Version
September 2016

Add-on Module

RF-LAMINATE

Design of Laminate Surfaces

Program Description

© Dlubal Software GmbH 2016
Am Zellweg 2
D-93464 Tiefenbach
Germany

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 GMBH calculates the 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 comprehensive material library. Furthermore, you can define new materials and add them to the library.

In RF-LAMINATE, you can create structures with different material models. Apart from isotropic and orthotropic material models, user-defined and hybrid models are available which allow for a combination of isotropic and orthotropic materials in one composition. The individual layers of orthotropic materials can be rotated by a specific angle β so that different properties can be considered in the relevant directions. You can also decide whether the shear coupling of the layers is to be considered in the calculation or not.

Due to its clear layout and intuitive 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. Yet it is not only an "optical" part of the main program: The results of the module, including graphical representations, can be incorporated in the global printout report. Therefore, the entire analysis can be easily and, above all, uniformly arranged and organized. The similar conception of all DLUBAL modules facilitates the 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 team from DLUBAL SOFTWARE GMBH.

1.2 Using the Manual

Topics such as operation system requirements or installation are described in the RFEM manual. Therefore, we put them aside in this description. We will rather focus on the specific features of the RF-LAMINATE module.

When describing RF-LAMINATE, we keep to the sequence and structure of the input and result windows of the module. The described **buttons** are introduced in the text in square brackets, for example [Details]. They are also displayed on the left margin. All terms mentioned in dialog boxes, windows or menus are written in *italics* so that you can easily find them in the program.

In this manual, an index for a quick search of certain terms is included. If you still cannot find what you need, please check our blog website https://www.dlubal.com/blog/en where you can browse the posts and find suitable suggestions.

1.3 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**.

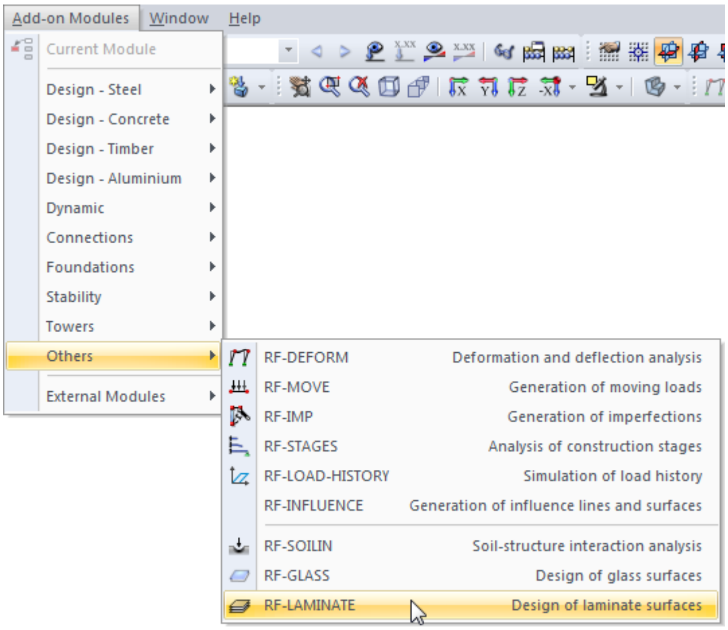


Figure 1.1: Main menu *Add-on Modules* → *Others* → *RF-LAMINATE*

Navigator

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

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

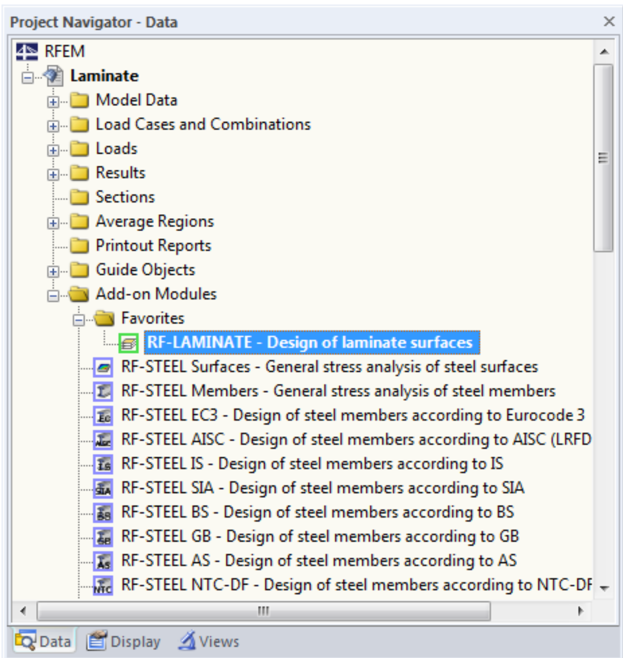
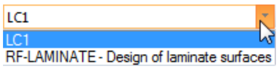


Figure 1.2: *Data* navigator *Add-on Modules* → *RF-LAMINATE*

Panel



If RF-LAMINATE results are already available in the 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 then display deformations or stresses.

The [RF-LAMINATE] button is displayed in the panel. You can start RF-LAMINATE by clicking that button.

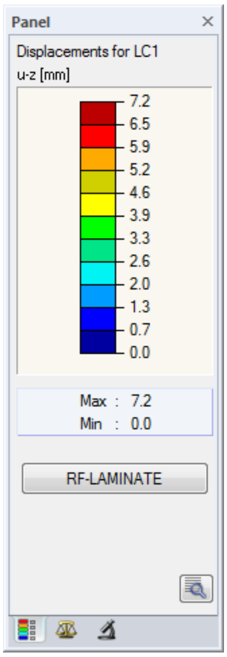


Figure 1.3: Panel button [RF-LAMINATE]

2 Theory

This chapter introduces the theoretical principles that are required for working with RF-LAMINATE.

2.1 Symbols

t	Thickness of composition [m]
t_i	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 modulus 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/m^3]
α_T	Coefficient of thermal expansion [K^{-1}]
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 Multilayered Structure Modeling

RF-LAMINATE is based on the plate theory. The calculation according to this theory has its limits in the case of plates with considerable thicknesses. An approximative criterion for the valid calculation according to the plate theory is given by the relation $t/L \leq 0.05$, where t is the thickness and L is the length of the plate side (or the characteristic dimension of the model). If the relation $t/L \leq 0.05$ is not satisfied, the solid element model should be considered.

Another problem in multilayer structure modeling arises when the stiffnesses of the layers differ significantly. An extreme example is a three-layered sandwich element consisting of a foam core surrounded by two thin metal sheets (see Figure 2.1). In this case, shear plays an important role. The line connecting the deformed points is no longer straight (see Figure 2.2). The 2D plate theory then yields incorrect results. It is recommended to use the solid element model in RFEM instead.

Figure 2.1: Three-layered sandwich element

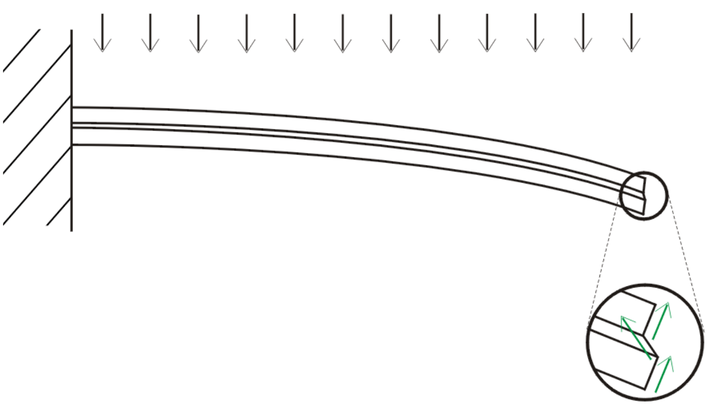
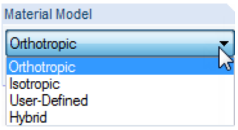


Figure 2.2: Shear distortion

2.3 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*

2.3.1 Orthotropic

The properties of an orthotropic material are distinct in each of the directions. Therefore, the material is defined by using two moduli of elasticity (E_x, E_y), three shear moduli (G_{yz}, G_{xz}, G_{xy}) and two Poisson's ratios (ν_{xy}, ν_{yx}).

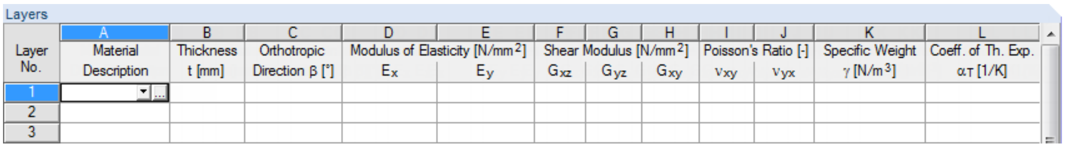


Figure 2.3: Orthotropic material model

The moduli of elasticity and the shear moduli must satisfy: $E_x \geq 0, E_y \geq 0, G_{yz} \geq 0, G_{xz} \geq 0, G_{xy} \geq 0$. The global stiffness matrix ***D*** has to be positive-definite.

Please note that – contrary to the isotropic material model where the values E, G and ν are mutually dependent according to Equation 2.14 – no such relation exists for the orthotropic material model. The values of E_x, E_y, ν_{xy} and G_{xy} are fully independent of each other.

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 CLT plates or rolled metal sheets.

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

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

in the case that the grain runs in x'-direction, that is $\mathrm{E_x} > \mathrm{E_y}$. In literature, you can sometimes also find the second way how to define the Poisson's ratios. Let us denote those ratios by overlines. For them, the equation $\overline{\nu}_{\mathrm{yx}}/\mathrm{E_x} = \overline{\nu}_{\mathrm{xy}}/\mathrm{E_y}$ is presumed, leading to the inequality $\overline{\nu}_{\mathrm{xy}} < \overline{\nu}_{\mathrm{yx}}$. 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, the material parameters are taken from standards. For example, the values of softwood timber of strength class C24 are given in EN 338, Table 1.

$$\begin{aligned} E_{0,\mathrm{mean}} &= 11{,}000\ \mathrm{N/mm^2} \\ E_{90,\mathrm{mean}} &= 370\ \mathrm{N/mm^2} \\ G_{\mathrm{mean}} &= 690\ \mathrm{N/mm^2} \end{aligned} \tag{2.3}$$

It is assumed by default that the grain runs in x'-direction. In this case, the values represent

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

where $\mathrm{G_{yz}}$ is the shear modulus corresponding to the rolling shear stress.

If the Poisson's ratios are not available, the values $\nu_{\mathrm{vx}} = \nu_{\mathrm{xv}} = 0$ can be used. Another possibility is to approximate the values according to HUBER's formulas ([1]).

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

For the softwood C24 mentioned above you get

$$\begin{aligned}
E_x &= 11{,}000\ \mathrm{MPa} \\
E_y &= 370\ \mathrm{MPa} \\
G_{xy} &= G_{xz} = 690\ \mathrm{MPa} \\
G_{yz} &= 69\ \mathrm{MPa} \\
\nu_{xy} &\approx \left(\frac{\sqrt{11{,}000 \cdot 370}}{2 \cdot 690} - 1\right)\sqrt{\frac{11{,}000}{370}} = 2.52 \\
\nu_{yx} &\approx \left(\frac{\sqrt{11{,}000 \cdot 370}}{2 \cdot 690} - 1\right)\sqrt{\frac{370}{11{,}000}} = 0.08
\end{aligned} \tag{2.6}$$

Example

Let us give an example that illustrates the relevance of the Poisson's ratios for orthotropic materials.

We consider the plane stress of a planar plate with the dimensions 1 m $\times$1 m. In the case of the plane stress condition for an orthotropic homogenous material, HOOKE's law takes the form

$$\begin{bmatrix} \varepsilon_\mathrm{x} \\ \varepsilon_\mathrm{y} \\ \gamma_\mathrm{xy} \end{bmatrix} = \begin{bmatrix} \frac{1}{\mathrm{E_x}} & -\frac{\nu_\mathrm{yx}}{\mathrm{E_y}} & 0 \\ -\frac{\nu_\mathrm{xy}}{\mathrm{E_x}} & \frac{1}{\mathrm{E_y}} & 0 \\ 0 & 0 & \mathrm{G_{xy}} \end{bmatrix} \begin{bmatrix} \sigma_\mathrm{x} \\ \sigma_\mathrm{y} \\ \tau_\mathrm{xy} \end{bmatrix} \tag{2.7}$$

Furthermore, we consider the stress conditions without the shear stress ($\tau_\mathrm{xy} = 0$). Equation 2.7 then implies that $\gamma_\mathrm{xy} = 0$. The matrix can be simplified to the form

$$\begin{bmatrix} \varepsilon_\mathrm{x} \\ \varepsilon_\mathrm{y} \end{bmatrix} = \begin{bmatrix} \frac{1}{\mathrm{E_x}} & -\frac{\nu_\mathrm{yx}}{\mathrm{E_y}} \\ -\frac{\nu_\mathrm{xy}}{\mathrm{E_x}} & \frac{1}{\mathrm{E_y}} \end{bmatrix} \begin{bmatrix} \sigma_\mathrm{x} \\ \sigma_\mathrm{y} \end{bmatrix} \tag{2.8}$$

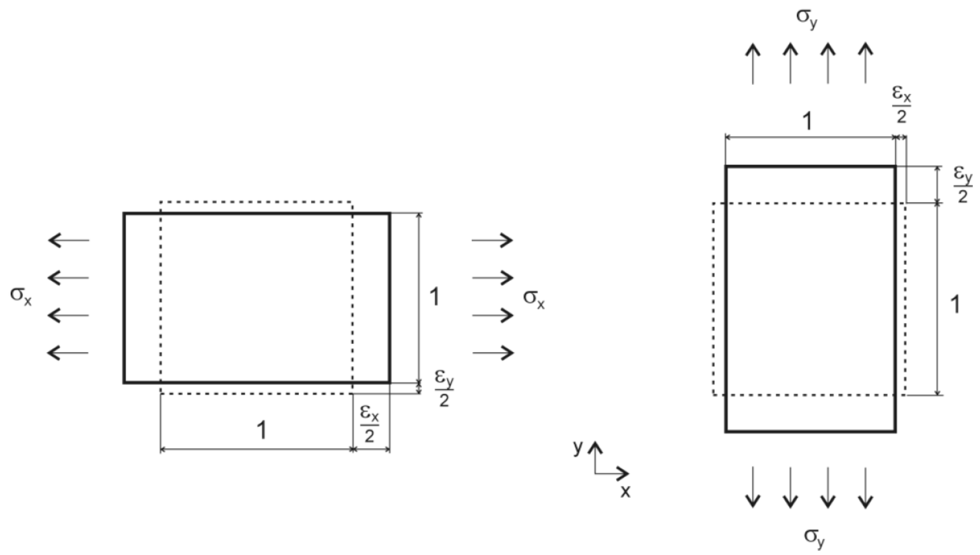


Figure 2.4: Plane stress of the plate in x-direction and y-direction

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

$$\varepsilon_x = \frac{\sigma_x}{E_x} \qquad (2.9)$$

$$\varepsilon_y = -\frac{\nu_{xy}}{E_x}\,\sigma_x$$

Hence, the relation for the Poisson's ratio ν_{xy}:

$$\nu_{xy} = -\frac{\varepsilon_y}{\varepsilon_x} \qquad (2.10)$$

We proceed accordingly for 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, we get

$$\varepsilon_x = -\frac{\nu_{yx}}{E_y}\,\sigma_y \qquad (2.11)$$

$$\varepsilon_y = \frac{\sigma_y}{E_y}$$

Hence, the relation for the Poisson's ratio ν_{yx}:

$$\nu_{yx} = -\frac{\varepsilon_x}{\varepsilon_y} \qquad (2.12)$$

Equation 2.10 and Equation 2.12 can be interpreted as follows: 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. It can be converted to the following schematic form:

$$\begin{bmatrix} \varepsilon_x \\ \varepsilon_y \end{bmatrix} = \begin{bmatrix} 1 & -\nu_{yx} \\ -\nu_{xy} & 1 \end{bmatrix} \begin{bmatrix} \frac{\sigma_x}{E_x} \\ \frac{\sigma_y}{E_y} \end{bmatrix} \qquad (2.13)$$

2.3.2 Isotropic

An isotropic material has identical mechanical properties in all directions. The material is defined by the modulus of elasticity E, the shear modulus G and the Poisson's ratio ν.

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 2.5: Isotropic material model

The modulus of elasticity and shear modulus must satisfy $E \geq 0$, $G \geq 0$. The global stiffness matrix $\boldsymbol{D}$ has to be positive-definite.

Examples of isotropic materials are glass or steel. For the modulus of elasticity E, the shear modulus G and the 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 (e.g. rubber).

2.3.3 User-Defined

The user-defined material model makes it possible to directly enter the stiffness matrix elements of individual layers. To calculate the shear elements of the global stiffness matrix, you need to fill in the shear moduli G_{xz} and G_{yz} as well.

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 2.6: User-defined material model

The stiffness matrix elements and shear moduli must satisfy: $d'_{11} \geq 0$, $d'_{22} \geq 0$, $d'_{33} \geq 0$, $G_{xz} \geq 0$ and $G_{yz} \geq 0$. The global stiffness matrix $\boldsymbol{D}$ has to be positive-definite.

2.3.4 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 2.7: Hybrid material model

The global stiffness matrix $\boldsymbol{D}$ has to be positive-definite.

An example of the hybrid material is a wood-concrete composite.

2.4 Stiffness Matrix

2.4.1 With Consideration of Shear Coupling

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

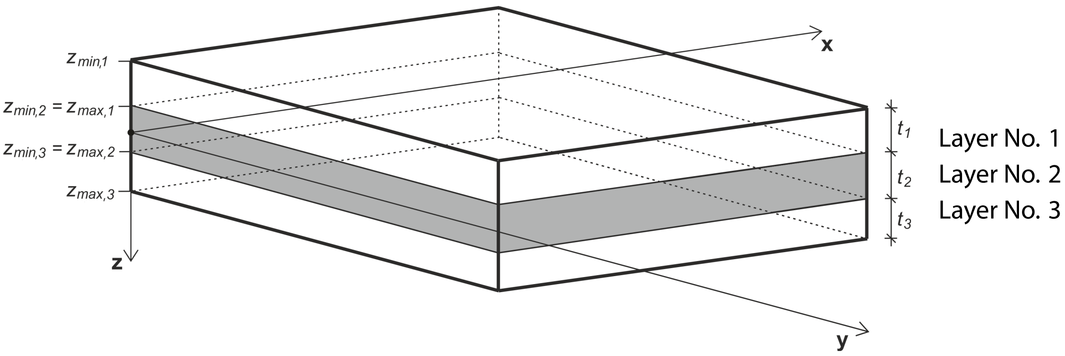


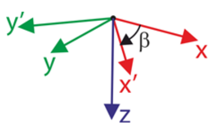
Figure 2.8: Layer scheme

The stiffness matrix for each layer $\boldsymbol{d}'_i$ (planar stiffness matrix) is calculated according to the following relation, using the moduli of elasticity, the shear modulus and Poisson's ratio of each layer.

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

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

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



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

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

where

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

The individual elements then are

$$d_{11,i} = c^4 d'_{11,i} + 2c^2 s^2 d'_{12,i} + s^4 d'_{22,i} + 4c^2 s^2 d'_{33,i}$$

$$d_{12,i} = c^2 s^2 d'_{11,i} + s^4 d'_{12,i} + c^4 d'_{12,i} + c^2 s^2 d'_{22,i} - 4c^2 s^2 d'_{33,i}$$

$$d_{13,i} = c^3 s d'_{11,i} + c s^3 d'_{12,i} - c^3 s d'_{12,i} - c s^3 d'_{22,i} - 2c^3 s d'_{33,i} + 2c s^3 d'_{33,i}$$

$$d_{22,i} = s^4 d'_{11,i} + 2c^2 s^2 d'_{12,i} + c^4 d'_{22,i} + 4c^2 s^2 d'_{33,i}$$

$$d_{23,i} = c s^3 d'_{11,i} + c^3 s d'_{12,i} - c s^3 d'_{12,i} - c^3 s d'_{22,i} + 2c^3 s d'_{33,i} - 2c s^3 d'_{33,i}$$

$$d_{33,i} = c^2 s^2 d'_{11,i} - 2c^2 s^2 d'_{12,i} + c^2 s^2 d'_{22,i} + \left(c^2 - s^2\right)^2 d'_{33,i}$$

The global stiffness matrix is

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

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

Bending and torsion

Shear

Membrane

Eccentricity

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

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

Stiffness matrix elements: Bending and torsion [Nm]

$$D_{11} = \sum_{i=1}^{n} \frac{z^3_{max,i} - z^3_{min,i}}{3} d_{11,i} \quad D_{12} = \sum_{i=1}^{n} \frac{z^3_{max,i} - z^3_{min,i}}{3} d_{12,i} \quad D_{13} = \sum_{i=1}^{n} \frac{z^3_{max,i} - z^3_{min,i}}{3} d_{13,i}$$

$$D_{22} = \sum_{i=1}^{n} \frac{z^3_{max,i} - z^3_{min,i}}{3} d_{22,i} \quad D_{23} = \sum_{i=1}^{n} \frac{z^3_{max,i} - z^3_{min,i}}{3} d_{23,i}$$

$$D_{33} = \sum_{i=1}^{n} \frac{z^3_{max,i} - z^3_{min,i}}{3} d_{33,i}$$

In case of a single layer plate of thickness *t*, the introduced relations lead to the familiar relation

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

Stiffness matrix elements: Eccentricity effects [Nm/m]

$$D_{16} = \sum_{i=1}^{n} \frac{z^2_{max,i} - z^2_{min,i}}{2} d_{11,i} \quad D_{17} = \sum_{i=1}^{n} \frac{z^2_{max,i} - z^2_{min,i}}{2} d_{12,i} \quad D_{18} = \sum_{i=1}^{n} \frac{z^2_{max,i} - z^2_{min,i}}{2} d_{13,i}$$

$$D_{27} = \sum_{i=1}^{n} \frac{z^2_{max,i} - z^2_{min,i}}{2} d_{22,i} \quad D_{28} = \sum_{i=1}^{n} \frac{z^2_{max,i} - z^2_{min,i}}{2} d_{23,i}$$

$$D_{38} = \sum_{i=1}^{n} \frac{z^2_{max,i} - z^2_{min,i}}{2} d_{33,i}$$

The eccentricity stiffness matrix elements are nonzero for unsymmetrical layer compositions, e.g. a two layered composition with identical orthotropic material for each layer where the second layer is rotated by 90 ° ($\beta_1 = 0$ °, $\beta_2 = 90$ °).

Layer No.	A Material Description	B Thickness t [mm]	C Orthotropic Direction β [°]	D Modulus of Elasticity [N/mm²] Ex	E Ey	F Shear Modulus [N/mm²] Gxz	G Gyz	H Gxy	I Poisson's Ratio [-] νxy	J νyx
1	C24	40.0	0.00	11000.0	370.0	690.0	69.0	690.0	0.000	0.000
2	C24	40.0	90.00	11000.0	370.0	690.0	69.0	690.0	0.000	0.000

Figure 2.9: Unsymmetrical layer composition

For symmetrical layer compositions, the eccentricity stiffness matrix is zero.

Layer No.	A Material Description	B Thickness t [mm]	C Orthotropic Direction β [°]	D Modulus of Elasticity [N/mm²] Ex	E Ey	F Shear Modulus [N/mm²] Gxz	G Gyz	H Gxy	I Poisson's Ratio [-] νxy	J νyx
1	C24	40.0	0.00	11000.0	370.0	690.0	69.0	690.0	0.000	0.000
2	C24	40.0	90.00	11000.0	370.0	690.0	69.0	690.0	0.000	0.000
3	C24	40.0	0.00	11000.0	370.0	690.0	69.0	690.0	0.000	0.000

Figure 2.10: Symmetrical layer composition

The bending and membrane stiffness matrix elements are coupled through the eccentricity stiffness matrix elements. Pure bending loading yields nonzero internal forces n_x, n_y, n_{xy}, and vice versa. Pure membrane loading yields nonzero internal moments m_x, m_y, m_{xy}.

Therefore, 2D models (plane XY, plane XZ, plane YZ) cannot be calculated in RF-LAMINATE as only membrane stiffness elements or only bending stiffness elements are used. The model type has to be set to **3D** in the *General Data* dialog box of RFEM.

Stiffness matrix elements: Membrane [N/m]

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

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

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

Stiffness matrix elements: Shear [N/m]

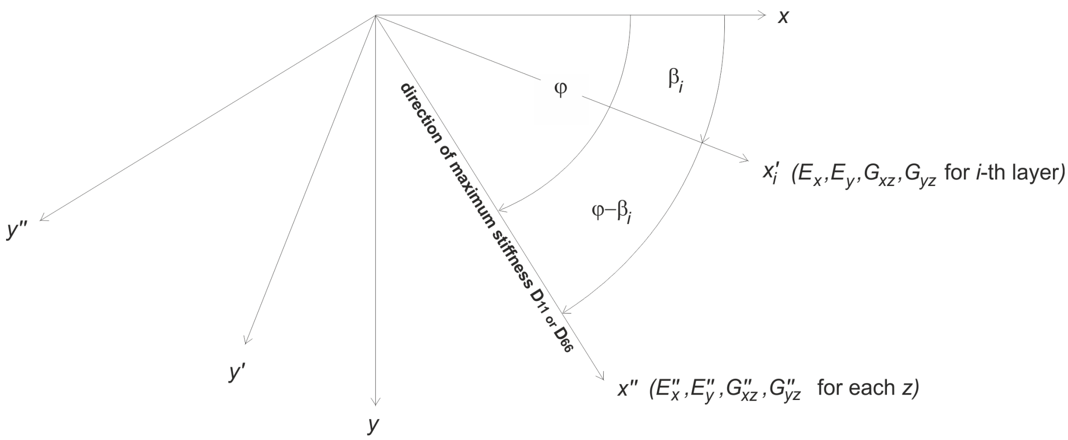


Figure 2.11: Calculation of shear matrix elements

The shear stiffness matrix elements are calculated according to the following algorithm.

1. Find the direction of maximum stiffness and the corresponding coordinate system x'', y''. The angle between the axes x and x'' is denoted by φ.
2. Transform the transversal shear stiffnesses $G_{xz,i}$, $G_{yz,i}$ for each layer from the coordinate system x', y' to the coordinate system x'', y'' in order to obtain $G''_{xz,i}$, $G''_{yz,i}$.

$$G''_{xz,i} = \cos^2(\varphi - \beta_i)\, G_{xz,i} + \sin^2(\varphi - \beta_i)\, G_{yz,i} \tag{2.22}$$

$$G''_{yz,i} = \sin^2(\varphi - \beta_i)\, G_{xz,i} + \cos^2(\varphi - \beta_i)\, G_{yz,i} \quad i = 1,\ldots,n$$

3. Transform the planar stiffness matrix $\boldsymbol{d}'_i$ for each layer from the coordinate system x', y' to the coordinate system x'', y'' in order to obtain the planar stiffness matrix $\boldsymbol{d}''_i$.

$$\boldsymbol{d}''_i = \boldsymbol{T}^{-T}_{3\times3,i} \boldsymbol{d}'_i \boldsymbol{T}^{-1}_{3\times3,i} \tag{2.23}$$

where

$$\boldsymbol{T}_{3\times3,i} = \begin{bmatrix} c^2 & s^2 & cs \\ s^2 & c^2 & -cs \\ -2cs & 2cs & c^2 - s^2 \end{bmatrix}, \text{where } c = \cos(\varphi - \beta_i), s = \sin(\varphi - \beta_i), \ i = 1,\ldots,n \tag{2.24}$$

From the stiffness matrix $\boldsymbol{d}''_i$, Young's moduli $E''_{x,i}$, $E''_{y,i}$ are extracted.

$$E''_{x,i} = d''_{11,i} + \frac{2d''_{12,i}d''_{13,i}d''_{23,i} - d''_{22,i}\left(d''_{13,i}\right)^2 - d''_{33,i}\left(d''_{12,i}\right)^2}{d''_{22,i}d''_{33,i} - \left(d''_{23,i}\right)^2} \tag{2.25}$$

$$E''_{y,i} = d''_{22,i} + \frac{2d''_{12,i}d''_{13,i}d''_{23,i} - d''_{11,i}\left(d''_{23,i}\right)^2 - d''_{33,i}\left(d''_{12,i}\right)^2}{d''_{11,i}d''_{33,i} - \left(d''_{13,i}\right)^2} \tag{2.26}$$

4. In the coordinate system x'', y'', calculate $D''_{44,calc}$, $D''_{55,calc}$ according to the GRASHOFF integral formula and consider $D''_{45} = 0$.

$$D''_{44,calc} = \frac{1}{\int\limits_{-t/2}^{t/2} \frac{1}{G''_{xz}(z)} \left(\frac{\int\limits_{z}^{t/2} E''_x(\bar{z})\left(\bar{z} - z_{0,x}\right) d\bar{z}}{\int\limits_{-t/2}^{t/2} E''_x(\bar{z})\left(\bar{z} - z_{0,x}\right)^2 d\bar{z}} \right)^2 dz}, \; z_{0,x} = \frac{\int\limits_{-t/2}^{t/2} E''_x(\bar{z})\,\bar{z}d\bar{z}}{\int\limits_{-t/2}^{t/2} E''_x(\bar{z})\, d\bar{z}} \tag{2.27}$$

$$D''_{55,calc} = \frac{1}{\int\limits_{-t/2}^{t/2} \frac{1}{G''_{yz}(z)} \left(\frac{\int\limits_{z}^{t/2} E''_y(\bar{z})\left(\bar{z} - z_{0,y}\right) d\bar{z}}{\int\limits_{-t/2}^{t/2} E''_y(\bar{z})\left(\bar{z} - z_{0,y}\right)^2 d\bar{z}} \right)^2 dz}, \; z_{0,y} = \frac{\int\limits_{-t/2}^{t/2} E''_y(\bar{z})\,\bar{z}d\bar{z}}{\int\limits_{-t/2}^{t/2} E''_y(\bar{z})\, d\bar{z}} \tag{2.28}$$

The values of stiffnesses D''_{44}, D''_{55} are given by the following equations:

$$D''_{44} = \max\left(D''_{44,calc}, \frac{48}{5\ell^2} \frac{1}{\frac{1}{\sum_{i=1}^{n} E''_{x,i}\frac{t_i^3}{12}} - \frac{1}{\sum_{i=1}^{n} E''_{x,i}\frac{z_{max,i}^3 - z_{min,i}^3}{3}}} \right) \tag{2.29}$$

$$D''_{55} = \max\left(D''_{55,calc}, \frac{48}{5\ell^2} \frac{1}{\frac{1}{\sum_{i=1}^{n} E''_{y,i}\frac{t_i^3}{12}} - \frac{1}{\sum_{i=1}^{n} E''_{y,i}\frac{z_{max,i}^3 - z_{min,i}^3}{3}}} \right) \tag{2.30}$$

where ℓ is the mean length of the lines surrounding the surface as a “box”.

5. Transform the values D''_{44}, D''_{55} from coordinate system x'', y'' back to coordinate system x, y (local coordinate system of surface) in order to obtain the stiffnesses D_{44}, D_{55}, D_{45}.

$$\begin{aligned} D_{44} &= \cos^2(\varphi)\, D''_{44} + \sin^2(\varphi)\, D''_{55} \\ D_{55} &= \sin^2(\varphi)\, D''_{44} + \cos^2(\varphi)\, D''_{55} \\ D_{45} &= \sin(\varphi)\cos(\varphi)\left(D''_{44} - D''_{55}\right) \end{aligned} \tag{2.31}$$

2.4.2 Without Consideration of Shear Coupling

We will now examine a plate consisting of n isotropic material layers. The individual layers are not shear-coupled. Each layer has the thickness t_i and the minimum and maximum z-coordinates $z_{min,i}$, $z_{max,i}$.

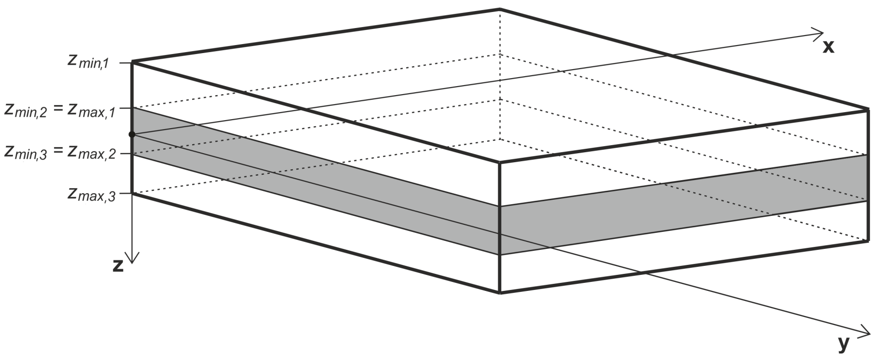


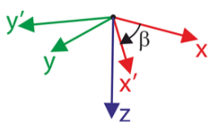
Figure 2.12: Layer scheme

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

$$\boldsymbol{d}_i' = \begin{bmatrix} d'_{11,i} & d'_{12,i} & 0 \\ & d'_{22,i} & 0 \\ \text{sym.} & & d'_{33,i} \end{bmatrix} = \begin{bmatrix} \dfrac{E_{x,i}}{1-\nu_{xy,i}^2 \dfrac{E_{y,i}}{E_{x,i}}} & \dfrac{\nu_{xy,i}\, E_{y,i}}{1-\nu_{xy,i}^2 \dfrac{E_{y,i}}{E_{x,i}}} & 0 \\ & \dfrac{E_y}{1-\nu_{xy,i}^2 \dfrac{E_{y,i}}{E_{x,i}}} & 0 \\ \text{sym.} & & G_{xy,i} \end{bmatrix} \qquad i = 1,\ldots,n \tag{2.32}$$

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

$$\boldsymbol{d}_i' = \begin{bmatrix} d'_{11,i} & d'_{12,i} & 0 \\ & d'_{22,i} & 0 \\ \text{sym.} & & d'_{33,i} \end{bmatrix} = \begin{bmatrix} \dfrac{E_i}{1-\nu_i^2} & \dfrac{\nu_i\, E_i}{1-\nu_i^2} & 0 \\ & \dfrac{E_i}{1-\nu_i^2} & 0 \\ \text{sym.} & & G_i \end{bmatrix}, \; G_i = \frac{E_i}{2\,(1+\nu_i)}, \quad i = 1,\ldots,n \tag{2.33}$$



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

$$\boldsymbol{d}_i = \begin{bmatrix} d_{11,i} & d_{12,i} & d_{13,i} \\ & d_{22,i} & d_{23,i} \\ \text{sym.} & & d_{33,i} \end{bmatrix} = \boldsymbol{T}^T_{3\times 3,i}\, \boldsymbol{d}_i'\, \boldsymbol{T}_{3\times 3,i} \tag{2.34}$$

where

$$\boldsymbol{T}_{3\times 3,i} = \begin{bmatrix} c^2 & s^2 & cs \\ s^2 & c^2 & -cs \\ -2cs & 2cs & c^2-s^2 \end{bmatrix} \quad \text{where } \; c = \cos(\beta_i)\,, s = \sin(\beta_i) \tag{2.35}$$

The individual elements then are

$$d_{11,i} = c^4 d'_{11,i} + 2c^2 s^2 d'_{12,i} + s^4 d'_{22,i} + 4c^2 s^2 d'_{33,i}$$

$$d_{12,i} = c^2 s^2 d'_{11,i} + s^4 d'_{12,i} + c^4 d'_{12,i} + c^2 s^2 d'_{22,i} - 4c^2 s^2 d'_{33,i}$$

$$d_{13,i} = c^3 s d'_{11,i} + c s^3 d'_{12,i} - c^3 s d'_{12,i} - c s^3 d'_{22,i} - 2c^3 s d'_{33,i} + 2c s^3 d'_{33,i}$$

$$d_{22,i} = s^4 d'_{11,i} + 2c^2 s^2 d'_{12,i} + c^4 d'_{22,i} + 4c^2 s^2 d'_{33,i}$$

$$d_{23,i} = c s^3 d'_{11,i} + c^3 s d'_{12,i} - c s^3 d'_{12,i} - c^3 s d'_{22,i} + 2c^3 s d'_{33,i} - 2c s^3 d'_{33,i}$$

$$d_{33,i} = c^2 s^2 d'_{11,i} - 2c^2 s^2 d'_{12,i} + c^2 s^2 d'_{22,i} + \left(c^2 - s^2\right)^2 d'_{33,i}$$

The global stiffness matrix is

$$\boldsymbol{D} = \begin{bmatrix} D_{11} & D_{12} & D_{13} & 0 & 0 & 0 & 0 & 0 \\ & D_{22} & D_{23} & 0 & 0 & 0 & 0 & 0 \\ & & D_{33} & 0 & 0 & 0 & 0 & 0 \\ & & & D_{44} & D_{45} & 0 & 0 & 0 \\ & & & & D_{55} & 0 & 0 & 0 \\ & & \text{sym.} & & & D_{66} & D_{67} & D_{68} \\ & & & & & & D_{77} & D_{78} \\ & & & & & & & D_{88} \end{bmatrix} \tag{2.36}$$

$$\begin{bmatrix} m_x \\ m_y \\ m_{xy} \\ v_x \\ v_y \\ n_x \\ n_y \\ n_{xy} \end{bmatrix} = \begin{bmatrix} D_{11} & D_{12} & D_{13} & 0 & 0 & 0 & 0 & 0 \\ & D_{22} & D_{23} & 0 & 0 & 0 & 0 & 0 \\ & & D_{33} & 0 & 0 & 0 & 0 & 0 \\ & & & D_{44} & D_{45} & 0 & 0 & 0 \\ & & & & D_{55} & 0 & 0 & 0 \\ & & \text{sym.} & & & D_{66} & D_{67} & D_{68} \\ & & & & & & D_{77} & D_{78} \\ & & & & & & & D_{88} \end{bmatrix} \begin{bmatrix} \kappa_x \\ \kappa_y \\ \kappa_{xy} \\ \gamma_{xz} \\ \gamma_{yz} \\ \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{bmatrix} \tag{2.37}$$

Bending and torsion

Shear

Membrane

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

$$\boldsymbol{D} = \begin{bmatrix} D_{11} & D_{12} & 0 & 0 & 0 & 0 & 0 & 0 \\ & D_{22} & 0 & 0 & 0 & 0 & 0 & 0 \\ & & D_{33} & 0 & 0 & 0 & 0 & 0 \\ & & & D_{44} & 0 & 0 & 0 & 0 \\ & & & & D_{55} & 0 & 0 & 0 \\ & & \text{sym.} & & & D_{66} & D_{67} & 0 \\ & & & & & & D_{77} & 0 \\ & & & & & & & D_{88} \end{bmatrix} \tag{2.38}$$

Stiffness matrix elements: Bending and Torsion [Nm]

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

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

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

Stiffness matrix elements: Membrane [N/m]

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

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

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

Stiffness matrix elements: Shear [N/m]

The shear stiffness matrix elements are calculated according to the following algorithm:

1. Find the direction of maximum stiffness and the corresponding coordinate system x'', y''. The angle between the axes x and x'' is denoted by φ.

2. Transform the transversal shear stiffnesses G_{xz}, G_{yz} for each layer from the coordinate system x', y' to the coordinate system x'', y'' in order to obtain $G''_{xz,i}, G''_{yz,i}$.

$$G''_{xz,i} = \cos^2(\varphi - \beta_i)\, G_{xz,i} + \sin^2(\varphi - \beta_i)\, G_{yz,i} \tag{2.39}$$

$$G''_{yz,i} = \sin^2(\varphi - \beta_i)\, G_{xz,i} + \cos^2(\varphi - \beta_i)\, G_{yz,i} \qquad i = 1,\ldots,n$$

3. In the coordinate system x'', y'', calculate D''_{44}, D''_{55} and consider $D''_{45} = 0$.

$$D''_{44} = \frac{5}{6} \sum_{i=1}^{n} G''_{xz,i} t_i \tag{2.40}$$

$$D''_{55} = \frac{5}{6} \sum_{i=1}^{n} G''_{yz,i} t_i \tag{2.41}$$

4. Transform the values D''_{44}, D''_{55} from coordinate system x'', y'' back to coordinate system x, y (local coordinate system of surface) in order to obtain the stiffnesses D_{44}, D_{55}, D_{45}.

$$\begin{aligned} D_{44} &= \cos^2(\varphi)\, D''_{44} + \sin^2(\varphi)\, D''_{55} \\ D_{55} &= \sin^2(\varphi)\, D''_{44} + \cos^2(\varphi)\, D''_{55} \\ D_{45} &= \sin(\varphi)\cos(\varphi)\left(D''_{44} - D''_{55}\right) \end{aligned} \tag{2.42}$$

3 Input Data

When you start RF-LAMINATE, a new window appears. In this window, a navigator is displayed on the left. It manages the windows and tables of all input data.

To select a window, click the corresponding entry in the navigator. To set the previous or next input window, use the buttons shown on the left. You can also use the function keys to select the next [F2] or previous [F3] window.

Details...

When you click the [Details] button, a dialog box appears where you can specify the stresses and result windows to be displayed (see Chapter 4.1, page 36).

Standard

The [Standard] button opens a dialog box which controls the safety and modification factors of the selected standard (see Chapter 4.2, page 45).

OK Cancel

[OK] saves the entered data (and results, if calculated). Thus, you exit RF-LAMINATE and return to the main program RFEM. To quit the module without saving any changes, click [Cancel].

3.1 General Data

In the *1.1 General Data* Window, you can select the surfaces and actions you want to design. The two tabs manage the load cases, load and result combinations for the ULS and SLS analyses.

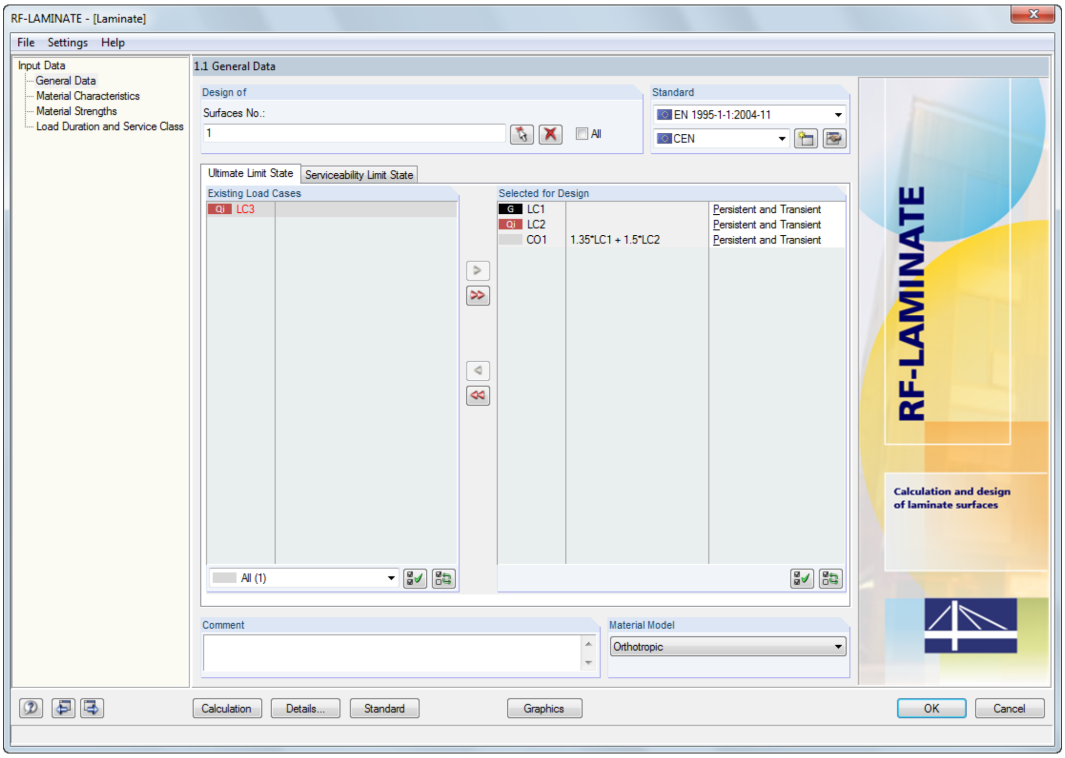


Figure 3.1: Window 1.1 *General Data*

Design of

If you want to design only specific *Surfaces*, clear the *All* check box. Then you can acces the text box and enter the numbers of the relevant surfaces. You can remove the list of numbers with the the [Delete] button. Use the [Select] button to surfaces the objects graphically in the RFEM work window.

Standard

In the drop-down list in the upper right corner of the window, you can select the standard whose parameters are relevant for the design and whose limit values of the deflection are to be applied.

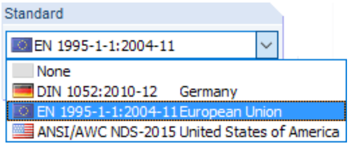


Figure 3.2: List of standards

For EN 1995-1-1 [2], the National annex can be selected from the list below.

Figure 3.3: List of National annexes

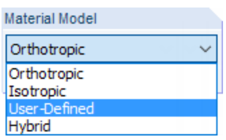
Use the [Edit] button to open a dialog box where you can check and, if necessary, adjust the parameters of the selected standard or National annex. This dialog box is described in Chapter 4.2 on page 45. You can also click the [Standard] button open the *Standard* dialog box. This button is available in all windows.

To create a user-defined standard or National annex, click the [New] button.

Comment

In this text box at the bottom of the window, you can enter additionnal notes or explanations.

Material Model



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

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

The material models are described in Chapter 2.2 on page 7.

3.1.1 Ultimate Limit State

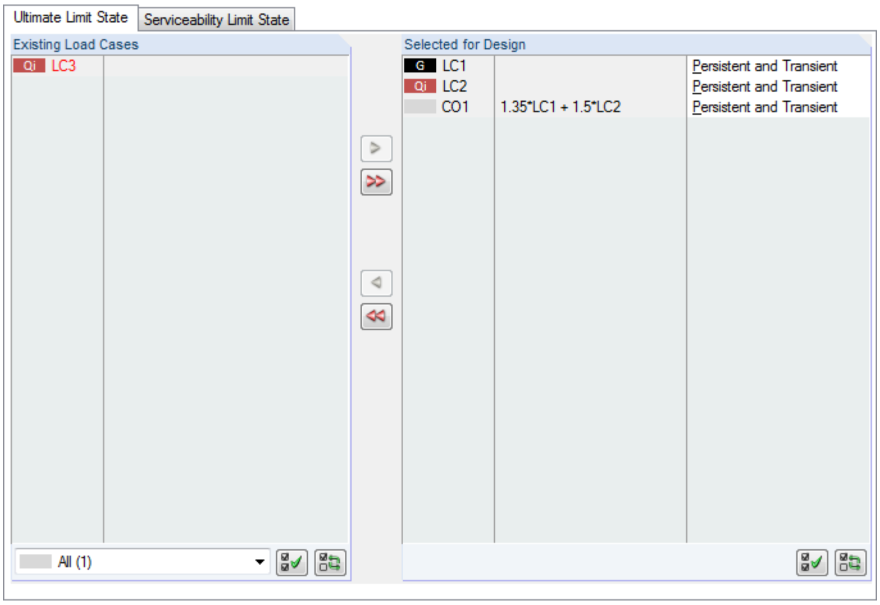


Figure 3.4: Window *1.1 General Data*, tab *Ultimate Limit State*

Existing Load Cases

This column lists all load cases, load and result combinations that have been created in RFEM.

Use the button to transfer selected entries to the *Selected for Design* table on the right. Alternatively, you can double-click the entries. To transfer the entire list to the right, use the button.

To add multiple entries of load cases, click the entries while pressing the [Ctrl] key, as common for Windows applications. Thus, you can transfer several load cases at the same time.

Load cases marked in red cannot be designed (see Figure 3.4): This happens when the load cases are defined without any load data or contain only imperfections.

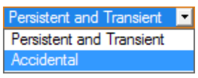
At the end of the list, several filter options are available. They will help you to assign the entries sorted by load case, load combination, or action category. The buttons have the following functions:

	Select all load cases in the list
	Invert the selection of load cases

Table 3.1: Buttons in the tab *Ultimate Limit State*

Selected for Design

The column on the right lists the load cases as well as the load and result combinations selected for design. Use the button or double-click the entries to remove selected entries from the list. The button transfers the entire list to the left.



You can assign the load cases, load and result combinations to the following design situations:

- *Persistent and transient*
- *Accidental*

This classification manages the partial factor γ_M of the material properties. You can check and adjust this factor in the *Standard* dialog box (see Chapter 4.2.1, page 46).

3.1.2 Serviceability Limit State

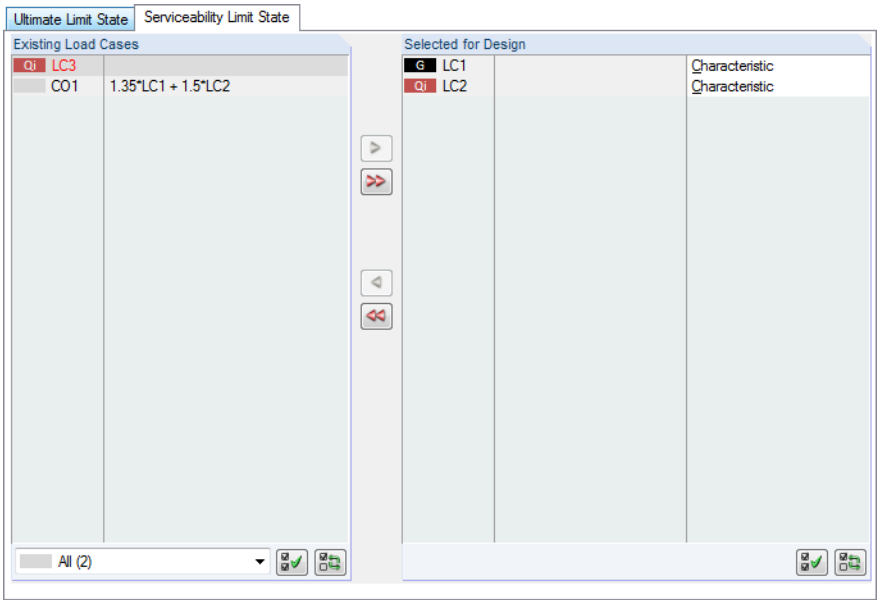


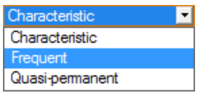
Figure 3.5: Window *1.1 General Data*, tab *Serviceability Limit State*

Existing Load Cases

This section lists all load cases, load and result combinations that have been created in RFEM.

Selected for Design

You can add or remove load cases, load combinations, and result combinations as described in Chapter 3.1.1. When a load case has been transferred, the item *Serviceability Data* is added in the navigator.



You can assign the load cases, load and result combinations to the following design situations:

- *Characteristic*
- *Frequent*
- *Quasi-permanent*

This classification controls the limit values that are to be applied for the deflection analysis. You can modify the limit values in the *Standard* dialog box (see Chapter 4.2.2, page 47).

3.2 Material Characteristics

In this window, the layers with the respective materials can be defined for the surfaces.

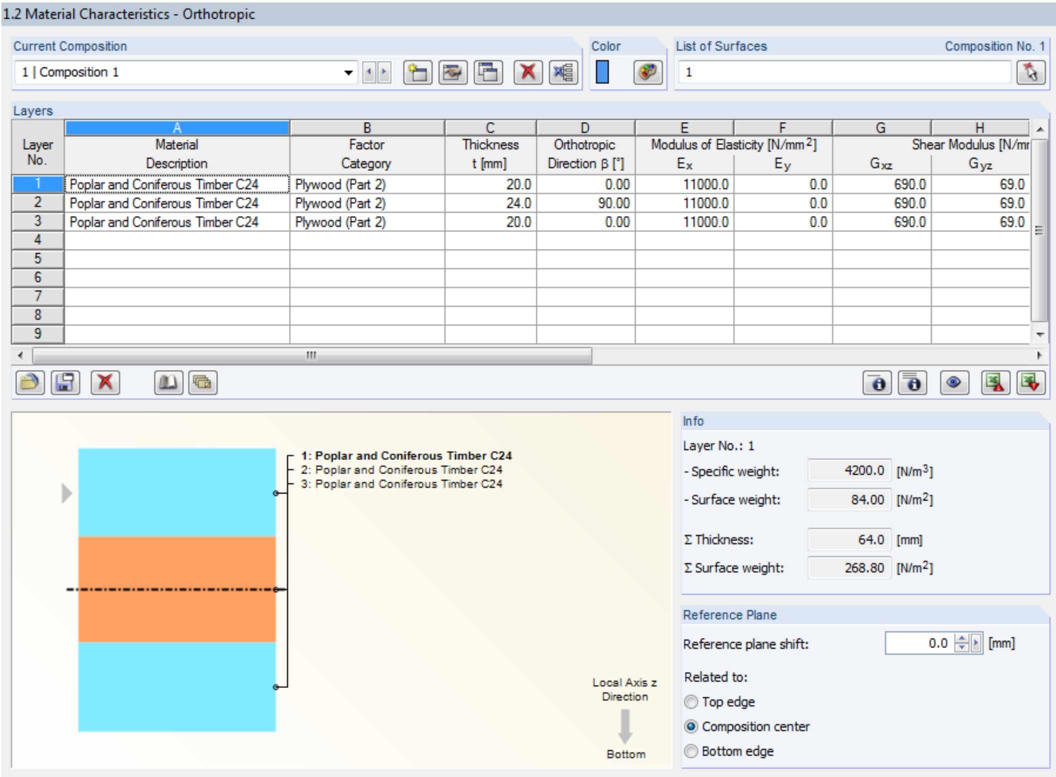


Figure 3.6: Window *1.2 Material Characteristics - Orthotropic*

Current Composition

In this window section, the active composition is displayed. The layers of the composition are listed in the table below. For each composition, the layers can be defined individually. You can create more compositions with various layers here.

The buttons have the following functions:

Button	Function
	Create new composition of layers
	Show details of current composition (see Figure 3.15, page 28)
	Copy current composition
	Delete current composition
	Delete all compositions

Table 3.2: Buttons for *Current Composition*

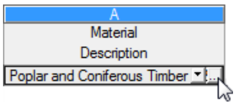
Color

Specific colors can be allocated to the compositions. Use the button to change the color of the current composition.

List of Surfaces

For each composition, the relevant surfaces can be defined in this window section. The button enables you to graphically select the surfaces in the work window of RFEM.

Layers



In this table, the individual layers of the current composition are to be defined. The material can be selected from the [Library] which contains a great number of materials with all required parameters. You can open the material library by clicking the button shown on the left. Alternatively, you place the pointer in the corresponding line of column A and click the button.

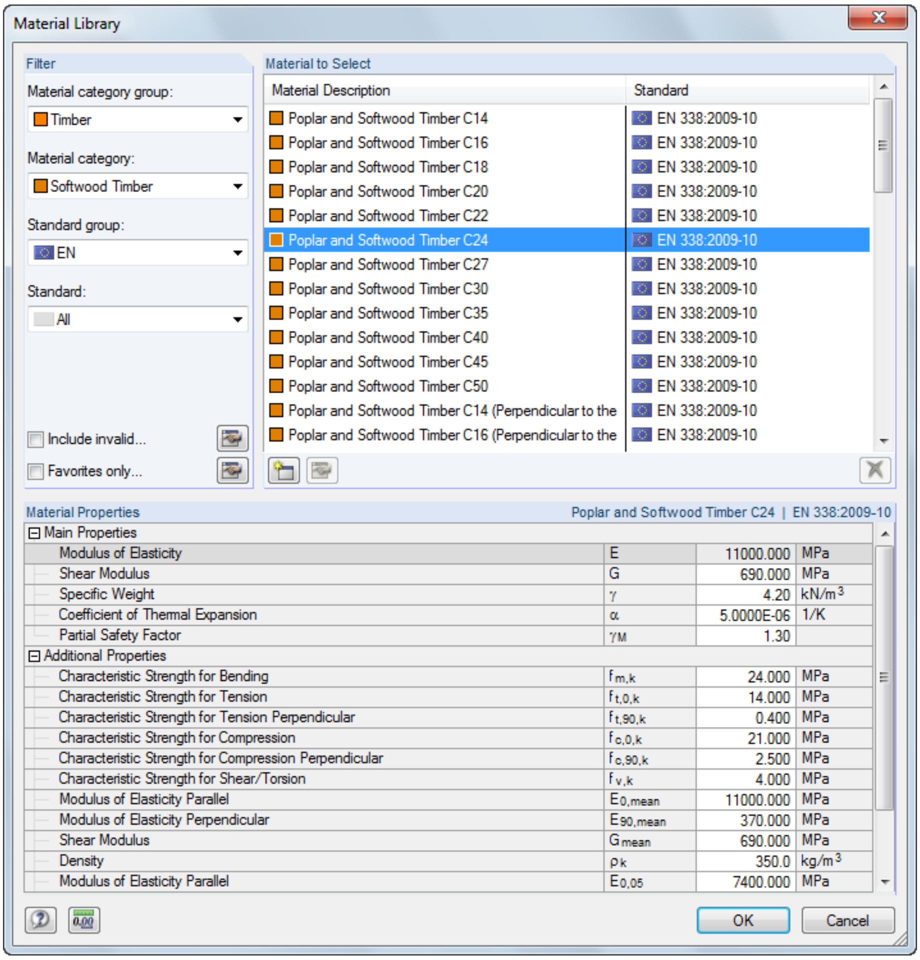


Figure 3.7: Material library

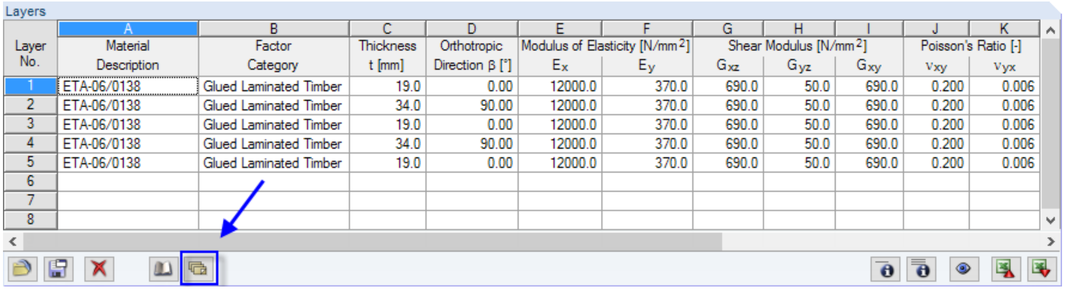
As the library is very extensive, various options for selection are available in the *Filter* section. You can filter the the list of materials by the criteria *Material category group*, *Material category*, *Standard group*, and *Standard*. In the list *Material to Select*, you can select the relevant material and check its parameters in the lower part of the dialog box.

Chapter 4.3 of the RFEM manual describes how materials can be filtered, added, or rearranged in the library.

When you click [OK], press the [↵] key or double-click a material, the material is imported to Window *1.2 Material Characteristics*. Then you can adjust all material parameters directly in the module.

Layer compositions from producers

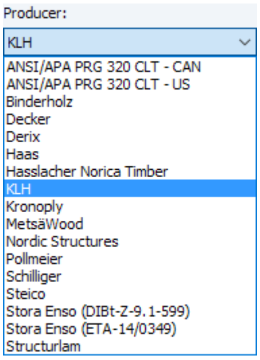
Furthermore, a library of layers can used to enter the entire composition at once. The database can be accessed by the [Import Layers from Library] button.



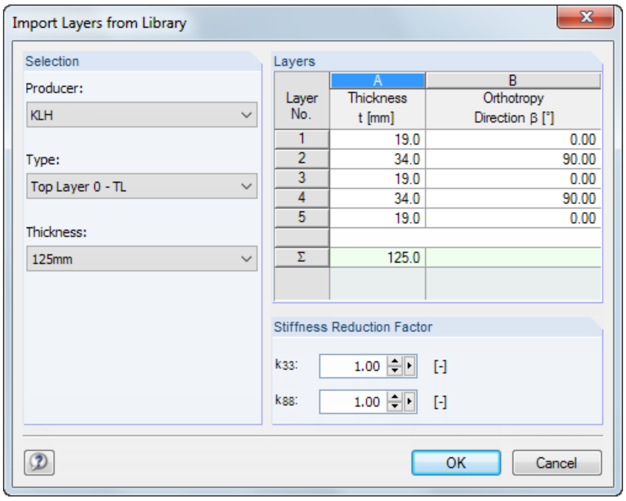
Layers

Layer No.	A Material Description	B Factor Category	C Thickness t [mm]	D Orthotropic Direction β [°]	E Modulus of Elasticity [N/mm²] E_x	F E_y	G Shear Modulus [N/mm²] G_{xz}	H G_{yz}	I G_{xy}	J Poisson's Ratio [-] ν_{xy}	K ν_{yx}
1	ETA-06/0138	Glued Laminated Timber	19.0	0.00	12000.0	370.0	690.0	50.0	690.0	0.200	0.006
2	ETA-06/0138	Glued Laminated Timber	34.0	90.00	12000.0	370.0	690.0	50.0	690.0	0.200	0.006
3	ETA-06/0138	Glued Laminated Timber	19.0	0.00	12000.0	370.0	690.0	50.0	690.0	0.200	0.006
4	ETA-06/0138	Glued Laminated Timber	34.0	90.00	12000.0	370.0	690.0	50.0	690.0	0.200	0.006
5	ETA-06/0138	Glued Laminated Timber	19.0	0.00	12000.0	370.0	690.0	50.0	690.0	0.200	0.006
6											
7											
8											

Figure 3.8: Button [Import Layers from Library]



In the library of layers, you can select the *Producer*, *Type* and *Thickness*.

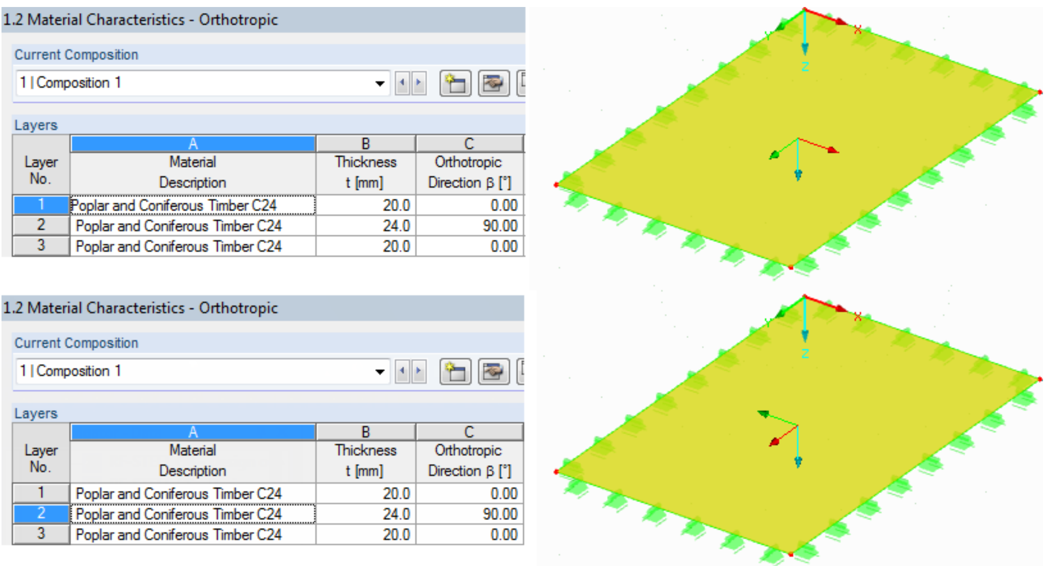


Layer No.	A Thickness t [mm]	B Orthotropy Direction β [°]
1	19.0	0.00
2	34.0	90.00
3	19.0	0.00
4	34.0	90.00
5	19.0	0.00
Σ	125.0	

Figure 3.9: Dialog box *Import Layers from Library*

The parameters of the imported layer composition can be modified in the *Layers* table, if necessary.

When you have chosen the orthotropic material model in Window *1.1 General Data*, the currently entered orthotropic direction β is displayed in the RFEM model in the background (see Figure 3.10). Thus, you can check your settings visually.



Layer No.	A Material Description	B Thickness t [mm]	C Orthotropic Direction β [°]
1	Poplar and Coniferous Timber C24	20.0	0.00
2	Poplar and Coniferous Timber C24	24.0	90.00
3	Poplar and Coniferous Timber C24	20.0	0.00

Figure 3.10: Display of the orthotropic direction β

Below the *Layers* table, several buttons are available. They have the following functions:

Button	Name	Function
	Load Layers	Load the composition that was saved before.
	Save Layers	Save the current composition as template for different models. The composition can be reloaded to any other composition via the button.
	Delete All Layers	Delete all data of the current composition.
	Material Library	Open the *Material Library* dialog box.
	Layer Library	Open the *Import Layers from Library* dialog box.
	Layer Matrix	Display the stiffness matrix elements of the current layer. → Chapter 2.4, page 12
	Composition Matrix	Display the stiffness matrix elements of the entire composition. → Chapter 2.4, page 12
	View Mode	Jump to the RFEM work window for graphical checks, without closing RF-LAMINATE.
	Excel Export	Export the current table to MS Excel or OpenOffice Calc. → Chapter 7.2, page 61
	Excel Import	Import the contents of a MS Excel or OpenOffice Calc sheet to the current table.

Table 3.3: Buttons for *Layers*

Info

Figure 3.11: Section *Info*

The *Info* section below the table provides information about the specific weight and surface weight of the current layer, and about the total thickness and total surface weight of the current composition.

Reference Plane

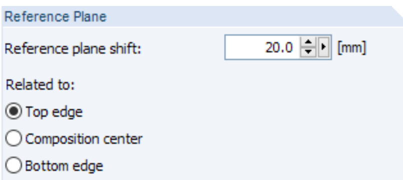


Figure 3.12: Section *Reference Plane*

If the surface is supported by eccentric bearings, the *shift* of the reference plane can be considered. Eccentricities are always relevant for asymmetric compositions. By the shift, the displaced center of gravity and the supports above or below the layers are accounted for.

The eccentricity elements of the stiffness matrix (see Equation 2.20, page 13) are calculated with respect to the defined shift. The shift of the reference plane basically means the place where supports are located. A dynamic graphic shows the reference plane so that you can check the input.

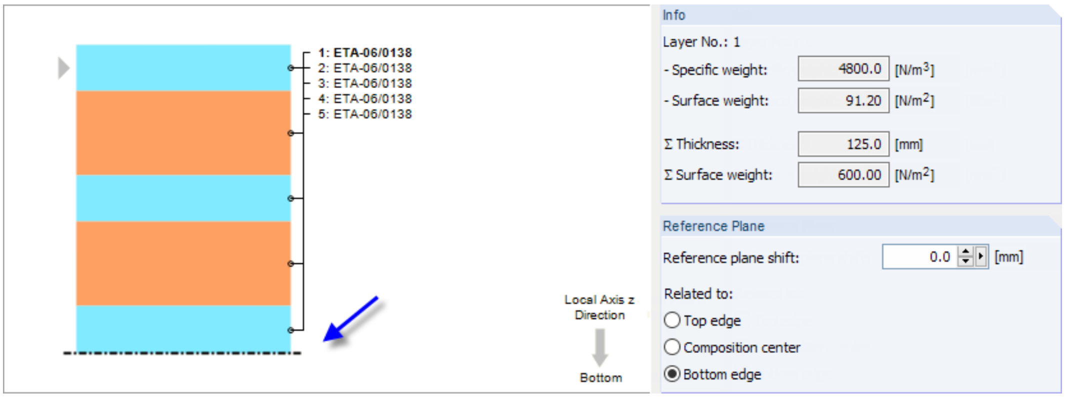


Figure 3.13: Shifted reference plane to *Bottom edge*

You can check the modified elements of the stiffness matrix by clicking the [Composition Matrix] button. In the *Extended Stiffness Matrix* dialog box, the eccentricity matrix elements are displayed.

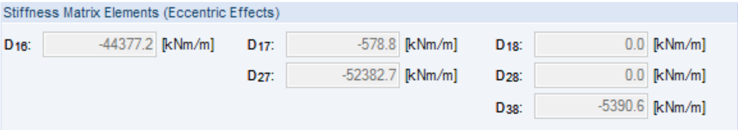


Figure 3.14: Info on *Stiffness Matrix Elements (Eccentricity Effects)*

Details of Composition

For each composition, the *Details of Composition* dialog box is available. You can open it by clicking the [Edit] button which is located to the right of the *Current Composition* list.

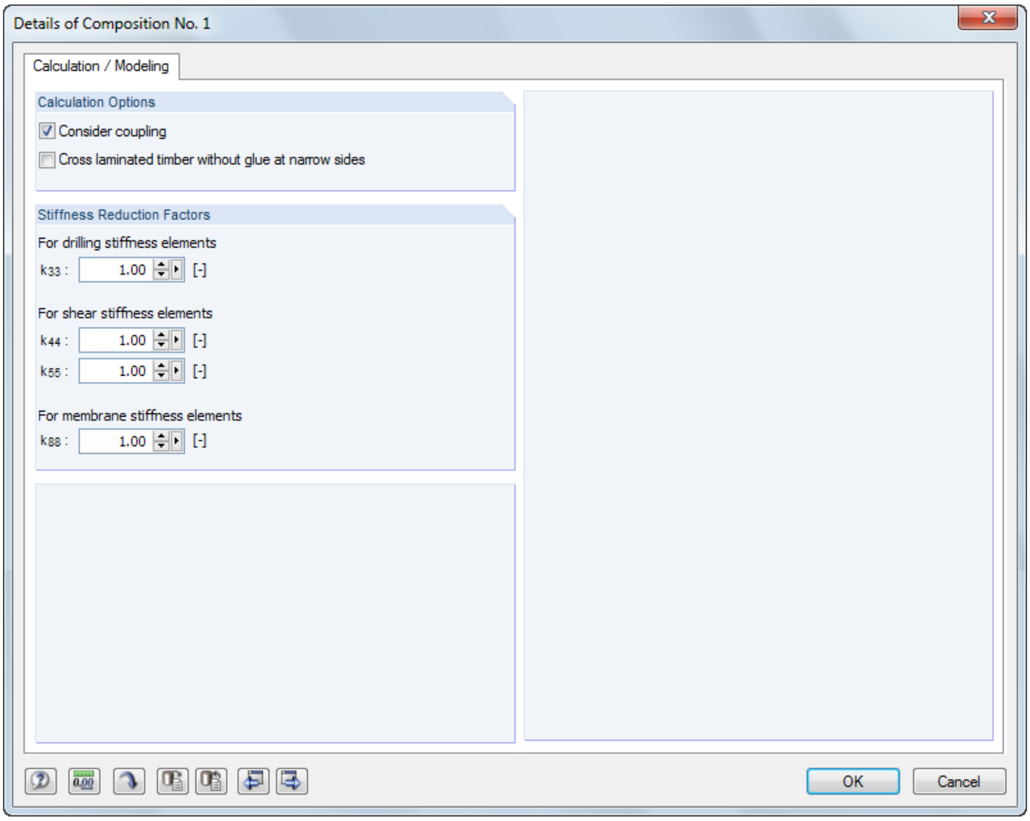


Figure 3.15: Dialog box *Details of Composition*

Calculation Options

In the upper dialog section, the check box *Consider coupling* is selected by default, which means that the shear coupling of layers is considered.

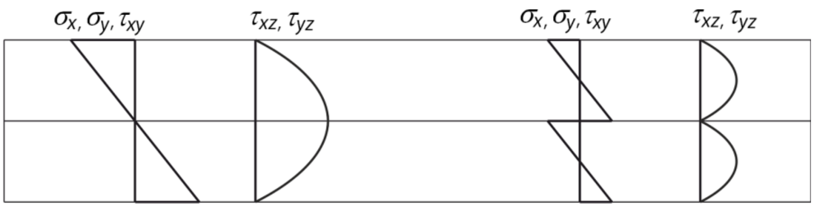


Figure 3.16: Basic bending stresses of two-layer plate – with shear coupling of layers (left) and without (right)

The approaches concerning shear coupling are described in Chapter 2.4.1 and Chapter 2.4.2.

The check box *Cross laminated timber without glue at narrow sides* can be applied to multi-layer plates made of cross laminated timber. For orthotropic material models, it is considered that $E_y = 0$ and the stiffness matrix element D_{88} is defined as follows:

$$D_{88} = \frac{1}{4} \sum_{i=1}^{n} t_i \, d_{33,i} \tag{3.1}$$

The reduction factor $\frac{1}{4}$ is recommended e.g. in DIN EN 1995-1-1, expression (NA.28).

For isotropic and user-defined material models, the stiffness matrix element D_{88} is defined as described in Equation 3.1.

Stiffness Reduction Factors

In this dialog section, you can reduce the drilling stiffness matrix element D_{33} by the factor k_{33}. The correction is possible only for plates having symmetric compositions and rotation angles that are multiples of 90 °. A correction is recommended in the standards ČSN 73 1702:2007, D.2.2(5) and DIN 1052:2008, D.2.2(5).

It is also possible to reduce the shear stiffness matrix elements D_{44} and D_{55} by the factors k_{44} and k_{55}. Those factors can only be applied for plates whose rotation angles are multiples of 90 °.

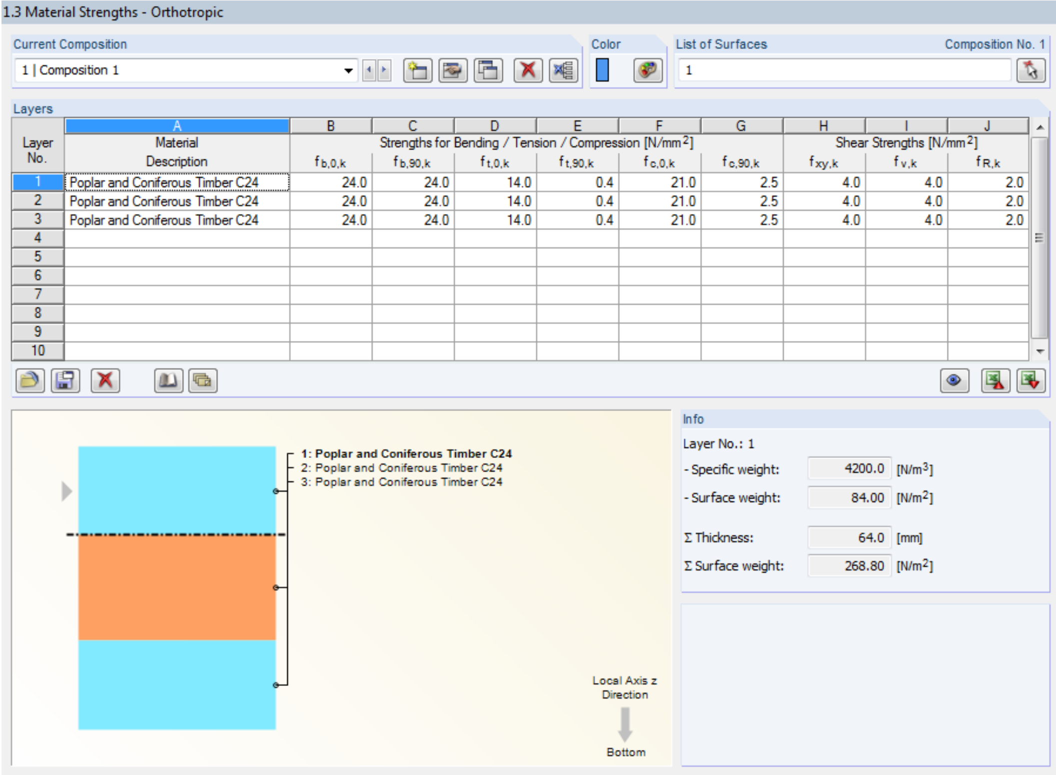
Finally, the membrane stiffness elements can be reduced by the factor k_{88}.

For symmetric compositions, the stiffness matrix is then equal to

$$\begin{bmatrix} m_x \\ m_y \\ m_{xy} \\ v_x \\ v_y \\ n_x \\ n_y \\ n_{xy} \end{bmatrix} = \begin{bmatrix} D_{11} & D_{12} & 0 & 0 & 0 & 0 & 0 & 0 \\ & D_{22} & 0 & 0 & 0 & 0 & 0 & 0 \\ & & k_{33}D_{33} & 0 & 0 & 0 & 0 & 0 \\ & & & k_{44}D_{44} & 0 & 0 & 0 & 0 \\ & & & & k_{55}D_{55} & 0 & 0 & 0 \\ & & \text{sym.} & & & D_{66} & D_{67} & 0 \\ & & & & & & D_{77} & 0 \\ & & & & & & & k_{88}D_{88} \end{bmatrix} \begin{bmatrix} \kappa_x \\ \kappa_y \\ \kappa_{xy} \\ \gamma_{xz} \\ \gamma_{yz} \\ \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{bmatrix} \tag{3.2}$$

3.3 Material Strengths

In Window 1.3, the characteristic strengths of the single layers are displayed. The values of each *Current Composition* are imported from the material library (see Figure 3.7, page 25).



Layer No.	Material Description	$f_{b,0,k}$	$f_{b,90,k}$	$f_{t,0,k}$	$f_{t,90,k}$	$f_{c,0,k}$	$f_{c,90,k}$	$f_{xy,k}$	$f_{v,k}$	$f_{R,k}$
1	Poplar and Coniferous Timber C24	24.0	24.0	14.0	0.4	21.0	2.5	4.0	4.0	2.0
2	Poplar and Coniferous Timber C24	24.0	24.0	14.0	0.4	21.0	2.5	4.0	4.0	2.0
3	Poplar and Coniferous Timber C24	24.0	24.0	14.0	0.4	21.0	2.5	4.0	4.0	2.0

Figure 3.17: Window *1.3 Material Strengths*

In the table, you can modify the values of the *Strengths for Bending / Tension / Compression* as well as of the *Shear Strengths*.

Below the table, there are the same buttons as in the previous Window *1.2 Material Characteristics*. They are described in Chapter 3.2 on page 27.

Again, the *Info* section provides information about the specific weight and surface weight of the current layer, and about the total thickness and total surface weight of the current composition.

3.4 Load Duration and Service Class

If the design is according to EN 1995-1-1:2004-11 or DIN 1052:2010-12 and an action has been selected in the *Ultimate Limit State* tab of Window 1.1, the *1.4 Load Duration and Service Class* window is displayed.

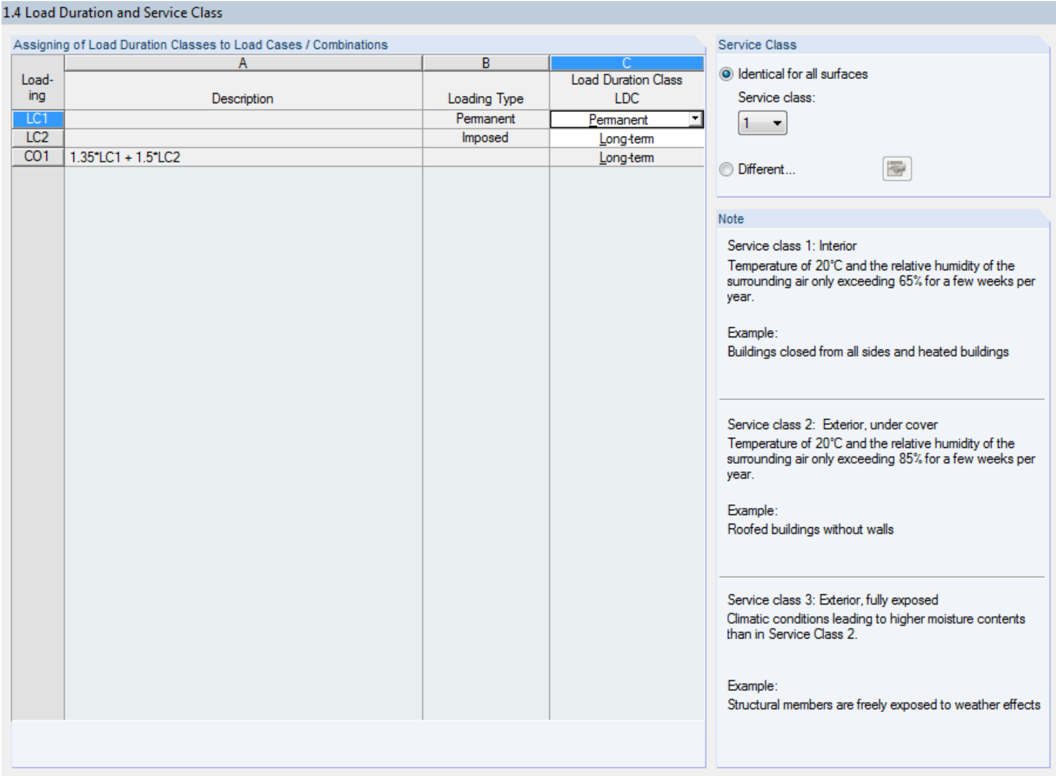


Figure 3.18: Window *1.4 Load Duration and Service Class*

In this window, the load duraction classes and service classes of the actions are to be assigned so that the respective climatic conditions can be accounted for.

Loading

In this table, all load cases and combinations that have been selected for the ULS design are listed. For load or result combinations, the contained load cases are included as well.

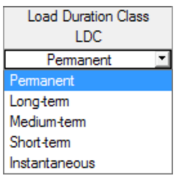
Description

The desciptions as defined in RFEM make it easier to classify the actions.

Loading Type

This column displays the action categories of the load cases according to their definitions in RFEM. The presettings of the next column are based on those loading types.

Load Duration Class LDC

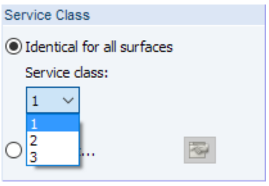


The load cases and their combinations have to allocated to specific of load-duration classes for the design. Those classes are described e.g. in EN 1995-1-1, Table 3.1. When an entry is selected from the list, the corresponding factor k_{mod} is automatically assigned according to the corresponding load-duration class and factor category.

Load and result combinations are classified in accordance with the governing load case.

You can check the values of k_{mod} in the *Standard* dialog box (see Chapter 4.2.1, page 46).

Service Class



By assigning the *Service Class* in the right part of the window, you can control the modification factors k_{mod} and the deflection analysis with respect to the environmental conditions. The service classes are described e.g. in EN 1995-1-1, Clause 2.3.1.3.

By default, all surfaces are allocated to one and the same service class. If you want to assign *Different* service classes, activate the corresponding option and click the button. A new dialog box opens where you can individually assign service classes to selected surfaces.

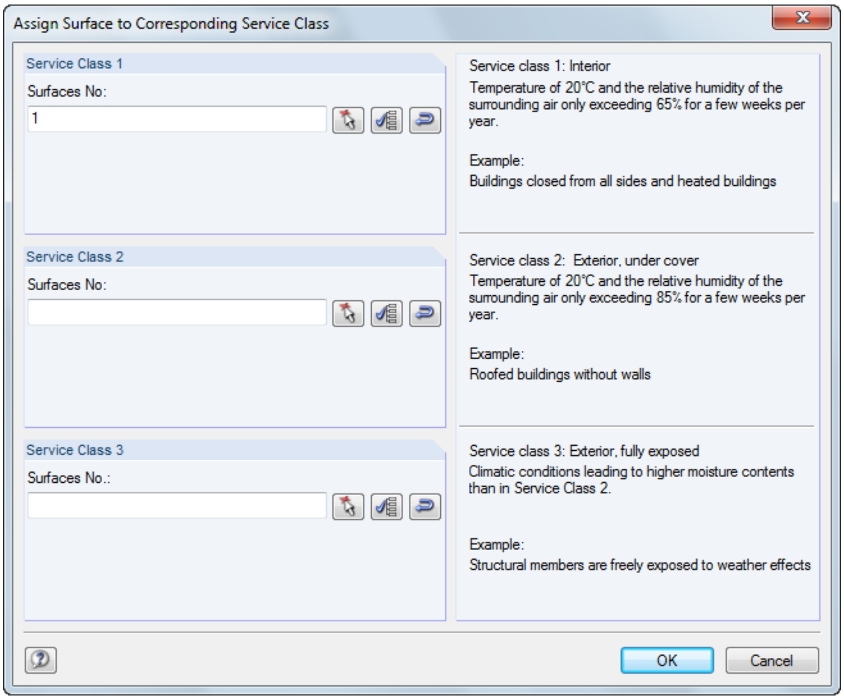


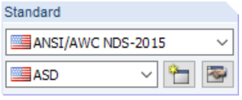
Figure 3.19: Dialog box *Assign Surface to Corresponding Service Class*

The bottons next to the text boxes have the following meanings:

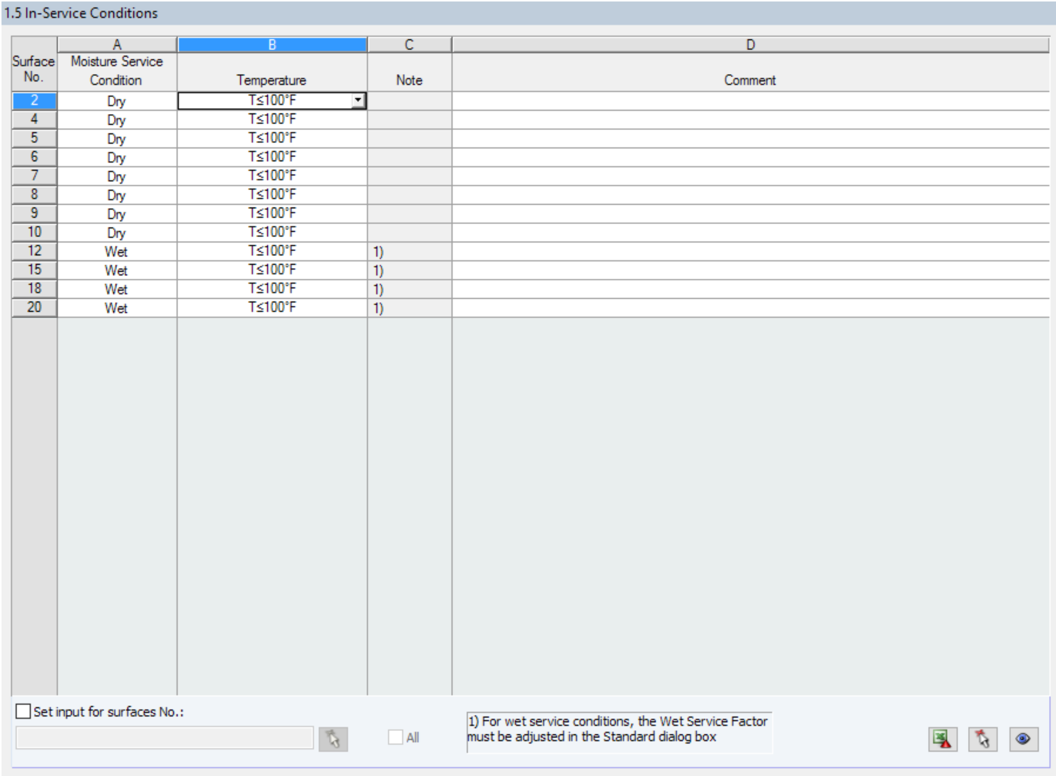
Button	Function
	Select surfaces graphically in the work window of RFEM.
	Assign all surfaces to this service class.
	Assign all surfaces that have not yet been selected to this service class.

Table 3.4: Buttons in dialog box *Assign Surface to Corresponding Service Class*

3.5 In-Service Conditions



If the design is carried out according to ANSI/AWC NDS-2015 [3], Window *1.5 In-Service Conditions* is shown. The settings of this window control the wet service factors, C_M, and temperature factors, C_t.

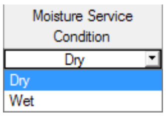


Surface No.	A Moisture Service Condition	B Temperature	C Note	D Comment
2	Dry	T≤100°F		
4	Dry	T≤100°F		
5	Dry	T≤100°F		
6	Dry	T≤100°F		
7	Dry	T≤100°F		
8	Dry	T≤100°F		
9	Dry	T≤100°F		
10	Dry	T≤100°F		
12	Wet	T≤100°F	1)	
15	Wet	T≤100°F	1)	
18	Wet	T≤100°F	1)	
20	Wet	T≤100°F	1)	

Figure 3.20: Window *1.5 In-Service Conditions*

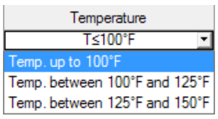
In this table, the in-service conditions can be specified for each surface selected for design.

Moisture Service Condition



By default, *Dry* moisture service conditions are set where the moisture content in service is less than 16 %. To change the service condition, use the button and open the list.

Temperature



For the design, elevated temperatures up to 150°F are possible. If required, the default temperature setting *T≤100°F* can also be modified via the button.

Note

When the settings have been changed, a note may be shown in this column. It is explained below the table.

Set input for surfaces No.

If this check box below the table is selected, the settings entered afterwards will be applied to the selected or to *All* surfaces. The surfaces can be selected by entering their numbers or by clicking them graphically via the [Select] button. That option is useful when you want to assign identical conditions to several surfaces. Please note that any settings that have been already defined cannot be changed subsequently by this function.

3.6 Serviceability Data

Window *1.6 Serviceability Data* contains the last input table for entering data. It is displayed when at least on action has been selected in the *Serviceability Limit State* tab of Window *1.1 General Data*.

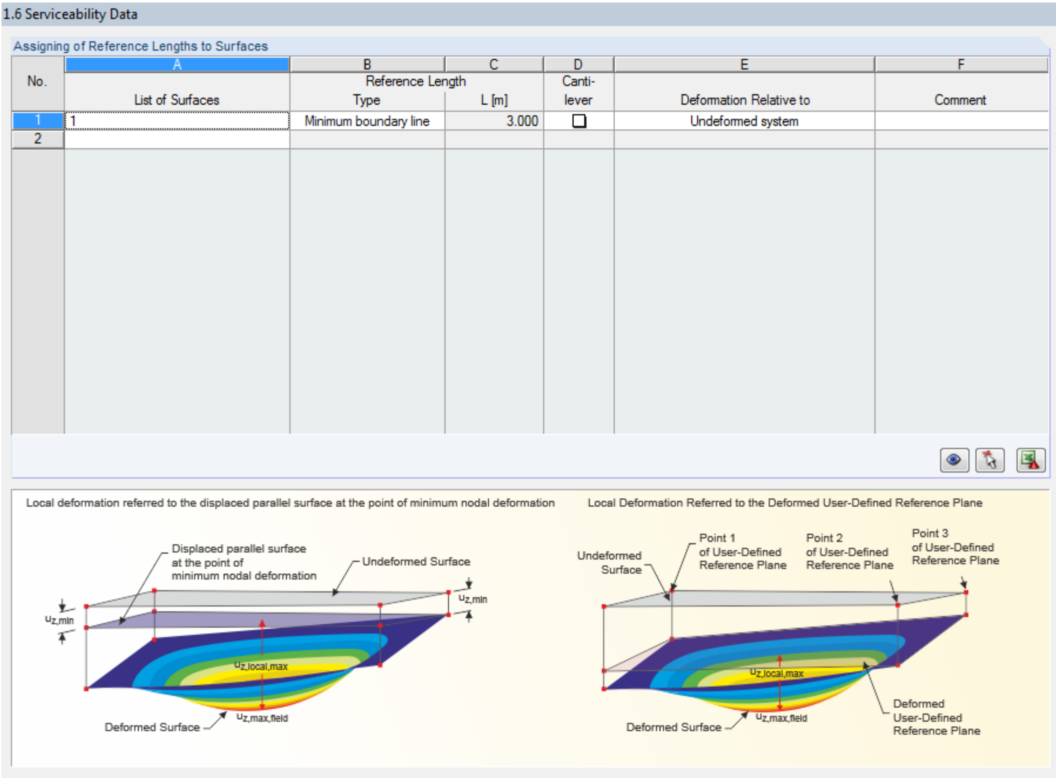


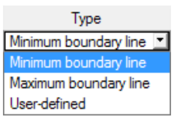
Figure 3.21: Window *1.6 Serviceability Data*

The settings of this window are important for the correct application of the limit deformations. You can check and, if necessary, adjust the limit values of the SLS design in the *Standard* dialog box (see Chapter 4.2.2, page 47).

List of Surfaces

In column A, specify the surfaces whose deformations are to be analyzed.

Reference Length



The *Type* of the reference length can be selected from the list. If the *Maximum boundary line* of a surface is set, the longest side of a surface is applied to determine the limit deformation of e.g. $\frac{\ell}{300}$. With the *Minimum boundary line*, the shortest line is used instead.

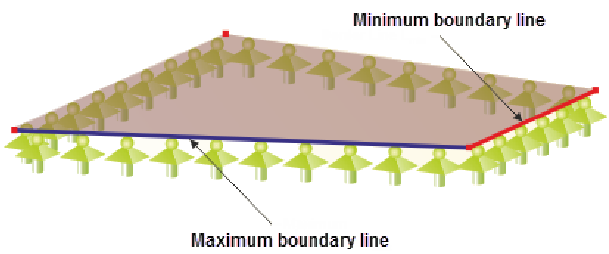
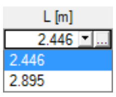


Figure 3.22: Maximum and minimum boundary line to determine $u_{z,max}$

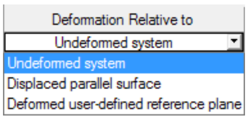


The *User-defined* option enables you to manually define the reference length of the surface. Having selected this entry, you can define the value in the *L* text box. It is also possible to select the length from the list or define it graphically via the [...] button in the work window of RFEM. It may be necessary to set the reference lengths manually for surfaces that are located within other surfaces, for example.

Cantilever

In column D, you can specify whether the surface is a cantilever or not.

Deformation Relative to



The deformation design criterion uses the deflection of a surface, i.e. the perpendicular deformation relative to the shortest line connecting the points of support. There are three possibilities how to calculate the local deformation $u_{z,local}$ which is then used in the design.

- *Undeformed system:* The deformation is related to the initial model.
- *Displaced parallel surface:* This option is recommended for elastic supports. The deformation $u_{z,local}$ is related to a virtual reference surface which is displaced parallel to the undeformed system. For the displacement vector of the reference surface, the minimal nodal deformation of the surface is applied.

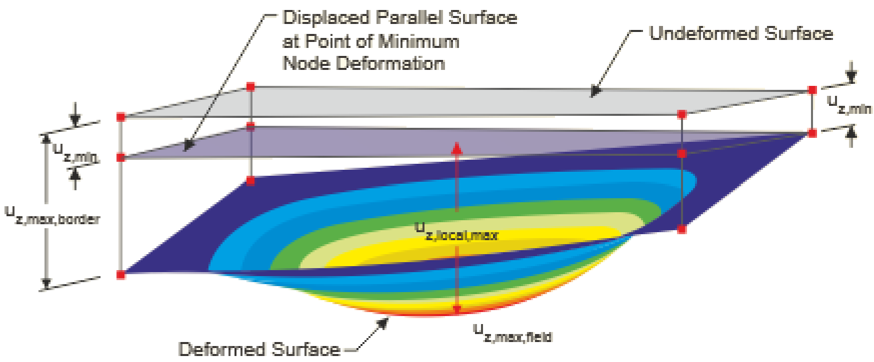


Figure 3.23: Displaced parallel surface, with smallest nodal deformation $u_{z,min}$ as displacement vector

- *Deformed reference plane:* If the deformations of the supports differ considerably, an inclined reference plane can be defined for the relevant deformation $u_{z,local}$. The plane is to be defined by three points of the undeformed system. The proram determines the deformations of those three points, places the reference plane in the displaced points, and then calculates the deformation $u_{z,local}$.

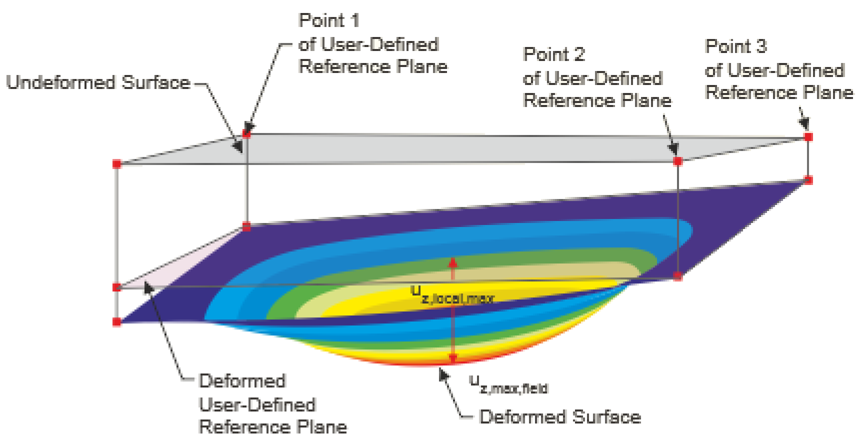


Figure 3.24: Displaced user-defined reference plane

4 Calculation

Details...

Before starting the calculation, you should check the detailed settings for the design. By clicking the [Details] button, you open the relevant dialog box which is described below.

Right at the start of the calculation, the program checks whether the global stiffness matrix is positive-definite (see Chapter 9.2, page 92).

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

The calculation then runs globally for the entire structure modeled in RFEM.

4.1 Details

The *Details* dialog box consists of these tabs:

- *Stresses*
- *Results*

The following buttons are common for both tabs:

Button	Name	Function
	Help	Call up the online help.
	Units and Decimal Places	Open the *Units and Decimal Places* dialog box that controls the units of RF-LAMINATE.
	Reset Dlubal Default	Set all parameters in the *Details* dialog box to the original DLUBALvalues.
	Default	Set all parameters in the *Details* dialog box according to the default setting that was saved before.
	Set As Default	Save the current setting as default. It can be reloaded to any other RF-LAMINATE case via the button.

Table 4.1: Buttons in *Details* dialog box

4.1.1 Stresses

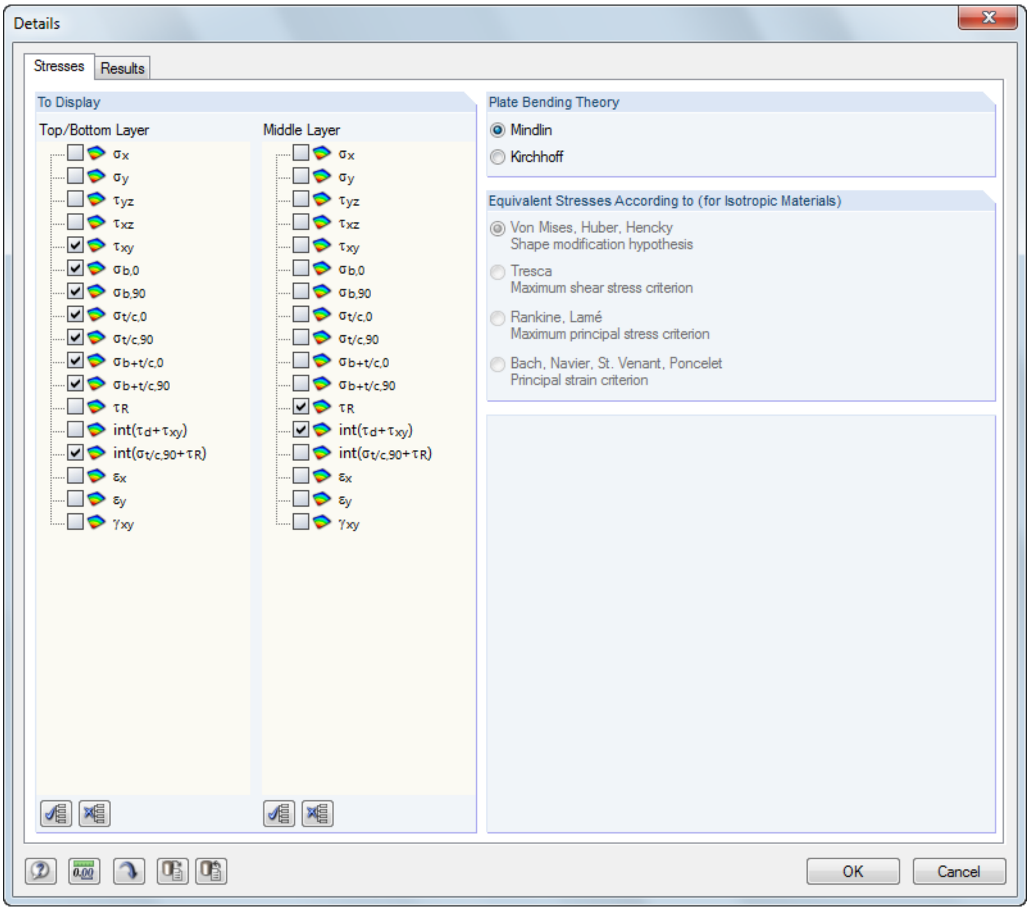
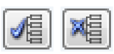


Figure 4.1: *Details* dialog box, *Stresses* tab

To Display

By selecting the appropriate check boxes in this dialog section, you determine which stresses are displayed in the result tables. The stresses are adjustable individually for *Top/Bottom Layer* and *Middle Layer*.



The [Select All] and [Deselect All] buttons facilitate selecting the stress types.

The basic stresses σ_x, σ_y, τ_{xy}, τ_{xz}, and τ_{yz} are calculated by the finite element method in RFEM. From those basic stresses, all other stresses are determined by the RF-LAMINATE module.

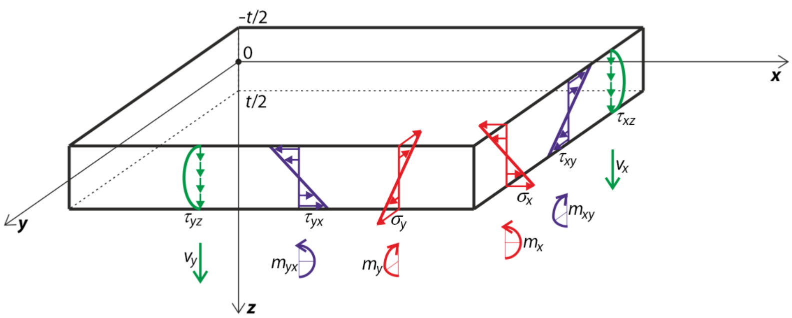


Figure 4.2: Basic stresses and sign convention for a single-layer plate subjected to bending

In Table 4.2, the equations are given that are valid for single-layer plates.

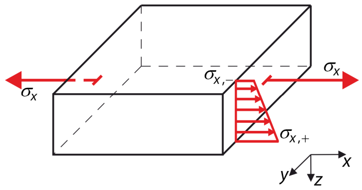
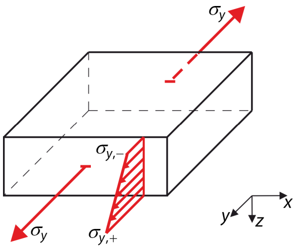
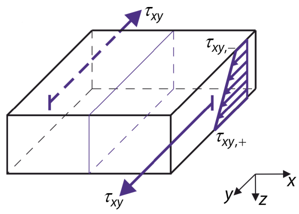
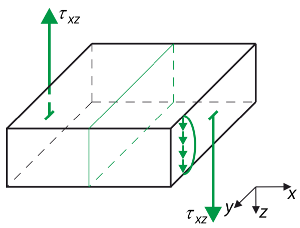
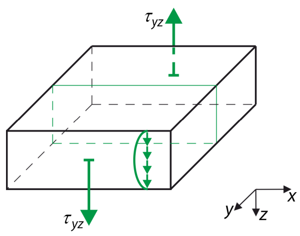
σ_x	Normal stress in *x*-direction • Stress on positive surface side $\sigma_{x,+} = \frac{n_x}{t} + \frac{6\,m_x}{t^2}$ where t = plate thickness • Stress on negative surface side $\sigma_{x,-} = \frac{n_x}{t} - \frac{6\,m_x}{t^2}$	
σ_y	Normal stress in *y*-direction • Stress on positive surface side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

Table 4.2: Basic stresses

In general, the stresses in the single layers are calculated from the total internal strains of the plate:

$$\varepsilon_{tot}^{T} = \left[\frac{\partial\varphi_y}{\partial x}, -\frac{\partial\varphi_x}{\partial y}, \frac{\partial\varphi_y}{\partial y} - \frac{\partial\varphi_x}{\partial x}, \frac{\partial w}{\partial x} + \varphi_y, \frac{\partial w}{\partial y} - \varphi_x, \frac{\partial u}{\partial x}, \frac{\partial v}{\partial y}, \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right] \tag{4.2}$$

The strains in the individual layers are calculated by using the relation

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

where z is the coordinate in z-direction in which the stress value is requested. For the e.g. i-th layer, the stress is calculated by using the relation

$$\boldsymbol{\sigma}\,(\boldsymbol{z}) = \boldsymbol{d_i}\varepsilon\,(\boldsymbol{z}) \tag{4.4}$$

where $\boldsymbol{d_i}$ is the partial stiffness matrix of the i-th layer.

According to the selected material model (isotropic or orthotropic) the selection for the stresses in details is changed.

Isotropic material model

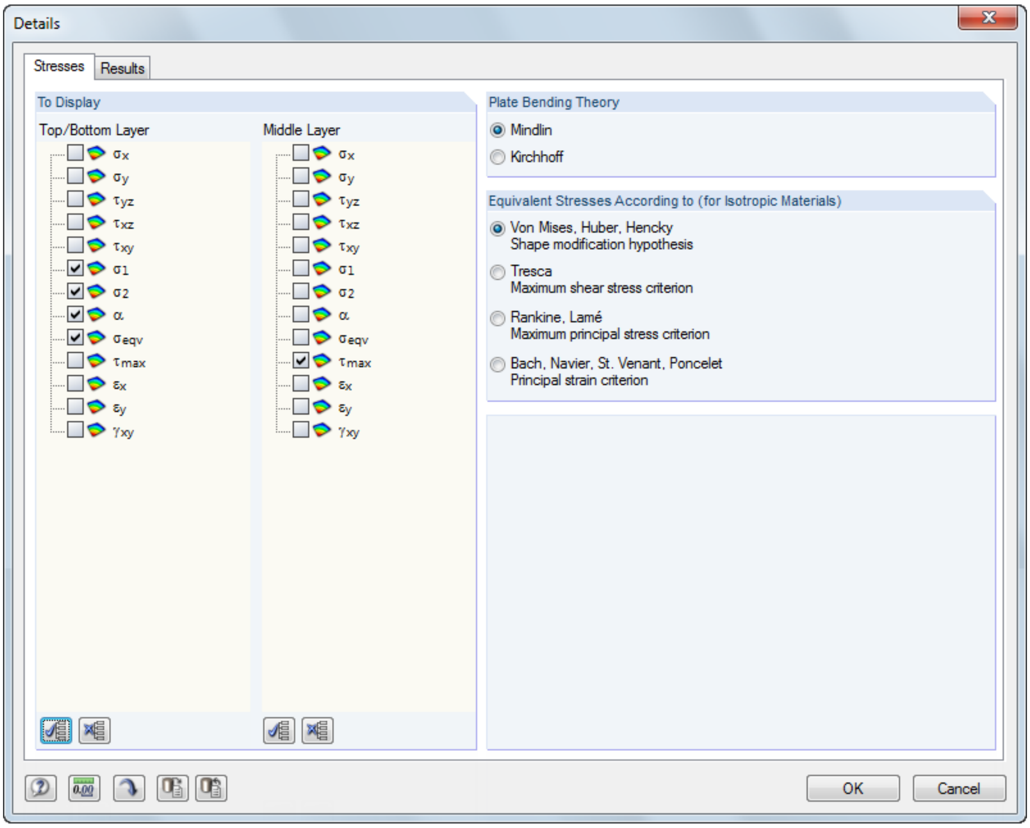


Figure 4.3: *Details* dialog box, *Stresses* tab for isotropic material model

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

τ_{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 Table 4.4. The effect of the shear stresses is neglected in the formulas τ_{xz} and τ_{yz}.

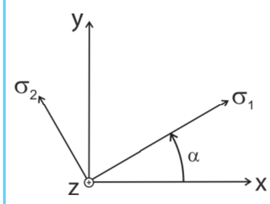
<table>
<tr><td>σ_1</td><td colspan="2">Principal stress
$$\sigma_1 = \frac{\sigma_x + \sigma_y + \sqrt{\left(\sigma_x - \sigma_y\right)^2 + 4\,\tau_{xy}^2}}{2}$$</td></tr>
<tr><td>σ_2</td><td colspan="2">Principal stress
$$\sigma_2 = \frac{\sigma_x + \sigma_y - \sqrt{\left(\sigma_x - \sigma_y\right)^2 + 4\,\tau_{xy}^2}}{2}$$</td></tr>
<tr><td>α</td><td>Angle between local axis x and direction of first principal stress
$$\alpha = \frac{1}{2}\mathrm{atan2}\left(2\tau_{xy}, \sigma_x - \sigma_y\right), \quad \alpha \in \left(-90\,°, 90\,°\right\rangle$$
The atan2 function is implemented in RFEM as follows:
$$\mathrm{atan2}\,(y,x) = \begin{cases} \arctan\frac{y}{x} & x > 0 \\ \arctan\frac{y}{x} + \pi & y \geq 0, x < 0 \\ \arctan\frac{y}{x} - \pi & y < 0, x < 0 \\ +\frac{\pi}{2} & y > 0, x = 0 \\ -\frac{\pi}{2} & y < 0, x = 0 \\ 0 & y = 0, x = 0 \end{cases}$$</td><td></td></tr>
<tr><td rowspan="4">σ_{eqv}</td><td colspan="2">Equivalent stress according to von Mises, Huber, Hencky – Shape modification hypothesis
$$\sigma_{eqv} = \sqrt{\sigma_x^2 + \sigma_y^2 - \sigma_x\sigma_y + 3\,\tau_{xy}^2}$$</td></tr>
<tr><td colspan="2">Equivalent stress according to Tresca – Maximum shear stress criterion
$$\sigma_{eqv} = \max\left[\sqrt{\left(\sigma_x - \sigma_y\right)^2 + 4\,\tau_{xy}^2}, \frac{\left|\sigma_x + \sigma_y\right| + \sqrt{\left(\sigma_x - \sigma_y\right)^2 + 4\,\tau_{xy}^2}}{2}\right]$$</td></tr>
<tr><td colspan="2">Equivalent stress according to Rankine, Lamé – Maximum principal stress criterion
$$\sigma_{eqv} = \frac{\left|\sigma_x + \sigma_y\right| + \sqrt{\left(\sigma_x - \sigma_y\right)^2 + 4\,\tau_{xy}^2}}{2}$$</td></tr>
<tr><td colspan="2">Equivalent stress according to Bach, Navier, St. Venant, Poncelet – Principal strain criterion
$$\sigma_{eqv} = \max\left[\frac{1-\nu}{2}\left|\sigma_x + \sigma_y\right| + \frac{1+\nu}{2}\sqrt{\left(\sigma_x - \sigma_y\right)^2 + 4\,\tau_{xy}^2}, \nu\left|\sigma_x + \sigma_y\right|\right]$$</td></tr>
</table>

Table 4.4: Stresses for isotropic material model

Orthotropic material model

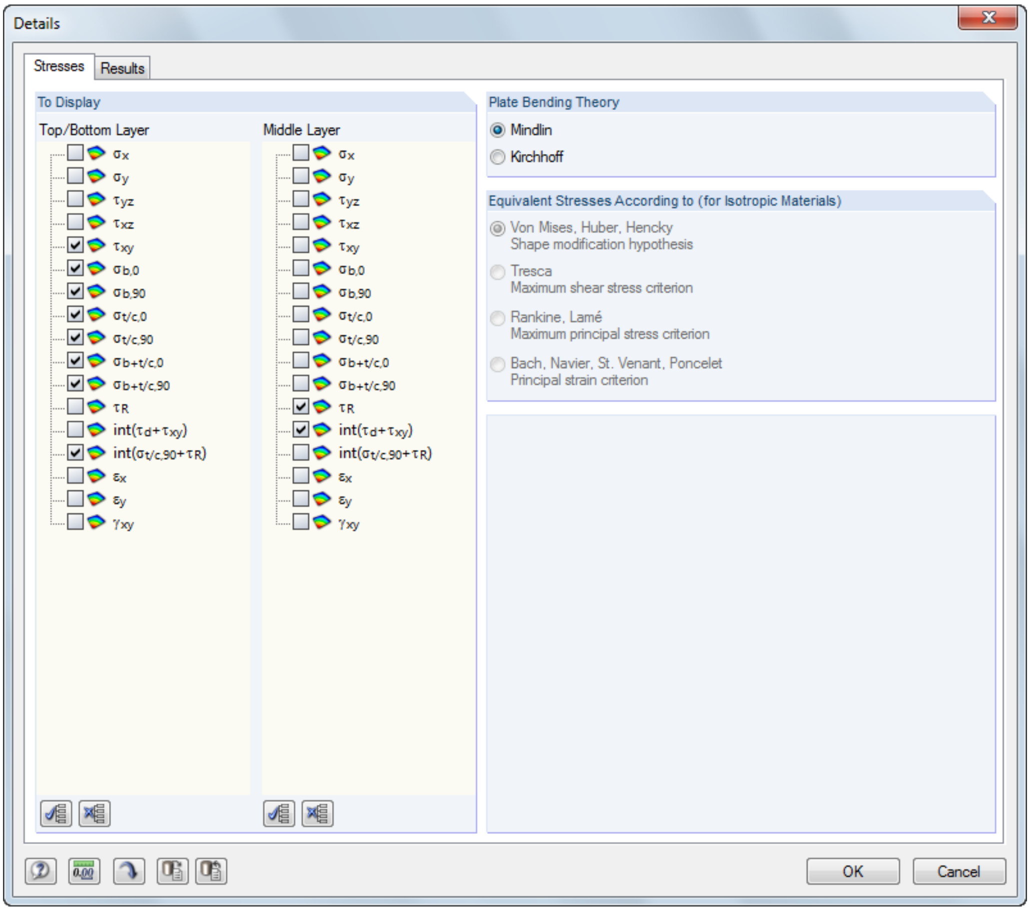
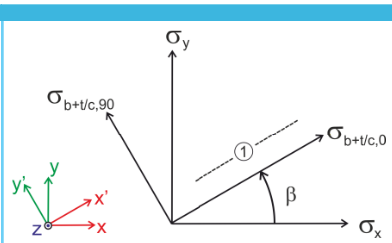


Figure 4.4: *Details* dialog box, *Stresses* tab for orthotropic material model

$\sigma_{b+t/c,0}$	Normal stress along the grain $\sigma_{b+t/c,0} = \sigma_x \cos^2\beta + \tau_{xy} \sin 2\beta + \sigma_y \sin^2\beta$	 *1 – Direction of grain
$\sigma_{b+t/c,90}$	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 $\sigma_{t/c,0} = \dfrac{\sigma_{b+t/c,0(\text{top})} + \sigma_{b+t/c,0(\text{middle})} + \sigma_{b+t/c,0(\text{bottom})}}{3}$	
$\sigma_{t/c,90}$	Tension/compression component of the normal stress perpendicular to the grain $\sigma_{t/c,90} = \dfrac{\sigma_{b+t/c,90(\text{top})} + \sigma_{b+t/c,90(\text{middle})} + \sigma_{b+t/c,90(\text{bottom})}}{3}$	

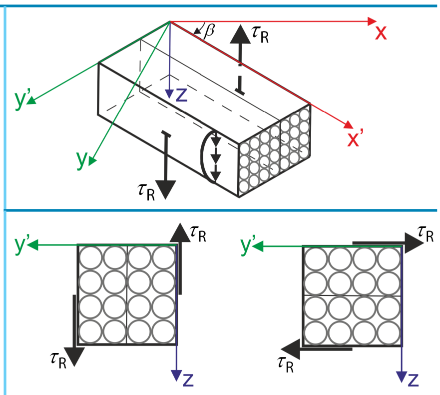
$\sigma_{b,0}$	Bending component of the normal stress along the grain $\sigma_{b,0} = \sigma_{b+t/c,0} - \sigma_{t/c,0}$
$\sigma_{b,90}$	Bending component of the normal stress perpendicular to the grain $\sigma_{b,90} = \sigma_{b+t/c,90} - \sigma_{t/c,90}$
τ_R	Rolling shear stress $\tau_R = -\tau_{xz} \sin\beta + \tau_{yz} \cos\beta$ 

Table 4.5: Stresses for orthotropic material model

The stresses $\sigma_{b+t/c,0}$, $\sigma_{b+t/c,90}$, $\sigma_{t/c,0}$, $\sigma_{t/c,90}$, $\sigma_{b,0}$, $\sigma_{b,90}$, and τ_R are expressed in the coordinate system of the grain x', y', z. As the grain can be rotated individually in each layer, discontinuities of the stress values may occur at the boundaries of the layers. The transformation formulas for those stresses are introduced in Equation 5.1 and Equation 5.2 on page 52.

The normal stress includes the tension/compression components and the bending components of the individual layers.

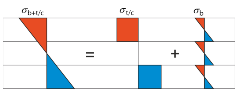
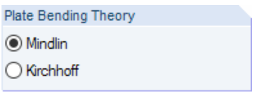


Figure 4.5: Normal stress – shares of tension/compression components and bending components

Plate Bending Theory



For surfaces, two bending theories are available:

- Mindlin
- Kirchhoff

The shear strain is considered for the calculation according to the MINDLIN theory, but not according to KIRCHHOFF.

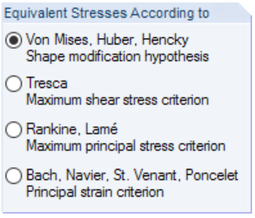
The bending theory according to MINDLIN is suitable for rather massive plates. For relatively thin plates, however, the bending theory according to KIRCHHOFF is recommended.

As the shear stresses τ_{xz} and τ_{yz} are not determined precisely according to KIRCHHOFF, they are calculated from the equilibrium conditions as follows.

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

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

Equivalent Stresses According to (for Isotropic Materials)



For isotropic materials, the equivalent stresses can be determined in four different ways. If the orthotropic material model has been selected, no equivalent stresses can be calculated.

Von Mises, Huber, Hencky – Shape modification hypothesis

This hypothesis is also known as HMH or as the energy criterion. The equivalent stress is calculated by using the relation

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

Tresca – Maximum shear stress criterion

Commonly, this equivalent stress is defined by the relation

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

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

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

and the resulting equation

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

Rankine, Lamé – Maximum principal stress criterion

This hypothesis is also known as the normal stress hypothesis. The RANKINE's stress is generally defined as the maximum of absolute values resulting from the principal stresses.

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

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

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

and the resulting equation

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

Bach, Navier, St. Venant, Poncelet – Principal strain criterion

According to this hypothesis, the equivalent stress is based on the principal deformation. It is assumed that the failure occurs in the direction of the maximum strain.

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

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

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

and the resulting equation

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

In all equations concerning the equivalent stress, the influence of the shear stresses τ_{xz} and τ_{yz} is neglected.

4.1.2 Results

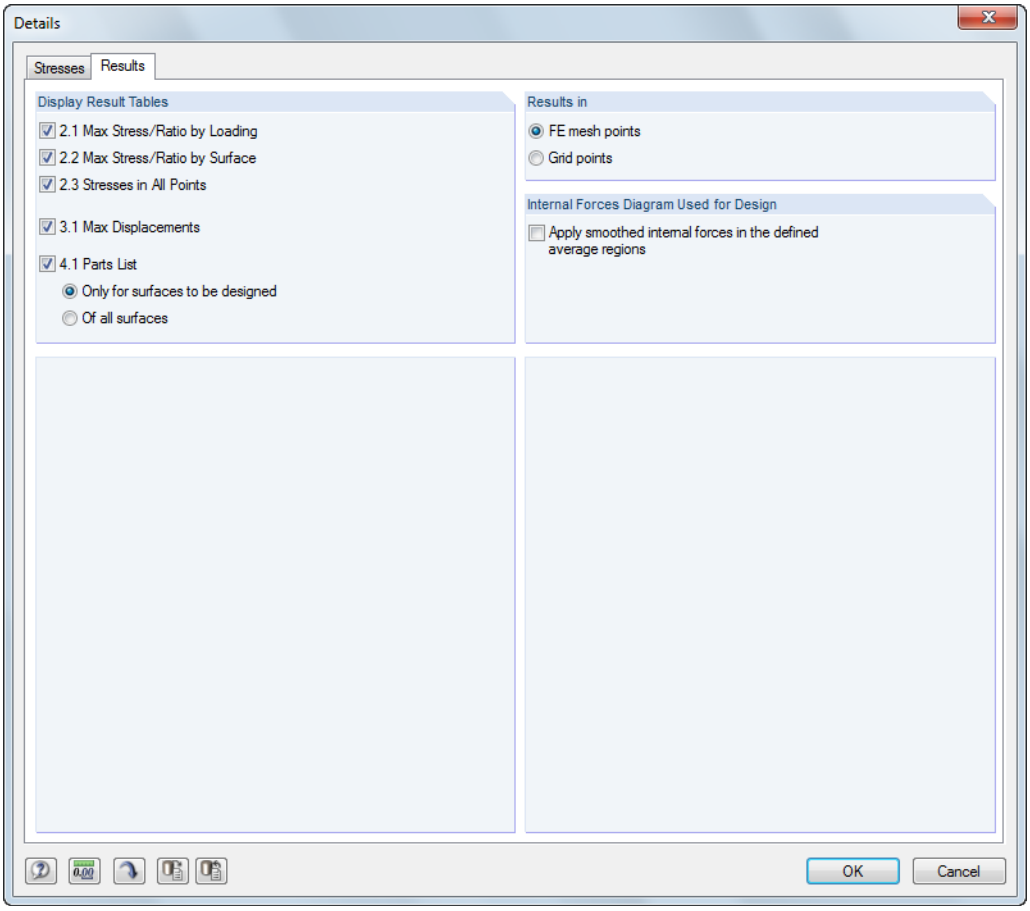


Figure 4.6: *Details* dialog box, *Results* tab

Display Result Tables

In this dialog section, you can select the result tables that are to be displayed after the calculation (stresses, displacements, parts list).

The result windows are described in Chapter 5.

Results in

By default, the stresses and displacements are displayed in all *FE mesh points*. Alternatively, you can set the results in the *Grid points* of each surface. Grid points can be defined in RFEM as a property of a surface (see RFEM manual, Chapter 8.13).

If a surface is rather small, the default grid point spacing of 0.5 m may produce very few grid points, or even only one grid point in the origin. In that case, the maximum values will not be covered by the result tables: the grid is not fine enough. You should then adapt the grid to the dimensions of the surface in RFEM so that more grid points are created.

Internal Forces Diagram Used for Design

If you select the check box *Apply smoothed internal forces in the defined average regions*, you can use the smoothed results of the average regions for the stress calculation in RF-LAMINATE. Details on the average regions can be found in the RFEM manual, Chapter 9.7.3.

4.2 Standard

Standard

To open the *Standard* dialog box, click the corresponding button. This button is available in every window of the RF-LAMINATE module.

In the upper right corner of Window *1.1 General Data*, you can select the standard whose parameters are relevant for the design and whose limit values of the deflection are to be applied (see Figure 3.2, page 21).

The following standards can be selected:

- None
- DIN 1052:2010-12 [4]
- EN 1995-1-1:2004-11 [2] with National annexes
- ANSI/AWC NDS-2015 [3]

If you select *None*, you can enter user-defined basic values for the material properties, γ_M, and for the serviceability limits that are independent of any specific standard.

The *Standard* dialog box is described exemplarily for **EN 1995-1-1:2004-11** to illustrate the relevant parameters.

For EN 1995-1-1, the design values of stresses (with subscript *d*) are calculated from the characteristic limit values of stresses (with subscript *k*) according to the following relation:

$$\begin{Bmatrix} f_{b,d} \\ f_{t,d} \\ f_{c,d} \\ f_{b,0,d} \\ f_{t,0,d} \\ f_{c,0,d} \\ f_{b,90,d} \\ f_{t,90,d} \\ f_{c,90,d} \\ f_{xy,d} \\ f_{v,d} \\ f_{\mathrm{eqv},d} \\ f_{R,d} \end{Bmatrix} = \frac{k_{\mathrm{mod}}}{\gamma_M} \begin{Bmatrix} f_{b,k} \\ f_{t,k} \\ f_{c,k} \\ f_{b,0,k} \\ f_{t,0,k} \\ f_{c,0,k} \\ f_{b,90,k} \\ f_{t,90,k} \\ f_{c,90,k} \\ f_{xy,k} \\ f_{v,k} \\ f_{\mathrm{eqv},k} \\ f_{R,k} \end{Bmatrix} \tag{4.17}$$

The *Standard - EN 1995-1-1* dialog box consists of these tabs:

- *Material Factors*
- *Serviceability Limits*

4.2.1 Material Factors

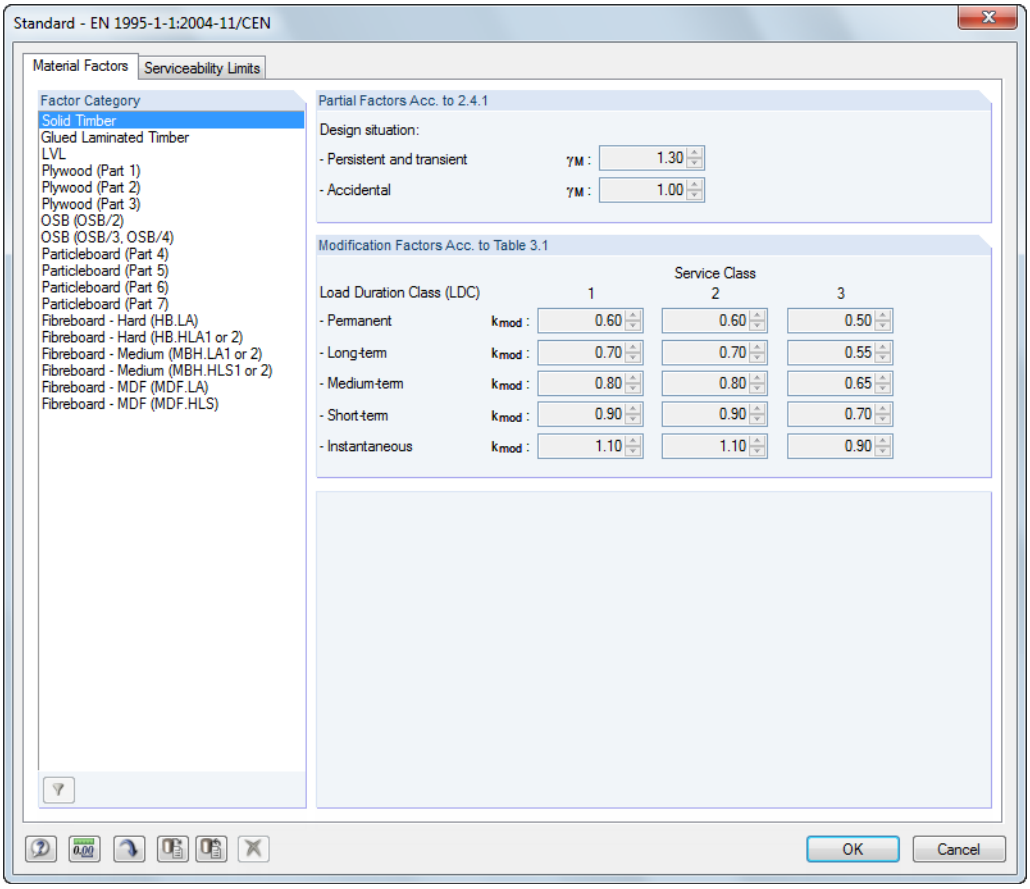


Figure 4.7: *Standard* dialog box for EN 1995-1-1, *Material Factors* tab

Factor Category

The material grades listed in the *Factor Category* correspond to the entries in column B of the *1.2 Material Characteristics* Window (see Figure 3.6, page 24). RF-LAMINATE presets the partial factors and modification factors according to the selected category.

To display all available categories in the list, use the [Include usused material categories] button.

If you want to apply user-defined factors, create a [New Standard or National Annex] in the *1.1 General Data* Window. Then you can define the relevant parameters in the *Material Factors* tab.

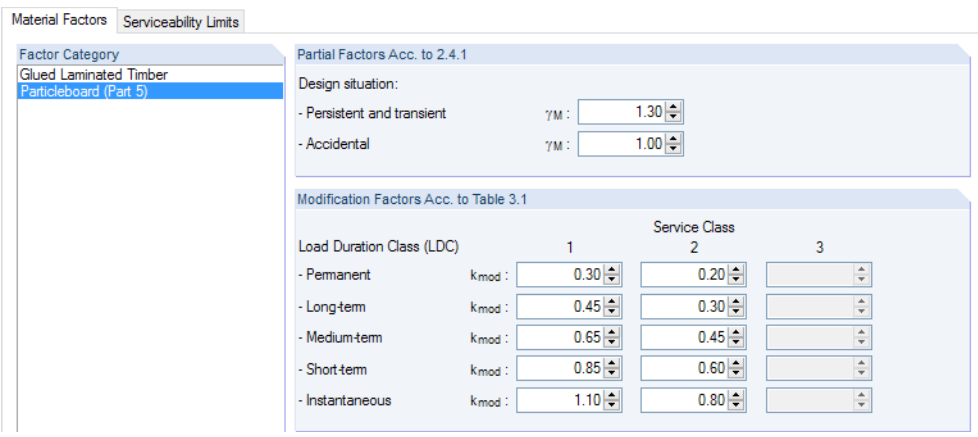
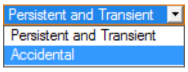


Figure 4.8: *Material Factors* tab of user-defined standard

For particleboard materials, service class 3 is not allowed (see Figure 4.8).

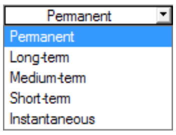
Partial Factors Acc. to 2.4.1



In this dialog section, you can check the partial factors of the material properties, γ_M, for each different design situation. The design situations are to be assigned to the selected load cases and combinations in the *Ultimate Limit State* tab of the *1.1 General Data* Window (see Chapter 3.1.1, page 22).

Modification Factors Acc. to Table 3.1

For the selected *Factor Category*, the values of the modification factor k_{mod} are displayed for the different load duration classes and service classes. They are specified in [2], Table 3.1.



The modification factor k_{mod} is assigned to the load cases according to the load duration and service classes as defined in the *1.4 Load Duration and Service Class* Window (see Chapter 3.4, Page 31).

4.2.2 Serviceability Limits

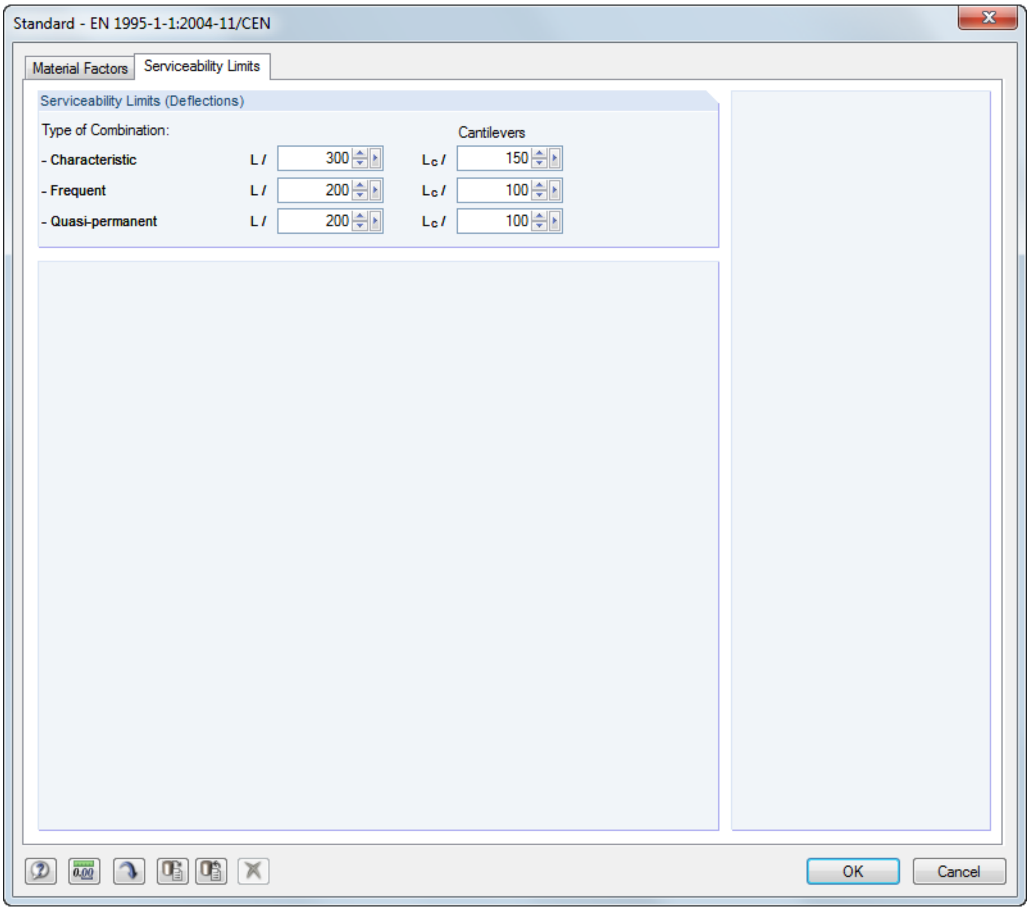
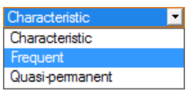
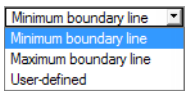


Figure 4.9: *Standard* dialog box for EN 1995-1-1, *Serviceability Limits* tab

The limit values of the allowable deflections are controlled by six text boxes. Thus, you can define specific limits for the different action combinations (*Characteristic*, *Frequent*, *Quasi-permanent*) as well as for surfaces supported on both sides or one side only (*Cantilevers*).



The load cases can be classified in the *Serviceability Limit State* tab of the *1.1 General Data* Window (see Chapter 3.1.2, page 23).



In the *1.6 Serviceability Data* Window, the reference length *L* of each surface is to be defined (see Chapter 3.6, page 34).

4.3 Starting Calculation

Calculation

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

You can also start the RF-LAMINATE calculation in the user interface of RFEM: Open the *To Calculate* dialog box by using the command from the main menu

Calculate → **To Calculate**.

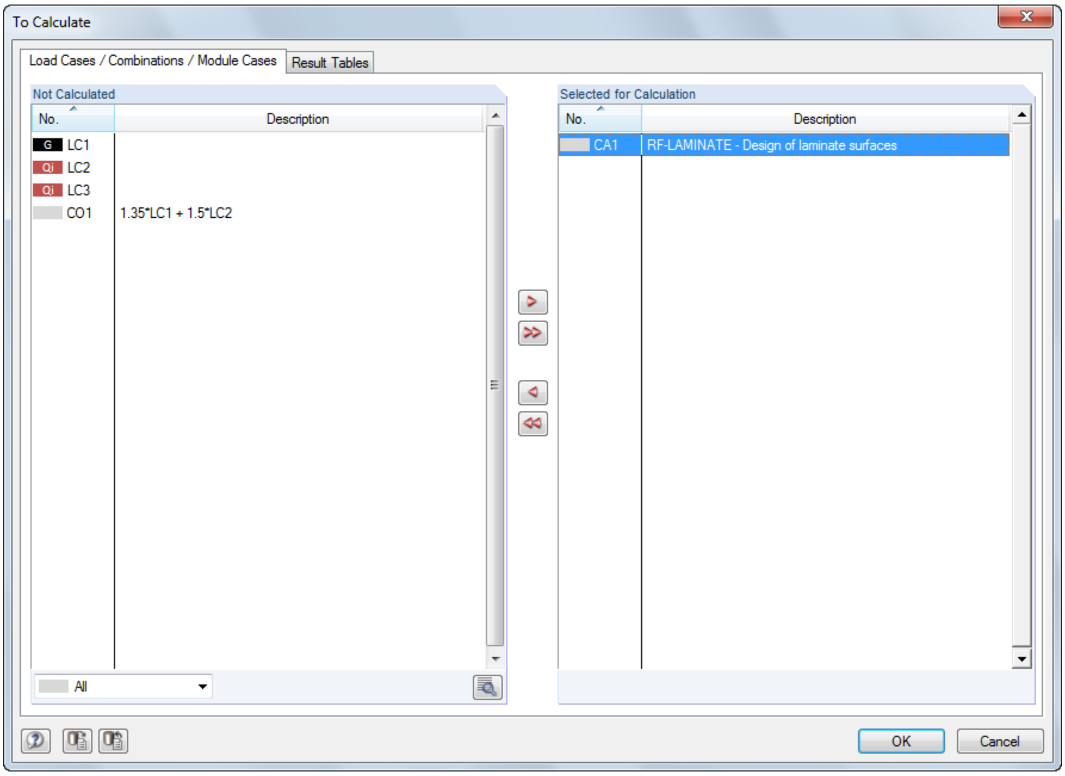


Figure 4.10: *To Calculate* dialog box in RFEM

If the RF-LAMINATE design case is missing in the *Not Calculated* list, select *Add-on Modules* or *All* below the list.

Add the selected design case to the list on the right with the [>] button. Then start the calculation with [OK].

It is also possible to start the calculation of RF-LAMINATE from the RFEM toolbar: set RF-LAMINATE in the list and then click the [Show Results] button.

Figure 4.11: Starting RF-LAMINATE calculation in toolbar

5 Results

Window *2.1 Max Stress Ratio by Loading* is shown immediately after the calculation.

Details...

In the *Details* dialog box, you can specify which result windows are to be displayed (see Chapter 4.1.2, page 44).

To select a result window, click the corresponding entry in the navigator. To set the previous or next window, use the buttons shown on the left. You can also use he function keys to select the next [F2] or previous [F3] window.

OK Cancel

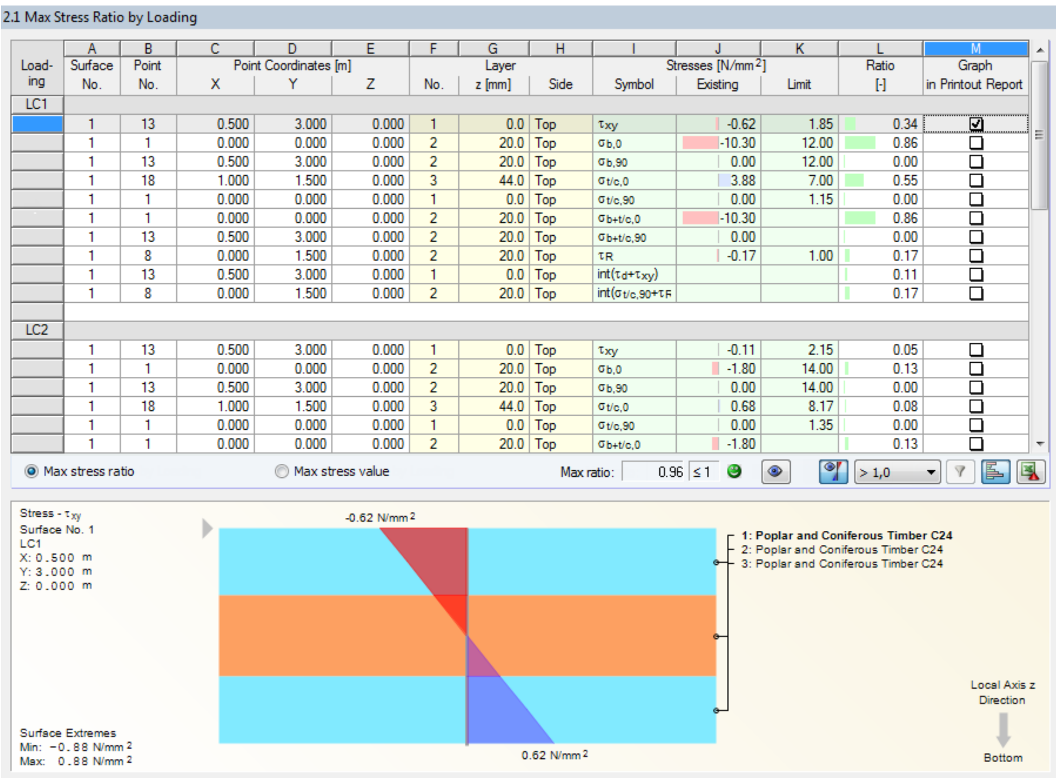
[OK] saves all data and closes RF-LAMINATE. To quit the module without saving, click [Cancel].

In the result windows, several buttons are available. They have the following functions:

Button	Name	Function
	View Mode	Jump to RFEM work window without closing RF-LAMINATE.
	Selection	Select surface or point graphically to display its results in table.
	Graphical Results	Display or hide results of current line in RFEM work window.
> 1,0 > 1,0 Max Define	Filter Parameters	Define criterion to filter results in tables: ratios greater than 1, maximum value, or user-defined limit.
	Color Bars	Display or hide colored relation scales in result tables.
	Excel Export	Export current table to MS Excel or OpenOffice Calc → Chapter 7.2, page 61.

Table 5.1: Buttons in result windows

5.1 Max Stress Ratio by Loading


2.1 Max Stress Ratio by Loading

Loading	A Surface No.	B Point No.	C Point Coordinates [m] X	D Y	E Z	F Layer No.	G z [mm]	H Side	I Stresses [N/mm²] Symbol	J Existing	K Limit	L Ratio [-]	M Graph in Printout Report
LC1													
	1	13	0.500	3.000	0.000	1	0.0	Top	τ_{xy}	-0.62	1.85	0.34	☑
	1	1	0.000	0.000	0.000	2	20.0	Top	$\sigma_{b,0}$	-10.30	12.00	0.86	☐
	1	13	0.500	3.000	0.000	2	20.0	Top	$\sigma_{b,90}$	0.00	12.00	0.00	☐
	1	18	1.000	1.500	0.000	3	44.0	Top	$\sigma_{t/c,0}$	3.88	7.00	0.55	☐
	1	1	0.000	0.000	0.000	1	0.0	Top	$\sigma_{t/c,90}$	0.00	1.15	0.00	☐
	1	1	0.000	0.000	0.000	2	20.0	Top	$\sigma_{b+t/c,0}$	-10.30		0.86	☐
	1	13	0.500	3.000	0.000	2	20.0	Top	$\sigma_{b+t/c,90}$	0.00		0.00	☐
	1	8	0.000	1.500	0.000	2	20.0	Top	τ_R	-0.17	1.00	0.17	☐
	1	13	0.500	3.000	0.000	1	0.0	Top	int($\tau_d+\tau_{xy}$)			0.11	☐
	1	8	0.000	1.500	0.000	2	20.0	Top	int($\sigma_{t/c,90}+\tau_R$			0.17	☐
LC2													
	1	13	0.500	3.000	0.000	1	0.0	Top	τ_{xy}	-0.11	2.15	0.05	☐
	1	1	0.000	0.000	0.000	2	20.0	Top	$\sigma_{b,0}$	-1.80	14.00	0.13	☐
	1	13	0.500	3.000	0.000	2	20.0	Top	$\sigma_{b,90}$	0.00	14.00	0.00	☐
	1	18	1.000	1.500	0.000	3	44.0	Top	$\sigma_{t/c,0}$	0.68	8.17	0.08	☐
	1	1	0.000	0.000	0.000	1	0.0	Top	$\sigma_{t/c,90}$	0.00	1.35	0.00	☐
	1	1	0.000	0.000	0.000	2	20.0	Top	$\sigma_{b+t/c,0}$	-1.80		0.13	☐

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

In this window, the maximum stress ratios (or maximum stress values) are displayed for every load case, load or result combination that was selected for design in Window *1.1 General Data*, tab *Ultimate Limit State*. The numbers of load cases, load and result combinations are shown in the headings of each table section.

There are two radio buttons below the table. They control whether the *Max stress ratio* or the *Max stress value* is listed for each type of stress in the table. For compositions with layers from different materials, there may be differences between the maximum ratios and the maximum stress values. The two options enable you to evaluate the results accordingly.

Surface No.

This column contains the numbers of those surfaces in which the maximum stress ratios or stress values occur. The results are shown for every designed load case.

Point No.

In this column, the numbers of the FE mesh nodes are displayed where the maximum stress ratios or stress values occur. The respective types of stresses are given in the *Symbol* Column.

Alternatively, the numbers of the grid points are listed, depending on the settings in the *Details* dialog box, tab *Results* (see Chapter 4.1.2, page 44). The grid points are an option to display the results independently of the FE mesh, according to their specification in RFEM for each surface.

Point Coordinates

The global coordinates X, Y, Z of each FE mesh point (or grid point) are specified in these columns.

Layer

In columns F to H, the numbers of the layers are listed with their *z*-coordinates and sides where the maximum stress ratios (or maximimum stress values) occur.

Stresses

Symbol

In column I, the types of stresses are described whose values are listed in the next column.

You can reduce or extend the list of stresses in the *Details* dialog box (see Chapter 4.1.1, page 37).

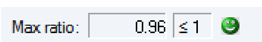
Existing

In this column, the calculated values of the stresses are listed. They are determined according to the equations that you can review in Table 4.2 to Table 4.5.

Limit

The limit values or limit stresses are based on the material properties specified in the *1.3 Material Strengths* Window and on the selected standard. Equation 4.17 on page 45 describes how the limit values are calculated according to EN 1995-1-1.

Ratio



The ratio of the calculated stress and limit stress is listed for every stress component. If the limit stress is not exceeded, the ratio is less than or equal to 1 and the stress design is satisfied. Thus, the entries in column L enable you to quickly assess the efficiency of the design.

Table 5.2 and Table 5.3 illustrate how the ratios are determined for the different types of stresses.

Isotropic material model

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

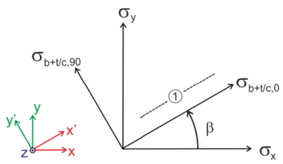
Table 5.2: Ratios for isotropic material model

Orthotropic material model

Stresses [Pa]	Ratios [–]	
$\sigma_{b,0}$	$\dfrac{\lvert\sigma_{b,0}\rvert}{f_{b,0,d}}$	
$\sigma_{b,90}$	$\dfrac{\lvert\sigma_{b,90}\rvert}{f_{b,90,d}}$	
$\sigma_{t/c,0}$	$= \begin{cases} \dfrac{\sigma_{t/c,0}}{f_{t,0,d}} & \text{if } \sigma_{t/c,0} > 0 \\ \dfrac{\lvert\sigma_{t/c,0}\rvert}{f_{c,0,d}} & \text{if } \sigma_{t/c,0} \leq 0 \end{cases}$	
$\sigma_{t/c,90}$	$= \begin{cases} \dfrac{\sigma_{t/c,90}}{f_{t,90,d}} & \text{if } \sigma_{t/c,90} > 0 \\ \dfrac{\lvert\sigma_{t/c,90}\rvert}{f_{c,90,d}} & \text{if } \sigma_{t/c,90} \leq 0 \end{cases}$	
$\sigma_{b+t/c,0}$	$= \begin{cases} \dfrac{\sigma_{t/c,0}}{f_{t,0,d}} + \dfrac{\lvert\sigma_{b,0}\rvert}{f_{b,0,d}} & \text{if } \sigma_{t/c,0} > 0 \\ \dfrac{\lvert\sigma_{t/c,0}\rvert}{f_{c,0,d}} + \dfrac{\lvert\sigma_{b,0}\rvert}{f_{b,0,d}} & \text{if } \sigma_{t/c,0} \leq 0 \end{cases}$	According to: ČSN 73 1702, (127), (128) DIN 1052, (127), (128) DIN EN 1995-1-1/NA, (NA.141), (NA.142)

$\sigma_{b+t/c,90}$	$= \begin{cases} \dfrac{\sigma_{t/c,90}}{f_{t,90,d}} + \dfrac{\lvert\sigma_{b,90}\rvert}{f_{b,90,d}} & \text{if } \sigma_{t/c,90} > 0 \\ \dfrac{\lvert\sigma_{t/c,90}\rvert}{f_{c,90,d}} + \dfrac{\lvert\sigma_{b,90}\rvert}{f_{b,90,d}} & \text{if } \sigma_{t/c,90} \leq 0 \end{cases}$	
$\tau_{x'y'}$	$\dfrac{\lvert\tau_{x'y'}\rvert}{f_{xy,d}}$	
$\tau_{y'z'}$	$\dfrac{\lvert\tau_{y'z'}\rvert}{f_{R,d}}$	
$\text{int}\left(\tau_{x'z'} + \tau_{x'y'}\right)$	$\dfrac{\tau^2_{x'z'}}{f^2_{v,d}} + \dfrac{\tau^2_{x'y'}}{f^2_{xy,d}}$	According to: ČSN 73 1702, (129) DIN 1052, (129) DIN EN 1995-1-1/NA, (NA.143)
$\text{int}\left(\sigma_{t/c,90} + \tau_{y'z'}\right)$	$= \begin{cases} \dfrac{\sigma_{t/c,90}}{f_{t,90,d}} + \dfrac{\lvert\tau_{y'z'}\rvert}{f_{R,d}} & \text{if } \sigma_{t/c,90} > 0 \\ \dfrac{\lvert\sigma_{t/c,90}\rvert}{f_{c,90,d}} + \dfrac{\lvert\tau_{y'z'}\rvert}{f_{R,d}} & \text{if } \sigma_{t/c,90} \leq 0 \end{cases}$	According to: ČSN 73 1702, (130), (131) DIN 1052, (130), (131) DIN EN 1995-1-1/NA, (NA.144), (NA.145)

Table 5.3: Ratios for orthotropic material model



*1 – Direction of grain

The stresses $\sigma_{b+t/c,0}$, $\sigma_{b+t/c,90}$, τ_d, and τ_R are defined in the coordinate system of the grain x', y', z. They are determined according the transformation formulas

$$\begin{bmatrix} \sigma_{b+t/c,0} \\ \sigma_{b+t/c,90} \\ * \end{bmatrix} = \underbrace{\begin{bmatrix} c^2 & s^2 & 2\,cs \\ s^2 & c^2 & -2\,cs \\ -cs & cs & c^2 - s^2 \end{bmatrix}}_{T^{-T}_{3\times3}} \begin{bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{bmatrix}, \quad \begin{bmatrix} \tau_d \\ \tau_R \end{bmatrix} = \underbrace{\begin{bmatrix} c & s \\ -s & c \end{bmatrix}}_{T_{2\times2}} \begin{bmatrix} \tau_{xz} \\ \tau_{yz} \end{bmatrix} \tag{5.1}$$

or, equivalently, in the non-matrix form

$$\begin{aligned} \sigma_{b+t/c,0} &= c^2\sigma_x + s^2\sigma_y + 2\,cs\tau_{xy} \\ \sigma_{b+t/c,90} &= s^2\sigma_x + c^2\sigma_y - 2\,cs\tau_{xy} \\ \tau_d &= c\tau_{xz} + s\tau_{yz} \\ \tau_R &= -s\tau_{xz} + c\tau_{yz} \end{aligned} \tag{5.2}$$

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 that are to be included in the printout report of RF-LAMINATE (see Chapter 6.2.2, page 60).

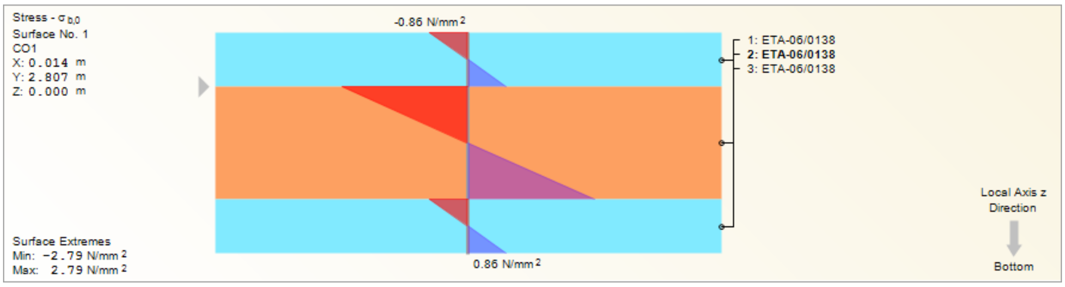


Figure 5.2: Stress diagram

5.2 Max Stress Ratio by Surface

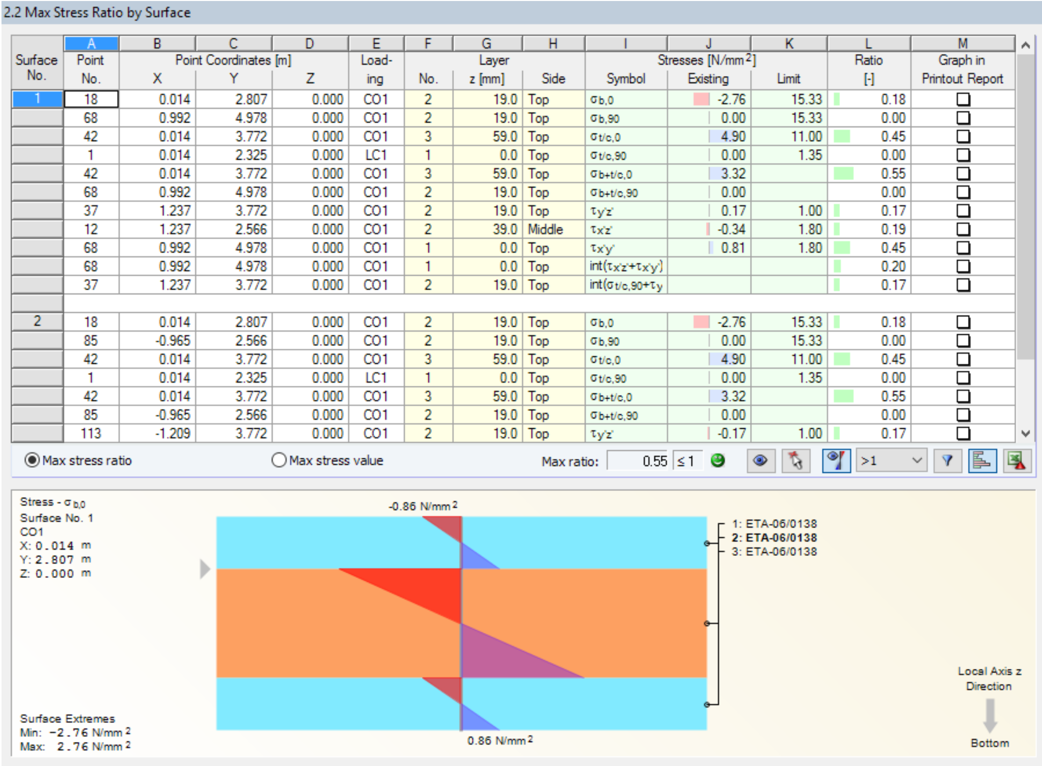


Figure 5.3: Window *2.2 Max Stress Ratio by Surface*

Max stress ratio   Max stress value

This result window contains the maximum stress ratios (or maximum stress values) of every designed surface. The columns of this table are described in the previous Chapter 5.1.

5.3 Max Stress Ratio by Composition

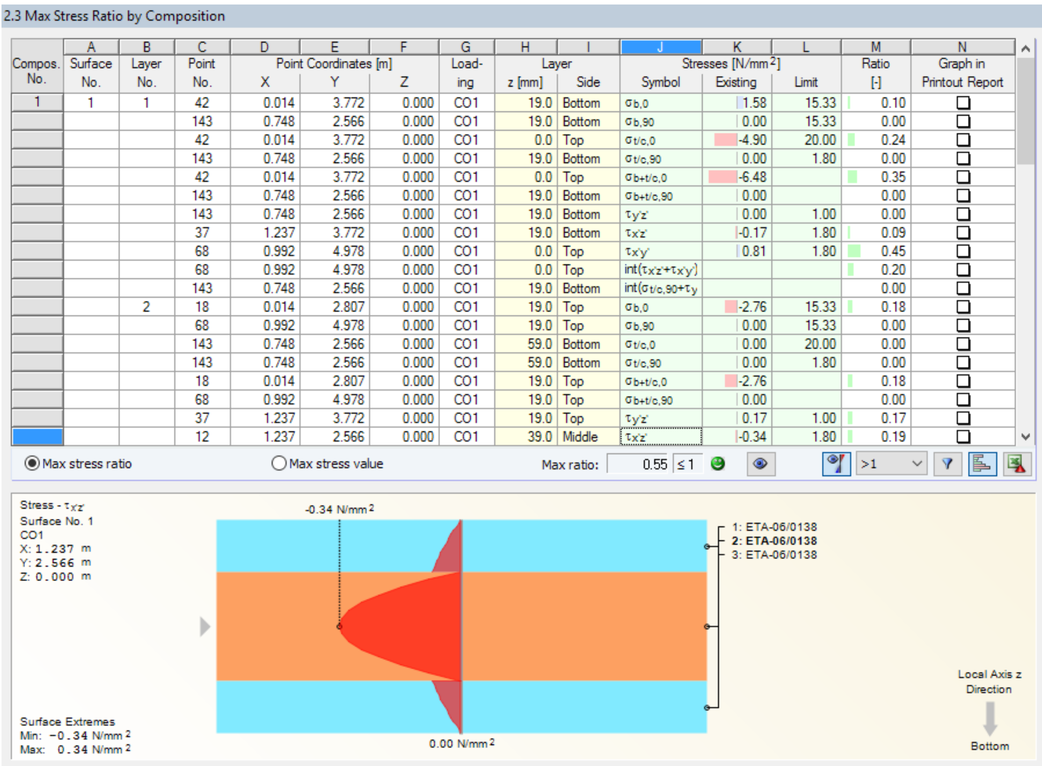
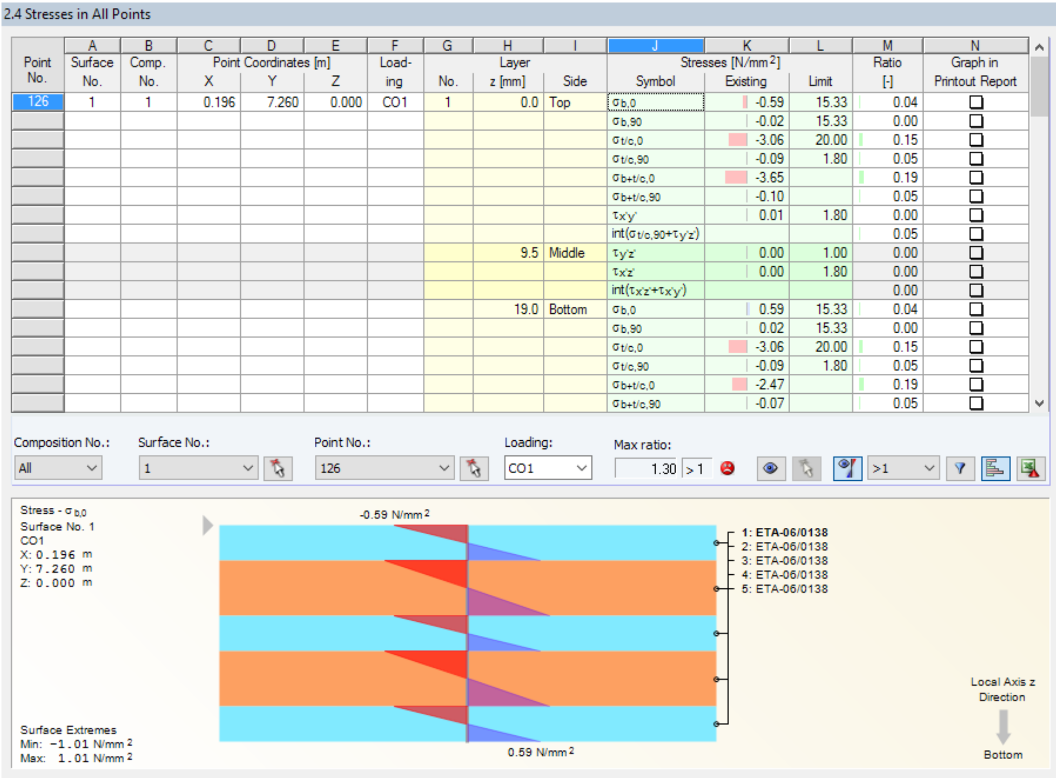


Figure 5.4: Window *2.3 Max Stress Ration by Composition*

In this window, the maximum stress ratios (or maximum stress values) are listed for every layer of each composition. The columns are described in Chapter 5.1.

5.4 Stresses in All Points



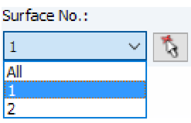
	A	B	C	D	E	F	G	H	I	J	K	L	M	N
Point No.	Surface No.	Comp. No.	Point Coordinates [m] X	Y	Z	Load-ing	Layer No.	z [mm]	Side	Stresses [N/mm²] Symbol	Existing	Limit	Ratio [-]	Graph in Printout Report
126	1	1	0.196	7.260	0.000	CO1	1	0.0	Top	$\sigma_{b,0}$	-0.59	15.33	0.04	☐
										$\sigma_{b,90}$	-0.02	15.33	0.00	☐
										$\sigma_{t/c,0}$	-3.06	20.00	0.15	☐
										$\sigma_{t/c,90}$	-0.09	1.80	0.05	☐
										$\sigma_{b+t/c,0}$	-3.65		0.19	☐
										$\sigma_{b+t/c,90}$	-0.10		0.05	☐
										$\tau_{x'y'}$	0.01	1.80	0.00	☐
										int($\sigma_{t/c,90}+\tau_{y'z'}$)			0.05	☐
								9.5	Middle	$\tau_{y'z'}$	0.00	1.00	0.00	☐
										$\tau_{x'z'}$	0.00	1.80	0.00	☐
										int($\tau_{x'z'}+\tau_{x'y'}$)			0.00	☐
								19.0	Bottom	$\sigma_{b,0}$	0.59	15.33	0.04	☐
										$\sigma_{b,90}$	0.02	15.33	0.00	☐
										$\sigma_{t/c,0}$	-3.06	20.00	0.15	☐
										$\sigma_{t/c,90}$	-0.09	1.80	0.05	☐
										$\sigma_{b+t/c,0}$	-2.47		0.19	☐
										$\sigma_{b+t/c,90}$	-0.07		0.05	☐

Figure 5.5: Window *2.4 Stresses in All Points*

In this window, the results can be evaluated for every FE mesh point or grid point of the designed layers. You can change the reference in the *Details* dialog box, tab *Results* (see Chapter 4.1.2, page 44).

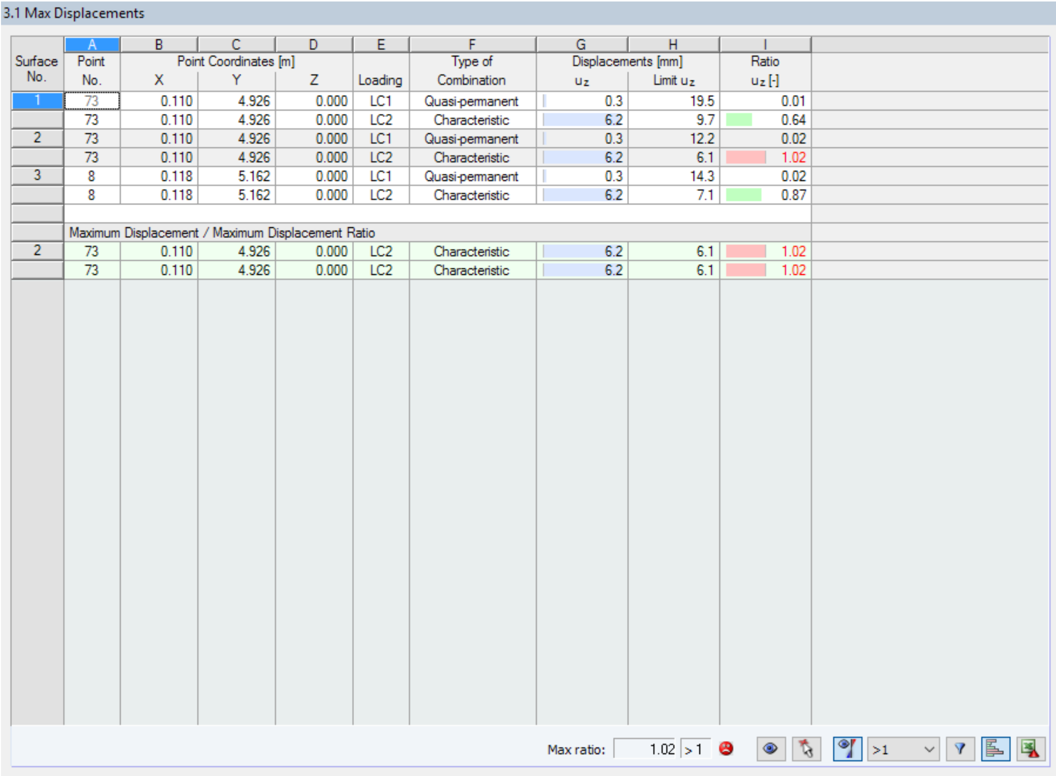
To reduce the number of results, you can select the stress components in the *Details* dialog box, tab *Stresses*, too.

The columns of this table are described in Chapter 5.1.



You can filter the data according to compositions, surfaces, points, and loadings. This selection is possible either from the lists below the table or by choosing the relevant point or surface in the work window via the button.

5.5 Max Displacements


3.1 Max Displacements

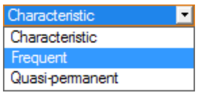
Surface No.	A Point No.	B Point Coordinates [m] X	C Y	D Z	E Loading	F Type of Combination	G Displacements [mm] u_z	H Limit u_z	I Ratio u_z [-]
1	73	0.110	4.926	0.000	LC1	Quasi-permanent	0.3	19.5	0.01
	73	0.110	4.926	0.000	LC2	Characteristic	6.2	9.7	0.64
2	73	0.110	4.926	0.000	LC1	Quasi-permanent	0.3	12.2	0.02
	73	0.110	4.926	0.000	LC2	Characteristic	6.2	6.1	1.02
3	8	0.118	5.162	0.000	LC1	Quasi-permanent	0.3	14.3	0.02
	8	0.118	5.162	0.000	LC2	Characteristic	6.2	7.1	0.87
	Maximum Displacement / Maximum Displacement Ratio								
2	73	0.110	4.926	0.000	LC2	Characteristic	6.2	6.1	1.02
	73	0.110	4.926	0.000	LC2	Characteristic	6.2	6.1	1.02

Figure 5.6: Window *3.1 Max Displacements*

This window is displayed when you have selected at least one load case or combination for the design in Window *1.1 General Data*, tab *Serviceability Limit State* (see Chapter 3.1.2, page 23). In the table, the maximum deflections are shown for every load case, load and result combination that was selected for the SLS design.

The results are listed by surface numbers.

Type of Combination



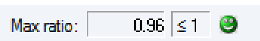
In this column, the design situations are shown that were defined for the relevant load cases and combinations (see Chapter 3.1.2, page 23).

Displacements

In the u_z column, the governing displacements are listed which occur in the direction of the local *z*-axes of the surfaces. Those axes are perpedicular to the plane of the surface.

The values of the *Limit* u_z column represent the maximum allowable deflections. Those values are determined from the reference lengths as defined in the *1.6 Serviceability Data* Window (see Chapter 3.6, page 34) and from the general limits as specified in the *Standard* dialog box, tab *Serviceability Limits* (see Chapter 4.2.2, page 47).

Ratio



In the last column, the ratios of the resulting displacement u_z (column G) and the limit displacement (column H) are shown. If no limits of the deformation are exceeded, the ratio is less than or equal to 1, and the deflection design is satisfied.

5.6 Parts List

4.1 Parts List

Surface No.	A Material Description	B Thickness t [mm]	C No. of Layers	D Area [m²]	E Coating [m²]	F Volume [m³]	G Weight [t]
1	ETA-06/0138	19.0	3	6.942	13.883	0.396	0.190
	ETA-06/0138	30.0	2	6.942	0.000	0.417	0.200
Σ		117.0	5	6.942	13.883	0.812	0.390
2	ETA-06/0138	19.0	3	3.471	6.942	0.198	0.095
	ETA-06/0138	30.0	2	3.471	0.000	0.208	0.100
Σ		117.0	5	3.471	6.942	0.406	0.195
3	ETA-06/0138	19.0	3	3.471	6.942	0.198	0.095
	ETA-06/0138	30.0	2	3.471	0.000	0.208	0.100
Σ		117.0	5	3.471	6.942	0.406	0.195
Σ Total				13.883	27.767	1.624	0.780

Figure 5.7: Window *4.1 Parts List*

The last result window gives an overall review of the surfaces. The data refers only to the designed surfaces by default. If you want to display the parts list of all surfaces contained in the model, change the setting in the *Details* dialog box, tab *Results* (see Chapter 4.1.2, page 44).

Surface No.

The parts list is sorted by surface numbers.

Material Description

In this column, the materials of the surfaces are specified.

Thickness t

The thicknesses of the layers which are listed in this column can be also checked in the *1.2 Material Characteristics* Window. Layers with identical thicknesses are summarized.

No. of Layers

This column specifies how many layers of the same material and thickness exist for each surface.

Area

For every surface, information about the surface area of the layers is given.

Coating

The surface coating is calculated from the upper and lower surface areas. The sides of the rather thin-walled 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. Those values are based on the volumes of the surfaces and the specific weight of each material.

Total

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

6 Printout

6.1 Printout Report

Like in RFEM, a printout report is created for the RF-LAMINATE data to which you can add graphics and comments. In the printout report, you can also select which input data and results of the module are to be included in the printout.

The printout report is described in the RFEM manual. In particular, Chapter 10.1.3.5 *Selecting Data of Add-on Modules* describes how input and output data from add-on modules can be arranged for the printout report.

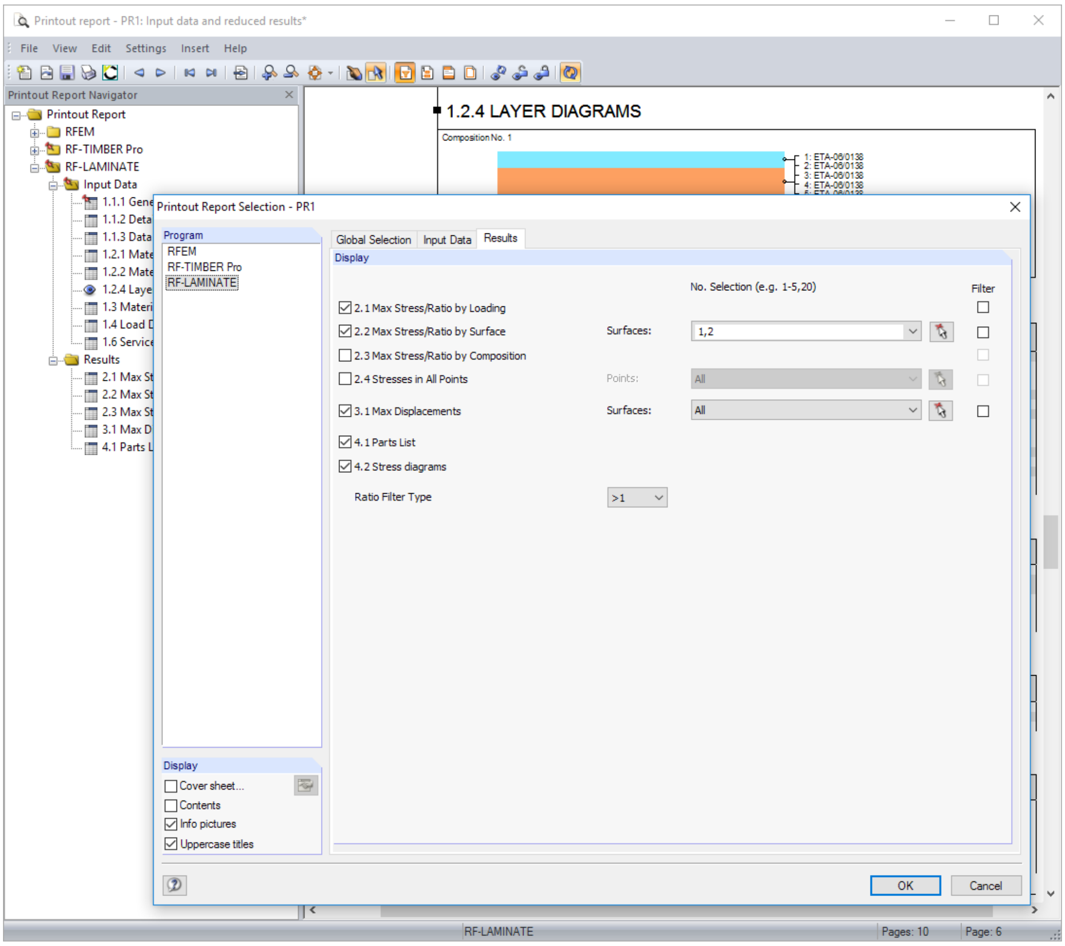


Figure 6.1: Selecting topics of RF-LAMINATE in printout report

You can create several printout reports for each model. Especially for complex structural systems, it is recommended to split the data into several printout reports. When you create a printout report only for the RF-LAMINATE data, for example, the data is processed much faster.

The printout report only includes the types of stresses that were selected for the display in the result windows. If you want to print the rolling shear stresses, for example, you have to activate the stresses $\tau_{y'z'}$ (τ_R) for the display in the *Details* dialog box. Chapter 4.1.1 on page 37 describes how those stresses can be selected.

6.2 Graphic Printout

6.2.1 Results on RFEM Model

In RFEM, you can add every view of the work window to the printout report or send it directly to a printer. In this way, you can prepare the stresses displayed in the RFEM model for the printout.

Printing graphics is described in the RFEM manual, Chapter 10.2.

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

File → Print Graphic

or by clicking the respective button in the toolbar.

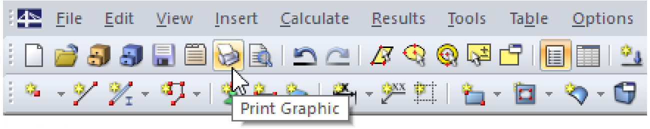


Figure 6.2: *Print Graphic* button in RFEM toolbar

The same button enables you to print the result diagrams of sections.

The following dialog box appears.

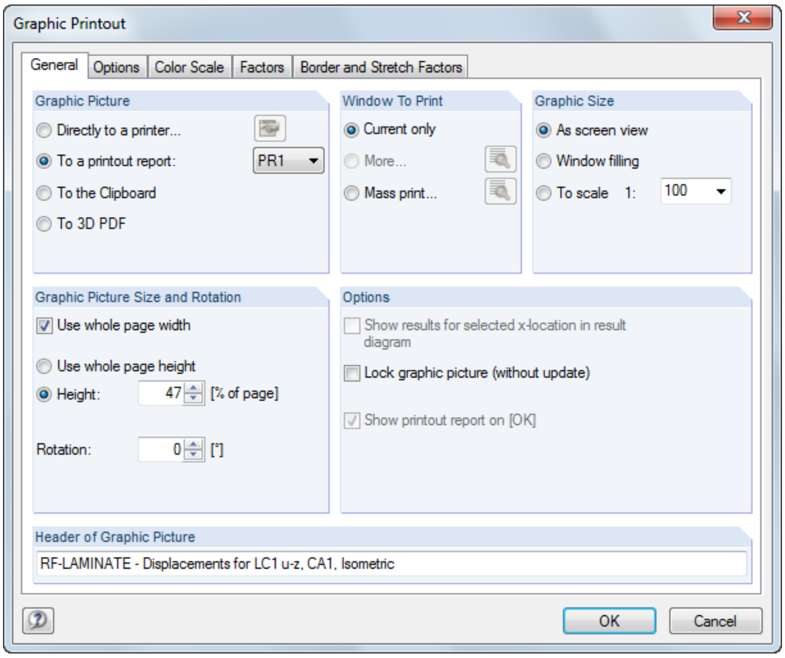
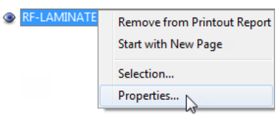


Figure 6.3: Dialog box *Graphic Printout*, tab *General*

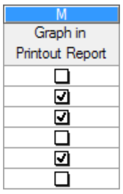
The *Graphic Printout* dialog box is described in detail in the RFEM manual, Chapter 10.2.

In the printout report, you can move the graphics to a different location by drag and drop.



Inserted images can be modified subsequently: right-click the item in the printout report navigator and select the *Properties* option in the shortcut menu. The *Graphic Printout* dialog box is displayed again so that you can change the settings.

6.2.2 Stress Diagrams



The Windows 2.1, 2.2 and 2.3 of RF-LAMINATE enable you to incorporate stress diagrams to the printout report. Select the relevant image(s) in the *Graph in Printout Report* column as seen on the left. According to the settings in Figure 6.4, the stress diagrams $\sigma_{b,0}$ in point 3 (surface 1) and point 4 (surface 2) are to be printed.

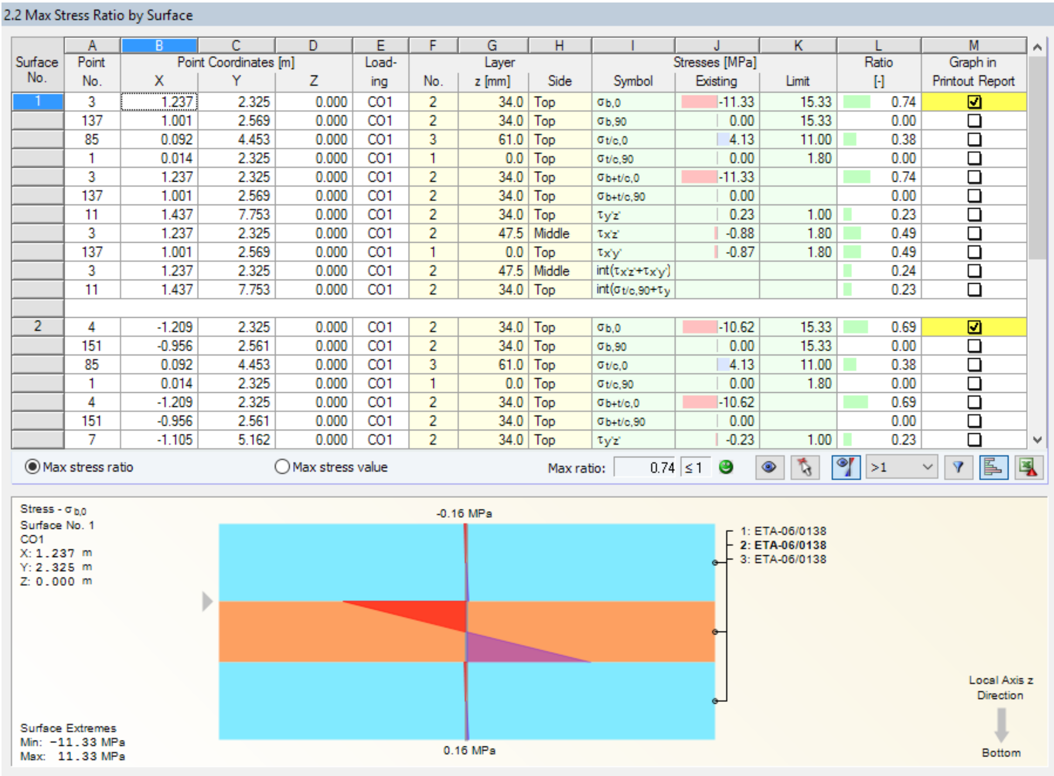


Figure 6.4: Window *2.2 Max Stress Ratio by Surface*

When you close the module with [OK] and open the printout report, the selected pictures are displayed in Chapter *4.2 Stress Diagrams*.

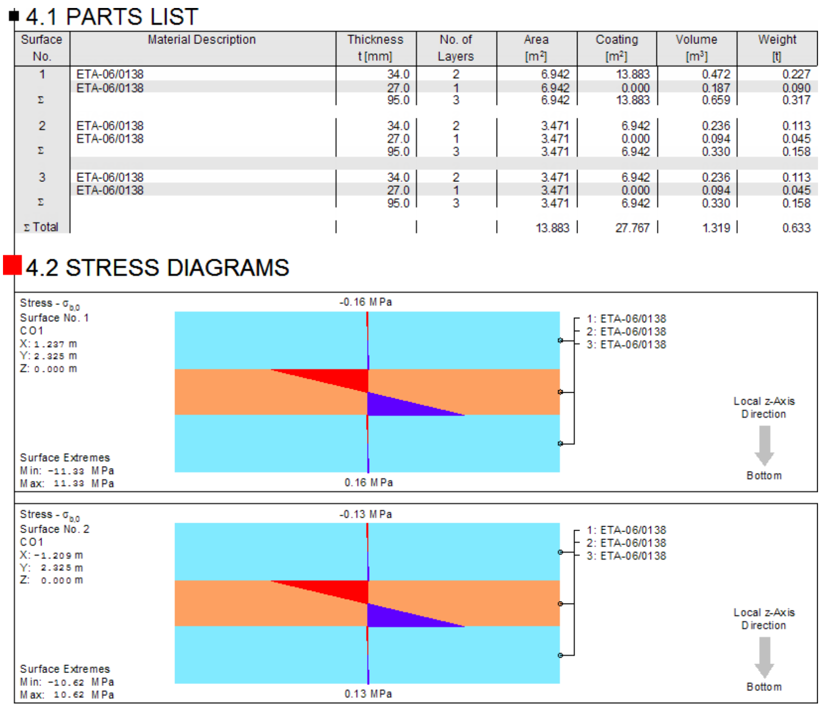


Figure 6.5: Stress diagrams in printout report

7 General Functions

This chapter describes the menu functions and export options for design results.

7.1 Units and Decimal Places

Units and decimal places for RFEM and all its add-on modules are managed in one dialog box. In RF-LAMINATE, you can open this dialog box from the main menu

Settings → **Units and Decimal Places**.

The dialog box is already familiar from RFEM. RF-LAMINATE is preset in the *Program / Module* list.

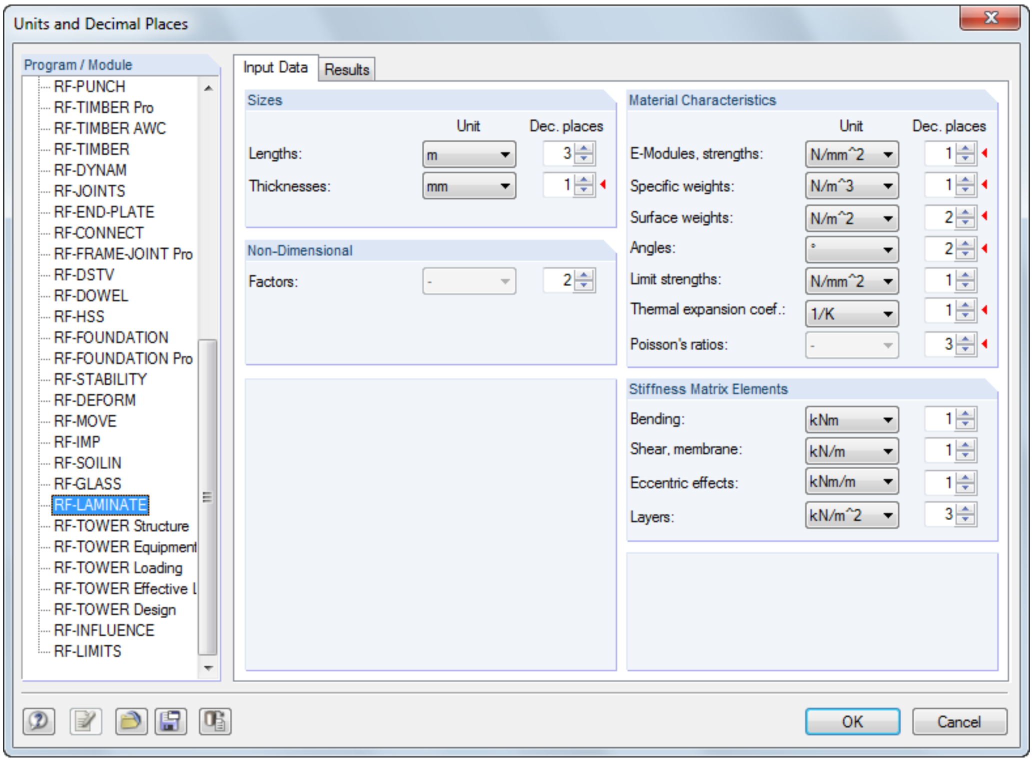


Figure 7.1: Dialog box *Units and Decimal Places*

In Figure 7.1, you can see that some units are marked with a red arrow, such as the thicknesses and material characteristics. 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.

You can save the settings as a user-defined profile to reuse them in other models. The functions are described in Chapter 11.1.3 of the RFEM manual.

7.2 Exporting Results

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

Clipboard

To copy selected cells of a result window to the Clipboard, use the [Ctrl]+[C] keys. Press [Ctrl]+[V] to insert the cells in a word processing program, for example. The headers of the table columns will not be transferred.

Printout report

Print the data of RF-LAMINATE to the global printout report (see Chapter 6.1, page 58). Then export the printout report by using the main menu

File → Export to RTF.

This function is described in Chapter 10.1.11 of the RFEM manual.

Excel / OpenOffice

RF-LAMINATE provides a function to directly export data to MS Excel, OpenOffice Calc, or the CSV file format. To open the corresponding dialog box, click

File → Export Tables

or use the corresponding button.

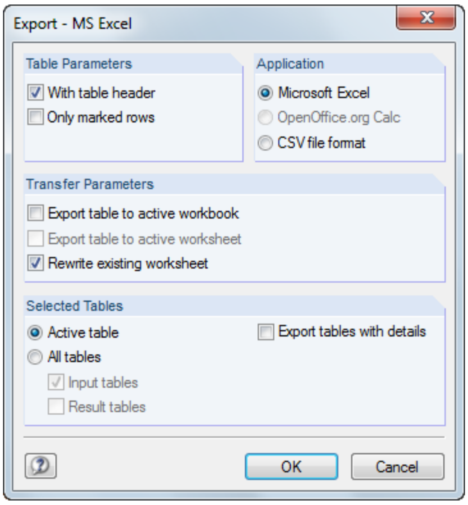


Figure 7.2: Dialog box *Export - MS Excel*

When you have selected the relevant parameters, start the export by clicking the [OK] button. Excel or OpenOffice need not run in the background; they will be started automatically.

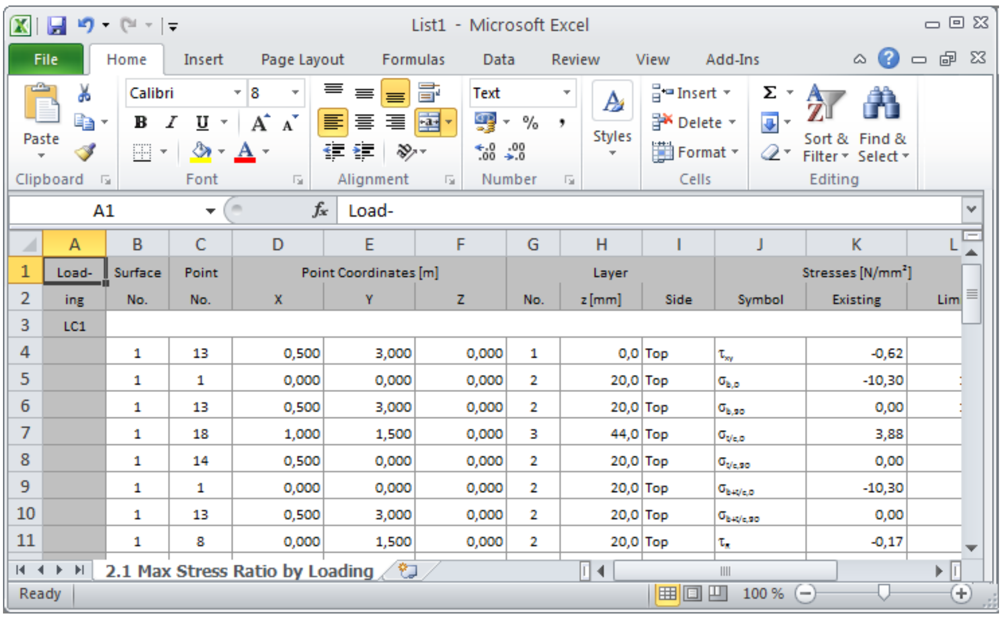


Figure 7.3: Results in MS Excel – Worksheet *2.1 Max Stress Ratio by Loading*

8 Examples

In this chapter, several examples are introduced.

8.1 Calculation of Stiffness Matrix Elements

The stiffness matrix elements of a three-layer plate is to be determined. The layers are as follows:

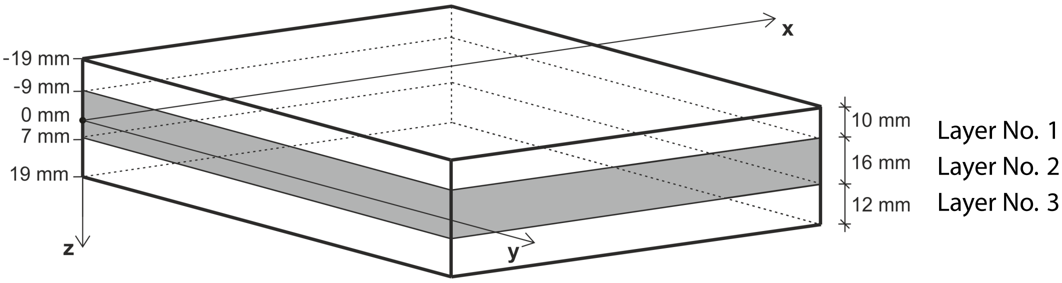


Figure 8.1: Layer scheme

The material characteristics of the layers are shown in Figure 8.2.

Layers

	A	B	C	D	E	F	G	H	I	J	K
Layer No.	Material Description	Thickness t [mm]	Orthotropic Direction β [°]	Modulus of Elasticity [N/mm²] Ex	Ey	Shear Modulus [N/mm²] Gxz	Gyz	Gxy	Poisson's Ratio [-] νxy	νyx	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: Material characteristics

At first, the stiffness matrices of the individual layers are calculated.

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

$$\boldsymbol{d}_1' = \begin{bmatrix} \dfrac{8{,}000}{1-0.2^2\frac{270}{8{,}000}} & \dfrac{0.2\cdot 270}{1-0.2^2\frac{270}{8{,}000}} & 0 \\ & \dfrac{270}{1-0.2^2\frac{270}{8{,}000}} & 0 \\ \text{sym.} & & 500 \end{bmatrix} = \begin{bmatrix} 8{,}010.81 & 54.07 & 0 \\ 54.07 & 270.36 & 0 \\ 0 & 0 & 500.00 \end{bmatrix} \text{MN/m}^2$$

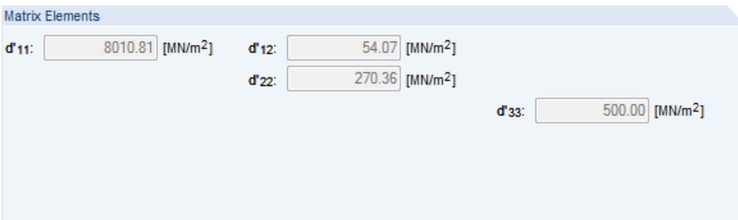


Figure 8.3: *Matrix Elements* of layer No. 1

$$\boldsymbol{d}_2' = \begin{bmatrix} \dfrac{7{,}000}{1-0.2^2\dfrac{230}{7{,}000}} & \dfrac{0.2\cdot 230}{1-0.2^2\dfrac{270}{7{,}000}} & 0 \\ & \dfrac{270}{1-0.2^2\dfrac{230}{7{,}000}} & 0 \\ \text{sym.} & & 440 \end{bmatrix} = \begin{bmatrix} 7{,}009.21 & 46.06 & 0 \\ 46.06 & 230.30 & 0 \\ 0 & 0 & 440.00 \end{bmatrix} \mathrm{MN/m^2}$$

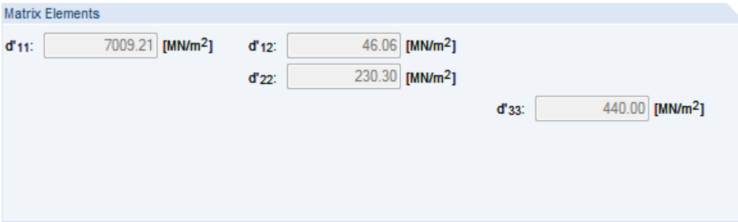


Figure 8.4: *Matrix Elements* of layer No. 2

$$\boldsymbol{d}_3' = \begin{bmatrix} \dfrac{8{,}000}{1-0.2^2\dfrac{270}{8{,}000}} & \dfrac{0.2\cdot 270}{1-0.2^2\dfrac{270}{8{,}000}} & 0 \\ & \dfrac{270}{1-0.2^2\dfrac{270}{8{,}000}} & 0 \\ \text{sym.} & & 500 \end{bmatrix} = \begin{bmatrix} 8{,}010.81 & 54.07 & 0 \\ 54.07 & 270.36 & 0 \\ 0 & 0 & 500.00 \end{bmatrix} \mathrm{MN/m^2}$$

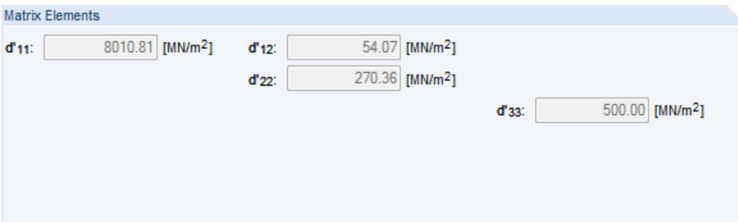
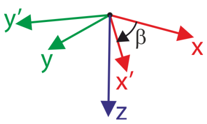


Figure 8.5: *Matrix Elements* of layer No. 3



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

$$\boldsymbol{d}_1 = \boldsymbol{d}_1' = \begin{bmatrix} 8{,}010.81 & 54.07 & 0 \\ 54.07 & 270.36 & 0 \\ 0 & 0 & 500.00 \end{bmatrix} \mathrm{MN/m^2}$$

$$\boldsymbol{d}_3 = \boldsymbol{d}_3' = \begin{bmatrix} 8{,}010.81 & 54.07 & 0 \\ 54.07 & 270.36 & 0 \\ 0 & 0 & 500.00 \end{bmatrix} \mathrm{MN/m^2}$$

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.

$$\boldsymbol{d}_i = \begin{bmatrix} \mathrm{d}_{11,i} & \mathrm{d}_{12,i} & \mathrm{d}_{13,i} \\ & \mathrm{d}_{22,i} & \mathrm{d}_{23,i} \\ \text{sym.} & & \mathrm{d}_{33,i} \end{bmatrix} = \boldsymbol{T}^T_{3\times 3,i}\,\boldsymbol{d}_i'\,\boldsymbol{T}_{3\times 3,i} \tag{8.2}$$

where

$$\boldsymbol{T}_{3\times 3,i} = \begin{bmatrix} \mathrm{c}^2 & \mathrm{s}^2 & \mathrm{cs} \\ \mathrm{s}^2 & \mathrm{c}^2 & -\mathrm{cs} \\ -2\mathrm{cs} & 2\mathrm{cs} & \mathrm{c}^2-\mathrm{s}^2 \end{bmatrix}, \quad \text{where} \quad \mathrm{c} = \cos(\beta_\mathrm{i}),\, \mathrm{s} = \sin(\beta_\mathrm{i}) \tag{8.3}$$

The individual elements then are

$$d_{11,i} = c^4 d'_{11,i} + 2c^2 s^2 d'_{12,i} + s^4 d'_{22,i} + 4c^2 s^2 d'_{33,i}$$

$$d_{12,i} = c^2 s^2 d'_{11,i} + s^4 d'_{12,i} + c^4 d'_{12,i} + c^2 s^2 d'_{22,i} - 4c^2 s^2 d'_{33,i}$$

$$d_{13,i} = c^3 s d'_{11,i} + c s^3 d'_{12,i} - c^3 s d'_{12,i} - c s^3 d'_{22,i} - 2c^3 s d'_{33,i} + 2c s^3 d'_{33,i}$$

$$d_{22,i} = s^4 d'_{11,i} + 2c^2 s^2 d'_{12,i} + c^4 d'_{22,i} + 4c^2 s^2 d'_{33,i}$$

$$d_{23,i} = c s^3 d'_{11,i} + c^3 s d'_{12,i} - c s^3 d'_{12,i} - c^3 s d'_{22,i} + 2c^3 s d'_{33,i} - 2c s^3 d'_{33,i}$$

$$d_{33,i} = c^2 s^2 d'_{11,i} - 2c^2 s^2 d'_{12,i} + c^2 s^2 d'_{22,i} + \left(c^2 - s^2\right)^2 d'_{33,i}$$

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

$$d_{11,2} = 0^4 \cdot 7{,}009.21 + 2 \cdot 0^2 \cdot 1^2 \cdot 46.06 + 1^4 \cdot 230.30 + 4 \cdot 0^2 \cdot 1^2 \cdot 440 = 230.30 \text{ MN/m}^2$$

$$d_{12,2} = 0^2 \cdot 1^2 \cdot 7{,}009.21 + 1^4 \cdot 46.06 + 0^4 \cdot 46.06 + 0^2 \cdot 1^2 \cdot 230.30 - 4 \cdot 0^2 \cdot 1^2 \cdot 440 = 46.06 \text{ MN/m}^2$$

$$d_{13,2} = 0^3 \cdot 1 \cdot 7{,}009.21 + 0 \cdot 1^3 \cdot 46.06 - 0^3 \cdot 1 \cdot 46.06 - 0 \cdot 1^3 \cdot 230.30 - 2 \cdot 0^3 \cdot 1 \cdot 440 + 2 \cdot 0 \cdot 1^3 \cdot 440 = 0 \text{MN/m}^2$$

$$d_{22,2} = 1^4 \cdot 7{,}009.21 + 2 \cdot 0^2 \cdot 1^2 \cdot 46.06 + 0^4 \cdot 230.30 + 4 \cdot 0^2 \cdot 1^2 \cdot 440 = 7{,}009.21 \text{ MN/m}^2$$

$$d_{23,2} = 0 \cdot 1^3 \cdot 7{,}009.21 + 0^3 \cdot 1 \cdot 46.06 - 0 \cdot 1^3 \cdot 46.06 - 0^3 \cdot 1 \cdot 230.30 + 2 \cdot 0^3 \cdot 1 \cdot 440 - 2 \cdot 0 \cdot 1^3 \cdot 440 = 0 \text{MN/m}^2$$

$$d_{33,2} = 0^2 \cdot 1^2 \cdot 7{,}009.21 - 2 \cdot 0^2 \cdot 1^2 46.06 + 0^2 \cdot 1^2 230.30 + \left(0^2 - 1^2\right)^2 \cdot 440 = 440.00 \text{ MN/m}^2$$

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

$$\boldsymbol{d_2} = \begin{bmatrix} 230.30 & 46.06 & 0 \\ 46.06 & 7{,}009.21 & 0 \\ 0 & 0 & 440.00 \end{bmatrix} \text{MN/m}^2$$

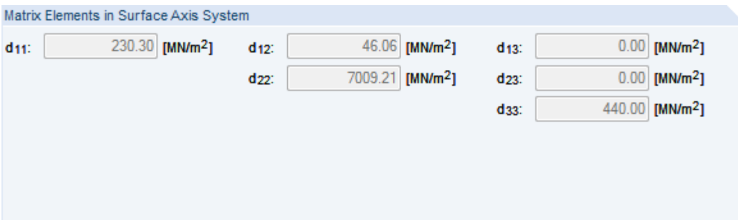


Figure 8.6: *Matrix Elements in Surface Axis System* of layer No. 2

8.1.1 With Shear Coupling of Layers

When the shear coupling of the layers is considered, the global stiffness matrix has the form

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

Stiffness matrix elements – bending and torsion

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

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

$$D_{33} = \sum_{i=1}^{n} \frac{z^3_{max,i} - z^3_{min,i}}{3} d_{33,i}$$

$$D_{11} = \frac{(-9 \cdot 10^{-3})^3 - (-19 \cdot 10^{-3})^3}{3} 8{,}010.81 \cdot 10^3 + \frac{(7 \cdot 10^{-3})^3 - (-9 \cdot 10^{-3})^3}{3} 230.30 \cdot 10^3 + \\ + \frac{(19 \cdot 10^{-3})^3 - (7 \cdot 10^{-3})^3}{3} 8{,}010.81 \cdot 10^3 = 33.85 \text{ kNm}$$

$$D_{12} = \frac{(-9 \cdot 10^{-3})^3 - (-19 \cdot 10^{-3})^3}{3} 54.07 \cdot 10^3 + \frac{(7 \cdot 10^{-3})^3 - (-9 \cdot 10^{-3})^3}{3} 46.06 \cdot 10^3 + \\ + \frac{(19 \cdot 10^{-3})^3 - (7 \cdot 10^{-3})^3}{3} 54.07 \cdot 10^3 = 0.24 \text{ kNm}$$

$$D_{13} = \frac{(-9 \cdot 10^{-3})^3 - (-19 \cdot 10^{-3})^3}{3} 0 \cdot 10^3 + \frac{(7 \cdot 10^{-3})^3 - (-9 \cdot 10^{-3})^3}{3} 0 \cdot 10^3 + \\ + \frac{(19 \cdot 10^{-3})^3 - (7 \cdot 10^{-3})^3}{3} 0 \cdot 10^3 = 0 \text{ kNm}$$

$$D_{22} = \frac{(-9 \cdot 10^{-3})^3 - (-19 \cdot 10^{-3})^3}{3} 270.36 \cdot 10^3 + \frac{(7 \cdot 10^{-3})^3 - (-9 \cdot 10^{-3})^3}{3} 7{,}009.21 \cdot 10^3 + \\ + \frac{(19 \cdot 10^{-3})^3 - (7 \cdot 10^{-3})^3}{3} 270.36 \cdot 10^3 = 3.64 \text{ kNm}$$

$$D_{23} = \frac{(-9 \cdot 10^{-3})^3 - (-19 \cdot 10^{-3})^3}{3} 0 \cdot 10^3 + \frac{(7 \cdot 10^{-3})^3 - (-9 \cdot 10^{-3})^3}{3} 0 \cdot 10^3 + \\ + \frac{(19 \cdot 10^{-3})^3 - (7 \cdot 10^{-3})^3}{3} 0 \cdot 10^3 = 0 \text{ kNm}$$

$$D_{33} = \frac{(-9 \cdot 10^{-3})^3 - (-19 \cdot 10^{-3})^3}{3} 500 \cdot 10^3 + \frac{(7 \cdot 10^{-3})^3 - (-9 \cdot 10^{-3})^3}{3} 440 \cdot 10^3 + \\ + \frac{(19 \cdot 10^{-3})^3 - (7 \cdot 10^{-3})^3}{3} 500 \cdot 10^3 = 2.26 \text{ kNm}$$

Stiffness matrix elements – eccentricity effects

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

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

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

$$D_{16}=\frac{(-9\cdot 10^{-3})^2-(-19\cdot 10^{-3})^2}{2}8{,}010.81\cdot 10^3+\frac{(7\cdot 10^{-3})^2-(-9\cdot 10^{-3})^2}{2}230.30\cdot 10^3+$$
$$+\frac{(19\cdot 10^{-3})^2-(7\cdot 10^{-3})^2}{2}8{,}010.81\cdot 10^3=124.49\text{ kNm/m}$$

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

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

$$D_{27}=\frac{(-9\cdot 10^{-3})^2-(-19\cdot 10^{-3})^2}{2}270.36\cdot 10^3+\frac{(7\cdot 10^{-3})^2-(-9\cdot 10^{-3})^2}{2}7{,}009.21\cdot 10^3+$$
$$+\frac{(19\cdot 10^{-3})^2-(7\cdot 10^{-3})^2}{2}270.36\cdot 10^3=-107.82\text{ kNm/m}$$

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

$$D_{38}=\frac{(-9\cdot 10^{-3})^2-(-19\cdot 10^{-3})^2}{2}500\cdot 10^3+\frac{(7\cdot 10^{-3})^2-(-9\cdot 10^{-3})^2}{2}440\cdot 10^3+$$
$$+\frac{(19\cdot 10^{-3})^2-(7\cdot 10^{-3})^2}{2}500\cdot 10^3=0.96\text{ kNm/m}$$

Stiffness matrix elements – membrane

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

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

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

$$D_{66}=10\cdot 10^{-3}\cdot 8{,}010.81\cdot 10^3+16\cdot 10^{-3}\cdot 230.30\cdot 10^3+12\cdot 10^{-3}\cdot 8{,}010.81\cdot 10^3=179{,}923\text{ N/m}$$

$$D_{67}=10\cdot 10^{-3}\cdot 54.07\cdot 10^3+16\cdot 10^{-3}\cdot 46.06\cdot 10^3+12\cdot 10^{-3}\cdot 54.07\cdot 10^3=1{,}927\text{ N/m}$$

$$D_{68}=10\cdot 10^{-3}\cdot 0\cdot 10^3+16\cdot 10^{-3}\cdot 46.06\cdot 10^3+12\cdot 10^{-3}\cdot 0\cdot 10^3=0\text{ N/m}$$

$$D_{77}=10\cdot 10^{-3}\cdot 270.36\cdot 10^3+16\cdot 10^{-3}\cdot 7{,}009.21\cdot 10^3+12\cdot 10^{-3}\cdot 270.36\cdot 10^3=118{,}095\text{ N/m}$$

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

$$D_{88}=10\cdot 10^{-3}\cdot 500\cdot 10^3+16\cdot 10^{-3}\cdot 440\cdot 10^3+12\cdot 10^{-3}\cdot 500\cdot 10^3=18{,}040\text{ N/m}$$

Stiffness matrix elements – shear

1. The angle $\varphi = 0\,°$ defines the coordinate system x'', y'' with the maximum stiffness.
2. The shear stiffnesses $G''_{xz,i}$, $G''_{yz,i}$ for each layer in the coordinate system x'', y'' are defined by the following formula.

$$G''_{xz,i} = \cos^2(\varphi - \beta_i)\, G_{xz,i} + \sin^2(\varphi - \beta_i)\, G_{yz,i}$$

$$G''_{yz,i} = \sin^2(\varphi - \beta_i)\, G_{xz,i} + \cos^2(\varphi - \beta_i)\, G_{yz,i} \qquad i = 1,\ldots,n \tag{8.5}$$

$$G''_{xz,1} = G''_{xz,3} = \cos^2(0\,°)\,500 + \sin^2(0\,°)\,50 = 500\ \mathrm{MN/m^2}$$

$$G''_{yz,1} = G''_{yz,3} = \sin^2(0\,°)\,500 + \cos^2(0\,°)\,50 = 50\ \mathrm{MN/m^2}$$

$$G''_{xz,2} = \cos^2(-90\,°)\,440 + \sin^2(-90\,°)\,44 = 44\ \mathrm{MN/m^2}$$

$$G''_{yz,2} = \sin^2(-90\,°)\,440 + \cos^2(-90\,°)\,44 = 440\ \mathrm{MN/m^2}$$

3. The planar stiffness matrix $\boldsymbol{d}''_i$ is defined

$$\boldsymbol{d}''_i = \boldsymbol{T}^{-T}_{3\times3,i}\boldsymbol{d}'_i\boldsymbol{T}^{-1}_{3\times3,i} \tag{8.6}$$

where

$$\boldsymbol{T}_{3\times3,i} = \begin{bmatrix} c^2 & s^2 & cs \\ s^2 & c^2 & -cs \\ -2cs & 2cs & c^2 - s^2 \end{bmatrix},\ \text{where } c = \cos(\varphi - \beta_i),\ s = \sin(\varphi - \beta_i),\ i = 1,\ldots,n \tag{8.7}$$

$$\boldsymbol{T}_{3\times3,1} = \boldsymbol{T}_{3\times3,3} = \begin{bmatrix} 1 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & 1 \end{bmatrix}, \quad \boldsymbol{T}_{3\times3,2} = \begin{bmatrix} 0 & 1 & 0 \\ 1 & 0 & 0 \\ 0 & 0 & -1 \end{bmatrix}$$

$$\boldsymbol{d}''_1 = \boldsymbol{d}''_3 = \begin{bmatrix} 8{,}010.81 & 54.07 & 0 \\ 54.07 & 270.36 & 0 \\ 0 & 0 & 500.00 \end{bmatrix} \mathrm{MN/m^2}$$

$$\boldsymbol{d}''_2 = \begin{bmatrix} 230.30 & 46.06 & 0 \\ 46.06 & 7{,}009.21 & 0 \\ 0 & 0 & 440.00 \end{bmatrix} \mathrm{MN/m^2}$$

From the stiffness matrix $\boldsymbol{d}''_i$, Young's moduli $E''_{x,i}$, $E''_{y,i}$ are extracted

$$E''_{x,i} = d''_{11,i} + \frac{2d''_{12,i}d''_{13,i}d''_{23,i} - d''_{22,i}\left(d''_{13,i}\right)^2 - d''_{33,i}\left(d''_{12,i}\right)^2}{d''_{22,i}d''_{33,i} - \left(d''_{23,i}\right)^2} \tag{8.8}$$

$$E''_{y,i} = d''_{22,i} + \frac{2d''_{12,i}d''_{13,i}d''_{23,i} - d''_{22,i}\left(d''_{23,i}\right)^2 - d''_{33,i}\left(d''_{12,i}\right)^2}{d''_{11,i}d''_{33,i} - \left(d''_{13,i}\right)^2} \tag{8.9}$$

$$E''_{x,1} = E''_{x,3} = 8{,}010.81 + \frac{2\cdot 54.07\cdot 0\cdot 0 - 270.36\,(0)^2 - 500.00\,(54.07)^2}{270.36\cdot 500.00 - (0)^2} = 8{,}000.00\mathrm{MN/m^2}$$

$$E''_{x,2} = 230.30 + \frac{2\cdot 46.06\cdot 0\cdot 0 - 7{,}009.21\,(0)^2 - 440.00\,(46.06)^2}{7{,}009.21\cdot 440.00 - (0)^2} = 230.00\ \mathrm{MN/m^2}$$

$$E''_{y,1} = E''_{y,3} = 270.36 + \frac{2\cdot 54.07\cdot 0\cdot 0 - 8{,}010.81\,(0)^2 - 500.00\,(54.07)^2}{8{,}010.81\cdot 500.00 - (0)^2} = 270.00\mathrm{MN/m^2}$$

$$E''_{y,2} = 7{,}009.21 + \frac{2\cdot 46.06\cdot 0\cdot 0 - 230.30\,(0)^2 - 440.00\,(46.06)^2}{230.30\cdot 440.00 - (0)^2} = 7{,}000\ \mathrm{MN/m^2}$$

4. In the coordinate system x'', y'', the values $D''_{44,calc}$ and $D''_{55,calc}$ are defined as follows.

$$D''_{44,calc} = \frac{1}{\int\limits_{-t/2}^{t/2} \frac{1}{G''_{xz}(z)} \left(\frac{\int_z^{t/2} E''_x(\bar{z})(\bar{z}-z_{0,x})\, d\bar{z}}{\int_{-t/2}^{t/2} E''_x(\bar{z})(\bar{z}-z_{0,x})^2\, d\bar{z}} \right)^2 dz}, \quad z_{0,x} = \frac{\int_{-t/2}^{t/2} E''_x(\bar{z})\bar{z}d\bar{z}}{\int_{-t/2}^{t/2} E''_x(\bar{z})\, d\bar{z}} \tag{8.10}$$

$$D''_{44,calc} = 2{,}128.07 \text{ kN/m}$$

$$D''_{55,calc} = \frac{1}{\int\limits_{-t/2}^{t/2} \frac{1}{G''_{yz}(z)} \left(\frac{\int_z^{t/2} E''_y(\bar{z})(\bar{z}-z_{0,y})\, d\bar{z}}{\int_{-t/2}^{t/2} E''_y(\bar{z})(\bar{z}-z_{0,y})^2\, d\bar{z}} \right)^2 dz}, \quad z_{0,y} = \frac{\int_{-t/2}^{t/2} E''_y(\bar{z})\bar{z}d\bar{z}}{\int_{-t/2}^{t/2} E''_y(\bar{z})\, d\bar{z}} \tag{8.11}$$

$$D''_{55,calc} = 7{,}085.28 \text{ kN/m}$$

The values of the stiffnesses D_{44} and D_{55} are given by the following formulas.

$$D''_{44} = \max \left(D''_{44,calc}, \frac{48}{5\,\ell^2} \frac{1}{\frac{1}{\sum_{i=1}^n E''_{x,i} \frac{t_i^3}{12}} - \frac{1}{\sum_{i=1}^n E''_{x,i} \frac{z_{max,i}^3 - z_{min,i}^3}{3}}} \right) \tag{8.12}$$

$$D''_{55} = \max \left(D''_{55,calc}, \frac{48}{5\,\ell^2} \frac{1}{\frac{1}{\sum_{i=1}^n E''_{y,i} \frac{t_i^3}{12}} - \frac{1}{\sum_{i=1}^n E''_{y,i} \frac{z_{max,i}^3 - z_{min,i}^3}{3}}} \right) \tag{8.13}$$

where ℓ is the mean length of the lines surrounding the surface as a “box”.

$$\sum_{i=1}^n E''_{x,i} \frac{t_i^3}{12} = 8{,}000{,}000 \frac{0.010^3}{12} + 230{,}000 \frac{0.016^3}{12} + 8{,}000{,}000 \frac{0.012^3}{12} = 1.897 \text{ kNm}$$

$$\sum_{i=1}^n E''_{x,i} \frac{z_{max,i}^3 - z_{min,i}^3}{3} = 8{,}000{,}000 \frac{(-0.009)^3 - (-0.019)^3}{3} +$$

$$+ 230{,}000 \frac{0.007^3 - (-0.009)^3}{3} + 8{,}000{,}000 \frac{0.019^3 - 0.007^3}{3} =$$

$$= 33.805 \text{ kNm}$$

$$D''_{44} = \max \left(2{,}128.07, \frac{48}{5 \cdot 1^2} \frac{1}{\frac{1}{1.897} - \frac{1}{33.805}} \right) = \max(2{,}128.07, 19.30) = 2{,}128.07 \text{kN/m}$$

$$\sum_{i=1}^n E''_{y,i} \frac{t_i^3}{12} = 270{,}000 \frac{0.010^3}{12} + 7{,}000{,}000 \frac{0.016^3}{12} + 270{,}000 \frac{0.012^3}{12} = 2.451 \text{ kNm}$$

$$\sum_{i=1}^n E''_{y,i} \frac{z_{max,i}^3 - z_{min,i}^3}{3} = 270{,}000 \frac{(-0.009)^3 - (-0.019)^3}{3} +$$

$$+ 7{,}000{,}000 \frac{0.007^3 - (-0.009)^3}{3} + 270{,}000 \frac{0.019^3 - 0.007^3}{3} =$$

$$= 3.640 \text{ kNm}$$

$$D''_{55} = \max\left(7{,}085.28, \frac{48}{5 \cdot 1^2}\frac{1}{\frac{1}{2.451} - \frac{1}{3.640}}\right) = \max\left(7{,}085.28, 72.03\right) = 7{,}085.28\text{kN/m}$$

5. The stiffnesses D_{44}, D_{55}, and D_{45} are obtained by transforming the values D''_{44}, D''_{55} from the coordinate system x'', y'' back to the coordinate system x, y (local coordinate system of surface).

$$\begin{aligned} D_{44} &= \cos^2(\varphi)\,D''_{44} + \sin^2(\varphi)\,D''_{55} \\ D_{55} &= \sin^2(\varphi)\,D''_{44} + \cos^2(\varphi)\,D''_{55} \\ D_{45} &= \sin(\varphi)\cos(\varphi)\left(D''_{44} - D''_{55}\right) \end{aligned} \tag{8.14}$$

$$\begin{aligned} D_{44} &= \cos^2(0\,°) \cdot 2{,}128.07 + \sin^2(0\,°) \cdot 7{,}085.28 = 2{,}128.07\text{ kNm} \\ D_{55} &= \sin^2(0\,°) \cdot 2{,}128.07 + \cos^2(0\,°) \cdot 7{,}085.28 = 7{,}085.28\text{ kNm} \\ D_{45} &= \sin(0\,°) \cdot \cos(0\,°) \cdot (2{,}128.07 - 7{,}085.28) = 0.00\text{ kNm} \end{aligned}$$

Global stiffness matrix

$$\boldsymbol{D} = \begin{bmatrix} 33.85 & 0.24 & 0 & 0 & 0 & 124.49 & 0.13 & 0 \\ & 3.64 & 0 & 0 & 0 & 0.13 & -107.82 & 0 \\ & & 2.26 & 0 & 0 & 0 & 0 & 0.96 \\ & & & 2{,}128.07 & 0 & 0 & 0 & 0 \\ & & & & 7{,}085.28 & 0 & 0 & 0 \\ & & \text{sym.} & & & 179{,}923 & 1{,}927 & 0 \\ & & & & & & 118{,}095 & 0 \\ & & & & & & & 18{,}040 \end{bmatrix}$$

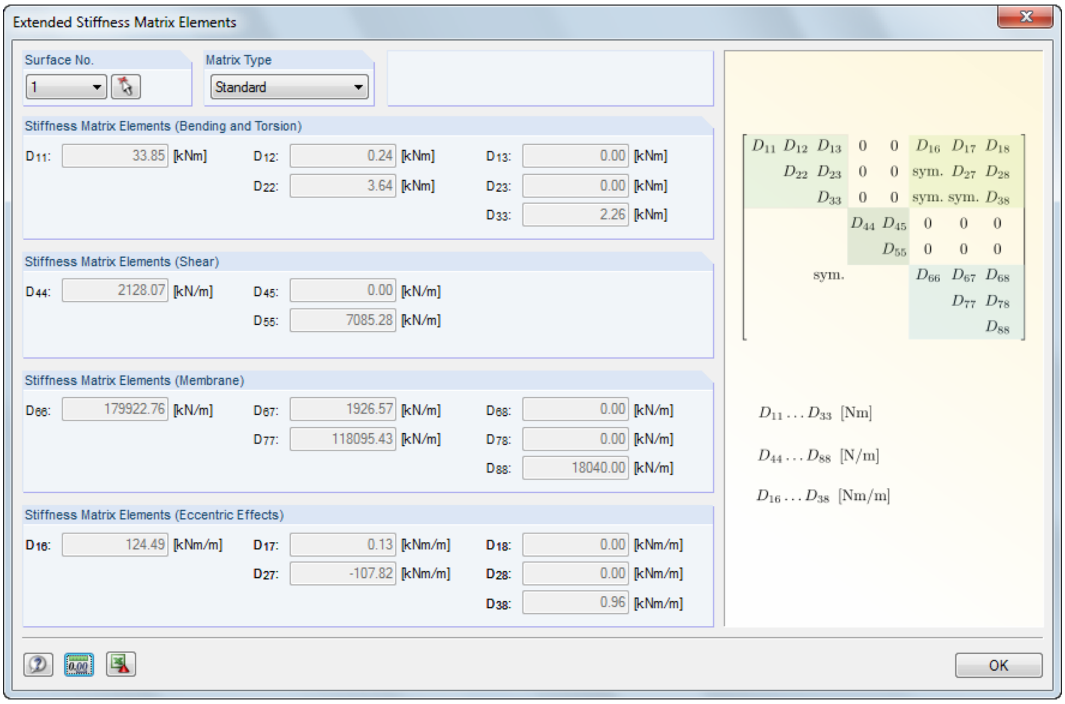


Figure 8.7: Dialog box *Extended Stiffness Matrix Elements* – with shear coupling of layers

8.1.2 Without Shear Coupling of Layers

The angles β_i are multiples of 90 °. Therefore, the global stiffness matrix has the form

$$
\boldsymbol{D} = \begin{bmatrix} D_{11} & D_{12} & 0 & 0 & 0 & 0 & 0 & 0 \\ & D_{22} & 0 & 0 & 0 & 0 & 0 & 0 \\ & & D_{33} & 0 & 0 & 0 & 0 & 0 \\ & & & D_{44} & 0 & 0 & 0 & 0 \\ & & & & D_{55} & 0 & 0 & 0 \\ & & \text{sym.} & & & D_{66} & D_{67} & 0 \\ & & & & & & D_{77} & 0 \\ & & & & & & & D_{88} \end{bmatrix} \tag{8.15}
$$

Stiffness matrix elements – bending and torsion

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

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

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

$$
\boldsymbol{d_1} = \begin{bmatrix} 8{,}010.81 & 54.07 & 0 \\ 54.07 & 270.36 & 0 \\ 0 & 0 & 500.00 \end{bmatrix} \text{MN/m}^2
$$

$$
\boldsymbol{d_2} = \begin{bmatrix} 230.30 & 46.06 & 0 \\ 46.06 & 7{,}009.21 & 0 \\ 0 & 0 & 440.00 \end{bmatrix} \text{MN/m}^2
$$

$$
\boldsymbol{d_3} = \begin{bmatrix} 8{,}010.81 & 54.07 & 0 \\ 54.07 & 270.36 & 0 \\ 0 & 0 & 500.00 \end{bmatrix} \text{MN/m}^2
$$

$$D_{11} = \frac{0.010^3}{12} 8{,}010.81 \cdot 10^3 + \frac{0.016^3}{12} 230.30 \cdot 10^3 + \frac{0.012^3}{12} 8{,}010.81 \cdot 10^3 = 1.900 \text{ kNm}$$

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

$$D_{22} = \frac{0.010^3}{12} 270.36 \cdot 10^3 + \frac{0.016^3}{12} 7{,}009.21 \cdot 10^3 + \frac{0.012^3}{12} 270.36 \cdot 10^3 = 2.454 \text{ kNm}$$

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

Stiffness matrix elements – membrane

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

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

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

$$D_{66} = 10 \cdot 10^{-3} \cdot 8{,}010.81 \cdot 10^3 + 16 \cdot 10^{-3} \cdot 230.30 \cdot 10^3 + 12 \cdot 10^{-3} \cdot 8{,}010.81 \cdot 10^3 = 179{,}923 \text{ N/m}$$

$$D_{67} = 10 \cdot 10^{-3} \cdot 54{,}07 \cdot 10^3 + 16 \cdot 10^{-3} \cdot 46{,}06 \cdot 10^3 + 12 \cdot 10^{-3} \cdot 54{,}07 \cdot 10^3 = 1{,}927 \text{ N/m}$$

$$D_{77} = 10 \cdot 10^{-3} \cdot 270{,}36 \cdot 10^3 + 16 \cdot 10^{-3} \cdot 7{,}009{,}21 \cdot 10^3 + 12 \cdot 10^{-3} \cdot 270{,}36 \cdot 10^3 = 118{,}095 \text{ N/m}$$

$$D_{77} = 10 \cdot 10^{-3} \cdot 500 \cdot 10^3 + 16 \cdot 10^{-3} \cdot 440 \cdot 10^3 + 12 \cdot 10^{-3} \cdot 500 \cdot 10^3 = 18{,}040 \text{ N/m}$$

Stiffness matrix elements – shear

1) The angle $\varphi = 0\,°$ defines the coordinate system x″, y″ with the maximum stiffness.

2) The shear stiffnesses $G''_{xz,i}$ and $G''_{yz,i}$ of each layer in the coordinate system x″, y″ are defined as follows.

$$G''_{xz,i} = \cos^2(\varphi - \beta_i)\, G_{xz,i} + \sin^2(\varphi - \beta_i)\, G_{yz,i}$$

$$G''_{yz,i} = \sin^2(\varphi - \beta_i)\, G_{xz,i} + \cos^2(\varphi - \beta_i)\, G_{yz,i} \qquad i = 1,\dots,n \tag{8.16}$$

$$G''_{xz,1} = G''_{xz,3} = \cos^2(0\,°)\, 500 + \sin^2(0\,°)\, 50 = 500\ \mathrm{MN/m^2}$$

$$G''_{yz,1} = G''_{yz,3} = \sin^2(0\,°)\, 500 + \cos^2(0\,°)\, 50 = 50\ \mathrm{MN/m^2}$$

$$G''_{xz,2} = \cos^2(-90\,°)\, 440 + \sin^2(-90\,°)\, 44 = 44\ \mathrm{MN/m^2}$$

$$G''_{yz,2} = \sin^2(-90\,°)\, 440 + \cos^2(-90\,°)\, 44 = 440\ \mathrm{MN/m^2}$$

3) In the coordinate system x'', y'', the values D''_{44} and D''_{55} are calculated according to the following formulas, considering $D''_{45} = 0$.

$$D''_{44} = \frac{5}{6}\sum_{i=1}^{n} G''_{xz,i}\, t_i \tag{8.17}$$

$$D''_{55} = \frac{5}{6}\sum_{i=1}^{n} G''_{yz,i}\, t_i \tag{8.18}$$

$$D''_{44} = \frac{5}{6} 500 \cdot 10^3 \cdot 0.010 + \frac{5}{6} \cdot 44 \cdot 10^3 \cdot 0.016 + \frac{5}{6} 500 \cdot 10^3 \cdot 0.012 = 9{,}753\ \mathrm{kN/m}$$

$$D''_{55} = \frac{5}{6} 50 \cdot 10^3 \cdot 0.010 + \frac{5}{6} \cdot 440 \cdot 10^3 \cdot 0.016 + \frac{5}{6} 50 \cdot 10^3 \cdot 0.012 = 6{,}783\ \mathrm{kN/m}$$

4) The stiffnesses D_{44}, D_{55}, and D_{45} are obtained by transforming the values D''_{44} and D''_{55} from the coordinate system x'', y'' back to the coordinate system x, y (local coordinate system of surface).

$$D_{44} = \cos^2(\varphi)\, D''_{44} + \sin^2(\varphi)\, D''_{55}$$

$$D_{55} = \sin^2(\varphi)\, D''_{44} + \cos^2(\varphi)\, D''_{55} \tag{8.19}$$

$$D_{45} = \sin(\varphi)\cos(\varphi)\left(D''_{44} - D''_{55}\right)$$

$$D_{44} = \cos^2(0\,°) \cdot 9{,}753 + \sin^2(0\,°) \cdot 6{,}783 = 9{,}753\ \mathrm{kNm}$$

$$D_{55} = \sin^2(0\,°) \cdot 9{,}753 + \cos^2(0\,°) \cdot 6{,}783 = 6{,}783\ \mathrm{kNm}$$

$$D_{45} = \sin(0\,°) \cdot \cos(0\,°) \cdot (9{,}753 - 6{,}783) = 0.00\ \mathrm{kNm}$$

Global stiffness matrix

$$\boldsymbol{D} = \begin{bmatrix} 1.900 & 0.028 & 0 & 0 & 0 & 0 & 0 & 0 \\ & 2.454 & 0 & 0 & 0 & 0 & 0 & 0 \\ & & 0.264 & 0 & 0 & 0 & 0 & 0 \\ & & & 9{,}753 & 0 & 0 & 0 & 0 \\ & & & & 6{,}783 & 0 & 0 & 0 \\ & & \text{sym.} & & & 179{,}923 & 1{,}927 & 0 \\ & & & & & & 118{,}095 & 0 \\ & & & & & & & 18{,}040 \end{bmatrix}$$

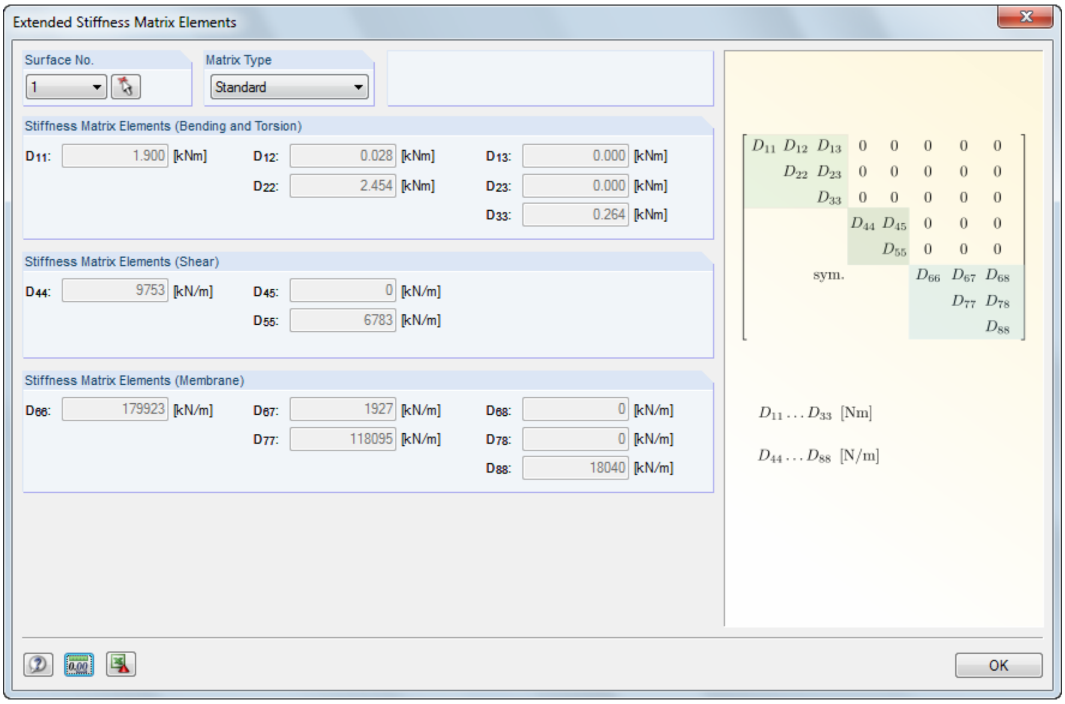


Figure 8.8: Dialog box *Extended Stiffness Matrix Elements* – without shear coupling of layers

8.2 Calculation of Stresses

For the three-layer plate of the previous example, the stresses are to be determined.

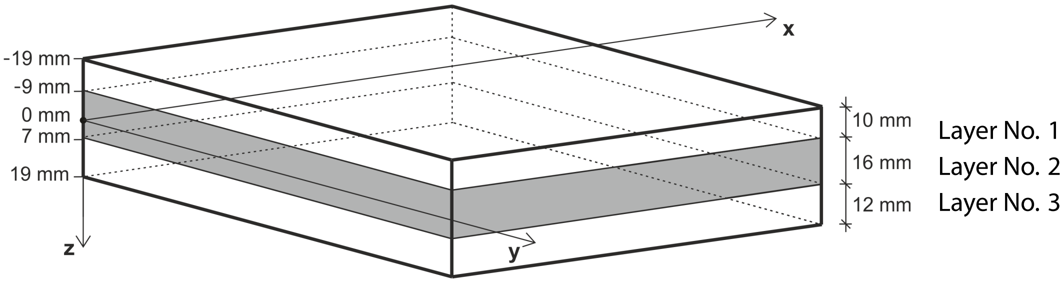


Figure 8.9: Layer scheme

The material characteristics are displayed in Figure 8.10.

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

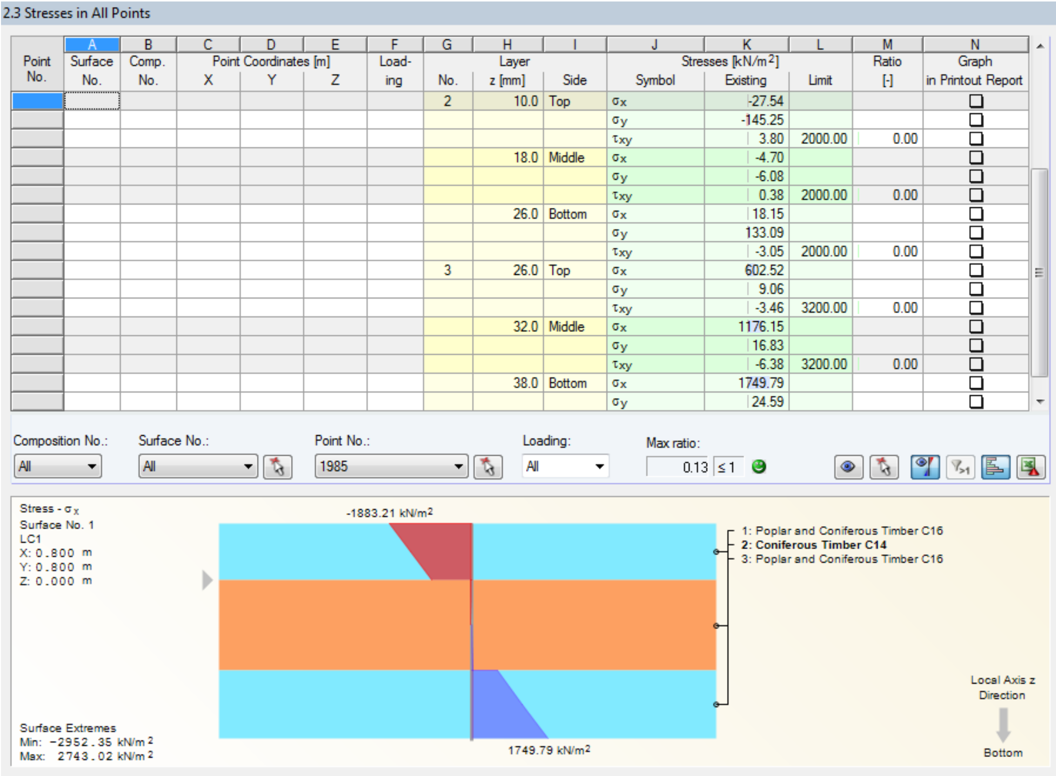
Figure 8.10: Material characteristics

In the previous example from Chapter 8.1, the stiffness matrix elements were calculated with and without considering shear coupling effects. The stresses of the plate differ accordingly.

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

8.2.1 Calculation of Stress Components

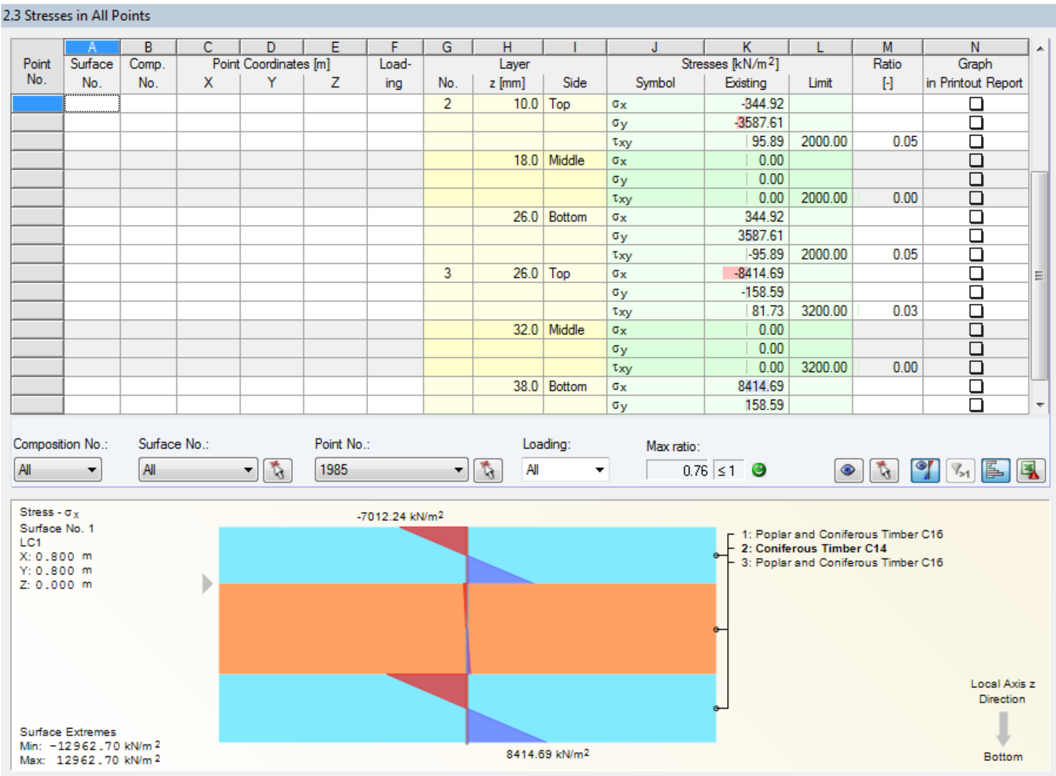
The finite element method of RFEM yields the stresses σ_x, σ_y, τ_{xy}, τ_{xz}, and τ_{yz}. Figure 8.11 and Figure 8.12 show the stress values in the point with the coordinates $[0.8, 0.8, 0]$ of the *Middle* layer. In the first picture, the shear coupling of layers is considered, in the second one it is not.



2.3 Stresses in All Points

Point No.	Surface No.	Comp. No.	X	Y	Z	Loading	Layer No.	z [mm]	Side	Symbol	Existing	Limit	Ratio [-]
							2	10.0	Top	σx	-27.54		
										σy	-145.25		
										τxy	3.80	2000.00	0.00
								18.0	Middle	σx	-4.70		
										σy	-6.08		
										τxy	0.38	2000.00	0.00
								26.0	Bottom	σx	18.15		
										σy	133.09		
										τxy	-3.05	2000.00	0.00
							3	26.0	Top	σx	602.52		
										σy	9.06		
										τxy	-3.46	3200.00	0.00
								32.0	Middle	σx	1176.15		
										σy	16.83		
										τxy	-6.38	3200.00	0.00
								38.0	Bottom	σx	1749.79		
										σy	24.59		

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



2.3 Stresses in All Points

Point No.	Surface No.	Comp. No.	X	Y	Z	Loading	Layer No.	z [mm]	Side	Symbol	Existing	Limit	Ratio [-]
							2	10.0	Top	σx	-344.92		
										σy	-3587.61		
										τxy	95.89	2000.00	0.05
								18.0	Middle	σx	0.00		
										σy	0.00		
										τxy	0.00	2000.00	0.00
								26.0	Bottom	σx	344.92		
										σy	3587.61		
										τxy	-95.89	2000.00	0.05
							3	26.0	Top	σx	-8414.69		
										σy	-158.59		
										τxy	81.73	3200.00	0.03
								32.0	Middle	σx	0.00		
										σy	0.00		
										τxy	0.00	3200.00	0.00
								38.0	Bottom	σx	8414.69		
										σy	158.59		

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

The calculation of the individual stress components is similar for both cases. Therefore, only the case with shear coupling of layers is presented with the following values.

Point	Side	σ_x [kPa]	σ_y [kPa]	τ_{xy} [kPa]
x = 0.8 m, y = 0.8 m, Layer No. 2	Top	−27.54	−145.25	3.80
	Middle	−4.70	−6.08	0.38
	Bottom	18.15	133.09	−3.05

Table 8.1: Stresses in layer No. 2 – with shear coupling

The middle layer is rotated by the angle $\beta = 90\,°$.

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

$$\sigma_{b+t/c,0(top)} = -27.54 \cos^2 90\,° + 3.80 \cdot \sin(2 \cdot 90\,°) - 145.25 \sin^2 90\,° = -145.25 \text{ kPa}$$

$$\sigma_{b+t/c,0(middle)} = -4.70 \cos^2 90\,° + 0.38 \cdot \sin(2 \cdot 90\,°) - 6.08 \sin^2 90\,° = -6.08 \text{ kPa}$$

$$\sigma_{b+t/c,0(bottom)} = 18.15 \cos^2 90\,° - 3.05 \cdot \sin(2 \cdot 90\,°) + 133.09 \sin^2 90\,° = 133.09 \text{ kPa}$$

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

$$\sigma_{b+t/c,90(top)} = -27.54 \sin^2 90\,° - 3.88 \cdot \sin(2 \cdot 90\,°) - 145.25 \cos^2 90\,° = -27.54 \text{ kPa}$$

$$\sigma_{b+t/c,90(middle)} = -4.70 \sin^2 90\,° - 0.38 \cdot \sin(2 \cdot 90\,°) - 6.08 \cos^2 90\,° = -4.70 \text{ kPa}$$

$$\sigma_{b+t/c,90(bottom)} = 18.15 \sin^2 90\,° - (-3.05) \cdot \sin(2 \cdot 90\,°) + 133.09 \cos^2 90\,° = 18.15 \text{ kPa}$$

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

$$\sigma_{t/c,0} = \frac{-145.25 - 6.08 + 133.09}{3} = -6.08 \text{ kPa}$$

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

$$\sigma_{t/c,90} = \frac{-27.54 - 4.70 + 18.15}{3} = -4.70 \text{ kPa}$$

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

$$\sigma_{b,0(top)} = -145.25 - (-6.08) = -139.17 \text{ kPa}$$

$$\sigma_{b,0(middle)} = -6.08 - (-6.08) = 0 \text{ kPa}$$

$$\sigma_{b,0(bottom)} = 133.09 - (-6.08) = 139.17 \text{ kPa}$$

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

$$\sigma_{b,90(top)} = -27.54 - (-4.70) = -22.84 \text{ kPa}$$

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

$$\sigma_{b,90(bottom)} = 18.15 - (-4.70) = 22.84 \text{ kPa}$$

8.2.2 Analysis in RF-LAMINATE Module

First create a *New Model* in RFEM.

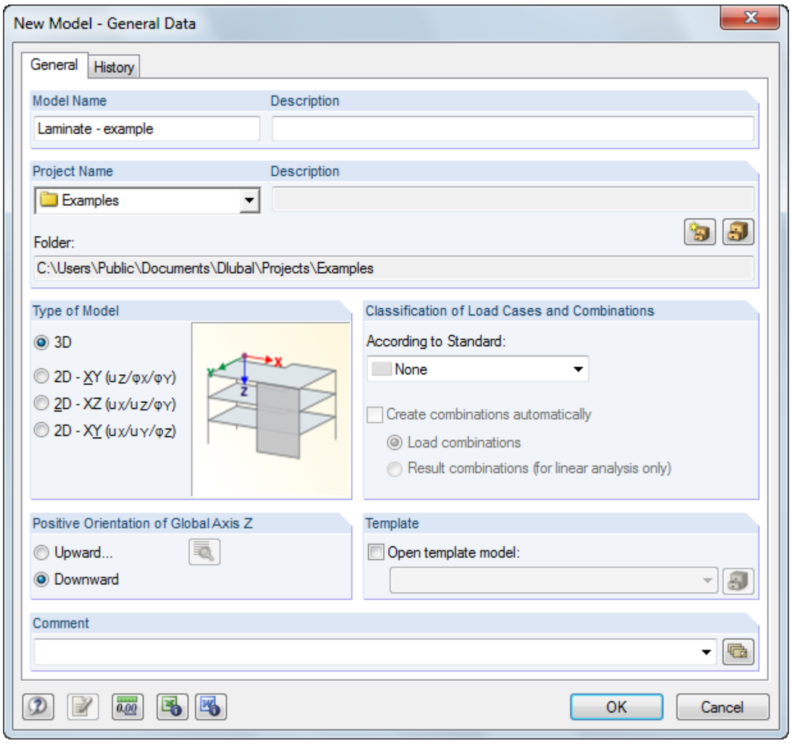


Figure 8.13: Creating new model

Having entered the basics, create a *New Rectangular Surface*. Select the stiffness type *Laminate*. Then define a plate with the dimensions 1.0 × 1.5 m.

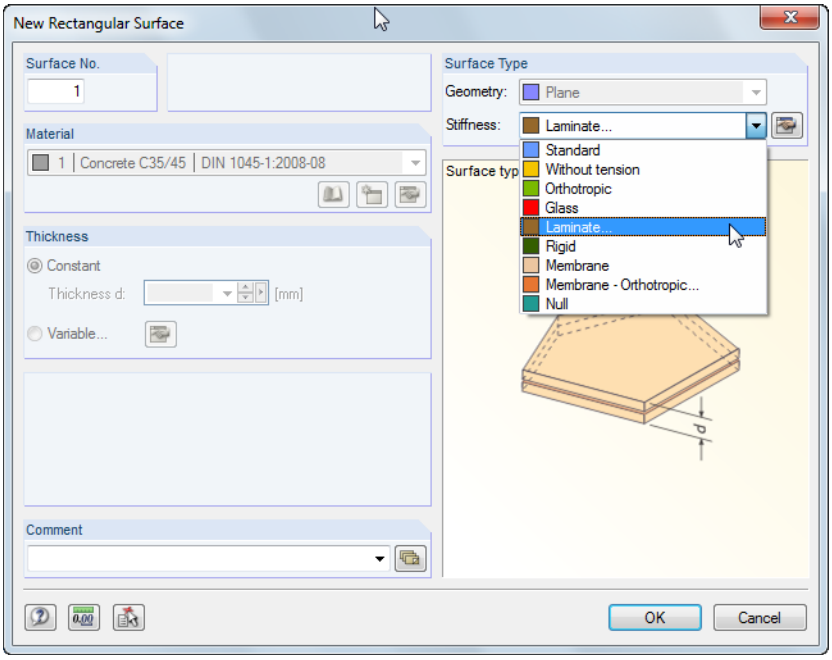
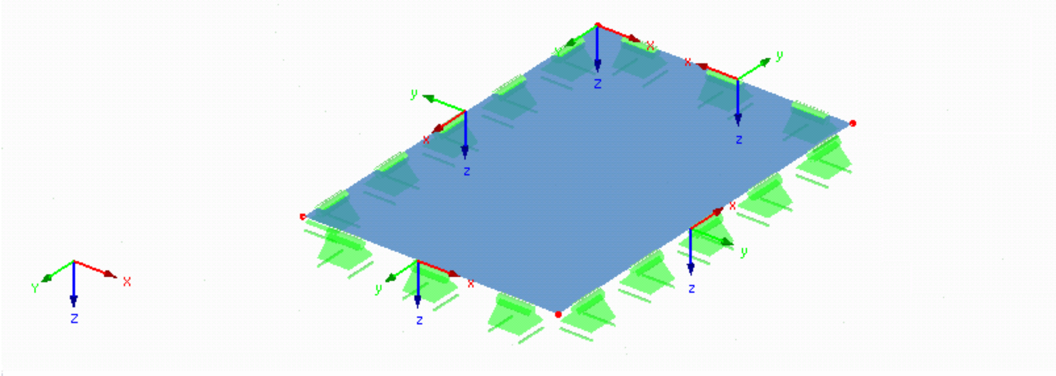


Figure 8.14: Selecting *Laminate* stiffness in *New Rectangular Surface* dialog box

Define the supports according to Figure 8.15.



Support No.	A On Lines No.	B Reference System	C Rotation β [°]	D Wall in Z	E Support or Spring [kN/m²] C_{ux}	F C_{uy}	G C_{uz}	H Rotational Restraint or Spring [kNm/rad/m] φ_x	I φ_y	J φ_z
1	1-4	Local	0.00	☐	☑	☐	☑	☐	☑	☑

Figure 8.15: Table *1.8 Line Supports*

Then create a *New Load Case*.

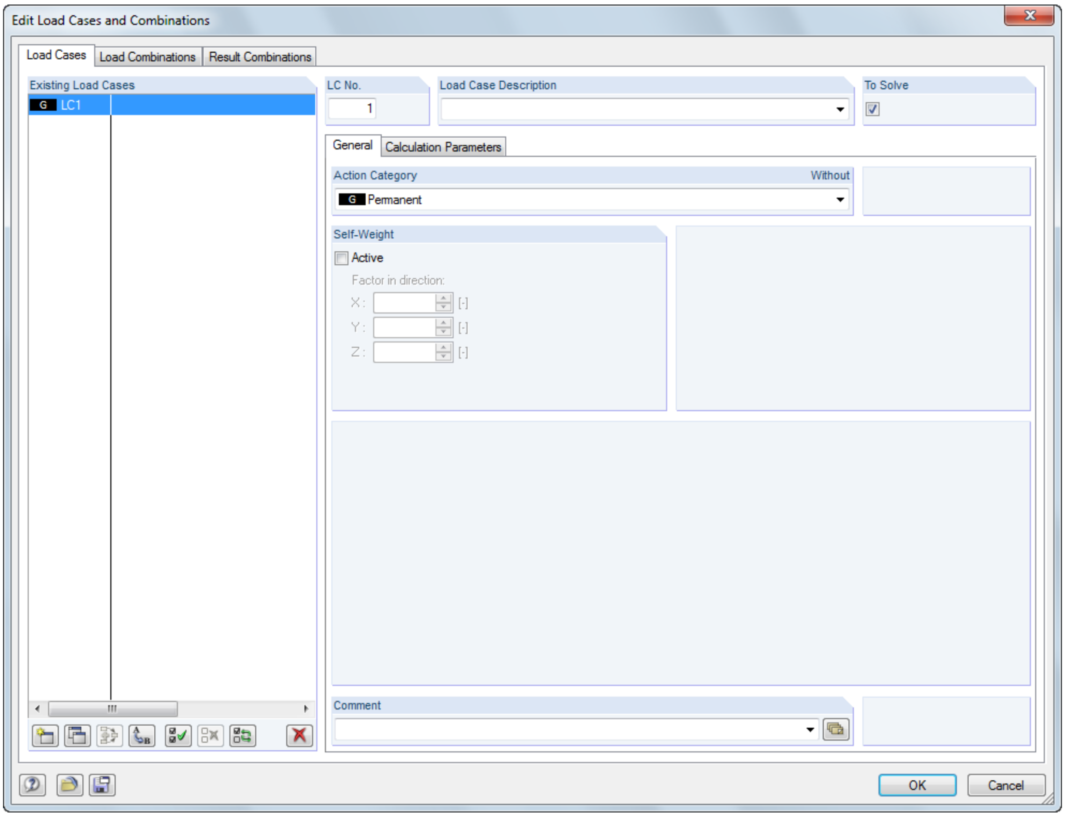


Figure 8.16: Dialog box *Edit Load Cases and Combinations*, tab *Load Cases*

Set the automatic self-weight as **not** *Active*.

Next, open the dialog box *New Surface Load* and enter the load of 5 kN/m^2.

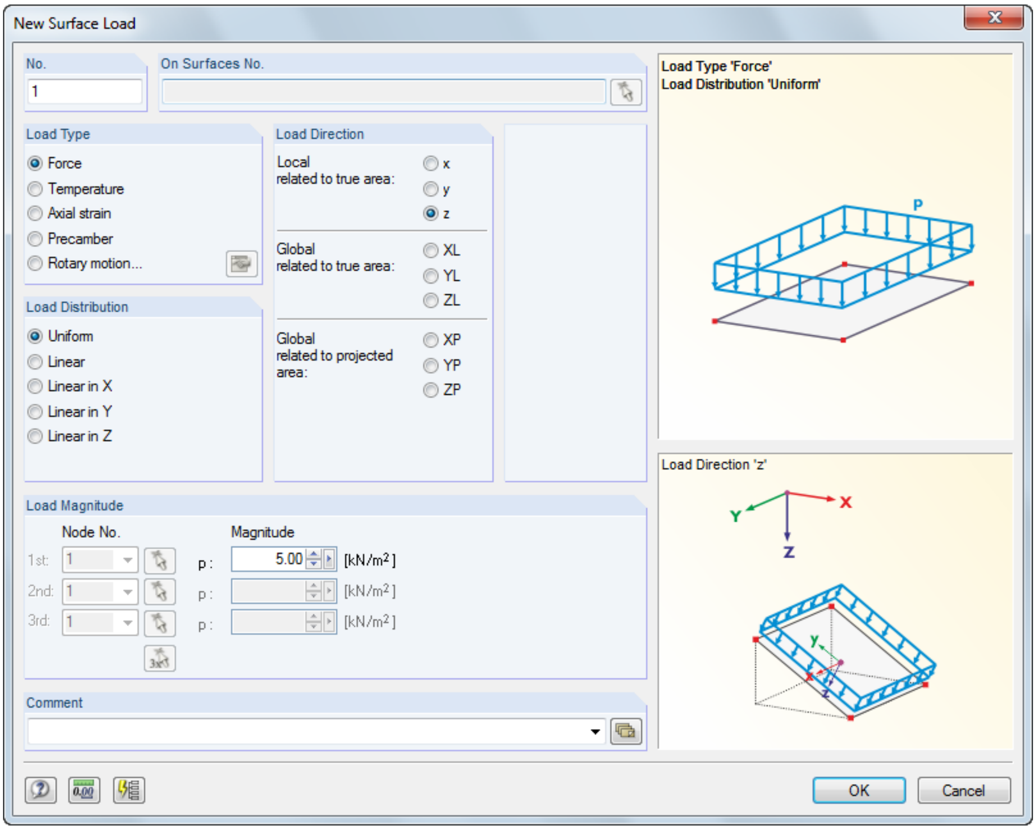


Figure 8.17: Dialog box *New Surface Load*

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

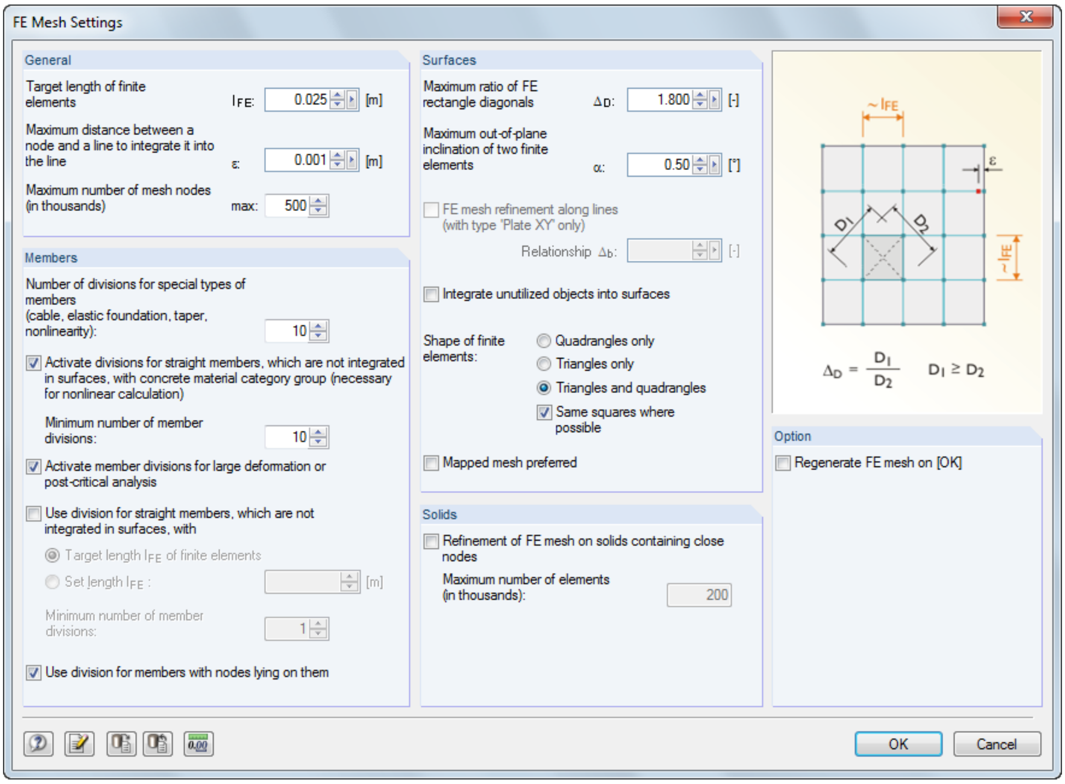


Figure 8.18: Dialog box *FE Mesh Settings*

Now open the RF-LAMINATE module (see Chapter 1.3, page 4).

In Window *1.1 General Data*, surface No. 1 is preset. If any standard is specified, change it to *None*.

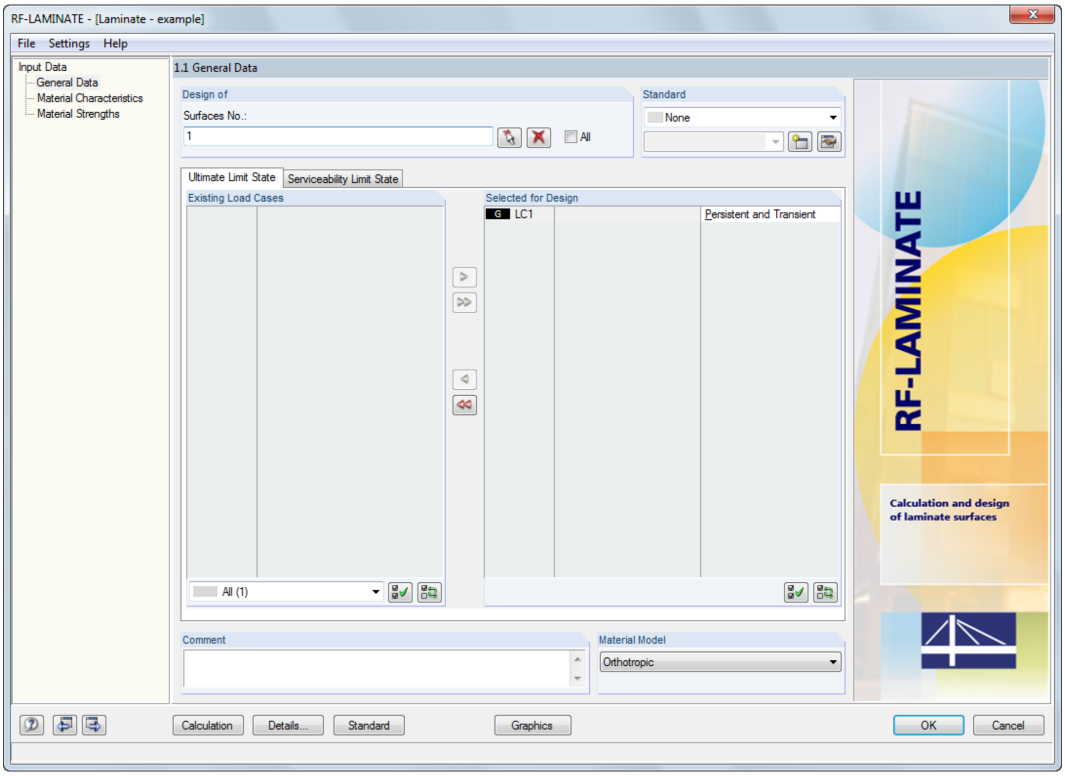


Figure 8.19: Window *1.1 General Data*

Select LC1 for the design and set the *Orthotropic* material model.

In Window *1.2 Material Characteristics*, select the individual layers from the [Library] of materials. Then assign this composition to surface No. 1.

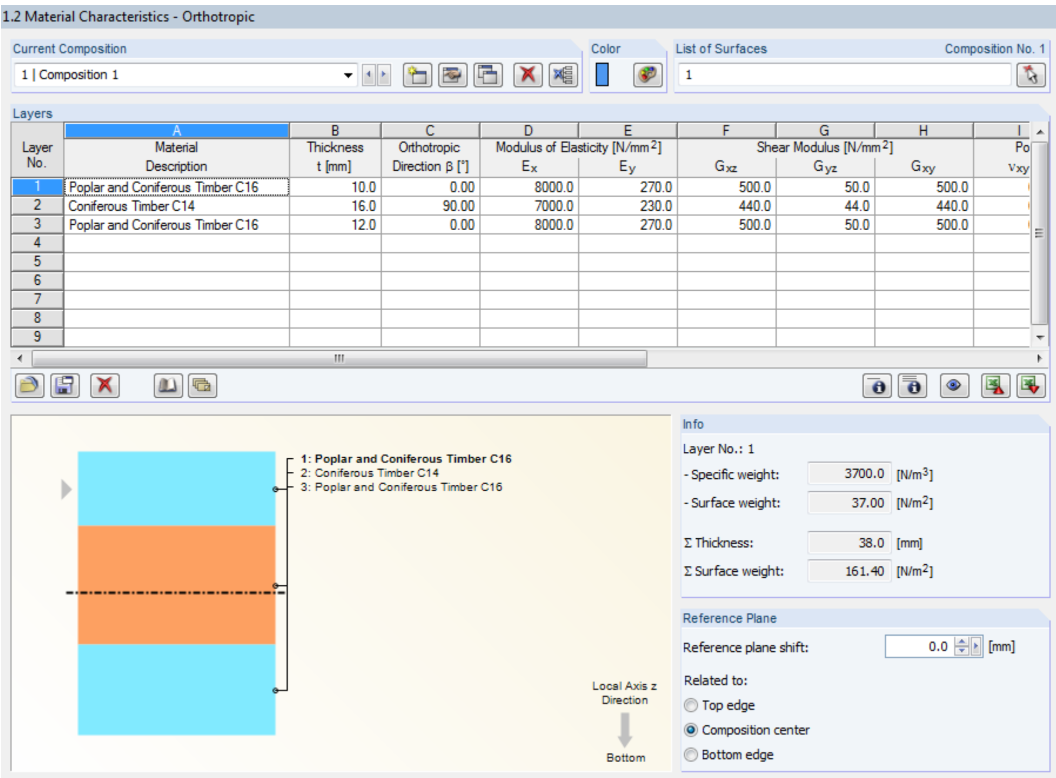


Figure 8.20: Window *1.2 Material Characteristics*

The characteristic strengths of the materials are displayed in Window *1.3 Material Strengths*.

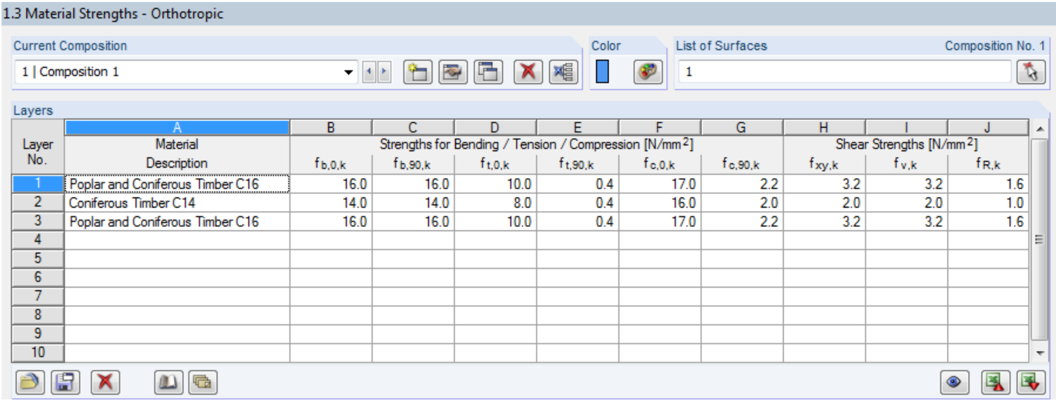


Figure 8.21: Window *1.3 Material Strengths*

Finally, check the settings in the *Details* dialog box.

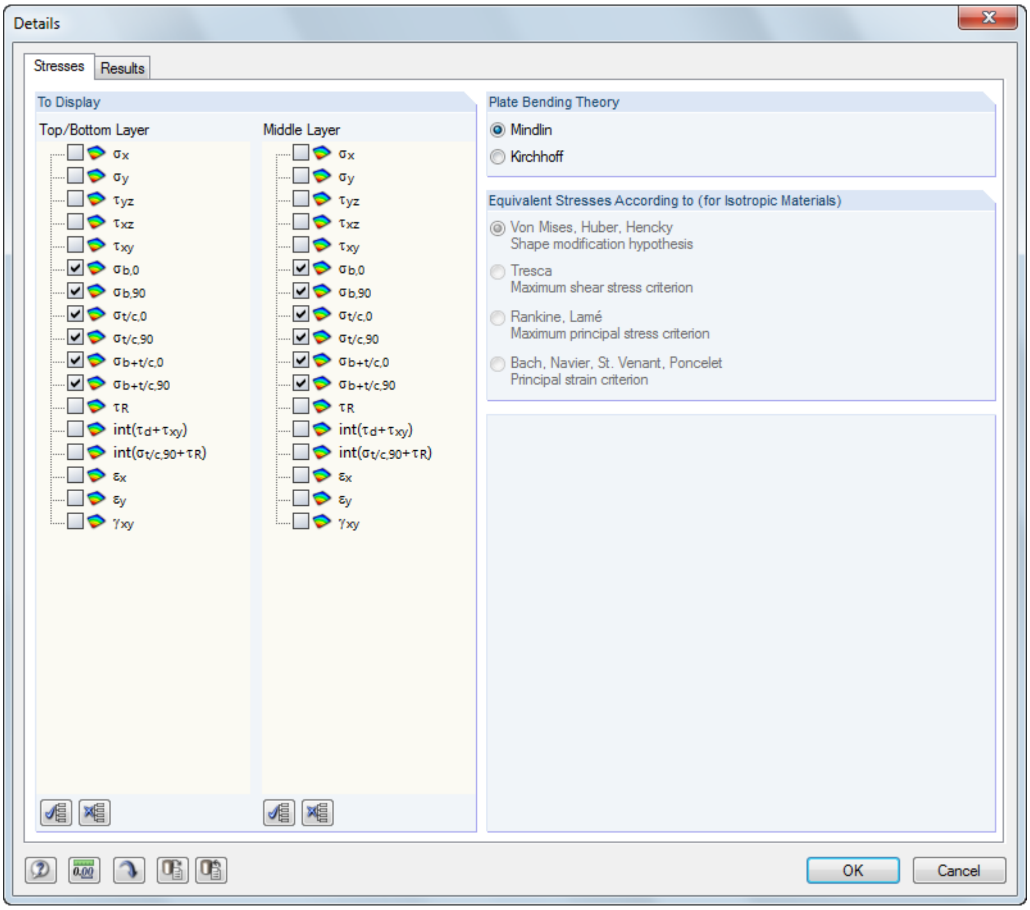
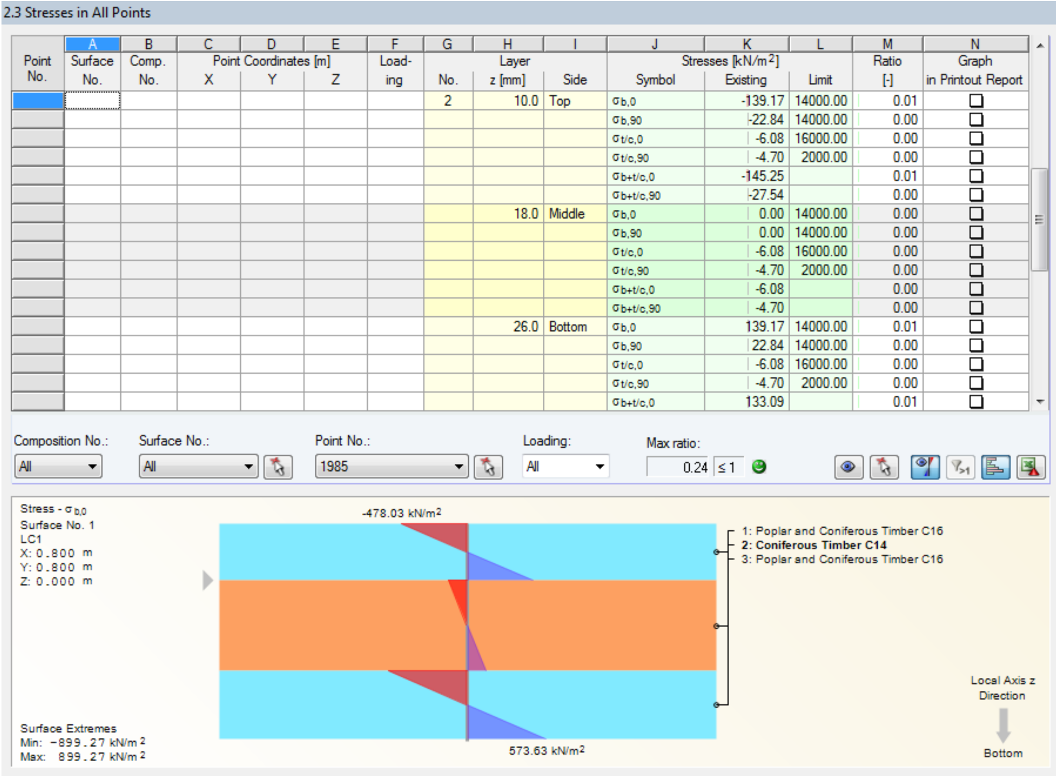


Figure 8.22: Dialog box *Details*, tab *Stresses*

Calculation

Then start the calculation.

You can now check the stress values in the result windows and compare them to ones that were calculated manually in Chapter 8.2.1 on page 75.



2.3 Stresses in All Points

Point No.	Surface No.	Comp. No.	X [m]	Y [m]	Z [m]	Loading	Layer No.	z [mm]	Side	Symbol	Stresses [kN/m²] Existing	Limit	Ratio [-]	Graph in Printout Report
							2	10.0	Top	$\sigma_{b,0}$	-139.17	14000.00	0.01	☐
										$\sigma_{b,90}$	-22.84	14000.00	0.00	☐
										$\sigma_{t/c,0}$	-6.08	16000.00	0.00	☐
										$\sigma_{t/c,90}$	-4.70	2000.00	0.00	☐
										$\sigma_{b+t/c,0}$	-145.25		0.01	☐
										$\sigma_{b+t/c,90}$	-27.54		0.00	☐
								18.0	Middle	$\sigma_{b,0}$	0.00	14000.00	0.00	☐
										$\sigma_{b,90}$	0.00	14000.00	0.00	☐
										$\sigma_{t/c,0}$	-6.08	16000.00	0.00	☐
										$\sigma_{t/c,90}$	-4.70	2000.00	0.00	☐
										$\sigma_{b+t/c,0}$	-6.08		0.00	☐
										$\sigma_{b+t/c,90}$	-4.70		0.00	☐
								26.0	Bottom	$\sigma_{b,0}$	139.17	14000.00	0.01	☐
										$\sigma_{b,90}$	22.84	14000.00	0.00	☐
										$\sigma_{t/c,0}$	-6.08	16000.00	0.00	☐
										$\sigma_{t/c,90}$	-4.70	2000.00	0.00	☐
										$\sigma_{b+t/c,0}$	133.09		0.01	☐

Figure 8.23: Window *2.3 Stresses in All Points*

8.3 Design of a Continuous Plate According to EC 5

The following example is taken from Chapter 10.2 in [5].

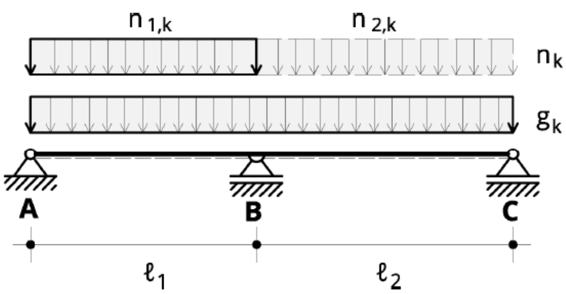


Figure 8.24: Model of two-span plate

System:	Ceiling category A
	$\ell_1 = \ell_2 = 5.20$ m
	$b = 3.50$ m
	Service class 1
	$\gamma_M = 1.25$
Loads:	LC1 - Permanent Load: $g_{2,k} = 2.0$ kN/m^2 + 1.21 kN/m^2 (self-weight)
	LC2 - Live Load: $n_k = 2.5$ kN/m^2, category A, load duration class: medium-term
	LC3 - Live Load: $n_k = 2.5$ kN/m^2, only on right side of the plate, category A, load duration class: medium-term
	CO1 = 1.35LC1 + 1.5LC2 – for ULS design
	CO2 = LC1 + LC3 – for SLS design (characteristic)
Setup:	BSP 220 L7S2

The model is analyzed according to the geometrically linear analysis. The FE mesh length is 0.5 m.

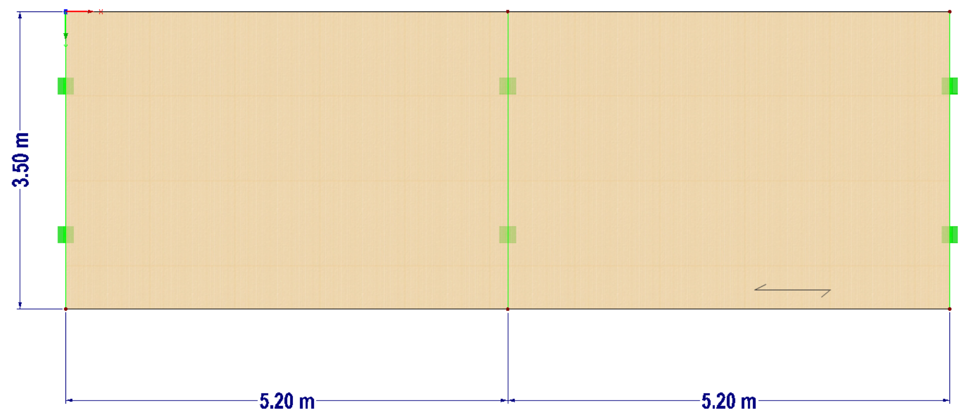


Figure 8.25: RFEM model

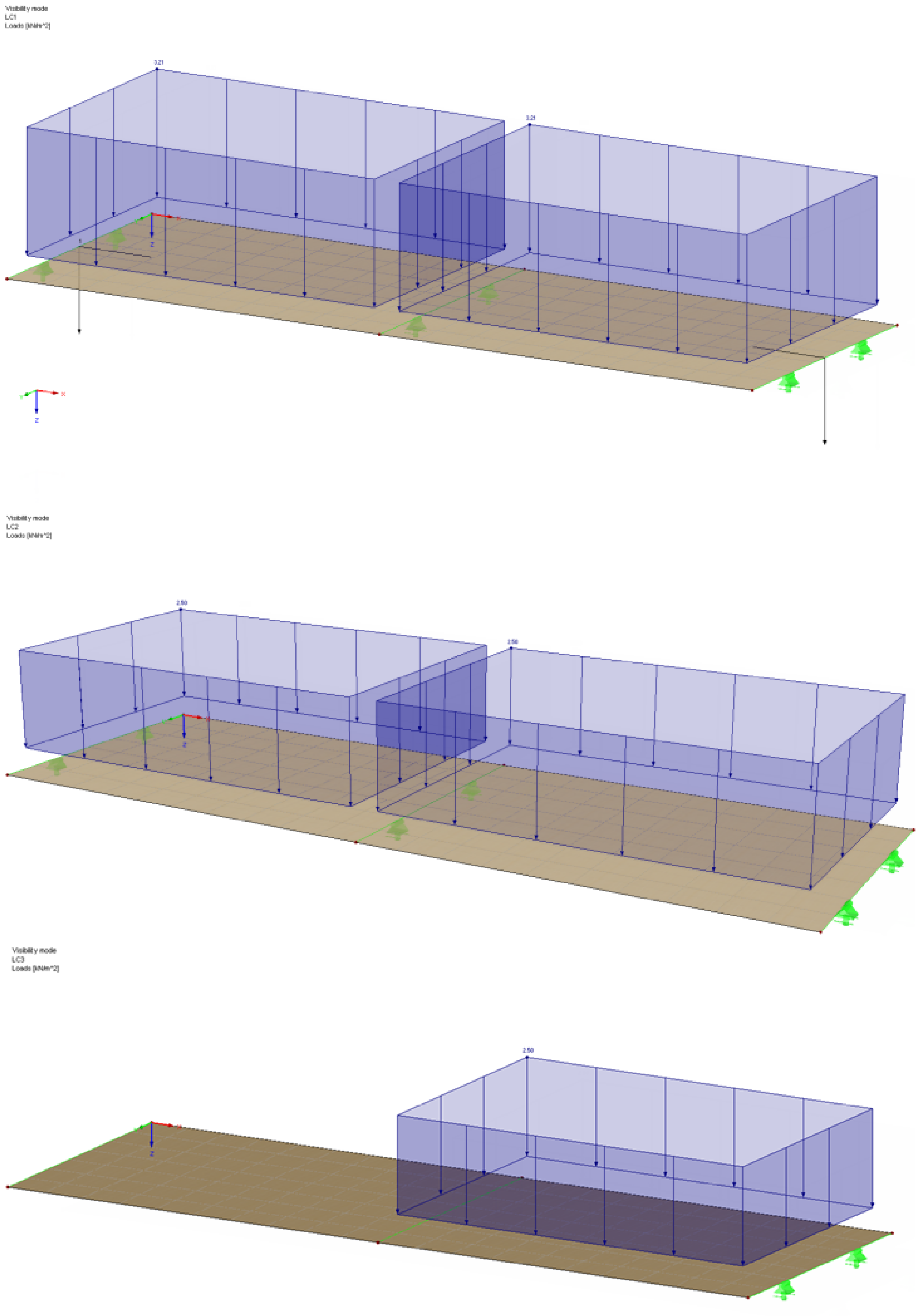


Figure 8.26: Load cases

When the model has been created in RFEM, the RF-LAMINATE module can be started.

The design is done according to EN 1995-1-1 with the German *DIN* annex. Select load combination CO1 for design and assign the *Persistent and Transient* design situation. The material model is *Orthotropic*.

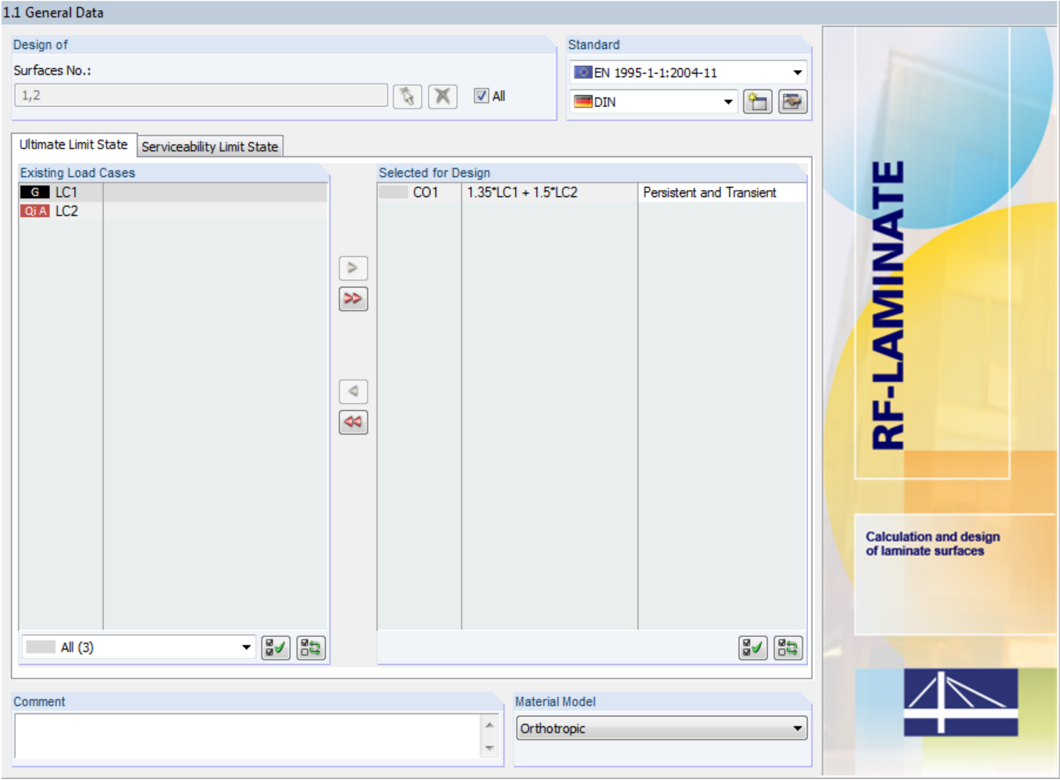


Figure 8.27: Window *1.1 General Data*

For the analysis of the deflections, select CO2 in the *Serviceability Limit State* tab.

The panel section is a STORA ENSO CLT 220 L7s2 from the approval [6]. It can be defined manually in Window *1.2 Material Characteristics - Orthotropic* or – faster – selected from the [Library]. The library, however, always uses the newest settings defined in the approvals from each producer. In order to be able to reproduce the results of this example, it is recommended to define the layers manually.

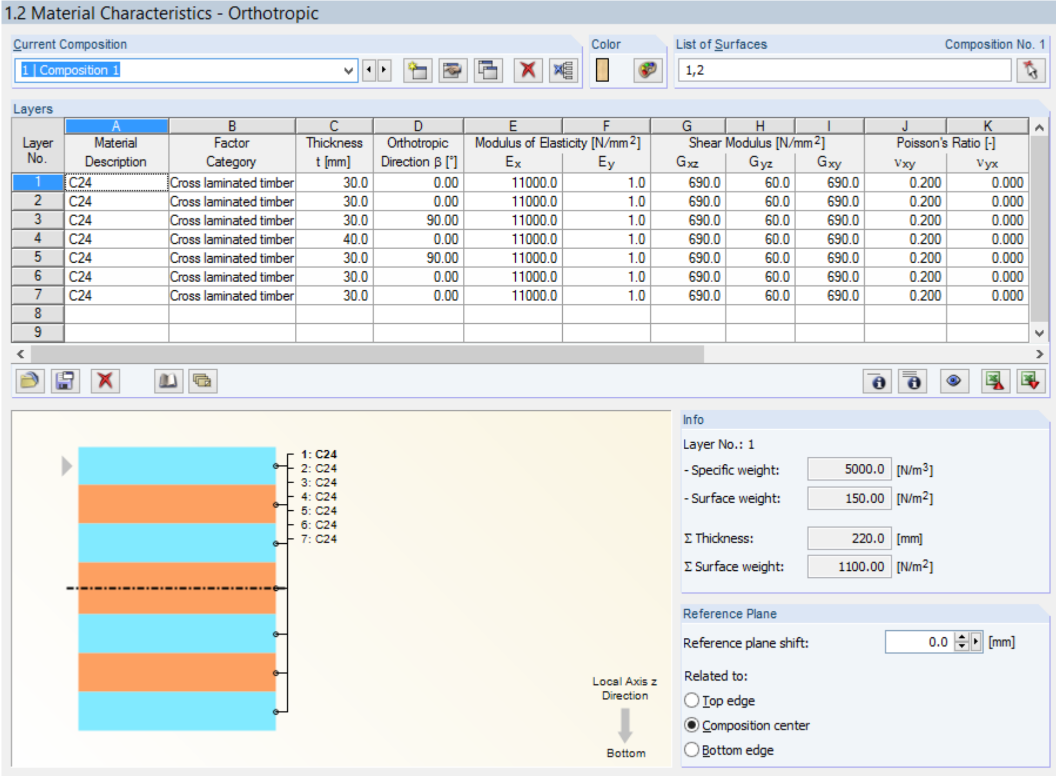


Figure 8.28: Window *1.2 Material Characteristics - Orthotropic*

Assign the factor category *Cross laminated timber* to all layers.

Standard

In the *Standard* dialog box, the safety factor is set to 1.3 for cross laminated timber. According to the recommendations of EC 5, it would also be possible to use to 1.25.

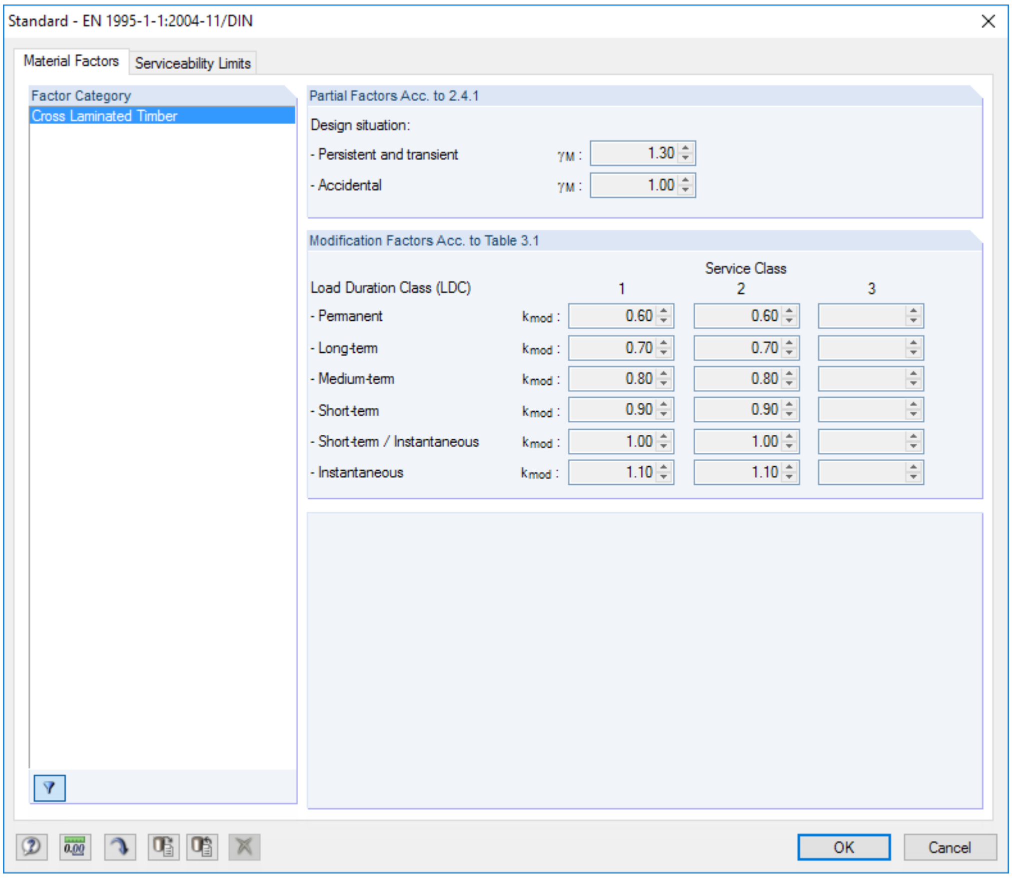


Figure 8.29: Dialog box *Standard - EN 1995-1-1:2004-11/DIN*

In Window *1.3 Material Strengths - Orthotropic*, the material strengths are defined.

Layers

Layer No.	A Material Description	B $f_{b,0,k}$	C $f_{b,90,k}$	D $f_{t,0,k}$	E $f_{t,90,k}$	F $f_{c,0,k}$	G $f_{c,90,k}$	H $f_{xy,k}$	I $f_{v,k}$	J $f_{R,k}$
		Strengths for Bending / Tension / Compression [N/mm²]						Shear Strengths [N/mm²]		
1	C24	24.0	0.0	14.0	0.4	21.0	2.5	4.0	4.0	1.5
2	C24	24.0	0.0	14.0	0.4	21.0	2.5	4.0	4.0	1.5
3	C24	24.0	0.0	14.0	0.4	21.0	2.5	4.0	4.0	1.5
4	C24	24.0	0.0	14.0	0.4	21.0	2.5	4.0	4.0	1.5
5	C24	24.0	0.0	14.0	0.4	21.0	2.5	4.0	4.0	1.5
6	C24	24.0	0.0	14.0	0.4	21.0	2.5	4.0	4.0	1.5
7	C24	24.0	0.0	14.0	0.4	21.0	2.5	4.0	4.0	1.5
8										
9										
10										

Figure 8.30: Material strengths

Results – ULS

The verification of the ultimate limit state is effectuated according to NA.9.3 of Germany.

The internal forces are similar to the example from [5]:

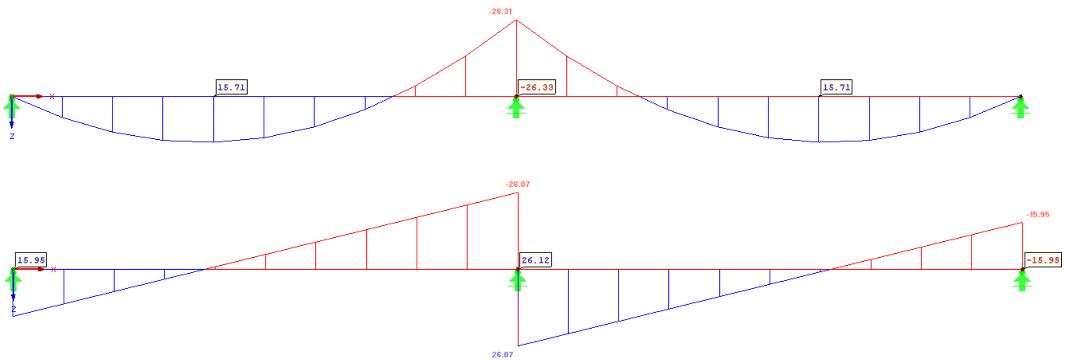


Figure 8.31: Bending moments and shear forces

$M = -26.31$ kNm

$V_x = 26.1$ kN

Bending stress:

$\sigma_{b+t/c,0} = 3.58 \text{ N/mm}^2$

Strength:

$$f_{m,d} = f_{b,0,d} = \frac{k_{mod}}{\gamma_M} f_{b,0,k} = 15.36 \text{ N/mm}^2$$

RF-LAMINATE distinguishes between the pressure and bending stresses as described in Chapter 5.1. However, this is not done here in order to compare the results with [5]. The entire bending stress is compared to the limit strength.

$3.58/15.36 = 0.25 < 1$

In [5], the ratio is 0.22.

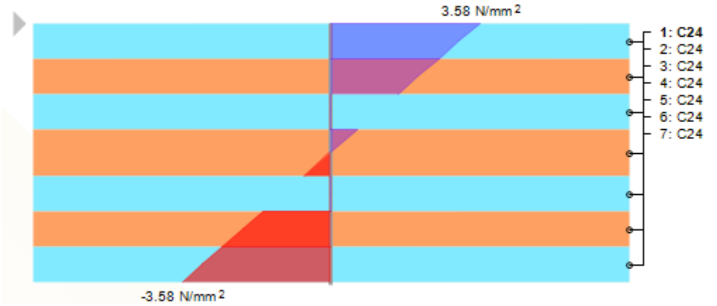


Figure 8.32: Bending stress in RF-LAMINATE

Results – SLS

For the serviceability limit state, the maximum deformation obtained for CO2 is 4.1 mm. It occurs at the distance of 3.1 m from the mid support.

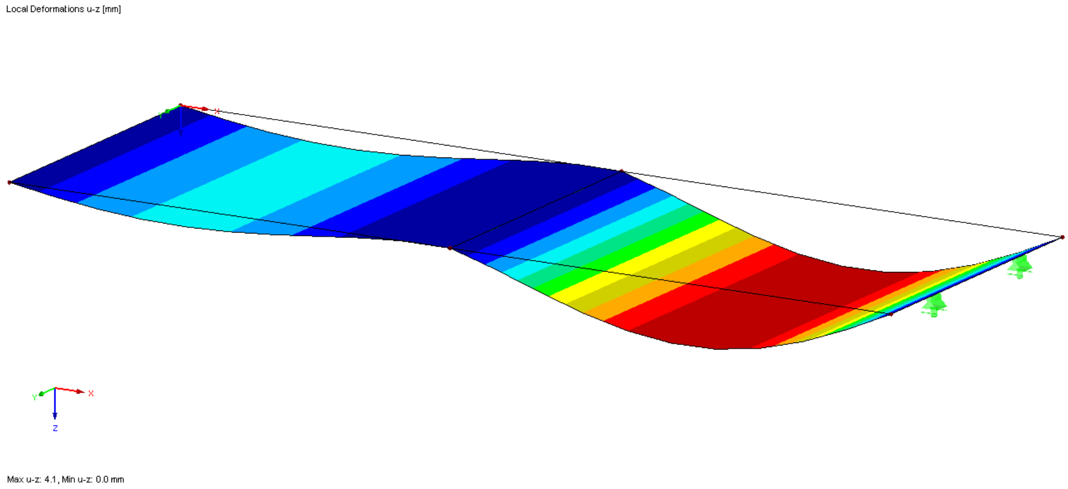


Figure 8.33: Deformations

Verification:

$w_{inst} = 4.1\text{ mm} < l/300 = 5{,}200/300 = 17.3\text{ mm}$

The calculated deformation of $w_{inst} = 4.1$ mm is similar to the one in [5] of 4.5 mm.

As [5] represents a beam design (1D) with the effective moment of inertia, the difference of 0.4 mm is comprehensible.

8.4 Shear Stiffness Matrix Element Calculation

For the surface of the previous example, the shear stiffness matrix elements are to be determined. The material characteristics of the layers are as follows.

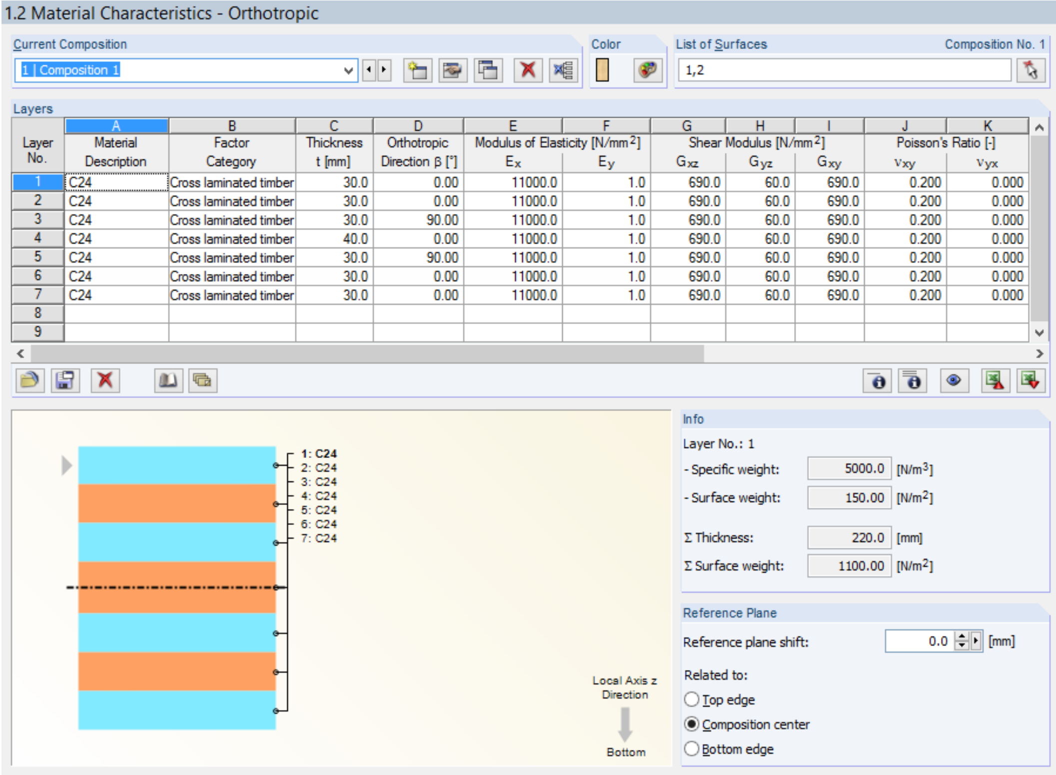


Figure 8.34: Window *1.2 Material Characteristics - Orthotropic*

With this composition of layers, the effects of shear stiffness limitation are demonstrated.

The extended stiffness matrix elements can be checked via the [Info] button as seen on the left.

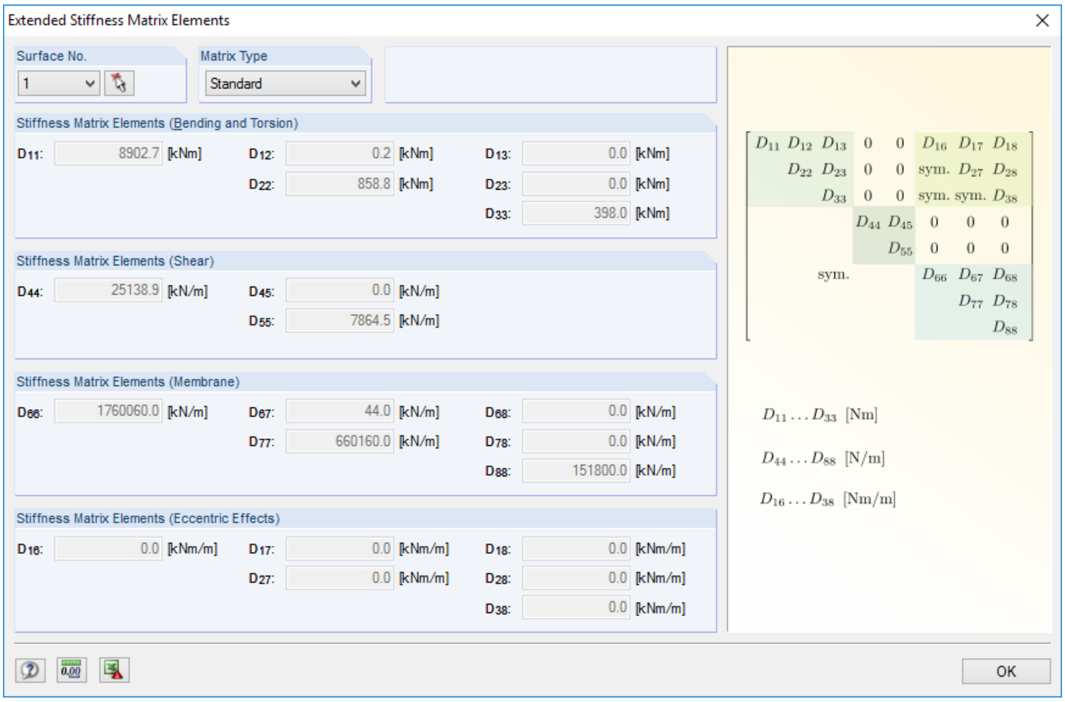


Figure 8.35: Extended stiffness matrix elements

The value of stiffness D''_{44} is given by the following formula:

$$D''_{44} = \max\left(D''_{44,calc}, D''_{min}\right) \max\left(D''_{44,calc}, \frac{48}{5\,\ell^2}\,\frac{1}{\dfrac{1}{\sum\limits_{i=1}^{n} E''_{x,i}\dfrac{t_i^3}{12}} - \dfrac{1}{\sum\limits_{i=1}^{n} E''_{x,i}\dfrac{z^3_{max,i} - z^3_{min,i}}{3}}}\right) \tag{8.20}$$

$$D''_{44} = \max\left(25{,}138.9, \frac{48}{5\cdot 3.5^2}\,\frac{1}{0.00623}\right) = \max\left(25{,}138.9, 125.8\right)$$

With the defined width of 3.5 m, the limitation D''_{min} is not activated.

The limitation D''_{min} is made to avoid shear transformation problems in very small areas and a very soft plate setup. For the layer composition of the example, the limit width of the surface would be 240 mm. Then the model would be as follows:

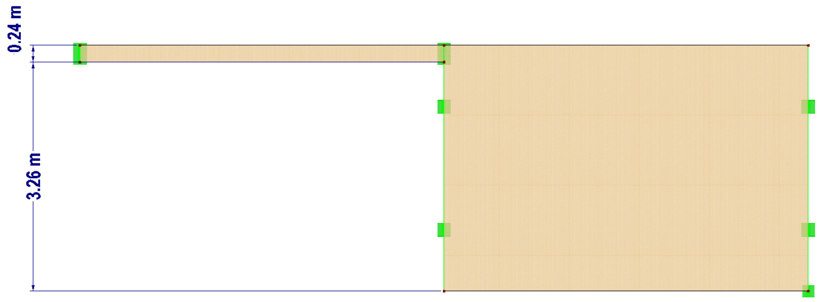


Figure 8.36: Geometry of surfaces with applied limit width

The shear stiffnesses for the x-orientation of this plate are:

	D44
One layer ($D44_1$)	$5/6 t_1 G'_x = 17{,}250$ kN/m
$D''_{44,calc}$	25,139 kN/m
$D''_{min} = D''_{44}$	26,752 kN/m

Table 8.2: Shear stiffnesses for x-orientation of plate

The value for D''_{min} will increase, however, when the surface becomes smaller. The value for $D''_{44,calc}$ takes into account that the shear stiffness of the entire plate increases because of the connection where one board of a surface crosses another one.

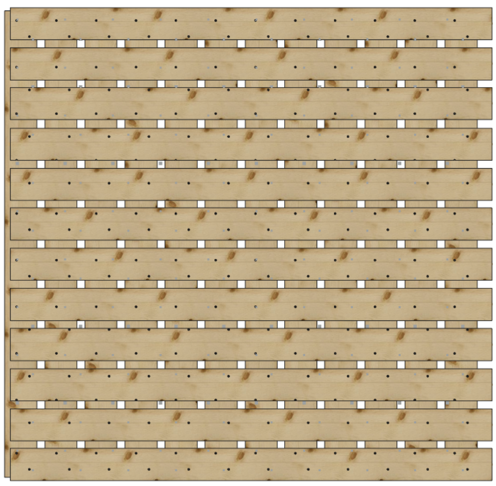


Figure 8.37: Exaggerated drawing of CLT plate

If the surface is very small only in parts as shown in Figure 8.39, the limitation is $D''_{min} = 125.8\,\mathrm{kN/m}$. This means that the value D''_{44} is equal to $D''_{44,calc}$.

The reduction factor k_{44} can applied in order to restrict the shear stiffness matrix element D_{44} for large shear forces that are transformed via the small side of the surface.

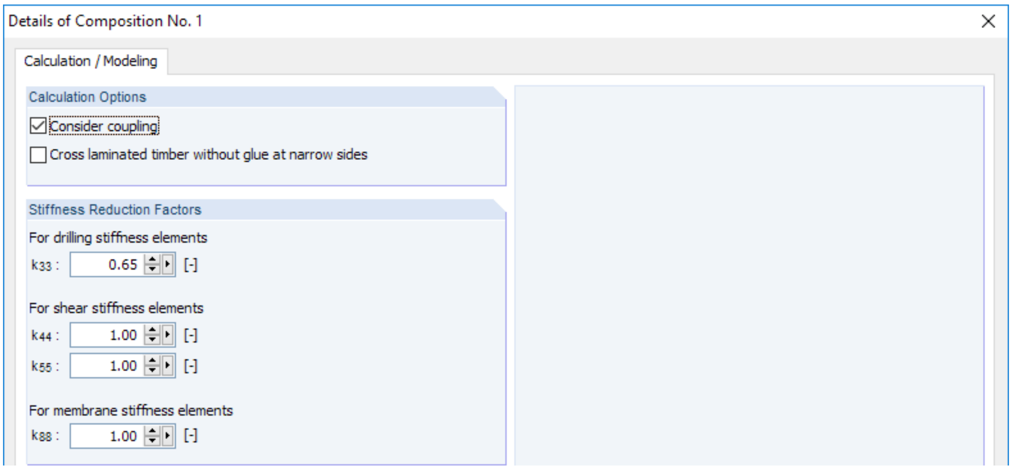


Figure 8.38: *Stiffness Reduction Factors* in *Details of Composition* dialog box

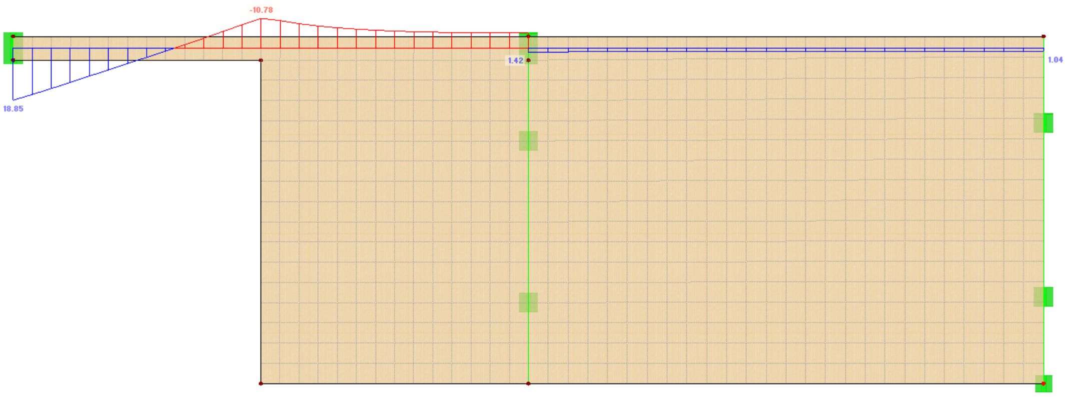


Figure 8.39: High shear force at support of narrow surface

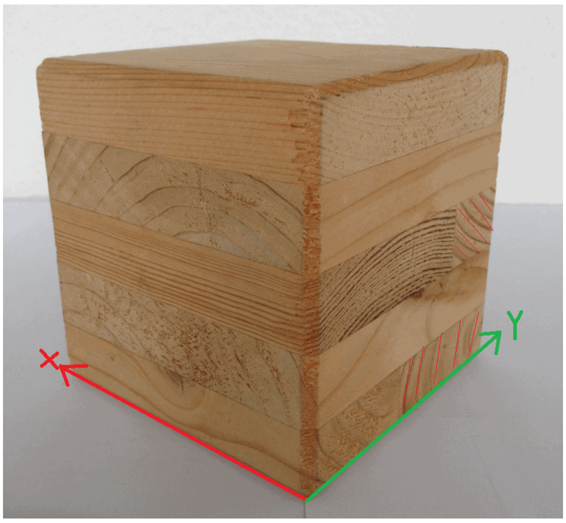


Figure 8.40: Shear failure in fibers, G_{xz} direction

As shown in Figure 8.40, the fibers opposite (soft side) of one layer tend to break due to rolling shear effects. This problem can be accounted for by modifying the shear stiffness elements as mentioned above.

9 Annexes

9.1 Transformation Relations

This chapter describes the relations that are required to transform the stresses, strains and stiffness matrices by rotating the coordinate system x, y, z to the coordinate system x', y', z' about the angle β. This angle β is defined as follows:

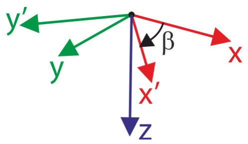


Figure 9.1: Definition of angle β

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

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

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

The stiffness matrix is transformed according to the relation

$$\boldsymbol{d} = \boldsymbol{T}^{T}_{3\times3}\boldsymbol{d}'\boldsymbol{T}_{3\times3} \quad \Leftrightarrow \quad \boldsymbol{d}' = \boldsymbol{T}^{-T}_{3\times3}\boldsymbol{d}\boldsymbol{T}^{-1}_{3\times3} \tag{9.3}$$

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

The transformation relations for shear stresses and strains are

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

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

The stiffness matrix is transformed according to the relation

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

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

9.2 Checking Positive Definiteness of Stiffness Matrix

The positive definiteness of the global stiffness matrix is indispensable for the calculation.

Generally, the global stiffness matrix has the shape

$$\boldsymbol{D}_{\mathbf{8\times 8}} = \begin{bmatrix} \boldsymbol{D}^{\text{bending}}_{3\times 3} & \mathbf{0} & \boldsymbol{D}^{\text{eccentric}}_{3\times 3} \\ \mathbf{0} & \boldsymbol{D}^{\text{shear}}_{2\times 2} & \mathbf{0} \\ \boldsymbol{D}^{\text{eccentric}}_{3\times 3} & \mathbf{0} & \boldsymbol{D}^{\text{membrane}}_{3\times 3} \end{bmatrix} = \begin{bmatrix} D_{11} & D_{12} & D_{13} & & & D_{16} & D_{17} & D_{18} \\ & D_{22} & D_{23} & & & \text{sym.} & D_{27} & D_{28} \\ & & D_{33} & & & \text{sym.} & \text{sym.} & D_{38} \\ & & & D_{44} & D_{45} & & & \\ & & & & D_{55} & & & \\ & & & & & D_{66} & D_{67} & D_{68} \\ & \text{sym.} & & & & & D_{77} & D_{78} \\ & & & & & & & D_{88} \end{bmatrix} \tag{9.9}$$

The following conditions are checked:

1. The matrix $\boldsymbol{D}$ must be positive-definite, i.e. all of its leading principal minors are positive.
2. All submatrices $\boldsymbol{D}^{\text{bending}}_{3\times 3}$, $\boldsymbol{D}^{\text{shear}}_{3\times 3}$, $\boldsymbol{D}^{\text{membrane}}_{3\times 3}$ must be positive-definite in a more restrictive sense, i.e. all of its leading principal minors must satisfy

$$\det \begin{bmatrix} D_{11} & & \\ & \ddots & \\ & & D_{ii} \end{bmatrix} \geq \sqrt{0.001}\,|D_{11}D_{22}\ldots D_{ii}|, \quad \text{where } i = 1,\ldots,n \text{ and } n = 2,3 \tag{9.10}$$

9.3 Two Equivalent Definitions of Poisson's Ratios

When defining an orthotropic material, there are theoretically two ways how to define the Poisson's ratios ν. RFEM uses the approach according to Equation 2.1 on page 8. It is characterized by the relation

$$\nu_{xy} > \nu_{yx} \tag{9.11}$$

if the grain runs in the x'-direction, that is $E_x > E_y$.

In literature, you can occasionally find an equivalent definition of the Poisson's ratios. Let us denote those Poisson's ratios by overlines. For them, the equation $\overline{\nu}_{yx}/E_x = \overline{\nu}_{xy}/E_y$ holds, leading to the inequality $\overline{\nu}_{xy} < \overline{\nu}_{yx}$. If you take the orthotropic material properties from a specific document, you can easily find out the applied orthotropy definition from the inequality between both Poisson's ratios. The stiffness matrix $\boldsymbol{D}$ is defined in both cases as follows:

$$\boldsymbol{D} = \begin{bmatrix} \frac{1}{E_x} & -\frac{\nu_{yx}}{E_y} & & & \\ -\frac{\nu_{xy}}{E_x} & \frac{1}{E_y} & & & \\ & & \frac{1}{G_{yz}} & & \\ & & & \frac{1}{G_{xz}} & \\ & & & & \frac{1}{G_{xy}} \end{bmatrix} = \begin{bmatrix} \frac{1}{E_x} & -\frac{\overline{\nu}_{yx}}{E_y} & & & \\ -\frac{\overline{\nu}_{xy}}{E_x} & \frac{1}{E_y} & & & \\ & & \frac{1}{G_{yz}} & & \\ & & & \frac{1}{G_{xz}} & \\ & & & & \frac{1}{G_{xy}} \end{bmatrix} \tag{9.12}$$

which yields the simple formula

$$\nu_{xy} = \overline{\nu}_{yx}$$
$$\nu_{yx} = \overline{\nu}_{xy} \tag{9.13}$$

In general orthotropic 3D cases, the analogous formulas can be used:

$$\nu_{yz} = \overline{\nu}_{zy} \qquad \nu_{xz} = \overline{\nu}_{zx}$$
$$\nu_{zy} = \overline{\nu}_{yz} \qquad \nu_{zx} = \overline{\nu}_{xz} \tag{9.14}$$

An example shows how to recognize the different definition of the Poisson's ratios and how to compute these values accepted by RFEM. The material properties are as follows:

$$E_x = 12{,}000 \text{ MPa}$$
$$E_y = 400 \text{ MPa}$$
$$\overline{\nu}_{xy} = 0.01 \tag{9.15}$$
$$\overline{\nu}_{yx} = \overline{\nu}_{xy} \cdot \frac{E_x}{E_y} = 0.01 \cdot \frac{12{,}000}{400} = 0.3$$

Realizing that $\overline{\nu}_{xy} < \overline{\nu}_{yx}$, we see that the definition is different than accepted by RFEM. Therefore, we apply Equation 9.13:

$$\nu_{xy} = \overline{\nu}_{yx} = 0.3$$
$$\nu_{yx} = \overline{\nu}_{xy} = 0.01 \tag{9.16}$$

Literature

[1] Huber M.T.. *The theory of crosswise reinforced ferroconcrete slabs and its application to various constructional problems involving rectangular slabs*. Der Bauingenieur, 1923.

[2] *Eurocode 5: Design of timber structures - Part 1-1: General - Common rules and rules for buildings*. CEN, Brussels, 2004.

[3] *National Design Specification for Wood Construction*. American Wood Council, Leesburg, VA, 2015.

[4] *DIN 1052:2008-12: Entwurf, Berechnung und Bemessung von Holztragwerken Allgemeine Bemessungsregeln und Regeln für den Hochbau*. Beuth Verlag GmbH, Berlin, 2008.

[5] Markus Wallner-Novak, Josef Koppelhuber and Kurt Pock. *Brettsperrholz Bemessung, Grundlagen für Statik und Konstruktion nach Eurocode*., 2013.

[6] Deutsches Institut für Bautechnik. *Allgemeine bauaufsichtliche Zulassung Z-9.1-559*., 2007.

[7] Holm Altenbach, Johannes Altenbach and Konstantin Naumenko. *Ebene Flächentragwerke: Grundlagen der Modellierung und Berechnung von Scheiben und Platten*. Springer, 2008.

[8] *Navrhování, výpočet a posuzování dřevěných stavebních konstrukcí : Obecná pravidla a pravidla pro pozemní stavby*. Český normalizační institut, Praha, 2007.

Index