4 Diubal

Version September 2013

Add-on Module

# **RF-LAMINATE**

# Calculation and Design of Laminate Surfaces

# **Program Description**

All rights, including those of translations, are reserved.

No portion of this book may be reproduced – mechanically, electronically or by any other means, including photocopying – without written permission of DLUBAL SOFTWARE GMBH.

© Dlubal Software GmbH Am Zellweg 2 D-93464 Tiefenbach

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

# Contents

	Contents	Page		Contents	Page
1.	Introduction	4	5.5	Parts List	52
1.1	Add-on Module RF-LAMINATE	4	6.	Printout	54
1.2	RF-LAMINATE - Team	5	6.1	Printout Report	54
1.3	Using the Manual	5	6.2	Printing RF-LAMINATE Graphics	54
1.4	Starting RF-LAMINATE	6	6.2.1	Results on the Model in RFEM	54
2.	Theory	8	6.2.2	Stress Diagrams	56
2.1	Symbols	8	7.	General Functions	57
2.2	Material Models	10	7.1	Units and Decimal Places	57
2.2.1	Orthotropic	10	7.2	Export of Results	58
2.2.2	Isotropic	13	7.3	Keyboard Shortcuts	59
2.2.3	User-Defined	13	8.	Examples	60
2.2.4	Hybrid	13	8.1	<b>Calculation of Stiffness Matrix</b>	
2.3	Stiffness Matrix	14		Elements	60
2.3.1	Shear Coupling of Layers Is Considered	14	8.1.1	Shear Coupling of Layers Is Considered	63
2.3.2	Shear Coupling of Layers Is Not Considered	17	8.1.2	Shear Coupling of Layers Is Not Considered	67
3.	Input Data	20	8.2	Calculation of Stresses	70
3.1	General Data	20	8.2.1	Calculation of Individual Stress Components	70
3.1.1	Ultimate Limit State Tab	21	8.2.2	Calculation Procedure in RF-LAMINAT	
3.1.2	Serviceability Limit State Tab	22	9.	Annexes	80
3.2	Material Characteristics	23	9.1	Transformation Relations	80
3.3	Material Strengths	29	9.2	Checking the Positive Definiteness	
3.4	Load Duration and Service Class	30	7.2	of the Stiffness Matrix	81
3.5	Serviceability Data	32	Α	Literature	82
4.	Calculation	33	В	Index	83
4.1	Details	33			
4.1.1	Design Tab	34			
4.1.2	Stresses Tab	35			
4.1.1	Results Tab	42			
4.2	Start Calculation	43			
5.	Results	44			
5.1	Max Stress/Ratio by Loading	45			
5.2	Max Stress/Ratio by Surface	49			
5.3	Stresses in All Points	50			
5.4	Max Displacements	51			



# 1. Introduction

# 1.1 Add-on Module RF-LAMINATE

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

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

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

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

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

Your DLUBAL team



### 1.2 RF-LAMINATE - Team

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

#### **Program coordination**

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

#### Programming

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

#### **Cross-section and material libraries**

Ing. Jan Rybín, Ph.D.

#### Design of a program, windows, dialog boxes and icons

Dipl.-Ing. Georg Dlubal

...

MgA. Robert Kolouch

#### Program testing

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

#### Manual, help system and translation

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

#### **Technical support and quality management**

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

### 1.3 Using the Manual

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

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

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

### 1.4 Starting RF-LAMINATE

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

#### Main menu

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

```
Add-on Modules \rightarrow Others \rightarrow RF-LAMINATE.
```

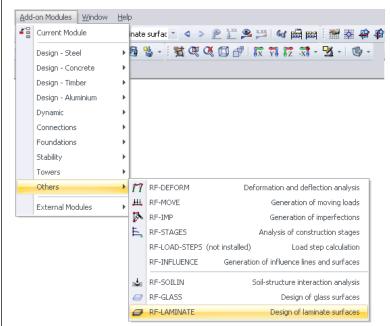


Figure 1.1: Main menu: Add-on Modules  $\rightarrow$  Others  $\rightarrow$  RF-LAMINATE

#### Navigator

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

Add-on Modules  $\rightarrow$  RF-LAMINATE – Design of laminate surfaces.

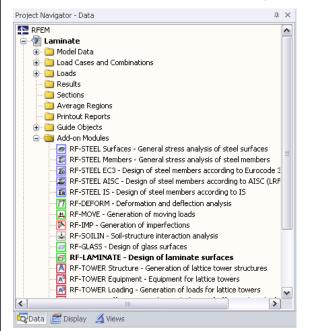


Figure 1.2: Navigator Data: Add-on Modules  $\rightarrow$  RF-LAMINATE

Panel

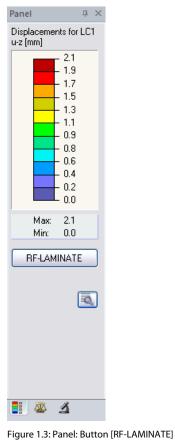




If RF-LAMINATE results are already available in a certain RFEM model, you can set the relevant RF-LAMINATE design case in the load case list in the RFEM toolbar. By using the [Show Results] button, you can display deformations or stresses.

ıbal

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





# 2. Theory

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

# 2.1 Symbols

t	Thickness of individual layers [m]
β	Orthotropy direction [°]
Ε	Young's modulus of elasticity [Pa]
E <sub>x</sub>	Young's modulus of elasticity in $x'$ -axis direction [Pa]
Ey	Young's modulus of elasticity in $y'$ -axis direction [Pa]
G	Shear modulus [Pa]
G <sub>xy</sub>	Shear moduli in $x'y'$ -plane [Pa]
G <sub>xz</sub>	Shear modulus in $x'z$ -plane [Pa]
G <sub>yz</sub>	Shear modulus in y'z -plane [Pa]
ν	Poisson's ratio [-]
V <sub>xy</sub> ,V <sub>yx</sub>	Poisson's ratios in $x'y'$ '-plane [-]
γ	Specific weight [N/m³]
$\alpha_{T}$	Coefficient of thermal expansion [1/K]
d' <sub>ij</sub>	Elements of partial stiffness matrix in coordinate system $x', y', z$ [Pa]
d <sub>ij</sub>	Elements of partial stiffness matrix in coordinate system $x, y, z$ [Pa]
D <sub>ij</sub>	Elements of global stiffness matrix [Nm, Nm/m, N/m]
$\sigma_x$ , $\sigma_y$	Normal stresses [Pa]
$ au_{yz}$ , $ au_{xz}$ , $ au_{xy}$	Shear stresses [Pa]
n	Number of layers [-]
Z	z-axis coordinate [m]
m <sub>x</sub>	Bending moment inducing stresses in $x$ -axis direction [Nm/m]
m <sub>y</sub>	Bending moment inducing stresses in <i>y</i> -axis direction [Nm/m]
m <sub>xy</sub>	Torsional moment [Nm/m]
$v_x, v_y$	Shear forces [N/m]
n <sub>x</sub>	Axial force in $x$ -axis direction [N/m]
n <sub>y</sub>	Axial force in $y$ -axis direction [N/m]
n <sub>xy</sub>	Shear flow [N/m]
f <sub>b,k</sub>	Characteristic value of strength for bending [Pa]
f <sub>t,k</sub>	Characteristic value of strength for tension [Pa]
f <sub>c,k</sub>	Characteristic value of strength for compression [Pa]
f <sub>b,0,k</sub>	Characteristic value of strength for bending along grain [Pa]

<i>f</i> <sub>t,0,k</sub>	Characteristic value of strength for tension along grain [Pa]
<i>f</i> <sub>c,0,k</sub>	Characteristic value of strength for compression along grain [Pa]
f <sub>b,90,k</sub>	Characteristic value of strength for bending perpendicular to grain [Pa]
f <sub>t,90,k</sub>	Characteristic value of strength for tension perpendicular to grain [Pa]
f <sub>c,90,k</sub>	Characteristic value of strength for compression perpendicular to grain [Pa]
f <sub>eqv,k</sub>	Characteristic value of equivalent strength [Pa]
f <sub>xy,k</sub>	Characteristic value of shear strength in plate plane [Pa]
f <sub>v,k</sub>	Characteristic value of shear strength [Pa]
f <sub>R,k</sub>	Characteristic value of rolling shear strength [Pa]
f <sub>b,d</sub>	Design value of strength for bending [Pa]
f <sub>t,d</sub>	Design value of strength for tension [Pa]
f <sub>c,d</sub>	Design value of strength for compression [Pa]
<i>f</i> <sub>b,0,d</sub>	Design value of strength for bending along grain [Pa]
<i>f</i> <sub>t,0,d</sub>	Design value of strength for tension along grain [Pa]
<i>f</i> <sub>c,0,d</sub>	Design value of strength for compression along grain [Pa]
f <sub>b,90,d</sub>	Design value of strength for bending perpendicular to grain [Pa]
<i>f</i> <sub>t,90,d</sub>	Design value of strength for tension perpendicular to grain [Pa]
f <sub>c,90,d</sub>	Design value of strength for compression perpendicular to grain [Pa]
f <sub>eqv,d</sub>	Design value of equivalent strength [Pa]
f <sub>xy,d</sub>	Design value of shear strength in plate plane [Pa]
f <sub>v,d</sub>	Design value of shear strength [Pa]
f <sub>R,d</sub>	Design value of rolling shear strength [Pa]

Dlubal



### 2.2 Material Models

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

- Orthotropic
- Isotropic
- User-Defined
- Hybrid

#### 2.2.1 Orthotropic

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

Layers													
	A	В	С	D	E	F	G	Н		J	K	L	1
Layer	Material	Thickness	Orthotropic	Modulus of Ela	sticity [N/mm <sup>2</sup> ]	Shear M	4odulus (I	V/mm <sup>2</sup> ]	Poisson'	s Ratio [-]	Specific Weight	Coeff. of Th. Exp.	Ì.
No.	Description	t [mm]	Direction β [*]	Ex	Ey	Gxz	Gyz	Gxy	₽ <sub>XY</sub>	¥ух	γ [N/m <sup>3</sup> ]	ατ [1/K]	
1	_												1
2													
ω													

Figure 2.1: Orthotropic material model

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

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

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

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

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

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

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

$$E_{0,mean} = 11000 \text{ N/mm}^2$$
  
 $E_{90,mean} = 370 \text{ N/mm}^2$  (2.3)  
 $G_{mean} = 690 \text{ N/mm}^2$ 

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

$$E_x = E_{0,mean}$$

$$E_y = E_{90,mean}$$

$$G_{xy} = G_{xz} = G_{mean}$$

$$G_{yz} = \frac{G_{mean}}{10}$$
(2.4)

Material Model	
Orthotropic	¥
Orthotropic Isotropic User-Defined	
Isotropic	
Hvbrid	



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

$$v_{xy} \approx \left(\frac{\sqrt{E_x E_y}}{2G_{xy}} - 1\right) \sqrt{\frac{E_x}{E_y}}$$

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

For the softwood mentioned above you get

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

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

#### Example:

Consider the plane stress of a plane plate with the dimensions  $1 \times 1$  m.

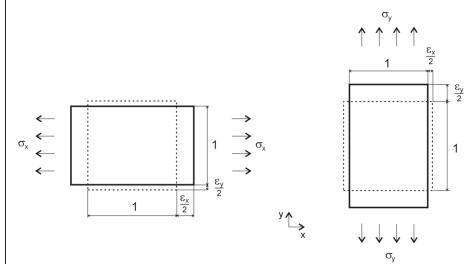


Figure 2.2: Plane stress of the plate in x-direction and y-direction

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

$$\begin{cases} \varepsilon_{x} \\ \varepsilon_{y} \\ \gamma_{xy} \end{cases} = \begin{bmatrix} \frac{1}{E_{x}} & -\frac{v_{xy}}{E_{x}} & 0 \\ -\frac{v_{yx}}{E_{y}} & \frac{1}{E_{y}} & 0 \\ 0 & 0 & G_{xy} \end{bmatrix} \begin{bmatrix} \sigma_{x} \\ \sigma_{y} \\ \tau_{xy} \end{bmatrix}$$
(2.7)



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

$$\begin{cases} \mathcal{E}_{x} \\ \mathcal{E}_{y} \end{cases} = \begin{bmatrix} \frac{1}{E_{x}} & -\frac{\nu_{xy}}{E_{x}} \\ -\frac{\nu_{yx}}{E_{y}} & \frac{1}{E_{y}} \end{bmatrix} \begin{cases} \sigma_{x} \\ \sigma_{y} \end{cases}$$
(2.8)

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

$$\varepsilon_{x} = \frac{\sigma_{x}}{E_{x}}$$

$$\varepsilon_{y} = -\frac{v_{xy}}{E_{x}}\sigma_{x}$$
(2.9)

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

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

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

$$\varepsilon_{x} = -\frac{v_{yx}}{E_{y}}\sigma_{y}$$

$$\varepsilon_{y} = \frac{\sigma_{y}}{E_{y}}$$
(2.11)

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

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

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

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

$$\begin{cases} \varepsilon_x \\ \varepsilon_y \end{cases} = \begin{bmatrix} 1 & -v_{yx} \\ -v_{xy} & 1 \end{bmatrix} \begin{cases} \frac{\sigma_x}{E_x} \\ \frac{\sigma_y}{E_y} \end{cases}$$

(2.13)



#### 2.2.2 Isotropic

An isotropic material has all mechanical properties the same in all directions. The material is defined by using the modulus of elasticity *E*, shear modulus *G*, Poisson's ratio  $\nu$ , specific weight  $\gamma$  and coefficient of thermal expansion  $\alpha_{T}$ .

A	В	С	D	E	F	G	Н	J.
Material	Thickness	Modulus of Elast.	Shear Modulus	Poisson's Ratio	Specific Weight	Coeff. of Th. Exp.		Tř
Description	t [mm]	E [N/mm <sup>2</sup> ]	G [N/mm <sup>2</sup> ]	v [-]	γ [N/m <sup>3</sup> ]	ατ [1/K]	Comment	
<b>-</b>								Т
								1

Figure 2.2: Isotropic material model

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

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

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

#### 2.2.3 User-Defined

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

Layers											
	A	В	С	D	E	F	G	Н		J	K
Layer	Material	Thickness	Orthotropic	Partial St	iffness Mati	ix Elements	s [kN/m²]	Shear Modu	ilus [N/mm <sup>2</sup> ]	Specific Weight	Coeff. of Th. Exp.
No.	Description	t [mm]	Direction ß [*]	ď11	ď 12	ď'22	d'33	Gxz	Gyz	γ [N/m <sup>3</sup> ]	ατ [1/K]
1	<b>_</b>										
2											
3											

Figure 2.3: User-defined material model

### 2.2.4 Hybrid

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

Layers											
	A	В	C	D	E	F	G	Н		J	<u> </u>
Layer	Material	Material	Thickness	Orthotropic	Modulus of Ela	asticity [N/mm <sup>2</sup> ]	Shear	Modulus [N	1/mm2]	Poisson	's Ratio
No.	Description	Model	t [mm]	Direction β [*]	E	Ey	G	Gyz	Gxy	v	I ₽90
1											
2		Orthotropic									
3		Isotropic									
4		User-Defined									

Figure 2.4: Hybrid material model

### 2.3 Stiffness Matrix

#### 2.3.1 Shear Coupling of Layers Is Considered

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

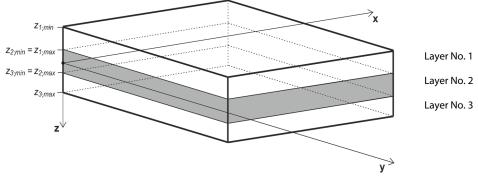


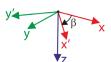
Figure 2.5: Layer scheme

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

$$\mathbf{d}'_{i} = \begin{bmatrix} d'_{i;11} & d'_{i;12} & 0\\ & d'_{i;22} & 0\\ sym. & d'_{i;33} \end{bmatrix} = \begin{bmatrix} \frac{E_{i;x}}{1 - v_{i;xy}^{2} \frac{E_{i;y}}{E_{i;x}}} & \frac{v_{i;xy}E_{i;y}}{1 - v_{i;xy}^{2} \frac{E_{i;y}}{E_{i;x}}} & 0\\ & \frac{E_{i;y}}{1 - v_{i;xy}^{2} \frac{E_{i;y}}{E_{i;x}}} & 0\\ & \frac{1 - v_{i;xy}^{2} \frac{E_{i;y}}{E_{i;x}}}{E_{i;x}} \end{bmatrix} i = 1, ..., n$$
(2.15)

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

$$\mathbf{d}'_{i} = \begin{bmatrix} d'_{i;11} & d'_{i;12} & 0\\ & d'_{i;22} & 0\\ \text{sym.} & d'_{i;33} \end{bmatrix} = \begin{bmatrix} \frac{E_{i}}{1-\nu_{i}^{2}} & \frac{\nu_{i}E_{i}}{1-\nu_{i}^{2}} & 0\\ & \frac{E_{i}}{1-\nu_{i}^{2}} & 0\\ & \text{sym.} & G_{i} \end{bmatrix} \quad i = 1, ..., n \quad \text{where } G_{i} = \frac{E_{i}}{2 \cdot (1+\nu_{i})} (2.16)$$



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

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

where

-

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

\_

The individual elements then are

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

The global stiffness matrix is

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

$$\begin{cases} m_x \\ m_y \\ m_{xy} \\ v_x \\ v_y \\ n_x \\ n_y \\ n_{xy} \end{cases} = \begin{bmatrix} D_{11} & D_{12} & D_{13} \\ D_{22} & D_{23} \\ & D_{33} & 0 & 0 \\ & D_{33} & 0 & 0 \\ & D_{33} & 0 & 0 \\ & D_{44} & D_{45} \\ & D_{55} & 0 & 0 & 0 \\ & D_{55} & 0 & 0 & 0 \\ & D_{66} & D_{67} & D_{68} \\ & D_{77} & D_{78} \\ & & & & & & & & \\ & & & & & & & & \\ & & & & & & & & & \\ & & & & & & & & & \\ & & & & & & & & & & \\ & & & & & & & & & & \\ & & & & & & & & & & \\ & & & & & & & & & & \\ & & & & & & & & & & \\ & & & & & & & & & & \\ & & & & & & & & & \\ & & & & & & & & & \\ & & & & & & & & & \\ & & & & & & & & & \\ & & & & & & & & & \\ & & & & & & & & \\ & & & & & & & & \\ & & & & & & & & \\ & & & & & & & & \\ & & & & & & & & \\ & & & & & & & & \\ & & & & & & & & \\ & & & & & & & & \\ & & & & & & & & \\ & & & & & & & & \\ & & & & & & & & \\ & & & & & & & & \\ & & & & & & & & \\ & & & & & & & & \\ & & & & & & & \\ & & & & & & & \\ & & & & & & & \\ & & & & & & & \\ & & & & & & \\ & & & & & & \\ & & & & & & \\ & & & & & & \\ & & & & & & \\ & & & & & & \\ & & & & & & \\ & & & & & & \\ & & & & & & \\ & & & & & & \\ & & & & & & \\ & & & & & & \\ & & & & & & \\ & & & & & & \\ & & & & & \\ & & & & & \\ & & & & & & \\ & & & & & \\ & & & & & \\ & & & & & \\ & & & & & \\ & & & & & \\ & & & & & \\ & & & & & \\ & & & & & \\ & & & & & \\ & & & & & \\ & & & & & \\ & & & & & & \\ & & & & & & \\ & & & & & & \\ & & & & & & \\ & & & & & & \\ & & & & & & \\ & & & & & & \\ & & & & & & \\ & & & & & & \\ & & & & & & \\ & & & & & & \\ & & & & & \\ & & & & & & \\ & & & & & & \\ & & & & & & \\ & & & & & \\ & & & & & & \\ & & & & & & \\ & & & & & & \\ & & & & & & \\ & & & & & & \\ & & & & & & \\ & & & & & & \\ & & & & & & \\ & & & & & \\ & & & & & & & \\ & & & & & & \\ & & & & & & \\ & & & & &$$

(2.20)

(2.19)

Bending and torsion Shear Membrane Eccentricity

If angles  $\beta_i$  are multiples of 90°, the global stiffness matrix has the simplified form

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

(2.21)

**Dlubal** 

Stiffness matrix elements (bending and torsion) [Nm]

$$D_{11} = \sum_{i=1}^{n} \frac{z_{i;\max}^3 - z_{i;\min}^3}{3} d_{i;11} \qquad D_{12} = \sum_{i=1}^{n} \frac{z_{i;\max}^3 - z_{i;\min}^3}{3} d_{i;12} \qquad D_{13} = \sum_{i=1}^{n} \frac{z_{i;\max}^3 - z_{i;\min}^3}{3} d_{i;13}$$
$$D_{22} = \sum_{i=1}^{n} \frac{z_{i;\max}^3 - z_{i;\min}^3}{3} d_{i;22} \qquad D_{23} = \sum_{i=1}^{n} \frac{z_{i;\max}^3 - z_{i;\min}^3}{3} d_{i;23}$$
$$D_{33} = \sum_{i=1}^{n} \frac{z_{i;\max}^3 - z_{i;\min}^3}{3} d_{i;33}$$

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

$$D_{ij} = \sum_{i=1}^{n=1} \frac{z_{i;\max}^3 - z_{i;\min}^3}{3} d_{i;ij} = \frac{\left(\frac{t}{2}\right)^5 - \left(-\frac{t}{2}\right)^5}{3} d_{1;ij} = \frac{2\left(\frac{t}{2}\right)^5}{3} d_{1;ij} = \frac{t^3}{12} d_{1;ij} \qquad i, j = 1, 2, 3$$

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

$$D_{16} = \sum_{i=1}^{n} \frac{z_{i;\max}^2 - z_{i;\min}^2}{2} d_{i;11} \qquad D_{17} = \sum_{i=1}^{n} \frac{z_{i;\max}^2 - z_{i;\min}^2}{2} d_{i;12} \qquad D_{18} = \sum_{i=1}^{n} \frac{z_{i;\max}^2 - z_{i;\min}^2}{2} d_{i;13}$$
$$D_{27} = \sum_{i=1}^{n} \frac{z_{i;\max}^2 - z_{i;\min}^2}{2} d_{i;22} \qquad D_{28} = \sum_{i=1}^{n} \frac{z_{i;\max}^2 - z_{i;\min}^2}{2} d_{i;23}$$
$$D_{38} = \sum_{i=1}^{n} \frac{z_{i;\max}^2 - z_{i;\min}^2}{2} d_{i;33}$$

Stiffness matrix elements (membrane) [N/m]

$$D_{66} = \sum_{i=1}^{n} t_i d_{i;11} \qquad D_{67} = \sum_{i=1}^{n} t_i d_{i;12} \qquad D_{68} = \sum_{i=1}^{n} t_i d_{i;13}$$
$$D_{77} = \sum_{i=1}^{n} t_i d_{i;22} \qquad D_{78} = \sum_{i=1}^{n} t_i d_{i;23}$$
$$D_{88} = \sum_{i=1}^{n} t_i d_{i;33}$$

#### Stiffness matrix elements (shear) [N/m]

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

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

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

where

$$\mathbf{G}_{i} = \begin{bmatrix} G_{i;11} & G_{i;12} \\ \text{sym.} & G_{i;22} \end{bmatrix} = \mathbf{T}_{2\times2;i}^{\mathsf{T}} \mathbf{G}_{i}^{\mathsf{T}} \mathbf{T}_{2\times2;i}$$
(2.24)

where

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

The individual elements then are

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

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

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

#### 2.3.2 Shear Coupling of Layers Is Not Considered

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

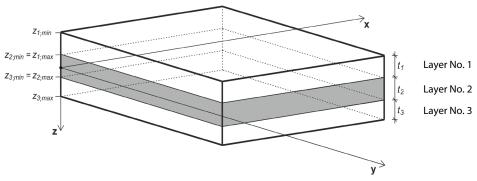


Figure 2.6: Layer scheme

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

$$\mathbf{d}'_{i} = \begin{bmatrix} d'_{i;11} & d'_{i;12} & 0\\ & d'_{i;22} & 0\\ & sym. & d'_{i;33} \end{bmatrix} = \begin{bmatrix} \frac{E_{i;x}}{1 - v_{i;xy}^{2}} \frac{E_{i;y}}{E_{i;x}} & \frac{v_{i;xy}E_{i;y}}{1 - v_{i;xy}^{2}} \frac{E_{i;y}}{E_{i;x}} & 0\\ & \frac{E_{i;y}}{1 - v_{i;xy}^{2}} \frac{E_{i;y}}{E_{i;x}} & 0\\ & \frac{1 - v_{i;xy}^{2}}{E_{i;x}} \frac{E_{i;y}}{E_{i;x}} \end{bmatrix}$$
  $i = 1, ..., n$  (2.26)

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

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

#### 2 Theory

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

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

where

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

The individual elements then are

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

The global stiffness matrix is

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

Bending and torsion Shear Membrane

Dlubal

Dlubal

(2.32)

If angles  $\beta_i$  are multiples of 90°, the global stiffness matrix has the simplified form

$$\mathbf{D} = \begin{bmatrix} D_{11} & D_{12} & 0 & 0 & 0 & 0 & 0 & 0 \\ D_{22} & 0 & 0 & 0 & 0 & 0 & 0 \\ D_{33} & 0 & 0 & 0 & 0 & 0 \\ D_{44} & 0 & 0 & 0 & 0 \\ D_{55} & 0 & 0 & 0 \\ Sym. & D_{66} & D_{67} & 0 \\ D_{77} & 0 \\ D_{88} \end{bmatrix}$$

Stiffness matrix elements (bending and torsion) [Nm]

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

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

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

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

Stiffness matrix elements (membrane) [N/m]

$$D_{66} = \sum_{i=1}^{n} t_i d_{i;11} \qquad D_{67} = \sum_{i=1}^{n} t_i d_{i;12}$$
$$D_{77} = \sum_{i=1}^{n} t_i d_{i;22}$$

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

Stiffness matrix elements (shear) [N/m]

$$D_{44} = \sum_{i=1}^{n} \frac{5}{6} G_{i;11} t_i \qquad D_{45} = \sum_{i=1}^{n} \frac{5}{6} G_{i;12} t_i$$
$$D_{55} = \sum_{i=1}^{n} \frac{5}{6} G_{i;22} t_i$$

where

$$\mathbf{G}_{i} = \begin{bmatrix} G_{i;11} & G_{i;12} \\ \text{sym.} & G_{i;22} \end{bmatrix} = \mathbf{T}_{2\times2;i}^{\mathsf{T}} \mathbf{G}_{i}^{\mathsf{T}} \mathbf{T}_{2\times2;i}$$
(2.33)

where

$$\mathbf{G}_{i}^{\prime} = \begin{bmatrix} G_{xz} & 0\\ 0 & G_{yz} \end{bmatrix} \text{ and } \mathbf{T}_{2\times2;i} = \begin{bmatrix} \cos(\beta_{i}) & \sin(\beta_{i})\\ -\sin(\beta_{i}) & \cos(\beta_{i}) \end{bmatrix}$$
(2.34)

The individual elements then are

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

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

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



# 3. Input Data

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

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

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

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

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

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

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

## 3.1 General Data

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

#### **Design of**

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

#### **Material Model**

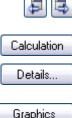
In this section, you select the material model. The following material models are available:

- Orthotropic
- Isotropic
- User-Defined
- Hybrid

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

#### Comment

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



Graphics						
OK						
Cancel						

Material Model	
Orthotropic	•
Orthotropic Isotropic User-Defined	
Isotropic	
User-Defined	
Lubrid	



#### 3.1.1 Ultimate Limit State Tab

Settings Help				
ut Data	1.1 General Data			
<ul> <li>General Data</li> <li>Material Characteristics</li> </ul>	Design of	Materi	ial Model	
- Material Strengths	Surfaces No.:	Othe	otropic 💌	
	1			
	Ultimate Limit State Serviceability Limit St	ale		
	Existing Load Cases	Selected for Design		111
	Qi *LC3	G LC1	Persistent and Transient	<b>2</b>
		Qi LC2	Persistent and Transient	
		C01 1.35*LC1 + 1.5	5°LC2 Persistent and Transient	RF-LAMINATE
				Z
				=
		>>		5
				5
				T
				LL.
				2
				Calculation and desig
				of laminate surfaces
	All (4)		2v 84	
	Comment			
	Calculation Details	Graphics		OK C.

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

#### **Existing Load Cases**

>

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

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

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

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

#### **Selected for Design**

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

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





#### RF-LAMINATE - [Laminate] File Settings Help Input Data General Data Material Characteristics Material Strengths Serviceability Data 1.1 General Data Orthotropic Surfaces No. ~ 🔊 🗙 🗆 Al Ultimate Limit State Serviceability Limit State **RF-LAMINATE** Qi \*LC3 CO1 G LC1 Qi LC2 Characteristic 1.35°LC1 + 1.5°LC2 > 4 Calculation and design of laminate surfaces All (4) ♥ 3 8 **1** Comm 9 🗗 🗟 Graphics Calculation Details... OK Cancel

#### 3.1.2 Serviceability Limit State Tab

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

#### **Existing Load Cases**

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

#### **Selected for Design**

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

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



# 3.2 Material Characteristics

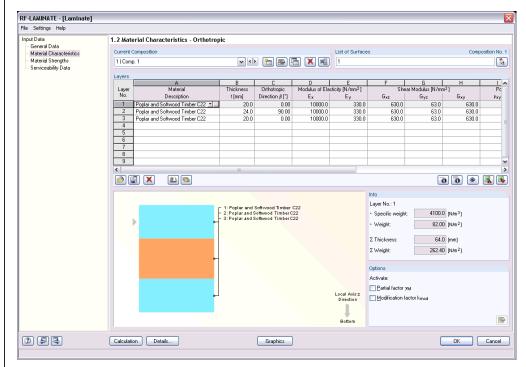


Figure 3.3: Window 1.2 Material Characteristics

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

The following buttons are available here:

Button	Name			
1	Create New Composition			
1	Edit Composition Details			
	Copy Current Composition			
×	Delete Current Composition			
Nun Mun	Delete All Compositions			
5	Select Surfaces			

Table 3.1: Buttons in the window Material Characteristics

10

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

#### 3 Input Data

4	$\checkmark$
DI	ubal

Calculation / Modeling     Calculation Options   Cross laminated timber without glue at narrow sides   Stiffness Reduction Factors For drilling stiffness elements K44: 100 @b [1] Partial Factors for Material Properties Design Shualion: Persistent/transient na: 1.00 @b     Accidental na: 1.00 @b  2 and b 1 10 10 10 10 10 10 10 10 10 10 10 10 1	tails of Composition No. 1		
Consider coupling   □ Cross laminated timber without glue at narrow sides     Stiffness Reduction Factors   For dilling stiffness elements   K3:   1.00 ⊕ [-]     Partial Factors for Material Properties   Design Situation:   • Persistent/transient   ym:   1.00 ⊕ [-]	Calculation / Modeling		
□ Cross laminated timber without glue at narrow sides         Stiffness Reduction Factors         For drilling stiffness elements         K3:       1.00 € h         []         For schear stiffness elements         K44:       1.00 € h         []         Partial Factors for Material Properties         Design Situation:         • Persistent/transient         mail         1.00 € h	Calculation Options		
Stiffness Reduction Factors For drilling stiffness elements K33: 1.00 () [-] For shear stiffness elements K44: 1.00 () [-] For shear stiffness elements K44: 1.00 () [-] Partial Factors for Material Properties Design Situation: • Persistent/transient min. 1.30 () • Accidental min. 1.00 ()	Consider coupling		
For drilling stiffness elements K33: 1.00 ♠ [-] For shear stiffness elements K44: 1.00 ♠ [-] K65: 1.00 ♠ [-] Partial Factors for Material Properties Design Situation: • Persistent/transient 1.30 ♠ • Accidental 1.00 ♠ •	Cross laminated timber withou	t glue at narrow sides	
K33: 1.00 mm   For shear stiffness elements   K44:   1.00 mm   Factors for Material Properties   Design Situation:   • Persistent/transient   * Accidental   * 7m:   1.30 mm	Stiffness Reduction Factors		
For shear stiffness elements K44: 1.00 h [] K66: 1.00 h [] Partial Factors for Material Properties Design Situation: • Persistent/transient min. 1.30 h • Accidental min. 1.00 h			
K44:       1.00 • [-]         K55:       1.00 • [-]         Partial Factors for Material Properties         Design Situation:         • Persistent/transient         • Accidental         • Mr:         1.00 • [-]	K33: 1.00 🕪 [·]		
K65:       1.00 m [·]         Partial Factors for Material Properties         Design Situation:         • Persistent/transient         • Accidental         • ym:         1.00 m	For shear stiffness elements		
K66:       1.00 • []         Partial Factors for Material Properties         Design Situation:         • Persistent/transient         • M:         1.00 • []	K44: 1.00 🗇 [·]		
Partial Factors for Material Properties Design Situation:   Persistent/transient mi: 1.30 @ h  Accidental mi: 1.00 @ h			
Design Situation: • Persistent/transient miles 1.30 melles • Accidental miles 1.00 melles			
<ul> <li>Persistent/transient milling b</li> <li>Accidental milling b</li> </ul>	Partial Factors for Material Prope	rties	
• Accidental 7M: 1.00 2	Design Situation:		
	<ul> <li>Persistent/transient</li> </ul>	-	
	<ul> <li>Accidental</li> </ul>	ум: 1.00 🚓 🕨	

Figure 3.4: Dialog box Details of Composition

#### **Calculation Options**

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

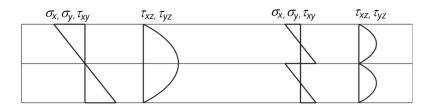


Figure 3.5: Basic stresses of the two-layer plate subjected to bending: with shear coupling of layers on the left, without shear coupling of layers on the right

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

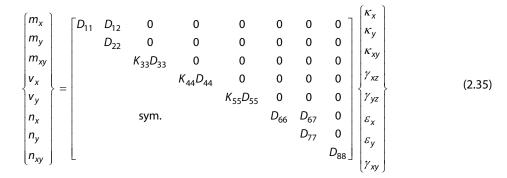
#### **Stiffness Reduction Factors**

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



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

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



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

#### **Partial Factors for Material Properties**

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

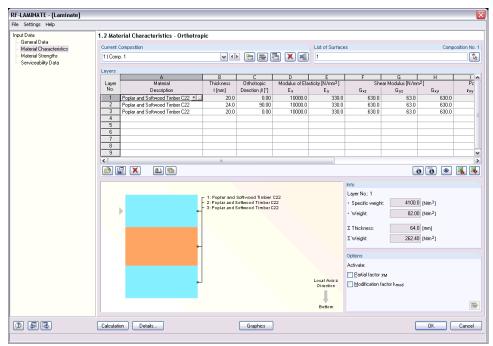
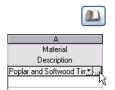


Figure 3.6: Window 1.2 Material Characteristics



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

Filter	Material to Select				
Material category group:	Material Description		Standard	ł	
Timber	🤜 📒 Poplar and Softwood T	imber C14	🔟 EN 3	38:2009-10	
	💳 📕 Poplar and Softwood T	imber C16	💽 EN 3	38:2009-10	
Material category:	Poplar and Softwood T	imber C18	💽 EN 3	38:2009-10	
Softwood Timber	🔽 📃 Poplar and Softwood T	imber C20	💽 EN 3	38:2009-10	
o	Poplar and Softwood T	imber C22	🔝 EN 3	38:2009-10	
Standard group:	Poplar and Softwood T	imber C24	🔟 EN 3	38:2009-10	
💿 EN	兰 📃 Poplar and Softwood T	imber C27	💽 EN 3	38:2009-10	
Standard:	Poplar and Softwood T	imber C30	💽 EN 3	38:2009-10	L
EN 338:2009-10	Poplar and Softwood T	imber C35	💽 EN 3	38:2009-10	
EN 336:2003-10	Poplar and Softwood T 📃	imber C40	🔟 EN 3	38:2009-10	
	Poplar and Softwood T	imber C45	💽 EN 3	38:2009-10	
	Poplar and Softwood T	imber C50	💽 EN 3	38:2009-10	
	Poplar and Softwood T	imber C14 (Perpendicular I	to the 🔝 EN 3	38:2009-10	
🗖 Include invalid	Poplar and Softwood T	imber C16 (Perpendicular I	to the 🔝 EN 3	38:2009-10	G
		1 C10/D I I I	a les pue	00 0000 10	ľ
Favorites only	9 1 2				X
Material Properties		Po	plar and Softwoo	od Timber C22   EN	338:2009-1
🗆 Main Properties					
<ul> <li>Modulus of Elasticity</li> </ul>		E	10000.000	MPa	ĺ
<ul> <li>Shear Modulus</li> </ul>		G	630.000		
<ul> <li>Specific Weight</li> </ul>		γ	4.10	kN/m <sup>3</sup>	
<ul> <li>Coefficient of Thermal Expa</li> </ul>	nsion	α	5.0000E-06	1/K	
<ul> <li>Partial Safety Factor</li> </ul>		ýМ	1.30		
Additional Properties					
<ul> <li>Characteristic Strength for B</li> </ul>	-	fm,k	22.000		
<ul> <li>Characteristic Strength for T</li> </ul>		ft,0,k	13.000		
<ul> <li>Characteristic Strength for T</li> </ul>	ension Perpendicular	ft,90,k	0.400		
<ul> <li>Characteristic Strength for C</li> </ul>	Compression	fc,0,k	20.000	MPa	
<ul> <li>Characteristic Strength for C</li> </ul>	Compression Perpendicular	fc,90,k	2.400	MPa	
<ul> <li>Characteristic Strength for S</li> </ul>	hear/Torsion	fv,k	3.800	MPa	
<ul> <li>Modulus of Elasticity Paralle</li> </ul>	1	E0,mean	10000.000	MPa	
<ul> <li>Modulus of Elasticity Perper</li> </ul>	ndicular	E90,mean	330.000	MPa	
		Gmean	630.000		
<ul> <li>Shear Modulus</li> </ul>					
— Shear Modulus — Density		Øk	34000	kg/m <sup>3</sup>	

Figure 3.7: Material library

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

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



#### 3 Input Data

Selection		Layers			
Producer:		No.	Thickness [mm]	Beta [*]	
Binderholz	~	1 2 3	20.0 26.0 20.0	0.00 90.00 0.00	
Туре:		Sum	66.0		
Thickness:					
BBS 66mm	~				

Figure 3.8: Dialog box Import Layers from Library

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

#### 1.2 Material Characteristics - Orthotropic

Current (	Composition		
1   Comp	o. 1	~ <	• 🛅 🔤 🛛
Layers			
	A	В	С
Layer	Material	Thickness	Orthotropic
No.	Description	t [mm]	Direction β [°]
1	Poplar and Softwood Timber C22 🗾	20.0	0.00
2	Poplar and Softwood Timber C22	24.0	90.00
3	Poplar and Softwood Timber C22	20.0	0.00

#### 1.2 Material Characteristics - Orthotropic

current o	composition		
1   Comp	o. 1	<b>v</b> (	) 🔁 🖻 [
Layers			
	A	В	С
Layer	Material	Thickness	Orthotropic
No.	Description	t [mm]	Direction β [*]
1	Poplar and Softwood Timber C22	20.0	0.00
2	Poplar and Softwood Timber C22 💌	24.0	90.00
3	Poplar and Softwood Timber C22	20.0	0.00

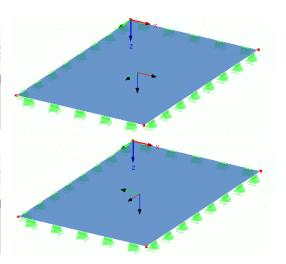


Figure 3.9: Display of the orthotropy direction  $\beta$ 

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

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



	Import Layers from Library	Opens the dialog box Import Layers from Library.
0	Show layer stiffness matrix elements	Displays elements of the stiffness matrix that is explained in Chapter 2.3.
0	Show extended stiffness ma- trix elements	Displays elements of the global stiffness matrix that is explained in Chapter 2.3.
۲	Jump to graphic to change view	Opens the RFEM work window for a graphical check, but does not quit the RF-LAMINATE module.
	Export to Microsoft Excel or OpenOffice.org Calc	Exports the contents of the current table to MS Excel or to the Calc application from the OpenOf- fice.org package $\rightarrow$ Chapter 7.2, page 58.
<b>B</b>	Import from Microsoft Excel or OpenOffice.org Calc	Imports the contents of a MS Excel table, the 1.2 Material Characteristics sheet, or from the Calc application of the OpenOffice.org package to Win- dow 1.2.

Table 3.2: Buttons in the window Material Characteristics

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

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

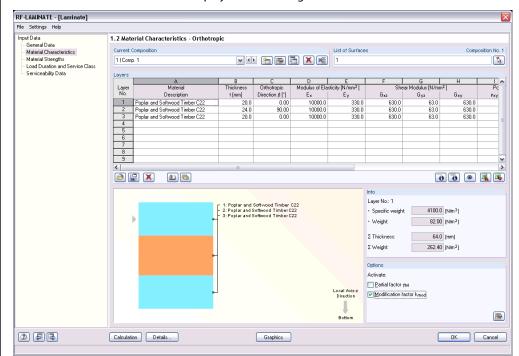


Figure 3.10: Window 1.2 Material Characteristics – the modification factor k<sub>mod</sub>



# 3.3 Material Strengths

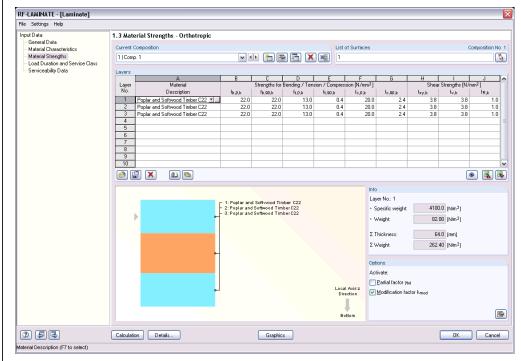


Figure 3.11: Window 1.3 Material Strengths

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

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

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



# 3.4 Load Duration and Service Class

RF-LAMINATE - [Laminate]						
File Settings Help						
Input Data General Data	1.4 Loa	d Duration and Service Class				
- Material Characteristics	Assignin	g of Load Duration Classes to Load Cases / Co	mbinations			Service Class
<ul> <li>Material Strengths</li> <li>Load Duration and Service Class</li> </ul>	Load	A	B	C Load Duration Class	D Factor	<ul> <li>Identical for all surfaces</li> </ul>
Serviceability Data	ing	Description	Loading Type	LDC	kmod	Service class:
,	LC1		Permanent	Permanent 🗾	0.60	1 🗸
	LC2		Imposed	Long-term	0.70	
	C01	1.35'LC1 + 1.5'LC2		Long-term	0.70	O Different
						Note
						Service class 1: Interior
						Temperature of 20°C and the relative humidity of the surrounding air only exceeding 65% for a few weeks per
						year.
						Example:
						Buildings closed from all sides and heated buildings
						Service class 2: Exterior, under cover
						Temperature of 20°C and the relative humidity of the surrounding air only exceeding 85% for a few weeks per year.
						Example:
						Roofed buildings without walls
						Service class 3: Exterior, fully exposed
						Climatic conditions leading to higher moisture contents than in Service Class 2.
						Example:
						Structural members are freely exposed to weather effect
2 5 5	Calculati	ion Details	Graphic	5		OK Cancel

Figure 3.12: Window 1.4 Load Duration and Service Class

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

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

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

15

Assign Surface to Corresponding Service Class	¥
Service Class 1 Surfaces No: 1 🚯 🖉 🥏	Service class 1: Interior Temperature of 20°C and the relative humidity of the surrounding air only exceeding 65% for a few weeks per year. Example: Buildings closed from all sides and heated buildings
Service Class 2 Surfaces No:	Service class 2: Exterior, under cover Temperature of 20°C and the relative humidity of the surrounding air only exceeding 85% for a few weeks per year. Example: Roofed buildings without walls
Service Class 3 Surfaces No.:	Service class 3: Exterior, fully exposed Climatic conditions leading to higher moisture contents than in Service Class 2. Example: Structural members are freely exposed to weather effects
٦	OK Cancel

Figure 3.13: Dialog box Assign Surface to Corresponding Service Class

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

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

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



# 3.5 Serviceability Data

RF-LAMINATE - [Laminate]								
File Settings Help								
Input Data — General Data	1.5 Serviceability Data Assigning of Reference Lengths to Surfaces							
<ul> <li>Material Characteristics</li> <li>Material Strengths</li> <li>Load Duration and Service Class</li> <li>Serviceability Data</li> </ul>	Assigning of R	A A a construction of the service of	В	C I	D	F		
	No.	Ref		Length	Canti	• • • • • • • • • • • • • • • • • • •		
		List of Surfaces	Manual	L [m]	lever	Comment		
	1 1			3.000				
	2							
						٩ 🚯		
9 <b>4 4</b>	Calculation	Details	ſ	Graphics		OK Cance		
	Carcolation	D'Ordany		Graphics		Calce		

Figure 3.14: Window 1.5 Serviceability Data

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

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



# 4. Calculation

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

Before the calculation begins, it is checked whether the global stiffness matrix is positivedefinite, see 9.2.

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

(4.1)

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

# 4.1 Details

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

- Design
- Stresses
- Results

The following buttons are common for all the tabs:

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

Table 4.1: Buttons in the Details



#### 4.1.1 Design Tab

Serviceability Limits (Deflections)					
Type of Combination:			ntilevers 150 🗇 🕨		
• Characteristic L /	300 🗇				
• Frequent L /					
• Quasi-permanent L /	200 📚 🕨	Lo/	100 😍 🕨		
Modification Factor					
Activate					
Load Duration Class (LDC):		1	Service Class 2	3	
<ul> <li>Permanent</li> </ul>	kmod :	1 0.60 🔿 🕨	0.60 🔿 🔪	0.50 🖘	
<ul> <li>Long-term</li> </ul>	kmod :	0.70 😍 🕨	0.70	0.55	
<ul> <li>Medium-term</li> </ul>	kmod :	0.80 😍 🕨	0.80	0.65	
Short-term	kmod :	0.90 🜩 🕨	0.90	0.70	
Instantaneous	kmod :	1.10 😍 🖢	1.10	0.90	
<ul> <li>Short-term / Instantaneous</li> </ul>	kmod :	1.00	1.00	0.80	
· Shoktenn / Instantaneous	K mod .	1.00 💌	1.00 💌 🖻	0.00 💌	

Figure 4.1: Dialog box Details - the Design tab

#### Serviceability Limits (Deflections)

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

#### **Modification Factor**

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



#### 4.1.2 Stresses Tab

Details					
Design Stresses Results					
To Display		Plate Bending Theory			
To Display Top/Bottom Layer	Middle Layer	Plate Bending Theory <ul> <li>Mindlin</li> <li>Kirchhoff</li> </ul> Equivalent Stresses According to (for lsotropic Materials) <ul> <li>Von Mises, Huber, Hencky Shape modification hypothesis</li> <li>Tresca Maximum shear stress criterion</li> <li>Rankine, Lamé Maximum principal stress criterion</li> </ul> Bach, Navier, St. Venant, Poncelet Principal strain criterion			
2 . <u>•</u>					

Figure 4.2: Dialog box Details - the Stresses tab

#### **To Display**

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

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

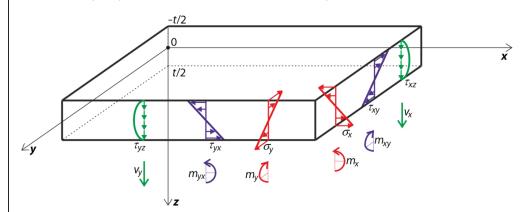


Figure 4.3: Basic stresses and the sign convention for the single layer plate subjected to bending



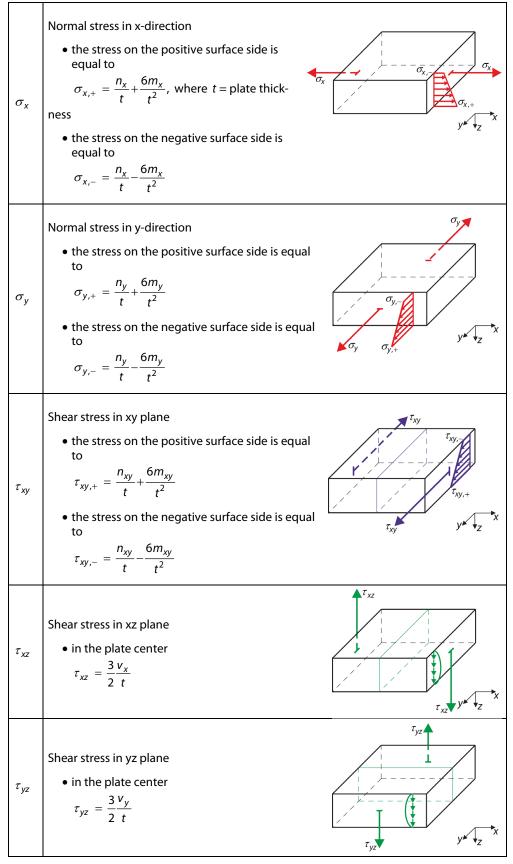


Table 4.2: Basic stresses

#### **4** Calculation

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

$$\boldsymbol{\varepsilon}_{\text{tot}}^{\mathsf{T}} = \left\{ \frac{\partial \varphi_{y}}{\partial x}, -\frac{\partial \varphi_{x}}{\partial y}, \frac{\partial \varphi_{y}}{\partial y} - \frac{\partial \varphi_{x}}{\partial x}, \frac{\partial w}{\partial x} + \varphi_{y}, \frac{\partial w}{\partial y} - \varphi_{x}, \frac{\partial u}{\partial x}, \frac{\partial v}{\partial y}, \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right\}$$
(4.2)

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

r

$$\boldsymbol{\varepsilon}(\boldsymbol{z}) = \begin{cases} \varepsilon_{x} \\ \varepsilon_{y} \\ \gamma_{xy} \end{cases} = \begin{cases} \frac{\partial u}{\partial x} \\ \frac{\partial v}{\partial y} \\ \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \end{cases} + \boldsymbol{z} \begin{cases} \frac{\partial \varphi_{y}}{\partial x} \\ -\frac{\partial \varphi_{x}}{\partial y} \\ \frac{\partial \varphi_{y}}{\partial y} - \frac{\partial \varphi_{x}}{\partial x} \end{cases}$$
(4.3)

h

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

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

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

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

#### **Isotropic material model**

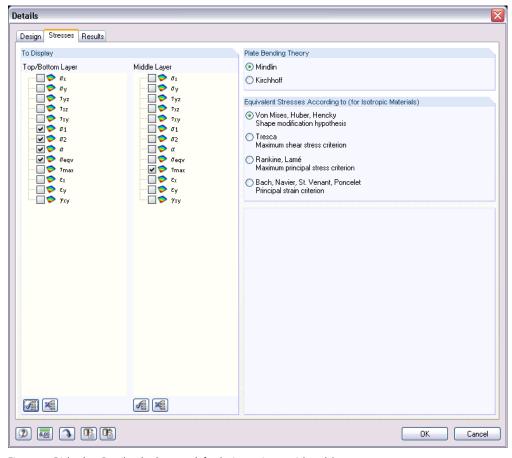


Figure 4.4: Dialog box Details - the Stresses tab for the isotropic material model

– Dlubal

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

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

Table 4.3: Maximum transversal shear stress

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

	Principal stress
$\sigma_1$	$\sigma_1 = \frac{\sigma_x + \sigma_y + \sqrt{\left(\sigma_x - \sigma_y\right)^2 + 4\tau_{xy}^2}}{2}$
	01 =2
	Principal stress
$\sigma_2$	$\sigma_2 = \frac{\sigma_x + \sigma_y - \sqrt{\left(\sigma_x - \sigma_y\right)^2 + 4\tau_{xy}^2}}{2}$
	$\frac{2}{1}$ The angle between the local axis x and the direction of the first principal stress
	y <sub>↑</sub>
	$\alpha = \frac{1}{2} \operatorname{atan2}(2\tau_{xy}, \sigma_x - \sigma_y),  \alpha \in (-90^\circ, 90^\circ)$ $\sigma_{2\gamma}$
	Function atan2 is implemented in RFEM this way $\alpha$
C.	$\begin{cases} \arctan \frac{y}{x} & x > 0 \\ z & \xrightarrow{\alpha} \\ x & \xrightarrow{\alpha} $
α	$\arctan \frac{y}{x} + \pi  y \ge 0, x < 0$
	$\arctan \frac{y}{2} - \pi  y < 0, x < 0$
	$\operatorname{atan2}(y,x) = \begin{cases} \operatorname{arctan} \frac{y}{x} - \pi & y < 0, x < 0 \\ + \frac{\pi}{2} & y > 0, x = 0 \end{cases}$
	-
	$-\frac{\pi}{2} \qquad y < 0, x = 0$
	0   y = 0, x = 0
	Equivalent stress according to von Mises, Huber, Hencky (Shape modification hypothesis)
	$\sigma_{\text{eqv}} = \sqrt{\sigma_x^2 + \sigma_y^2 - \sigma_x \sigma_y + 3\tau_{xy}^2}$
	Equivalent stress according to Tresca
	(Maximum shear stress criterion)
	$\sigma_{\text{eqv}} = \max\left[\sqrt{\left(\sigma_x - \sigma_y\right)^2 + 4\tau_{xy}^2}, \frac{\left \sigma_x + \sigma_y\right  + \sqrt{\left(\sigma_x - \sigma_y\right)^2 + 4\tau_{xy}^2}}{2}\right]$
σ	$\begin{bmatrix} \nabla_{eqv} - \max \left[ \sqrt{(\nabla_x - \nabla_y)} + 4\lambda_{xy}, \frac{1}{2} \right] \end{bmatrix}$
$\sigma_{ m eqv}$	Equivalent stress according to Rankine, Lamé (Maximum principal stress criterion)
	$\sigma_{\text{conv}} = \frac{\left \sigma_{x} + \sigma_{y}\right  + \sqrt{\left(\sigma_{x} - \sigma_{y}\right)^{2} + 4\tau_{xy}^{2}}$
	$\sigma_{\rm eqv} = \frac{1}{2}$
	Equivalent stress according to Bach, Navier, St. Venant, Poncelet (Principal strain criterion)
	$\sigma_{\text{eqv}} = \max\left[\frac{1-\nu}{2}\left \sigma_x + \sigma_y\right  + \frac{1+\nu}{2}\sqrt{\left(\sigma_x - \sigma_y\right)^2 + 4\tau_{xy}^2}, \nu\left \sigma_x + \sigma_y\right \right]$

Table 4.4: Stresses for the isotropic material model



#### Orthotropic material model

To Display		Plate Bending Theory
Top/Bottom Layer	Middle Layer	Mindlin
$\sigma_{\rm X}$	$\sigma_x$	C Kirchhoff
$ \begin{array}{c} & & & & \\ & & & & & \\ & & & & & \\ & & & & & \\ & & & & & \\ & & & & & \\ & & & & & \\ & & & & & \\ & & & & & \\ & & & & & \\ & & & & & \\ & & & & & \\ & & & & & \\ & & & & & \\ & & & & & \\ & & & & & \\ & & & & & \\ & & & & \\ & & & & \\ & & & & \\ & & & & \\ & & & & \\ & & & $	$ \begin{array}{c} \checkmark \qquad 7yz \\ \hline \qquad 7yz \\ \hline \qquad 7xz \\ \hline \qquad 7xy \\ \hline \qquad 9b,0 \\ \hline \qquad 9b,0 \\ \hline \qquad 9b,0 \\ \hline \qquad 9c,(c,0) \\ \hline \ 9c,(c,0) \\ $	Equivalent Stresses According to (for Isotropic Materials) <ul> <li>Von Mises, Huber, Hencky Shape modification hypothesis</li> <li>Tresca Maximum shear stress criterion</li> <li>Rankine, Lamé Maximum principal stress criterion</li> <li>Bach, Navier, St. Venant, Poncelet Principal strain criterion</li> </ul>

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

	Normal stress along the grain $\sigma_y$ $\sigma_{b+t/c,0} = \sigma_x \cos^2 \beta + \tau_{xy} \sin 2\beta + \sigma_y \sin^2 \beta$
$\sigma_{\rm b+t/c,0}$	$y_{i} \xrightarrow{y'_{i}} x_{i}$ $y_{i} \xrightarrow{y'_{i}} x_{i}$ $1 - \text{grain direction}$
$\sigma_{\rm b+t/c.90}$	Normal stress perpendicular to the grain
<sup>o</sup> b+t/c,90	$\sigma_{\rm b+t/c,90} = \sigma_x \sin^2 \beta - \tau_{xy} \sin 2\beta + \sigma_y \cos^2 \beta$
	Tension/compression component of the normal stress along the grain
$\sigma_{\rm t/c,0}$	$\sigma_{t/c,0} = \frac{\sigma_{b+t/c,0(top)} + \sigma_{b+t/c,0(middle)} + \sigma_{b+t/c,0(bottom)}}{3}$
C.	Tension/compression component of the normal stress perpendicular to the grain
$\sigma_{\rm t/c,90}$	$\sigma_{t/c,90} = \frac{\sigma_{b+t/c,90(top)} + \sigma_{b+t/c,90(middle)} + \sigma_{b+t/c,90(bottom)}}{3}$
$\sigma_{ m b,0}$	Bending component of the normal stress along the grain
0,0	$\sigma_{\rm b,0} = \sigma_{\rm b+t/c,0} - \sigma_{\rm t/c,0}$
$\sigma_{ m b,90}$	Bending component of the normal stress perpendicular to the grain
- 0,90	$\sigma_{\rm b,90} = \sigma_{\rm b+t/c,90} - \sigma_{\rm t/c,90}$

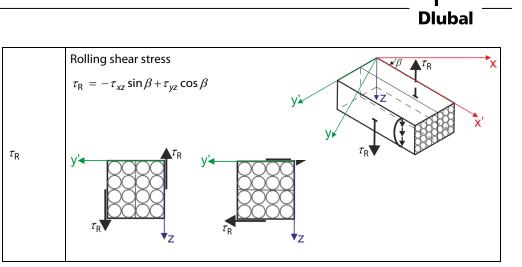


Table 4.5: Stresses for the orthotropic material model

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

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

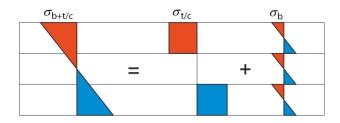


Figure 4.6: Decomposition of the normal stress into the tension/compression component and the bending component

#### **Plate Bending Theory**

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

- Mindlin
- Kirchhoff

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

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

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

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



#### Equivalent Stresses According to (for Isotropic Materials)

You can determine the equivalent stresses in four different ways:

# Equivalent stress according to von Mises, Huber, Hencky (Shape modification hypothesis)

This hypothesis is known as HMH, the VON MISES hypothesis, or as the energy criterion. The equivalent stress according to this hypothesis is calculated by using the relation

$$\sigma_{\text{eqv}} = \sqrt{\sigma_x^2 + \sigma_y^2 - \sigma_x \sigma_y + 3\tau_{xy}^2}$$
(4.7)

### Equivalent stress according to Tresca

#### (Maximum shear stress theory)

This equivalent stress is generally defined by using the relation

$$\sigma_{\text{eqv}} = \max(|\sigma_1 - \sigma_2|, |\sigma_1 - \sigma_3|, |\sigma_2 - \sigma_3|)$$
(4.8)

which is on the condition  $\,\sigma_{3}\,$  =0 , simplified to

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

and the resulting equation

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

#### Equivalent stress according to Rankine, Lamé (Maximum principal stress criterion)

This hypothesis is known as the normal stress hypothesis or as the equivalent stress according to RANKINE. The Rankine's stress is generally defined as the maximum of absolute values of principal stresses

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

which is on the condition  $\,\sigma_{3}\,$  =0 , simplified to

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

and the resulting equation

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

#### Equivalent stress according to Bach, Navier, St. Venant, Poncelet (Principal strain criterion)

The principal deformation hypothesis or the equivalent stress according to BACH. The equivalent strain is defined as the maximum strain that originates in the direction of principal stresses

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

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

$$\sigma_{\text{eqv}} = \max(|\sigma_1 - v\sigma_2|, |\sigma_2 - v\sigma_1|, v|\sigma_1 + \sigma_2|)$$
(4.15)

and the resulting equation

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

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



## 4.1.1 Results Tab

Details	N 1997 - 1997 - 1997 - 1997 - 1997 - 1997 - 1997 - 1997 - 1997 - 1997 - 1997 - 1997 - 1997 - 1997 - 1997 - 1997				
Design Stresses Results					
Display Result Tables	Results in				
✓ 2.1 Max Stress/Ratio by Loading	• FE mesh points				
☑ 2.2 Max Stress/Ratio by Surface	O Grid points				
✓ 2.3 Stresses in All Points					
✓ 3.1 Max Displacements	Internal Forces Diagram Used for Design Apply smoothed internal forces in the defined				
	smooth ranges				
<ul> <li>4.1 Parts List</li> <li>Only for surfaces to be designed</li> </ul>					
O Df all surfaces					
	OK Cancel				

Figure 4.7: Dialog box Details – the Results tab

#### **Display Result Tables**

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

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

#### **Results in**

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

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

#### **Internal Forces Diagram Used for Design**

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



## 4.2 Start Calculation

Calculation

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

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

Calculate  $\rightarrow$  To Calculate....

To Calculate					
Load Cases / C	Combinations / Module Cases Result Tables				
Not Calculated			Selected for Calcu		
No.	Description		No. 🔺	Description	n 🦳 🧖
G LC1 G LC2 G LC3 C01 CA1	1.35°LC1 + 1.5°LC2 RF-LAMINATE - Design of laminate surfaces	<ul><li></li><li></li><li></li><li></li><li></li><li></li><li></li><li></li><li></li><li></li><li></li><li></li><li></li><li></li><li></li><li></li><li></li><li></li><li></li><li></li><li></li><li></li><li></li><li></li><li></li><li></li><li></li><li></li><li></li><li></li><li></li><li></li></ul>			
All	~				
					OK Cancel

Figure 4.8: Dialog box To Calculate in RFEM

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



۲

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

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



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



# 5. Results





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

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

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

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

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

Table 5.1: Buttons in result windows

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



## 5.1 Max Stress/Ratio by Loading

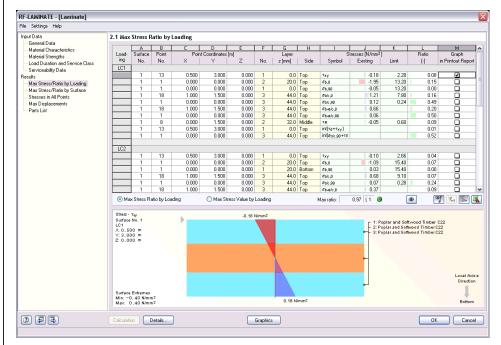


Figure 5.1: Window 2.1 Max Stress/Ratio by Loading

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

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

Individual columns of Table 2.1 are described below.

#### Surface No.

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

#### **Point No.**

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

Details...

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

#### **Point Coordinates**

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



#### Layer

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

#### Stresses

#### **Symbol and Existing**

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

#### Limit

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

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

$\begin{cases} f_{b,d} \\ f_{t,d} \\ f_{c,d} \\ f_{b,0,d} \\ f_{t,0,d} \\ f_{c,0,d} \\ f_{c,90,d} \\ f_{c,90,d} \\ f_{c,90,d} \\ f_{xy,d} \\ f_{v,d} \\ f_{eqv,d} \\ f_{eqv,d} \\ f_{R,d} \end{cases}$	$\rangle = \frac{k_{mod}}{\gamma_M}$	$ \begin{cases} 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_{c,90,k} \\ f_{c,90,k} \\ f_{c,90,k} \\ f_{xy,k} \\ f_{xy,k} \\ f_{eqv,k} \\ f_{eqv,k} \\ f_{R,k} \end{cases} $	
--	--------------------------------------	---	--

(5.1)

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

#### Ratio

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

Max ratio: 0.97 🖾 1 🥹



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

#### Isotropic material model

Stresses [Pa]	Ratios [-]
$\sigma_{x}$	$= \begin{cases} \frac{\sigma_{t/c,x}}{f_{t,d}} + \frac{ \sigma_{b,x} }{f_{b,d}} & \text{if } \sigma_{t/c,x} > 0\\ \frac{ \sigma_{t/c,x} }{f_{c,d}} + \frac{ \sigma_{b,x} }{f_{b,d}} & \text{if } \sigma_{t/c,x} \le 0 \end{cases}$
$\sigma_y$	$= \begin{cases} \frac{\sigma_{t/c,y}}{f_{t,d}} + \frac{ \sigma_{b,y} }{f_{b,d}} & \text{if } \sigma_{t/c,y} > 0\\ \frac{ \sigma_{t/c,y} }{f_{c,d}} + \frac{ \sigma_{b,y} }{f_{b,d}} & \text{if } \sigma_{t/c,y} \le 0 \end{cases}$
$\sigma_1$	$= \begin{cases} \frac{\sigma_1}{f_{t,d}} & \text{if } \sigma_1 > 0\\ \frac{ \sigma_1 }{f_{c,d}} & \text{if } \sigma_1 \le 0 \end{cases}$
$\sigma_2$	$= \begin{cases} \frac{\sigma_2}{f_{t,d}} & \text{if } \sigma_2 > 0\\ \frac{ \sigma_2 }{f_{c,d}} & \text{if } \sigma_2 \le 0 \end{cases}$
$\sigma_{ m eqv}$	$\frac{\left \sigma_{\rm eqv}\right }{f_{\rm eqv,d}}$
$ au_{max}$	$\frac{\left \tau_{\max}\right }{f_{v,d}}$
τ <sub>xz</sub>	$\frac{\left \tau_{xz}\right }{f_{v,d}}$
τ <sub>xy</sub>	$\frac{\left \tau_{xy}\right }{f_{v,d}}$
τ <sub>yz</sub>	$\frac{\left \tau_{yz}\right }{f_{\rm v,d}}$

Table 5.2: Ratios for the isotropic material model



### Orthotropic material model

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

Table 5.3: Ratios for the orthotropic material model



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

$$\begin{cases} \sigma_{b+t/c,0} \\ \sigma_{b+t/c,90} \\ * \end{cases} = \begin{bmatrix} c^2 & s^2 & 2cs \\ s^2 & c^2 & -2cs \\ -cs & cs & c^2 - s^2 \end{bmatrix} \begin{bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{bmatrix}, \quad \begin{cases} \tau_d \\ \tau_R \\ \end{cases} = \begin{bmatrix} c & s \\ -s & c \\ T_{2\times 2} \end{bmatrix} \begin{bmatrix} \tau_{xz} \\ \tau_{yz} \end{bmatrix}$$
(5.2)

or, equivalently, in the non-matrix form

\_

$$\sigma_{b+t/c,0} = c^{2} \sigma_{x} + s^{2} \sigma_{y} + 2cs \tau_{xy}$$

$$\sigma_{b+t/c,90} = s^{2} \sigma_{x} + c^{2} \sigma_{y} - 2cs \tau_{xy}$$

$$\tau_{d} = c \tau_{xz} + s \tau_{yz}$$

$$\tau_{R} = -s \tau_{xz} + c \tau_{yz}$$
(5.3)

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

#### **Graph in Printout Report**

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

## 5.2 Max Stress/Ratio by Surface

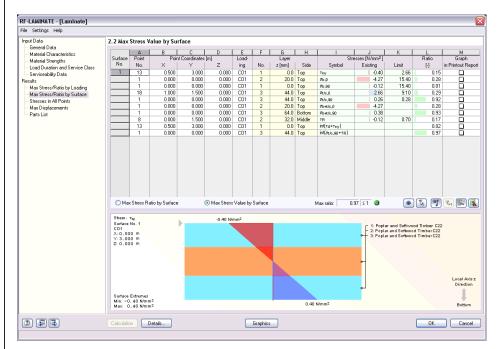


Figure 5.2: Window 2.2 Max Stress/Ratio by Surface

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

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



## 5.3 Stresses in All Points

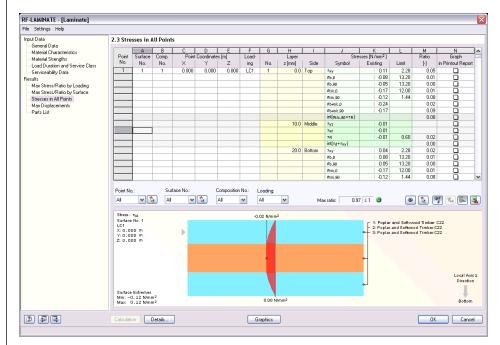


Figure 5.3: Window 2.3 Stresses in All Points

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

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

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



Details...

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



## 5.4 Max Displacements

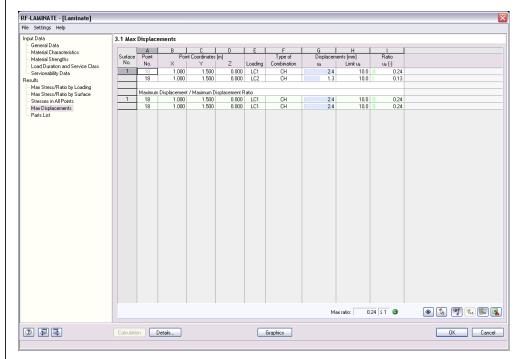


Figure 5.4: Window 3.1 Max Displacements

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

The data is listed according to individual surfaces.

#### Displacements

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

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

#### Ratio

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

Max ratio: 0.24 🖸 🥹



## 5.5 Parts List

jile <u>S</u> ettings <u>H</u> elp										
Input Data	4.1 Parts Li	4.1 Parts List								
<ul> <li>General Data</li> <li>Material Characteristics</li> </ul>		A	B	C	D	E	F	G		
- Material Strengths	Surface No.	Material Description	Thickness t [mm]	No. of	Area [m <sup>2</sup> ]	Coating	Volume [m <sup>3</sup> ]	Weight		
- Load Duration and Service Class	1	Poplar and Softwood Timber C22	t (mm) 20.0	Layers 2	[m²] 6.000	[m <sup>2</sup> ] 12.000	[m <sup>3</sup> ] 0.240	[t] 0.098		
- Serviceability Data Results		Poplar and Softwood Timber C22	24.0	1	6.000	0.000	0.240	0.059		
Max Stress/Ratio by Loading	Σ		64.0	3	6.000	12.000	0.384	0.157		
Max Stress/Ratio by Surface	ΣTotal				6.000	12.000	0.384	0.157		
<ul> <li>Stresses in All Points</li> <li>Max Displacements</li> </ul>	2100				0.000	12.000	0.304	0.157		
Parts List										
								۹	1	F
<b>9 4 5</b>									ОК	Canc

Figure 5.5: Window 4.1 Parts List

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

#### Surface No.

This column contains the numbers of individual surfaces.

#### **Material Description**

Surfaces are classified according to materials.

#### Thickness

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

#### No. of Layers

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

#### Area

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

#### Coating

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

#### Volume

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



### Weight

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

### Total

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



# 6. Printout

# 6.1 Printout Report

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

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

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

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

## 6.2 Printing RF-LAMINATE Graphics

## 6.2.1 Results on the Model in RFEM

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

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

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

#### $\textbf{File} \rightarrow \textbf{Print Graphic...}$

or by clicking the relevant button in the toolbar.



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

Then, the following dialog box appears.

Graphic Printout							
General Options Color Spectrum Factors							
Graphic Picture         Directly to a printer         To a printout report:         PR1         To the Clipboard	Window To Print <ul> <li>Current only</li> <li>More</li> <li>Mass print</li> </ul>	Oraphic Size					
Graphic Picture Size and Rotation         ✓ Use whole page width         O Use whole page height         ④ Height:       50 🐑 [% of page]	Options Show results for selected diagram Lock graphic picture (wit	hout update)					
Rotation: 0 (1) Header of Graphic Picture RF-LAMINATE - Surfaces Tau-xy, CA1, Isometric							
D		OK Cancel					

Figure 6.2: Dialog box Graphic Printout - the General tab

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

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

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

Graphic Printout	_			. 🛛
Properties Options Color Spectrum	Factors			
Script	Symbols		Frame	
Proportional	<ul> <li>Proportional</li> </ul>		🔘 None	
<ul> <li>Constant</li> </ul>	🔘 Constant		<ul> <li>Framed</li> </ul>	
Factor: 1	Factor: 1 🐑		🔲 Title box	
Print Quality		Color		
O Standard (max 1000 x 1000 Pixels)	)	O Grayscal	e	
Maximum (max 5000 x 5000 Pixels)	)	🔘 Texts an	d lines in black	
<ul> <li>User-defined</li> </ul>		All colore	d	
Max number of pixels:	1000 牵			
D			OK	Cancel
gure 6.3: Dialog box Graphic Pri	ntout - the Options to	ab		

RF-LAMINATE - Surfa
 Remove from Printout Report
 Start with New Page
 Selection...
 Properties...



## 6.2.2 Stress Diagrams

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

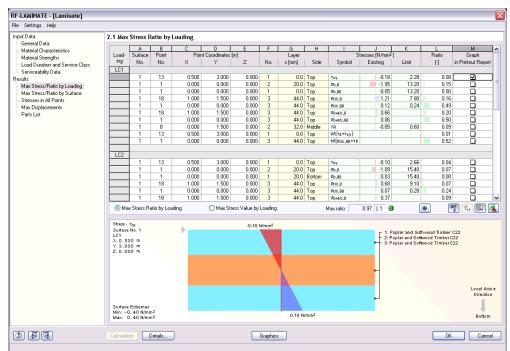


Figure 6.4: Window 2.1 Max Stress/Ratio by Loading

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

<b>•</b> 4.1	PARTS LIST						
Part	Material Description	Thickness	Layer	Area	Coating	Volume	Weight
No.		t [mm]	Count	[m <sup>2</sup> ]	[m²]	[m³]	[t]
1	Poplar and Softwood Timber C22	20.0	2	6.000	12.000	0.240	0.098
	Poplar and Softwood Timber C22	24.0	1	6.000	0.000	0.144	0.059
Σ		64.0	3	6.000	12.000	0.384	0.157
Σ Total	1	1		6.000	12.000	0.384	0.157

#### 4.2 STRESS DIAGRAMS

I

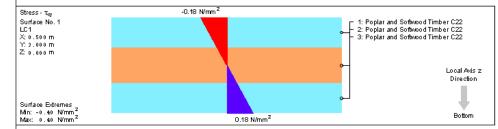


Figure 6.5: Stress diagrams in the printout report



# 7. General Functions

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

# 7.1 Units and Decimal Places

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

#### Settings $\rightarrow$ Units and Decimal Places....

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

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

Units and Decimal Places						
Program / Module	Input Data Results					
PLATE-BUCKLING						
RF-CONCRETE Surfac	Sizes			Material Characteristics		
		Unit	Dec. places		Unit	Dec. places
RF-CONCRETE Colum	Lengths:	m 🗸	3 📚	E-Modules, strengths:	N/mm^2 🔽	1 🚭 🖣
RF-PUNCH	Thicknesses:	mm 💌	1 🚭 🖣	Specific weights:	N/m^3 🗸	1.
RF-TIMBER Pro	Thicknesses:	mm 💌	· •	Specific weights:	N/m 3 🖤	
RF-TIMBER				Surface weights:	N/m^2 💌	2 🚭 🖣
RF-DYNAM RF-END-PLATE	Non-Dimensional			Angles:	• 🗸	2 🚔 🖣
RF-END-PLATE				3 T		×
BE-FRAME-JOINT Pro	Factors:	- 🗸	2 🚔	Limit strengths:	N/mm^2 💌	1 🤤
RF-DSTV				Thermal expansion coef.:	1/K 💌	1 🚭 🖣
BF-DOWEL				Poisson's ratios:		
- BF-HSS				Poisson's ratios:	· ×	3 🌩 🖣
RF-FOUNDATION				Stiffness Matrix Elements		
				Stiffness Matrix Elements		
RF-STABILITY				Bending:	kNm 💌	1 🚔
- RF-DEFORM				Shear, membrane:	kN/m 💌	1 🚔
- RF-MOVE						
BF-IMP				Eccentric effects:	kNm/m 💌	1 🤤
- RF-SOILIN				Layers:	kN/m^2 🔽	3 🗢
- RF-GLASS						
RF-LAMINATE RF-TOWER Structure						
- RF-TOWER Equipmen						
RF-TOWER Loading						
BF-TOWER Effective L						
RF-TOWER Design						
RF-INFLUENCE						
2 🛛 🖹 🖺					ОК	Cancel

Figure 7.1: Dialog box Units and Decimal Places



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



## 7.2 Export of Results

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

#### Clipboard

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

#### **Printout report**

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

File  $\rightarrow$  Export to RTF....

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

#### Excel / OpenOffice

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

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

File  $\rightarrow$  Export Tables....

The following dialog box for the data export opens:

Export - MS Excel	
Table Parameters	Application
✓ With table header	O Microsoft Excel
Only marked rows	⊙ OpenOffice.org Calc
	CSV file format
Transfer Parameters	
Export table to active workbook	
Export table to active worksheet	
Rewrite existing worksheet	
Selected Tables	
<ul> <li>Active table</li> </ul>	Export tables with details
◯ All tables	
🗹 Input tables	
Result tables	
	OK Cancel

Figure 7.2: Dialog box Export - MS Excel

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

X	🚽 🤊	0 - 0	Ŧ			List1	- M	icrosoft	Excel				-	• 23
Fi	le	Home	Insert	Page L	ayout	Formulas		Data	Review	View A	dd-Ins	~	<b>?</b> - 1	er XX
Pas	<mark>٦</mark> (°,	Calibi	<u> </u>	× 8 × A A A ×		<b>■</b> ≣ ≫,•		ext <b>9</b> - % 30 - 300	' st	A Inser yles ▼ Form	te - 💽 -	Sort & Fir		
Clipt	ooard T		Font	G.	Alignr			Number	Gi .	Cells		Editing		
	B1	16	- (	n	<i>f</i> <sub>x</sub> 1									~
	А	В	С	D	E	F	G	Н	1	J	К	L	М	Ę
1	Load-	Surface	FE Mesh	Point	Coordinate	es [m]		Layer		Stre	esses [N/mm²	]	Ratio	
2	ing	No.	Point No.	x	Y	z	No.	z [mm]	Side	Symbol	Existing	Limit	[-]	
3	LC1													
4		1	1	0,000	0,000	0,000	З	64,0	bottom	τ <sub>xy</sub>	-0,24	1,44	0,17	
5		1	62	0,000	0,750	0,000	2	32,0	middle	τ.,	0,09	1,44	0,06	
6		1	1229	0,500	0,200	0,000	2	20,0	top	$\sigma_{b,o}$	-0,51	13,20	0,04	
7		1	1251	0,500	0,750	0,000	2	20,0	top	σ <sub>6,90</sub>	-0,02	13,20	0,00	
8		1	1251	0,500	0,750	0,000	з	44,0	top	$\sigma_{t/c,0}$	1,07	7,80	0,14	
9		1	1230	0,500	0,225	0,000	1	0,0	top	σ <sub>t/c,90</sub>	-0,03	1,44	0,02	
10		1	1251	0,500	0,750	0,000	з	64,0	bottom	σ <sub>b+t/c,0</sub>	1,56		0,17	
11		1	1230	0,500	0,225	0,000	1	0,0	top	σ <sub>b+t/c,so</sub>	-0,05		0,03	-
I4 ∢ Rea	A -	2.1 Max	Stresse	s by Loa	ding 🥂	1/			J	▲ [	III 100 %	Θ		►   +

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

# 7.3 Keyboard Shortcuts

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

Help
Next Window
Previous Window
Selection in Tables
Copy the Last Table Row
Calculator
Copy a Table Row to Next Row
Copy to the Clipboard
Find in the Table
Find and Replace in the Table
Insert a Row in the Table
Go to Row with a Given Number
Delete Table Rows
Save Data
Insert Data from the Clipboard
Exclude from Table
Empty the Current Table Row
Undo

Table 7.1: Keyboard shortcuts



# 8. Examples

Several examples are introduced in the following chapter.

# 8.1 Calculation of Stiffness Matrix Elements

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

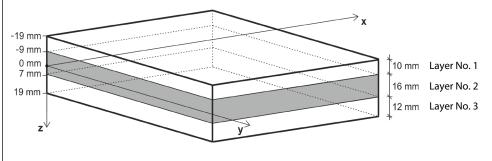


Figure 8.1: Layer scheme

La	ayers											
		A	В	С	D	E	F	G	Н		1	K
	Layer	Material	Thickness	Orthotropic	Modulus of Ela	asticity [N/mm <sup>2</sup> ]	Shear	Modulus (N	/mm <sup>2</sup> ]	Poisson's	Ratio [-]	Specific Weight
	No.	Description	t (mm)	Direction β [*]	Ex	Ey	Gxz	Gyz	Gxy	Vxy	₽ух	γ [N/m <sup>3</sup> ]
	1	Poplar and Coniferous Timber C16	10.0	0.00	8000.0	270.0	500.0	50.0	500.0	0.200	0.007	3700.0
	2	Coniferous Timber C14	16.0	90.00	7000.0	230.0	440.0	44.0	440.0	0.200	0.007	5000.0
	3	Poplar and Coniferous Timber C16	12.0	0.00	8000.0	270.0	500.0	50.0	500.0	0.200	0.007	3700.0

Figure 8.2: Table 1.2 Material Characteristics

At first, you calculate the stiffness matrices of individual layers

$$\mathbf{d}'_{i} = \begin{bmatrix} d'_{i;11} & d'_{i;12} & 0\\ & d'_{i;22} & 0\\ \text{sym.} & d'_{i;33} \end{bmatrix} = \begin{bmatrix} \frac{E_{i;x}}{1 - v_{i;xy}^{2} \frac{E_{i;y}}{E_{i;x}}} & \frac{V_{i;xy}E_{i;y}}{1 - v_{i;xy}^{2} \frac{E_{i;y}}{E_{i;x}}} & 0\\ & \frac{E_{i;y}}{1 - v_{i;xy}^{2} \frac{E_{i;y}}{E_{i;x}}} & 0\\ & \frac{1 - v_{i;xy}^{2} \frac{E_{i;y}}{E_{i;x}}}{E_{i;x}} \end{bmatrix} \quad i = 1, ..., n$$
(8.1)

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

#### 8 Examples

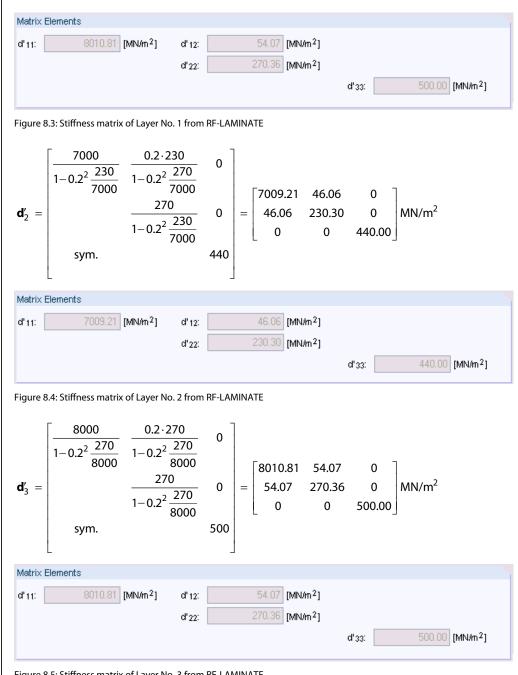


Figure 8.5: Stiffness matrix of Layer No. 3 from RF-LAMINATE

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

$$\mathbf{d}_{1} = \mathbf{d}_{1}' = \begin{bmatrix} 8010.81 & 54.07 & 0\\ 54.07 & 270.36 & 0\\ 0 & 0 & 500.00 \end{bmatrix} \text{MN/m}^{2}$$
$$\mathbf{d}_{3} = \mathbf{d}_{3}' = \begin{bmatrix} 8010.81 & 54.07 & 0\\ 54.07 & 270.36 & 0\\ 0 & 0 & 500.00 \end{bmatrix} \text{MN/m}^{2}$$

### 8 Examples

ı Dlubal

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

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

where

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

The individual elements then are

$$\begin{aligned} d_{i;11} &= c^4 d'_{i;11} + 2c^2 s^2 d'_{i;12} + s^4 d'_{i;22} + 4c^2 s^2 d'_{i;33} \\ d_{i;12} &= c^2 s^2 d'_{i;11} + s^4 d'_{i;12} + c^4 d'_{i;12} + c^2 s^2 d'_{i;22} - 4c^2 s^2 d'_{i;33} \\ d_{i;13} &= c^3 s d'_{i;11} + c^3 s'_{i;12} - c^3 s d'_{i;22} - 2c^3 s d'_{i;33} + 2cs^3 d'_{i;33} \\ d_{i;22} &= s^4 d'_{i;11} + 2c^2 s^2 d'_{i;12} + c^4 d'_{i;22} + 4c^2 s^2 d'_{i;33} \\ d_{i;23} &= cs^3 d'_{i;11} + c^3 s d'_{i;12} - c^3 s d'_{i;22} - 2c^3 s d'_{i;33} - 2cs^3 d'_{i;33} \\ d_{i;33} &= c^2 s^2 d'_{i;11} + 2c^2 s^2 d'_{i;12} + c^2 s^2 d'_{i;22} + (c^2 - s^2)^2 d'_{i;33} \\ c &= \cos 90^\circ = 0, \ s = \sin 90^\circ = 1 \\ d_{2;11} &= 0^4 \cdot 7009.21 + 2 \cdot 0^2 \cdot 1^2 \cdot 46.06 + 1^4 \cdot 230.30 + 4 \cdot 0^2 \cdot 1^2 \cdot 440 = 230.30 \ \text{MN/m}^2 \\ d_{2;12} &= 0^2 \cdot 1^2 \cdot 7009.21 + 1^4 \cdot 46.06 + 0^4 \cdot 46.06 + 0^2 \cdot 1^2 \cdot 230.30 - 4 \cdot 0^2 \cdot 1^2 \cdot 440 = 46.06 \ \text{MN/m}^2 \\ d_{2;13} &= 0^3 \cdot 1 \cdot 7009.21 + 0 \cdot 1^3 \cdot 46.06 - 0^3 \cdot 1 \cdot 46.06 - 0 \cdot 1^3 \cdot 230.30 - 2 \cdot 0^3 \cdot 1 \cdot 440 + 2 \cdot 0 \cdot 1^3 \cdot 440 = \\ &= 0 \ \text{MN/m}^2 \\ d_{2;22} &= 1^4 \cdot 7009.21 + 2 \cdot 0^2 \cdot 1^2 \cdot 46.06 + 0^4 \cdot 230.30 + 4 \cdot 0^2 \cdot 1^2 \cdot 440 = 7009.21 \ \text{MN/m}^2 \\ d_{2;23} &= 0 \cdot 1^3 \cdot 7009.21 + 0^3 \cdot 1 \cdot 46.06 - 0 \cdot 1^3 \cdot 46.06 - 0^3 \cdot 1 \cdot 230.30 + 2 \cdot 0^3 \cdot 1 \cdot 440 - 2 \cdot 0 \cdot 1^3 \cdot 440 = \\ &= 0 \ \text{MN/m}^2 \\ d_{2;33} &= 0^2 \cdot 1^2 \cdot 7009.21 - 2 \cdot 0^2 \cdot 1^2 \cdot 46.06 + 0^2 \cdot 1^2 \cdot 230.30 + (0^2 - 1^2)^2 \ 440 = 440.00 \ \text{MN/m}^2 \end{aligned}$$

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

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

Matrix	Elements in Surface Axis Syste	em			
d 11:	230.30 [MN/m <sup>2</sup> ]	d 12:	46.06 [MN/m <sup>2</sup> ]	d 13:	0.00 [MN/m <sup>2</sup> ]
		d 22:	7009.21 [MN/m <sup>2</sup> ]	d 23:	0.00 [MN/m <sup>2</sup> ]
				d33:	440.00 [MN/m <sup>2</sup> ]

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

. Dlubal

(8.4)

## 8.1.1 Shear Coupling of Layers Is Considered

The global stiffness matrix has the form

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

Stiffness matrix elements (bending and torsion)

$$D_{11} = \sum_{i=1}^{n} \frac{z_{i;\max}^3 - z_{i;\min}^3}{3} d_{i;11} \qquad D_{12} = \sum_{i=1}^{n} \frac{z_{i;\max}^3 - z_{i;\min}^3}{3} d_{i;12} \qquad D_{13} = \sum_{i=1}^{n} \frac{z_{i;\max}^3 - z_{i;\min}^3}{3} d_{i;13}$$
$$D_{22} = \sum_{i=1}^{n} \frac{z_{i;\max}^3 - z_{i;\min}^3}{3} d_{i;22} \qquad D_{23} = \sum_{i=1}^{n} \frac{z_{i;\max}^3 - z_{i;\min}^3}{3} d_{i;23}$$
$$D_{33} = \sum_{i=1}^{n} \frac{z_{i;\max}^3 - z_{i;\min}^3}{3} d_{i;33}$$

$$D_{11} = \frac{(-9 \cdot 10^{-3})^3 - (-19 \cdot 10^{-3})^3}{3} 8010.81 \cdot 10^3 + \frac{(7 \cdot 10^{-3})^3 - (-9 \cdot 10^{-3})^3}{3} 230.30 \cdot 10^3 + \frac{(19 \cdot 10^{-3})^3 - (7 \cdot 10^{-3})^3}{3} 8010.81 \cdot 10^3 = 33.85 \text{ kNm}$$

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

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

$$D_{22} = \frac{(-9 \cdot 10^{-3})^3 - (-19 \cdot 10^{-3})^3}{3} 270.36 \cdot 10^3 + \frac{(7 \cdot 10^{-3})^3 - (-9 \cdot 10^{-3})^3}{3} 7009.21 \cdot 10^3 + \frac{(19 \cdot 10^{-3})^3 - (7 \cdot 10^{-3})^3}{3} 270.36 \cdot 10^3 = 3.64 \text{ kNm}$$

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

$$D_{33} = \frac{(-9 \cdot 10^{-3})^3 - (-19 \cdot 10^{-3})^3}{3} 500 \cdot 10^3 + \frac{(7 \cdot 10^{-3})^3 - (-9 \cdot 10^{-3})^3}{3} 440.00 \cdot 10^3 + \frac{(19 \cdot 10^{-3})^3 - (7 \cdot 10^{-3})^3}{3} 500 \cdot 10^3 = 2.26 \text{ kNm}$$

Program RF-LAMINATE © 2013 Dlubal Software GmbH

+

#### Stiffness matrix elements (eccentricity effects)

$$D_{16} = \sum_{i=1}^{n} \frac{z_{i,max}^{2} - z_{i,min}^{2}}{2} d_{i;11} \qquad D_{17} = \sum_{i=1}^{n} \frac{z_{i,max}^{2} - z_{i,min}^{2}}{2} d_{i;12} \qquad D_{18} = \sum_{i=1}^{n} \frac{z_{i,max}^{2} - z_{i,min}^{2}}{2} d_{i;13} \\ D_{27} = \sum_{i=1}^{n} \frac{z_{i,max}^{2} - z_{i,min}^{2}}{2} d_{i;22} \qquad D_{28} = \sum_{i=1}^{n} \frac{z_{i,max}^{2} - z_{i,min}^{2}}{2} d_{i;23} \\ D_{38} = \sum_{i=1}^{n} \frac{z_{i,max}^{2} - z_{i,min}^{2}}{2} d_{i;33} \\ D_{16} = \frac{(-9 \cdot 10^{-3})^{2} - (-19 \cdot 10^{-3})^{2}}{2} 8010.81 \cdot 10^{3} + \frac{(7 \cdot 10^{-3})^{2} - (-9 \cdot 10^{-3})^{2}}{2} 230.30 \cdot 10^{3} + \frac{(19 \cdot 10^{-3})^{2} - (-19 \cdot 10^{-3})^{2}}{2} 8010.81 \cdot 10^{3} = 124.49 \text{ kNm/m} \\ D_{17} = \frac{(-9 \cdot 10^{-3})^{2} - (-19 \cdot 10^{-3})^{2}}{2} 54.07 \cdot 10^{3} = 0.13 \text{ kNm/m} \\ D_{18} = \frac{(-9 \cdot 10^{-3})^{2} - (-19 \cdot 10^{-3})^{2}}{2} 0 \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} = 0 \text{ kNm/m} \\ D_{27} = \frac{(-9 \cdot 10^{-3})^{2} - (-19 \cdot 10^{-3})^{2}}{2} 270.36 \cdot 10^{3} + \frac{(7 \cdot 10^{-3})^{2} - (-9 \cdot 10^{-3})^{2}}{2} 7009.21 \cdot 10^{3} + \frac{(19 \cdot 10^{-3})^{2} - (-19 \cdot 10^{-3})^{2}}{2} 270.36 \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} - (-19 \cdot 10^{-3})^{2}}{2} 0 \cdot 10^{3} + \frac{(19 \cdot 10^{-3})^{2} - (-19 \cdot 10^{-3})^{2}}{2} 0 \cdot 10^{3} = 0 \text{ kNm/m} \\ D_{28} = \frac{(-9 \cdot 10^{-3})^{2} - (-19 \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} = 0 \text{ kNm/m} \\ D_{38} = \frac{(-9 \cdot 10^{-3})^{2} - (-19 \cdot 10^{-3})^{2}}{2} 500 \cdot 10^{3} + \frac{(7 \cdot 10^{-3})^{2} - (-9 \cdot 10^{-3})^{2}}{2} 440.00 \cdot 10^{3} + \frac{(19 \cdot 10^{-3})^{2} - (-19 \cdot 10^{-3})^{2}}{2} 500 \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} = 0.96 \text{ kNm/m} \\ D_{38} = \frac{(-9 \cdot 10^{-3})^{2} - (-19 \cdot 10^{-3})^{2}}{2} 500 \cdot 10^{3} = 0.96 \text{ kNm/m} \\ D_{38} = \frac{(-9 \cdot 10^{-3})^{2} - (-10 \cdot 10^{-3})^{2}}{2} 500 \cdot 10^{3} = 0.96 \text{ kNm/m} \\ D_{38} = \frac{(-9 \cdot 10^{-3})^{2} - (-10 \cdot 10^{-3})^{2}}{2} 500 \cdot 10^{3} = 0.96 \text{ kNm/m} \\ D_{38} = \frac{(-9 \cdot 10^{-3})^{2}$$

4 Diubal

Stiffness matrix elements (membrane)

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

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

$$D_{68} = \sum_{i=1}^{n} t_i d_{i;13}$$

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

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

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

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

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

$$D_{68} = 10 \cdot 10^{-3} \cdot 0.10^3 + 16 \cdot 10^{-3} \cdot 0.10^3 + 12 \cdot 10^{-3} \cdot 0.10^3 = 0 \text{ N/m}$$

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

$$D_{78} = 10 \cdot 10^{-3} \cdot 0.10^3 + 16 \cdot 10^{-3} \cdot 440 \cdot 10^3 + 12 \cdot 10^{-3} \cdot 500 \cdot 10^3 = 18040 \text{ N/m}$$

#### Stiffness matrix elements (shear)

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

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

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

where

$$\mathbf{G}_{i} = \begin{bmatrix} G_{i;11} & G_{i;12} \\ \text{sym.} & G_{i;22} \end{bmatrix} = \mathbf{T}_{2\times2;i}^{\mathsf{T}} \mathbf{G}_{i}^{\mathsf{T}} \mathbf{T}_{2\times2;i}$$
(8.7)

where

$$\mathbf{G}_{i}^{\prime} = \begin{bmatrix} G_{xz} & 0\\ 0 & G_{yz} \end{bmatrix} \text{ and } \mathbf{T}_{2\times2;i} = \begin{bmatrix} \cos(\beta_{i}) & \sin(\beta_{i})\\ -\sin(\beta_{i}) & \cos(\beta_{i}) \end{bmatrix}$$
(8.8)

The individual elements then are

$$\begin{aligned} G_{i;11} &= c^2 G_{i;xz} + s^2 G_{i;yz} \\ G_{i;12} &= cs G_{i;xz} - cs G_{i;yz} \\ G_{i;22} &= s^2 G_{i;xz} + c^2 G_{i;yz} \text{, where } c = \cos(\beta_i) \text{, } s = \sin(\beta_i) \end{aligned}$$

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

 $D_{44} = 5000.00 \text{ kN/m}$   $D_{55} = 5866.67 \text{ kN/m}$   $\mathbf{G}_{1} = \mathbf{G}_{1}' = \begin{bmatrix} 500 & 0 \\ 0 & 50 \end{bmatrix}$   $\mathbf{G}_{3} = \mathbf{G}_{3}' = \begin{bmatrix} 500 & 0 \\ 0 & 50 \end{bmatrix}$   $\mathbf{G}_{2}' = \begin{bmatrix} 440 & 0 \\ 0 & 44 \end{bmatrix}, \ c = \cos 90^{\circ} = 0, \ s = \sin 90^{\circ} = 1$ 

 $G_{2.11} = 0^2 \cdot 440 + 1^2 \cdot 44 = 44$  MPa

 $G_{2;12} = 0 \cdot 1 \cdot 440 - 0 \cdot 1 \cdot 44 = 0 \text{ MPa}$   $G_{2;22} = 1^2 \cdot 440 + 0^2 \cdot 44 = 440 \text{ MPa}$   $G_2 = \begin{bmatrix} 44 & 0 \\ 0 & 440 \end{bmatrix}$   $\max\left(\frac{5}{6}500 \cdot 10; \frac{5}{6}44 \cdot 16; \frac{5}{6}500 \cdot 12\right) \le D_{44} \le \frac{5}{6}\max(500; 44; 500) \cdot (10 + 16 + 12)$   $5000 \le 5000 \le 15833$   $\max\left(\frac{5}{6}50 \cdot 10; \frac{5}{6}440 \cdot 16; \frac{5}{6}50 \cdot 12\right) \le D_{55} \le \frac{5}{6}\max(50; 440; 59) \cdot (10 + 16 + 12)$   $5866.67 \le 5866.67 \le 13933.33$ 

#### **Global stiffness matrix**

	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
<b>D</b> =				5000.00	0	0	0	0
<b>D</b> =					5866.67	0	0	0
			sym.			179923	1927	0
							118095	0
	L							18040

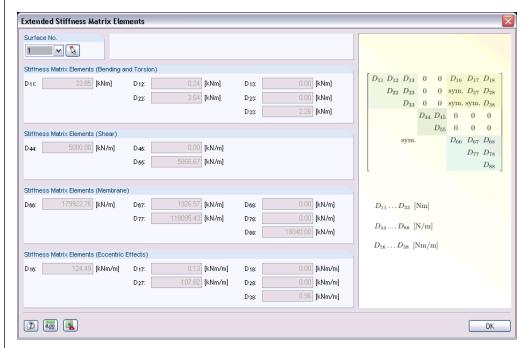


Figure 8.7: Dialog box Extended Stiffness Matrix Elements from RF-LAMINATE – with shear coupling of layers

## 8.1.2 Shear Coupling of Layers Is Not Considered

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

$$\mathbf{D} = \begin{bmatrix} D_{11} & D_{12} & 0 & 0 & 0 & 0 & 0 & 0 \\ D_{22} & 0 & 0 & 0 & 0 & 0 & 0 \\ D_{33} & 0 & 0 & 0 & 0 & 0 \\ D_{44} & 0 & 0 & 0 & 0 \\ D_{55} & 0 & 0 & 0 \\ Sym. & D_{66} & D_{67} & 0 \\ D_{77} & 0 \\ D_{88} \end{bmatrix}$$

(8.9)

#### Stiffness matrix elements (bending and torsion)

$$D_{11} = \sum_{i=1}^{n} \frac{t_i^3}{12} d_{i;11} \qquad D_{12} = \sum_{i=1}^{n} \frac{t_i^3}{12} d_{i;12}$$
$$D_{22} = \sum_{i=1}^{n} \frac{t_i^3}{12} d_{i;22}$$

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

$$\mathbf{d}_{1} = \begin{bmatrix} 8010.81 & 54.07 & 0 \\ 54.07 & 270.36 & 0 \\ 0 & 0 & 500.00 \end{bmatrix} \text{MN/m}^{2}$$

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

$$\mathbf{d}_{3} = \begin{bmatrix} 8010.81 & 54.07 & 0 \\ 54.07 & 270.36 & 0 \\ 0 & 0 & 500.00 \end{bmatrix} \text{MN/m}^{2}$$

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

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

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

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



Stiffness matrix elements (membrane)

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

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

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

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

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

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

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

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

Stiffness matrix elements (shear)

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

where

$$\mathbf{G}_{i} = \begin{bmatrix} G_{i;11} & G_{i;12} \\ \text{sym.} & G_{i;22} \end{bmatrix} = \mathbf{T}_{2\times2;i}^{\mathsf{T}} \mathbf{G}_{i}' \mathbf{T}_{2\times2;i}, \text{ where } \mathbf{G}_{i}' = \begin{bmatrix} G_{xz} & 0 \\ 0 & G_{yz} \end{bmatrix} \text{ and}$$
$$\mathbf{T}_{2\times2;i} = \begin{bmatrix} \cos(\beta_{i}) & \sin(\beta_{i}) \\ -\sin(\beta_{i}) & \cos(\beta_{i}) \end{bmatrix}$$

The individual elements then are

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

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

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

$$G_1 = G_1' = \begin{bmatrix} 500 & 0 \\ 0 & 50 \end{bmatrix}$$

$$G_3 = G_3' = \begin{bmatrix} 500 & 0 \\ 0 & 50 \end{bmatrix}$$

$$G_2' = \begin{bmatrix} 440 & 0 \\ 0 & 44 \end{bmatrix}, c = cos 90^\circ = 0, s = sin 90^\circ = 1$$

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

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

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

$$G_2 = \begin{bmatrix} 44 & 0 \\ 0 & 440 \end{bmatrix}$$

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

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



#### **Global stiffness matrix**

	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
<b>D</b> =				9753	0	0	0	0
<b>D</b> =					6783	0	0	0
			sym.			179923	1927	0
							118095	0
	L							18040

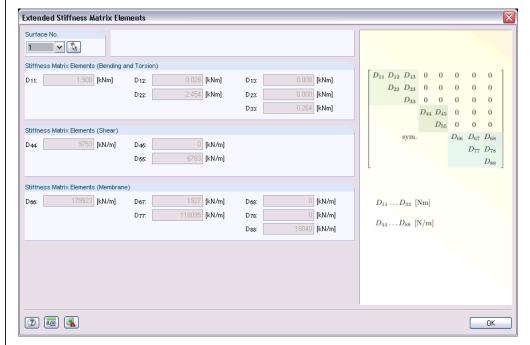


Figure 8.8: Dialog box Extended Stiffness Matrix Elements from RF-LAMINATE – without shear coupling of layers



## 8.2 Calculation of Stresses

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

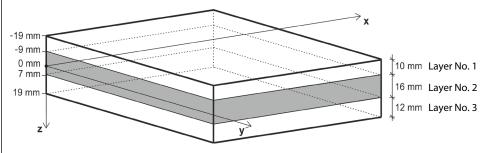


Figure 8.9: Layer scheme

Layers											
	A	В	C	D	E	F	G	Н		1	K
Layer	Material	Thickness	Orthotropic	Modulus of Ela	isticity [N/mm <sup>2</sup> ]	Shear	Modulus (N	/mm <sup>2</sup> ]	Poisson's	Ratio [-]	Specific Weight
No.	Description	t (mm)	Direction β [*]	Ex	Ey	Gxz	Gyz	Gxy	Vxy	Vух	γ [N/m <sup>3</sup> ]
1	Poplar and Coniferous Timber C16	10.0	0.00	8000.0	270.0	500.0	50.0	500.0	0.200	0.007	3700.0
2	Coniferous Timber C14	16.0	90.00	7000.0	230.0	440.0	44.0	440.0	0.200	0.007	5000.0
3	Poplar and Coniferous Timber C16	12.0	0.00	8000.0	270.0	500.0	50.0	500.0	0.200	0.007	3700.0

Figure 8.10: Table 1.2 Material Characteristics

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

The plate with the dimensions 1.0 x 1.5 m is simply supported and loaded with a surface load of 5  $kN/m^2$ .

## 8.2.1 Calculation of Individual Stress Components

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

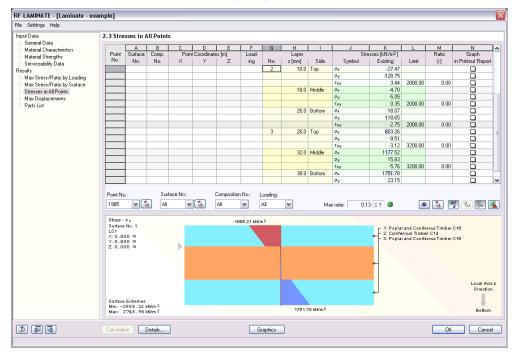


Figure 8.11: Window 2.3 Stresses in All Points - the example with the shear coupling of layers



INATE - [Laminate - ex	ample]														
ings Help															
a	2.3 Stre	sses in <i>l</i>	ll Points												
ral Data		A	в	С	D	E	F	G	н		1 1	ĸ		м	N
rial Characteristics	Point	Surface	D Comp.		: Coordinate		Load-	u	Layer			sses [kN/m <sup>2</sup> ]		Ratio	Graph
rial Strengths ceability Data	No.	No.	No.	×	Y	Z	ing	No.	z [mm]	Side	Symbol	Existing	Limit		in Printout Repo
ceability Data								2		Top	σχ	-344.92		.,	
Stress/Ratio by Loading											σv	3587.61			
Stress/Ratio by Surface											Txy	95.89	2000.00	0.05	
ses in All Points									18.0	Middle	σx	0.00			
Displacements											σy	0.00			
List											Txy	0.00	2000.00	0.00	
List									26.0	Bottom	σx	344.92			
											σ¥	3587.61			
											Txy	-95.89	2000.00	0.05	
								3	26.0	Top	σχ	-8414.69			Ē
											σv	-158.59			
											TXV	81.73	3200.00	0.03	
									32.0	Middle	σx	0.00			
											Øy	0.00			
											T <sub>XY</sub>	0.00	3200.00	0.00	
									38.0	Bottom	σχ	8414.69			
											σv	158.59			Ē
	1985 Stress - Surface LC1 X: 0.80 Y: 0.80	No. 1 00 m	AI AI			All .	-7012.24 kM	AI	~	Ma	ax ratio: 0.76	5 ≤ 1 - 1: Poplar - 2: Conife - 3: Poplar	rous Timber	rous Timber	
	Z: 0.00	00 m													
		Extremes	kN/m <sup>2</sup>							39 kN/m <sup>2</sup>					Local A Directi

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

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

Point	Side	$\sigma_{\rm x}$ [kPa]	$\sigma_y$ [kPa]	$ au_{xy}$ [kPa]
x = 0.8 m,	top	-27.47	-128.75	3.44
y = 0.8 m,	middle	-4.70	-5.05	0.35
Layer No. 2	bottom	18.07	118.65	-2.75

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

The middle layer is rotated of the angle  $\beta = 90^{\circ}$ 

 $\sigma_{\rm b+t/c,0} \,=\, \sigma_x \cos^2\beta + \tau_{xy} \sin 2\beta + \sigma_y \sin^2\beta$  $\sigma_{\rm b+t/c,0(top)} = -27.47\cos^2 90^\circ + 3.44 \cdot \sin(2 \cdot 90^\circ) - 128.75\sin^2 90^\circ = -128.75 \text{ kPa}$  $\sigma_{b+t/c,0(middle)} = -4.70\cos^2 90^\circ + 0.35 \cdot \sin(2 \cdot 90^\circ) - 5.05\sin^2 90^\circ = -5.05 \text{ kPa}$  $\sigma_{\rm b+t/c,0(bottom)} = 18.07\cos^2 90^\circ - 2.75 \cdot \sin(2 \cdot 90^\circ) + 118.65\sin^2 90^\circ = 118.65\,\rm kPa$ 

$$\begin{aligned} \sigma_{b+t/c,90} &= \sigma_x \sin^2 \beta - \tau_{xy} \sin 2\beta + \sigma_y \cos^2 \beta \\ \sigma_{b+t/c,90(top)} &= -27.47 \sin^2 90^\circ - 3.44 \sin(2 \cdot 90^\circ) - 128.75 \cos^2 90^\circ = -27.47 \text{ kPa} \\ \sigma_{b+t/c,90(middle)} &= -4.70 \sin^2 90^\circ - 0.35 \sin(2 \cdot 90^\circ) - 5.05 \cos^2 90^\circ = -4.70 \text{ kPa} \\ \sigma_{b+t/c,90(bottom)} &= 18.07 \sin^2 90^\circ - (-2.75) \sin(2 \cdot 90^\circ) + 118.65 \cos^2 90^\circ = 18.07 \text{ kPa} \end{aligned}$$

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

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



```
\begin{aligned} \sigma_{t/c,90} &= \frac{\sigma_{b+t/c,90(top)} + \sigma_{b+t/c,90(middle)} + \sigma_{b+t/c,90(bottom)}}{3} \\ \sigma_{t/c,90} &= \frac{-27.47 - 4.70 + 18.07}{3} = -4.70 \text{ kPa} \\ \sigma_{b,0} &= \sigma_{b+t/c,0} - \sigma_{t/c,0} \\ \sigma_{b,0(top)} &= -128.75 - (-5.05) = -123.70 \text{ kPa} \\ \sigma_{b,0(middle)} &= -5.05 - (-5.05) = 0 \text{ kPa} \\ \sigma_{b,0(bottom)} &= 118.65 - (-5.05) = 123.70 \text{ kPa} \\ \sigma_{b,90} &= \sigma_{b+t/c,90} - \sigma_{t/c,90} \\ \sigma_{b,90(top)} &= -27.47 - (-4.70) = -22.77 \text{ kPa} \\ \sigma_{b,90(middle)} &= -4.70 - (-4.70) = 0 \text{ kPa} \\ \sigma_{b,90(bottom)} &= 18.07 - (-4.70) = 22.77 \text{ kPa} \end{aligned}
```

### 8.2.2 Calculation Procedure in RF-LAMINATE

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

**D** 

ew Model - General Data General History	
	escription
	escription
Type of Model	Classification of Load Cases and Combinations
<ul> <li>3D</li> <li>2D - ⊻Y (uz/φx/φY)</li> <li>2D - ½Y (uz/φx/φY)</li> <li>2D - ½Y (ux/uz/φY)</li> <li>2D - ½Y (ux/uY/φz)</li> </ul>	According to Standard: None Create combinations automatically Load combinations Result combinations (for linear analysis only)
Positive Orientation of Global Axis Z	Template
<ul> <li>○ Upward</li> <li>○ Downward</li> </ul>	Open template model:
Comment	
2 📝 🔤 🖪 🖪	OK Cancel

Figure 8.13: Creating a new model

### 8 Examples



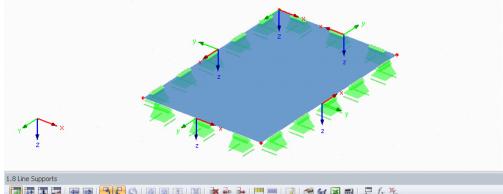
1

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

New Rectangular Surface	×
Surface No.	Surface Type
1	Geometry: Plane
Material	Stiffness: 📕 Laminate 💌 💽
1 Concrete C30/37 DIN 1045-1:2008-08	Surface typ Othout tension Othotropic Glass
Thickness	Laminate
Constant     Thickness d:	Rigid Membrane Membrane - Orthotropic
🔿 Variable 💽	
	P
Comment	
	OK Cancel

Figure 8.14: Dialog box New Rectangular Surface

Define the surfaces according to figure 8.15.



4			3	<b>X -</b>		14 1 🔀 64	🗶 📰 ab=	$\int x f_x$		
	A	В	С	D	E	F	G	Н		J
Support		Reference	Rotation	Wall	Supp	ort or Spring [k]	√/m²]	Rotational Re	estraint or Spring	g [kNm/rad/m]
No.	On Lines No.	System	β[*]	in Z	Cux	Cuy	Cuz	ψx	Фу	ψz
1	1-4	Local	0.00		√				1	<b>V</b>

Figure 8.15: Table Line Supports

<u>\*</u>

Now create a New Load Case.

ad Cases Load Combinations Result I	Combinations		
risting Load Cases	LC No. Load Case Description		To Solve
G LC1	1	~	
	General Calculation Parameters		
	Action Type		
	G Permanent	~	
	Self-Weight		
	Active		
	×: 💽 🔅 🖸		
	Y: Wassesse		
	Z:		
	Comment		
🔁 🔁 🛃 🛃 🛤 👪		v 💿	

DΙι

ıbal

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

Next, fill in the dialog box New Surface Load.

New Surface Load				
No. On Surfaces N	D.		<b>1</b>	Load Type 'Force' Load Distribution 'Uniform'
Load Type	Load Direction			
<ul> <li>Force</li> <li>Temperature</li> <li>Axial strain</li> </ul>	Local related to true area:	○ x ○ y ⊙ z		
Precamber     Rotary motion	Global related to true area:	O XL O YL		
Load Distribution	·	⊙ZL		
Uniform     Linear     Linear in X     Linear in Y     Linear in Z	Global related to projected area:	○XP ○YP ○ZP		
				Load Direction 'z'
Load Magnitude				X
Node No. Mag	nitude 5.00 (kN/m²)			ż
2nd: 1 📝 🗞 p: 3rd: 1 📝 🗞 p:	<pre> [kN/m<sup>2</sup>]</pre>			and the second s
the				
Comment			<b>v</b> 🖪	
2 🔤 🌿				OK Cancel

Figure 8.17: Dialog box New Surface Load



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

FE Mesh Settings			
General		Surfaces	
Target length of finite elements	IFE: 0.025 (m)	Maximum ratio of FE rectangle diagonals Δ <sub>D</sub> : 1.800 ⊕ [·]	,∼ IFE,
Maximum distance between a node and a line to integrate it into the line	ε: 0.001 €≽ [m]	Maximum out-of-plane inclination of two finite elements α: 0.50 () [']	3
Maximum number of mesh nodes (in thousands)	max: 500 牵	FE mesh refinement along lines (with type 'Plate XY' only)	0 ** 3
Members		Relationship ∆ <sub>b</sub> :	
Number of divisions for special type members (cable, elastic foundation, taper,	es of	Integrate unutilized objects into surfaces	
nonlinearity):	10 🚭	Shape of finite OQuadrangles only elements: OTriangles only	DI DI
in surfaces, with concrete mate for nonlinear calculation)		<ul> <li>Triangles only</li> <li>Triangles and quadrangles</li> </ul>	$\Delta_{D} = \frac{D_{I}}{D_{2}} \qquad D_{I} \ge D_{2}$
Minimum number of member divisions:	10 📚		Option
Activate member divisions for la post-critical analysis	rge deformation or	Mapped mesh preferred	Regenerate FE mesh on [OK]
Use division for straight member	rs, which are not	Solids	
integrated in surfaces, with Target length IFE of finite ele	ements	Refinement of FE mesh on solids containing close nodes	
◯ Set length IFE :	[m]	Maximum number of elements (in thousands): 200	
Minimum number of member divisions:	1		
Use division for members with n	odes lying on them		
			OK Cancel

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

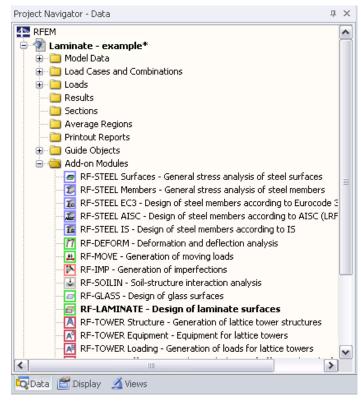


Figure 8.19: Navigator Data: Add-on Modules  $\rightarrow$  RF-LAMINATE

Figure 8.18: Dialog box FE Mesh Settings



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

Settings Help				
Input Data General Data	1.1 General Data			
Material Characteristics	Design of		Material Model	
laterial Strengths	Surfaces No.:		Orthotropic 🗸	
	1	🐴 🗙 🗖 All		
	Ultimate Limit State Serviceability Limit State			
	Existing Load Cases	Selected for Design		ш
		G LC1	Persistent and Transient	
				<b>RF-LAMINATE</b>
				2
				=
		>>		5
				5
				4
		4		
				L L
				~
				Calculation and design
				of laminate surfaces
	All (1)		20 K	
	Comment			
		1		
<b>4</b>		Graphics		OK Car

Figure 8.20: Window 1.1 General Data

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

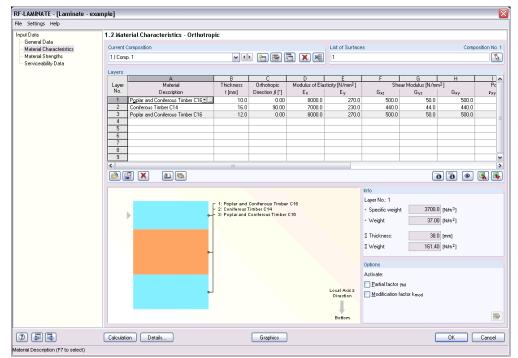


Figure 8.21: Window 1.2 Material Characteristics



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

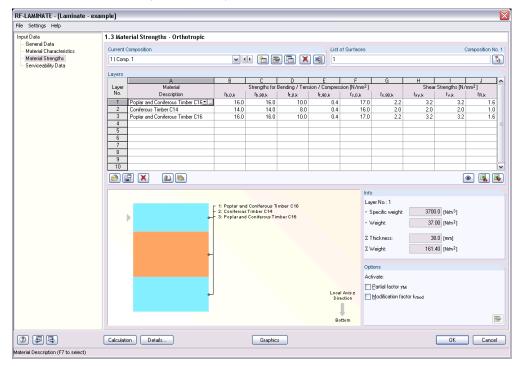


Figure 8.22: Window 1.3 Material Strengths

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

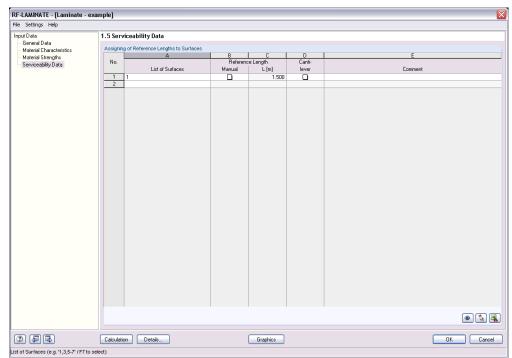


Figure 8.23: Window 1.5 Serviceability Data



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

Details		
Design Stresses Results		
To Display		Plate Bending Theory
Top/Bottom Layer $\sigma_x$ $\sigma_y$	Middle Layer $\sigma_x$ $\sigma_y$	Mindlin     Kirchhoff Equivalent Stresses According to (for Isotropic Materials)     Von Mises, Huber, Hencky
$ \begin{array}{c} & & & & & & \\ & & & & & & & \\ & & & & $	$ \begin{array}{c} & & & & & \\ & & & & & & \\ & & & & & & $	Verification hypothesis     Tresca     Maximum shear stress criterion     Rankine, Lamé     Maximum principal stress criterion     Bach, Navier, St. Venant, Poncelet     Principal strain criterion

Figure 8.24: Dialog box Details - the Stresses tab

E-SOLVER	Running       RFEM - Calculation by FEM       RF-LAMINATE       Partial Steps       Calculation of Stresses       Initialization       Reading the results       Calculating Stresses       Calculating Extremes       Saving Results       Surface No. 1 (1/1)	Number of surfaces Number of LC Number of CO Number of RC	
		Cancel	▼ Graph

Figure 8.25: Dialog box FE-Calculation...

#### 8 Examples



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

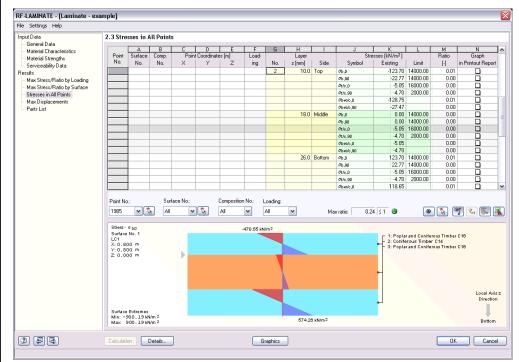


Figure 8.26: Window 2.3 Stresses in All Points



9. Annexes

### 9.1 Transformation Relations

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

y'y β x' z

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

$$\begin{cases} \sigma'_{x} \\ \sigma'_{y} \\ \tau'_{xy} \end{cases} = \begin{bmatrix} c^{2} & s^{2} & 2cs \\ s^{2} & c^{2} & -2cs \\ -cs & cs & c^{2} - s^{2} \end{bmatrix} \begin{cases} \sigma_{x} \\ \sigma_{y} \\ \tau_{xy} \end{cases}, \quad \begin{cases} \sigma_{b+t/c,0} \\ \sigma_{b+t/c,90} \end{cases} \equiv \begin{cases} \sigma'_{x} \\ \sigma'_{y} \end{cases}$$
(9.1)  
$$\overrightarrow{T_{3\times3}^{-T}}$$
$$\begin{cases} \varepsilon'_{x} \\ \varepsilon'_{y} \\ \gamma'_{xy} \end{cases} = \begin{bmatrix} c^{2} & s^{2} & cs \\ s^{2} & c^{2} & -cs \\ -2cs & 2cs & c^{2} - s^{2} \end{bmatrix} \begin{cases} \varepsilon_{x} \\ \varepsilon_{y} \\ \gamma'_{xy} \end{cases}$$
(9.2)

The stiffness matrix is transformed according to the relation

 $\mathbf{T}_{3 \times 3}$ 

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

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

The transformation relations for shear stresses and strains are the following

$$\begin{cases} \tau'_{xz} \\ \tau'_{yz} \end{cases} = \begin{bmatrix} c & s \\ -s & c \end{bmatrix} \begin{cases} \tau_{xz} \\ \tau_{yz} \end{cases}, \quad \begin{cases} \tau_d \\ \tau_R \end{cases} \equiv \begin{cases} \tau'_{xz} \\ \tau'_{yz} \end{cases}$$
(9.5)

$$\begin{cases} \gamma'_{xz} \\ \gamma'_{yz} \end{cases} = \begin{bmatrix} c & s \\ -s & c \end{bmatrix} \begin{cases} \gamma_{xz} \\ \gamma_{yz} \end{cases}$$
(9.6)  
$$\mathbf{T}_{2\times 2}$$

The stiffness matrix is transformed according to the relation

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

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



# 9.2 Checking the Positive Definiteness of the Stiffness Matrix

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

Generally, the global stiffness matrix has the shape

$$\mathbf{D}_{8\times8} = \begin{bmatrix} \mathbf{D}_{3\times3}^{\text{bending}} & \mathbf{0} & \mathbf{D}_{3\times3}^{\text{eccentric}} \\ \mathbf{0} & \mathbf{D}_{2\times2}^{\text{shear}} & \mathbf{0} \\ \mathbf{D}_{3\times3}^{\text{eccentric}} & \mathbf{0} & \mathbf{D}_{3\times3}^{\text{membrane}} \end{bmatrix} = \\ \begin{bmatrix} D_{11} & D_{12} & D_{13} & & D_{16} & D_{17} & D_{18} \\ D_{22} & D_{23} & & \text{sym. } D_{27} & D_{28} \\ & D_{33} & & \text{sym. } \text{sym. } D_{38} \\ & & D_{44} & D_{45} & & \\ & & & D_{55} & & \\ & & & & D_{66} & D_{67} & D_{68} \\ & & & & & & D_{77} & D_{78} \\ & & & & & & & & D_{88} \end{bmatrix}$$
(9.9)

The following conditions are checked:

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

$$\det \begin{bmatrix} D_{11} & & \\ & \ddots & \\ & & D_{ii} \end{bmatrix} \ge \sqrt{0.001} \prod_{i=1} |D_{ii}|, \text{ pro } i = 1, \dots, n \tag{9.10}$$



### A Literature

- [1] ALTENBACH, Holm ; ALTENBACH, Johannes; NAUMENKO, Konstantin. Ebene Flächentragwerke. Grundlagen der Modellierung und Berechnung von Scheiben und Platten. [s.l.] : Springer Verlag, 2008. 491 p.
- [2] DIN 1052. Entwurf, Berechnung und Bemessung von Holzbauwerken : Allgemein Bemessungsregeln und Bemessungsregeln für den Hochbau. [s.l.] : [s.n.], 2008. 239 p.
- [3] ČSN 73 1702. Navrhování, výpočet a posuzování dřevěných stavebních konstrukcí : Obecná pravidla a pravidla pro pozemní stavby. Praha : Český normalizační institut, November 2007. 174 p.
- [4] Huber M.T.: The theory of crosswise reinforced ferroconcrete slabs and its application to various constructional problems involving rectangular slabs, Der Bauingenieur, 1923, 4(12), 354-360 and 4(13), 392-395.



## B Index

Α
Action combination
В
Bach41
Browsing module windows20, 44
c
Calculation20, 33
Calculation start43
Characteristic strength
Closing RF-LAMINATE20, 44
Comment20
Coordinates45
Cross laminated timber without side glue24
D
Decimal places
Deflection22, 34
Design 20, 34
Details
Details of composition23
E
Equivalent stress41
Export of results58
Export of tables28
F
FE mesh point42, 45
Filter
G
General data20
Graphics
Grid point
н
Hybrid material model13
1
Import of tables
Installation
Isotropic material model13
К
Keyboard shortcuts
Kirchhoff40

#### I

L	
Library	28
Limit stress	. 46
Limit values	. 22
Load case	, 30
Load duration and service class	. 30
М	
Material characteristics	. 23
Material library	, 27
Material model 10,	, 20
Material strengths	. 29
Max displacements	. 51
Max. stress/ratio by loading	. 45
Max. stress/ratio by surface	. 49
Mindlin	. 40
Mises	. 41
Modification factor	, 46
Module windows	. 20
N	
Navigator	, 44
ο	
Orthotropic material model	. 10
Ρ	
Panel	7
Partial factor	29
Parts list	. 52
Plate bending theory	. 40
Positive-definite	, 81
Printing graphic	. 54
Printout report	. 54
R	
Rankine	. 41
Ratio 46,	51
Reference length 32,	, 51
Results 42,	, 44
S	
Sections	. 54
Serviceability data	. 32
Shear coupling of layers14, 17, 24,	63
Starting RF-LAMINATE	6
Stiffness matrix	. 60

Stiffness reduction	
Stress	35, 70
Stress components	54
Stresses in all points	
Symbols	8

### Т

Tresca	41
U	
Units	
User profile	57
User-defined material model	13

Dlubal