Dlubal

Version
September 2013

Add-on Module

RF-LAMINATE

Calculation and Design of Laminate Surfaces

Program Description

© Dlubal Software GmbH
Am Zellweg 2 D-93464 Tiefenbach

Tel.: +49 9673 9203-0
Fax: +49 9673 9203-51
E-Mail: info@dlubal.com
Web: www.dlubal.com

Contents

1. Introduction

1.1 Add-on Module RF-LAMINATE

The add-on module RF-LAMINATE from Dlubal Software calculates deformations and stresses of laminate surfaces. For example, you can use RF-LAMINATE to design cross laminated timber, glued-laminated timber or OSB boards. The module is well suitable for more than just timber structures because you can create various layer compositions with any materials that can be selected from the available material library. Of course, you can also create other materials, which can be added to the library.

In RF-LAMINATE, you can create structures with different material models. Not only isotropic and orthotropic material models are available, but also user-defined models and hybrid models, which allow for a combination of isotropic and orthotropic materials in one composition. For orthotropic materials, individual layers can be rotated of angle β and you can take into account different properties in a required direction. Furthermore in RF-LAMINATE, you can decide whether or not you want to consider shear coupling of individual layers in the calculation.

Thanks to its clear layout and intuitive module windows for entering data, the module facilitates your work. In this manual, all necessary information is provided for working with RF-LAMINATE, including typical examples.

Like other modules, RF-LAMINATE is fully integrated into the RFEM program. However, it is not only an visual part of the program. Results from the module, including graphical representations, can be incorporated to the RFEM printout report. Therefore, the entire analysis can be easily and, above all, uniformly arranged and organized. The same structure of all Dlubal modules facilitates work with RF-LAMINATE as well.

We wish you much success during your work with the main program RFEM and its add-on module RF-LAMINATE.

Your DLUBAL team

1.2 RF-LAMINATE - Team

The following people were involved in the development of RF-LAMINATE:

Program coordination

Dipl.-Ing. Georg Dlubal
Dipl.-Ing. (FH) Younes El Frem
Ing. Pavel Bartoš

Programming

doc. Ing. Ivan Němec, CSc.
Mgr. Petr Zajíček
Ing. Lukáš Weis
Mgr. Vítězslav Štembera, Ph.D.

Cross-section and material libraries

Ing. Jan Rybín, Ph.D.

Design of a program, windows, dialog boxes and icons

Dipl.-Ing. Georg Dlubal
MgA. Robert Kolouch

Program testing

Mgr. Vítězslav Štembera, Ph.D.
Ing. Iva Horčičková
Dipl.-Ing. (FH) Bastian Kuhn

Manual, help system and translation

Ing. Iva Horčičková
Mgr. Vítězslav Štembera, Ph.D.
Mgr. Ing. Hana Macková
Mgr. Michaela Kryšková
Dipl.-Ing. (FH) Bastian Kuhn
Dipl.-Ing. (FH) Robert Vogl
Dipl.-Ü. Gundel Pietzcker

Technical support and quality management

Dipl.-Ing. (BA) Markus Baumgärtel
Dipl.-Ing. (BA) Sandy Matula
Dipl.-Ing. (FH) Steffen Clauß
Dipl.-Ing. (FH) Matthias Entenmann
Dipl.-Ing. Frank Faulstich
Dipl.-Ing. (FH) René Flori
M.Sc. Dipl.-Ing. (BA) Shaobin Ding
Dipl.-Ing. (FH) Bastian Kuhn
Dipl.-Ing. (FH) Andreas Hörold
Dipl.-Ing. (FH) Stefan Frenzel
Dipl.-Ing. (FH) Walter Fröhlich
M.Sc. Dipl.-Ing. Frank Lobisch
Dipl.-Ing. (FH) Alexander Meierhofer
M. Eng. Dipl.-Ing. (BA) Andreas Niemeier
M. Eng. Dipl.-Ing. (FH) Walter Rustler
M.Sc. Dipl.-Ing. (BA) Frank Sonntag
Dipl.-Ing. (FH) Christian Stautner
Dipl.-Ing. (FH) Robert Vogl
Dipl.-Ing. (FH) Andreas Wopperer

1.3 Using the Manual

Topics such as operation system requirements or an installation procedure are described in detail in the manual for the main program RFEM, therefore we put them aside in this manual. On the contrary, we focus on RF-LAMINATE special features.

When describing RF-LAMINATE, we keep to a sequence and structure of input and result windows. The described **icons** (buttons) are introduced in the text in square brackets, for example [Details]. The buttons are also displayed on the left margin. **Names** of dialog boxes, windows and individual menus are marked in the text by using *italics*, in order to find them in the program easily.

An index for a quick search of certain terms is included in this manual too. If you still cannot find what you need, please check our Web site www.dlubal.com where you can browse *FAQ* pages and find suitable suggestions.

1.4 Starting RF-LAMINATE

The add-on module RF-LAMINATE can be started from RFEM in several ways.

Main menu

You can start RF-LAMINATE by using the command from the RFEM main menu

Add-on Modules → Others → RF-LAMINATE.

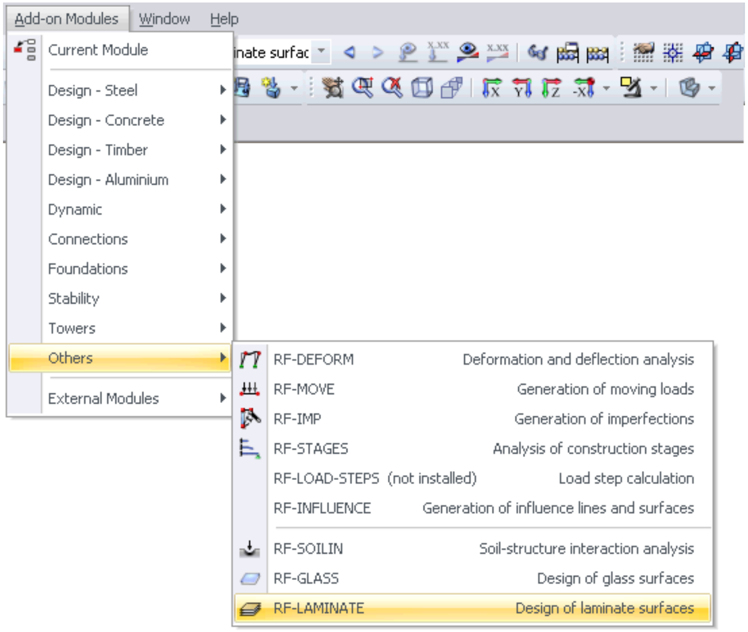


Figure 1.1: Main menu: *Add-on Modules → Others → RF-LAMINATE*

Navigator

You can also start RF-LAMINATE from the *Data* navigator by clicking the item

Add-on Modules → RF-LAMINATE – Design of laminate surfaces.

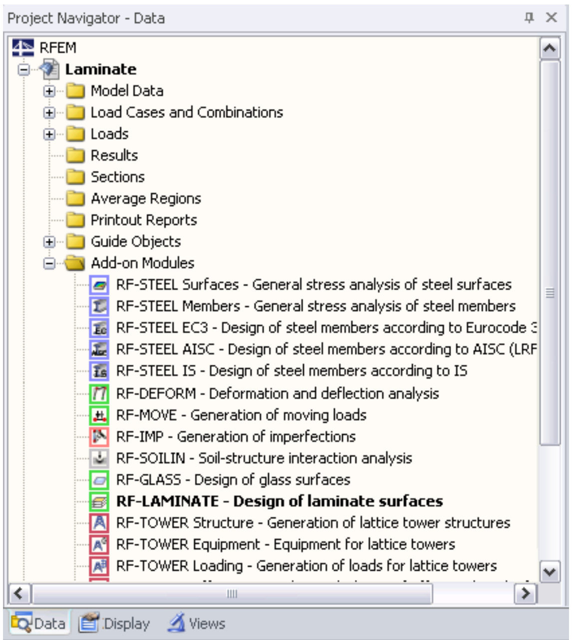
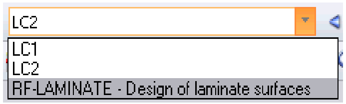


Figure 1.2: Navigator *Data*: *Add-on Modules → RF-LAMINATE*



Panel

If RF-LAMINATE results are already available in a certain RFEM model, you can set the relevant RF-LAMINATE design case in the load case list in the RFEM toolbar. By using the [Show Results] button, you can display deformations or stresses.

The [RF-LAMINATE] button is now available in the panel; you can start RF-LAMINATE by using this button.

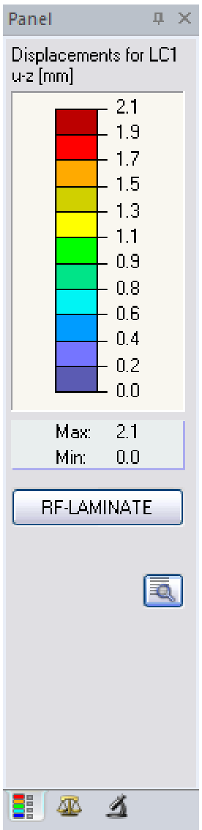


Figure 1.3: Panel: Button [RF-LAMINATE]

2. Theory

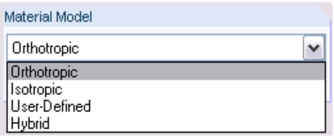
Theoretical principles that are required for working with RF-LAMINATE are introduced in this chapter.

2.1 Symbols

t	Thickness of individual layers [m]
β	Orthotropy direction [°]
E	Young's modulus of elasticity [Pa]
E_x	Young's modulus of elasticity in x' -axis direction [Pa]
E_y	Young's modulus of elasticity in y' -axis direction [Pa]
G	Shear modulus [Pa]
G_{xy}	Shear moduli in $x'y'$ -plane [Pa]
G_{xz}	Shear modulus in $x'z$ -plane [Pa]
G_{yz}	Shear modulus in $y'z$ -plane [Pa]
ν	Poisson's ratio [-]
ν_{xy}, ν_{yx}	Poisson's ratios in $x'y'$ '-plane [-]
γ	Specific weight [N/m³]
α_T	Coefficient of thermal expansion [1/K]
d'_{ij}	Elements of partial stiffness matrix in coordinate system x', y', z [Pa]
d_{ij}	Elements of partial stiffness matrix in coordinate system x, y, z [Pa]
D_{ij}	Elements of global stiffness matrix [Nm, Nm/m, N/m]
σ_x, σ_y	Normal stresses [Pa]
$\tau_{yz}, \tau_{xz}, \tau_{xy}$	Shear stresses [Pa]
n	Number of layers [-]
z	z-axis coordinate [m]
m_x	Bending moment inducing stresses in x -axis direction [Nm/m]
m_y	Bending moment inducing stresses in y -axis direction [Nm/m]
m_{xy}	Torsional moment [Nm/m]
v_x, v_y	Shear forces [N/m]
n_x	Axial force in x -axis direction [N/m]
n_y	Axial force in y -axis direction [N/m]
n_{xy}	Shear flow [N/m]
$f_{b,k}$	Characteristic value of strength for bending [Pa]
$f_{t,k}$	Characteristic value of strength for tension [Pa]
$f_{c,k}$	Characteristic value of strength for compression [Pa]
$f_{b,0,k}$	Characteristic value of strength for bending along grain [Pa]

$f_{t,0,k}$	Characteristic value of strength for tension along grain [Pa]
$f_{c,0,k}$	Characteristic value of strength for compression along grain [Pa]
$f_{b,90,k}$	Characteristic value of strength for bending perpendicular to grain [Pa]
$f_{t,90,k}$	Characteristic value of strength for tension perpendicular to grain [Pa]
$f_{c,90,k}$	Characteristic value of strength for compression perpendicular to grain [Pa]
$f_{eqv,k}$	Characteristic value of equivalent strength [Pa]
$f_{xy,k}$	Characteristic value of shear strength in plate plane [Pa]
$f_{v,k}$	Characteristic value of shear strength [Pa]
$f_{R,k}$	Characteristic value of rolling shear strength [Pa]
$f_{b,d}$	Design value of strength for bending [Pa]
$f_{t,d}$	Design value of strength for tension [Pa]
$f_{c,d}$	Design value of strength for compression [Pa]
$f_{b,0,d}$	Design value of strength for bending along grain [Pa]
$f_{t,0,d}$	Design value of strength for tension along grain [Pa]
$f_{c,0,d}$	Design value of strength for compression along grain [Pa]
$f_{b,90,d}$	Design value of strength for bending perpendicular to grain [Pa]
$f_{t,90,d}$	Design value of strength for tension perpendicular to grain [Pa]
$f_{c,90,d}$	Design value of strength for compression perpendicular to grain [Pa]
$f_{eqv,d}$	Design value of equivalent strength [Pa]
$f_{xy,d}$	Design value of shear strength in plate plane [Pa]
$f_{v,d}$	Design value of shear strength [Pa]
$f_{R,d}$	Design value of rolling shear strength [Pa]

2.2 Material Models



As already mentioned in the introduction, you can create individual layers of a structure from any material and from different material models in RF-LAMINATE. The following material models are available:

- *Orthotropic*
- *Isotropic*
- *User-Defined*
- *Hybrid*

2.2.1 Orthotropic

Properties of an orthotropic material are distinct in different directions. Therefore, the material is defined by using two moduli of elasticity E_x and E_y, three shear moduli G_{yz}, G_{xz} and G_{xy}, two Poisson's ratios ν_{xy} and ν_{yx}, specific weight γ and coefficient of thermal expansion α_T.

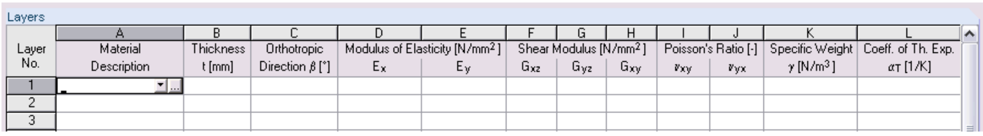


Figure 2.1: Orthotropic material model

The moduli of elasticity and Poisson's ratios are in the following mutual relation

$$\frac{\nu_{yx}}{E_y} = \frac{\nu_{xy}}{E_x} \tag{2.1}$$

Examples of the orthotropic material are timber or rolled metal sheets.

Please note that when defining an orthotropic material, there are theoretically two ways how to define Poisson's ratios. The way used in RFEM is described by using Equation (2.1) and is characterized by the relation

$$\nu_{xy} > \nu_{yx} \tag{2.2}$$

in case that the grain runs in x'-direction, that is $E_x > E_y$. In literature, you can also find the second way of the definition rarely, given by the equation $\nu_{yx} / E_x = \nu_{xy} / E_y$, leading to the inequality $\nu_{xy} < \nu_{yx}$. If you take the orthotropic material properties from a certain document, you can easily find out the applied orthotropy definition from the inequality between both Poisson's ratios.

In practice, material parameters are taken from standards. Let us show that on the example of softwood timber of strength class C24, whose values are given in standard EN 338, in Table 1.

$$\begin{aligned} E_{0,\text{mean}} &= 11000 \text{ N/mm}^2 \\ E_{90,\text{mean}} &= 370 \text{ N/mm}^2 \\ G_{\text{mean}} &= 690 \text{ N/mm}^2 \end{aligned} \tag{2.3}$$

It is assumed by default that the grain runs in x'-direction. In this case the values have the following meaning

$$\begin{aligned} E_x &= E_{0,\text{mean}} \\ E_y &= E_{90,\text{mean}} \\ G_{xy} &= G_{xz} = G_{\text{mean}} \\ G_{yz} &= \frac{G_{\text{mean}}}{10} \end{aligned} \tag{2.4}$$

where G_{yz} is the shear modulus corresponding to the rolling shear stress. To find out the Poisson's ratios, it is convenient sometimes to use approximate Huber's formulas (see Huber [4])

$$\nu_{xy} \approx \left(\frac{\sqrt{E_x E_y}}{2G_{xy}} - 1 \right) \sqrt{\frac{E_x}{E_y}}$$
$$\nu_{yx} \approx \left(\frac{\sqrt{E_x E_y}}{2G_{xy}} - 1 \right) \sqrt{\frac{E_y}{E_x}} \tag{2.5}$$

For the softwood mentioned above you get

$$E_x = 11000 \text{ MPa}$$
$$E_y = 370 \text{ MPa}$$
$$G_{xy} = G_{xz} = 690 \text{ MPa}$$
$$G_{yz} = 69 \text{ MPa} \tag{2.6}$$
$$\nu_{xy} \approx \left(\frac{\sqrt{11000 \cdot 370}}{2 \cdot 690} - 1 \right) \sqrt{\frac{11000}{370}} \doteq 2.52$$
$$\nu_{yx} \approx \left(\frac{\sqrt{11000 \cdot 370}}{2 \cdot 690} - 1 \right) \sqrt{\frac{370}{11000}} \doteq 0.08$$

Now, let us introduce an example that illustrates the meaning of Poisson's ratios in the case of an orthotropic material.

Example:

Consider the plane stress of a plane plate with the dimensions 1×1 m.

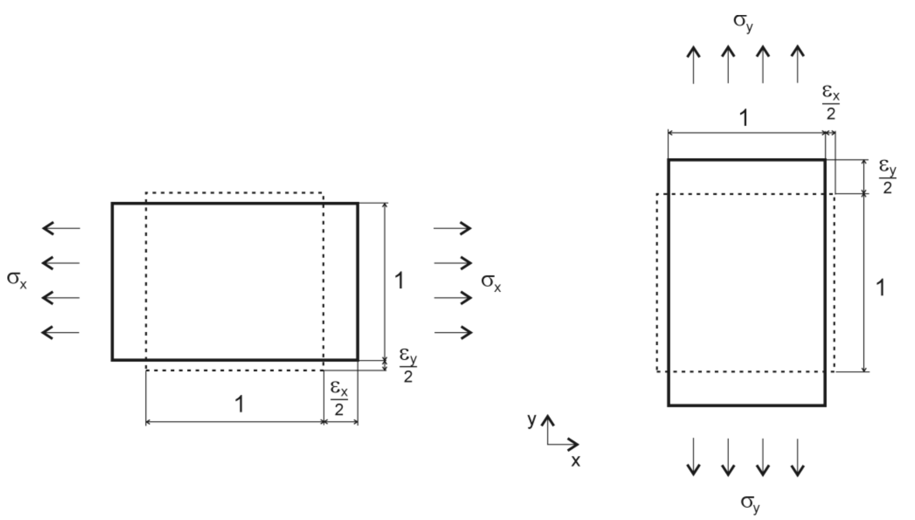


Figure 2.2: Plane stress of the plate in *x*-direction and *y*-direction

In the case of the plane stress condition for an orthotropic homogenous material, Hooke's law takes the form

$$\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} = \begin{bmatrix} \frac{1}{E_x} & -\frac{\nu_{xy}}{E_x} & 0 \\ -\frac{\nu_{yx}}{E_y} & \frac{1}{E_y} & 0 \\ 0 & 0 & G_{xy} \end{bmatrix} \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} \tag{2.7}$$

Furthermore, consider stress conditions without the shear stress $\tau_{xy} = 0$. Relation (2.7) then implies that $\gamma_{xy} = 0$ and it can be simplified to the form

$$\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \end{Bmatrix} = \begin{bmatrix} \dfrac{1}{E_x} & -\dfrac{\nu_{xy}}{E_x} \\ -\dfrac{\nu_{yx}}{E_y} & \dfrac{1}{E_y} \end{bmatrix} \begin{Bmatrix} \sigma_x \\ \sigma_y \end{Bmatrix} \tag{2.8}$$

At first, consider the stress in x-direction, where the stress is given by the relation $\sigma_x \neq 0,\ \sigma_y = 0$. By the substitution to Equation (2.8), you get

$$\begin{aligned} \varepsilon_x &= \frac{\sigma_x}{E_x} \\ \varepsilon_y &= -\frac{\nu_{xy}}{E_x}\sigma_x \end{aligned} \tag{2.9}$$

By using the combination of Equations (2.9) and (2.1), you get the relation for the Poisson's ratio ν_{xy}

$$\nu_{xy} = -\frac{\varepsilon_y}{\varepsilon_x} \tag{2.10}$$

You proceed accordingly in case of the stress in y-direction, where the stress is given by the relation $\sigma_x = 0,\ \sigma_y \neq 0$. By the substitution to Equation (2.8), you get

$$\begin{aligned} \varepsilon_x &= -\frac{\nu_{yx}}{E_y}\sigma_y \\ \varepsilon_y &= \frac{\sigma_y}{E_y} \end{aligned} \tag{2.11}$$

By using the combination of Equations (2.11) and (2.1), you get the relation for the Poisson's ratio ν_{yx}

$$\nu_{yx} = -\frac{\varepsilon_x}{\varepsilon_y} \tag{2.12}$$

Equations (2.10) and (2.12) can be interpreted in the following way: the Poisson's ratio ν_{ij} is equal to the negative contraction ratio in direction j at the extension in direction i.

The case of the combined stress can be described by Equation (2.8) that can be converted to the following schematic form:

$$\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \end{Bmatrix} = \begin{bmatrix} 1 & -\nu_{yx} \\ -\nu_{xy} & 1 \end{bmatrix} \begin{Bmatrix} \dfrac{\sigma_x}{E_x} \\ \dfrac{\sigma_y}{E_y} \end{Bmatrix} \tag{2.13}$$

2.2.2 Isotropic

An isotropic material has all mechanical properties the same in all directions. The material is defined by using the modulus of elasticity E, shear modulus G, Poisson's ratio ν, specific weight γ and coefficient of thermal expansion α_T.

Layers

Layer No.	A Material Description	B Thickness t [mm]	C Modulus of Elast. E [N/mm²]	D Shear Modulus G [N/mm²]	E Poisson's Ratio ν [-]	F Specific Weight γ [N/m³]	G Coeff. of Th. Exp. αT [1/K]	H Comment
1								
2								
3								

Figure 2.2: Isotropic material model

Examples of the isotropic material are glass or steel. For the modulus of elasticity E, shear modulus G and Poisson's ratio ν the following relation applies

$$G = \frac{E}{2(1+\nu)} \tag{2.14}$$

The value of the Poisson's ratio value is in the range $\langle -0.999, 0.5 \rangle$, where the limit value $\nu = 0.5$ corresponds to a voluminously incompressible material (in practice for example rubber).

2.2.3 User-Defined

A user-defined material model makes it possible to enter stiffness matrix elements of individual layers directly. For the calculation of shear elements of the global stiffness matrix, you need to fill in shear moduli G_{xz} and G_{yz}. as well. The material is further characterized by the specific weight γ and by the coefficient of thermal expansion α_T.

Layers

Layer No.	A Material Description	B Thickness t [mm]	C Orthotropic Direction β [°]	D Partial Stiffness Matrix Elements [kN/m²] d'11	E d'12	F d'22	G d'33	H Shear Modulus [N/mm²] Gxz	I Gyz	J Specific Weight γ [N/m³]	K Coeff. of Th. Exp. αT [1/K]
1											
2											
3											

Figure 2.3: User-defined material model

2.2.4 Hybrid

A hybrid material model allows for a combination of isotropic and orthotropic layers.

Layers

Layer No.	A Material Description	B Material Model	C Thickness t [mm]	D Orthotropic Direction β [°]	E Modulus of Elasticity [N/mm²] E	F Ey	G Shear Modulus [N/mm²] G	H Gyz	I Gxy	J Poisson's Ratio ν	K νyx
1											
2		Orthotropic									
3		Isotropic									
4		User-Defined									

Figure 2.4: Hybrid material model

2.3 Stiffness Matrix

2.3.1 Shear Coupling of Layers Is Considered

Consider a plate consisting of n layers of a generally orthotropic material. Each layer has the thickness t_i and minimum and maximum z-coordinate $z_{i;\min}, z_{i;\max}$.

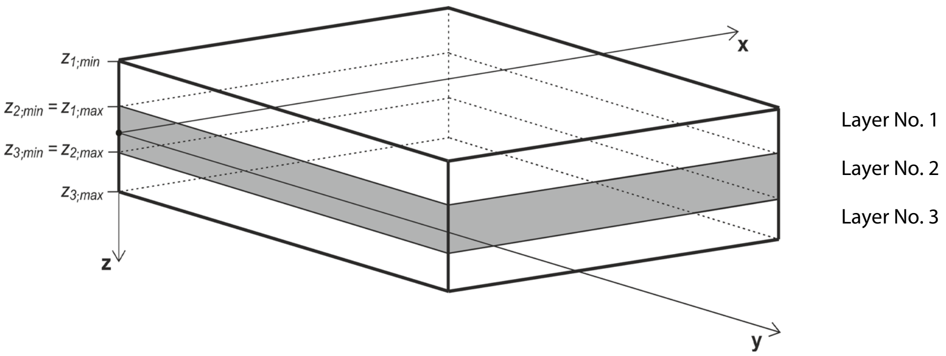


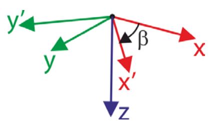
Figure 2.5: Layer scheme

The stiffness matrix for each layer is $\mathbf{d}'_i$ according to the following relation

$$\mathbf{d}'_i = \begin{bmatrix} d'_{i;11} & d'_{i;12} & 0 \\ & d'_{i;22} & 0 \\ \text{sym.} & & d'_{i;33} \end{bmatrix} = \begin{bmatrix} \dfrac{E_{i;x}}{1-\nu_{i;xy}^2 \dfrac{E_{i;y}}{E_{i;x}}} & \dfrac{\nu_{i;xy} E_{i;y}}{1-\nu_{i;xy}^2 \dfrac{E_{i;y}}{E_{i;x}}} & 0 \\ & \dfrac{E_{i;y}}{1-\nu_{i;xy}^2 \dfrac{E_{i;y}}{E_{i;x}}} & 0 \\ \text{sym.} & & G_{i;xy} \end{bmatrix} \quad i = 1,\ldots,n \tag{2.15}$$

For isotropic materials, where $E_{i;x} = E_{i;y}$ applies, the stiffness matrix has the simplified form

$$\mathbf{d}'_i = \begin{bmatrix} d'_{i;11} & d'_{i;12} & 0 \\ & d'_{i;22} & 0 \\ \text{sym.} & & d'_{i;33} \end{bmatrix} = \begin{bmatrix} \dfrac{E_i}{1-\nu_i^2} & \dfrac{\nu_i E_i}{1-\nu_i^2} & 0 \\ & \dfrac{E_i}{1-\nu_i^2} & 0 \\ \text{sym.} & & G_i \end{bmatrix} \quad i = 1,\ldots,n \quad \text{where } G_i = \frac{E_i}{2 \cdot (1+\nu_i)} \tag{2.16}$$



Because layers with orthotropic materials can be rotated arbitrarily by the angle β, it is necessary to transform stiffness matrices of individual layers to a uniform coordinate system x, y (local coordinate system of a surface).

$$\mathbf{d}_i = \begin{bmatrix} d_{i;11} & d_{i;12} & d_{i;13} \\ & d_{i;22} & d_{i;23} \\ \text{sym.} & & d_{i;33} \end{bmatrix} = \mathbf{T}_{3\times3;i}{}^{\mathrm{T}} \mathbf{d}'_i \mathbf{T}_{3\times3;i} \tag{2.17}$$

where

$$\mathbf{T}_{3\times3;i} = \begin{bmatrix} c^2 & s^2 & cs \\ s^2 & c^2 & -cs \\ -2cs & 2cs & c^2-s^2 \end{bmatrix}, \text{ where } c = \cos(\beta_i),\ s = \sin(\beta_i) \tag{2.18}$$

The individual elements then are

$$
\begin{aligned}
d_{i;11} &= c^4 d'_{i;11} + 2c^2 s^2 d'_{i;12} + s^4 d'_{i;22} + 4c^2 s^2 d'_{i;33} \\
d_{i;12} &= c^2 s^2 d'_{i;11} + s^4 d'_{i;12} + c^4 d'_{i;12} + c^2 s^2 d'_{i;22} - 4c^2 s^2 d'_{i;33} \\
d_{i;13} &= c^3 s d'_{i;11} + c s^3 d'_{i;12} - c^3 s d'_{i;12} - c s^3 d'_{i;22} - 2c^3 s d'_{i;33} + 2c s^3 d'_{i;33} \\
d_{i;22} &= s^4 d'_{i;11} + 2c^2 s^2 d'_{i;12} + c^4 d'_{i;22} + 4c^2 s^2 d'_{i;33} \\
d_{i;23} &= c s^3 d'_{i;11} + c^3 s d'_{i;12} - c s^3 d'_{i;12} - c^3 s d'_{i;22} + 2c^3 s d'_{i;33} - 2c s^3 d'_{i;33} \\
d_{i;33} &= c^2 s^2 d'_{i;11} - 2c^2 s^2 d'_{i;12} + c^2 s^2 d'_{i;22} + \left(c^2 - s^2\right)^2 d'_{i;33}
\end{aligned}
$$

The global stiffness matrix is

$$
\mathbf{D} = \begin{bmatrix}
D_{11} & D_{12} & D_{13} & 0 & 0 & D_{16} & D_{17} & D_{18} \\
 & D_{22} & D_{23} & 0 & 0 & \text{sym.} & D_{27} & D_{28} \\
 & & D_{33} & 0 & 0 & \text{sym.} & \text{sym.} & D_{38} \\
 & & & D_{44} & D_{45} & 0 & 0 & 0 \\
 & & & & D_{55} & 0 & 0 & 0 \\
 & & \text{sym.} & & & D_{66} & D_{67} & D_{68} \\
 & & & & & & D_{77} & D_{78} \\
 & & & & & & & D_{88}
\end{bmatrix} \tag{2.19}
$$

$$
\begin{Bmatrix} m_x \\ m_y \\ m_{xy} \\ v_x \\ v_y \\ n_x \\ n_y \\ n_{xy} \end{Bmatrix} =
\begin{bmatrix}
D_{11} & D_{12} & D_{13} & 0 & 0 & D_{16} & D_{17} & D_{18} \\
 & D_{22} & D_{23} & 0 & 0 & \text{sym.} & D_{27} & D_{28} \\
 & & D_{33} & 0 & 0 & \text{sym.} & \text{sym.} & D_{38} \\
 & & & D_{44} & D_{45} & 0 & 0 & 0 \\
 & & & & D_{55} & 0 & 0 & 0 \\
 & & \text{sym.} & & & D_{66} & D_{67} & D_{68} \\
 & & & & & & D_{77} & D_{78} \\
 & & & & & & & D_{88}
\end{bmatrix}
\begin{Bmatrix} \kappa_x \\ \kappa_y \\ \kappa_{xy} \\ \gamma_{xz} \\ \gamma_{yz} \\ \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} \tag{2.20}
$$

Bending and torsion
Shear
Membrane
Eccentricity

If angles β_i are multiples of 90°, the global stiffness matrix has the simplified form

$$
\mathbf{D} = \begin{bmatrix}
D_{11} & D_{12} & 0 & 0 & 0 & D_{16} & D_{17} & 0 \\
 & D_{22} & 0 & 0 & 0 & \text{sym.} & D_{27} & 0 \\
 & & D_{33} & 0 & 0 & 0 & 0 & D_{38} \\
 & & & D_{44} & D_{45} & 0 & 0 & 0 \\
 & & & & D_{55} & 0 & 0 & 0 \\
 & & \text{sym.} & & & D_{66} & D_{67} & 0 \\
 & & & & & & D_{77} & 0 \\
 & & & & & & & D_{88}
\end{bmatrix} \tag{2.21}
$$

Stiffness matrix elements (bending and torsion) [Nm]

$$D_{11} = \sum_{i=1}^{n} \frac{z_{i;\max}^3 - z_{i;\min}^3}{3} d_{i;11} \qquad D_{12} = \sum_{i=1}^{n} \frac{z_{i;\max}^3 - z_{i;\min}^3}{3} d_{i;12} \qquad D_{13} = \sum_{i=1}^{n} \frac{z_{i;\max}^3 - z_{i;\min}^3}{3} d_{i;13}$$

$$D_{22} = \sum_{i=1}^{n} \frac{z_{i;\max}^3 - z_{i;\min}^3}{3} d_{i;22} \qquad D_{23} = \sum_{i=1}^{n} \frac{z_{i;\max}^3 - z_{i;\min}^3}{3} d_{i;23}$$

$$D_{33} = \sum_{i=1}^{n} \frac{z_{i;\max}^3 - z_{i;\min}^3}{3} d_{i;33}$$

Note: in case of the single layer plate of thickness t , the introduced relations lead to the familiar relation

$$D_{ij} = \sum_{i=1}^{n=1} \frac{z_{i;\max}^3 - z_{i;\min}^3}{3} d_{i;ij} = \frac{\left(\frac{t}{2}\right)^3 - \left(-\frac{t}{2}\right)^3}{3} d_{1;ij} = \frac{2\left(\frac{t}{2}\right)^3}{3} d_{1;ij} = \frac{t^3}{12} d_{1;ij} \qquad i,j = 1,2,3$$

Stiffness matrix elements (eccentricity effects) [Nm/m]

$$D_{16} = \sum_{i=1}^{n} \frac{z_{i;\max}^2 - z_{i;\min}^2}{2} d_{i;11} \qquad D_{17} = \sum_{i=1}^{n} \frac{z_{i;\max}^2 - z_{i;\min}^2}{2} d_{i;12} \qquad D_{18} = \sum_{i=1}^{n} \frac{z_{i;\max}^2 - z_{i;\min}^2}{2} d_{i;13}$$

$$D_{27} = \sum_{i=1}^{n} \frac{z_{i;\max}^2 - z_{i;\min}^2}{2} d_{i;22} \qquad D_{28} = \sum_{i=1}^{n} \frac{z_{i;\max}^2 - z_{i;\min}^2}{2} d_{i;23}$$

$$D_{38} = \sum_{i=1}^{n} \frac{z_{i;\max}^2 - z_{i;\min}^2}{2} d_{i;33}$$

Stiffness matrix elements (membrane) [N/m]

$$D_{66} = \sum_{i=1}^{n} t_i\, d_{i;11} \qquad D_{67} = \sum_{i=1}^{n} t_i\, d_{i;12} \qquad D_{68} = \sum_{i=1}^{n} t_i\, d_{i;13}$$

$$D_{77} = \sum_{i=1}^{n} t_i\, d_{i;22} \qquad D_{78} = \sum_{i=1}^{n} t_i\, d_{i;23}$$

$$D_{88} = \sum_{i=1}^{n} t_i\, d_{i;33}$$

Stiffness matrix elements (shear) [N/m]

The exact calculation procedure for shear elements of the stiffness matrix is not introduced, but the following relations apply

$$\max_i \left(\frac{5}{6} G_{i;11} t_i \right) \le D_{44} \le \frac{5}{6} \max_i \left(G_{i;11} \right) \sum_{i=1}^{n} t_i \tag{2.22}$$

$$\max_i \left(\frac{5}{6} G_{i;22} t_i \right) \le D_{55} \le \frac{5}{6} \max_i \left(G_{i;22} \right) \sum_{i=1}^{n} t_i \tag{2.23}$$

where

$$\mathbf{G}_i = \begin{bmatrix} G_{i;11} & G_{i;12} \\ \text{sym.} & G_{i;22} \end{bmatrix} = \mathbf{T}_{2\times2;i}{}^{\mathrm{T}} \mathbf{G}'_i \mathbf{T}_{2\times2;i} \tag{2.24}$$

where

$$\mathbf{G}'_i = \begin{bmatrix} G_{xz} & 0 \\ 0 & G_{yz} \end{bmatrix} \text{ and } \mathbf{T}_{2\times2;i} = \begin{bmatrix} \cos(\beta_i) & \sin(\beta_i) \\ -\sin(\beta_i) & \cos(\beta_i) \end{bmatrix} \tag{2.25}$$

The individual elements then are

$$G_{i;11} = c^2 G_{i;xz} + s^2 G_{i;yz}$$

$$G_{i;12} = csG_{i;xz} - csG_{i;yz}$$

$$G_{i;22} = s^2 G_{i;xz} + c^2 G_{i;yz}, \text{ where } c = \cos(\beta_i),\ s = \sin(\beta_i)$$

2.3.2 Shear Coupling of Layers Is Not Considered

Now consider a plate consisting of n isotropic material layers, where the individual layers are not shear coupled. Each layer has the thickness t_i and minimum and maximum z-coordinate $z_{i;\min}, z_{i;\max}$.

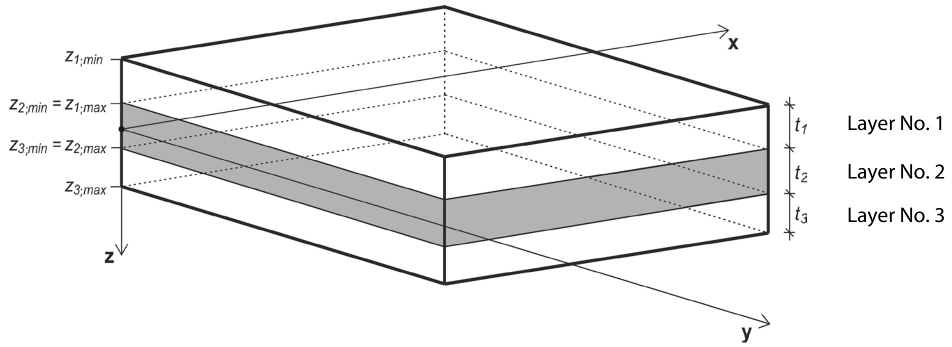


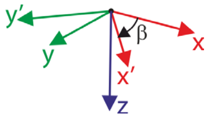
Figure 2.6: Layer scheme

The stiffness matrix for each layer is $\mathbf{d}'_i$ according to the following relation

$$\mathbf{d}'_i = \begin{bmatrix} d'_{i;11} & d'_{i;12} & 0 \\ & d'_{i;22} & 0 \\ \text{sym.} & & d'_{i;33} \end{bmatrix} = \begin{bmatrix} \dfrac{E_{i;x}}{1-\nu_{i;xy}^2 \dfrac{E_{i;y}}{E_{i;x}}} & \dfrac{\nu_{i;xy} E_{i;y}}{1-\nu_{i;xy}^2 \dfrac{E_{i;y}}{E_{i;x}}} & 0 \\ & \dfrac{E_{i;y}}{1-\nu_{i;xy}^2 \dfrac{E_{i;y}}{E_{i;x}}} & 0 \\ \text{sym.} & & G_{i;xy} \end{bmatrix} \quad i = 1,...,n \tag{2.26}$$

For isotropic materials, where $E_{i;x} = E_{i;y}$ applies, the stiffness matrix has the simplified form

$$\mathbf{d}'_i = \begin{bmatrix} d'_{i;11} & d'_{i;12} & 0 \\ & d'_{i;22} & 0 \\ \text{sym.} & & d'_{i;33} \end{bmatrix} = \begin{bmatrix} \dfrac{E_i}{1-\nu_i^2} & \dfrac{\nu_i E_i}{1-\nu_i^2} & 0 \\ & \dfrac{E_i}{1-\nu_i^2} & 0 \\ \text{sym.} & & G_i \end{bmatrix}, \quad G_i = \frac{E_i}{2\cdot(1+\nu_i)}, \quad i = 1,...,n \tag{2.27}$$



Because layers with orthotropic materials can be rotated arbitrarily by the angle β, it is necessary to transform stiffness matrices of individual layers to a uniform coordinate system x, y (local coordinate system of a surface).

$$\mathbf{d}_i = \begin{bmatrix} d_{i;11} & d_{i;12} & d_{i;13} \\ & d_{i;22} & d_{i;23} \\ \text{sym.} & & d_{i;33} \end{bmatrix} = \mathbf{T}_{3\times3;i}{}^{\mathrm{T}} \mathbf{d}'_i \mathbf{T}_{3\times3;i} \tag{2.28}$$

where

$$\mathbf{T}_{3\times3;i} = \begin{bmatrix} c^2 & s^2 & cs \\ s^2 & c^2 & -cs \\ -2cs & 2cs & c^2-s^2 \end{bmatrix}, \text{where } c = \cos(\beta_i),\ s = \sin(\beta_i) \tag{2.29}$$

The individual elements then are

$$\begin{aligned}
d_{i;11} &= c^4 d'_{i;11} + 2c^2s^2 d'_{i;12} + s^4 d'_{i;22} + 4c^2s^2 d'_{i;33} \\
d_{i;12} &= c^2s^2 d'_{i;11} + s^4 d'_{i;12} + c^4 d'_{i;12} + c^2s^2 d'_{i;22} - 4c^2s^2 d'_{i;33} \\
d_{i;13} &= c^3s d'_{i;11} + cs^3 d'_{i;12} - c^3s d'_{i;12} - cs^3 d'_{i;22} - 2c^3s d'_{i;33} + 2cs^3 d'_{i;33} \\
d_{i;22} &= s^4 d'_{i;11} + 2c^2s^2 d'_{i;12} + c^4 d'_{i;22} + 4c^2s^2 d'_{i;33} \\
d_{i;23} &= cs^3 d'_{i;11} + c^3s d'_{i;12} - cs^3 d'_{i;12} - c^3s d'_{i;22} + 2c^3s d'_{i;33} - 2cs^3 d'_{i;33} \\
d_{i;33} &= c^2s^2 d'_{i;11} - 2c^2s^2 d'_{i;12} + c^2s^2 d'_{i;22} + \left(c^2 - s^2\right)^2 d'_{i;33}
\end{aligned}$$

The global stiffness matrix is

$$\mathbf{D} = \begin{bmatrix} D_{11} & D_{12} & D_{13} & 0 & 0 & 0 & 0 & 0 \\ & D_{22} & D_{23} & 0 & 0 & 0 & 0 & 0 \\ & & D_{33} & 0 & 0 & 0 & 0 & 0 \\ & & & D_{44} & D_{45} & 0 & 0 & 0 \\ & & & & D_{55} & 0 & 0 & 0 \\ & & \text{sym.} & & & D_{66} & D_{67} & D_{68} \\ & & & & & & D_{77} & D_{78} \\ & & & & & & & D_{88} \end{bmatrix} \tag{2.30}$$

$$\begin{Bmatrix} m_x \\ m_y \\ m_{xy} \\ v_x \\ v_y \\ n_x \\ n_y \\ n_{xy} \end{Bmatrix} = \begin{bmatrix} D_{11} & D_{12} & D_{13} & 0 & 0 & 0 & 0 & 0 \\ & D_{22} & D_{23} & 0 & 0 & 0 & 0 & 0 \\ & & D_{33} & 0 & 0 & 0 & 0 & 0 \\ & & & D_{44} & D_{45} & 0 & 0 & 0 \\ & & & & D_{55} & 0 & 0 & 0 \\ & & \text{sym.} & & & D_{66} & D_{67} & D_{68} \\ & & & & & & D_{77} & D_{78} \\ & & & & & & & D_{88} \end{bmatrix} \begin{Bmatrix} \kappa_x \\ \kappa_y \\ \kappa_{xy} \\ \gamma_{xz} \\ \gamma_{yz} \\ \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} \tag{2.31}$$

Bending and torsion
Shear
Membrane

If angles β_i are multiples of 90°, the global stiffness matrix has the simplified form

$$\mathbf{D} = \begin{bmatrix} D_{11} & D_{12} & 0 & 0 & 0 & 0 & 0 & 0 \\ & D_{22} & 0 & 0 & 0 & 0 & 0 & 0 \\ & & D_{33} & 0 & 0 & 0 & 0 & 0 \\ & & & D_{44} & 0 & 0 & 0 & 0 \\ & & & & D_{55} & 0 & 0 & 0 \\ & & \text{sym.} & & & D_{66} & D_{67} & 0 \\ & & & & & & D_{77} & 0 \\ & & & & & & & D_{88} \end{bmatrix} \qquad (2.32)$$

Stiffness matrix elements (bending and torsion) [Nm]

$$D_{11} = \sum_{i=1}^{n} \frac{t_i^3}{12} d_{i;11} \qquad D_{12} = \sum_{i=1}^{n} \frac{t_i^3}{12} d_{i;12}$$

$$D_{22} = \sum_{i=1}^{n} \frac{t_i^3}{12} d_{i;22}$$

$$D_{33} = \sum_{i=1}^{n} \frac{t_i^3}{12} d_{i;33}$$

Stiffness matrix elements (membrane) [N/m]

$$D_{66} = \sum_{i=1}^{n} t_i \, d_{i;11} \qquad D_{67} = \sum_{i=1}^{n} t_i \, d_{i;12}$$

$$D_{77} = \sum_{i=1}^{n} t_i \, d_{i;22}$$

$$D_{88} = \sum_{i=1}^{n} t_i \, d_{i;33}$$

Stiffness matrix elements (shear) [N/m]

$$D_{44} = \sum_{i=1}^{n} \frac{5}{6} G_{i;11} t_i \qquad D_{45} = \sum_{i=1}^{n} \frac{5}{6} G_{i;12} t_i$$

$$D_{55} = \sum_{i=1}^{n} \frac{5}{6} G_{i;22} t_i$$

where

$$\mathbf{G}_i = \begin{bmatrix} G_{i;11} & G_{i;12} \\ \text{sym.} & G_{i;22} \end{bmatrix} = \mathbf{T}_{2\times2;i}{}^{\mathrm{T}} \mathbf{G}'_i \mathbf{T}_{2\times2;i} \qquad (2.33)$$

where

$$\mathbf{G}'_i = \begin{bmatrix} G_{xz} & 0 \\ 0 & G_{yz} \end{bmatrix} \text{ and } \mathbf{T}_{2\times2;i} = \begin{bmatrix} \cos(\beta_i) & \sin(\beta_i) \\ -\sin(\beta_i) & \cos(\beta_i) \end{bmatrix} \qquad (2.34)$$

The individual elements then are

$$G_{i;11} = c^2 G_{i;xz} + s^2 G_{i;yz}$$

$$G_{i;12} = csG_{i;xz} - csG_{i;yz}$$

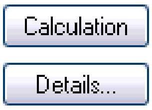
$$G_{i;22} = s^2 G_{i;xz} + c^2 G_{i;yz}$$, where $c = \cos(\beta_i)$, $s = \sin(\beta_i)$

3. Input Data

Input data of a structure is entered in windows, where calculation results are displayed as well.

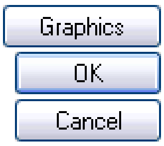
After you start RF-LAMINATE, a new window opens and on the left side a navigator is displayed, which contains all currently accessible windows.

The windows can be opened either by clicking their names in the RF-LAMINATE navigator, or you can browse through them by using the [<] and [>] buttons, displayed here on the left, or also by the keys [F2] and [F3].



By using the [Calculation] button, you can run the calculation after all input data is entered.

When you click the [Details...] button, dialog box appears where you can set the limit deformation, plate bending theory and other calculation parameters (see Chapter 4.1, page 33).



By clicking the [Graphics] button, you can display the RFEM workspace.

When you click the [OK] button, the entered data is saved and RF-LAMINATE is closed, whereas by clicking the [Cancel] button, you quit the module without saving any changes.

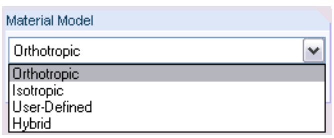
3.1 General Data

In Window 1.1 *General Data*, you select the surfaces and loads for the design. You can specify load cases, load combinations or result combinations for the ultimate limit state design and for the serviceability limit state design separately in the corresponding tabs.

Design of

To select surfaces that you want to design, there is a text box where you can enter numbers of individual surfaces. The *All* check box facilitates this selection. By the [Select Surfaces] button, you can select the surfaces graphically in the RFEM work window. You can quickly delete the list of preset surface numbers by using the [Delete Current List of Surfaces] button or select all the text by double-clicking in the text box and rewrite it manually.

Material Model



In this section, you select the material model. The following material models are available:

- *Orthotropic*
- *Isotropic*
- *User-Defined*
- *Hybrid*

For more information about material models see Chapter 2.2, page 10.

Comment

This text box is located in the bottom part of the window and you can quote your own notes or explanations for the current RF-LAMINATE case there.

Dlubal

3.1.1 Ultimate Limit State Tab

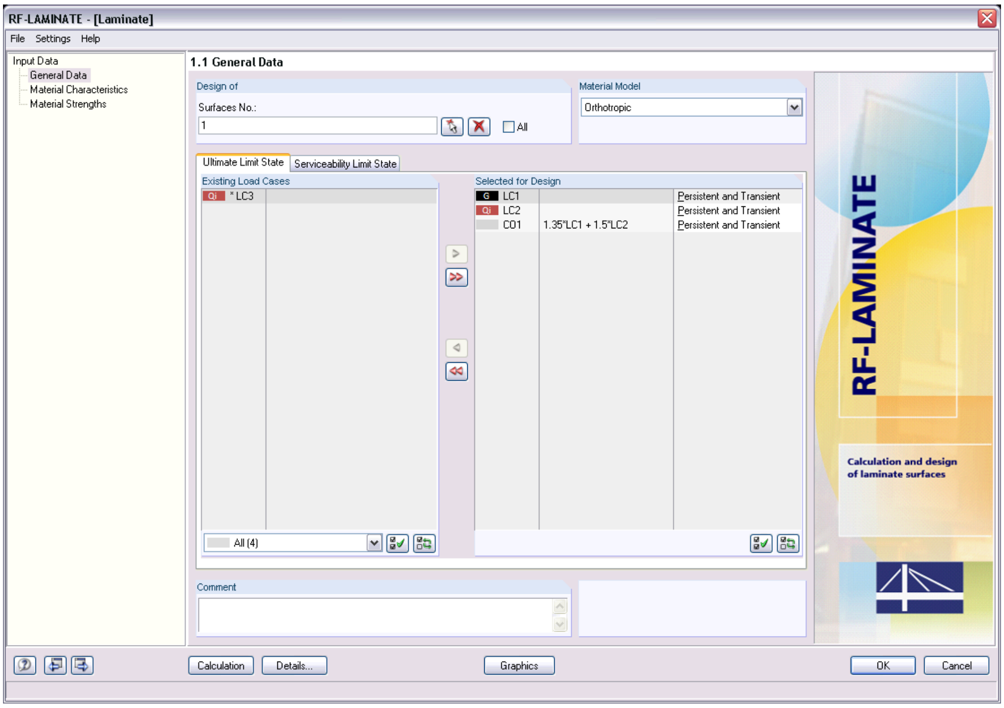


Figure 3.1: Window 1.1 *General Data* – the *Ultimate Limit State* tab

Existing Load Cases

In this section, the list of all load cases, load combinations and result combinations, which were created in RFEM, is displayed. By using the [▶] button, you can add the selected load cases, load combinations or result combinations to the list on the right *Selected for Design*. You can also select individual items by double-clicking. By clicking the [▶▶] button, you add all items to the list on the right at once.

You can also perform a multiple selection of load cases by using the [Ctrl] key, as usual in Windows. In this way, you can select and add to the list on the right several load cases at the same time.

In the bottom part of the section, you can use the buttons [Select All] and [Invert Selection], which facilitate the selection of requested load cases, load and result combinations together with the roll-out list.

If load cases or load combinations are marked with an asterisk (*), as you can see for example in figure 3.1 at LC3, you cannot design them. In such a case, no loads were assigned to these load cases or load combinations or they contain only imperfections.

Selected for Design

The selected loads for the design are listed in the right column. By clicking the [◀] button, you can remove the selected load cases, load and result combinations from the list. Here, you can select items with a double-click as well. By clicking the [◀◀] button, you delete the entire list.

In this section, you can also assign the design situation *Persistent and Transient* or *Accidental* to individual load cases, load and result combinations. The partial factors for material properties are assigned on the basis of this selection. The values of the partial factors can be modified for individual compositions in the *Details of Composition* dialog box. You can open this dialog box by clicking the [Edit Composition Details...] button in Window 1.2 or 1.3.

3.1.2 Serviceability Limit State Tab

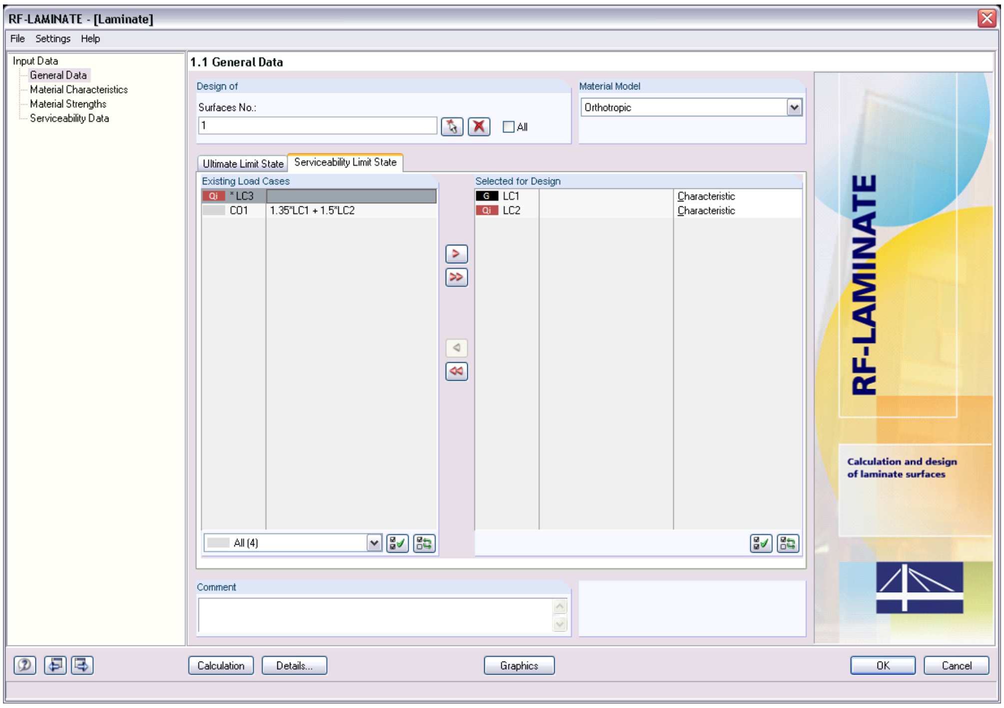


Figure 3.2: Window 1.1 *General Data* – the *Serviceability Limit State* tab

Existing Load Cases

In this section, the list of all load cases, load combinations or result combinations, which were created in RFEM, is displayed. After you add items to the list on the right *Selected for Design*, Window 1.5 *Serviceability Data* is displayed in the navigator.

Selected for Design

The addition of load cases, load and result combinations to the list for the design, and their removal is carried out the same way like in the previous tab (see Chapter 3.1.1, page 21).

In *Selected for Design*, the action combination is assigned to individual load cases, load and result combinations as well, either *Characteristic, Frequent* or *Quasi-permanent*. Different limit values for the deflection are applied on the basis of this selection. You can modify the limit values in the *Details* dialog box, in the *Design* tab. Open the *Details* dialog box by clicking the [Details...] button.

3.2 Material Characteristics

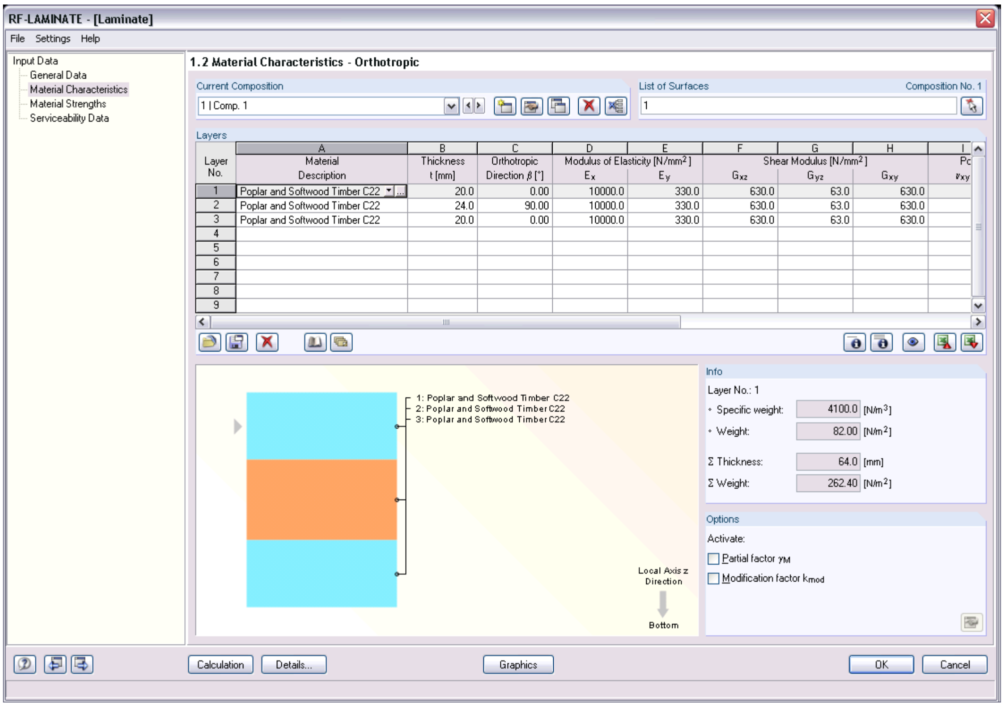


Figure 3.3: Window 1.2 *Material Characteristics*

In this window, you define layer compositions for individual surfaces of a structure. The selected composition is displayed in the *Current Composition* section. You can specify individual layers for each composition. You can create more compositions with various layers here. For each composition, you need to define corresponding surfaces in the section *List of Surfaces*.

The following buttons are available here:

Button	Name
	Create New Composition
	Edit Composition Details...
	Copy Current Composition
	Delete Current Composition
	Delete All Compositions
	Select Surfaces

Table 3.1: Buttons in the window *Material Characteristics*

For each composition, the dialog box *Details of Composition* is available. Open it by clicking the [Edit Composition Details...] button. Now, individual sections of the dialog box *Details of Composition* are described.

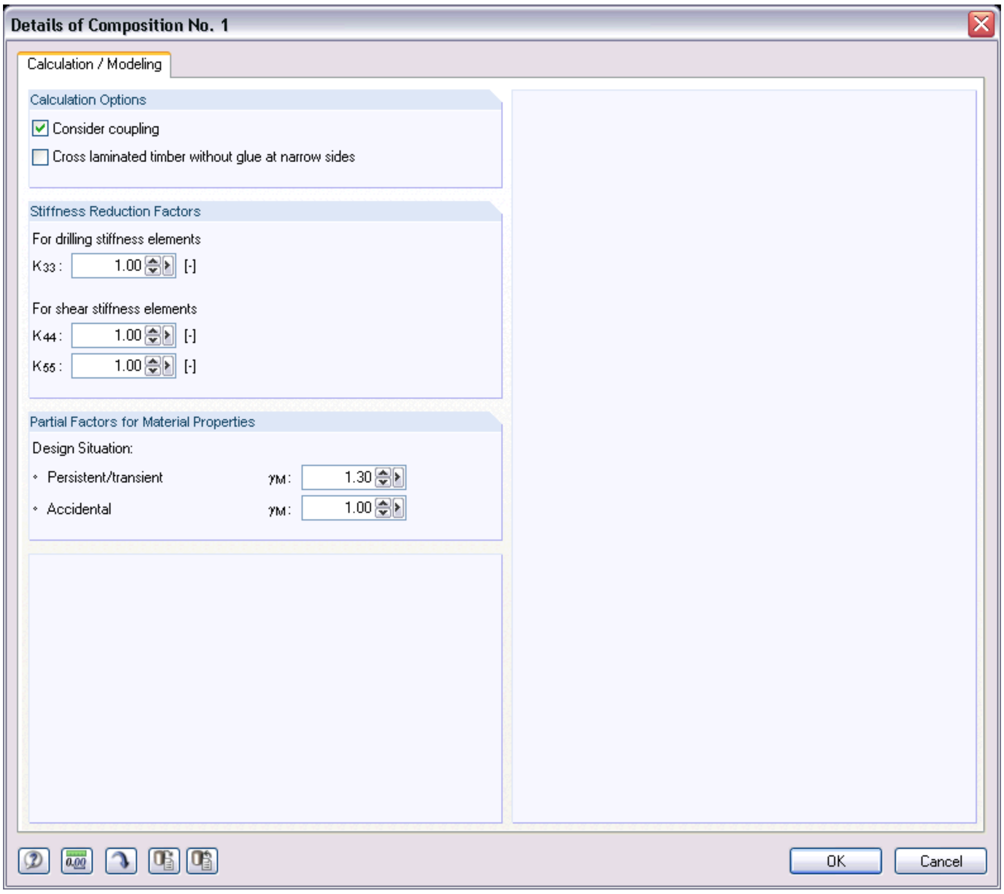


Figure 3.4: Dialog box *Details of Composition*

Calculation Options

In the section *Calculation Options*, the check box *Consider coupling* is selected by default, which means that the shear coupling of layers is considered. More information about both approaches is in Chapters 2.3.1 and 2.3.2.

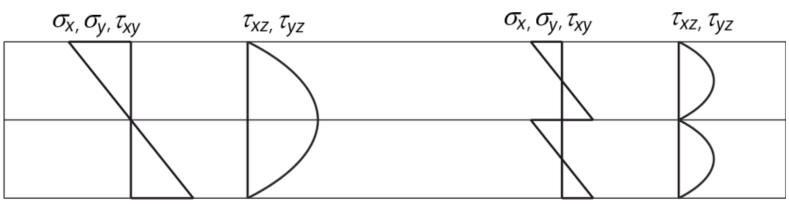


Figure 3.5: Basic stresses of the two-layer plate subjected to bending: with shear coupling of layers on the left, without shear coupling of layers on the right

Below, the check box *Cross laminated timber without glue at narrow sides* is available. This option is suitable for plates from boards without side glue, when it is considered that $E_y = 0$.

Stiffness Reduction Factors

In the section *Stiffness Reduction Factors*, you can reduce the torsional stiffness matrix element D_{33} by using the factor K_{33}. The correction is possible only for plates with the symmetric composition, for which the rotation angles are multiples of 90°. The correction is recommended in the standard ČSN 73 1702:2007, Chapter D.2.2 (5), page 127 (DIN 1052:2008, Chapter D.2.2 (5), page 175).

Next, you can reduce shear stiffness matrix elements D_{44} and D_{55} by using reduction factors K_{44} and K_{55}. The correction is possible only for plates, for which the rotation angles are multiples of 90°.

The stiffness matrix is then equal to (the case of the symmetric composition is shown here)

$$\begin{Bmatrix} m_x \\ m_y \\ m_{xy} \\ v_x \\ v_y \\ n_x \\ n_y \\ n_{xy} \end{Bmatrix} = \begin{bmatrix} D_{11} & D_{12} & 0 & 0 & 0 & 0 & 0 & 0 \\ & D_{22} & 0 & 0 & 0 & 0 & 0 & 0 \\ & & K_{33}D_{33} & 0 & 0 & 0 & 0 & 0 \\ & & & K_{44}D_{44} & 0 & 0 & 0 & 0 \\ & & & & K_{55}D_{55} & 0 & 0 & 0 \\ & & \text{sym.} & & & D_{66} & D_{67} & 0 \\ & & & & & & D_{77} & 0 \\ & & & & & & & D_{88} \end{bmatrix} \begin{Bmatrix} \kappa_x \\ \kappa_y \\ \kappa_{xy} \\ \gamma_{xz} \\ \gamma_{yz} \\ \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} \quad (2.35)$$

In Window 1.2 *Material Characteristics*, you can display the modified stiffness matrix by clicking the [Show extended stiffness matrix elements] button.

Partial Factors for Material Properties

If you select the check box *Partial factor* γ_M in Window 1.2, you can rewrite partial safety factors for material properties γ_M for offered design situations in this section. The design situations are assigned to individual load cases, load and result combinations in Window 1.1 *General Data*, in the tab *Ultimate Limit State* (see Chapter 3.1.1, page 21).

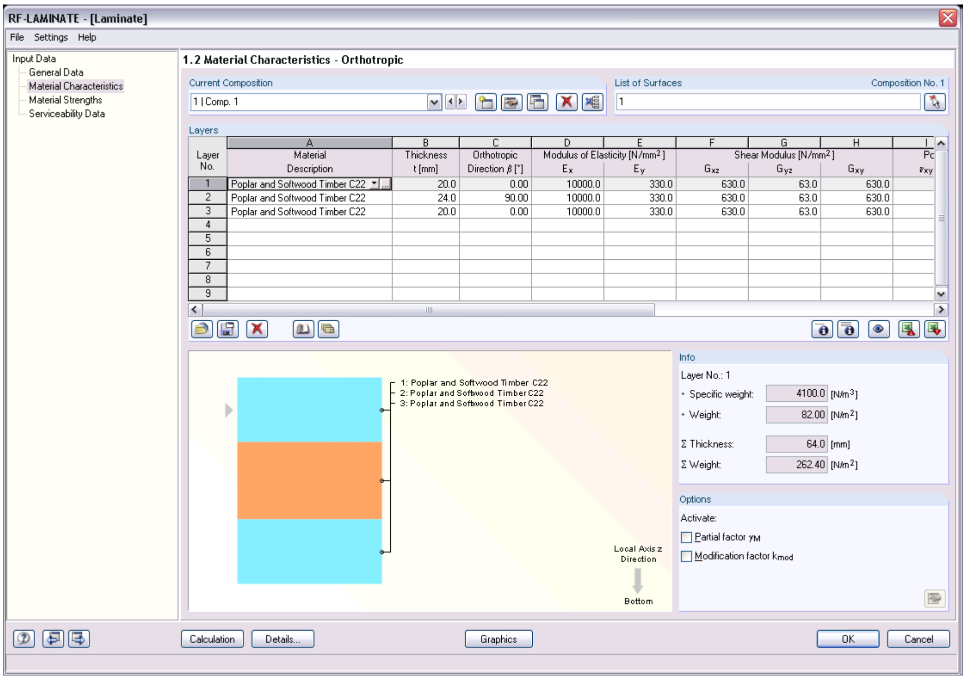
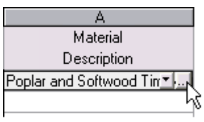


Figure 3.6: Window 1.2 *Material Characteristics*



In Window 1.2 *Material Characteristics*, in the *Layers* section, you enter individual layers for the current composition. You can select the relevant material directly from the library, where a large number of materials with all the necessary parameters are already predefined. You can open the material library by clicking the [Import Material from Library...] button or you can place the pointer to the corresponding line in column A and click the [...] button.

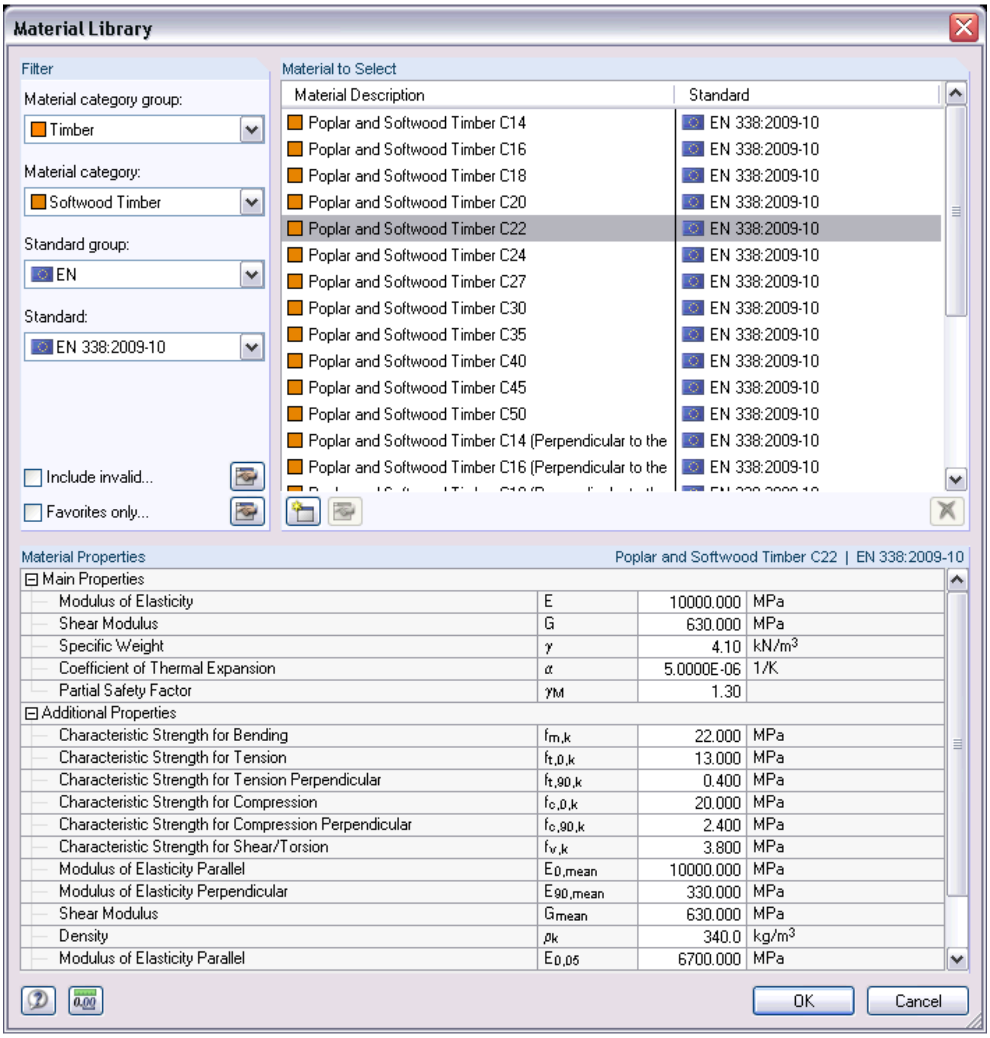
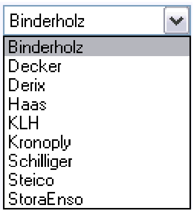


Figure 3.7: Material library

The material library is very extensive, therefore various filters are available. The offer of materials can be reduced in the list by using the criteria *Material category group*, *Material category*, *Standard group* and *Standard*. In the material library, in the list *Material to Select*, you can choose the required material and then check its parameters in the lower part of the dialog box. When you click the [OK] button, press the [↵] key or double-click a material, you import the material to Window 1.2 *Material Characteristics* of RF-LAMINATE. Then it is possible to adjust all material parameters directly in the module.



Furthermore, using the [Import Layers from Library...] button, you can enter the entire composition at once. There, you have the library of layers where you can select the *Producer*, *Type* and *Thickness*.

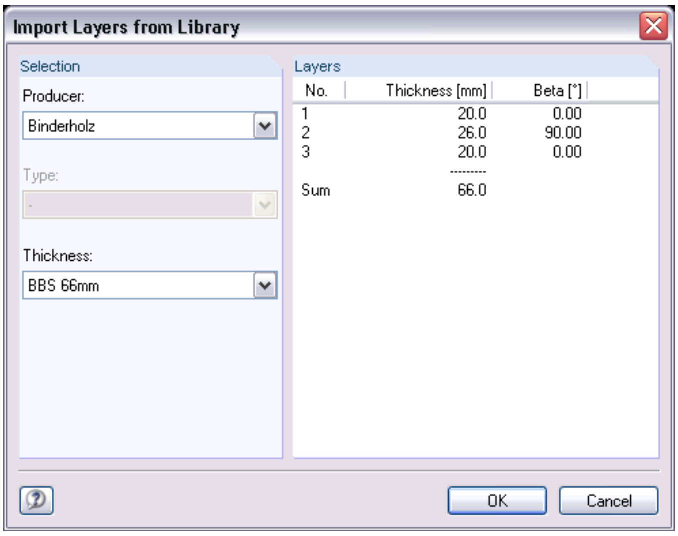


Figure 3.8: Dialog box *Import Layers from Library*

If you select the orthotropic material model in Window 1.1 *General Data*, the orthotropy direction according to the currently entered β is displayed in the RFEM graphic when you enter individual layers in Window 1.2 *Material Characteristics*, see the following picture. In this way, you can check your settings visually.

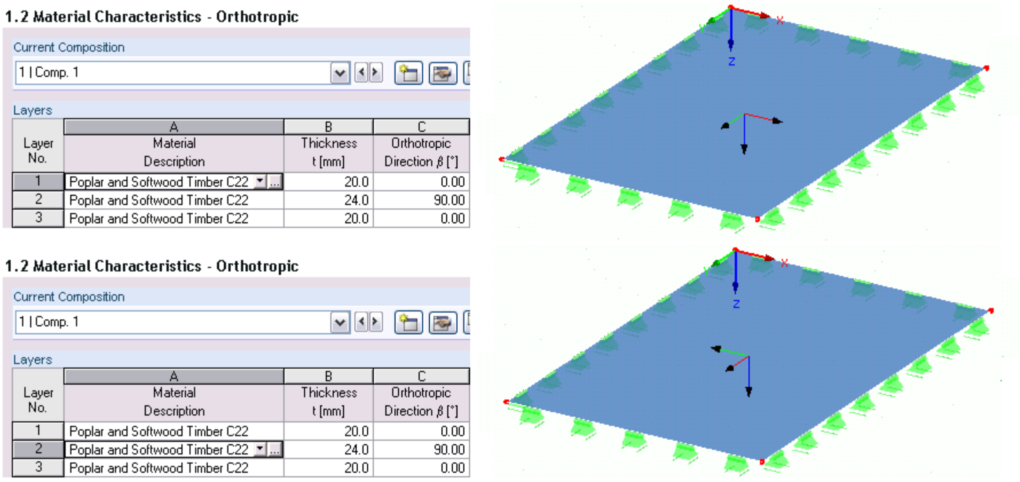


Figure 3.9: Display of the orthotropy direction β

Below the *Layers* section, a number of useful buttons are available. The buttons have the following functions:

Button	Name	Function
	Load Saved Layers...	Loads the composition that was saved before.
	Save Layers As...	Saves the entered composition from Window 1.2. This composition can be then reloaded to any other RF-LAMINATE composition by using the [Load Saved Layers...] button.
	Delete All Layers	Deletes all data from Window 1.2.
	Import Material from Library...	Opens the dialog box *Material Library*.

	Import Layers from Library...	Opens the dialog box *Import Layers from Library*.
	Show layer stiffness matrix elements	Displays elements of the stiffness matrix that is explained in Chapter 2.3.
	Show extended stiffness matrix elements	Displays elements of the global stiffness matrix that is explained in Chapter 2.3.
	Jump to graphic to change view	Opens the RFEM work window for a graphical check, but does not quit the RF-LAMINATE module.
	Export to Microsoft Excel or OpenOffice.org Calc	Exports the contents of the current table to MS Excel or to the Calc application from the OpenOffice.org package → Chapter 7.2, page 58.
	Import from Microsoft Excel or OpenOffice.org Calc	Imports the contents of a MS Excel table, the 1.2 Material Characteristics sheet, or from the Calc application of the OpenOffice.org package to Window 1.2.

Table 3.2: Buttons in the window *Material Characteristics*

In Window 1.2 *Material Characteristics*, under the table on the right, *Info* is provided about the specific weight and weight of the current layer, and about the total thickness and total weight of the selected composition.

Furthermore, in the *Options* section, there are check boxes to consider the partial factor γ_M and modification factor k_{mod} for the design. You can set the values of the partial factors for each composition separately in the dialog box *Details of Composition*. The values of the modification factors for all compositions are the same and you can edit them in the *Details* dialog box. If you select the check box *Modification factor* k_{mod}, see figure 3.10, another Window 1.4 *Load Duration and Service Class* is displayed in the navigator.

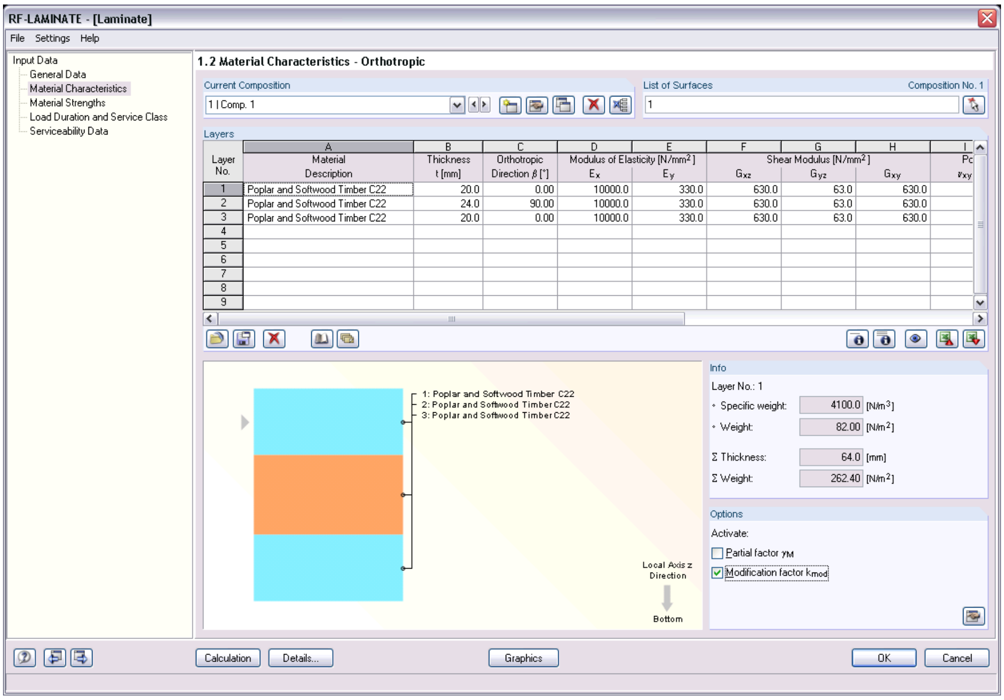


Figure 3.10: Window 1.2 *Material Characteristics* – the modification factor k_{mod}

3.3 Material Strengths

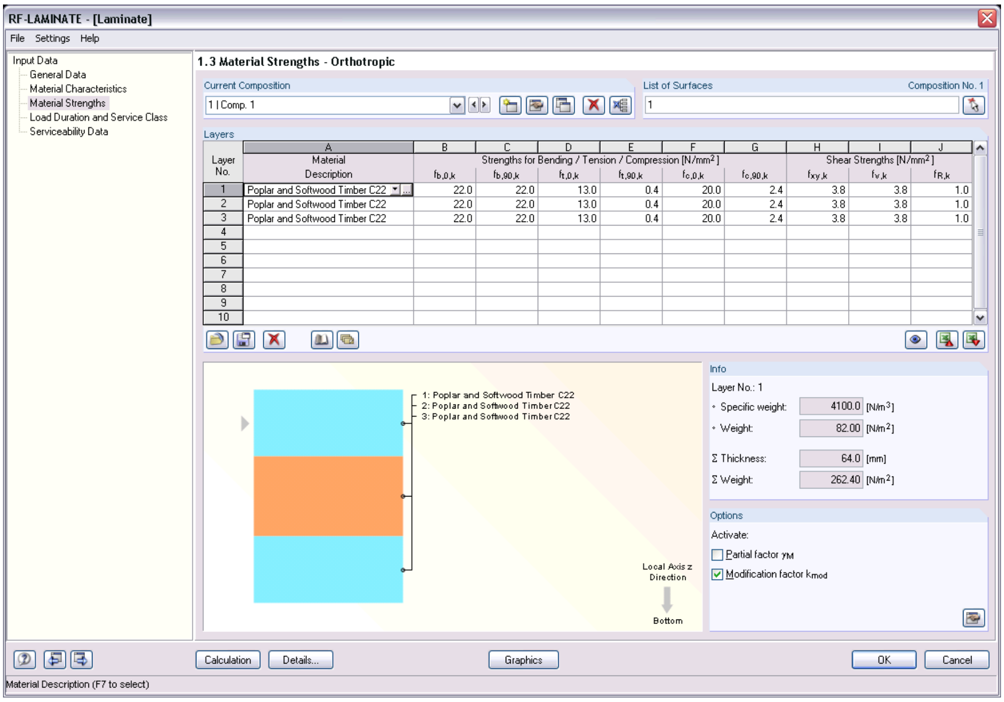


Figure 3.11: Window 1.3 *Material Strengths*

In Window 1.3, all characteristic strengths are displayed, imported for individual layers of a current composition from the material library. You can change these values simply by editing the values in Window 1.3.

Below the table, there are the same buttons as in the previous window, which are described in Chapter 3.2 on page 23.

Again, under the table on the right, *Info* is provided about the specific weight and weight of the current layer, and about the total thickness and total weight of the selected composition. There are also check boxes to consider the partial factor γ_M and the modification factor k_{mod}.

3.4 Load Duration and Service Class

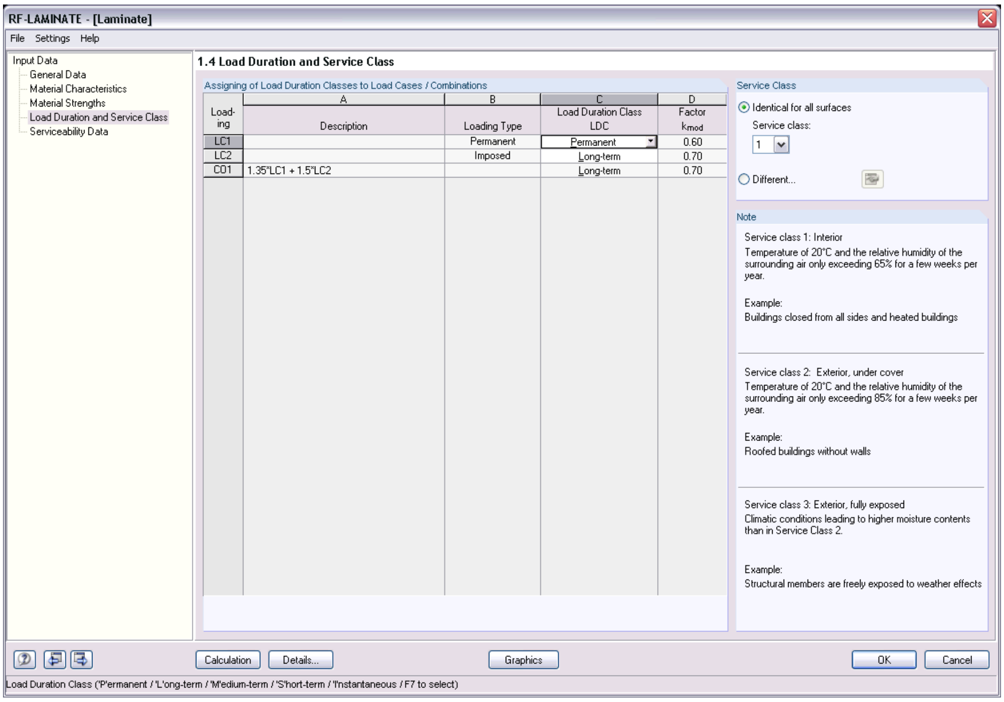


Figure 3.12: Window 1.4 *Load Duration and Service Class*

If you select the check box *Modification factor* k_{mod} in Window 1.2 *Material Characteristics* or in Window 1.3 *Material Strengths*, Window 1.4 *Load Duration and Service Class* is displayed. Here, all load cases and combinations that were selected for the design in Window 1.1 are displayed. In columns A and B, *Description* and *Loading Type* as defined in RFEM are shown. In column C, the *Load Duration Class - LDC* is set, where the assignment to various classes follows the standard EN 1995-1-1:2006. The classification of load combinations and result combinations adhere to the governing load automatically. When you fill in column C, the corresponding factor k_{mod} is supplied automatically. You can preset its value in the *Details* dialog box under the *Design* tab.

In the section *Service Class* , you can assign service classes to surfaces. The service classes are required for determination of the modification factor k_{mod}. The description of individual service classes is provided directly in the window and is based on the standard EN 1995-1-1:2006, Chapter 2.3.1.3.

The service class can be set the same for all surfaces, or you can select *Different*…. Then click the [Assign surface to corresponding service classes…] button and display the dialog box where you can assign individual surfaces to different service classes.

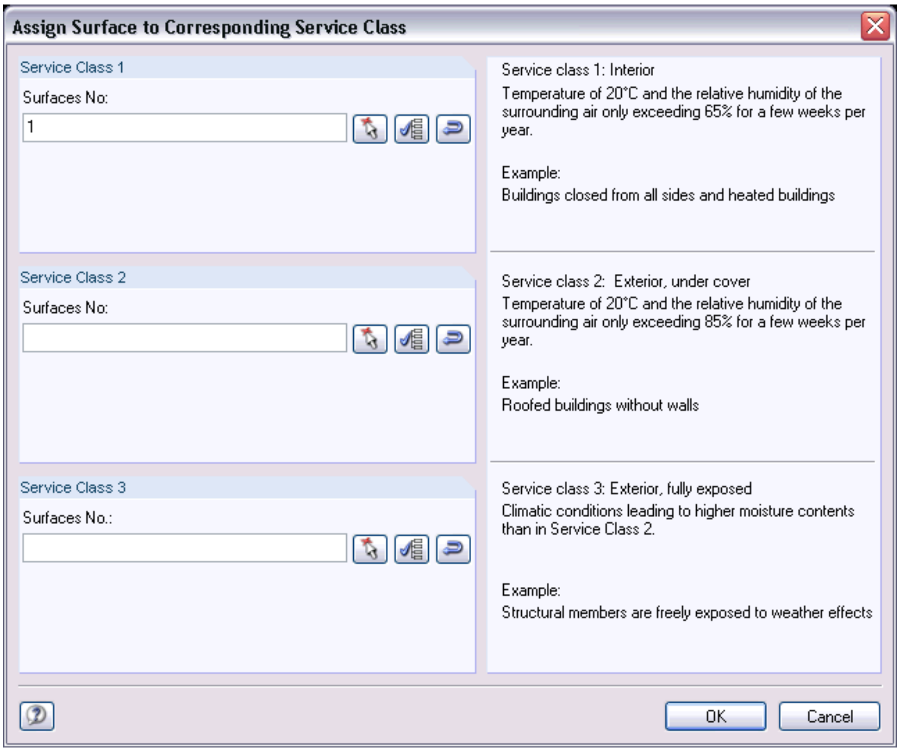


Figure 3.13: Dialog box *Assign Surface to Corresponding Service Class*

In the dialog box *Assign Surface to Corresponding Service Class*, the following buttons are available:

Button	Function
	Performs the graphical selection of surfaces that will be assigned to a corresponding service class.
	Assigns all surfaces to a corresponding service class.
	Assigns all surfaces that were not selected yet to a corresponding service class.

Table 3.3: Buttons in the dialog box *Assign Surface to Corresponding Service Class*

3.5 Serviceability Data

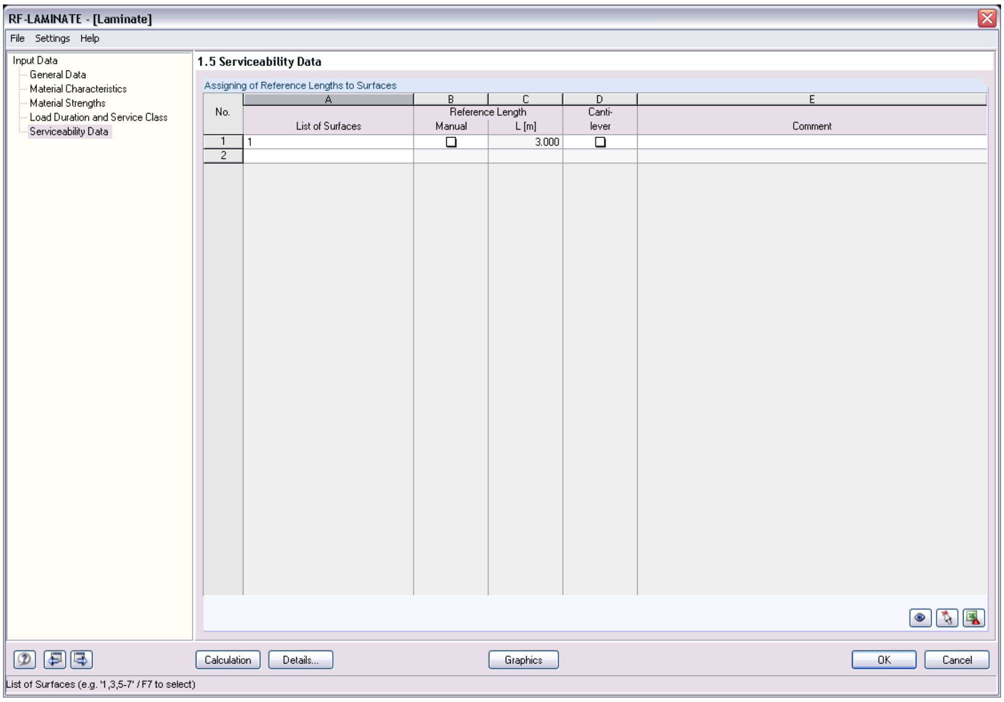
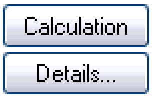


Figure 3.14: Window 1.5 *Serviceability Data*

The last input window is 1.5 *Serviceability Data*. In column A, you can enter individual surfaces. In column B, choose whether you want to enter *Reference Length L* manually or not. If you do not select the *Manual* check box, the length of the longest boundary line of the relevant surface is set automatically. In column D, select whether there is a *Cantilever* or not and it is possible to write your own *Comment* in column E.

All entered data is important for the correct application of limit deformations. You can check and if necessary modify these values in the *Details* dialog box, in the *Design* tab (see Chapter 4.1.1, page 34).

4. Calculation



Before you start the calculation (by using the button with the same name), it is necessary to check the detailed settings for the design. Open the relevant dialog box, which is described in the following chapter, by clicking the [Details…] button.

Before the calculation begins, it is checked whether the global stiffness matrix is positive-definite, see 9.2.

$$\mathbf{D} = \begin{bmatrix} D_{11} & D_{12} & D_{13} & 0 & 0 & D_{16} & D_{17} & D_{18} \\ & D_{22} & D_{23} & 0 & 0 & \text{sym.} & D_{27} & D_{28} \\ & & D_{33} & 0 & 0 & \text{sym.} & \text{sym.} & D_{38} \\ & & & D_{44} & D_{45} & 0 & 0 & 0 \\ & & & & D_{55} & 0 & 0 & 0 \\ & & \text{sym.} & & & D_{66} & D_{67} & D_{68} \\ & & & & & & D_{77} & D_{78} \\ & & & & & & & D_{88} \end{bmatrix} \qquad (4.1)$$

The calculation then runs globally with the entire structure that was modeled in RFEM.

4.1 Details

For the reason of clarity, the *Details* dialog box is divided into the following tabs:

- *Design*
- *Stresses*
- *Results*

The following buttons are common for all the tabs:

Button	**Name**	**Function**
	Help (F1)	Calls up the online help for RF-LAMINATE.
	Units and Decimal Places...	Opens the dialog box *Units and Decimal Places*, where you can set units for RF-LAMINATE.
	Reset Dlubal Standard Values	Sets all values in the *Details* dialog box to the original Dlubal values.
	Default	Sets all parameters in the *Details* dialog box according to the default setting that was saved previously.
	Set As Default	Saves the current setting as default. You can then reload this setting to any RF-LAMINATE case by using the previous button [Default].

Table 4.1: Buttons in the *Details*

4.1.1 Design Tab

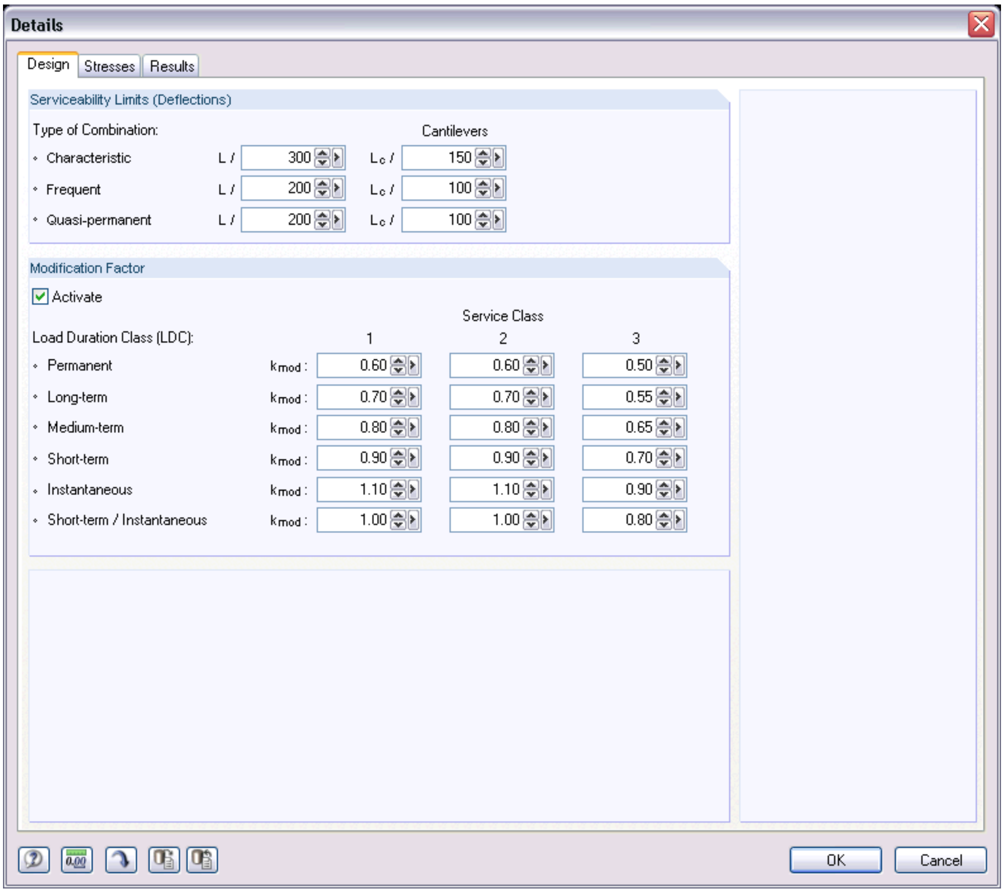


Figure 4.1: Dialog box *Details* – the *Design* tab

Serviceability Limits (Deflections)

The limit values of allowable deflections are set in six text boxes. In this way, you can enter specific data for various action combinations (characteristic, frequent and quasi-permanent) and for surfaces supported on both sides or one side only. The action combinations are assigned to load cases, load combinations and result combinations in the tab *Serviceability Limit State* in Window 1.1 *General Data* (see Chapter 3.1.2, page 22). Reference lengths L are entered for individual surfaces in Window 1.5 *Serviceability Data* (see Chapter 3.5, page 32).

Modification Factor

Values of the modification factor k_{mod} are provided there according to the *Load Duration Class (LDC)* and *Service Class*. The values follow the standard EN 1995-1-1:2006, Table 3.1. You can rewrite numeric values of the modification factor. The modification factor is then assigned to individual load cases according to the corresponding load duration class in Window 1.4 *Load Duration and Service Class* (see Chapter 3.4, page 30).

4.1.2 Stresses Tab

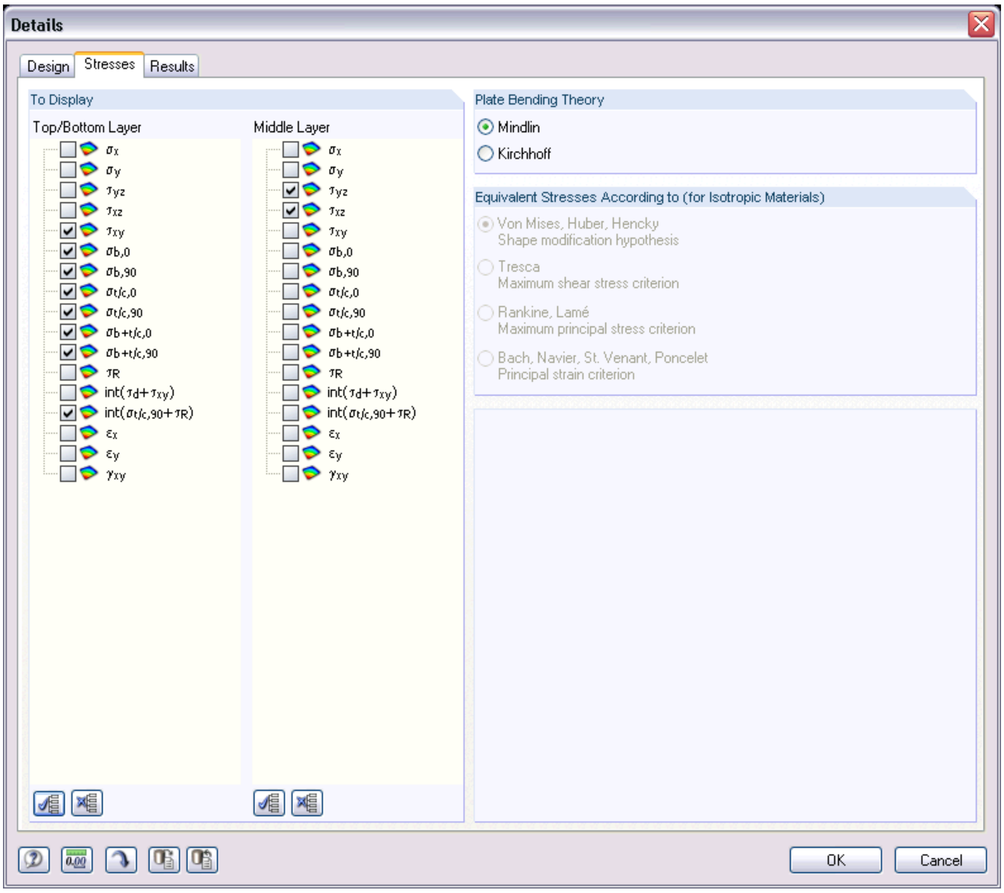


Figure 4.2: Dialog box *Details* – the *Stresses* tab

To Display

In this section, you can choose which stresses you want to display in the result windows by using check boxes. The stresses are separated for middle layers and for top/bottom layers. The buttons [Select All] and [Deselect All] are available to facilitate the selection.

Basic stresses $\sigma_x, \sigma_y, \tau_{xy}, \tau_{xz}, \tau_{yz}$ are calculated by the finite element method in RFEM, other stresses are then calculated from these basic stresses in the RF-LAMINATE module. Formulas valid for a single layer plate are introduced in the following table.

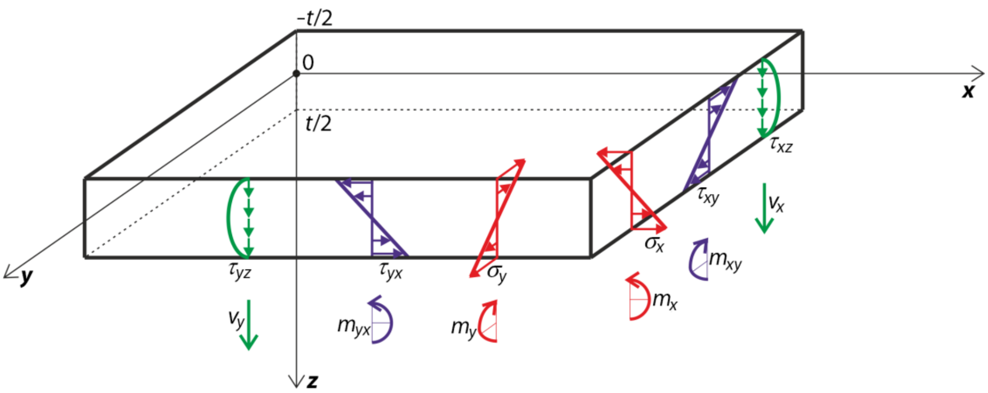


Figure 4.3: Basic stresses and the sign convention for the single layer plate subjected to bending

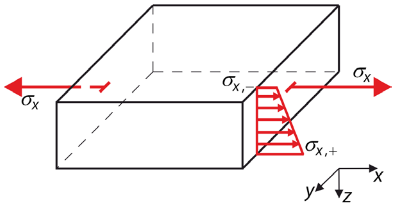
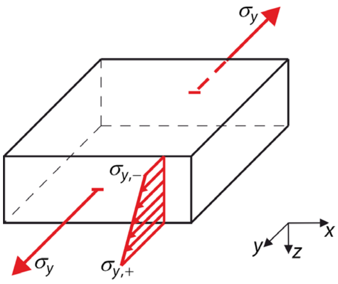
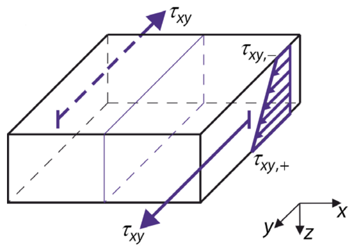
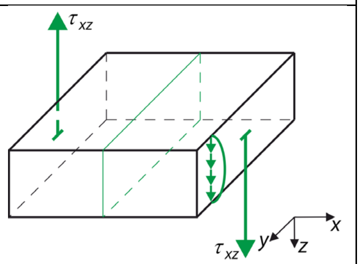
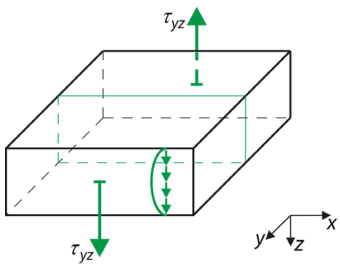
σ_x	Normal stress in x-direction • the stress on the positive surface side is equal to $\sigma_{x,+} = \frac{n_x}{t} + \frac{6m_x}{t^2}$, where t = plate thickness • the stress on the negative surface side is equal to $\sigma_{x,-} = \frac{n_x}{t} - \frac{6m_x}{t^2}$ 
σ_y	Normal stress in y-direction • the stress on the positive surface side is equal to $\sigma_{y,+} = \frac{n_y}{t} + \frac{6m_y}{t^2}$ • the stress on the negative surface side is equal to $\sigma_{y,-} = \frac{n_y}{t} - \frac{6m_y}{t^2}$ 
τ_{xy}	Shear stress in xy plane • the stress on the positive surface side is equal to $\tau_{xy,+} = \frac{n_{xy}}{t} + \frac{6m_{xy}}{t^2}$ • the stress on the negative surface side is equal to $\tau_{xy,-} = \frac{n_{xy}}{t} - \frac{6m_{xy}}{t^2}$ 
τ_{xz}	Shear stress in xz plane • in the plate center $\tau_{xz} = \frac{3}{2}\frac{v_x}{t}$ 
τ_{yz}	Shear stress in yz plane • in the plate center $\tau_{yz} = \frac{3}{2}\frac{v_y}{t}$ 

Table 4.2: Basic stresses

Generally, the stresses in individual layers are calculated from the total internal strains of the plate

$$\boldsymbol{\varepsilon}_{\text{tot}}^{\text{T}} = \left\{ \frac{\partial \varphi_y}{\partial x},\ -\frac{\partial \varphi_x}{\partial y},\ \frac{\partial \varphi_y}{\partial y} - \frac{\partial \varphi_x}{\partial x},\ \frac{\partial w}{\partial x} + \varphi_y,\ \frac{\partial w}{\partial y} - \varphi_x,\ \frac{\partial u}{\partial x},\ \frac{\partial v}{\partial y},\ \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right\} \tag{4.2}$$

The strains in individual layers are calculated by using the relation

$$\boldsymbol{\varepsilon}(z) = \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} = \begin{Bmatrix} \frac{\partial u}{\partial x} \\ \frac{\partial v}{\partial y} \\ \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \end{Bmatrix} + z \begin{Bmatrix} \frac{\partial \varphi_y}{\partial x} \\ -\frac{\partial \varphi_x}{\partial y} \\ \frac{\partial \varphi_y}{\partial y} - \frac{\partial \varphi_x}{\partial x} \end{Bmatrix} \tag{4.3}$$

where z is the coordinate in z -direction, where the stress value is requested. If there is for example i – th layer, the stress is calculated by using the relation

$$\boldsymbol{\sigma}(z) = \mathbf{d}_i \boldsymbol{\varepsilon}(z) \tag{4.4}$$

where $\mathbf{d}_i$ is the partial stiffness matrix of the i – th layer.

Now, it is necessary to divide stresses according to the material model - isotropic or orthotropic.

Isotropic material model

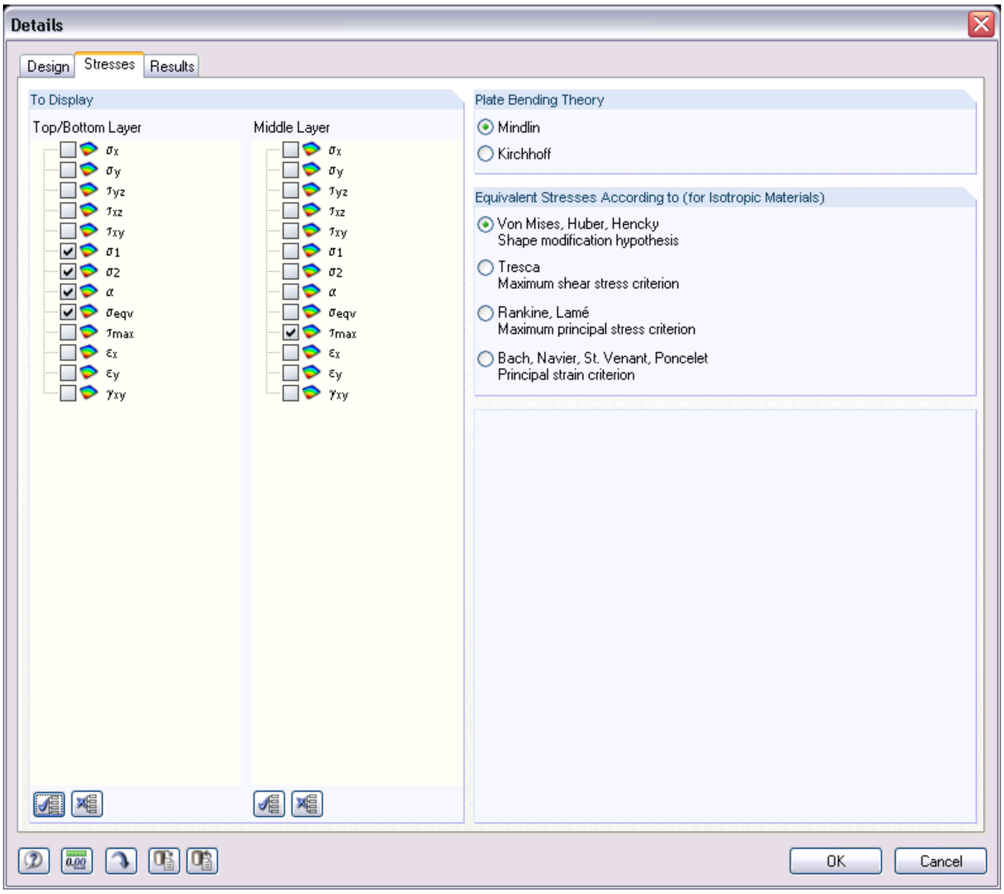


Figure 4.4: Dialog box *Details* – the *Stresses* tab for the isotropic material model

The effect of the transversal shear stresses is expressed by the quantity:

$\tau_{\max}$	Maximum transversal shear stress $\tau_{\max} = \sqrt{\tau_{yz}^2 + \tau_{xz}^2}$

Table 4.3: Maximum transversal shear stress

The relations for the calculation of principal and equivalent stresses are introduced in the following table. The effect of the shear stresses is neglected in the formulas τ_{xz} and τ_{yz}.

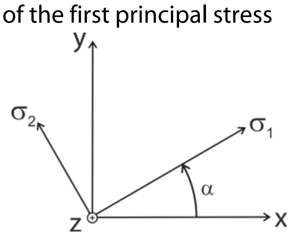
σ_1	Principal stress $\sigma_1 = \dfrac{\sigma_x + \sigma_y + \sqrt{\left(\sigma_x - \sigma_y\right)^2 + 4\tau_{xy}^2}}{2}$
σ_2	Principal stress $\sigma_2 = \dfrac{\sigma_x + \sigma_y - \sqrt{\left(\sigma_x - \sigma_y\right)^2 + 4\tau_{xy}^2}}{2}$
α	The angle between the local axis x and the direction of the first principal stress $\alpha = \dfrac{1}{2}\operatorname{atan2}\left(2\tau_{xy},\ \sigma_x - \sigma_y\right), \quad \alpha \in (-90^\circ, 90^\circ\rangle$  Function atan2 is implemented in RFEM this way $\operatorname{atan2}(y,x) = \begin{cases} \arctan\dfrac{y}{x} & x > 0 \\ \arctan\dfrac{y}{x} + \pi & y \ge 0, x < 0 \\ \arctan\dfrac{y}{x} - \pi & y < 0, x < 0 \\ +\dfrac{\pi}{2} & y > 0, x = 0 \\ -\dfrac{\pi}{2} & y < 0, x = 0 \\ 0 & y = 0, x = 0 \end{cases}$
σ_{eqv}	Equivalent stress according to von Mises, Huber, Hencky (Shape modification hypothesis) $\sigma_{eqv} = \sqrt{\sigma_x^2 + \sigma_y^2 - \sigma_x\sigma_y + 3\tau_{xy}^2}$
	Equivalent stress according to Tresca (Maximum shear stress criterion) $\sigma_{eqv} = \max\left[\sqrt{\left(\sigma_x - \sigma_y\right)^2 + 4\tau_{xy}^2},\ \dfrac{\left\|\sigma_x + \sigma_y\right\| + \sqrt{\left(\sigma_x - \sigma_y\right)^2 + 4\tau_{xy}^2}}{2}\right]$
	Equivalent stress according to Rankine, Lamé (Maximum principal stress criterion) $\sigma_{eqv} = \dfrac{\left\|\sigma_x + \sigma_y\right\| + \sqrt{\left(\sigma_x - \sigma_y\right)^2 + 4\tau_{xy}^2}}{2}$
	Equivalent stress according to Bach, Navier, St. Venant, Poncelet (Principal strain criterion) $\sigma_{eqv} = \max\left[\dfrac{1-\nu}{2}\left\|\sigma_x + \sigma_y\right\| + \dfrac{1+\nu}{2}\sqrt{\left(\sigma_x - \sigma_y\right)^2 + 4\tau_{xy}^2},\ \nu\left\|\sigma_x + \sigma_y\right\|\right]$

Table 4.4: Stresses for the isotropic material model

Orthotropic material model

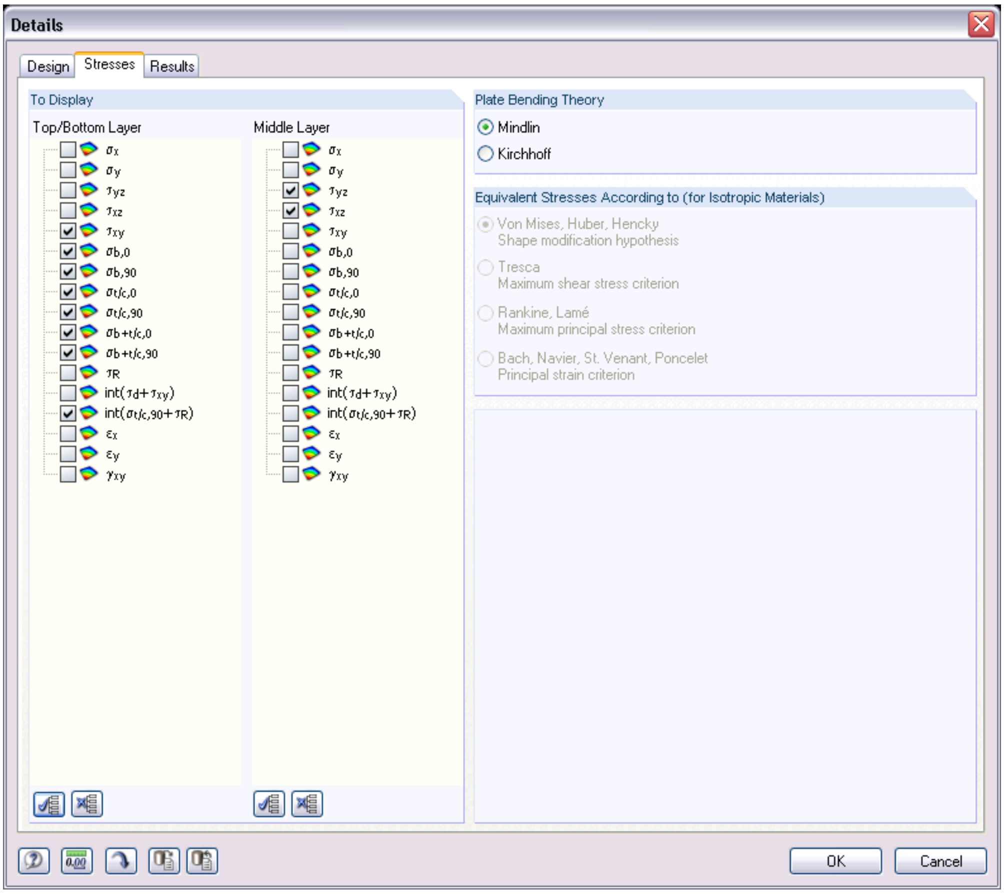
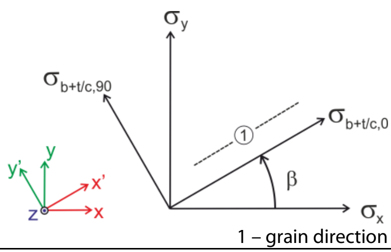


Figure 4.5: Dialog box *Details* – the *Stresses* tab for the orthotropic material model

$\sigma_{b+t/c,0}$	Normal stress along the grain $\sigma_{b+t/c,0} = \sigma_x \cos^2\beta + \tau_{xy} \sin 2\beta + \sigma_y \sin^2\beta$ 
$\sigma_{b+t/c,90}$	Normal stress perpendicular to the grain $\sigma_{b+t/c,90} = \sigma_x \sin^2\beta - \tau_{xy} \sin 2\beta + \sigma_y \cos^2\beta$
$\sigma_{t/c,0}$	Tension/compression component of the normal stress along the grain $\sigma_{t/c,0} = \frac{\sigma_{b+t/c,0(top)} + \sigma_{b+t/c,0(middle)} + \sigma_{b+t/c,0(bottom)}}{3}$
$\sigma_{t/c,90}$	Tension/compression component of the normal stress perpendicular to the grain $\sigma_{t/c,90} = \frac{\sigma_{b+t/c,90(top)} + \sigma_{b+t/c,90(middle)} + \sigma_{b+t/c,90(bottom)}}{3}$
$\sigma_{b,0}$	Bending component of the normal stress along the grain $\sigma_{b,0} = \sigma_{b+t/c,0} - \sigma_{t/c,0}$
$\sigma_{b,90}$	Bending component of the normal stress perpendicular to the grain $\sigma_{b,90} = \sigma_{b+t/c,90} - \sigma_{t/c,90}$

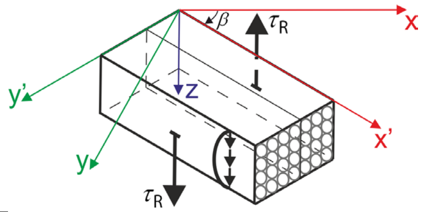
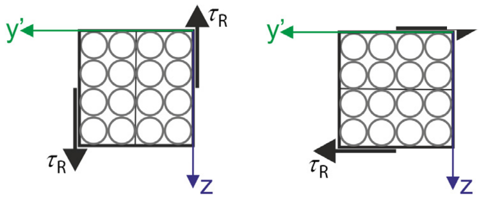
τ_R	Rolling shear stress $\tau_R = -\tau_{xz} \sin\beta + \tau_{yz} \cos\beta$  

Table 4.5: Stresses for the orthotropic material model

It is important to realize that the stresses $\sigma_{b+t/c,0}$, $\sigma_{b+t/c,90}$, $\sigma_{t/c,0}$, $\sigma_{t/c,90}$, $\sigma_{b,0}$, $\sigma_{b,90}$, τ_R are expressed in the coordinate system of the grain x', y', z, which can be rotated differently in every layer in general. That is why discontinuities in stress values of individual layers can occur at their boundaries. The transformation formulas for these stresses are introduced in Chapter 5.1.

The normal stress contains the bending component and the tension/compression component for individual layers.

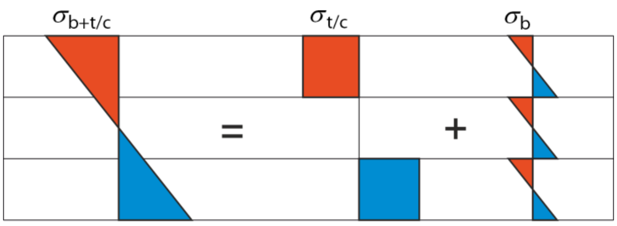


Figure 4.6: Decomposition of the normal stress into the tension/compression component and the bending component

Plate Bending Theory

For surfaces, you can choose the bending theory according to:

- Mindlin
- Kirchhoff

The shear strain is considered for calculation according to the Mindlin theory, but not according to the Kirchhoff theory. The bending theory according to Mindlin is suitable for massive plates, the bending theory according to Kirchhoff for relatively thin plates.

Because the shear stresses τ_{xz} and τ_{yz} are not determined exactly in the Kirchhoff's theory, they are calculated from equilibrium conditions. You can calculate them by using the following relations

$$\tau_{xz,max} = \frac{3}{2}\frac{v_x}{t} = 1.5\frac{v_x}{t} \quad (4.5)$$

$$\tau_{yz,max} = \frac{3}{2}\frac{v_y}{t} = 1.5\frac{v_y}{t} \quad (4.6)$$

Equivalent Stresses According to (for Isotropic Materials)

You can determine the equivalent stresses in four different ways:

Equivalent stress according to von Mises, Huber, Hencky
(Shape modification hypothesis)
This hypothesis is known as HMH, the VON MISES hypothesis, or as the energy criterion. The equivalent stress according to this hypothesis is calculated by using the relation

$$\sigma_{\mathrm{eqv}} = \sqrt{\sigma_x^2 + \sigma_y^2 - \sigma_x \sigma_y + 3\tau_{xy}^2} \tag{4.7}$$

Equivalent stress according to Tresca
(Maximum shear stress theory)
This equivalent stress is generally defined by using the relation

$$\sigma_{\mathrm{eqv}} = \max\left(\left|\sigma_1 - \sigma_2\right|,\ \left|\sigma_1 - \sigma_3\right|,\ \left|\sigma_2 - \sigma_3\right|\right) \tag{4.8}$$

which is on the condition $\sigma_3 = 0$, simplified to

$$\sigma_{\mathrm{eqv}} = \max\left(\left|\sigma_1 - \sigma_2\right|,\ \left|\sigma_1\right|,\ \left|\sigma_2\right|\right) \tag{4.9}$$

and the resulting equation

$$\sigma_{\mathrm{eqv}} = \max\left[\sqrt{\left(\sigma_x - \sigma_y\right)^2 + 4\tau_{xy}^2},\ \frac{\left|\sigma_x + \sigma_y\right| + \sqrt{\left(\sigma_x - \sigma_y\right)^2 + 4\tau_{xy}^2}}{2}\right] \tag{4.10}$$

Equivalent stress according to Rankine, Lamé
(Maximum principal stress criterion)
This hypothesis is known as the normal stress hypothesis or as the equivalent stress according to RANKINE. The Rankine's stress is generally defined as the maximum of absolute values of principal stresses

$$\sigma_{\mathrm{eqv}} = \max\left(\left|\sigma_1\right|,\ \left|\sigma_2\right|,\ \left|\sigma_3\right|\right) \tag{4.11}$$

which is on the condition $\sigma_3 = 0$, simplified to

$$\sigma_{\mathrm{eqv}} = \max(\left|\sigma_1\right|,\ \left|\sigma_2\right|) \tag{4.12}$$

and the resulting equation

$$\sigma_{\mathrm{eqv}} = \frac{\left|\sigma_x + \sigma_y\right| + \sqrt{\left(\sigma_x - \sigma_y\right)^2 + 4\tau_{xy}^2}}{2} \tag{4.13}$$

Equivalent stress according to Bach, Navier, St. Venant, Poncelet
(Principal strain criterion)
The principal deformation hypothesis or the equivalent stress according to BACH. The equivalent strain is defined as the maximum strain that originates in the direction of principal stresses

$$\sigma_{\mathrm{eqv}} = \max\left(\left|\sigma_1 - \nu\left(\sigma_2 + \sigma_3\right)\right|,\ \left|\sigma_2 - \nu\left(\sigma_1 + \sigma_3\right)\right|,\ \left|\sigma_3 - \nu\left(\sigma_1 + \sigma_2\right)\right|\right) \tag{4.14}$$

which is on the condition $\sigma_3 = 0$, simplified to

$$\sigma_{\mathrm{eqv}} = \max\left(\left|\sigma_1 - \nu\sigma_2\right|,\ \left|\sigma_2 - \nu\sigma_1\right|,\ \nu\left|\sigma_1 + \sigma_2\right|\right) \tag{4.15}$$

and the resulting equation

$$\sigma_{\mathrm{eqv}} = \max\left[\frac{1-\nu}{2}\left|\sigma_x + \sigma_y\right| + \frac{1+\nu}{2}\sqrt{\left(\sigma_x - \sigma_y\right)^2 + 4\tau_{xy}^2},\ \nu\left|\sigma_x + \sigma_y\right|\right] \tag{4.16}$$

In the formulas for the equivalent stress, influence of shear stresses τ_{xz} and τ_{yz}. is neglected.

4.1.1 Results Tab

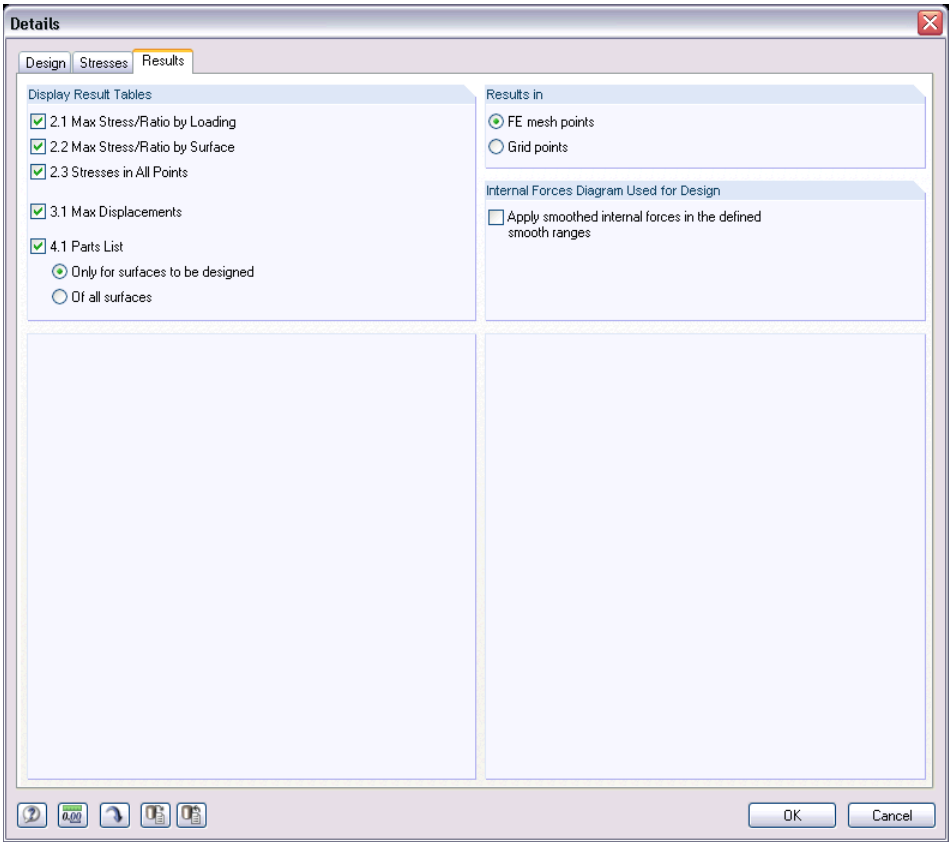


Figure 4.7: Dialog box *Details* – the *Results* tab

Display Result Tables

In this section, you can specify which result windows you want to display. There are various windows with result values of stresses, displacements and parts lists, which you can switch on or off by using the relevant check boxes.

Individual result windows are described in Chapter 5. *Results* on page 44.

Results in

Stresses and displacements are displayed in all FE mesh points by default. However, you have the option to display results in grid points that you have defined and that are saved in RFEM as a surface property (see the RFEM manual, Chapter 8.12).

In the case of smaller surfaces, the default grid point spacing of 0.5 m can cause that only a small amount of points exists on a surface or even one point, the initial grid point. In such a case, you generally do not get any maximum values because the resulting grid is too coarse. The spacing of grid points then should be adapted to the surface dimensions in RFEM, in order to create more grid points.

Internal Forces Diagram Used for Design

In this section, you can select the check box *Apply smoothed internal forces in the defined smooth ranges* and thus use the smooth ranges that were specified in RFEM during the stress calculation in RF-LAMINATE.

4.2 Start Calculation

In all RF-LAMINATE input windows, you can start the calculation by clicking the [Calculation] button.

You can also start the RF-LAMINATE calculation from the RFEM user interface. Add-on modules are displayed in RFEM in the dialog box *To Calculate*, in the tab *Load Cases / Combinations / Module Cases*. You can open this dialog box in RFEM by using the command from the main menu

Calculate → To Calculate....

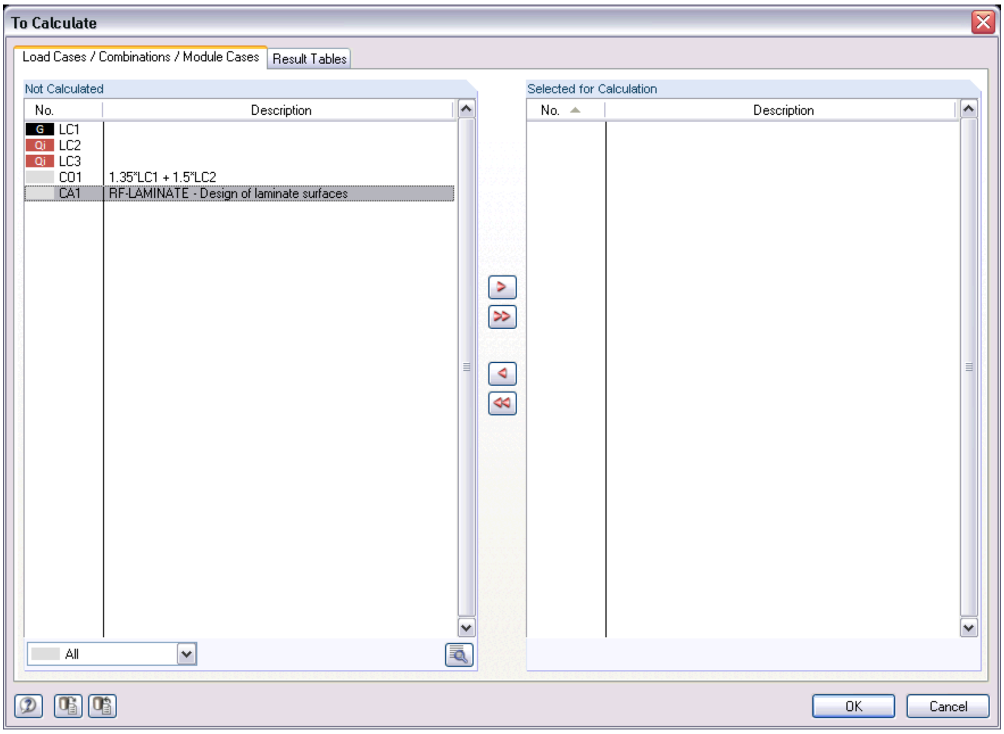


Figure 4.8: Dialog box *To Calculate* in RFEM

If an RF-LAMINATE design case is missing in the *Not Calculated* list, you have to select *Add-on Modules* in the bottom part of the dialog box.

By using the [▶] button, add the selected design case to the list on the right. Then, start the calculation by using the [OK] button.

You can also start the calculation of an RF-LAMINATE design case from the toolbar. You set the requested design case in the list and then click the [Show Results] button.

Figure 4.9: Direct calculation of the RF-LAMINATE design case in RFEM

5. Results

Window 2.1 *Max Stress/Ratio by Loading* is displayed immediately after the calculation ends. In the *Details* dialog box, in the *Results* tab, you can specify which result windows you want to display (see Chapter 4.1.1 on page 42).

All result windows are accessible from the RF-LAMINATE navigator. To browse through individual windows, you can use the [<] and [>] buttons, displayed here on the left margin, or the function keys [F2] and [F3].

By clicking the [OK] button, you save all input data and results and close RF-LAMINATE, whereas by clicking the [Cancel] button, you quit the module without saving the data.

In the result windows, a number of useful buttons is available. The buttons have the following functions:

Button	Name	Function
	Jump to graphic to change view	Opens the RFEM work window for a graphical check, but does not quit the RF-LAMINATE module.
	Select	Opens the RFEM work window for a graphical selection of surfaces or points.
	Show current results in RFEM graphic	Displays results from the current line in the RFEM graphical window in the background.
	Show rows with ratio > 1	Displays only the rows with the design ratio >1 in tables.
	Show color bars in table	Displays the colored background in result tables according to the relation scale.
	Export to Microsoft Excel or OpenOffice.org Calc	Exports contents of a current table to MS Excel or to the Calc application from the OpenOffice.org package → Chapter 7.2, page 58.

Table 5.1: Buttons in result windows

In this chapter, individual windows are described gradually according to their sequence.

5.1 Max Stress/Ratio by Loading

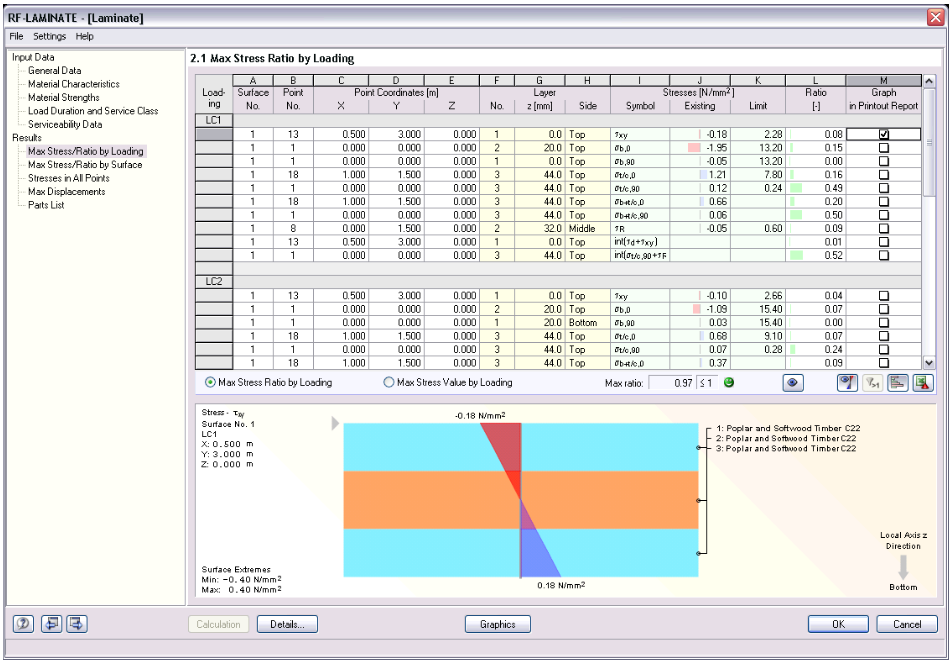


Figure 5.1: Window 2.1 *Max Stress/Ratio by Loading*

If the calculation proceeds successfully, Window 2.1 *Max Stress/Ratio by Loading* opens. In this window, maximum stress values or maximum ratios are displayed for each load case, load combination or result combination that you selected for the design in Window 1.1 *General Data*, in the tab *Ultimate Limit State*. The numbers of load cases, load and result combinations are shown in the heading of each section.

Regarding the fact that compositions with layers from different materials are often designed in RF-LAMINATE, it can happen that the maximum ratio is found in a different location than the maximum stress value. For this reason, below the table you can choose whether you want to display the maximum stress ratios or maximum stress values in the table.

Individual columns of Table 2.1 are described below.

Surface No.

This column contains the numbers of surfaces, for which maximum values of stress components or ratios were found during the calculation. The data is shown for individual load cases.

Point No.

In this column, the numbers of FE mesh points are displayed, or of grid points, where the stress values or ratios reach the maximum. The grid points represent the option to display results independently of the FE mesh, in a regular spacing that you defined. Their count and arrangement are set in the *Grid* tab in the dialog box *Edit Surface* in RFEM.

Details...

In the *Details* dialog box, in the *Results* tab, you can specify whether you want to evaluate the results in FE mesh points or in grid points (see Chapter 4.1.1, page 42).

Point Coordinates

The displayed global coordinates X, Y, Z specify the point with the maximum value of the given stress or its ratio.

Layer

In columns F - H, the layer number is listed, z –coordinate and side of the given layer, where maximum stress values or ratios occur.

Stresses

Symbol and Existing

Here, the maximum stress values that you selected in the *Stresses* tab, the *Details* dialog box, are displayed. The symbol and the existing numeric value are shown.

Limit

The limit value (limit stress) results from the material selection for individual layers in Window 1.2 *Material Characteristics*.

If you use the modification factor k_{mod} or the partial factor γ_M, then the design stress values (with subscript d) are calculated from the characteristic limit stress values (with subscript k), according to

$$\begin{Bmatrix} f_{b,d} \\ f_{t,d} \\ f_{c,d} \\ f_{b,0,d} \\ f_{t,0,d} \\ f_{c,0,d} \\ f_{b,90,d} \\ f_{t,90,d} \\ f_{c,90,d} \\ f_{xy,d} \\ f_{v,d} \\ f_{eqv,d} \\ f_{R,d} \end{Bmatrix} = \frac{k_{mod}}{\gamma_M} \begin{Bmatrix} f_{b,k} \\ f_{t,k} \\ f_{c,k} \\ f_{b,0,k} \\ f_{t,0,k} \\ f_{c,0,k} \\ f_{b,90,k} \\ f_{t,90,k} \\ f_{c,90,k} \\ f_{xy,k} \\ f_{v,k} \\ f_{eqv,k} \\ f_{R,k} \end{Bmatrix} \tag{5.1}$$

If the modification factor and the partial factor are not used, it is considered that $k_{mod} = 1$ and $\gamma_M = 1$.

Ratio

The ratio of the design stress and limit stress is calculated for every stress component. The ratio of the surface in the corresponding FE mesh point or grid point is listed for each selected kind of stress. If the limit stress is not exceeded, the ratio is less than or equal to 1 and the stress design is satisfied. Thus the entry in column L allows you to quickly assess the efficiency of the design.

The following tables show the ratio calculation for particular kinds of stress.

Isotropic material model

Stresses [Pa]	Ratios [-]
σ_x	$= \begin{cases} \dfrac{\sigma_{t/c,x}}{f_{t,d}} + \dfrac{\lvert\sigma_{b,x}\rvert}{f_{b,d}} & \text{if } \sigma_{t/c,x} > 0 \\ \dfrac{\lvert\sigma_{t/c,x}\rvert}{f_{c,d}} + \dfrac{\lvert\sigma_{b,x}\rvert}{f_{b,d}} & \text{if } \sigma_{t/c,x} \le 0 \end{cases}$
σ_y	$= \begin{cases} \dfrac{\sigma_{t/c,y}}{f_{t,d}} + \dfrac{\lvert\sigma_{b,y}\rvert}{f_{b,d}} & \text{if } \sigma_{t/c,y} > 0 \\ \dfrac{\lvert\sigma_{t/c,y}\rvert}{f_{c,d}} + \dfrac{\lvert\sigma_{b,y}\rvert}{f_{b,d}} & \text{if } \sigma_{t/c,y} \le 0 \end{cases}$
σ_1	$= \begin{cases} \dfrac{\sigma_1}{f_{t,d}} & \text{if } \sigma_1 > 0 \\ \dfrac{\lvert\sigma_1\rvert}{f_{c,d}} & \text{if } \sigma_1 \le 0 \end{cases}$
σ_2	$= \begin{cases} \dfrac{\sigma_2}{f_{t,d}} & \text{if } \sigma_2 > 0 \\ \dfrac{\lvert\sigma_2\rvert}{f_{c,d}} & \text{if } \sigma_2 \le 0 \end{cases}$
σ_{eqv}	$\dfrac{\lvert\sigma_{eqv}\rvert}{f_{eqv,d}}$
τ_{max}	$\dfrac{\lvert\tau_{max}\rvert}{f_{v,d}}$
τ_{xz}	$\dfrac{\lvert\tau_{xz}\rvert}{f_{v,d}}$
τ_{xy}	$\dfrac{\lvert\tau_{xy}\rvert}{f_{v,d}}$
τ_{yz}	$\dfrac{\lvert\tau_{yz}\rvert}{f_{v,d}}$

Table 5.2: Ratios for the isotropic material model

Orthotropic material model

Stresses [Pa]	Ratios [-]	
$\sigma_{b,0}$	$\frac{\|\sigma_{b,0}\|}{f_{b,0,d}}$	
$\sigma_{b,90}$	$\frac{\|\sigma_{b,90}\|}{f_{b,90,d}}$	
$\sigma_{t/c,0}$	$= \begin{cases} \frac{\sigma_{t/c,0}}{f_{t,0,d}} & \text{if } \sigma_{t/c,0} > 0 \\ \frac{\|\sigma_{t/c,0}\|}{f_{c,0,d}} & \text{if } \sigma_{t/c,0} \le 0 \end{cases}$	
$\sigma_{t/c,90}$	$= \begin{cases} \frac{\sigma_{t/c,90}}{f_{t,90,d}} & \text{if } \sigma_{t/c,90} > 0 \\ \frac{\|\sigma_{t/c,90}\|}{f_{c,90,d}} & \text{if } \sigma_{t/c,90} \le 0 \end{cases}$	
$\sigma_{b+t/c,0}$	$= \begin{cases} \frac{\sigma_{t/c,0}}{f_{t,0,d}} + \frac{\|\sigma_{b,0}\|}{f_{b,0,d}} & \text{if } \sigma_{t/c,0} > 0 \\ \frac{\|\sigma_{t/c,0}\|}{f_{c,0,d}} + \frac{\|\sigma_{b,0}\|}{f_{b,0,d}} & \text{if } \sigma_{t/c,0} \le 0 \end{cases}$	According to: ČSN 73 1702, (127), (128) DIN 1052, (127), (128) DIN EN 1995-1-1/NA, (NA.130), (NA.131)
$\sigma_{b+t/c,90}$	$= \begin{cases} \frac{\sigma_{t/c,90}}{f_{t,90,d}} + \frac{\|\sigma_{b,90}\|}{f_{b,90,d}} & \text{if } \sigma_{t/c,90} > 0 \\ \frac{\|\sigma_{t/c,90}\|}{f_{c,90,d}} + \frac{\|\sigma_{b,90}\|}{f_{b,90,d}} & \text{if } \sigma_{t/c,90} \le 0 \end{cases}$	
τ_{xy}	$\frac{\|\tau_{xy}\|}{f_{xy,d}}$	
τ_R	$\frac{\|\tau_R\|}{f_{R,d}}$	
$\text{int}\left(\tau_d + \tau_{xy}\right)$	$\frac{\tau_d^2}{f_{v,d}^2} + \frac{\tau_{xy}^2}{f_{xy,d}^2}, \quad \tau_d = \tau_{xz} \cos\beta + \tau_{yz} \sin\beta$	According to: ČSN 73 1702, (129) DIN 1052, (129) DIN EN 1995-1-1/NA, (NA.132)
$\text{int}\left(\sigma_{t/c,90} + \tau_R\right)$	$= \begin{cases} \frac{\sigma_{t/c,90}}{f_{t,90,d}} + \frac{\|\tau_R\|}{f_{R,d}} & \text{if } \sigma_{t/c,90} > 0 \\ \frac{\|\sigma_{t/c,90}\|}{f_{c,90,d}} + \frac{\|\tau_R\|}{f_{R,d}} & \text{if } \sigma_{t/c,90} \le 0 \end{cases}$	According to: ČSN 73 1702, (130), (131) DIN 1052, (130), (131) DIN EN 1995-1-1/NA, (NA.133), (NA.134)

Table 5.3: Ratios for the orthotropic material model

The stresses $\sigma_{\mathrm{b+t/c,0}}$, $\sigma_{\mathrm{b+t/c,90}}$, τ_{d}, τ_{R} are defined in the coordinate system of the grain x', y', z and the following transformation formulas apply

$$\begin{Bmatrix} \sigma_{\mathrm{b+t/c,0}} \\ \sigma_{\mathrm{b+t/c,90}} \\ * \end{Bmatrix} = \underbrace{\begin{bmatrix} c^2 & s^2 & 2cs \\ s^2 & c^2 & -2cs \\ -cs & cs & c^2-s^2 \end{bmatrix}}_{\mathbf{T}_{3\times3}^{-T}} \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix}, \quad \begin{Bmatrix} \tau_{\mathrm{d}} \\ \tau_{\mathrm{R}} \end{Bmatrix} = \underbrace{\begin{bmatrix} c & s \\ -s & c \end{bmatrix}}_{\mathbf{T}_{2\times2}} \begin{Bmatrix} \tau_{xz} \\ \tau_{yz} \end{Bmatrix} \tag{5.2}$$

or, equivalently, in the non-matrix form

$$\begin{aligned} \sigma_{\mathrm{b+t/c,0}} &= \mathrm{c}^2\sigma_x + \mathrm{s}^2\sigma_y + 2\mathrm{cs}\tau_{xy} \\ \sigma_{\mathrm{b+t/c,90}} &= \mathrm{s}^2\sigma_x + \mathrm{c}^2\sigma_y - 2\mathrm{cs}\tau_{xy} \\ \tau_{\mathrm{d}} &= \mathrm{c}\tau_{xz} + \mathrm{s}\tau_{yz} \\ \tau_{\mathrm{R}} &= -\mathrm{s}\tau_{xz} + \mathrm{c}\tau_{yz} \end{aligned} \tag{5.3}$$

where $s = \sin\beta$, $c = \cos\beta$ and β is the rotation angle of the considered layer.

Graph in Printout Report

In the last column of the table, you can select the stress diagrams by thickness, displayed in the lower part of the window, that you want to import to the printout report of RF-LAMINATE, see Chapter 6.2.2, page 56.

5.2 Max Stress/Ratio by Surface

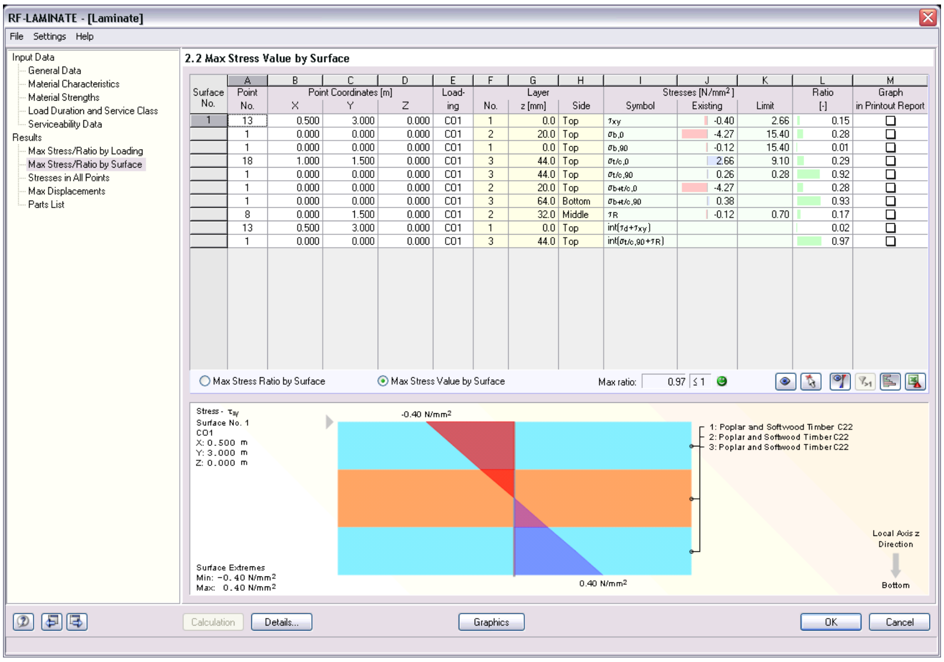


Figure 5.2: Window 2.2 *Max Stress/Ratio by Surface*

This result window contains the maximum stresses/ratios for every designed surface. The data is listed according to individual surfaces.

Individual table columns are described in Chapter 5.1 *Max Stress/Ratio by Loading* on page 45.

5.3 Stresses in All Points

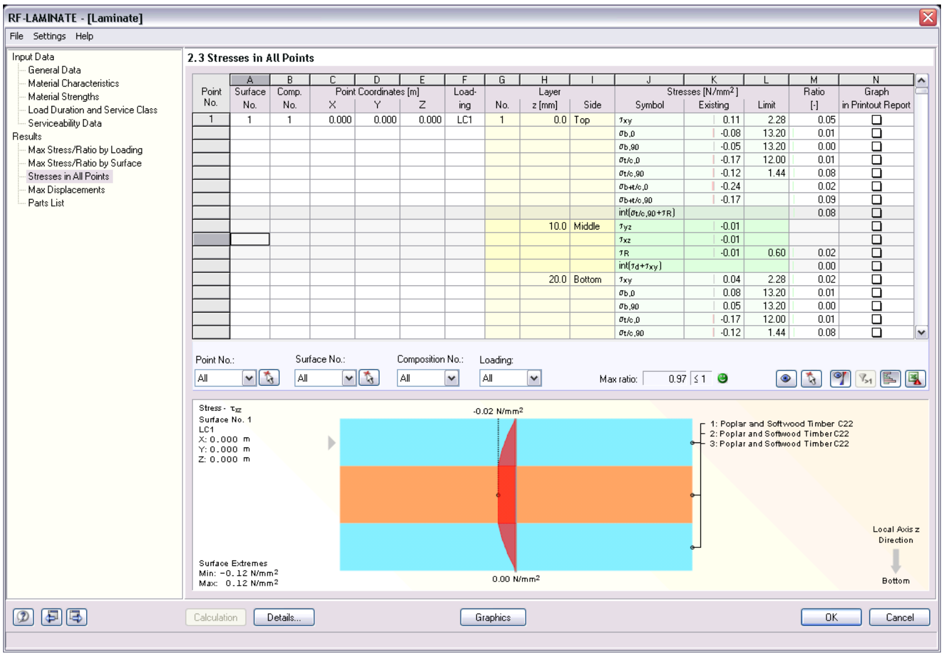
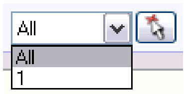


Figure 5.3: Window 2.3 *Stresses in All Points*

In this window, the stresses and ratios are displayed in every FE mesh point, or in every grid point of the designed surfaces. In the *Details* dialog box, in the *Results* tab (see Chapter 4.1.1, page 42), you can set whether you want to display the results in FE mesh points or in grid points.

In the *Details* dialog box, in the *Stresses* tab, you can also specify which stress components you want to display in the window.

Individual table columns are described in Chapter 5.1 *Max Stress/Ratio by Loading* on page 45.



If needed, you can filter the data in the table according to numbers of FE mesh points, numbers of surfaces, compositions or loadings. Selection is possible either from the lists under the table or you can select a certain point or surface in the graphical window by using the [Select] button.

5.4 Max Displacements

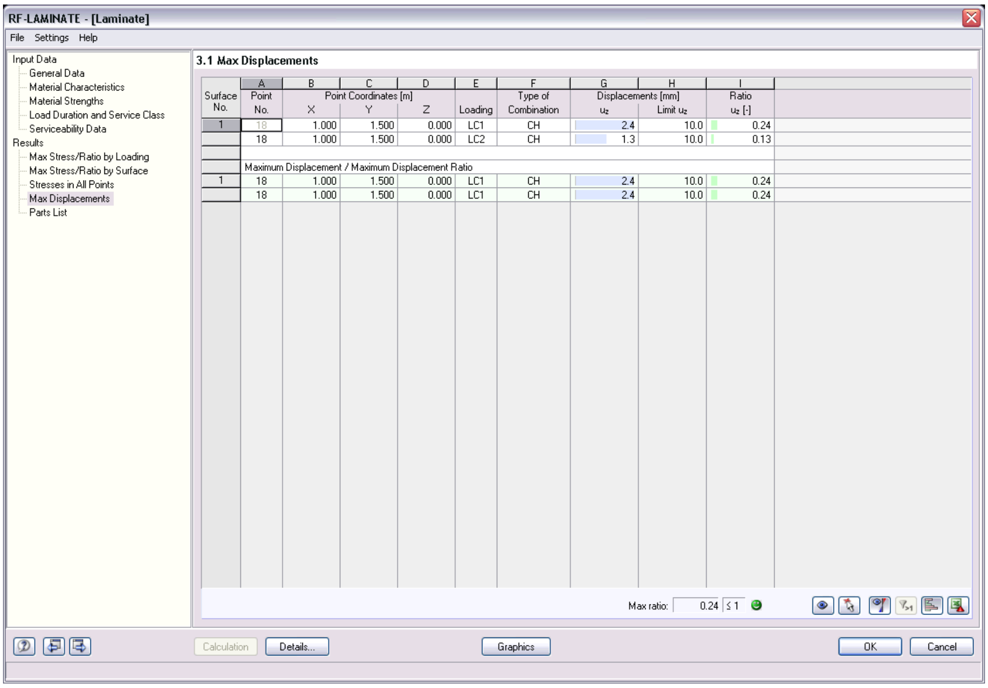


Figure 5.4: Window 3.1 *Max Displacements*

This window is displayed only if you have selected at least one load case for the design in Window 1.1 *General Data*, in the tab *Serviceability Limit State* (see Chapter 3.1.2, page 22). Here, the maximum values of displacements are shown from load cases, load combinations or result combinations that you selected for the serviceability limit state design.

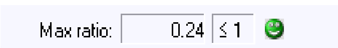
The data is listed according to individual surfaces.

Displacements

In the column *Displacements*, you can see the displacements in the direction of local axes z of given surfaces, which are governing for the deformation analysis. You can display the local axes of the surfaces in the *Display* navigator in RFEM.

In column I, the limit values of deformations are displayed. These values are determined from the reference lengths L, which are entered in Window 1.5 *Serviceability Data*, see page 32, and from the general limit values defined for the serviceability design in the *Details* dialog box, in the *Design* tab (Chapter 4.1.1, page 34).

Ratio



In the last column, ratios of the resulting displacement u_z (column G) and the limit displacement (column H) are shown. If the limit deformations are not exceeded, the ratio is less than or equal to 1 and the deformation design is satisfied.

5.5 Parts List

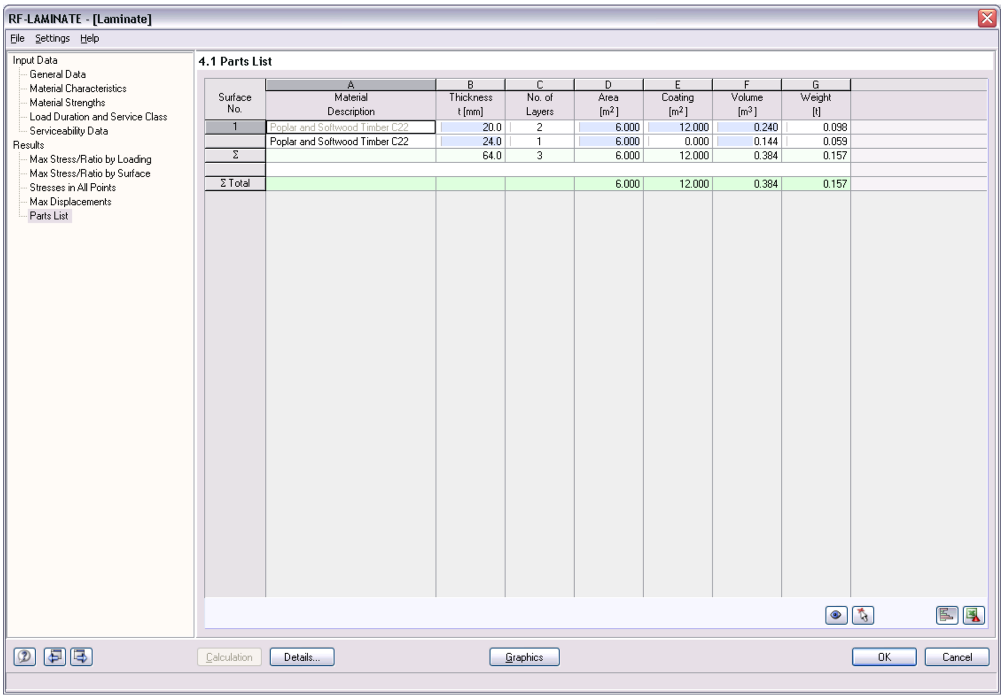


Figure 5.5: Window 4.1 *Parts List*

The last result window shows the overall review of surfaces. The data refers only to designed surfaces by default. If you want to display the parts list of all surfaces in the structure, you can modify the setting in the *Details* dialog box, in the *Results* tab (Chapter 4.1.1, page 42).

Surface No.

This column contains the numbers of individual surfaces.

Material Description

Surfaces are classified according to materials.

Thickness

In column B, the thickness of layers t is displayed. The thicknesses can be also found in input Window 1.2 *Material Characteristics.*

No. of Layers

This column shows the number of layers in the structure that have the given material and thickness.

Area

Here, information about the area of every surface is displayed.

Coating

The total surface coating is calculated based on the upper and lower surface. Side surfaces are neglected.

Volume

The volume is calculated as the product of the thickness and surface area.

Weight

In the last column, the weight of every surface is displayed. This entry is calculated based on the surface volume and the specific weight of the selected material.

Total

In the last table row, you can see the total sums of individual columns.

6. Printout

6.1 Printout Report

As usual in RFEM, a printout report is created for the design results in RF-LAMINATE at first. You can insert graphical displays or your own comments to the printout report. In the printout report, you can also select which RF-LAMINATE result windows you want to print.

The printout report is described in detail in the RFEM manual. Chapter 10.1.3.4 *Selecting Data of Add-on Modules* is important in particular. This chapter describes the selection of input and output data in add-on modules.

You can create several printout reports for each model. Especially in the case of complex structures, it is recommended to create a few smaller printout reports instead of one large report. If you create for example a separate printout report only for RF-LAMINATE data, the printout report is processed relatively fast.

The stress components, which were selected for the display in result windows in RF-LAMINATE, appear in the printout report.

6.2 Printing RF-LAMINATE Graphics

6.2.1 Results on the Model in RFEM

Graphical displays of the performed designs can be incorporated in the printout report or sent directly to a printer. In Chapter 10.2 *Direct Graphic Printout* in the RFEM manual, the printing of graphical displays is described in detail.

Every image that is displayed in the graphical window in RFEM can be incorporated in the printout report. In the same way, you can take over result diagrams on sections to the printout report as well, by clicking the [Print] button in the respective window.

You can print the current RF-LAMINATE graphical display in the RFEM work window by using the command from the main menu

File → Print Graphic…

or by clicking the relevant button in the toolbar.

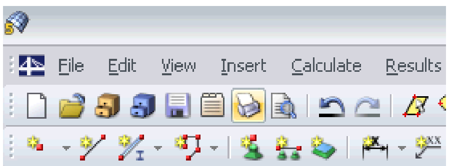


Figure 6.1: Button *Print Graphic* in the toolbar in the main window

Then, the following dialog box appears.

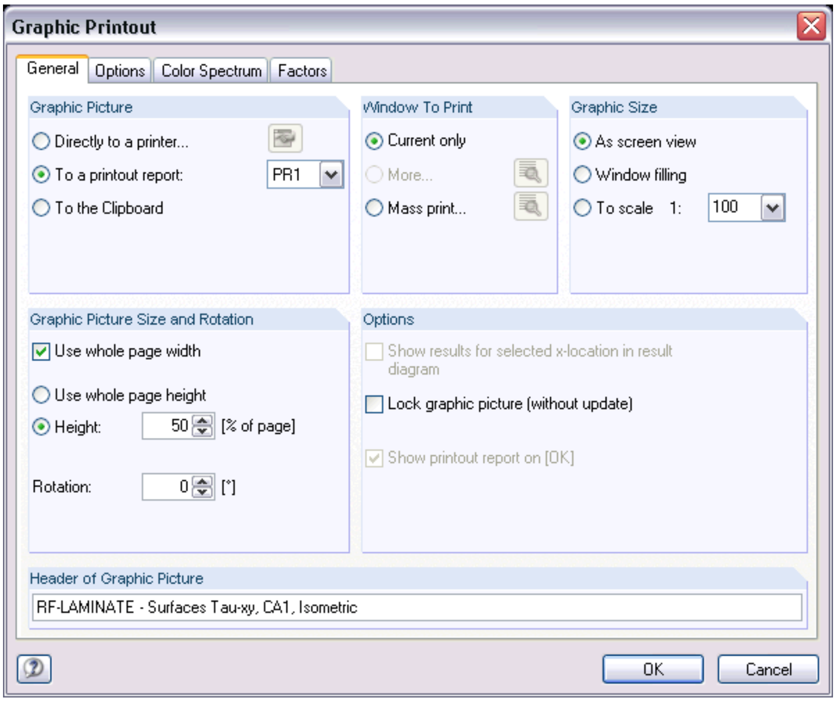
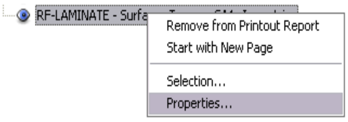


Figure 6.2: Dialog box *Graphic Printout* - the *General* tab

This dialog box is described in detail in Chapter 10.2 *Direct Graphic Printout* in the RFEM manual. Other tabs *Options*, *Color Spectrum* and *Factors* are described there as well.

You can move the RF-LAMINATE graphics to another place in the printout report, by using the drag & drop function.



Inserted images can be also modified additionally: right-click the corresponding item in the printout report navigator and choose *Properties...* in the local menu. The dialog box *Graphic Printout* is displayed again, and you can set possible changes there.

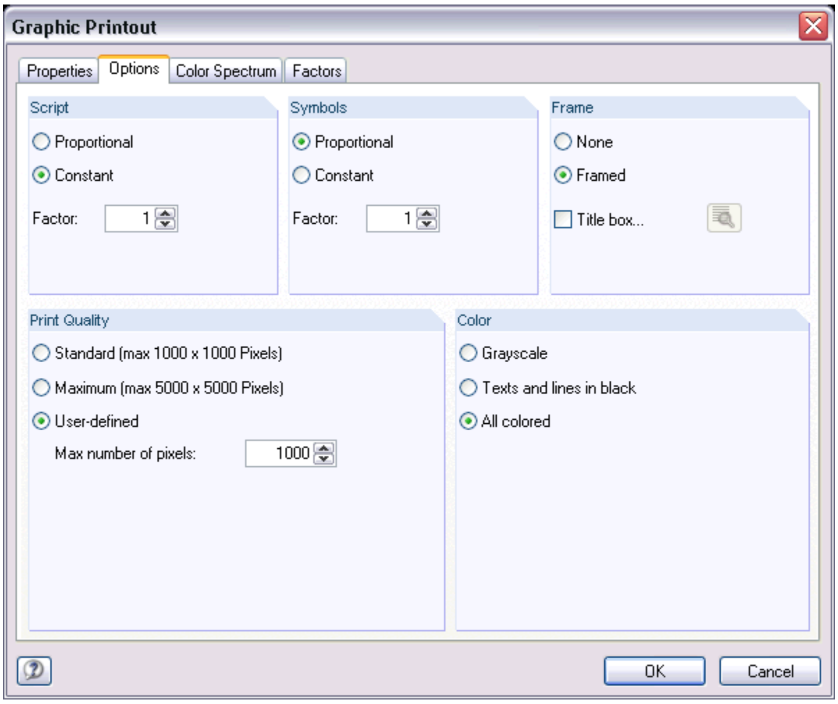


Figure 6.3: Dialog box *Graphic Printout* - the *Options* tab

6.2.2 Stress Diagrams

In Windows 2.1, 2.2 and 2.3 of RF-LAMINATE, you can incorporate a selected picture to the printout report by using the check boxes in the column *Graph in Printout Report*. In figure 6.4, it is apparent that the stress diagram τ_{xy} in point [0.5, 3.0, 0.0] will be displayed in the printout report.

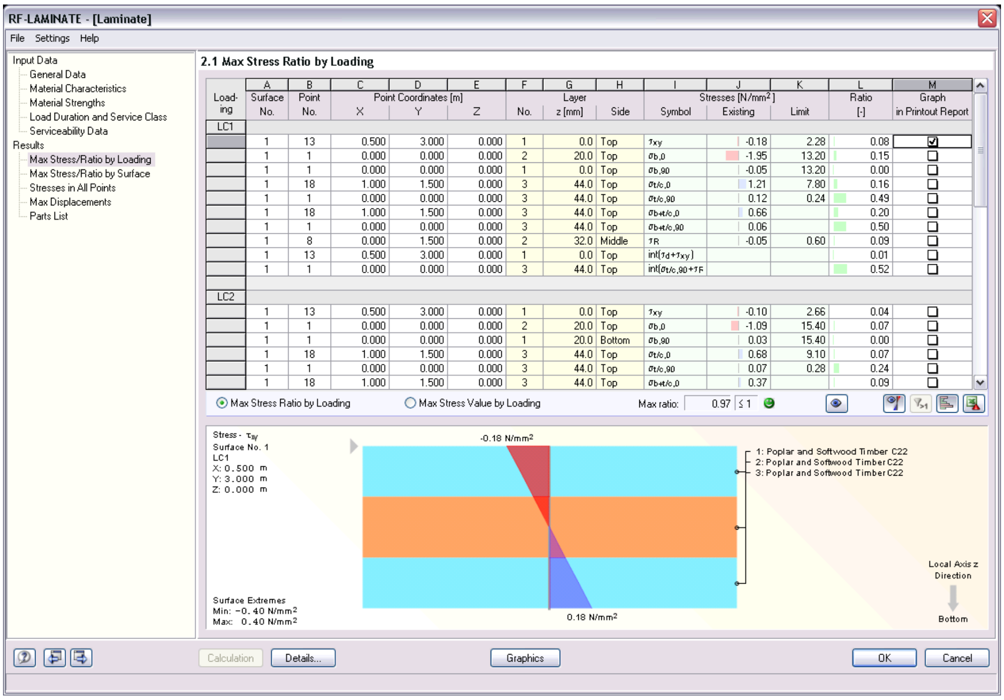


Figure 6.4: Window 2.1 *Max Stress/Ratio by Loading*

The selected pictures are then displayed in the printout report in Chapter 4.2 *Stress Diagrams*.

4.1 PARTS LIST

Part No.	Material Description	Thickness t [mm]	Layer Count	Area [m²]	Coating [m²]	Volume [m³]	Weight [t]
1	Poplar and Softwood Timber C22	20.0	2	6.000	12.000	0.240	0.098
	Poplar and Softwood Timber C22	24.0	1	6.000	0.000	0.144	0.059
Σ		64.0	3	6.000	12.000	0.384	0.157
Σ Total				6.000	12.000	0.384	0.157

4.2 STRESS DIAGRAMS

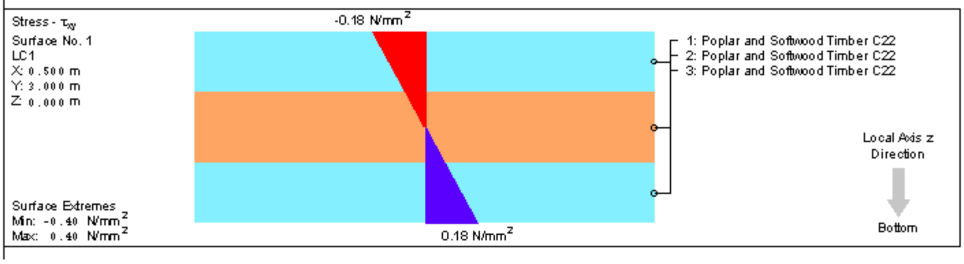


Figure 6.5: Stress diagrams in the printout report

7. General Functions

Commonly used functions from the main menu and export options for design results are described in this chapter.

7.1 Units and Decimal Places

Units and decimal places for RFEM and all its add-on modules are set in one central dialog box. In RF-LAMINATE, you can open the dialog box for the setting of units by using the command from the main menu

Settings → **Units and Decimal Places…**.

The dialog box that is already familiar from RFEM opens. The RF-LAMINATE module is already preset in this dialog box.

In figure 7.1, you can see that some units are marked with a red arrow (the thicknesses and material characteristics in this case). This marking is used for a quick orientation in the dialog box *Units and Decimal Places*, for the currently opened RF-LAMINATE window. In this case, Window *1.2 Material Characteristics* is opened in the module, therefore it is very easy to find and then change the units related to this window.

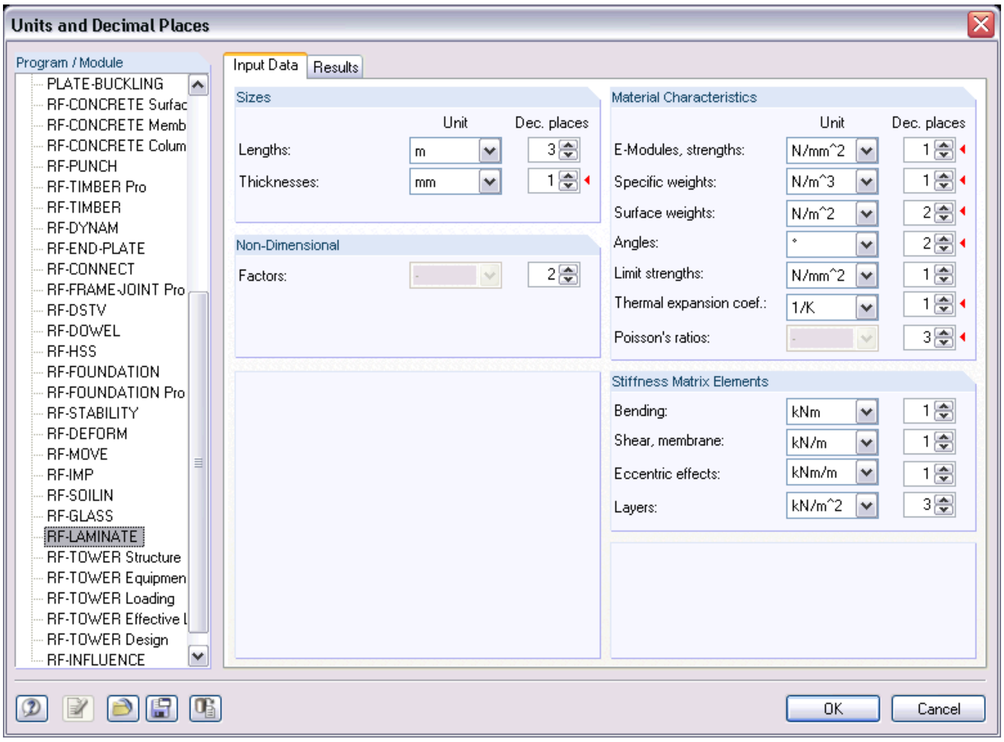


Figure 7.1: Dialog box *Units and Decimal Places*

You can save the settings as a user-defined profile and then use it in other models. You can find the description of this function in Chapter 11.1.3 *Units and Decimal Places* in the RFEM manual.

7.2 Export of Results

You can also transfer the analysis results to other programs in a variety of ways.

Clipboard

You can copy the marked rows in the RF-LAMINATE result window to the Clipboard by using the buttons [Ctrl]+[C] and then transfer for example to a text processor by using [Ctrl]+[V]. The headings of table columns are not exported.

Printout report

RF-LAMINATE data can be incorporated in the central printout report (Chapter 6.1, page 54) and then exported by using the command from the main menu

File → Export to RTF....

This function is described in Chapter 10.1.11 *Export Printout Report* in the RFEM manual.

Excel / OpenOffice

RF-LAMINATE provides a direct export of data to MS Excel or to the Calc application from the OpenOffice.org package.

You call up this function from the main menu of RF-LAMINATE

File → Export Tables....

The following dialog box for the data export opens:

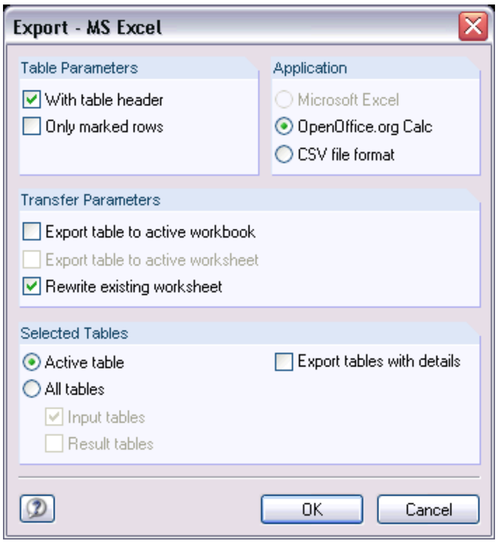


Figure 7.2: Dialog box *Export - MS Excel*

As soon as you select the required parameters, you can start the export by clicking the [OK] button. Excel or Calc need not run in the background, it is automatically started before the export.

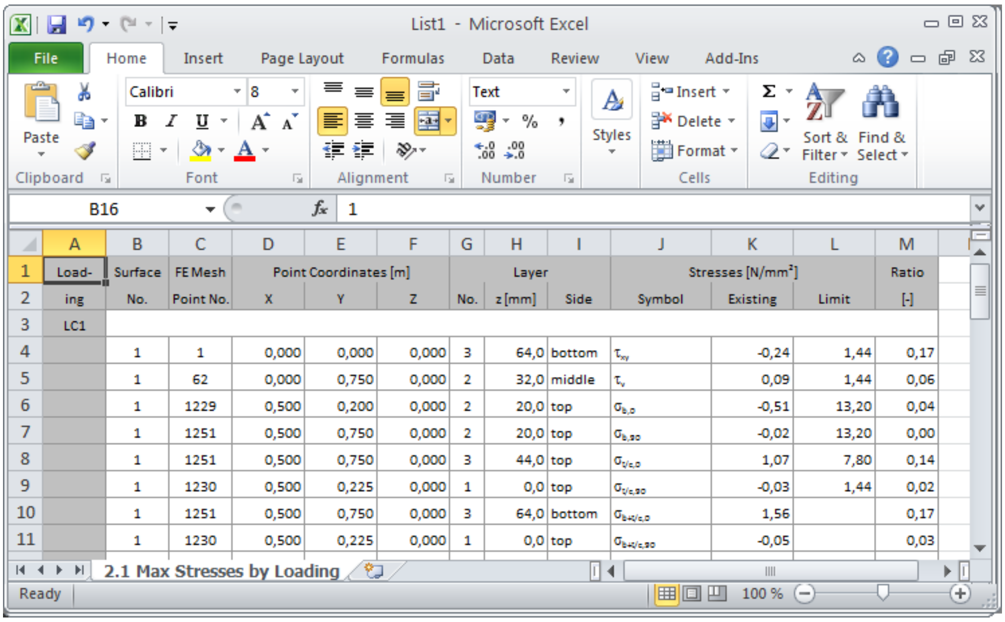


Figure 7.3: Results in MS Excel: Table 2.1 Max Stress/Ratio by Loading

7.3 Keyboard Shortcuts

In individual windows and dialog boxes of RF-LAMINATE, you can call up frequently used functions from the keyboard quickly:

[F1]	Help
[F2]	Next Window
[F3]	Previous Window
[F7]	Selection in Tables
[F8]	Copy the Last Table Row
[F9]	Calculator
[Ctrl+2]	Copy a Table Row to Next Row
[Ctrl+C]	Copy to the Clipboard
[Ctrl+F]	Find in the Table
[Ctrl+H]	Find and Replace in the Table
[Ctrl+I]	Insert a Row in the Table
[Ctrl+L]	Go to Row with a Given Number
[Ctrl+R]	Delete Table Rows
[Ctrl+S]	Save Data
[Ctrl+V]	Insert Data from the Clipboard
[Ctrl+X]	Exclude from Table
[Ctrl+Y]	Empty the Current Table Row
[Ctrl+Z]	Undo

Table 7.1: Keyboard shortcuts

8. Examples

Several examples are introduced in the following chapter.

8.1 Calculation of Stiffness Matrix Elements

Consider a plate consisting of three layers that are shown in figure 8.1 and their material characteristics are displayed in figure 8.2.

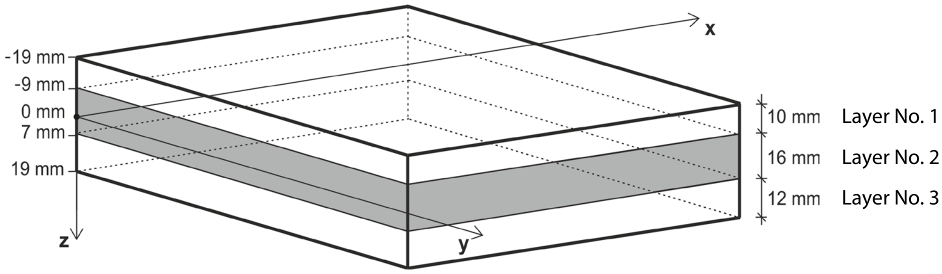


Figure 8.1: Layer scheme

Layers

Layer No.	A Material Description	B Thickness t [mm]	C Orthotropic Direction β [°]	D Modulus of Elasticity [N/mm²] E_x	E E_y	F Shear Modulus [N/mm²] G_{xz}	G G_{yz}	H G_{xy}	I Poisson's Ratio [-] ν_{xy}	J ν_{yx}	K Specific Weight γ [N/m³]
1	Poplar and Coniferous Timber C16	10.0	0.00	8000.0	270.0	500.0	50.0	500.0	0.200	0.007	3700.0
2	Coniferous Timber C14	16.0	90.00	7000.0	230.0	440.0	44.0	440.0	0.200	0.007	5000.0
3	Poplar and Coniferous Timber C16	12.0	0.00	8000.0	270.0	500.0	50.0	500.0	0.200	0.007	3700.0

Figure 8.2: Table 1.2 *Material Characteristics*

At first, you calculate the stiffness matrices of individual layers

$$\mathbf{d}'_i = \begin{bmatrix} d'_{i;11} & d'_{i;12} & 0 \\ & d'_{i;22} & 0 \\ \text{sym.} & & d'_{i;33} \end{bmatrix} = \begin{bmatrix} \dfrac{E_{i;x}}{1-\nu_{i;xy}^2 \dfrac{E_{i;y}}{E_{i;x}}} & \dfrac{\nu_{i;xy} E_{i;y}}{1-\nu_{i;xy}^2 \dfrac{E_{i;y}}{E_{i;x}}} & 0 \\ & \dfrac{E_{i;y}}{1-\nu_{i;xy}^2 \dfrac{E_{i;y}}{E_{i;x}}} & 0 \\ \text{sym.} & & G_{i;xy} \end{bmatrix} \quad i = 1, \ldots, n \tag{8.1}$$

$$\mathbf{d}'_1 = \begin{bmatrix} \dfrac{8000}{1-0.2^2 \dfrac{270}{8000}} & \dfrac{0.2 \cdot 270}{1-0.2^2 \dfrac{270}{8000}} & 0 \\ & \dfrac{270}{1-0.2^2 \dfrac{270}{8000}} & 0 \\ \text{sym.} & & 500 \end{bmatrix} = \begin{bmatrix} 8010.81 & 54.07 & 0 \\ 54.07 & 270.36 & 0 \\ 0 & 0 & 500.00 \end{bmatrix} \text{MN/m}^2$$

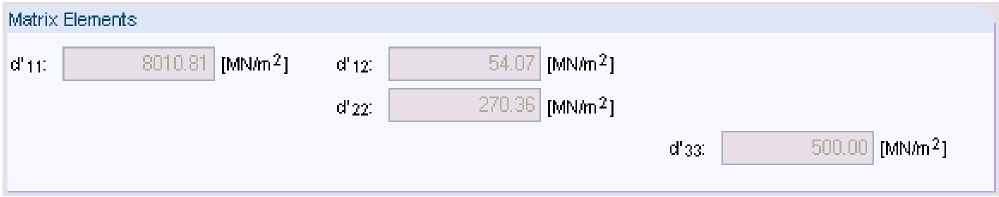


Figure 8.3: Stiffness matrix of Layer No. 1 from RF-LAMINATE

$$\mathbf{d}_2' = \begin{bmatrix} \dfrac{7000}{1-0.2^2\dfrac{230}{7000}} & \dfrac{0.2\cdot 230}{1-0.2^2\dfrac{270}{7000}} & 0 \\ & \dfrac{270}{1-0.2^2\dfrac{230}{7000}} & 0 \\ \text{sym.} & & 440 \end{bmatrix} = \begin{bmatrix} 7009.21 & 46.06 & 0 \\ 46.06 & 230.30 & 0 \\ 0 & 0 & 440.00 \end{bmatrix} \text{MN/m}^2$$

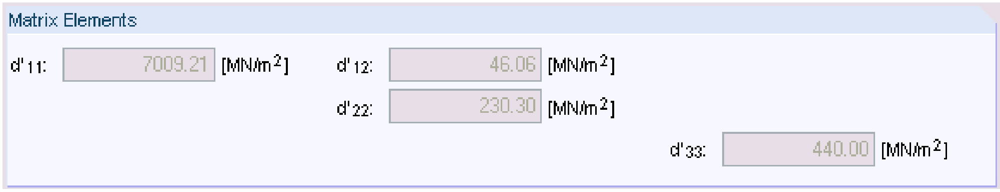


Figure 8.4: Stiffness matrix of Layer No. 2 from RF-LAMINATE

$$\mathbf{d}_3' = \begin{bmatrix} \dfrac{8000}{1-0.2^2\dfrac{270}{8000}} & \dfrac{0.2\cdot 270}{1-0.2^2\dfrac{270}{8000}} & 0 \\ & \dfrac{270}{1-0.2^2\dfrac{270}{8000}} & 0 \\ \text{sym.} & & 500 \end{bmatrix} = \begin{bmatrix} 8010.81 & 54.07 & 0 \\ 54.07 & 270.36 & 0 \\ 0 & 0 & 500.00 \end{bmatrix} \text{MN/m}^2$$

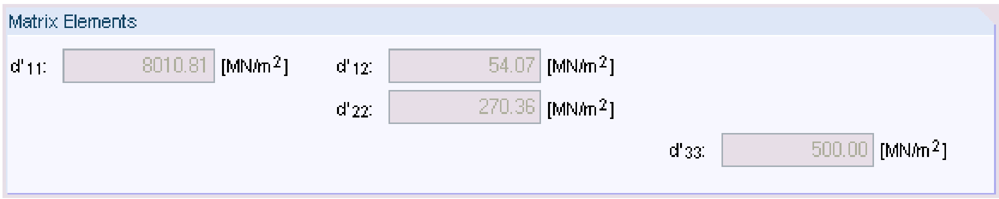
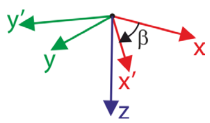


Figure 8.5: Stiffness matrix of Layer No. 3 from RF-LAMINATE



Now, you have to rotate the layers to the same coordinate system x, y (local coordinate system of the surface). Layers No. 1 and 3 have the orthotropy direction $\beta = 0°$, therefore it applies that

$$\mathbf{d}_1 = \mathbf{d}_1' = \begin{bmatrix} 8010.81 & 54.07 & 0 \\ 54.07 & 270.36 & 0 \\ 0 & 0 & 500.00 \end{bmatrix} \text{MN/m}^2$$

$$\mathbf{d}_3 = \mathbf{d}_3' = \begin{bmatrix} 8010.81 & 54.07 & 0 \\ 54.07 & 270.36 & 0 \\ 0 & 0 & 500.00 \end{bmatrix} \text{MN/m}^2$$

Because Layer No. 2 is rotated by the angle $\beta = 90°$, it is necessary to transform the stiffness matrix of Layer No. 2 to the coordinate system x, y.

$$\mathbf{d}_i = \begin{bmatrix} d_{i;11} & d_{i;12} & d_{i;13} \\ & d_{i;22} & d_{i;23} \\ \text{sym.} & & d_{i;33} \end{bmatrix} = \mathbf{T}_{3\times3;i}{}^{\mathrm{T}} \mathbf{d}'_i \mathbf{T}_{3\times3;i} \tag{8.2}$$

where

$$\mathbf{T}_{3\times3;i} = \begin{bmatrix} c^2 & s^2 & cs \\ s^2 & c^2 & -cs \\ -2cs & 2cs & c^2 - s^2 \end{bmatrix}, \text{where } c = \cos(\beta_i),\ s = \sin(\beta_i) \tag{8.3}$$

The individual elements then are

$$d_{i;11} = c^4 d'_{i;11} + 2c^2 s^2 d'_{i;12} + s^4 d'_{i;22} + 4c^2 s^2 d'_{i;33}$$
$$d_{i;12} = c^2 s^2 d'_{i;11} + s^4 d'_{i;12} + c^4 d'_{i;12} + c^2 s^2 d'_{i;22} - 4c^2 s^2 d'_{i;33}$$
$$d_{i;13} = c^3 s d'_{i;11} + cs^3 d'_{i;12} - c^3 s d'_{i;12} - cs^3 d'_{i;22} - 2c^3 s d'_{i;33} + 2cs^3 d'_{i;33}$$
$$d_{i;22} = s^4 d'_{i;11} + 2c^2 s^2 d'_{i;12} + c^4 d'_{i;22} + 4c^2 s^2 d'_{i;33}$$
$$d_{i;23} = cs^3 d'_{i;11} + c^3 s d'_{i;12} - cs^3 d'_{i;12} - c^3 s d'_{i;22} + 2c^3 s d'_{i;33} - 2cs^3 d'_{i;33}$$
$$d_{i;33} = c^2 s^2 d'_{i;11} - 2c^2 s^2 d'_{i;12} + c^2 s^2 d'_{i;22} + \left(c^2 - s^2\right)^2 d'_{i;33}$$

$$c = \cos 90° = 0,\ s = \sin 90° = 1$$

$$d_{2;11} = 0^4 \cdot 7009.21 + 2 \cdot 0^2 \cdot 1^2 \cdot 46.06 + 1^4 \cdot 230.30 + 4 \cdot 0^2 \cdot 1^2 \cdot 440 = 230.30\ \text{MN/m}^2$$

$$d_{2;12} = 0^2 \cdot 1^2 \cdot 7009.21 + 1^4 \cdot 46.06 + 0^4 \cdot 46.06 + 0^2 \cdot 1^2 \cdot 230.30 - 4 \cdot 0^2 \cdot 1^2 \cdot 440 = 46.06\ \text{MN/m}^2$$

$$d_{2;13} = 0^3 \cdot 1 \cdot 7009.21 + 0 \cdot 1^3 \cdot 46.06 - 0^3 \cdot 1 \cdot 46.06 - 0 \cdot 1^3 \cdot 230.30 - 2 \cdot 0^3 \cdot 1 \cdot 440 + 2 \cdot 0 \cdot 1^3 \cdot 440 =$$
$$= 0\ \text{MN/m}^2$$

$$d_{2;22} = 1^4 \cdot 7009.21 + 2 \cdot 0^2 \cdot 1^2 \cdot 46.06 + 0^4 \cdot 230.30 + 4 \cdot 0^2 \cdot 1^2 \cdot 440 = 7009.21\ \text{MN/m}^2$$

$$d_{2;23} = 0 \cdot 1^3 \cdot 7009.21 + 0^3 \cdot 1 \cdot 46.06 - 0 \cdot 1^3 \cdot 46.06 - 0^3 \cdot 1 \cdot 230.30 + 2 \cdot 0^3 \cdot 1 \cdot 440 - 2 \cdot 0 \cdot 1^3 \cdot 440 =$$
$$= 0\ \text{MN/m}^2$$

$$d_{2;33} = 0^2 \cdot 1^2 \cdot 7009.21 - 2 \cdot 0^2 \cdot 1^2 \cdot 46.06 + 0^2 \cdot 1^2 \cdot 230.30 + \left(0^2 - 1^2\right)^2 440 = 440.00\ \text{MN/m}^2$$

The total planar stiffness matrix of Layer No. 2 is then

$$\mathbf{d}_2 = \begin{bmatrix} 230.30 & 46.06 & 0 \\ 46.06 & 7009.21 & 0 \\ 0 & 0 & 440.00 \end{bmatrix} \text{MN/m}^2$$

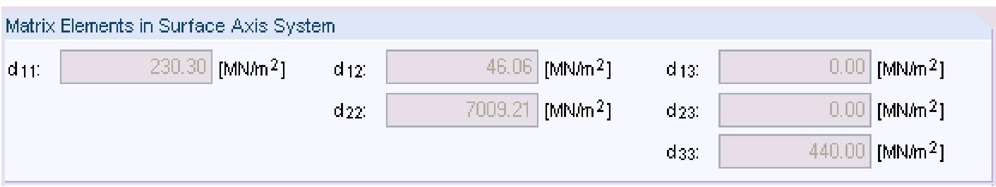


Figure 8.6: Stiffness matrix of layer No. 2 from RF-LAMINATE

8.1.1 Shear Coupling of Layers Is Considered

The global stiffness matrix has the form

$$\mathbf{D} = \begin{bmatrix} D_{11} & D_{12} & D_{13} & 0 & 0 & D_{16} & D_{17} & D_{18} \\ & D_{22} & D_{23} & 0 & 0 & \text{sym.} & D_{27} & D_{28} \\ & & D_{33} & 0 & 0 & \text{sym.} & \text{sym.} & D_{38} \\ & & & D_{44} & D_{45} & 0 & 0 & 0 \\ & & & & D_{55} & 0 & 0 & 0 \\ & & \text{sym.} & & & D_{66} & D_{67} & D_{68} \\ & & & & & & D_{77} & D_{78} \\ & & & & & & & D_{88} \end{bmatrix} \tag{8.4}$$

Stiffness matrix elements (bending and torsion)

$$D_{11} = \sum_{i=1}^{n} \frac{z_{i;\max}^3 - z_{i;\min}^3}{3} d_{i;11} \qquad D_{12} = \sum_{i=1}^{n} \frac{z_{i;\max}^3 - z_{i;\min}^3}{3} d_{i;12} \qquad D_{13} = \sum_{i=1}^{n} \frac{z_{i;\max}^3 - z_{i;\min}^3}{3} d_{i;13}$$

$$D_{22} = \sum_{i=1}^{n} \frac{z_{i;\max}^3 - z_{i;\min}^3}{3} d_{i;22} \qquad D_{23} = \sum_{i=1}^{n} \frac{z_{i;\max}^3 - z_{i;\min}^3}{3} d_{i;23}$$

$$D_{33} = \sum_{i=1}^{n} \frac{z_{i;\max}^3 - z_{i;\min}^3}{3} d_{i;33}$$

$$D_{11} = \frac{(-9\cdot10^{-3})^3-(-19\cdot10^{-3})^3}{3}8010.81\cdot10^3 + \frac{(7\cdot10^{-3})^3-(-9\cdot10^{-3})^3}{3}230.30\cdot10^3 + \\ + \frac{(19\cdot10^{-3})^3-(7\cdot10^{-3})^3}{3}8010.81\cdot10^3 = 33.85\text{ kNm}$$

$$D_{12} = \frac{(-9\cdot10^{-3})^3-(-19\cdot10^{-3})^3}{3}54.07\cdot10^3 + \frac{(7\cdot10^{-3})^3-(-9\cdot10^{-3})^3}{3}46.06\cdot10^3 + \\ + \frac{(19\cdot10^{-3})^3-(7\cdot10^{-3})^3}{3}54.07\cdot10^3 = 0.24\text{ kNm}$$

$$D_{13} = \frac{(-9\cdot10^{-3})^3-(-19\cdot10^{-3})^3}{3}0\cdot10^3 + \frac{(7\cdot10^{-3})^3-(-9\cdot10^{-3})^3}{3}0\cdot10^3 + \\ + \frac{(19\cdot10^{-3})^3-(7\cdot10^{-3})^3}{3}0\cdot10^3 = 0\text{ kNm}$$

$$D_{22} = \frac{(-9\cdot10^{-3})^3-(-19\cdot10^{-3})^3}{3}270.36\cdot10^3 + \frac{(7\cdot10^{-3})^3-(-9\cdot10^{-3})^3}{3}7009.21\cdot10^3 + \\ + \frac{(19\cdot10^{-3})^3-(7\cdot10^{-3})^3}{3}270.36\cdot10^3 = 3.64\text{ kNm}$$

$$D_{23} = \frac{(-9\cdot10^{-3})^3-(-19\cdot10^{-3})^3}{3}0\cdot10^3 + \frac{(7\cdot10^{-3})^3-(-9\cdot10^{-3})^3}{3}0\cdot10^3 + \\ + \frac{(19\cdot10^{-3})^3-(7\cdot10^{-3})^3}{3}0\cdot10^3 = 0\text{ kNm}$$

$$D_{33} = \frac{(-9\cdot10^{-3})^3-(-19\cdot10^{-3})^3}{3}500\cdot10^3 + \frac{(7\cdot10^{-3})^3-(-9\cdot10^{-3})^3}{3}440.00\cdot10^3 + \\ + \frac{(19\cdot10^{-3})^3-(7\cdot10^{-3})^3}{3}500\cdot10^3 = 2.26\text{ kNm}$$

Stiffness matrix elements (eccentricity effects)

$$D_{16} = \sum_{i=1}^{n} \frac{z_{i;\max}^2 - z_{i;\min}^2}{2} d_{i;11} \qquad D_{17} = \sum_{i=1}^{n} \frac{z_{i;\max}^2 - z_{i;\min}^2}{2} d_{i;12} \qquad D_{18} = \sum_{i=1}^{n} \frac{z_{i;\max}^2 - z_{i;\min}^2}{2} d_{i;13}$$

$$D_{27} = \sum_{i=1}^{n} \frac{z_{i;\max}^2 - z_{i;\min}^2}{2} d_{i;22} \qquad D_{28} = \sum_{i=1}^{n} \frac{z_{i;\max}^2 - z_{i;\min}^2}{2} d_{i;23}$$

$$D_{38} = \sum_{i=1}^{n} \frac{z_{i;\max}^2 - z_{i;\min}^2}{2} d_{i;33}$$

$$D_{16} = \frac{(-9\cdot 10^{-3})^2 - (-19\cdot 10^{-3})^2}{2} 8010.81\cdot 10^3 + \frac{(7\cdot 10^{-3})^2 - (-9\cdot 10^{-3})^2}{2} 230.30\cdot 10^3 + \\ + \frac{(19\cdot 10^{-3})^2 - (7\cdot 10^{-3})^2}{2} 8010.81\cdot 10^3 = 124.49 \text{ kNm/m}$$

$$D_{17} = \frac{(-9\cdot 10^{-3})^2 - (-19\cdot 10^{-3})^2}{2} 54.07\cdot 10^3 + \frac{(7\cdot 10^{-3})^2 - (-9\cdot 10^{-3})^2}{2} 46.06\cdot 10^3 + \\ + \frac{(19\cdot 10^{-3})^2 - (7\cdot 10^{-3})^2}{2} 54.07\cdot 10^3 = 0.13 \text{ kNm/m}$$

$$D_{18} = \frac{(-9\cdot 10^{-3})^2 - (-19\cdot 10^{-3})^2}{2} 0\cdot 10^3 + \frac{(7\cdot 10^{-3})^2 - (-9\cdot 10^{-3})^2}{2} 0\cdot 10^3 + \\ + \frac{(19\cdot 10^{-3})^2 - (7\cdot 10^{-3})^2}{2} 0\cdot 10^3 = 0 \text{ kNm/m}$$

$$D_{27} = \frac{(-9\cdot 10^{-3})^2 - (-19\cdot 10^{-3})^2}{2} 270.36\cdot 10^3 + \frac{(7\cdot 10^{-3})^2 - (-9\cdot 10^{-3})^2}{2} 7009.21\cdot 10^3 + \\ + \frac{(19\cdot 10^{-3})^2 - (7\cdot 10^{-3})^2}{2} 270.36\cdot 10^3 = -107.82 \text{ kNm/m}$$

$$D_{28} = \frac{(-9\cdot 10^{-3})^2 - (-19\cdot 10^{-3})^2}{2} 0\cdot 10^3 + \frac{(7\cdot 10^{-3})^2 - (-9\cdot 10^{-3})^2}{2} 0\cdot 10^3 + \\ + \frac{(19\cdot 10^{-3})^2 - (7\cdot 10^{-3})^2}{2} 0\cdot 10^3 = 0 \text{ kNm/m}$$

$$D_{38} = \frac{(-9\cdot 10^{-3})^2 - (-19\cdot 10^{-3})^2}{2} 500\cdot 10^3 + \frac{(7\cdot 10^{-3})^2 - (-9\cdot 10^{-3})^2}{2} 440.00\cdot 10^3 + \\ + \frac{(19\cdot 10^{-3})^2 - (7\cdot 10^{-3})^2}{2} 500\cdot 10^3 = 0.96 \text{ kNm/m}$$

Stiffness matrix elements (membrane)

$$D_{66} = \sum_{i=1}^{n} t_i\, d_{i;11} \qquad D_{67} = \sum_{i=1}^{n} t_i\, d_{i;12} \qquad D_{68} = \sum_{i=1}^{n} t_i\, d_{i;13}$$

$$D_{77} = \sum_{i=1}^{n} t_i\, d_{i;22} \qquad D_{78} = \sum_{i=1}^{n} t_i\, d_{i;23}$$

$$D_{88} = \sum_{i=1}^{n} t_i\, d_{i;33}$$

$$D_{66} = 10 \cdot 10^{-3} \cdot 8010.81 \cdot 10^3 + 16 \cdot 10^{-3} \cdot 230.30 \cdot 10^3 + 12 \cdot 10^{-3} \cdot 8010.81 \cdot 10^3 = 179923 \text{ N/m}$$

$$D_{67} = 10 \cdot 10^{-3} \cdot 54.07 \cdot 10^3 + 16 \cdot 10^{-3} \cdot 46.06 \cdot 10^3 + 12 \cdot 10^{-3} \cdot 54.07 \cdot 10^3 = 1927 \text{ N/m}$$

$$D_{68} = 10 \cdot 10^{-3} \cdot 0 \cdot 10^3 + 16 \cdot 10^{-3} \cdot 0 \cdot 10^3 + 12 \cdot 10^{-3} \cdot 0 \cdot 10^3 = 0 \text{ N/m}$$

$$D_{77} = 10 \cdot 10^{-3} \cdot 270.36 \cdot 10^3 + 16 \cdot 10^{-3} \cdot 7009.21 \cdot 10^3 + 12 \cdot 10^{-3} \cdot 270.36 \cdot 10^3 = 118095 \text{ N/m}$$

$$D_{78} = 10 \cdot 10^{-3} \cdot 0 \cdot 10^3 + 16 \cdot 10^{-3} \cdot 0 \cdot 10^3 + 12 \cdot 10^{-3} \cdot 0 \cdot 10^3 = 0 \text{ N/m}$$

$$D_{88} = 10 \cdot 10^{-3} \cdot 500 \cdot 10^3 + 16 \cdot 10^{-3} \cdot 440 \cdot 10^3 + 12 \cdot 10^{-3} \cdot 500 \cdot 10^3 = 18040 \text{ N/m}$$

Stiffness matrix elements (shear)

As already mentioned in Chapter 2.3, the calculation procedure for shear elements of the stiffness matrix is not introduced. However, it can be verified that the following relation applies

$$\max_i \left(\frac{5}{6} G_{i;11} t_i \right) \le D_{44} \le \frac{5}{6} \max_i \left(G_{i;11} \right) \sum_{i=1}^{n} t_i \tag{8.5}$$

$$\max_i \left(\frac{5}{6} G_{i;22} t_i \right) \le D_{55} \le \frac{5}{6} \max_i \left(G_{i;22} \right) \sum_{i=1}^{n} t_i \tag{8.6}$$

where

$$\mathbf{G}_i = \begin{bmatrix} G_{i;11} & G_{i;12} \\ \text{sym.} & G_{i;22} \end{bmatrix} = \mathbf{T}_{2\times2;i}{}^{\mathrm{T}} \mathbf{G}'_i \mathbf{T}_{2\times2;i} \tag{8.7}$$

where

$$\mathbf{G}'_i = \begin{bmatrix} G_{xz} & 0 \\ 0 & G_{yz} \end{bmatrix} \text{ and } \mathbf{T}_{2\times2;i} = \begin{bmatrix} \cos(\beta_i) & \sin(\beta_i) \\ -\sin(\beta_i) & \cos(\beta_i) \end{bmatrix} \tag{8.8}$$

The individual elements then are

$$G_{i;11} = c^2 G_{i;xz} + s^2 G_{i;yz}$$

$$G_{i;12} = cs G_{i;xz} - cs G_{i;yz}$$

$$G_{i;22} = s^2 G_{i;xz} + c^2 G_{i;yz} \text{ , where } c = \cos(\beta_i),\ s = \sin(\beta_i)$$

The shear stiffness matrix elements adapted from RF-LAMINATE are the following

$$D_{44} = 5000.00 \text{ kN/m}$$

$$D_{55} = 5866.67 \text{ kN/m}$$

$$\mathbf{G}_1 = \mathbf{G}'_1 = \begin{bmatrix} 500 & 0 \\ 0 & 50 \end{bmatrix}$$

$$\mathbf{G}_3 = \mathbf{G}'_3 = \begin{bmatrix} 500 & 0 \\ 0 & 50 \end{bmatrix}$$

$$\mathbf{G}'_2 = \begin{bmatrix} 440 & 0 \\ 0 & 44 \end{bmatrix},\ c = \cos 90° = 0,\ s = \sin 90° = 1$$

$$G_{2;11} = 0^2 \cdot 440 + 1^2 \cdot 44 = 44 \text{ MPa}$$

$$G_{2;12} = 0 \cdot 1 \cdot 440 - 0 \cdot 1 \cdot 44 = 0 \text{ MPa}$$

$$G_{2;22} = 1^2 \cdot 440 + 0^2 \cdot 44 = 440 \text{ MPa}$$

$$\mathbf{G}_2 = \begin{bmatrix} 44 & 0 \\ 0 & 440 \end{bmatrix}$$

$$\max\left(\frac{5}{6}500 \cdot 10; \frac{5}{6}44 \cdot 16; \frac{5}{6}500 \cdot 12\right) \leq D_{44} \leq \frac{5}{6}\max(500;44;500) \cdot (10+16+12)$$

$$5000 \leq 5000 \leq 15833$$

$$\max\left(\frac{5}{6}50 \cdot 10; \frac{5}{6}440 \cdot 16; \frac{5}{6}50 \cdot 12\right) \leq D_{55} \leq \frac{5}{6}\max(50;440;59) \cdot (10+16+12)$$

$$5866.67 \leq 5866.67 \leq 13933.33$$

Global stiffness matrix

$$\mathbf{D} = \begin{bmatrix} 33.85 & 0.24 & 0 & 0 & 0 & 124.49 & 0.13 & 0 \\ & 3.64 & 0 & 0 & 0 & 0.13 & -107.82 & 0 \\ & & 2.26 & 0 & 0 & 0 & 0 & 0.96 \\ & & & 5000.00 & 0 & 0 & 0 & 0 \\ & & & & 5866.67 & 0 & 0 & 0 \\ & & \text{sym.} & & & 179923 & 1927 & 0 \\ & & & & & & 118095 & 0 \\ & & & & & & & 18040 \end{bmatrix}$$

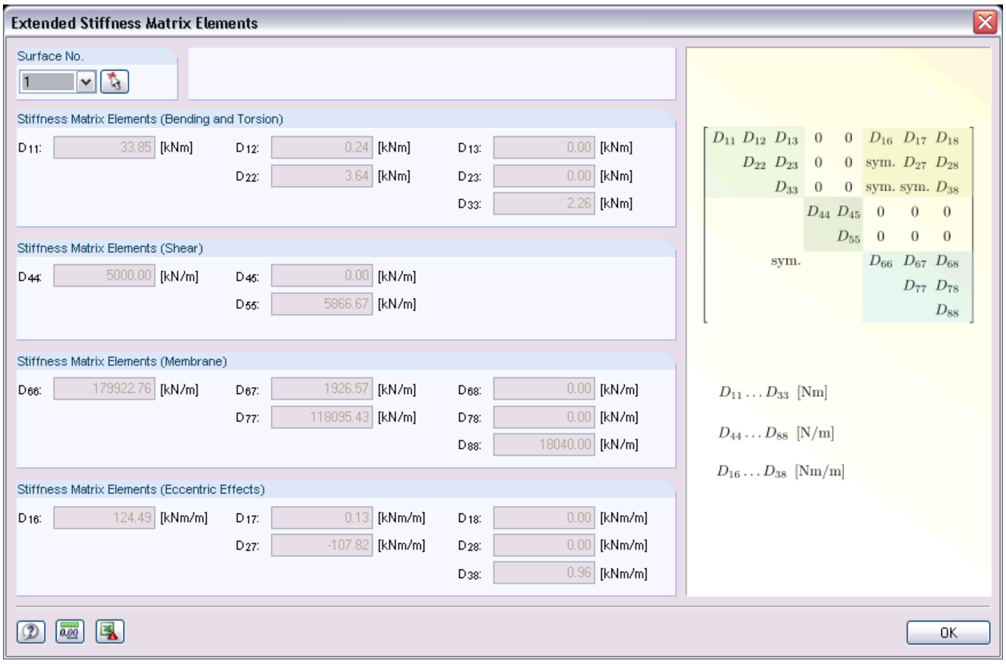


Figure 8.7: Dialog box *Extended Stiffness Matrix Elements* from RF-LAMINATE – with shear coupling of layers

8.1.2 Shear Coupling of Layers Is Not Considered

Because angles β_i are multiples of $90°$, the global stiffness matrix has the form

$$\mathbf{D} = \begin{bmatrix} D_{11} & D_{12} & 0 & 0 & 0 & 0 & 0 & 0 \\ & D_{22} & 0 & 0 & 0 & 0 & 0 & 0 \\ & & D_{33} & 0 & 0 & 0 & 0 & 0 \\ & & & D_{44} & 0 & 0 & 0 & 0 \\ & & & & D_{55} & 0 & 0 & 0 \\ & & \text{sym.} & & & D_{66} & D_{67} & 0 \\ & & & & & & D_{77} & 0 \\ & & & & & & & D_{88} \end{bmatrix} \qquad (8.9)$$

Stiffness matrix elements (bending and torsion)

$$D_{11} = \sum_{i=1}^{n} \frac{t_i^3}{12} d_{i;11} \qquad D_{12} = \sum_{i=1}^{n} \frac{t_i^3}{12} d_{i;12}$$

$$D_{22} = \sum_{i=1}^{n} \frac{t_i^3}{12} d_{i;22}$$

$$D_{33} = \sum_{i=1}^{n} \frac{t_i^3}{12} d_{i;33}$$

$$\mathbf{d}_1 = \begin{bmatrix} 8010.81 & 54.07 & 0 \\ 54.07 & 270.36 & 0 \\ 0 & 0 & 500.00 \end{bmatrix} \text{MN/m}^2$$

$$\mathbf{d}_2 = \begin{bmatrix} 230.30 & 46.06 & 0 \\ 46.06 & 7009.21 & 0 \\ 0 & 0 & 440.00 \end{bmatrix} \text{MN/m}^2$$

$$\mathbf{d}_3 = \begin{bmatrix} 8010.81 & 54.07 & 0 \\ 54.07 & 270.36 & 0 \\ 0 & 0 & 500.00 \end{bmatrix} \text{MN/m}^2$$

$$D_{11} = \frac{0.010^3}{12} 8010.81 \cdot 10^3 + \frac{0.016^3}{12} 230.30 \cdot 10^3 + \frac{0.012^3}{12} 8010.81 \cdot 10^3 = 1.900 \text{ kNm}$$

$$D_{12} = \frac{0.010^3}{12} 54.07 \cdot 10^3 + \frac{0.016^3}{12} 46.06 \cdot 10^3 + \frac{0.012^3}{12} 54.07 \cdot 10^3 = 0.028 \text{ kNm}$$

$$D_{22} = \frac{0.010^3}{12} 270.36 \cdot 10^3 + \frac{0.016^3}{12} 7009.21 \cdot 10^3 + \frac{0.012^3}{12} 270.36 \cdot 10^3 = 2.454 \text{ kNm}$$

$$D_{33} = \frac{0.010^3}{12} 500 \cdot 10^3 + \frac{0.016^3}{12} 440.00 \cdot 10^3 + \frac{0.012^3}{12} 500 \cdot 10^3 = 0.264 \text{ kNm}$$

Stiffness matrix elements (membrane)

$$D_{66} = \sum_{i=1}^{n} t_i \, d_{i;11} \qquad D_{67} = \sum_{i=1}^{n} t_i \, d_{i;12}$$

$$D_{77} = \sum_{i=1}^{n} t_i \, d_{i;22}$$

$$D_{88} = \sum_{i=1}^{n} t_i \, d_{i;33}$$

$$D_{66} = 10 \cdot 10^{-3} \cdot 8010.81 \cdot 10^3 + 16 \cdot 10^{-3} \cdot 230.30 \cdot 10^3 + 12 \cdot 10^{-3} \cdot 8010.81 \cdot 10^3 = 179923 \text{ N/m}$$

$$D_{67} = 10 \cdot 10^{-3} \cdot 54.07 \cdot 10^3 + 16 \cdot 10^{-3} \cdot 46.06 \cdot 10^3 + 12 \cdot 10^{-3} \cdot 54.07 \cdot 10^3 = 1927 \text{ N/m}$$

$$D_{77} = 10 \cdot 10^{-3} \cdot 270.36 \cdot 10^3 + 16 \cdot 10^{-3} \cdot 7009.21 \cdot 10^3 + 12 \cdot 10^{-3} \cdot 270.36 \cdot 10^3 = 118095 \text{ N/m}$$

$$D_{88} = 10 \cdot 10^{-3} \cdot 500 \cdot 10^3 + 16 \cdot 10^{-3} \cdot 440 \cdot 10^3 + 12 \cdot 10^{-3} \cdot 500 \cdot 10^3 = 18040 \text{ N/m}$$

Stiffness matrix elements (shear)

$$D_{44} = \sum_{i=1}^{n} \frac{5}{6} G_{i;11} t_i \qquad D_{55} = \sum_{i=1}^{n} \frac{5}{6} G_{i;22} t_i$$

where

$$\mathbf{G}_i = \begin{bmatrix} G_{i;11} & G_{i;12} \\ \text{sym.} & G_{i;22} \end{bmatrix} = \mathbf{T}_{2\times2;i}{}^{\mathrm{T}} \mathbf{G}'_i \mathbf{T}_{2\times2;i}, \text{ where } \mathbf{G}'_i = \begin{bmatrix} G_{xz} & 0 \\ 0 & G_{yz} \end{bmatrix} \text{ and}$$

$$\mathbf{T}_{2\times2;i} = \begin{bmatrix} \cos(\beta_i) & \sin(\beta_i) \\ -\sin(\beta_i) & \cos(\beta_i) \end{bmatrix}$$

The individual elements then are

$$G_{i;11} = c^2 G_{i;xz} + s^2 G_{i;yz}$$

$$G_{i;12} = csG_{i;xz} - csG_{i;yz}$$

$$G_{i;22} = s^2 G_{i;xz} + c^2 G_{i;yz}, \text{ where } c = \cos(\beta_i), \ s = \sin(\beta_i)$$

$$\mathbf{G}_1 = \mathbf{G}'_1 = \begin{bmatrix} 500 & 0 \\ 0 & 50 \end{bmatrix}$$

$$\mathbf{G}_3 = \mathbf{G}'_3 = \begin{bmatrix} 500 & 0 \\ 0 & 50 \end{bmatrix}$$

$$\mathbf{G}'_2 = \begin{bmatrix} 440 & 0 \\ 0 & 44 \end{bmatrix}, \ c = \cos 90° = 0, \ s = \sin 90° = 1$$

$$G_{2;11} = 0^2 \cdot 440 + 1^2 \cdot 44 = 44 \text{ MPa}$$

$$G_{2;12} = 0 \cdot 1 \cdot 440 - 0 \cdot 1 \cdot 44 = 0 \text{ MPa}$$

$$G_{2;22} = 1^2 \cdot 440 + 0^2 \cdot 44 = 440 \text{ MPa}$$

$$\mathbf{G}_2 = \begin{bmatrix} 44 & 0 \\ 0 & 440 \end{bmatrix}$$

$$D_{44} = \frac{5}{6} 500 \cdot 10^3 \cdot 0.010 + \frac{5}{6} 44 \cdot 10^3 \cdot 0.016 + \frac{5}{6} 500 \cdot 10^3 \cdot 0.012 = 9753 \text{ kN/m}$$

$$D_{55} = \frac{5}{6} 50 \cdot 10^3 \cdot 0.010 + \frac{5}{6} 440 \cdot 10^3 \cdot 0.016 + \frac{5}{6} 50 \cdot 10^3 \cdot 0.012 = 6783 \text{ kN/m}$$

Global stiffness matrix

$$
\mathbf{D} = \begin{bmatrix}
1.900 & 0.028 & 0 & 0 & 0 & 0 & 0 & 0 \\
 & 2.454 & 0 & 0 & 0 & 0 & 0 & 0 \\
 & & 0.264 & 0 & 0 & 0 & 0 & 0 \\
 & & & 9753 & 0 & 0 & 0 & 0 \\
 & & & & 6783 & 0 & 0 & 0 \\
 & & \text{sym.} & & & 179923 & 1927 & 0 \\
 & & & & & & 118095 & 0 \\
 & & & & & & & 18040
\end{bmatrix}
$$

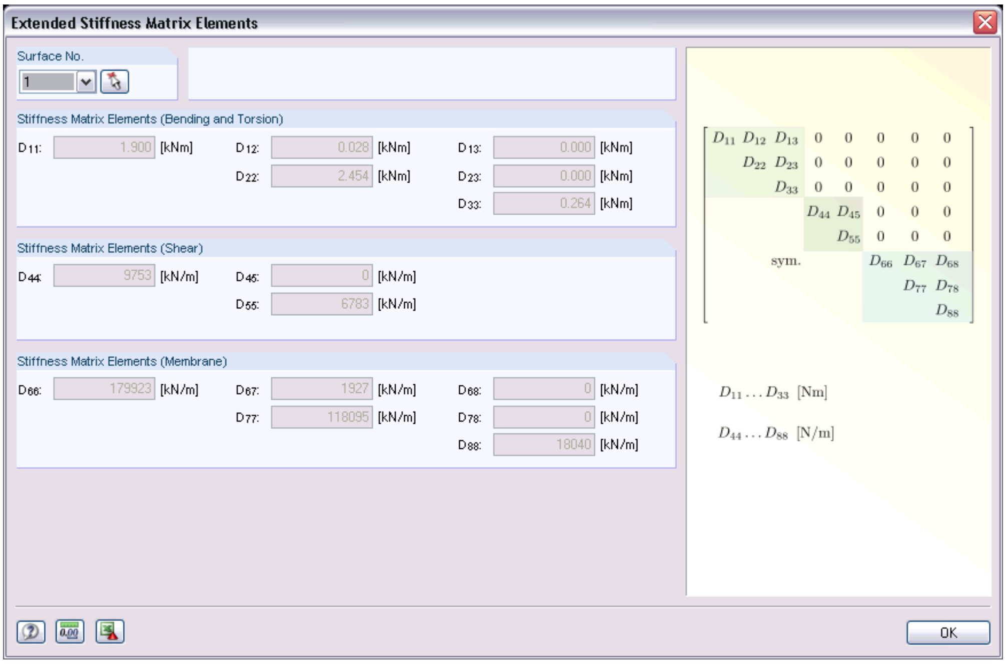


Figure 8.8: Dialog box *Extended Stiffness Matrix Elements* from RF-LAMINATE – without shear coupling of layers

8.2 Calculation of Stresses

Consider a plate from the previous example, consisting of three layers with material characteristics displayed in figure 8.10.

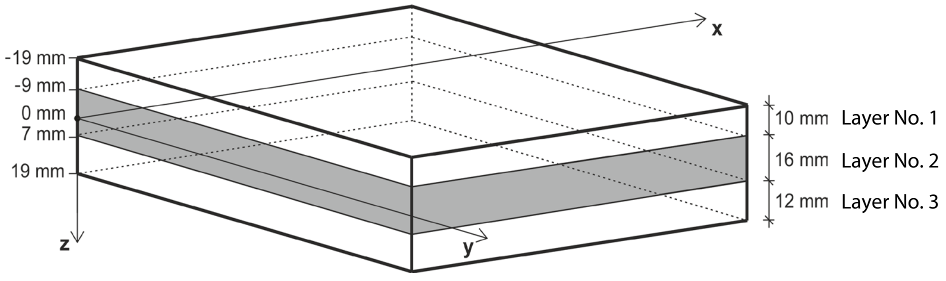


Figure 8.9: Layer scheme

Layers

Layer No.	A Material Description	B Thickness t [mm]	C Orthotropic Direction β [°]	D Modulus of Elasticity [N/mm²] E_x	E E_y	F Shear Modulus [N/mm²] G_{xz}	G G_{yz}	H G_{xy}	I Poisson's Ratio [-] ν_{xy}	J ν_{yx}	K Specific Weight γ [N/m³]
1	Poplar and Coniferous Timber C16	10.0	0.00	8000.0	270.0	500.0	50.0	500.0	0.200	0.007	3700.0
2	Coniferous Timber C14	16.0	90.00	7000.0	230.0	440.0	44.0	440.0	0.200	0.007	5000.0
3	Poplar and Coniferous Timber C16	12.0	0.00	8000.0	270.0	500.0	50.0	500.0	0.200	0.007	3700.0

Figure 8.10: Table 1.2 *Material Characteristics*

In the previous example from Chapter 8.1, the calculation of stiffness matrix elements is presented for the case of the shear coupling of layers and also for the case without the shear coupling of layers. Now, the calculation of stresses is carried out.

The plate with the dimensions 1.0 x 1.5 m is simply supported and loaded with a surface load of 5 kN/m².

8.2.1 Calculation of Individual Stress Components

By using the finite element method in RFEM, you get the stresses $\sigma_x, \sigma_y, \tau_{xy}, \tau_{xz}, \tau_{yz}$. The stress values in the point with coordinates [0.8, 0.8, 0], in the middle layer, are shown in the following pictures. In the first case it is the structure where the shear coupling of layers is considered, in the second case the shear coupling of layers is not considered.

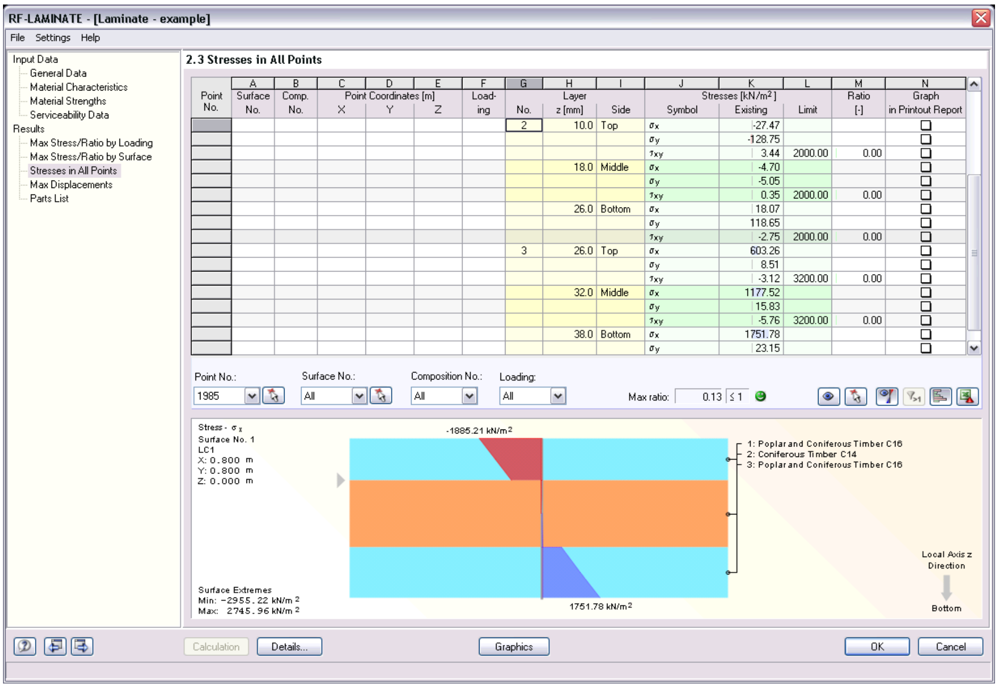


Figure 8.11: Window 2.3 *Stresses in All Points* – the example with the shear coupling of layers

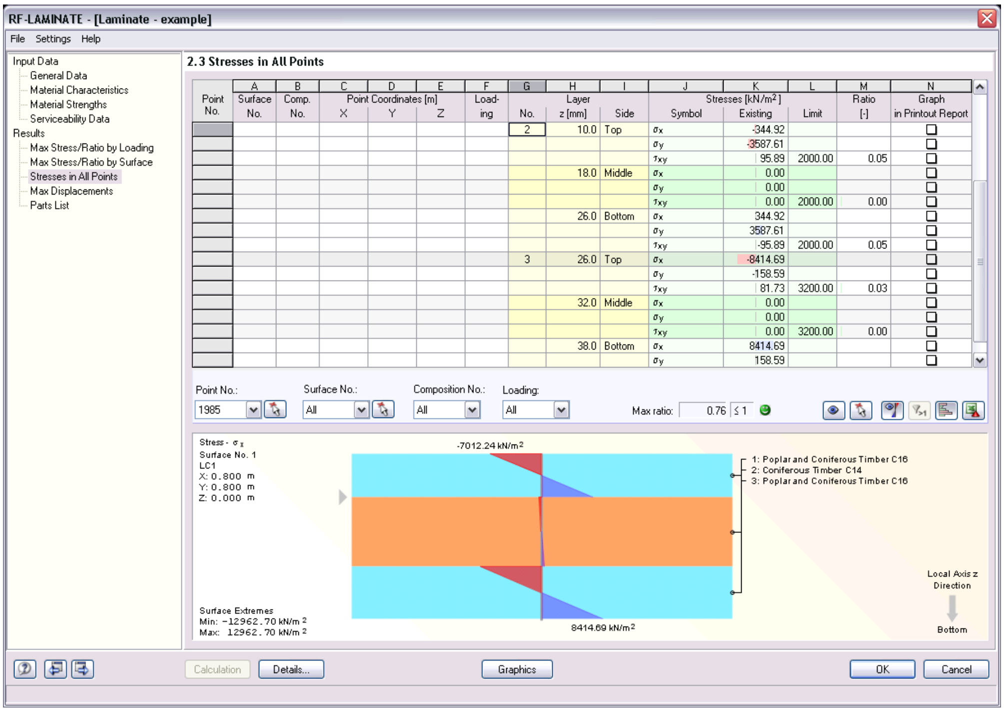


Figure 8.12: Window 2.3 *Stresses in All Points* – the example without the shear coupling of layers

Because the calculation of individual stress components for both cases is the same, only the case with the shear coupling of layers is presented here, therefore with the following values.

Point	*Side*	σ_x [kPa]	σ_y [kPa]	τ_{xy} [kPa]
x = 0.8 m,	top	-27.47	-128.75	3.44
y = 0.8 m,	middle	-4.70	-5.05	0.35
Layer No. 2	bottom	18.07	118.65	-2.75

Table 8.1: Stresses in Layer No. 2 in point [0.8;0.8]

The middle layer is rotated of the angle $\beta = 90°$

$$\sigma_{b+t/c,0} = \sigma_x \cos^2\beta + \tau_{xy}\sin 2\beta + \sigma_y \sin^2\beta$$

$$\sigma_{b+t/c,0(top)} = -27.47\cos^2 90° + 3.44 \cdot \sin(2 \cdot 90°) - 128.75\sin^2 90° = -128.75 \text{ kPa}$$

$$\sigma_{b+t/c,0(middle)} = -4.70\cos^2 90° + 0.35 \cdot \sin(2 \cdot 90°) - 5.05\sin^2 90° = -5.05 \text{ kPa}$$

$$\sigma_{b+t/c,0(bottom)} = 18.07\cos^2 90° - 2.75 \cdot \sin(2 \cdot 90°) + 118.65\sin^2 90° = 118.65 \text{ kPa}$$

$$\sigma_{b+t/c,90} = \sigma_x \sin^2\beta - \tau_{xy}\sin 2\beta + \sigma_y \cos^2\beta$$

$$\sigma_{b+t/c,90(top)} = -27.47\sin^2 90° - 3.44\sin(2 \cdot 90°) - 128.75\cos^2 90° = -27.47 \text{ kPa}$$

$$\sigma_{b+t/c,90(middle)} = -4.70\sin^2 90° - 0.35\sin(2 \cdot 90°) - 5.05\cos^2 90° = -4.70 \text{ kPa}$$

$$\sigma_{b+t/c,90(bottom)} = 18.07\sin^2 90° - (-2.75)\sin(2 \cdot 90°) + 118.65\cos^2 90° = 18.07 \text{ kPa}$$

$$\sigma_{t/c,0} = \frac{\sigma_{b+t/c,0(top)} + \sigma_{b+t/c,0(middle)} + \sigma_{b+t/c,0(bottom)}}{3}$$

$$\sigma_{t/c,0} = \frac{-128.75 - 5.05 + 118.65}{3} = -5.05 \text{ kPa}$$

$$\sigma_{t/c,90} = \frac{\sigma_{b+t/c,90(top)} + \sigma_{b+t/c,90(middle)} + \sigma_{b+t/c,90(bottom)}}{3}$$

$$\sigma_{t/c,90} = \frac{-27.47 - 4.70 + 18.07}{3} = -4.70 \text{ kPa}$$

$$\sigma_{b,0} = \sigma_{b+t/c,0} - \sigma_{t/c,0}$$

$$\sigma_{b,0(top)} = -128.75 - (-5.05) = -123.70 \text{ kPa}$$

$$\sigma_{b,0(middle)} = -5.05 - (-5.05) = 0 \text{ kPa}$$

$$\sigma_{b,0(bottom)} = 118.65 - (-5.05) = 123.70 \text{ kPa}$$

$$\sigma_{b,90} = \sigma_{b+t/c,90} - \sigma_{t/c,90}$$

$$\sigma_{b,90(top)} = -27.47 - (-4.70) = -22.77 \text{ kPa}$$

$$\sigma_{b,90(middle)} = -4.70 - (-4.70) = 0 \text{ kPa}$$

$$\sigma_{b,90(bottom)} = 18.07 - (-4.70) = 22.77 \text{ kPa}$$

8.2.2 Calculation Procedure in RF-LAMINATE

First, you need to create a *New Model* in RFEM.

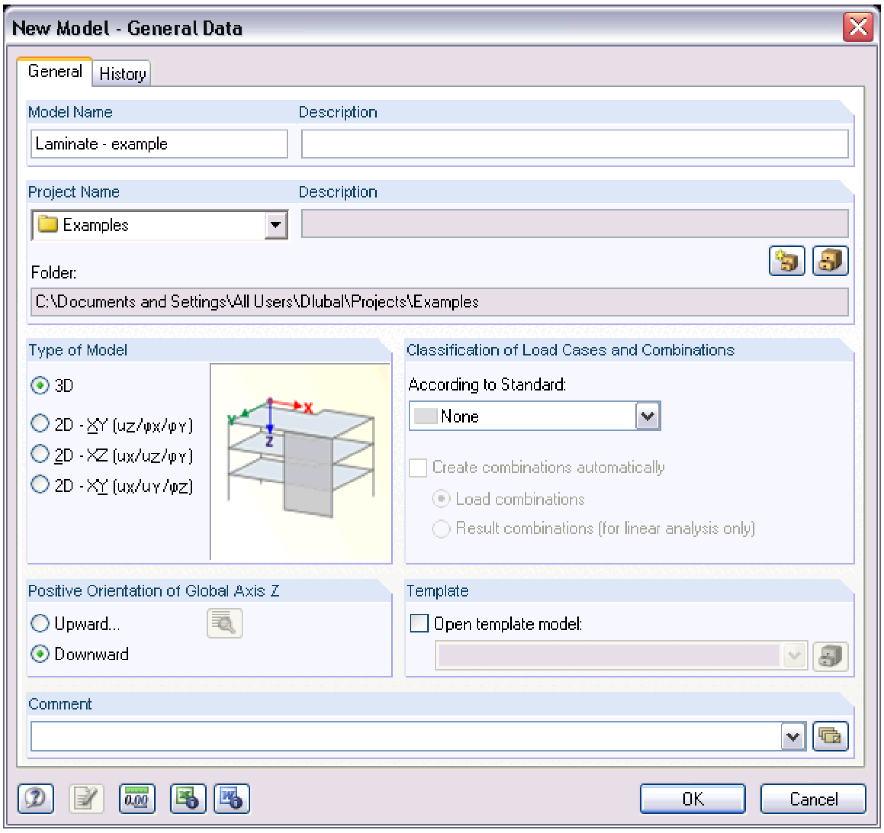


Figure 8.13: Creating a new model

After entering a new model, create a *New Surface*. Choose *Laminate* as the surface type and define the plate dimensions as 1.0 x 1.5 m.

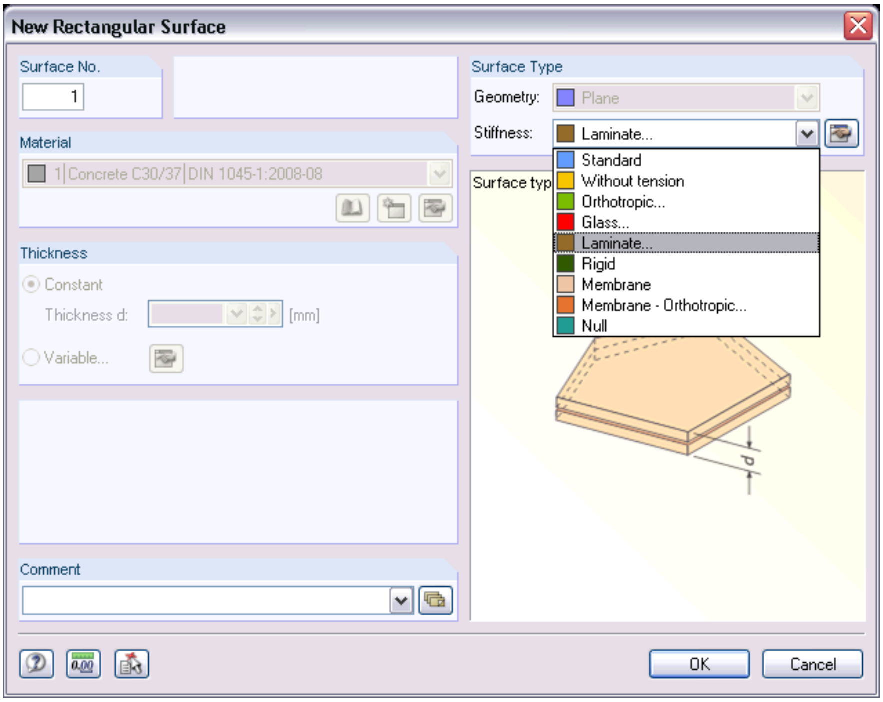
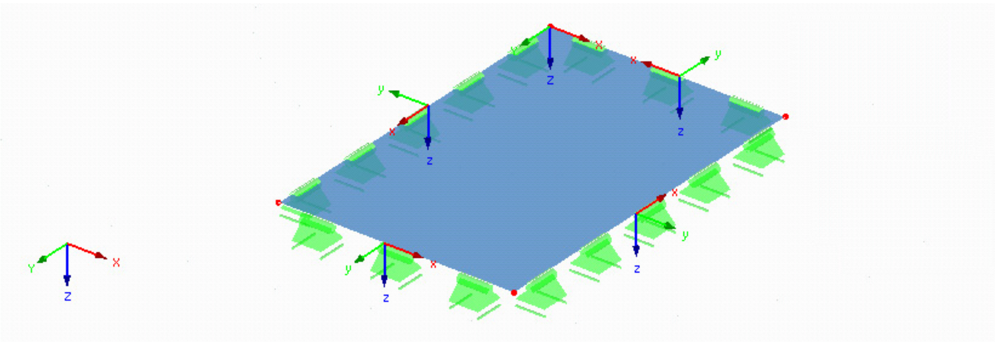
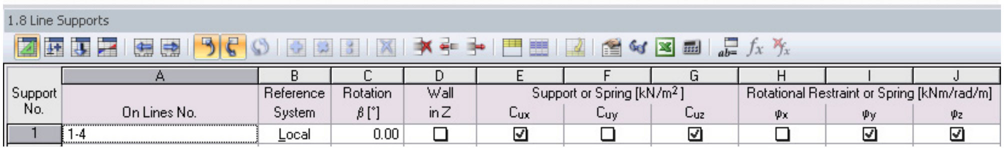


Figure 8.14: Dialog box *New Rectangular Surface*

Define the surfaces according to figure 8.15.




1.8 Line Supports

	A	B	C	D	E	F	G	H	I	J
Support No.	On Lines No.	Reference System	Rotation β [°]	Wall in Z	Support or Spring [kN/m²] Cux	Cuy	Cuz	Rotational Restraint or Spring [kNm/rad/m] φx	φy	φz
1	1-4	Local	0.00	☐	☑	☐	☑	☐	☑	☑

Figure 8.15: Table *Line Supports*

Now create a *New Load Case*.

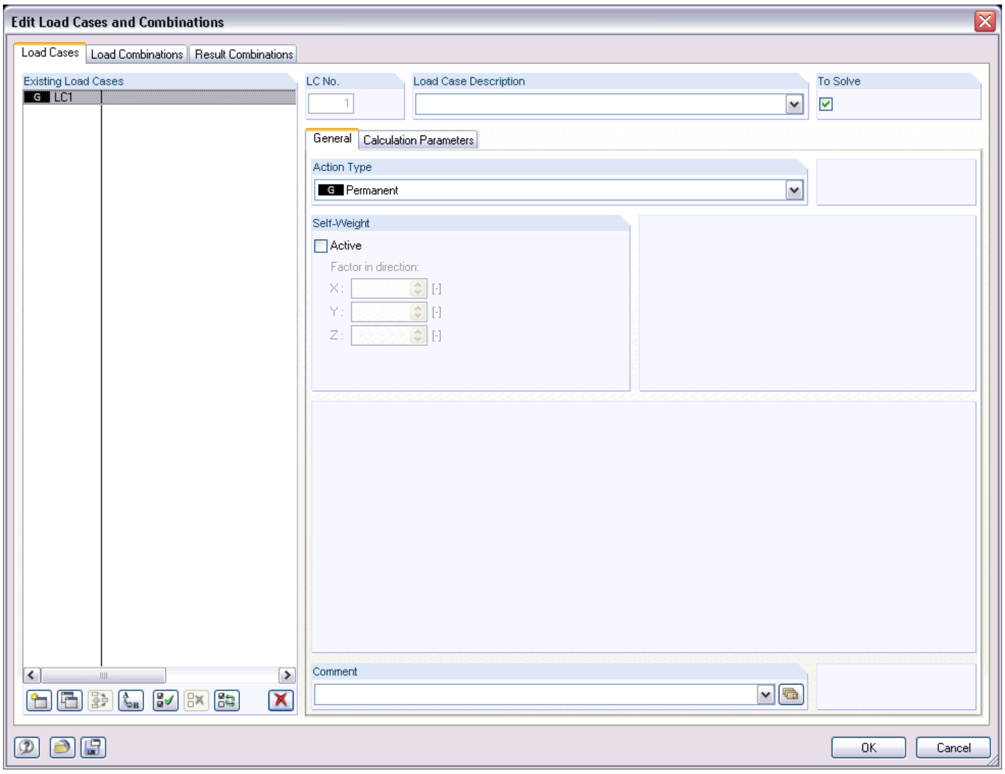


Figure 8.16: Dialog box *Edit Load Cases and Combinations* - the tab *Load Cases*

Next, fill in the dialog box *New Surface Load*.

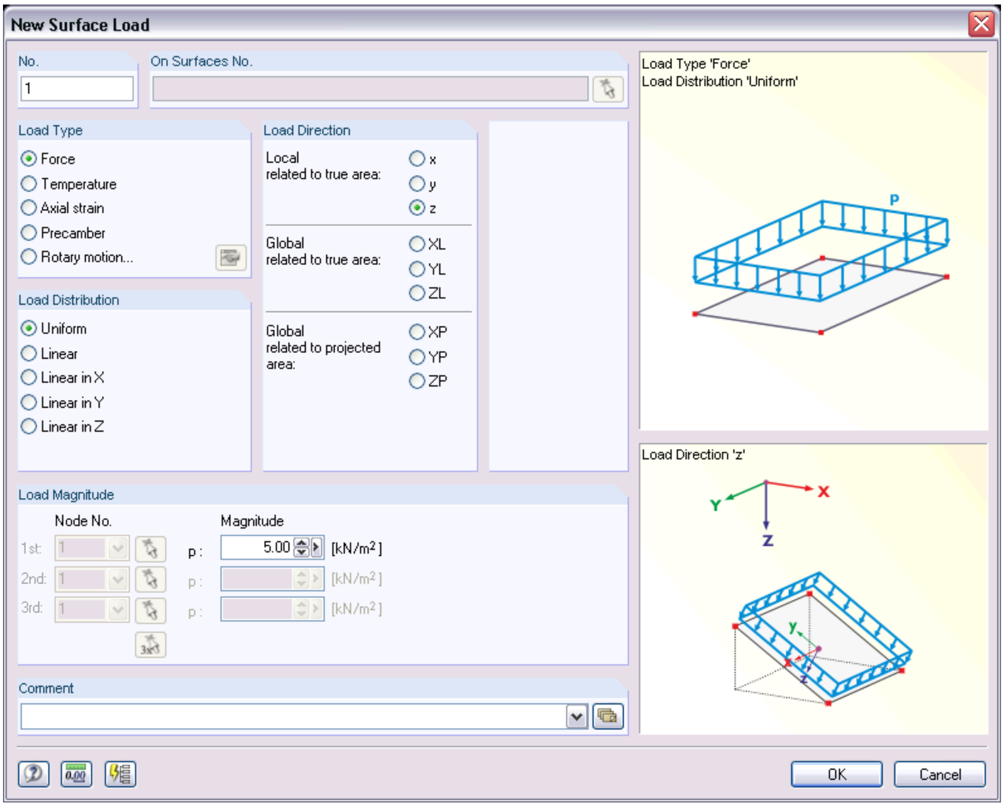


Figure 8.17: Dialog box *New Surface Load*

In the dialog box *FE Mesh Settings*, set the length of finite elements to 25 mm.

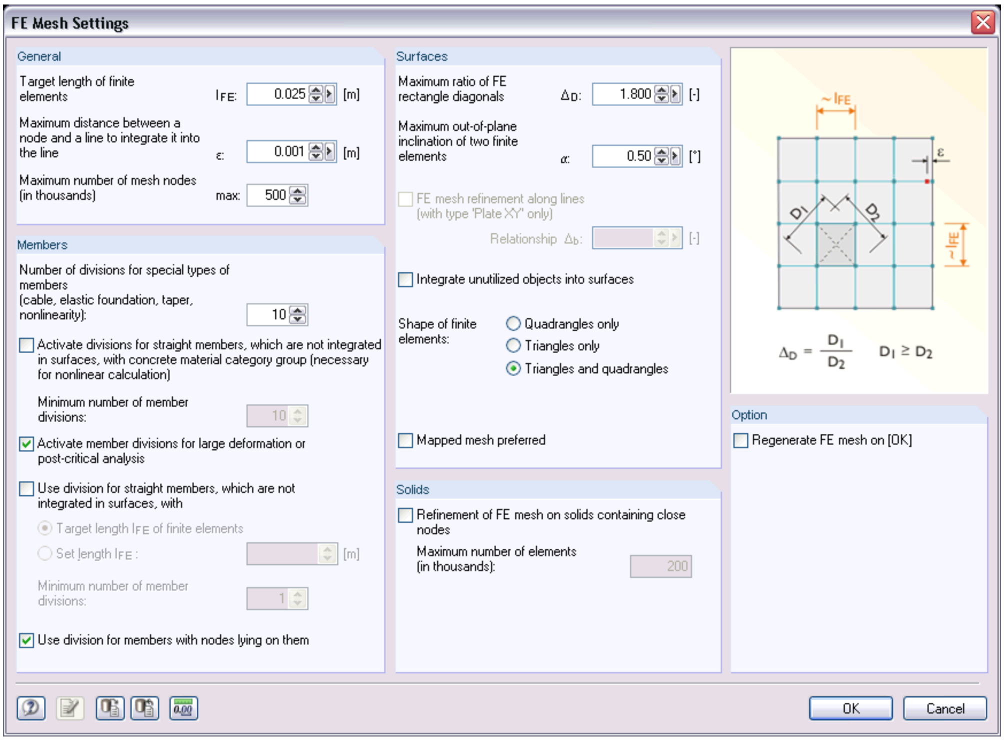


Figure 8.18: Dialog box *FE Mesh Settings*

Now you can open the RF-LAMINATE module and fill in the individual input windows.

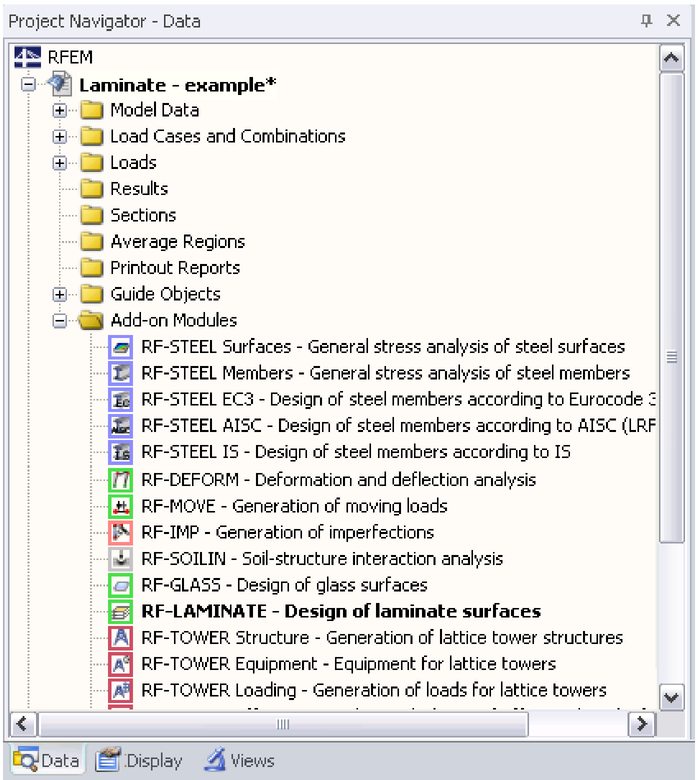


Figure 8.19: Navigator *Data*: *Add-on Modules* → *RF-LAMINATE*

In Window 1.1 *General Data*, select Surface No. 1. Then select the orthotropic material model and select LC1 for the design.

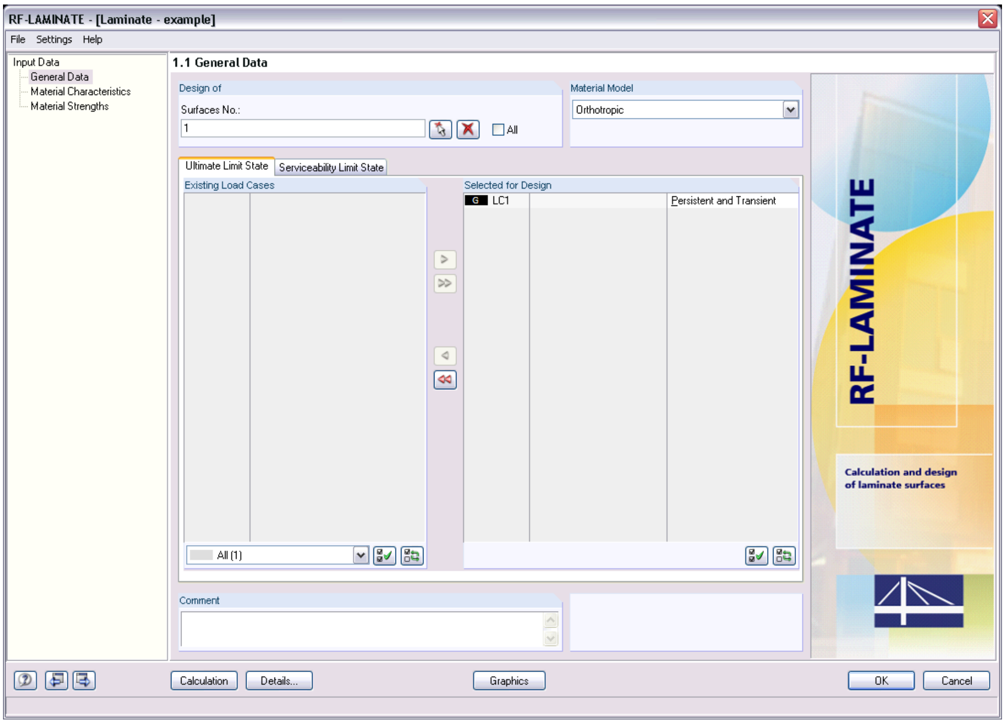


Figure 8.20: Window 1.1 *General Data*

In Window 1.2 *Material Characteristics*, select individual layers from the material library and assign the created composition to Surface No. 1.

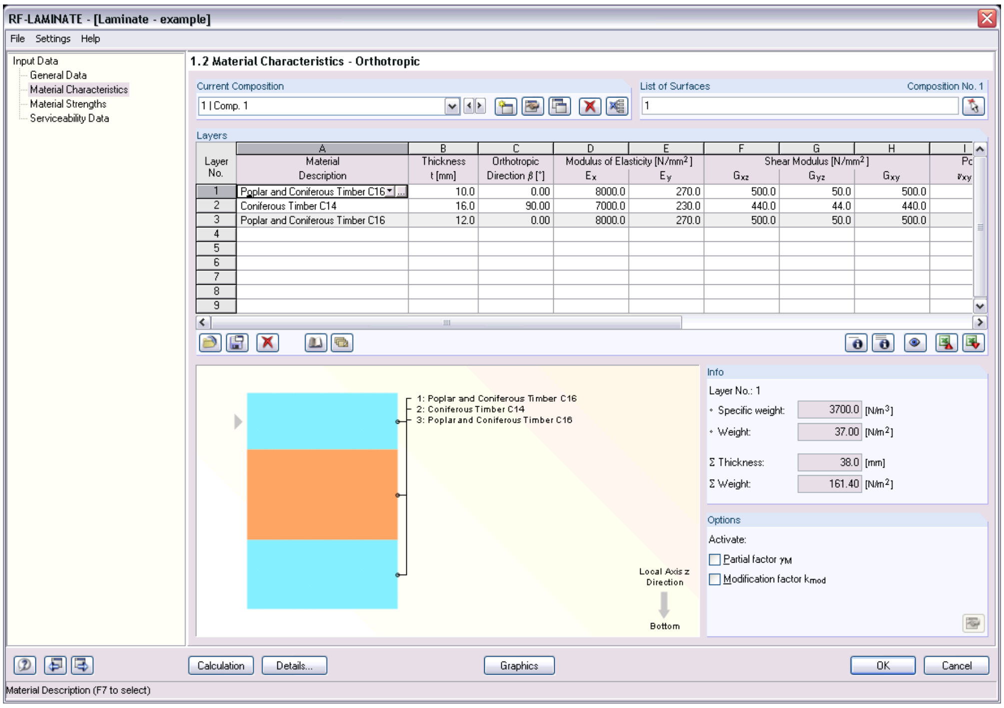


Figure 8.21: Window 1.2 *Material Characteristics*

In following the next window, 1.3 *Material Strengths*, characteristic strength values are imported from the material library automatically.

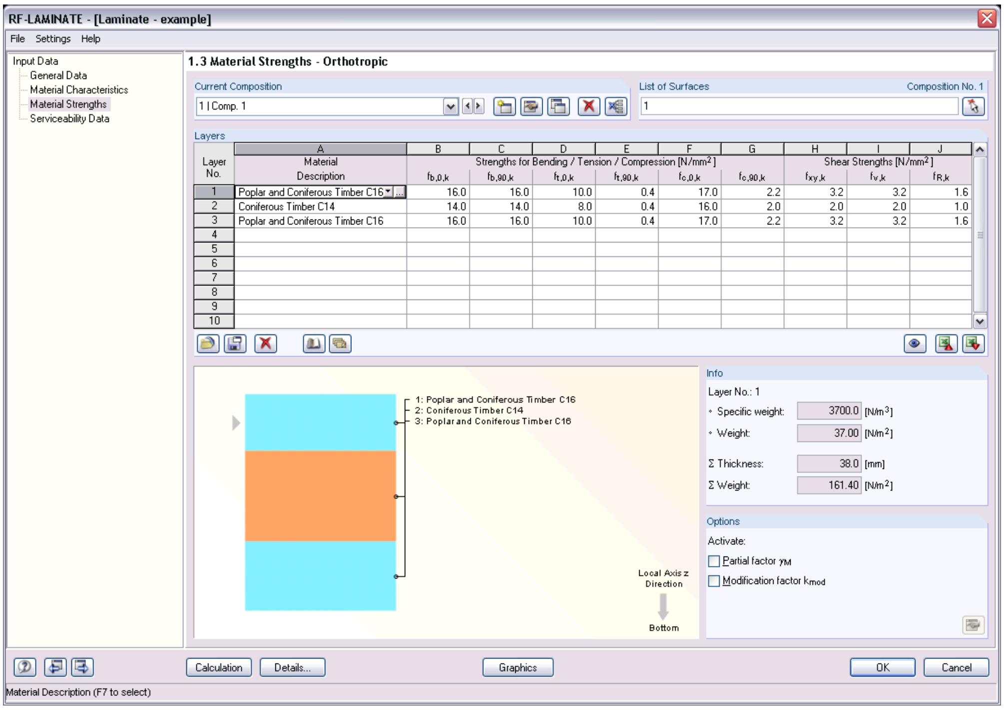


Figure 8.22: Window 1.3 *Material Strengths*

In the last input window, enter Surface No. 1 to the *List of Surfaces*. Because the *Manual* check box is not selected, the *Reference Length L* is completed automatically.

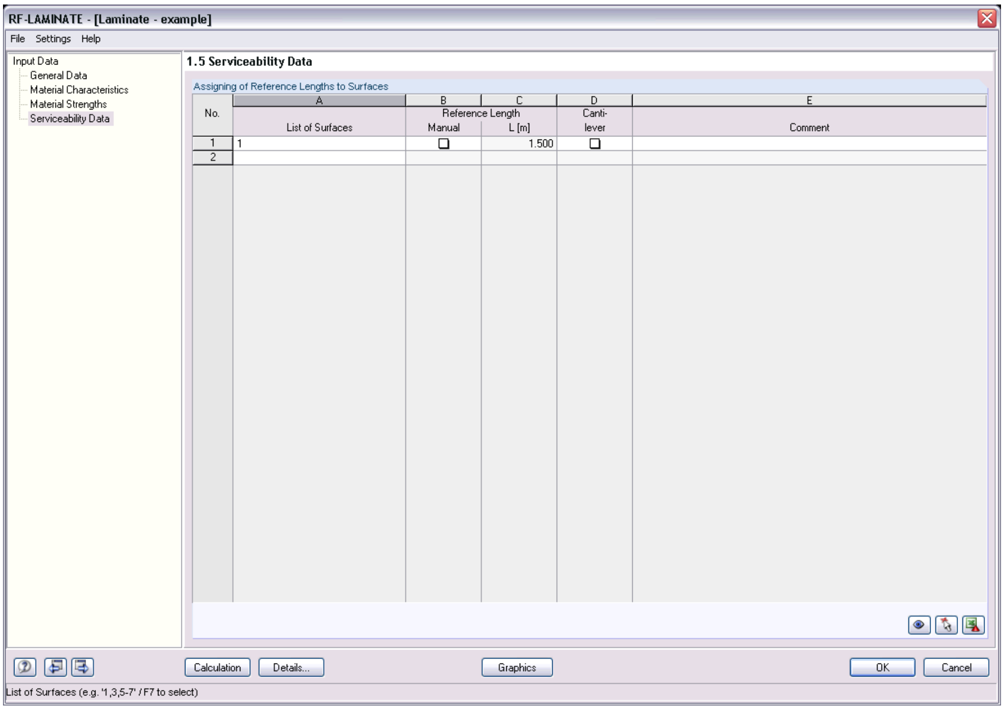


Figure 8.23: Window 1.5 *Serviceability Data*

Then, check the settings in the *Details* dialog box and start the calculation.

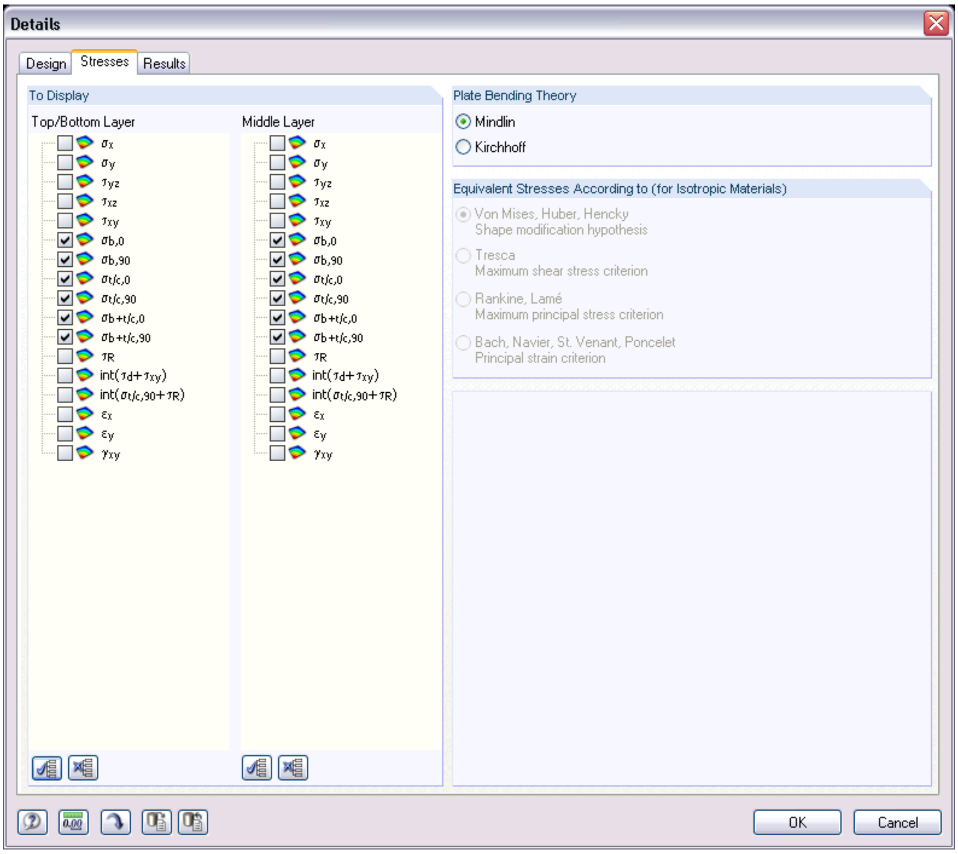


Figure 8.24: Dialog box *Details* – the *Stresses* tab

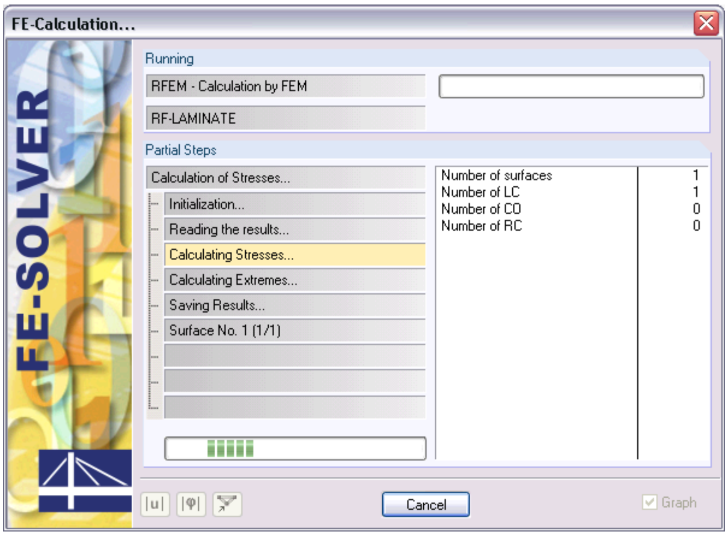


Figure 8.25: Dialog box *FE-Calculation…*

You can check the stress values in result windows and make sure that they match the calculation introduced in the previous chapter.

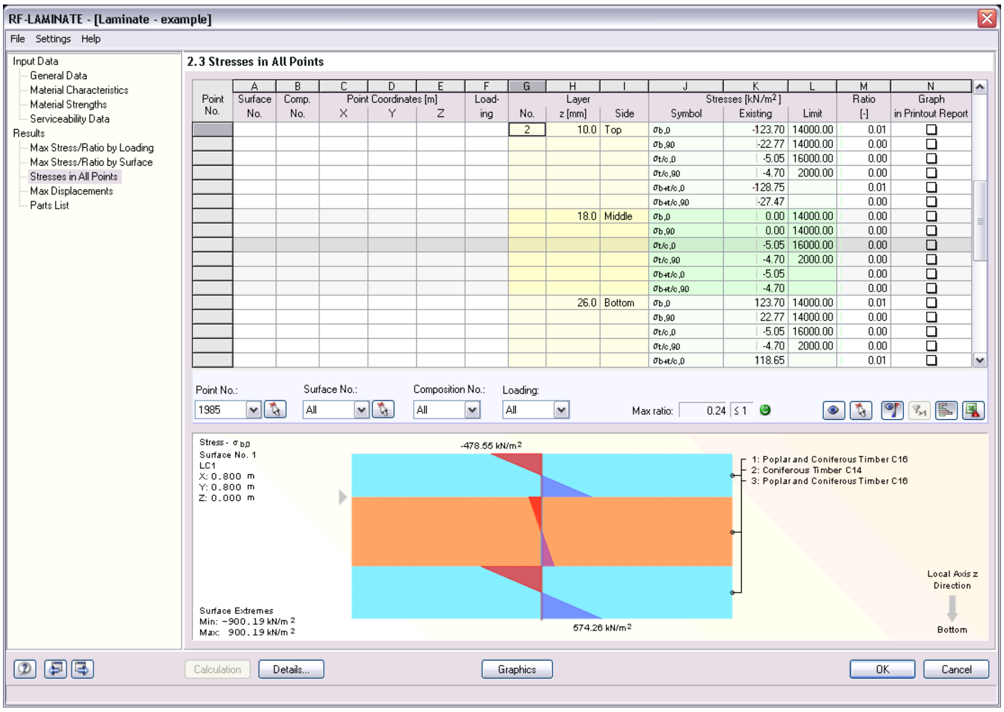
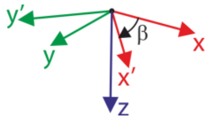


Figure 8.26: Window 2.3 *Stresses in All Points*

9. Annexes

9.1 Transformation Relations

The relations necessary for the transformation of stresses, strains and stiffness matrices at the rotation of the coordinate system x, y, z to the coordinate system x', y', z of angle β are summarized here. Angle β is defined as



The quantities related to the system x, y, z, such as stresses, strains and elements of stiffness matrices, are marked without an acute accent (´), the quantities in the system x', y', z are marked with an acute accent. The transformation relations for plane stresses and strains are the following

$$\begin{Bmatrix}\sigma'_x\\ \sigma'_y\\ \tau'_{xy}\end{Bmatrix} = \underbrace{\begin{bmatrix} c^2 & s^2 & 2cs\\ s^2 & c^2 & -2cs \\ -cs & cs & c^2-s^2\end{bmatrix}}_{\mathbf{T}_{3\times3}^{-\mathrm{T}}} \begin{Bmatrix}\sigma_x\\ \sigma_y\\ \tau_{xy}\end{Bmatrix}, \quad \begin{Bmatrix}\sigma_{\mathrm{b+t/c,0}}\\ \sigma_{\mathrm{b+t/c,90}}\end{Bmatrix} \equiv \begin{Bmatrix}\sigma'_x\\ \sigma'_y\end{Bmatrix} \tag{9.1}$$

$$\begin{Bmatrix}\varepsilon'_x\\ \varepsilon'_y\\ \gamma'_{xy}\end{Bmatrix} = \underbrace{\begin{bmatrix} c^2 & s^2 & cs\\ s^2 & c^2 & -cs \\ -2cs & 2cs & c^2-s^2\end{bmatrix}}_{\mathbf{T}_{3\times3}} \begin{Bmatrix}\varepsilon_x\\ \varepsilon_y\\ \gamma_{xy}\end{Bmatrix} \tag{9.2}$$

The stiffness matrix is transformed according to the relation

$$\mathbf{d} = \mathbf{T}_{2\times2}^{\mathrm{T}}\mathbf{d}'\mathbf{T}_{2\times2} \quad \Leftrightarrow \quad \mathbf{d}' = \mathbf{T}_{2\times2}^{-\mathrm{T}}\mathbf{d}\mathbf{T}_{2\times2}^{-1} \tag{9.3}$$

$$\mathbf{d} = \begin{bmatrix} d_{11} & d_{12} & d_{13}\\ & d_{22} & d_{23}\\ \text{sym.} & & d_{33}\end{bmatrix}, \quad \mathbf{d}' = \begin{bmatrix} d'_{11} & d'_{12} & 0\\ & d'_{22} & 0\\ \text{sym.} & & d'_{33}\end{bmatrix} \tag{9.4}$$

The transformation relations for shear stresses and strains are the following

$$\begin{Bmatrix}\tau'_{xz}\\ \tau'_{yz}\end{Bmatrix} = \underbrace{\begin{bmatrix} c & s\\ -s & c\end{bmatrix}}_{\mathbf{T}_{2\times2}} \begin{Bmatrix}\tau_{xz}\\ \tau_{yz}\end{Bmatrix}, \quad \begin{Bmatrix}\tau_d\\ \tau_{\mathrm{R}}\end{Bmatrix} \equiv \begin{Bmatrix}\tau'_{xz}\\ \tau'_{yz}\end{Bmatrix} \tag{9.5}$$

$$\begin{Bmatrix}\gamma'_{xz}\\ \gamma'_{yz}\end{Bmatrix} = \underbrace{\begin{bmatrix} c & s\\ -s & c\end{bmatrix}}_{\mathbf{T}_{2\times2}} \begin{Bmatrix}\gamma_{xz}\\ \gamma_{yz}\end{Bmatrix} \tag{9.6}$$

The stiffness matrix is transformed according to the relation

$$\mathbf{G} = \mathbf{T}_{2\times2}^{\mathrm{T}}\mathbf{G}'\mathbf{T}_{2\times2} \quad \Leftrightarrow \quad \mathbf{G}' = \mathbf{T}_{2\times2}\mathbf{G}\mathbf{T}_{2\times2}^{\mathrm{T}} \tag{9.7}$$

$$\mathbf{G} = \begin{bmatrix} G_{11} & G_{12}\\ \text{sym.} & G_{22}\end{bmatrix}, \quad \mathbf{G}' = \begin{bmatrix} G'_{11} & 0\\ 0 & G'_{22}\end{bmatrix} \tag{9.8}$$

9.2 Checking the Positive Definiteness of the Stiffness Matrix

The positive definiteness of the global stiffness matrix in necessary for calculation.

Generally, the global stiffness matrix has the shape

$$\mathbf{D}_{8\times8}=\begin{bmatrix}\mathbf{D}_{3\times3}^{\text{bending}} & \mathbf{0} & \mathbf{D}_{3\times3}^{\text{eccentric}}\\ \mathbf{0} & \mathbf{D}_{2\times2}^{\text{shear}} & \mathbf{0}\\ \mathbf{D}_{3\times3}^{\text{eccentric}} & \mathbf{0} & \mathbf{D}_{3\times3}^{\text{membrane}}\end{bmatrix}=$$

$$=\begin{bmatrix}D_{11} & D_{12} & D_{13} & & & D_{16} & D_{17} & D_{18}\\ & D_{22} & D_{23} & & & \text{sym.} & D_{27} & D_{28}\\ & & D_{33} & & & \text{sym.} & \text{sym.} & D_{38}\\ & & & D_{44} & D_{45} & & & \\ & & & & D_{55} & & & \\ & & & & & D_{66} & D_{67} & D_{68}\\ & \text{sym.} & & & & & D_{77} & D_{78}\\ & & & & & & & D_{88}\end{bmatrix} \qquad (9.9)$$

The following conditions are checked:

1. Matrix $\mathbf{D}$ is positive-definite (that is all of its leading principal minors are positive).
2. In addition, it is required that the submatrices $\mathbf{D}_{3\times3}^{\text{bending}}$, $\mathbf{D}_{2\times2}^{\text{shear}}$, $\mathbf{D}_{3\times3}^{\text{membrane}}$ are positive-definite in a more restrictive sense – all of its leading principal minors must satisfy:

$$\det\begin{bmatrix}D_{11} & & \\ & \ddots & \\ & & D_{ii}\end{bmatrix}\geq\sqrt{0.001}\prod_{i=1}\left|D_{ii}\right|,\quad \text{pro}\quad i=1,\ldots,n \qquad (9.10)$$

A Literature

[1] ALTENBACH, Holm ; ALTENBACH, Johannes; NAUMENKO, Konstantin. Ebene Flächentragwerke. Grundlagen der Modellierung und Berechnung von Scheiben und Platten. [s.l.] : Springer Verlag, 2008. 491 p.

[2] DIN 1052. Entwurf, Berechnung und Bemessung von Holzbauwerken : Allgemein Bemessungsregeln und Bemessungsregeln für den Hochbau. [s.l.] : [s.n.], 2008. 239 p.

[3] ČSN 73 1702. Navrhování, výpočet a posuzování dřevěných stavebních konstrukcí : Obecná pravidla a pravidla pro pozemní stavby. Praha : Český normalizační institut, November 2007. 174 p.

[4] Huber M.T.: The theory of crosswise reinforced ferroconcrete slabs and its application to various constructional problems involving rectangular slabs, Der Bauingenieur, 1923, 4(12), 354-360 and 4(13), 392-395.

B Index