the **File | Export ▸ Lay-up to NASTRAN...** menu item. The command operates relative to the active Data window if the Lay-ups tab is selected: the lay-up object to be exported is the one selected in the list.

A text file is created that contains all the necessary commands and data, which can be read by NASTRAN for inclusion in composite material finite elements. All relevant material data are included. Please note that only linear materials are supported and that the exported laminate thickness and material properties are adjusted for fibre volume fraction effects.

LUSAS

LAP can export lay-up information to LUSAS (FEA software) by means of the **File | Export ▸ Lay-up to LUSAS...** menu item. The command operates relative to the active Data window if the Lay-ups tab is selected: the lay-up object to be exported is the one selected in the list.

A text command file is created that contains all the necessary commands and data, which can be read by LUSAS for use with composite material finite elements. All relevant material data are included. Please note that only linear materials are supported.

Similarly, just the material properties can be exported to LUSAS by means of the **File | Export ▸ Material(s) to LUSAS...** menu item. Again, the command operates relative to the active Data window if the Materials tab is selected: the material object to be exported is the one selected in the list or, if there is no selection, all materials in the data file are exported in one operation.

CoDAT (M/Vision)

LAP material properties can also be exported to M/Vision (material database software) using the NPL CoDAT schema, by means of the **File | Export ▸ Material(s) to CoDAT (M/Vision)...** menu item. The command operates relative to the active Data window if the Materials tab is selected: the material object to be exported is the one selected in the list or, if there is no selection, all materials in the data file are exported in one operation.

Text files

ASCII text files can be generated with LAP, to include selected data in a simple format. This functionality is intended for data transfer to spreadsheets or other custom applications.

The format for these text files has been discussed with the **Import** command (page 56). Data export is achieved by means of the **File | Export ▸ All Data (text)...** and **File | Export ▸ Lay-up + Load Data (text)...** menu items. The first variant includes all data in the current file, while the second includes only the data relevant to the current solution set. These text datafiles are comma-delimited.

Note that the Description fields for any type of object are not exported.

Exporting LAP results in text form to other applications has been discussed with the various Results windows (page 21): it is the **Copy (text)** command that is used for that, via the Windows clipboard. In addition, it is possible to export a standard set of "key" results in text format, via the **File | Export ▸ Key Results (text)...** command. These text results files are tab-delimited.

Furthermore, the various printing facilities can be used to export LAP data and results in text form. To achieve this, the standard Windows printer driver **Generic / Text Only on FILE:** must be present in the Windows configuration (install it if not present, manufacturer=Generic). Selecting to print a report to this device creates a file (filename is user-specified) that contains the ASCII text of all items present in the report.

Other

LAP does not support any other proprietary data file formats for export, but other software may provide such an interface. For example, the MSC.Patran Laminate Modeler module includes a two-way interface to LAP. In addition, setting up such functionality within LAP may be simple depending on the requirements, so if you have a request regarding export interfacing, please get in contact with the developers to find out what would be involved.

Using the Custom failure criterion

The *Additional Failure Criteria* module enables the use of a custom failure criterion, the results of which are used in the graphical and tabular displays of layer failure indices (page 25), or in the Batch Solution design procedure (page 38).

The computations for the Custom criterion are carried out outside the LAP executable, in a Windows .DLL file (Dynamically Loaded Library) which must be prepared by the user. The LAP installation CD includes a dummy library, complete with C source code and the necessary files to build it with Microsoft Developer Studio 97 (Visual C++ 5.0). If the user has access to this compiler, all that is required is to modify the file FAILCRIT.C so that the necessary failure criterion calculations are included, and then build the library. Alternatively, the provided source code may be used with any C compiler that can produce 32-bit Windows DLLs. Additional instructions are included with the source code. Finally, the custom library must be named LAPF1.DLL and placed in the same folder as the LAP executable.

The data passed from LAP to the library are:

- the local stresses in the fibre axes,
- the local material strength components (ultimate stresses),
- the local strains in the fibre axes,
- the local material ultimate strains.

Stresses and strains include all hygrothermal and mechanical effects, and all data are corrected for fibre volume fraction effects. The custom function must return a positive index, for which a value equal to or greater than 1 signifies failure.

Furthermore, the library provides a function that allows users to define a name for their custom criterion, so that the LAP results can be presented in a more meaningful way.

Product Information and Support

Contacts

Software information, orders, upgrades, telephone and electronic mail support can be obtained from:

Anaglyph Ltd
Suite 33
10 Barley Mow Passage
London W4 4PH
United Kingdom.

Tel:	(+44-20) 8987 6056
Fax:	(+44-20) 8994 1533
E-mail:	support@anaglyph.co.uk
Internet:	www.anaglyph.co.uk

Comments and suggestions are most welcome.

Acknowledgements

Parts of the present software where the Unnotched and Notched compressive strengths are calculated according to the BFS (Budiansky-Fleck-Soutis) failure criterion are based on the equivalent parts of the Composite Compressive Strength Modeller (CCSM) software package [3]. Anaglyph are grateful for access to documentation and source code, which has greatly simplified the development effort.

In particular, the help of the following parties is acknowledged:

Prof. C.Soutis, Imperial College
Dr. M.Sutcliffe, Cambridge University Engineering Department
Dr. X.J.Xin, Cambridge University Engineering Department
Prof. N.A.Fleck, Cambridge University Engineering Department
Prof. P.T.Curtis, Dstl Farnborough.

References

1. RM Jones (1975) "Mechanics of Composite Materials", McGraw-Hill, ISBN 0-07-032790-4.
2. BD Agarwal and LJ Broutman (1980) "Analysis and Performance of Fiber Composites", John Wiley & Sons.
3. MPF Sutcliffe, XJ Xin, NA Fleck and PT Curtis (1999) "Composite Compressive Strength Modeller, Version 1.4".
4. C Soutis, PT Curtis and NA Fleck (1993) "Compressive failure of notched carbon fibre composites" Proc.R. Soc. London A, Vol. 440, pp 241-256.
5. C Soutis and PT Curtis (1996) "Prediction of the post-impact compressive strength of CFRP laminated composites", Composites Science and Technology, 56, pp 677-684.
6. C Soutis and EC Edge (1997) "A method for the production of carpet plots for notched compression strength of carbon fibre reinforced plastic multidirectional laminates", Proc.Instn Mech.Engrs, 211 (G), pp 251-261.
7. B Budiansky and NA Fleck (1993) "Compressive failure of fibre composites" J.Mech.Phys.Solids, 41(1), pp.183-211.
8. S Slaughter, NA Fleck and B Budiansky (1992) "Microbuckling of fibre composites: the role of multi-axial loading and creep" J. Eng. Mater. Tech., 115(5), pp.308-313.
9. PM Jelf and NA Fleck (1994) "The failure of composite tubes due to combined compression and torsion" J. Materials Science 29 3080-3086.
10. MPF Sutcliffe and NA Fleck (1993) "Effect of geometry upon compressive failure of notched composites" Int. J. Fracture, 59, pp.115-132.