Quick Start Guide for

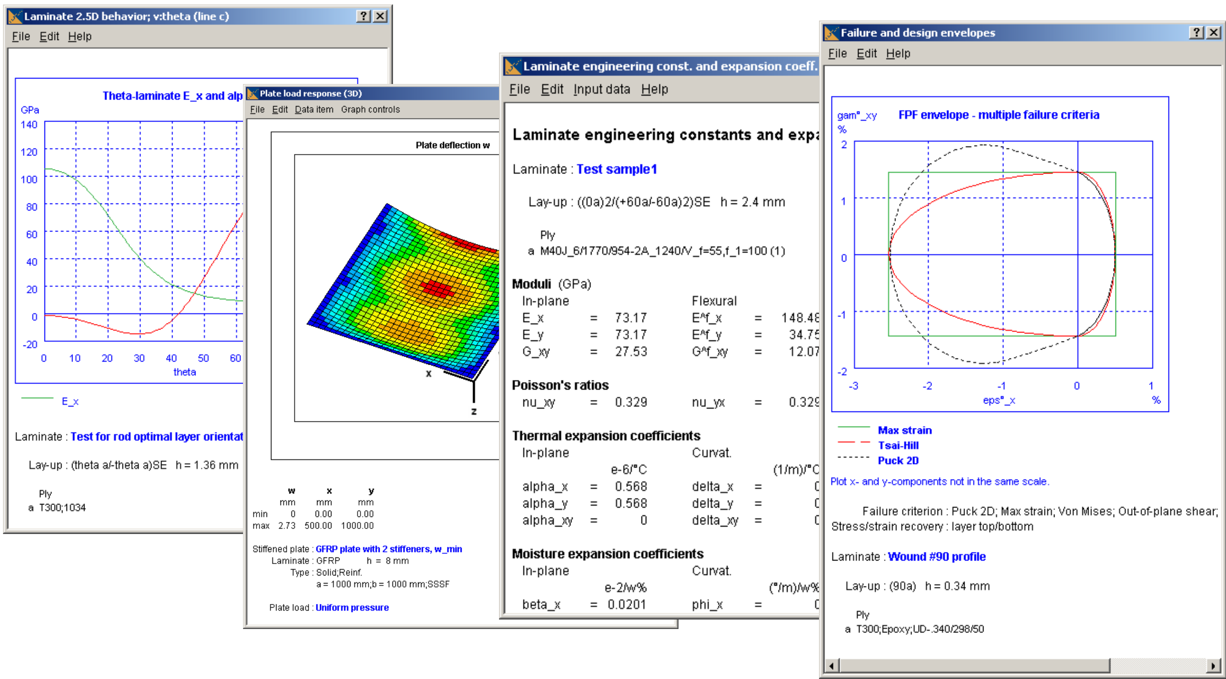
ESAComp

Version 4.0

The Quick Start Guide demonstrates the following basic functions:

- How to specify a ply
- How to specify a laminate
- Laminate 2.5D behavior analysis
- Laminate load response
- Laminate FPF analysis
- Plate analysis

To get more help on using ESAComp, please refer to the online User's Guide.



Key Features of ESAComp 4.0

Analysis capabilities for fiber/matrix micromechanics, plies, laminates, plates and stiffened panels, beams, bonded joints, and mechanical joints.

Design capabilities. Evaluation and creation of laminates based on the multiobjective design approach.

Versatile result viewing capabilities. Comparison tools, parametric studies, and graphic result displays.

Data Bank of commonly used composite materials.

FE export and import interfaces for ABAQUS®, ANSYS®, I-DEAS®, LS-DYNA® (export only), NASTRAN® and NISA® (export only). Add-on modules for further integration with ANSYS® and MSC.Patran®.

User extensions. Analysis extensions and batch mode.

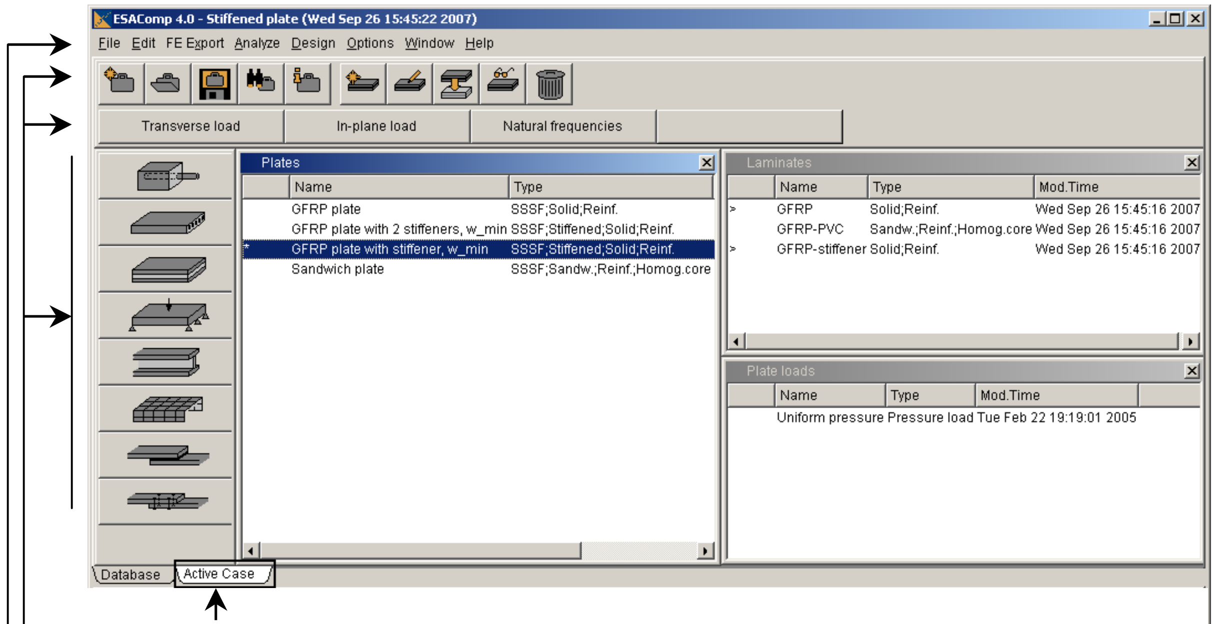
Comprehensive documentation consisting of online help and theoretical background documents.

Support services. Help desk, training, and development of customized extensions.

Platforms: MS Windows, Linux and Unix

Main window, active case

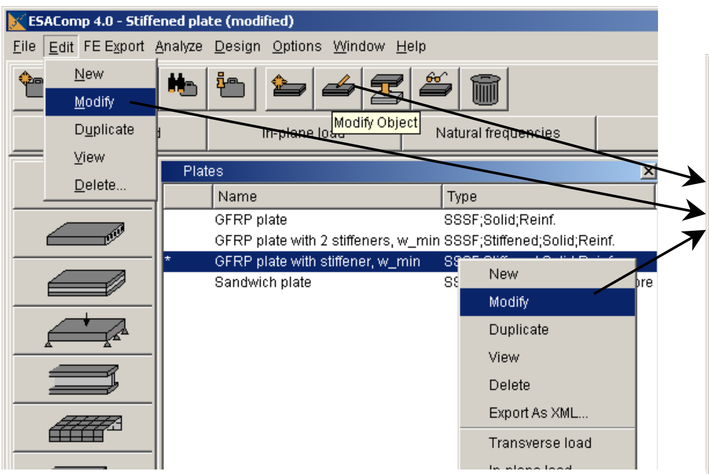
Classic ESAComp main window includes drop down menus, button bars and predefined window modes.

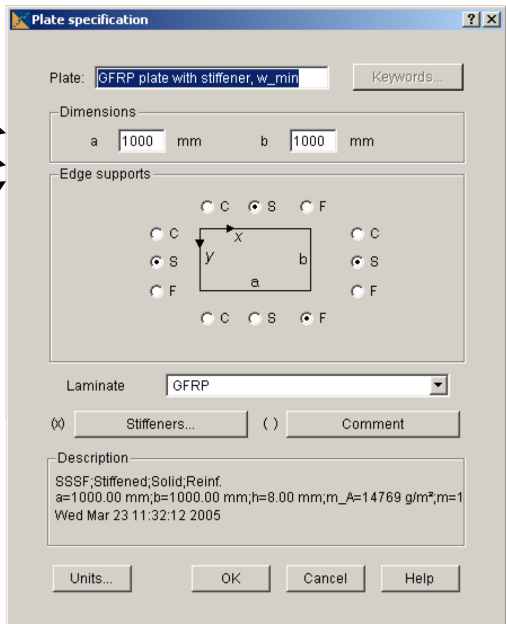


Active Case tab opens the classic ESAComp GUI.

Main window drop-down menus give access to all major functions.

Button bars provide quick access to frequently used file/edit functions and analyses. The vertical button bar on the left is for selecting the window mode: Fibers and Matrices, Plies, Laminates, … A set of object windows is opened according to the selection. The analysis button bar changes accordingly.



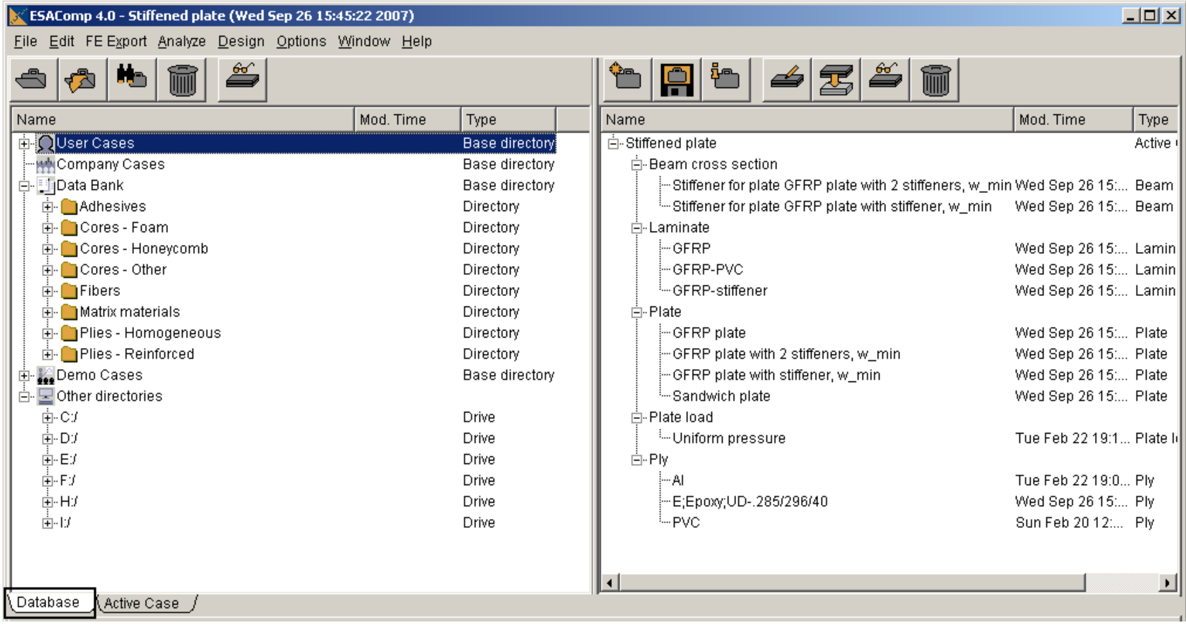


Object and analysis specification windows are accessed using drop-down menus and button bars.

Right-mouse-button menu provides a handy way to work with edit and analysis functions of the active object window.

Main window, database

At start-up ESAComp is run in the database mode. Tree type of view of ESAComp database is shown in the left window pane and the active case in the right window pane.



Switching between the database mode and the active case mode is done using tabs.

The browser type ESAComp GUI allows expanding and collapsing directories, case files, ESAComp object types, and ESAComp objects.

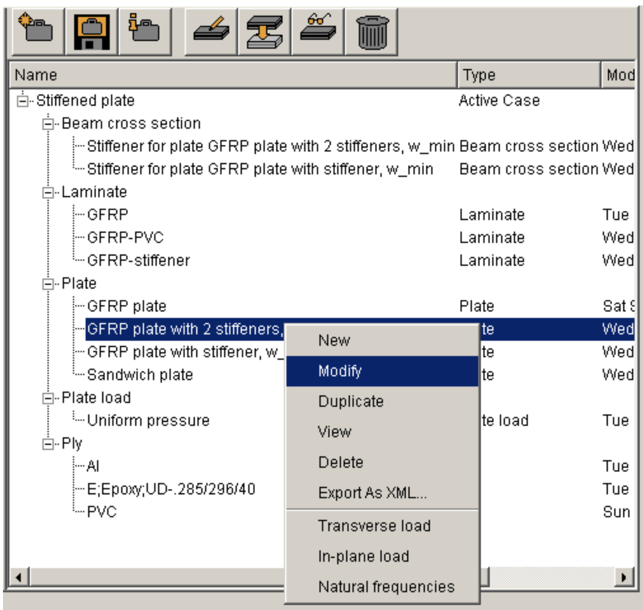
Button bar is grouped for database and active case related functions.

With database browser objects can be viewed on both window panes by double-clicking,

A case file (e.g. material data set from a specific supplier) can be opened from the database tree (File menu, button bar, mouse menu).

Objects can be imported from the database to the active case with drag-and-drop.

In the active case window pane object related functions and analyses can be accessed with mouse menu.

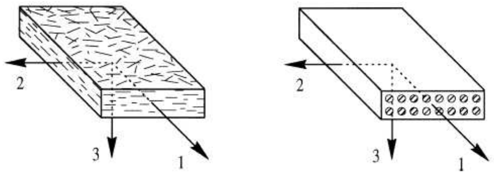


Demo 1: How to specify a ply

This demo shows how to specify a unidirectional carbon/epoxy ply with ESAComp. Plies may also be imported from the database or created using micromechanics tool.

Main window is opened when ESAComp is started. Open the **Plies** window mode using the button bar and select **New** from the right mouse button edit menu.

In the **Ply specification** window define **Ply identification** and specify the ply to be a **Reinforced ply, UD**, which is **Transversely isotropic** in the **23**-plane.



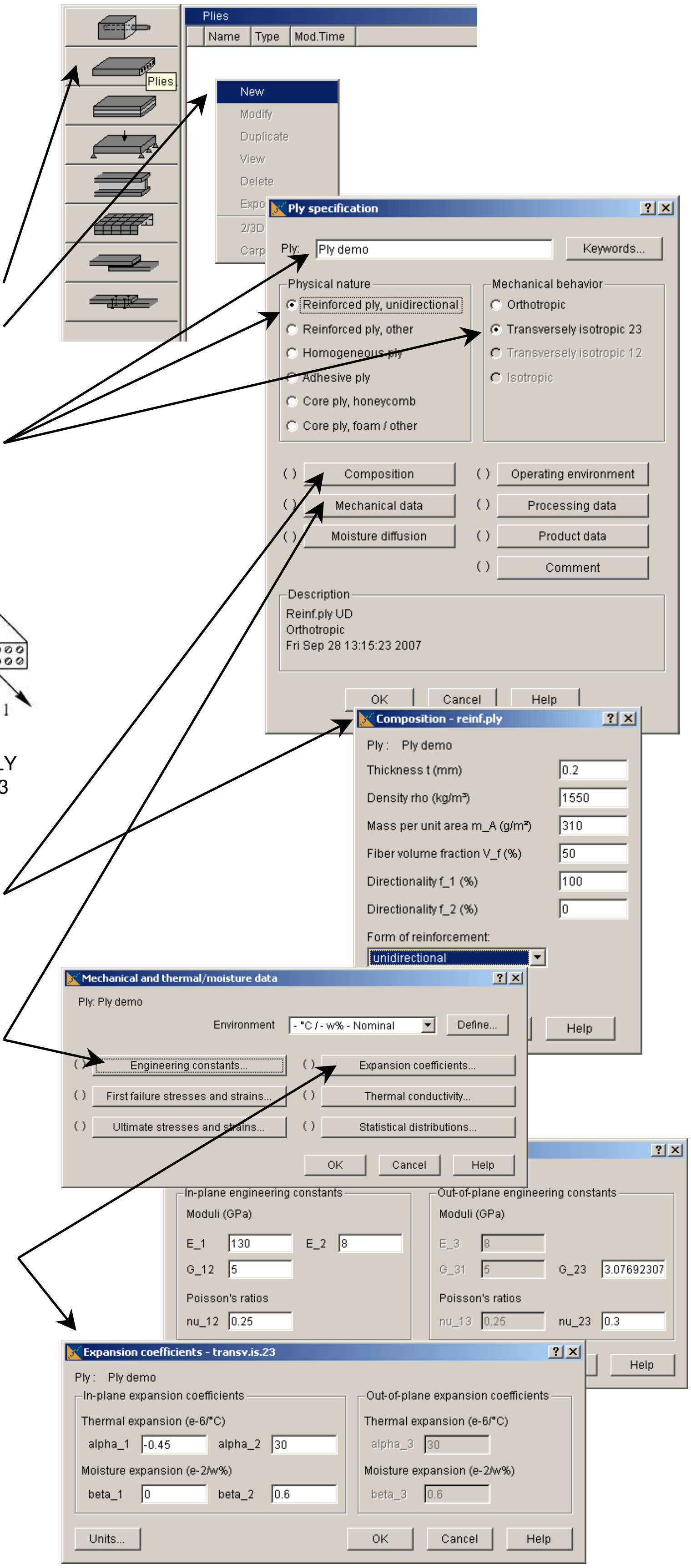
TRANSVERSELY ISOTROPIC 12 TRANSVERSELY ISOTROPIC 23

Click **Composition** and type in the ply composition data. Note that most of this data is not required for typical analyses. Click **OK**.

Click **Mechanical data** and specify the **Engineering constants...**
Some of the engineering constants are calculated by ESAComp in accordance with the mechanical behavior of the ply. Only the in-plane constants are required for typical laminate analyses. Click **OK**.

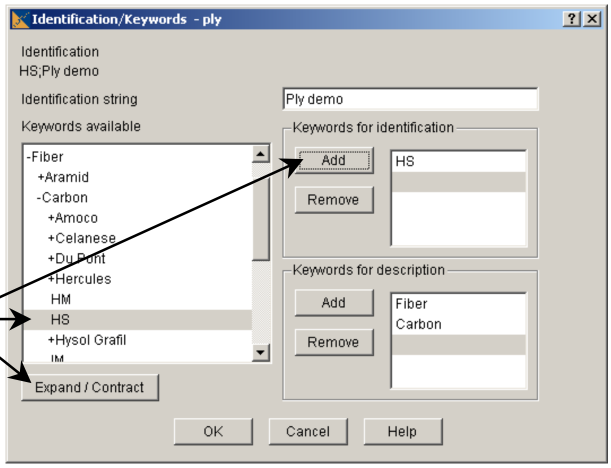
Specify also the **Expansion coefficients...** Click **OK**.

You can also specify e.g. the **First failure stresses and strains…** for FPF analyses and **Ultimate stresses and strains…** for DLF analyses.



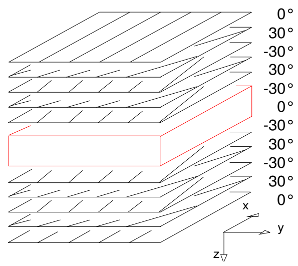
When you have specified the mechanical data, click **OK** to get back to the **Ply specification** window. You can also specify the **Processing data, Product data**, and **Comment** for the ply.

The ply identification can be a combination of a user defined id-string and/or predefined keywords. Select **Keywords…** in the **Ply specification** window.



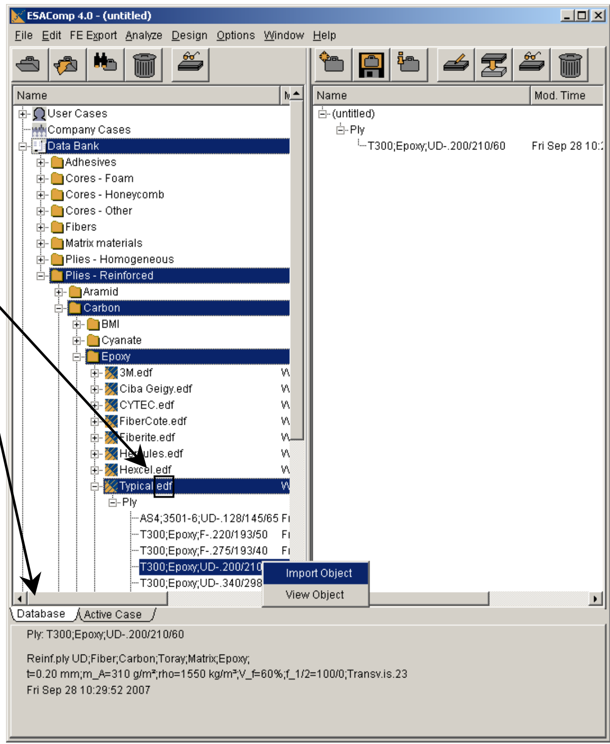
Select *Fiber* from the **Keywords available** list and click **Expand**. Repeat the **Expand** with *Carbon*. Select *HS (High Strength)* and **Add** it in the identification. The upper level keywords, *Fiber* and *Carbon,* appear automatically in the description. Description *Carbon*, for instance, is used in the FPF analysis if Puck failure criteria are used. Similarly you can specify keywords for the resin, e.g. *Matrix/Epoxy*. Click **OK**. Your ply is now identified as *HS;Epoxy;Ply demo*.

Click **OK** in the **Ply specification** window.



Demo 2: How to specify a laminate

This demo shows how to specify a sandwich laminate with ESAComp. The laminate is made up of 10 carbon/epoxy skin plies and an aluminum honeycomb core. The laminate lay-up is (0/+30/-30/+30/-30/0/-30/+30/-30/+30/0). The plies are imported into the active case from the *ESAComp Data Bank*.

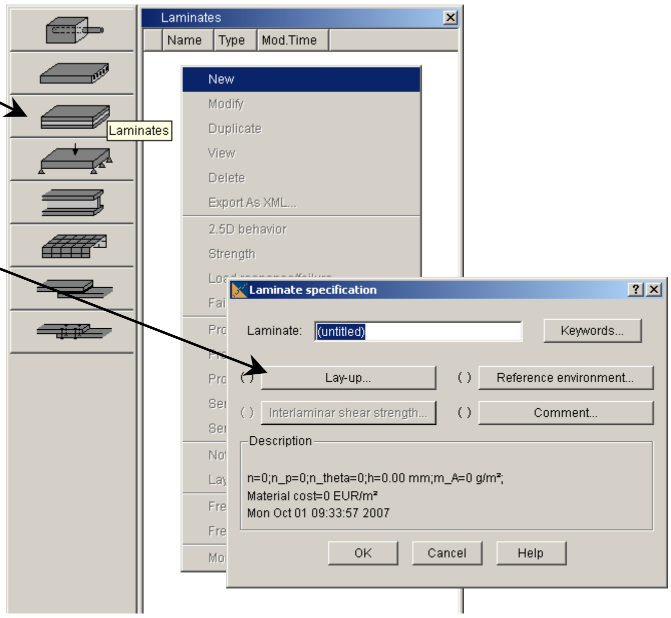


ESAComp objects such as plies can be imported from ESAComp data files (***.edf**) using the **database mode**. ESAComp data files, which are referred as cases, are stored on four levels: **User**, **Company**, **Data Bank, and Demo.**

To import the ply browse: *Data Bank*, directory *Plies – Reinforced*, directory *Carbon*, directory *Epoxy*, ESAComp data file *Typical*, and select the skin ply *T300;Epoxy;UD-.200/210/60*. The import can be made using button bar, mouse menu, or drag-and-drop.

Import accordingly the core material *1/8-2024-0.0015;HON-/80* from *Cores - Honeycomb/Aluminum/2024/Hexcel.*

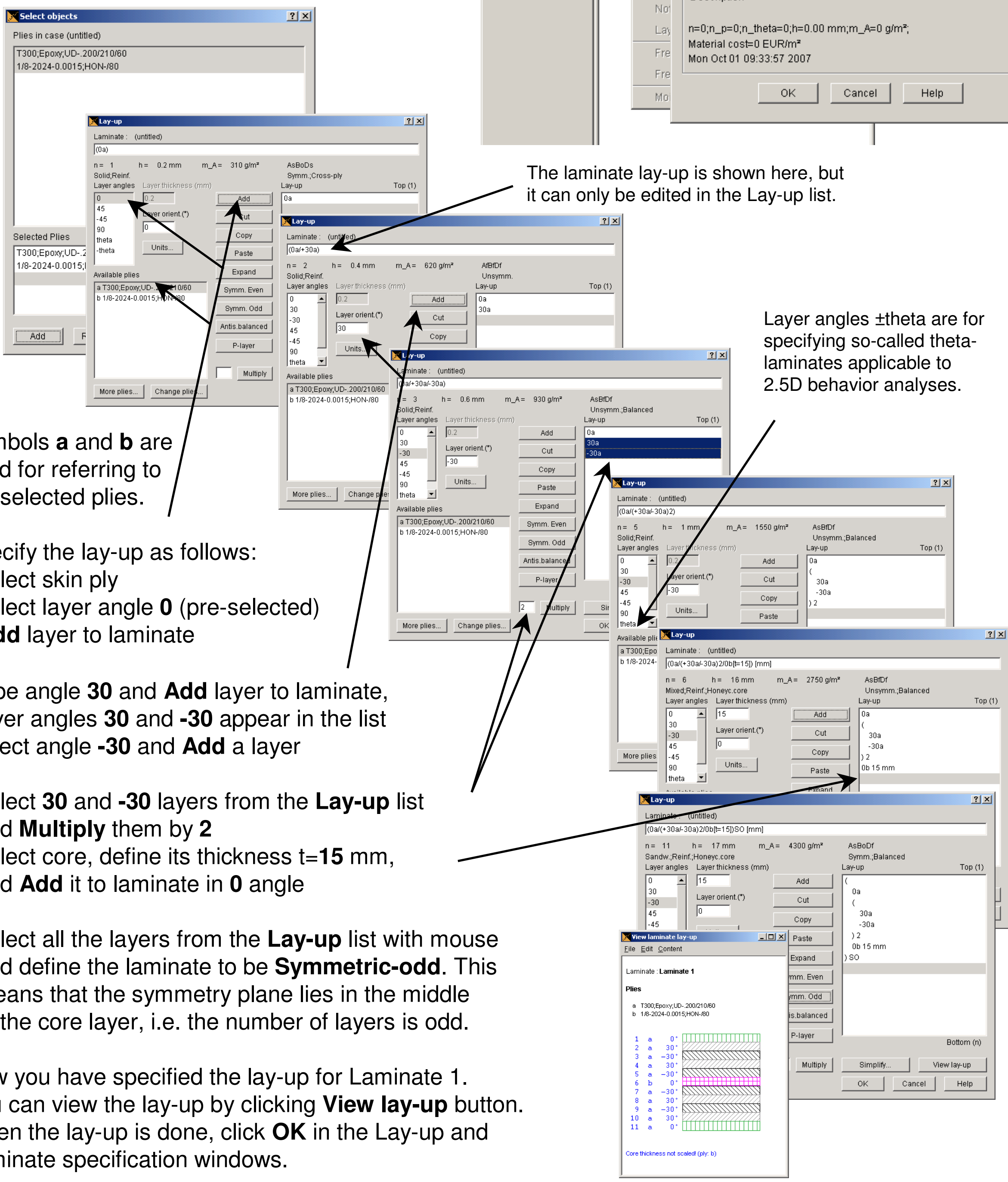
The imported plies are shown in the active case window pane.



In the active case mode open the **Laminates** window mode using the button bar and select **New** from the right mouse button edit menu.

In the **Laminate specification** window define **Laminate identification.** Select **Lay-up...** from the Laminate specification window.

Select the plies to be used in the laminate. **Add** (or double-click) them into the **Selected plies** list. Click **OK**.



The laminate lay-up is shown here, but it can only be edited in the Lay-up list.

Layer angles ±theta are for specifying so-called theta-laminates applicable to 2.5D behavior analyses.

Symbols **a** and **b** are used for referring to the selected plies.

Specify the lay-up as follows:
- select skin ply
- select layer angle **0** (pre-selected)
- **Add** layer to laminate

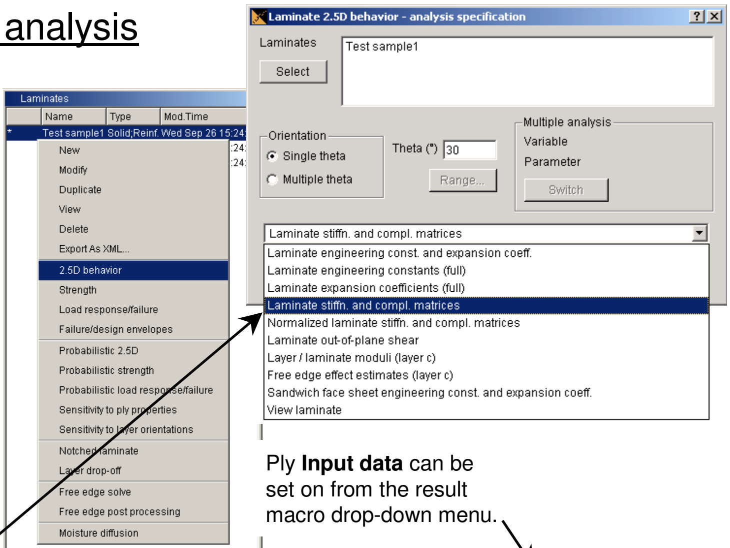
- type angle **30** and **Add** layer to laminate, layer angles **30** and **-30** appear in the list
- select angle **-30** and **Add** a layer

- select **30** and **-30** layers from the **Lay-up** list and **Multiply** them by **2**
- select core, define its thickness t=**15** mm, and **Add** it to laminate in **0** angle

- select all the layers from the **Lay-up** list with mouse and define the laminate to be **Symmetric-odd**. This means that the symmetry plane lies in the middle of the core layer, i.e. the number of layers is odd.

Now you have specified the lay-up for Laminate 1. You can view the lay-up by clicking **View lay-up** button. When the lay-up is done, click **OK** in the Lay-up and Laminate specification windows.

Demo 3: Laminate 2.5D behavior analysis



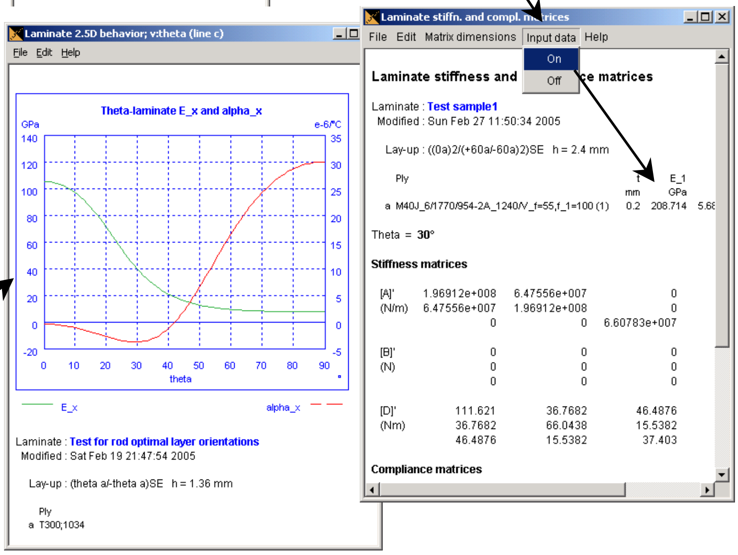
Laminate *2.5D behavior* calculates laminate stiffness and expansion coefficients assuming in plane stress state. Also, out-of-plane shear stiffness values are computed.

The demonstrated laminates are imported into the active case from the *ESAComp Demo* level. Alternatively, the case file including the laminate is opened.

To analyze laminate 2.5D behavior, select **2.5D behavior** from the right mouse button analysis menu (case: *Short beam test*).

Result macros are selected from the **Analysis specification** window drop down menu.

Ply **Input data** can be set on from the result macro drop-down menu.



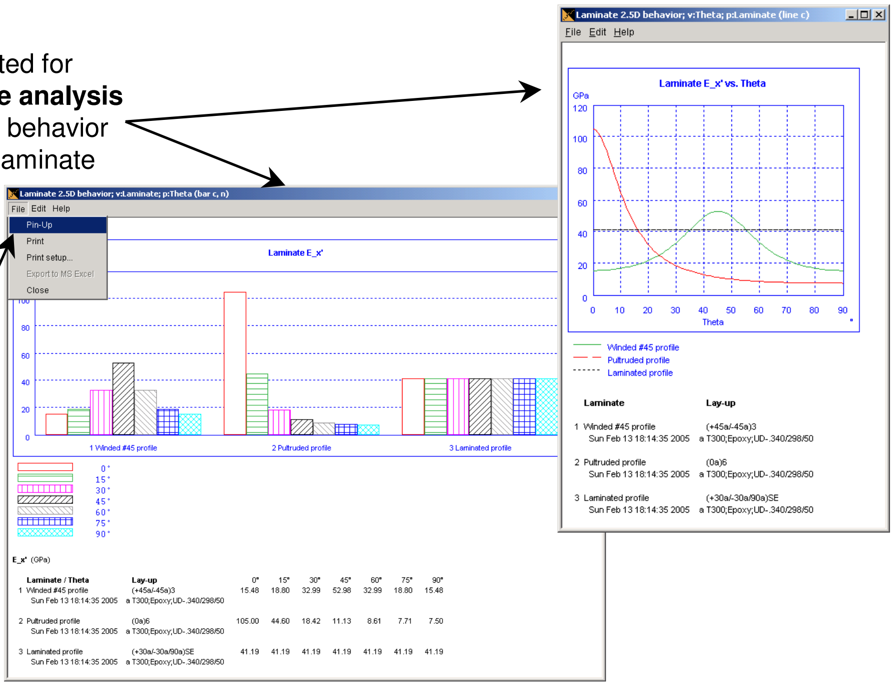
A **Single** or **Multiple** values can be given for the laminate orientation **Theta**. This provides a way to study laminate properties in the global coordinate system as a function of the laminate orientation.

2.5D behavior analyses are applicable to ***theta***-laminates and *p*-laminates as well. In p-laminates thickness proposition of the selected p-layers are varied (case: *Moisture diffusion*).

Different options for result **Display…** and **Unit/format…** can be defined. Specification windows can be found under main window **Options** drop-down menu.

Several laminates can be selected for comparative studies. In **Multiple analysis** with **Multiple theta** option 2.5D behavior analysis variable can be either laminate orientation or laminates (case: *Free torsion of thin-…*).

You can have several result windows open simultaneously. When analysis specification window is closed, all result windows are closed unless **Pin-up** is used.



Demo 4: Laminate load response

Laminate *Load response* includes laminate 2.5D behavior analysis results subjected to the applied loads. As a result layer strains and stresses in different coordinate systems can be determined.

Start by opening the case: *4-point bending* located under demo level and open the **Laminates** window mode, which includes **Plies** and **Laminate loads** objects as well.

Object windows can be arranged also **horizontally** and **vertically** using **Window** drop-down menu from the main window .

When new load is created click first **Laminate loads** title bar and use edit **New Object** button for example.

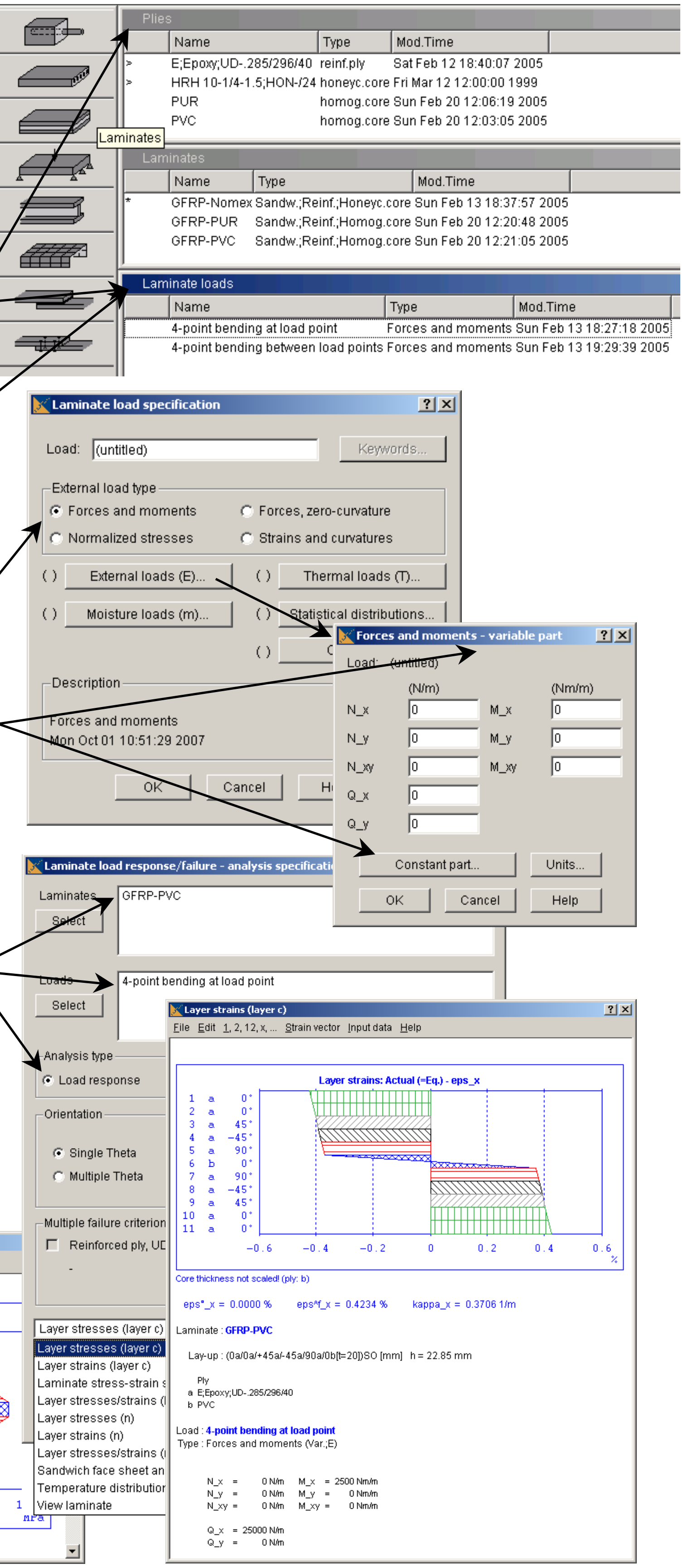
In the **Laminate load specification** window define **Load id.** Select **External load type** using **Radio buttons** and define **External loads (E)…**

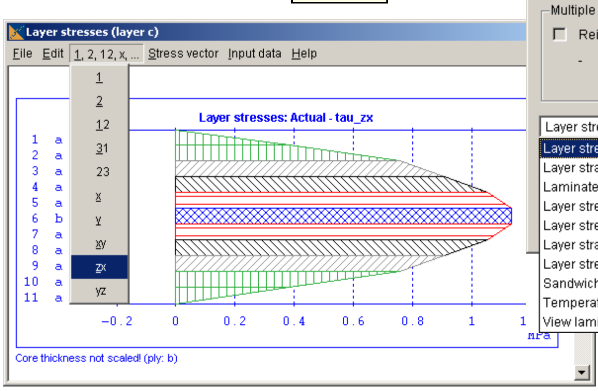
In addition to the **variable part**, a **constant part** can be specified. This partitioning of load into two load vectors may reflect, for instance, the different physical natures of the applied loads.

In this demo, existing objects of the demo case are used. Click **Load response/ failure** analysis specification button to open **Analysis specification window**.

Select **Laminate**, **Load**, and **Analysis type**. Result macros are selected from the **Analysis specification** window drop-down menu.

View object specification and object comments using edit **View Object** button.





Demo 5: Laminate FPF analysis

Laminate *FPF analysis* calculates how the laminate is able to withstand the load. Analysis is based on selected failure criteria and ply first failure strengths.

Click **Load response/failure** analysis specification button to open **Analysis specification window** (case: *4-point bending*).

Select **Laminate**, **Load**, and **Analysis type**. Several laminates and loads can be selected for comparative studies like in earlier example. Result macros are selected from the **Analysis specification** window drop down menu.

Result macros include e.g. laminate and layer reserve factors/margin of safety, failure modes, and critical layer(s).

Analysis takes into account failure within layers and interlaminar shear failure in between layers. In addition, core shear failure and face sheet wrinkling are considered for sandwich laminates.

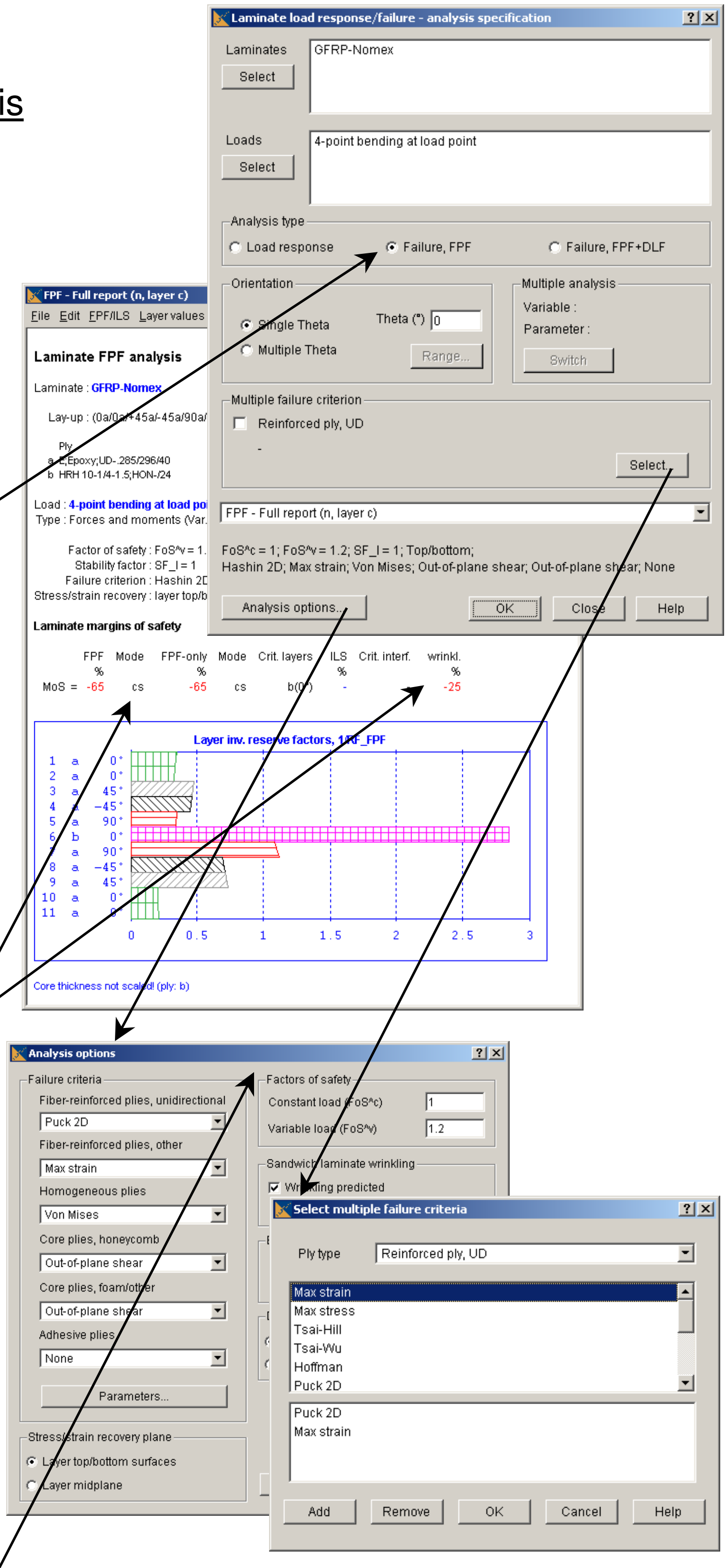
Analysis results indicate criticality of **core shear**, *cs*, and **wrinkling**. Graphics displays criticality of individual layers as per "standard FPF". Inverse reserve factor, 1/RF, is a very informative way to identify the most critical layer.

When analysis type *Failure, FPF* is selected, the **Analysis options...** can be changed.

FPF analysis can also contain **Multiple failure criteria** selected from the list of pre-defined failure criteria. Software supports use of user defined criteria as well.

Effective load vector(s) used in the FPF analysis are computed by multiplying the specified nominal load vector(s) by **Factors of safety**.

Laminate FPF analysis forms the basis of the failure analysis for various structural elements like plates, beams, and mechanical joints.

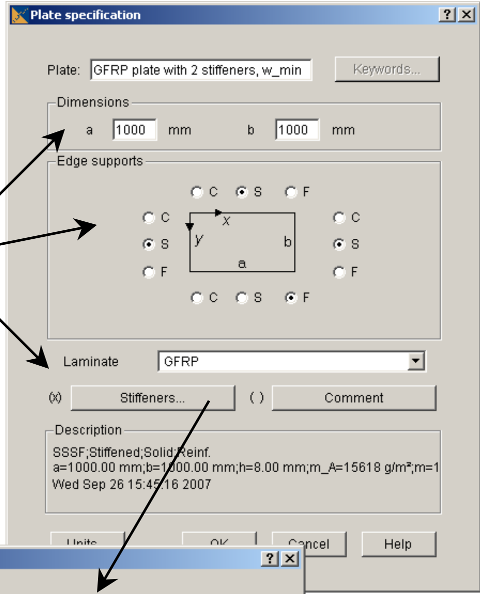


Several result windows can be open simultaneously. When analysis specification window is changed, open result windows become inactive. Their content cannot be changed from drop-down menus of the result windows.

Demo 6: Plate analysis

Plate analyses include load response and failure of a rectangular plate under transverse loads, plate buckling due to in-plane loads, and plate natural frequencies.

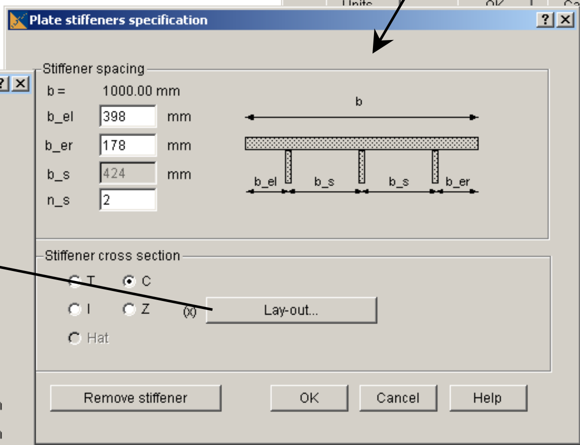
Plate creation is started by clicking edit **New Object** button when plate object is active. In **Plate specification** window plate **Dimensions**, **Edge Supports**, and used **Laminate** are specified.

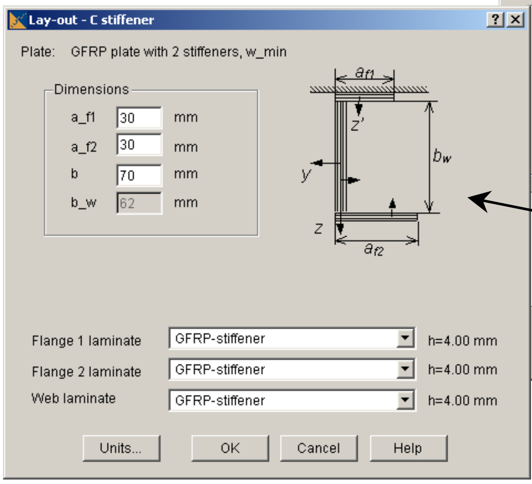


Laminate(s) to be used in the plate need to be created first. Laminate may be unsymmetric and unbalanced.

T, I, C, and **Z**-stiffeners may be defined in plate *x-direction*. Stiffeners have equal spacing. Edge distance can be different at the two edges.

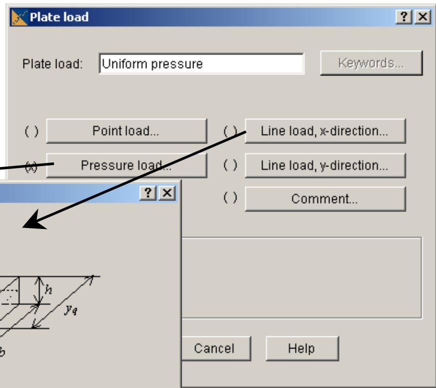
Any feasible combination of clamped, simply supported and free edges is allowed.

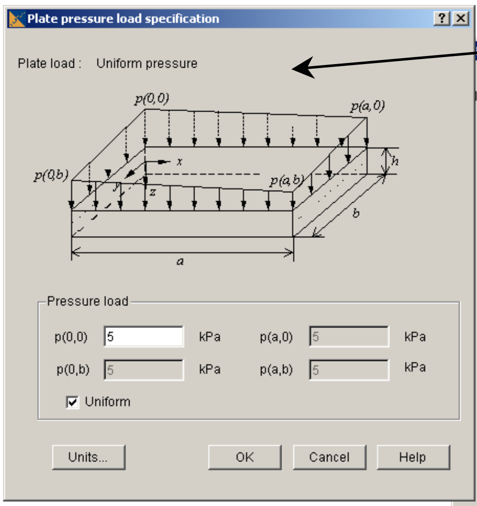


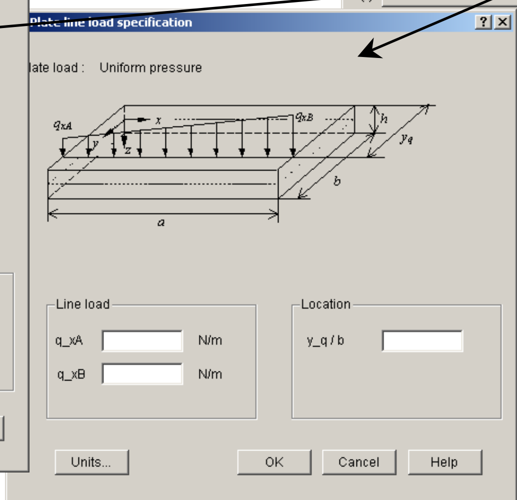


Stiffener cross section is specified similar to beam object cross sections. Stiffener top flange can be left out.

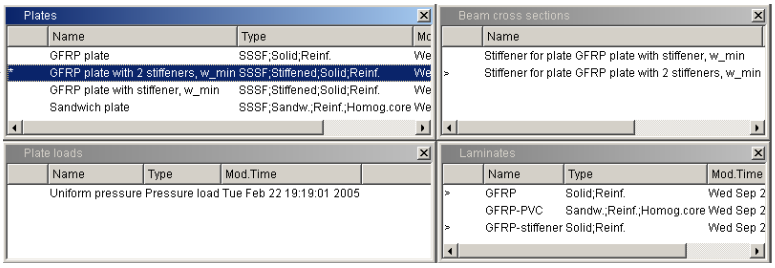
Plate transverse load may contain a point load, pressure load, line load in the *x-* and *y-direction* or any combination of these.



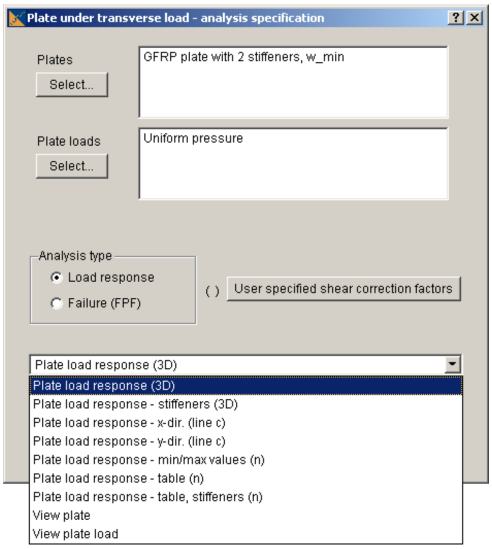


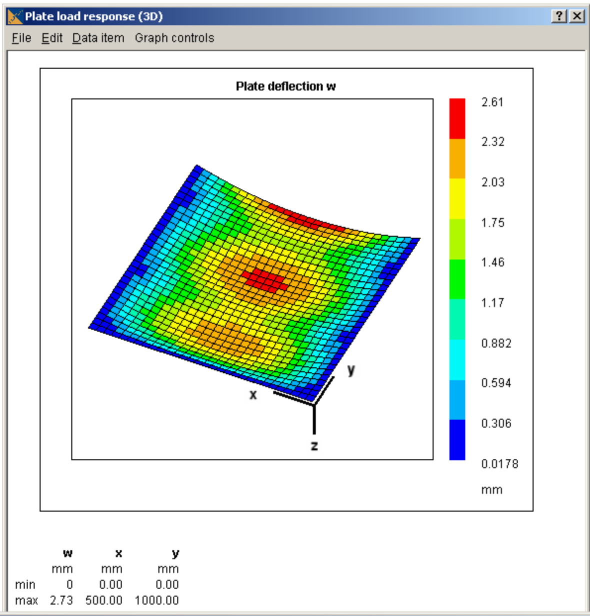


Individual object windows can be added from **Window** drop-down menu. When a **single object** is selected (marked with *) the related objects are highlighted with ">". A change in the properties of an object reflects to all related objects. Before the change is performed a confirmation is required.



Stiffener cross sections can be edited only through stiffened plate specification.





Plates and stiffener results are shown in separate displays. Both 3D contours and numeric tables are available. For shell elements laminate FPF analysis is performed at each element and for beams at various locations (case: *Stiffened plate*).

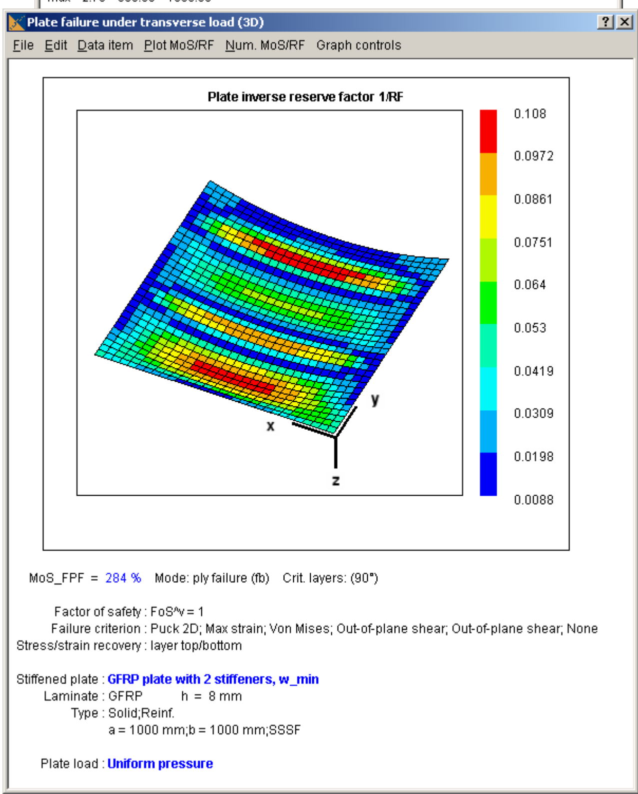


Plate analysis is based on ESAComp built-in FE solver. Plate is modeled with MITC4 shell elements and beams are based on Timoshenko beam theory. Shell element is based on FSDT and therefore, ply transverse shear moduli must be specified.

Laminate in-plane loads are used in the buckling analysis. The natural frequency analysis provides first five natural frequencies.

Componeering Inc.
Itämerenkatu 8
FI-00180 Helsinki
Finland

www.componeering.com
esacomp@componeering.com

Tel. +358 9 4342 1550
Fax +358 9 4342 1551

11/2007 © Componeering Inc.