**Diubal** 

Version October 2013

Add-on Module

# **RF-GLASS**

Design of Single Layer, Laminated, and Insulating Glass

### 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	4.3.3	Standard – None	52
1.1	Add-on Module RF-GLASS	4	4.4	Start calculation	54
1.2	RF-GLASS Team	5	5.	Results	55
1.3	Using the Manual	6	5.1	Max Stress/Ratio by Loading	56
1.4	Open the RF-GLASS module	6	5.2	Max Stress/Ratio by Surface	58
2.	Theoretical background	8	5.3	Stresses in All Points	59
2.1	Symbols	8	5.4	Line Support Reactions	60
2.2	Types of Glass Structures	9	5.5	Nodal Support Reactions	61
2.2.1	Single-Layer Glass	9	5.6	Max Displacements	62
2.2.2	Laminated Glass	9	5.7	Gas Pressure	63
2.2.3	Insulating Glass	10	5.8	Parts List	63
2.3	Stiffness Matrix	12	6.	Printout	65
2.3.1	2D - Consideration of Shear Couplin		6.1	Printout Report	65
	Between Layers	12	6.2	Printing RF-GLASS Graphics	65
2.3.2	3D	14	6.2.1	Results on the RFEM Model	65
2.3.3	2D - Shear Coupling Between Layers Not Considered	5 14	6.2.2	Results in Layers	67
3.	Input Data	17	7.	<b>General Functions</b>	68
3.1	General Data	17	7.1	Units and Decimal Places	68
3.1.1	Ultimate Limit State	21	7.2	Export of Results	69
3.1.2	Serviceability Limit State	22	7.3	Connection to RFEM	70
3.2	Layers	23	8.	Examples	71
3.3	Line Supports	27	8.1	Determination of Stiffness Matrix	
3.4	Nodal Supports	31		Elements	71
3.5	Boundary Members	34	8.2	Insulating glass	76
3.6	Climatic Load Parameters	35	8.2.1	Calculation in RF-GLASS	77
3.7	Load Duration	37	8.2.2	Check of Calculation	83
3.8	Serviceability Data	38	8.3	Insulating Glass According to TRLV, Annex A	89
4.	Calculation	39	8.4	Curved Insulating Glass	92
4.1	Details	39	8.4.1	Calculation in RF-GLASS	93
4.1.1	Stresses	40	8.4.2	Check of Calculation	99
4.1.2	Results	45	9.	Appendix A	103
4.2	Details of Composition	46	9.1	Stiffness Matrix Check for Positive	
4.3	Standard	49		Definiteness	103
4.3.1	Standard – DIN 18008:2010-12	49	Α	Literature	104
4.3.2	Standard – TRLV:2006-08	51	В	Index	105



# I. Introduction

### 1.1 Add-on Module RF-GLASS

The add-on module RF-GLASS from DLUBAL SOFTWARE calculates deformations and stresses of glass surfaces. It allows you to generate all glass types like single layer, laminated, and insulation glass. Furthermore, you can consider shear coupling between the layers.

This module provides an extensive material library containing the common types of glass, foils, and gases. This library includes all essential material parameters according to the standards E DIN EN 13474, DIN 18008-1:2010-12, the technical rules TRLV:2006-08, as well as DIBt approval. Of course, you can also add other materials to the library.

For insulating glass, the calculation considers not only external loads, but also changes of temperature, atmospheric pressure, and altitude that influence an intermediate gas layer. The module also provides a simplified calculation according to Annex A of the standard DIN 18008-1:2010-12 or TRLV:2006-08.

This manual provides all necessary information for working with RF-GLASS. At the end of the manual, you find typical examples for glass design.

Like other modules, RF-GLASS is also fully integrated into RFEM. It is, however, not just an optical part of the main program: Results from the glass calculation, including graphical representations, can be transferred to the RFEM printout report. This allows for an easy and, above all, clearly arranged glass design. The clear layout of the program with its intuitive tables and dialog boxes as well as the uniform structure of the Dlubal add-on modules facilitate working with RF-GLASS.

We hope you will enjoy working with RFEM 5 and RF-GLASS.

Your team from DLUBAL SOFTWARE GMBH



### 1.2 RF-GLASS Team

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

#### **Program coordination**

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

#### Programming

Doc. Ing. Ivan Němec, CSc. Mgr. Petr Zajíček Ing. Pavel Bartoš

lng. Lukáš Weis Mgr. Vítězslav Štembera, Ph.D.

#### Cross-section and material data base

Ing. Jan Rybín, Ph.D.

#### Program design, dialog figures, icons

Dipl.-Ing. Georg Dlubal

MgA. Robert Kolouch

#### **Program supervision**

Mgr. Vítězslav Štembera, Ph.D. Ing. Iva Horčičková Dipl.-Ing. (FH) Ulrich Lex

#### Manual, help system, and translation

Ing. Iva Horčičková Mgr. Vítězslav Štembera, Ph.D. Ing. Fabio Borriello Ing. Dmitry Bystrov Eng.° Rafael Duarte Ing. Jana Duníková Ing. Lara Freyer Alessandra Grosso Ing. Chelsea Jennings Jan Jeřábek Ing. Ladislav Kábrt Ing. Aleksandra Kociołek Mgr. Michaela Kryšková Dipl.-Ing. Tingting Ling Ing. Roberto Lombino Eng.° Nilton Lopes Mgr. Ing. Hana Macková Ing. Téc. Ind. José Martínez MA Translation Anton Mitleider Dipl.-Ü. Gundel Pietzcker Mgr. Petra Pokorná Ing. Zoja Rendlová Dipl.-Ing. Jing Sun Ing. Marcela Svitáková Dipl.-Ing. (FH) Robert Vogl Ing. Marcin Wardyn

#### Technical support and quality management

M.Eng. Cosme Asseya Dipl.-Ing. (BA) Markus Baumgärtel Dipl.-Ing. Moritz Bertram M.Sc. Sonja von Bloh Dipl.-Ing. (FH) Steffen Clauß Dipl.-Ing. Frank Faulstich Dipl.-Ing. (FH) René Flori Dipl.-Ing. (FH) Stefan Frenzel Dipl.-Ing. (FH) Walter Fröhlich Dipl.-Ing. (FH) Walter Fröhlich Dipl.-Ing. (FH) Andreas Hörold Dipl.-Ing. (FH) Paul Kieloch Dipl.-Ing. (FH) Bastian Kuhn Dipl.-Ing. (FH) Ulrich Lex Dipl.-Ing. (BA) Sandy Matula Dipl.-Ing. (FH) Alexander Meierhofer M. Eng. Dipl.-Ing. (BA) Andreas Niemeier Dipl.-Ing. (FH) Gerhard Rehm M. Eng. Dipl.-Ing. (FH) Walter Rustler M.Sc. Dipl.-Ing. (FH) Walter Rustler M.Sc. Dipl.-Ing. (FH) Christian Stautner Dipl.-Ing. (FH) Lukas Sühnel Dipl.-Ing. (FH) Robert Vogl



### 1.3 Using the Manual

Topics like system requirements or installation are described in detail in the RFEM manual. Therefore, they do not need to be introduced here. Instead, the present manual focuses on the special features of the add-on module RF-GLASS.

The description of the module keeps to the sequence and structure of the input and output module windows. The text of the manual shows the described **buttons** in square brackets, for example [View mode]. At the same time, they are pictured on the left. In addition, **expressions** used in dialog boxes, tables, and menus are set in *italics* to clarify the explanations.

At the end of the manual, you find an index. However, if you still cannot find what you are looking for, please check our website **www.dlubal.com**, where you can go through our *FAQ* pages by selecting particular criteria.

### 1.4 Open the RF-GLASS module

There are several possibilities to open the add-on module RF-GLASS.

#### Main menu

۲

To open RF-GLASS, select on the RFEM menu

```
\textbf{Add-on Modules} \rightarrow \textbf{Others} \rightarrow \textbf{RF-GLASS}.
```

Add	-on Modules Windo	w	<u>H</u> elp	5	
<b>4</b> 0	Current Module			- < > <u>P</u>	🚰 🕰 🔛 🖌 🕼 📾 🖬 😤 🗳
	Design - Steel	×	웧	- 🛛 💥 🥰 🏹 🗇	) 🗗   🛱 📬 🛱 🛪 - 🛂 -   🌚 - 🤅
	Design - Concrete	×			
	Design - Timber	×			
	Design - Aluminium	•			
	Dynamic	×			
	Connections	×			
	Foundations	×			
	Stability	×			
	Towers	*			
	Others	•	17	RF-DEFORM	Deformation and deflection analysis
	External Modules	•	<u>,+++</u>	RF-MOVE	Generation of moving loads
		-	Þ	RF-IMP	Generation of imperfections
			E,	RF-STAGES	Analysis of construction stages
			₺_	RF-LOAD-HISTORY	Simulation of load history
				RF-INFLUENCE	Generation of influence lines and surfaces
			÷	RF-SOILIN	Soil-structure interaction analysis
			0	RF-GLASS	Design of glass surfaces
			ø	RF-LAMINATE	Design of laminate surfaces

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

#### Navigator

Alternatively, you can start RF-GLASS in the Data navigator:

```
\label{eq:Add-onModules} \mathsf{Add}\text{-}\mathsf{onModules} \mathop{\rightarrow} \mathsf{RF}\text{-}\mathsf{GLASS}\text{-}\mathsf{Design}\,\mathsf{ofglass}\,\mathsf{surfaces}.
```

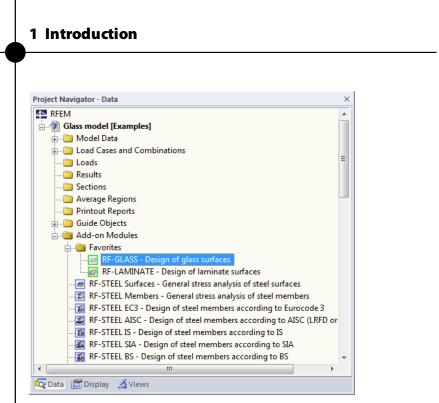


Figure 1.2: Data navigator: Add-on Modules  $\rightarrow$  RF-GLASS

#### Panel

If there are already RF-GLASS results in the RFEM model, you can start the add-on module in the panel:

Set the relevant RF-GLASS design case in the load case list, which is located in the RFEM toolbar. Then, click [Show results] to display the deformations or stresses graphically.

In the panel, you can now use the [RF-GLASS] button to open the module.

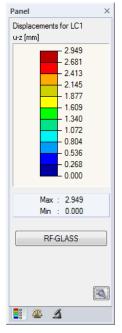
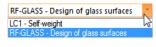


Figure 1.3: Panel button [RF-GLASS]



RF-GLASS

<u>\_</u>



# 2. Theoretical background

This chapter briefly explains the theoretical principles of the RF-GLASS module.

## 2.1 Symbols

t	Thickness of composition [m]
t <sub>i</sub>	Thickness of individual layers [m]
Ε	Modulus of elasticity [Pa]
G	Shear modulus [Pa]
ν	Poisson's ratio [-]
γ	Specific weight [N/m³]
$\alpha_{\rm T}$	Coefficient of thermal expansion [1/K]
$\sigma_{limit}$	Limit stress [Pa]
λ	Thermal conductivity [W/(mK)]
$d_{ij}$	Elements of the partial stiffness matrix [Pa]
D <sub>ij</sub>	Elements of the global stiffness matrix [Nm, Nm/m, N/m]
$\sigma_x$ , $\sigma_y$	Normal stresses [Pa]
$\tau_{yz}$ , $\tau_{xz}$ , $\tau_{xy}$	Shear stresses [Pa]
n	Number of layers [-]
Ζ	z -axis coordinate [m]
Т	Temperature [K]
p	Pressure [Pa]
Н	Altitude [m]
V	Volume [m <sup>3</sup> ]
m <sub>x</sub>	Bending moment inducing stresses in $x$ -axis direction [Nm/m]
m <sub>y</sub>	Bending moment inducing stresses in $y$ -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]



### 2.2 Types of Glass Structures

As already mentioned in the introduction, we distinguish between single layer glass, laminated glass, and insulating glass. Modeling of the different glass types is described in the following chapters.

### 2.2.1 Single-Layer Glass

Single-layer glass is the simplest composition case. For single-layer glass, you can use:

- 2D calculation (plate theory)
- 3D calculation (modeling by using solids)

Calculation according to the plate theory has its limits in the case of plates with an extreme thickness. These are modeled by using solids. An approximation criterion for a valid calculation according to the plate theory is given by the relation  $t/L \le 0.05$ , where t is the thickness and L is the length of the plate side (or the characteristic dimension of the model).

#### 2.2.2 Laminated Glass

Laminated glass consists of at least two glass panes, connected by an intermediate layer, which in most cases is made up of a foil or resin.

For laminated glass, you can use:

- 2D calculation with shear coupling between layers (plate theory)
- 3D calculation (modeling by using solids)
- 2D calculation without shear coupling of layers (plate theory)

#### 2D calculation without shear coupling between layers

The stiffness, which is calculated on the basis of the layer composition, is assigned to one or more selected surfaces. The surface is then modeled by using common surface elements.

#### **3D** calculation

For laminated glass, the foil connecting individual glass panes is usually much thinner than the glass. The product of the foil thickness and its shear modulus  $t \cdot G$  is about 3-7 decimal places smaller than the product of the glass thickness and the shear modulus of glass. This means that there is a significant shear distortion in glass and foil (see Figure 2.2), and the 2D plate theory yields incorrect results. In this case, it is recommended to use the 3D calculation which yields accurate results, but is more time-consuming.

#### 2D calculation without shear coupling between layers

It is also possible to calculate according to the 2D plate theory without shear coupling between layers. Individual glass panes can then "slide" over each other. This calculation is recommended for long-term loads, when the shear resistance of a connecting foil should not be considered, because its properties depend on the load duration and temperature.

The three mentioned options are shown in Figure 2.1.

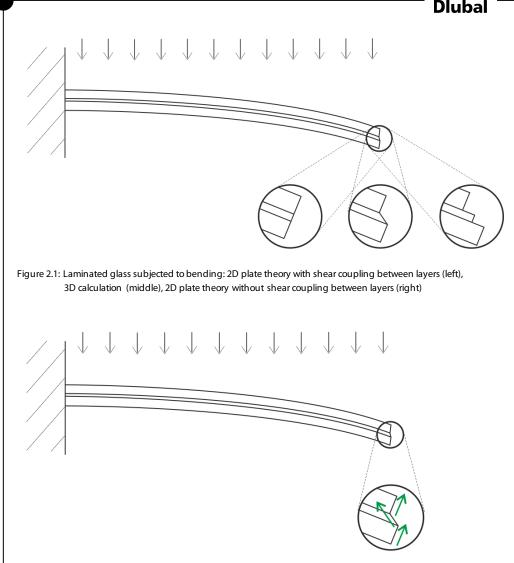


Figure 2.2: Shear distortion in laminated glass (3D calculation)

#### 2.2.3 Insulating Glass

This type of glass is always calculated by large deformation analysis, with an application of the NEWTON-RAPHSON method.

Insulating glass consists of individual glass panes, intermediate gas layer, spacer, primary and secondary seal. All these components are essential for the overall behavior of the glass. Besides a composition of individual layers, you can specify properties of the secondary seal and climatic load parameters in RF-GLASS.

Insulating glass is calculated in 3D, therefore all layers are modeled by solids. Consequently, it is only possible to create an insulating glass when the *Local* calculation type is selected (see Chapter 3.1, page 17). A layer of the *Gas* type is modeled by using a solid element, created especially for this calculation. The ideal gas law is then considered in the calculation. Glass is produced at temperature  $T_p$ , pressure  $p_p$ , and initial gas volume  $V_0$  (of a certain intermediate layer).

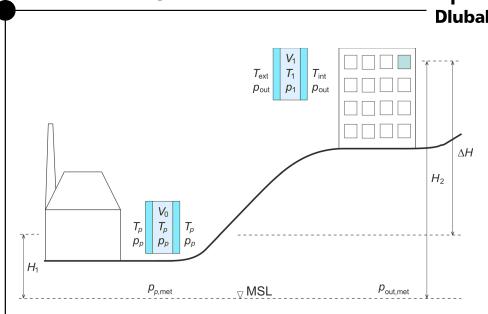


Figure 2.3: Climatic load parameters for manufacturing (left), mount (right), MSL = mean sea level

A load due to a temperature change is converted to a change of an ambient pressure  $p_{out}$  by using the coefficient  $c_1$ . The ambient pressure  $p_{out}$  comprises the atmospheric pressure change converted to the sea level  $\Delta p_{met}$ , the influence of gas heating  $\Delta T$ , and the pressure change due to the altitude  $\Delta H$ . It is determined as follows:

$$p_{\rm out} = p_p + \Delta p_{\rm met} - c_1 \Delta T - c_2 \Delta H \tag{2.1}$$

$$\Delta p_{\rm met} = p_{\rm out,met} - p_{p,\rm met} \tag{2.2}$$

$$\Delta T = T_1 - T_p \tag{2.3}$$

$$\Delta H = H_2 - H_1 \tag{2.4}$$

where

$$c_1 = \frac{p_p}{T_p} \text{ Pa/K}$$
(2.5)

$$c_2 = 12 \text{ Pa/m}$$
 (2.6)

Moreover, the solution satisfies the equilibrium equation

$$p_p V_0 = p_1 V_1$$
 (2.7)

H <sub>1</sub>	Altitude at manufacturing	$p_{p,\mathrm{met}}$	Atmospheric pressure at sea level (manufacturing)
H <sub>2</sub>	Altitude at mount	p <sub>out,met</sub>	Atmospheric pressure at sea level (mount)
$\Delta H$	Difference in altitude $H_2 - H_1$	p <sub>p</sub>	Pressure during manufacturing
Τ <sub>p</sub>	Temperature during manu- facturing	p <sub>out</sub>	Ambient pressure during mount
T <sub>ext</sub>	Temperature on the external glass side (mount)	$p_1$	Gas pressure during mount
T <sub>int</sub>	Temperature on the internal glass side (mount)	V <sub>0</sub>	lnitial gas volume
<i>T</i> <sub>1</sub>	Gas temperature (mount)	<i>V</i> <sub>1</sub>	Final gas volume

Table 2.1: Symbols for insulating glass



### 2.3 Stiffness Matrix

As an isotropic material, glass is defined by the modulus of elasticity E, the shear modulus G and Poisson's ratio v:

$$G = \frac{E}{2 \cdot (1 + \nu)} \tag{2.8}$$

#### 2.3.1 2D - Consideration of Shear Coupling Between Layers

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

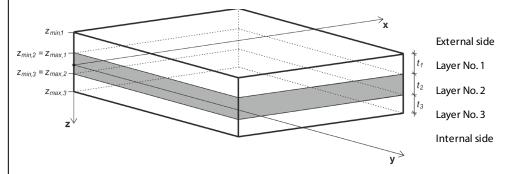


Figure 2.4: Layer composition

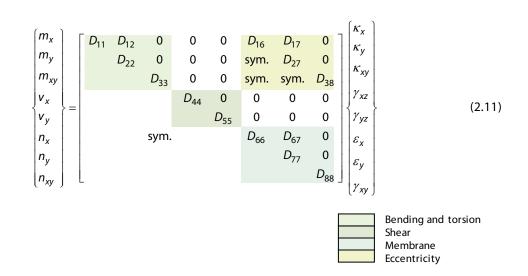
The stiffness matrix for each layer  $d_i$  is defined as follows:

$$\boldsymbol{d}_{i} = \begin{bmatrix} d_{11,i} & d_{12,i} & 0\\ & d_{22,i} & 0\\ \text{sym.} & & d_{33,i} \end{bmatrix} = \begin{bmatrix} \frac{E_{i}}{1-v_{i}^{2}} & \frac{v_{i}E_{i}}{1-v_{i}^{2}} & 0\\ & \frac{E_{i}}{1-v_{i}^{2}} & 0\\ \text{sym.} & & G_{i} \end{bmatrix}, \quad G_{i} = \frac{E_{i}}{2 \cdot (1+v_{i})} \quad i = 1, ..., n \quad (2.9)$$

The global stiffness matrix is:

$$\boldsymbol{D} = \begin{bmatrix} D_{11} & D_{12} & 0 & 0 & 0 & D_{16} & D_{17} & 0 \\ D_{22} & 0 & 0 & 0 & sym. & D_{27} & 0 \\ D_{33} & 0 & 0 & sym. & sym. & D_{38} \\ & & D_{44} & 0 & 0 & 0 & 0 \\ & & & D_{55} & 0 & 0 & 0 \\ & & & & D_{66} & D_{67} & 0 \\ & & & & & D_{77} & 0 \\ & & & & & & & D_{88} \end{bmatrix}$$
(2.10)

### 2 Theoretical background



#### Stiffness matrix elements (bending and torsion) [Nm]

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

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

Dlubal

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

$$D_{16} = \sum_{i=1}^{n} \frac{z_{\max,i}^2 - z_{\min,i}^2}{2} d_{11,i} \qquad D_{17} = \sum_{i=1}^{n} \frac{z_{\max,i}^2 - z_{\min,i}^2}{2} d_{12,i}$$
$$D_{27} = \sum_{i=1}^{n} \frac{z_{\max,i}^2 - z_{\min,i}^2}{2} d_{22,i}$$

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

Stiffness matrix elements (membrane) [N/m]

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

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

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

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

. Dlubal

Stiffness matrix elements (shear) [N/m]

$$D_{44} = D_{55} = \max\left(D_{44/55, \text{calc}}, \frac{48}{5l^2} \frac{1}{\frac{1}{\sum_{i=1}^{n} E_i \frac{t_i^3}{12}} - \frac{1}{\sum_{i=1}^{n} E_i \frac{z_{\text{max},i}^3 - z_{\text{min},i}^3}{3}}\right)$$
(2.12)

where I is the middle size of the surface bounding box. The value  $D_{44/55, calc}$  is given by:

$$D_{44/55,\text{calc}} = \frac{1}{\int_{-t/2}^{t/2} \frac{1}{G(z)} \left( \int_{-t/2}^{t/2} \frac{d_{11}(\overline{z})(\overline{z} - z_0) d\overline{z}}{\int_{-t/2}^{z} \frac{d_{11}(\overline{z})(\overline{z} - z_0) d\overline{z}}{\int_{-t/2}^{z} \frac{d_{11}(\overline{z})(\overline{z} - z_0)^2 d\overline{z}}{\int_{-t/2}^{z} \frac{d_{11}(\overline{z})(\overline{z} - z_0)^2 d\overline{z}}} \right)^2} dz, \quad z_0 = \frac{\int_{-t/2}^{t/2} d_{11}(\overline{z}) d\overline{z}}{\int_{-t/2}^{t/2} d_{11}(\overline{z}) d\overline{z}}, \quad t = \sum_{i=1}^n t_i$$
(2.13)

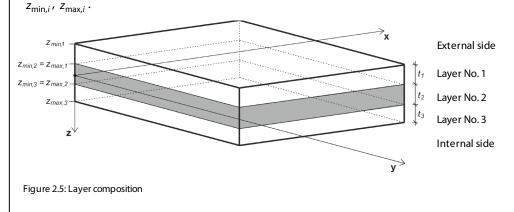
#### 2.3.2 3D

If the model is created by means of solids, the following stiffness matrix is used:

$$\begin{cases} \sigma_{X} \\ \sigma_{y} \\ \sigma_{z} \\ \tau_{yz} \\ \tau_{xz} \\ \tau_{xy} \end{cases} = \begin{bmatrix} \frac{1}{E} & -\frac{\nu}{E} & 0 & 0 & 0 \\ & \frac{1}{E} & -\frac{\nu}{E} & 0 & 0 & 0 \\ & & \frac{1}{E} & 0 & 0 & 0 \\ & & & \frac{1}{G} & 0 & 0 \\ & & & & \frac{1}{G} & 0 & 0 \\ & & & & & \frac{1}{G} & 0 \\ & & & & & \frac{1}{G} \end{bmatrix}^{-1} \begin{cases} \varepsilon_{X} \\ \varepsilon_{y} \\ \varepsilon_{z} \\ \gamma_{yz} \\ \gamma_{xz} \\ \gamma_{xy} \end{cases}, \quad G = \frac{E}{2 \cdot (1 + \nu)}$$
(2.14)

#### 2.3.3 2D - Shear Coupling Between Layers Not Considered

Now, consider a plate consisting of n isotropic materials without shear coupling of the individual layers. Each layer has the thickness  $t_i$  and a minimum and maximum z-coordinate



#### Program RF-GLASS© 2013 Dlubal Software GmbH

Dlubal

The stiffness matrix for each layer  $d_i$  is determined as follows:

$$\boldsymbol{d}_{i} = \begin{bmatrix} d_{11,i} & d_{12,i} & 0\\ & d_{22,i} & 0\\ \text{sym.} & & d_{33,i} \end{bmatrix} = \begin{bmatrix} \frac{E_{i}}{1-v_{i}^{2}} & \frac{v_{i}E_{i}}{1-v_{i}^{2}} & 0\\ & \frac{E_{i}}{1-v_{i}^{2}} & 0\\ & \frac{1-v_{i}^{2}}{2} & 0\\ \text{sym.} & & G_{i} \end{bmatrix}, \quad G_{i} = \frac{E_{i}}{2\cdot(1+v_{i})} \quad i = 1, ..., n \quad (2.15)$$

The global stiffness matrix is:

Stiffness matrix elements (bending and torsion) [Nm]

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

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

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

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

Membrane

**Diubal** 

Stiffness matrix elements (membrane) [N/m]

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

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

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

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

Stiffness matrix elements (shear) [N/m]

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



When you start RF-GLASS, a new window appears. The navigator on the left side contains the available module windows.

The design-relevant data is to be defined in several input windows.



To open a module window, click the appropriate item in the navigator. To select the previous or next window, use the buttons shown on the left. To browse through the windows, you can use the keys [F2] (next) and [F3] (previous).

Calculation

Details...

Standard
Graphics
ОК
Cancel

Having entered all necessary data, you can start the [Calculation].

By clicking [Details], you open the dialog box where you can specify the stresses and results windows to be displayed (see Chapter 4.1, page 39).

To set the limit deflections and other calculation parameters, click [Standard].

To display the RFEM work window, click [Graphics].

To save the entered data and exit RF-GLASS, click [OK]. To exit the module without saving the entered data, click [Cancel].

### 3.1 General Data

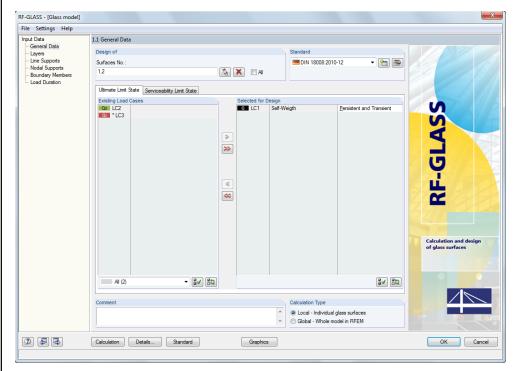


Figure 3.1: Window 1.1 General Data

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



#### Design of

In the upper section, you specify the surfaces for the design. If you want to analyze only particular surfaces, clear the selection of the *All* check box: Thus, an input field becomes available where you can enter the relevant surfaces. You can select the list of the preset numbers by double-clicking it, and then overwrite the entry manually. The [^] button allows you to graphically select the surfaces in the RFEM work window. You can delete the list of the already preset surface numbers by clicking [X].

#### Standard

In the Item list in the upper right corner of the window, you can select the standard from which the parameters will be applied for the design and the limit values of the deflection.

The following standards can be selected:

- DIN 18008:2010-12
- TRLV:2006-08
- None

Use the [Edit] button to open a dialog box where you can check and, if necessary, adjust the parameters of the selected standard. The dialog box is described in Chapter 4.3 on page 49.

To create a user-defined standard, click [Create new standard...].

In addition to that, you can click the [Standard] button from all windows. This button also allows you to open the *Standard* dialog box.

#### Calculation type

In the section Calculation type, you can choose:

- Local Individual glass surfaces
- Global Whole model in RFEM

If you select *Local - Individual glass surfaces*, the calculation of the selected surfaces in RF-GLASS is done in independent systems. The surfaces are analyzed separately, without interaction with the model created in RFEM. Line supports, nodal supports and boundary members can be set directly in Windows 1.3, 1.4 and 1.5 of the module. The supports and members entered in RF-GLASS are considered only in the module; they do not influence RFEM specifications. For this selection, 3D calculation (using 3D finite elements) of glass surfaces is possible.

If you select *Global* - *Whole model in RFEM*, the calculation proceeds directly with the model created in RFEM. Therefore, it is not necessary to define supports and boundary members directly in RF-GLASS. Consequently, Windows 1.3, 1.4 and 1.5 are not available with this option (as you can see in Figure 3.2). If the *Global* calculation type is selected, only 2D calculation (plate theory) is possible. It is not possible to create insulating glass (set gas layer in Table 1.2) which is always modeled by solid elements (see Chapter 2.2.3, page 10).

Standard

6

X

Settings Help Data	1.1 General Data				
eneral Data	Design of		Standard		
ayers bad Duration	Surfaces No.:		DIN 18008:2010		12
	1.2	🐴 🗙 🔳 Al	Div 10000.2010	0-12 🔻 🎦 💽	
	Ultimate Limit State Serviceability Limit State	te			
	Existing Load Cases	Selected for	Desian		
	Qs LC2	G LC1	Self-Weigth	Persistent and Transient	S
	Qi * LC3				RF-GLASS
		>>			
					U
					T
		4			LL I
					Calculation and desig
					of glass surfaces
	All (2)	2v 82			
	Comment		Calculation Type		
			Cocal - Individual	class surfaces	
			<ul> <li>Global - Whole me</li> </ul>		

Figure 3.2: Window 1.1 General Data – Global calculation type

The following example shows how the model in RFEM and the *Calculation type* in RF-GLASS significantly influences the RF-GLASS calculation. Consider the glass structure in the following picture.

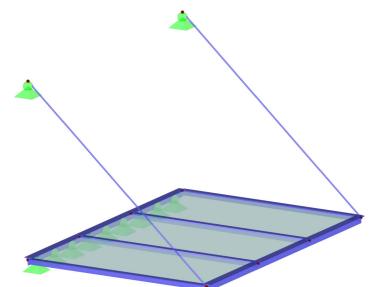


Figure 3.3: RFEM model

• The model is created as one surface. Then, two lines with members are inserted. *Local calculation of individual glass surfaces* is selected in RF-GLASS. The model used for the calculation is in the following picture.



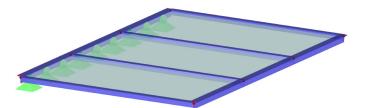


Figure 3.4: RF-GLASS model with one surface of the type Glass and Local calculation of individual glass surfaces

• This example can also be modeled in RFEM with three surfaces. However, in this case three separate models are created in RF-GLASS – see Figure 3.5, Figure 3.6 and Figure 3.7. You can see that supports or members that are created on common lines or nodes (in this case member) are valid for both surfaces.

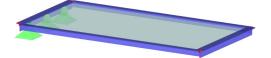


Figure 3.5: Model of surface No. 1 in RF-GLASS in the case of three surfaces of the type Glass and Local calculation of individual glass surfaces

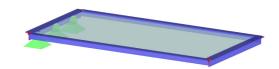


Figure 3.6: Model of surface No. 2 in RF-GLASS in the case of three surfaces of the type Glass and Local calculation of individual glass surfaces

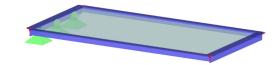


Figure 3.7: Model of surface No. 3 in RF-GLASS in the case of three surfaces of the type Glass and Local calculation of individual glass surfaces

• If *Global calculation with whole model in RFEM* is selected, calculation is done with the same model as in RFEM.

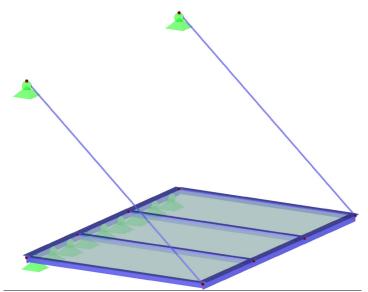


Figure 3.8: Model in the case of Global calculation with whole structure in RFEM



#### Comment

This comment text box is located at the bottom of the window. You can enter notes or explanations for the RF-GLASS case.

#### 3.1.1 Ultimate Limit State

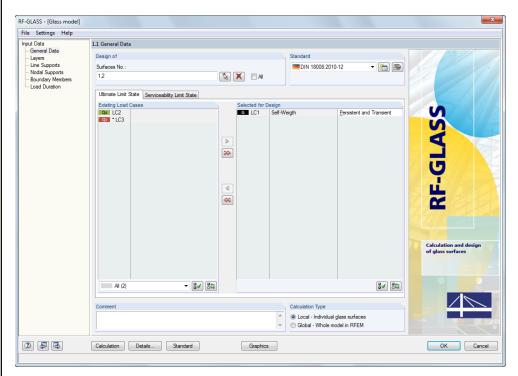


Figure 3.9: Window 1.1 General Data, tab Ultimate Limit State

#### **Existing Load Cases**

This section contains the list of all load cases, load combinations, and result combinations created in RFEM.

By using the [▶] button, you can transfer selected entries to the list *Selected for Design* on the right. You can also transfer items by double-clicking them. To transfer the entire list to the right, click [▶▶].

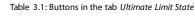
You can also make a multiple selection of load cases by pressing the [Ctrl] key and clicking the respective items, as is usual in Windows. In this way, you can select and transfer several load cases to the list on the right at once.

If a load case or load combination is marked with an asterisk (\*), as for example LC3 in Figure 3.9, you cannot design it: This indicates a load case without load data or an imperfection load case. However, this does not apply to insulating glass: This type of glass can also be loaded by a change of temperature, atmospheric pressure, or altitude (see Chapter 3.6, page 35). If at least one gas layer is defined in the 1.2 *Layers* window, the asterisk (\*) disappears in the 1.1 *General Data* window at the load case without the assigned load data so that you can select it for the design.

Filter options are available at the bottom of the list. These options make it easier to assign the entries sorted by load cases, load combinations, or action categories. The buttons have the following functions:



Selects all load cases in the list
Inverts the selection of load cases



Because the calculation of insulating glass always proceeds by the large deformation analysis, it is not possible to calculate result combinations for insulating glass.

#### **Selected for Design**

5

4

4

Standard

The right part of the module window lists the load cases, load combinations, and result combinations selected for design. To remove the selected items from the list, click [] or double-click them. To transfer the entire list to the left, click [].

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

- Persistent and transient
- Accidental

This classification manages the partial safety factor  $\gamma_M$  of the material properties. You can check and adjust this factor in the *Standard* dialog box (see Chapter 4.3).

### 3.1.2 Serviceability Limit State

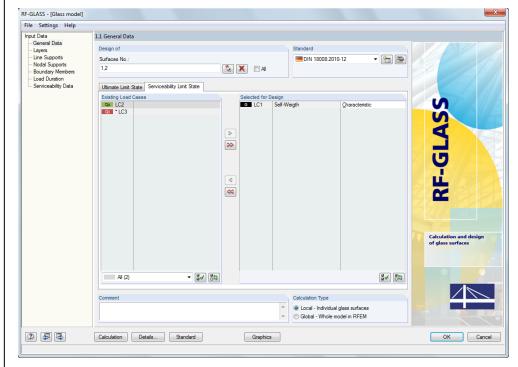


Figure 3.10: Window 1.1 General Data, tab Serviceability Limit State

#### **Existing Load Cases**

This section lists all load cases, load combinations, and result combinations, which were created in RFEM. After you transfer items to the *Selected for Design* list on the right, the additional window 1.8 *Serviceability Data* appears in the navigator.



Standard

Selected for Design

As described in Chapter 3.1.1, you can add or remove load cases, load combinations, and result combination.

In this section, you assign a design situation to the individual load cases, load combinations, and result combinations, either *Characteristic, Frequent*, or *Quasi-permanent*. Based on this selection, different limit values apply for the deflection. You can adjust these limit values in the *Standard* dialog box (see Chapter 4.3).

# 3.2 Layers

urrent (	Composition				List of Surface	es		Composition No
1   Comp	o. 1	▼	• 🎦 🗃		1			1
ayers								
	A	В	C	D	E	F	G	Н
Layer	Layer	Material	Thickness	Modulus of Elast.	Shear Modulu	is Poisson's Ratio	Specific Weight	Coeff. of Th. E
No.	Туре	Description	t [mm]	E [MPa]	G [MPa]	v [-]	γ [kN/m <sup>3</sup> ]	αт [1/К]
1	Glass	Thermally Toughened Float Glass	10.00	70000.000	28455.	300 0.230	25.00	9.0
2	Foil	PVB 22 °C loading until 3 min	0.38	3.000	1.	001 0.499	10.70	8.0
3	Glass	Themally Toughened Float Glass	10.00	70000.000	28455.	300 0.230	25.00	9.0
4	Glass	15						
5	Foil							
6	Gas							
7	-							
8								
9								
<u>)</u>	<b>}</b> 🗙						0	• 🛐
							0	
							00	
						Info		
			Taushaaad Elas	t Glass		Info Layer No.: 3	00	
		r 1: Thermally	Toughened Floa	t Glass min		Layer No.: 3		
		- 1: Thermally - 2: PVB 22 *	' Toughened Floa C loading until 3 r y Toughened Floa	min			0.250 [kN/m <sup>2</sup> ]	
		- 1: Thermally - 2: PVB 22 *	C loading until 3 r	min		Layer No.: 3 Surface weight:	0.250 [kN/m <sup>2</sup> ]	
		- 1: Thermally - 2: PVB 22 *	C loading until 3 r	min		Layer No.: 3	0.250 [kN/m <sup>2</sup> ] 20.38 [mm]	
		- 1: Thermally - 2: PVB 22 *	C loading until 3 r	min		Layer No.: 3 Surface weight:	0.250 [kN/m <sup>2</sup> ]	
		- 1: Thermally - 2: PVB 22 *	C loading until 3 r	min		Layer No.: 3 Surface weight: Σ Thickness:	0.250 [kN/m <sup>2</sup> ] 20.38 [mm]	
		- 1: Thermally - 2: PVB 22 *	C loading until 3 r	min		Layer No.: 3 Surface weight: Σ Thickness:	0.250 [kN/m <sup>2</sup> ] 20.38 [mm]	
		- 1: Thermally - 2: PVB 22 *	C loading until 3 r	min		Layer No.: 3 Surface weight: Σ Thickness:	0.250 [kN/m <sup>2</sup> ] 20.38 [mm]	
		- 1: Thermally - 2: PVB 22 *	C loading until 3 r	min		Layer No.: 3 Surface weight: Σ Thickness:	0.250 [kN/m <sup>2</sup> ] 20.38 [mm]	
		- 1: Thermally - 2: PVB 22 *	C loading until 3 r	min		Layer No.: 3 Surface weight: Σ Thickness:	0.250 [kN/m <sup>2</sup> ] 20.38 [mm]	
		- 1: Thermally - 2: PVB 22 *	C loading until 3 r	min		Layer No.: 3 Surface weight: Σ Thickness:	0.250 [kN/m <sup>2</sup> ] 20.38 [mm]	
		- 1: Thermally - 2: PVB 22 *	C loading until 3 r	min		Layer No.: 3 Surface weight: Σ Thickness:	0.250 [kN/m <sup>2</sup> ] 20.38 [mm]	
		- 1: Thermally - 2: PVB 22 *	C loading until 3 r	min	Local Axis z	Layer No.: 3 Surface weight: Σ Thickness:	0.250 [kN/m <sup>2</sup> ] 20.38 [mm]	
		- 1: Thermally - 2: PVB 22 *	C loading until 3 r	min	Local Axis z Direction	Layer No.: 3 Surface weight: Σ Thickness:	0.250 [kN/m <sup>2</sup> ] 20.38 [mm]	
		- 1: Thermally - 2: PVB 22 *	C loading until 3 r	min		Layer No.: 3 Surface weight: Σ Thickness:	0.250 [kN/m <sup>2</sup> ] 20.38 [mm]	
		- 1: Thermally - 2: PVB 22 *	C loading until 3 r	min		Layer No.: 3 Surface weight: Σ Thickness:	0.250 [kN/m <sup>2</sup> ] 20.38 [mm]	

Figure 3.11: Window 1.2 Layers

In this window, you define layer compositions for individual surfaces of a glass 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 in this window:

Button	Name
4	Create New Composition
	Edit Composition Details
	Copy Current Composition
×	Delete Current Composition
Num	Delete All Compositions
₹ <b>3</b>	Select Surfaces

Table 3.1: Buttons in the Layers window



For each composition, the *Details of Composition* dialog box is available. To open the dialog box, which is described in Chapter 4.2, click [Edit Composition Details...].

In Window 1.2 *Layers*, in the *Layers* section, you can define individual layers for the current composition. Column A *Layer Type* provides the three options **Glass**, **Foil**, and **Gas**.

You can select the materials from the library, which already contains a large number of materials. To open the material library, click the button shown on the left. You can also place the cursor in the relevant field of column B *Material Description*, and then click the appearing [...] button or press [F7].

In the *Filter* section of the material library (see Figure 3.4), the material category appropriate for the layer type selected in column A is preset.

You can reduce the selection possibilities of materials by using the drop-down lists *Standard group* or *Standard*. In the *Material to Select* list on the right, you can select a material and check its parameters in the lower part of the dialog box.

To import a material in the 1.2 *Layers* window, click [OK]. Alternatively, you can press [...] or double-click the material. Then, you can adjust all material parameters directly in the module.

	Material to Select			_
Naterial category group:	Material Description	Standard		
Glass and Foil	👻 🔲 Float Glass, Horizontal Glazing	TRLV:2	006-08	
	Float Glass, Vertical Glazing	TRLV:2	006-08	
Naterial category:	Rolled Glass, Horizontal Glazing	TRLV:2	006-08	
Glass	Rolled Glass, Vertical Glazing	TRLV:2	006-08	
itandard group:	Thermally Toughened Float Glass	TRLV:2	006-08	
2 .	Thermally Toughened Patterned Glass	TRLV:2	006-08	
i DIN	Thermally Toughened Enamelled Glass	TRLV:2	006-08	
itandard:	Heat Strengthened Float Glass	TRLV:2	006-08	
TRLV:2006-08	➡ Heat Strengthened Enamelled Glass	TRLV:2	006-08	
11129.2000-00	Laminated Float Glass, Horizontal Glazing	TRLV:2	006-08	
	Laminated Float Glass, Vertical Glazing	TRLV:2	006-08	
	Laminated Heat Strengthened Glass	TRLV:2	006-08	
	Laminated Heat Strengthened Enamelled Glass	TRLV:2	006-08	
Include invalid		1—		
Favorites only				7
laterial Properties	F	loat Glass Horiz	contal Glazing   TF	21.1/-2006
Main Properties				
Modulus of Elasticity		E	70000.0	
<ul> <li>Shear Modulus</li> </ul>		G	28455.3	N/mm <sup>2</sup>
Poisson's Ratio		v	0.230	
<ul> <li>Specific Weight</li> <li>Specific is at a Thermal Formula</li> </ul>		γ		kN/m <sup>3</sup>
<ul> <li>Coefficient of Thermal Expanded</li> <li>Additional Properties</li> </ul>	nsion	α	9.0000E-06	1/1
		Gallow	12.0	-
<ul> <li>Allowable Stress</li> </ul>				N/mm <sup>2</sup>

Figure 3.12: Material library

For the TRLV regulation (Technical regulation for the use of glazing with linear supports), the material library distinguishes between vertical and horizontal glazing. The following figure illustrates the difference. For glass types that are not distinguished in such a way, the parameters are the same for both glazing types.



15

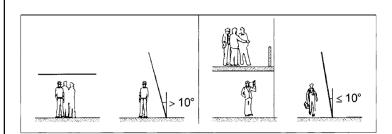


Figure 3.13: Horizontal glazing (left), vertical glazing (right) [6]

Individual layers can have a solid (glass or foil) or a gaseous state. If a composition contains a gas layer (that is, for insulating glass), then the program shows which side is considered as the external and which as the internal one (see Figure 3.14). This piece of information is important for entering further parameters in the 1.6 *ClimaticLoad Parameters* window. This module window appears in the navigator if you specify a gas layer (see Chapter 3.6, page 35).



A gas layer must always be enclosed on both sides by layers of a solid material (glass or foil).

2 LComp. 2         B         C         Module         Pages         E         C         C         C         Module         Pages         C         C         C         C         No.         E         F         G         C         C         C         Module         Pages         Specific Weight Thickness         Module         Pages         C         G         Module         Pages         C         C         C         Module         C         Images         C <thc< th=""> <thc< th="">         C</thc<></thc<>									;	.2 Layer
Leyers           Layer         A         B         C         D         E         F         G           Layer         Description         If Imit Cheess         Modulus of Bast.         Sheer Modulus         Poisson's Ratio         Specific Weight         Coef           1         Glass         Thermally Toughened Roat Glass         10.00         70000.000         28455.300         0.230         25.00           2         Gas         Dy Ar         10.00         70000.000         28455.300         0.230         25.00           3         Glass         Thermally Toughened Roat Glass         10.00         70000.000         28455.300         0.230         25.00           4         Thermally Toughened Roat Glass         10.00         70000.000         28455.300         0.230         25.00           5         -         -         -         -         -         -         -           6         - </td <td>position No. 2</td> <td>Compos</td> <td></td> <td></td> <td>List of Surfaces</td> <td></td> <td></td> <td></td> <td>Composition</td> <td>Current</td>	position No. 2	Compos			List of Surfaces				Composition	Current
A         B         C         D         E         F         G           Layer         Layer         Material         Thickness         Modulus of Elast.         Sheer Modulus         Poisson's Ratio         Specific Weight         Coef           1         Glass         Themaly Toughened Float Glass         10.00         70000.000         28455.300         0.230         25.00           2         Ggas         Dry Ar         10.00         70000.000         28455.300         0.230         25.00           3         Glass         Themaly Toughened Float Glass         10.00         70000.000         28455.300         0.230         25.00           4         5	\$				2		•• 🛅 🔤	▼	o. 2	2   Com
Layer         Material         Thickness         Modulus of Elast.         Shear Modulus         Poisson's Ratio         Specific Weight y (k1/m³)         Coeffic           1         Glass         Thermally Toughened Roat Glass         10.00         70000.000         28455.300         0.230         2.500         0.01           3         Glass         Thermally Toughened Roat Glass         10.00         70000.000         28455.300         0.230         2.500         0.01           4         Femaly Toughened Roat Glass         10.00         70000.000         28455.300         0.230         2.500         0.01           5         Image: State St										Layers
No.         Type         Description         t [mm]         E [MPa]         G [MPa]         v [-]         r [kN/m <sup>3</sup> ]         v           2         Gas         Thermally Toughened Plot Glass         10.00         70000.000         28455.300         0.230         25.00           3         Glass         Thermally Toughened Plot Glass         10.00         70000.000         28455.300         0.230         25.00           4         -         -         -         -         -         -         -         -         0.01           5         -	H 🔺								A	
Type         Thermally Toughened Roat Glass         Turning         Turning <thturning< th=""> <thturning< th="">         Tur</thturning<></thturning<>	ff. of Th. E	ht Coeff.	Specific Weight	Poisson's Ratio	Shear Modulus	Modulus of Elast.	Thickness	Material	Layer	Layer
2         Ggs         Dy Ar         10.00         0.01           3         Glass         Thermally Toughened Roat Glass         10.00         70000.000         28455.300         0.230         25:00           5         -	αт [1/К]	α.7	γ [kN/m <sup>3</sup> ]	v [-]	G [MPa]	E [MPa]	t [mm]	Description	Туре	No.
3         Gass         Thermally Toughened Roat Glass         10.00         70000.000         28455.300         0.230         25.00           4	9.0	5.00	25.00	0.230	28455.300	70000.000	10.00	Thermally Toughened Float Glass	Glass	1
4         2         1		0.01	0.01				10.00	Dry Air	Gas	2
5     -     -     -       7     -     -     -       8     -     -     -       9     -     -     -       •     III     -     -       •     IIII     -     -       •     IIII     -     -       •     IIIIIIIIIIIIIIIIIIIIIIIIIIIIIIIIIIII	9.0 😑	5.00	25.00	0.230	28455.300	70000.000	10.00	Thermally Toughened Float Glass	Glass	3
6     7	=								-	4
7       8       1										5
8										6
9       III         Image: State and Side       Info         External Side       1: Thermally Toughened Float Glass         2: Dry Atr       3: Thermally Toughened Float Glass         3: Thermally Toughened Float Glass       Surface weight:         0       0         1: Thermally Toughened Float Glass       Surface weight:         0.250 [NM/m <sup>2</sup> ]       Σ Thickness:         30.00 [mm]       Σ Surface weight:         0.500 [NM/m <sup>2</sup> ]       Directon										7
Image: Solution of the second seco										8
External Side  External Side External Side  External Side  External Side  External Side  External Side  External Side  External Side  External Side  External Side  Externa										9
External Side	•							III		•
External Side	🛐 😼	ð 💿 [	<b>e</b>							
1: Thermally Toughened Float Glass     Layer No.: 1       2: Dry Ar     Surface weight:     0.250 [NM/m <sup>2</sup> ]       3: Thermally Toughened Float Glass     Σ Thickness:     30.00 [mm]       5: Surface weight:     0.500 [NM/m <sup>2</sup> ]				•	Info					
Surface weight: 0.250 [kN/m <sup>2</sup> ] 2: Diy Ar 3: Thermally Toughened Float Glass 3: Thermally Toughened Float Glass 5: Thickness: 30.00 [mm] 2: Surface weight: 0.500 [kN/m <sup>2</sup> ] 2: Direction				ver No.: 1	Lav				E	
•         Σ Thickness: 30.00 [mm]           •         Σ Surface weight: 0.500 [kN/m²]           •         Local Axis z Direction		√/m²]	0.250 [kN/m <sup>2</sup> ]					<ul> <li>2: Dry Air</li> </ul>		
Local Axis z     Directon						t Glass	Toughened Floa	- 3: Thermally		
e Local Axis z Directon		m]	30.00 [mm]	hickness:	ΣT					
e Local Axis z Directon		√/m <sup>2</sup> 1	0.500 [kN/m <sup>2</sup> ]	urface weight:	Σ.5					
Direction										
Direction										
Direction								~		
Direction										
Direction										
Direction										
Direction										
								لو		
					Direction					
Internal Side Bottom					Bottom			nternal Side		
Bulum					Dottom					

Figure 3.14: Window 1.2 Layers - insulated glass

Below the table in Window 1.2, you can find a number of useful buttons. The buttons have the following functions:

Button	Name	Function
	Load saved layers	Loads a previously saved composition
	Save layers as	Saves the composition entered in Window 1.2 which can then be loaded in other RF-GLASS models
×	Delete all layers	Deletes all data in Window 1.2.
	Import material from library	Opens the dialog box Material Library



0	Show layer stiffness matrix elements	Displays elements of the stiffness matrix (see Chapter 2.3, page 12).
0	Show extended stiffness matrix elements	Displays elements of the global stiffness matrix (see Chapter 2.3, page 12).
۲	Jump to graphic to change view	Jumps to the RFEM user interface allowing for a graphical evaluation without exiting RF-GLASS
	Export to MS Excel/ OpenOffice.org Calc	Exports contents of a current module window to MS Excel or OpenOffice.org Calc → Chapter 7.2, page 69
<b>B</b>	Import from Microsoft Excel/ OpenOffice.org Calc	Imports contents of a MS Excel or OpenOffice.org Calc table to Window 1.2

Table 3.2: Button in Window 1.2 Layers

In the lower right part of the 1.2 *Layers* window, you can find information on the weight of the selected layer as well as the total thickness and weight of a model.



#### Line Supports 3.3 1.3 Line Supports Current Composition List of Surfaces Composition No. 1 1 | Comp. 1 • • E 🔄 🔄 🗙 🛋 1 \$ Support Type Support Suppor No. On Lines No Туре Hinged - type 1 1-4 2 Hinged - type 2 Hinged - type 3 Hinged - type 3 Hinged - type 4 Hinged - type 5 Hinged - type 6 Hinged - type 7 Symm Rigid User-defined ۵ 🐧 🛋 Layer No. Reference lkNm/°/m Support Rotatio Spring [kN/m<sup>2</sup>] β [°] syster Loca 0.00 0.00

Figure 3.15: Window 1.3 Line Supports

If you select the *Local* calculation in Window 1.1, the analysis in RF-GLASS requires a precise structural model. To this end, you can choose from nine types of predefined line supports or define your own type. The supports entered in RF-GLASS are used only for this module; they do not influence RFEM specifications.

In column A On Lines No., you specify the lines at which the support acts. In column B, you can select a standard *Support Type* (*Hinged-type 1* through *type 7*, *Symmetry*, and *Rigid*), or specify a user-defined support. The user-defined support is to be specified in the lower table of this window. A dynamic graphic shows the locations of the line supports on the layers, allowing you to check your entries. All predefined line supports are related to the local coordinate system that is defined for RF-GLASS in the following way: Axis *x* is the center line of the selected line, axis *y* is in the plane of a surface defined in RFEM, and axis *z* is perpendicular to the RFEM surface.

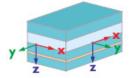
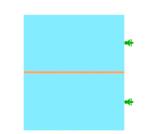


Figure 3.16: Local coordinate system of RF-GLASS

5

For laminated glass, there is a difference between 2D and 3D calculation with regard to boundary conditions for predefined line supports of the type *Hinged* (type 1, 3, 5, 7). If the calculation is in 2D (according to the plate theory), the supports are hinged. In the 3D calculation, however (solid model), the supports are partially rigid. The following figure illustrates the difference:



Dlubal

Figure 3.17: Line supports of the type Hinged - type 5: 2D calculation (left) and 3D calculation (right)

The predefined support types are explained in the following table:

Hinged - type 1						
2D calculation	3D calculation	Boundary conditions				
xo→y zv	x → y z → Boundary conditions on center lines of layers of type Glass	$u_x = u_y = u_z = 0$ $\varphi_z = 0$				
	Hinged - type 2					
2D calculation	3D calculation	Boundary conditions				
xo→y zv	xo→y zv	$u_x = u_y = u_z = 0$ $\varphi_z = 0$				
	Boundary conditions on bottom edge of lowest layer of type Glass <i>Hinged - type 3</i>					
2D calculation	3D calculation	Boundary conditions				
	Boundary conditions on center lines of layers of type Glass	$u_z = 0$ $\varphi_z = 0$				
	Hinged - type 4					
2D calculation	3D calculation	Boundary conditions				
xo→y zv	x → y z ↓ Boundary conditions on bottom edge of lowest layer of type Glass	$u_z = 0$ $\varphi_z = 0$				



	Hinged - type 5	1
2D calculation	3D calculation	Boundary conditions
xœ→y zv	xo→y z↓	
		$u_x = u_y = 0$
	-+	$\varphi_z = 0$
	Boundary conditions on center lines of layers of type Glass	
	Hinged - type 6	
2D calculation	3D calculation	Boundary conditions
xœ→y zv	xo→y z↓	
		$u_x = u_y = 0$
		$\varphi_z = 0$
	Boundary conditions on bottom edge of lowest layer of type Glass	
	Hinged - type 7	
2D calculation	3D calculation	Boundary conditions
xœ→y zv	xo→y z↓	$u_x = u_z = 0$ $\varphi_y = \varphi_z = 0$
	Boundary conditions on center lines of layers of type Glass	
	Symmetry	
a model. The condition contain terial of the side surface, which	ommended for cases when you want ns not only correct line supports but a n does not cause stiffening of the mod	lso an appropriate ma- el.
2D calculation	3D calculation	Boundary conditions
xœ→y zv	Boundary conditions on all lines of all layers	$u_y = 0$ $\varphi_x = \varphi_z = 0$



	Rigid	
2D calculation	3D calculation	Boundary conditions
X⊕→Y ZV	X → Y Z ↓ Boundary conditions on all lines of all layers	$u_x = u_y = u_z = 0$ $\varphi_x = \varphi_y = \varphi_z = 0$

Table 3.3: Predefined types of line supports

User-defined supports can be entered in the lower table (see Figure 3.15) – for glass layers which contain the lines of this table. In this table, you select the *Support Location* and define the *Reference System*. You can choose either the local coordinate system of RF-GLASS (can be defined directly in the table) or the global coordinate system. Furthermore, you can specify a rotation of the local coordinate system about axis *x* with the angle  $\beta$  and define individual degrees of freedom.

As for the predefined supports, a graphic illustrates the selected lines with the chosen line supports.

In Window 1.3, three buttons are available, which have the following functions:

Button	Name	Function
۲	View mode	Jumps to the RFEM work window for a graphical check without exiting RF-GLASS
1	Graphical selection	Allows you to graphically select a line in the RFEM work window
X	MS Excel	Exports contents of a current window to MS Excel or OpenOffice.org Calc ( $\rightarrow$ Chapter 7.2, page 69)

Table 3.4: Buttons in the window Line Supports



### 3.4 Nodal Supports

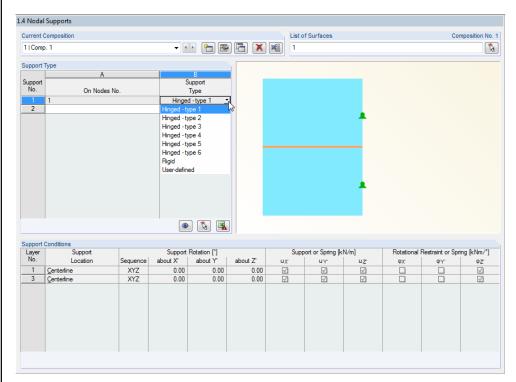
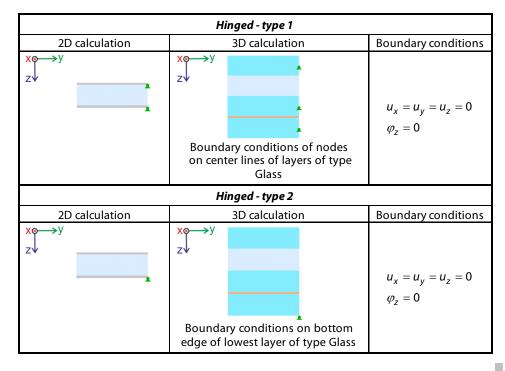


Figure 3.18: Window 1.4 Nodal Supports

In this module window, you can define nodal supports. In column A *On Nodes No.*, you select the nodes, at which the support acts. In column B, you can select a standard *Support Type* (*Hinged - type 1* to *type 6*, *Rigid*), or a user-defined support. The user-defined support is to be specified in the lower table. A dynamic graphic shows the exact locations of the nodal supports at the individual layers, allowing you to check your entries.

The predefined support types are explained in the following table:





	Hinged - type 3						
2D calculation	3D calculation	Boundary conditions					
x ↔ y z ↓	xo→y zv	$u_z = 0$					
	Boundary conditions of nodes on center lines of layers of type Glass	$\varphi_z = 0$					
Hinged - type 4							
2D calculation	3D calculation	Boundary conditions					
x ↔ y z ↓	xo→y zv	<i>u<sub>z</sub></i> = 0					
	Boundary conditions on bottom edge of lowest layer of type Glass	$\varphi_z = 0$					
	Hinged - type 5						
2D calculation	3D calculation	Boundary conditions					
xo→y z↓	xo→y zv	$u_x = u_y = 0$					
	Boundary conditions of nodes	$\varphi_z = 0$					
on center lines of layers of type Glass <i>Hinged - type 6</i>							
2D calculation	3D calculation	Boundary condi- tions					
xo→y zv	xo→y zv						
		$u_x = u_y = 0$ $\varphi_z = 0$					
	Boundary conditions on bottom edge of lowest layer of type Glass						
	Rigid						
2D calculation	3D calculation	Boundary condi- tions					
xo→y zv	X						
	Roundary conditions of nodes on all	$u_x = u_y = u_z = 0$ $\varphi_x = \varphi_y = \varphi_z = 0$					
	Boundary conditions of nodes on all center lines of all layers						

Table 3.5: Predefined types of nodal supports



User-defined supports can be entered in the lower table (see Figure 3.18) – for glass layers which contain nodes of this table. First, you specify the *Support Location* and, if necessary, a *Support Rotation*. Then, you can define the degrees of freedom in detail.

Window 1.4 provides the same buttons as Window 1.3 (see Table 3.4, page 30).



### 3.5 Boundary Members

Current Composition					List of Surfaces Composition No. 1							
1   Comp. 1		•	•• 🛅	s) 🖪	X	1						1
Assigning o	of Reference Lengths to Surfaces											
	А	B	C	D	E	F	G	H		J	K	L
Member No.				Cross-sec		Member		Relea		Eccentricity	Division	Comme
	On Lines No.	Layer	Location	Start	End	Туре	β["]	Start	End	No.	No.	
1 1		3	<u>C</u> enterline	1	1	<u>A</u> ngle	0.00	0	0	0	0	
2												
3												
4												
5												
6												
7												
8												
9												
10												
11												
12												
13												
14 15												
15												
17												
1/												
18												
20												
20												-
21												-
22						_						-
23												-
25												
25												-
20												
28												
20												-

Figure 3.19: Window 1.5 Boundary Members

In this window, you can define members at the boundary of the glass surface.

In column A *On Lines No.*, you select the line on which the member lies. In column B, you can specify the *Layer* on whose center line the member is located. Only layers of the type "Glass" are available for selection. In column C, you specify the *Location* of the boundary members at the glass layer (*Upper/Lower edge, Centerline*).

In columns D and E, you select the member's *Cross-section No*. at the start and end of the line. The cross-section must be defined in RFEM beforehand. In columns F and G, you can enter a possible *Member Rotation*. Columns G and H are used for the definition of *Releases* at member ends. In column J, you can define an *Eccentricity*, and in column K a *Division*.

In column L, you can write your own Comment.

Window 1.5 provides the same buttons as Window 1.3 (see Table 3.4, page 30).



### 3.6 Climatic Load Parameters

This window appears only if at least one gas layer is selected in Window 1.2, that is, if you specified an insulating glass (see Chapter 3.2 *Layers*).

Single layer and laminated glass without gas layers can be loaded only by load cases defined in RFEM (defined in Window 1.1). Insulated glass, on the other hand, can also be loaded by climatic loads. These are defined in Window 1.6.

Current (	Composition				List of Surfa	ices	Composition No.
2   Comp	o. 2		•••	2 🔁 🗙	2		3
limatic L	.oad Parameter	s - Summer					
Use							
		Manufacturing		Mo		Difference	
Tempera	ture:	19.0 🚔 [°C]	Temperature Exte	emal:	28.0 🚔 [°C]	9.0 🔶 [°C]	
			Gas	e -	39.0 🚔 [°C]	20.0 → [°C]	
		Manufacturing	Inte	mal:	28.0 ≑ [°C]	9.0 🔶 [°C]	
tmosph	eric pressure:	0.103 🚔 [N/mm <sup>2</sup> ]	Atmospheric pressu	ure:	0.101 🚔 [N/mm <sup>2</sup> ]	-0.002 🔶 [N/mm <sup>2</sup> ]	
Vititude:		0.0 🚔 [m]	Altitude:		0.0 ≑ [m]	0.0 × [m]	
empera	ture:	Manufacturing	Temperature Exte Gas Inte		unt -10.0 🛫 [°C] 2.0 丈 [°C] 19.0 🐳 [°C]	Difference -37.0 (*) [*C] -25.0 (*) [*C] -8.0 (*) [*C]	
ter e en la	eric pressure:	Manufacturing	Atmospheric pressu		0.103 🚔 [N/mm <sup>2</sup> ]	0.004 ⊕ [N/mm <sup>2</sup> ]	
Atitude:	enc pressure.	0.0 ÷ [m]	Attitude:		0.0 👻 [m]	0.0 × [m]	
orce Lo	ad Distribution				Options		
No.		Description	Load Part [%] o External	n Glass Side Internal	Calculation accord	ling to DIN 18008-2:2010-12, Appendix A tangular surfaces supported by line support	Hinged - type 7
LC1	Self-Weigth		100.0	0.0			
LC2			100.0	0.0			

Figure 3.20: Window 1.6 Climatic Load Parameters for Insulating Glass

*Climatic Load Parameters* are divided into summer and winter loads. The layout of both parameter sets is the same. Therefore, individual climatic load parameters are explained on the example of **Climatic Load Parameters - Summer** (see also the following figure).

First, you have to select the Use check box for the relevant set of parameters.

On the left side, you specify the load parameters *Temperature*, *Atmospheric pressure*, and *Alti-tude* at the time of the glass *Manufacturing*. On the right, you enter the parameters that are valid after the glass *mount*, that is, when the glass pane is used.

The *Temperature*, which is the same for all insulating glass components during manufacturing, can differ for these components after the mount. External temperature, internal temperature, and gas temperature are to be defined differently for the designs.

The *Difference* between the conditions during manufacturing and after the mount is then displayed on the right.

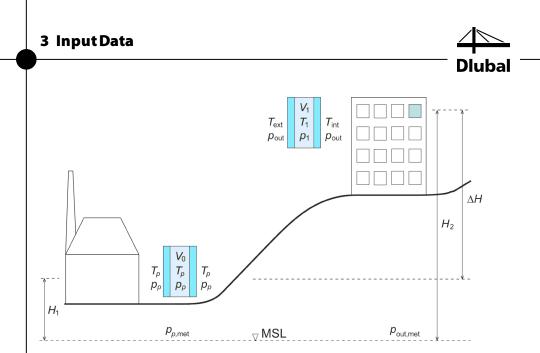


Figure 3.21: Climatic load parameters for manufacturing (left), mount (right), MSL = mean sea level

In the *Load Distribution* section, you can specify how the loads defined in RFEM are distributed to the external and internal glass side. The actions selected for the design are already preset. The 1.2 *Layers* window specifies the position of the sides:

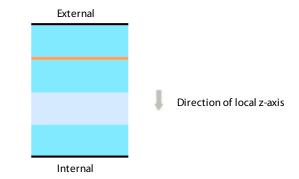


Figure 3.22: External and internal side of insulating glass

For special models, you can use a simplified calculation according to DIN 18008-2:2010-12, Annex A or according to the German technical rules TRLV, Annex A, by selecting the relevant check box in the *Settings* section. For this, the following conditions must be satisfied:

- Rectangular surface without openings
- Exactly one gas layer
- Line support of the type *Hinged type 7* on all boundary lines
- Loading only by surface load

Kirchhoff's plate theory and the linear static analysis are always applied for the calculation according to TRLV, Annex A (see [1], [2], [5]).

On the bottom right, there are three buttons with the following functions:

Button	Name	Function
	Default	Sets climatic load parameters according to the saved default settings
	Set as Default	Saves current climatic load parameters as new default
3	Load Default Dlubal Values	Sets the original presetting according to DIN 18008-2:2010-12, Table 3

Table 3.6: Buttons in window 1.6 Climatic Load Parameters



### 3.7 Load Duration

oigini	ent of Reference Lengths to					-
oad-	A	B	C	D	E Coefficient	F
bad- ing	Description	LC Type	Load Duration Class - LDC	Manual	Coefficient k mod	Comment
		Permanent				Comment
.C1 .C2	Self-Weigth	Snow / ice	Permanent		0.25	
		Show / Ice	<u>M</u> edium-term		0.40	

Figure 3.23: Window 1.7 Load Duration

If you design according to the standard *DIN 18008:2010-12* and select a load case in the *Ultimate Limit State* tab of Window 1.1, the 1.7 *Load Duration* window is displayed.

This window lists all load cases, load combinations, and result combinations that were selected for the design. Columns A and B show the *Description* and *LCType* defined in RFEM.

In column C, you can specify the *Load Duration Class - LDC*. These classes follow the standard DIN 18008-2:2010-12, Table 6. The classification of load combinations automatically follows the governing load. If you select an entry in column C, the corresponding coefficient  $k_{mod}$  is automatically set in column E.

You can check the values of the coefficients  $k_{mod}$  the *Standard* dialog box. To open it, click

In the bottom right corner, you find the [Export] button that allows you to export the table

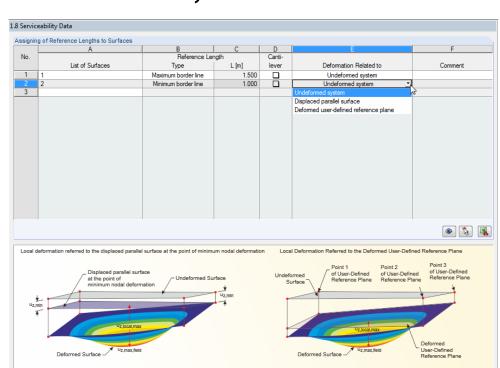
Standard

3

Standard.

contents to MS Excel or OpenOffice.org Calc.





### 3.8 Serviceability Data

Figure 3.24: Window 1.8 Serviceability Data

The 1.8 Serviceability Data window is the last input window.

In column A, you specify the surfaces whose deformations you want to analyze.

In column B, you choose the type of the *Reference Length L*. If you select the *Maximum border line*, the length of the longest boundary line of the selected surface is set automatically.

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

In column E, you specify the system to which deformation is related. If the calculation type *Local – Individual glass surfaces* is selected, only the option *Undeformed system* is available.

In column F, you can write your own Comment.

The specifications of this window are important for the correct application of the limit deformations. You can check and, if necessary, adjust these limit values for the serviceability limit state design in the *Standard* dialog box (see Chapter 4.3).

Standard

# 4. Calculation

Calculation Details... Before you start the [Calculation], it is necessary to check the detail settings for the design. To open the relevant dialog box, which is described in Chapter 4.1, click [Details].

# 4.1 Details

The Details dialog box is divided in the following tabs:

- Stresses
- Results

The following buttons are available in all tabs:

Button	Name	Function
0.00	Units and Decimal Places	Opens the dialog box <i>Units and Decimal Places</i> → Chapter 7.1, page 68
3	Dlubal Standard Values	Sets the original Dlubal settings in the <i>Details</i> dialog box
	Default	Sets all parameters in the <i>Details</i> dialog box according to the previously saved default settings
	Set as Default	Saves the current settings as user-defined standard

Table 4.1: Buttons in the Details



### 4.1.1 Stresses

Stresses Results			
To Display		Equivalent Stresses According to	
Top/Bottom Layer         Image: Constraint of the system         Image: Constraint of the system	Middle Layer          Image: Constraint of the system         Image: Constr	<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 4.1: Dialog box Details, tab Stresses

#### **To Display**

偱

×E

In this section, you choose which stresses you want to display in the result tables by selecting the appropriate check boxes. The stresses are divided in the categories *Top/Bottom Layer* and *Middle Layer*. The buttons [Select All] and [Deselect All] facilitate the selection.

The basic stresses  $\sigma_x$ ,  $\sigma_y$ ,  $\tau_{xy}$ ,  $\tau_{xz}$ ,  $\tau_{yz}$  are calculated by the finite element method in RFEM. Further stresses are calculated from these basic stresses in the RF-GLASS module. Table 4.2 presents the formulas valid for a single layer plate.

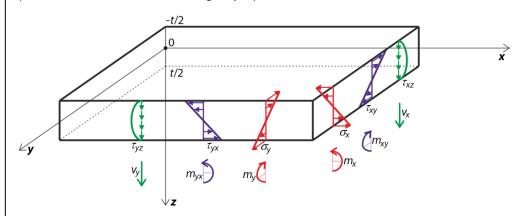


Figure 4.2: Basic stresses and sign convention for single layer plate subjected to bending



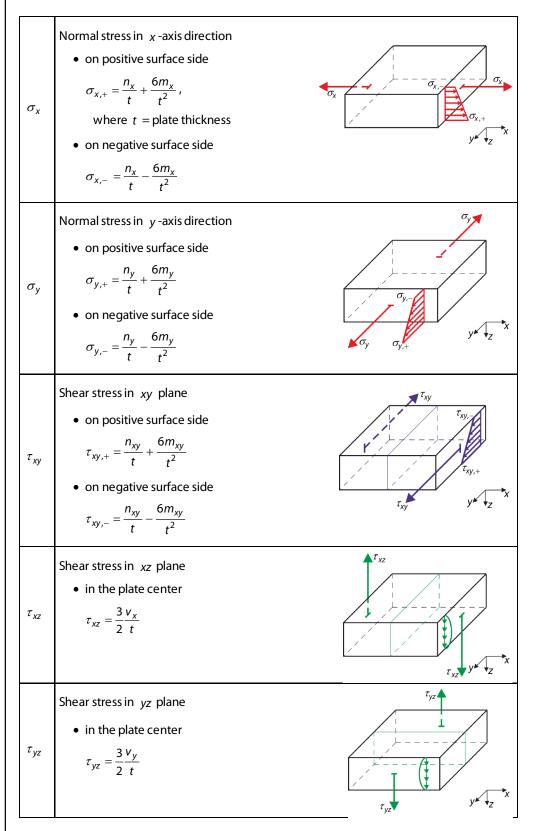


Table 4.2: Basic stresses

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

5

$$\boldsymbol{\varepsilon}_{\text{tot}}^{\mathsf{T}} = \left\{ \frac{\partial \varphi_{y}}{\partial x}, -\frac{\partial \varphi_{x}}{\partial y}, \frac{\partial \varphi_{y}}{\partial y} - \frac{\partial \varphi_{x}}{\partial x}, \frac{\partial w}{\partial x} + \varphi_{y}, \frac{\partial w}{\partial y} - \varphi_{x}, \frac{\partial u}{\partial x}, \frac{\partial v}{\partial y}, \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right\}$$
(4.1)

The strains in individual layers are calculated according to the following relation:

ſ

$$\boldsymbol{\varepsilon}(z) = \begin{cases} \varepsilon_{x} \\ \varepsilon_{y} \\ \gamma_{xy} \end{cases} = \begin{cases} \frac{\partial u}{\partial x} \\ \frac{\partial v}{\partial y} \\ \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \end{cases} + z \begin{cases} \frac{\partial \varphi_{y}}{\partial x} \\ -\frac{\partial \varphi_{x}}{\partial y} \\ \frac{\partial \varphi_{y}}{\partial y} - \frac{\partial \varphi_{x}}{\partial x} \end{cases},$$
(4.2)

where z: is the coordinate in z-axis direction, where the stress value is requested.

If there is, for example, *i* - layer, the stress is calculated according to the following relation:

$$\boldsymbol{\sigma}(z) = \boldsymbol{d}_i \boldsymbol{\varepsilon}(z) \tag{4.3}$$

where  $d_i$ : is the partial stiffness matrix of the *i* - th layer.

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

,

2



Table 4.4 shows the formula for the calculation of the maximum and equivalent stresses.

	Principal stress				
$\sigma_1$	$\sigma_1 = \frac{\sigma_x + \sigma_y + \sqrt{\left(\sigma_x - \sigma_y\right)^2 + 4\tau_{xy}^2}}{2}$				
	Principal stress				
$\sigma_2$	$\sigma_2 = \frac{\sigma_x + \sigma_y - \sqrt{\left(\sigma_x - \sigma_y\right)^2 + 4\tau_{xy}^2}}{2}$				
	Angle between the local $x$ -axis and the direction of the first principal stress				
	$\alpha = \frac{1}{2} \operatorname{atan2}(2\tau_{xy}, \sigma_x - \sigma_y), \ \alpha \in (-90^\circ, 90^\circ]$				
	Function atan2 is implemented in RFEM as follows: $\sigma_2$ arctan $\frac{y}{x}$ $x > 0$				
α	$\arctan \frac{y}{x} + \pi  y \ge 0, x < 0$				
	atan2(y,x) = $\begin{cases} \arctan \frac{y}{x} - \pi & y < 0, x < 0 \\ + \frac{\pi}{2} & y > 0, x = 0 \\ - \frac{\pi}{2} & y < 0, x = 0 \\ 0 & y = 0, x = 0 \end{cases}$				
	$+\frac{1}{2}$ $y > 0, x = 0$				
	$ \begin{array}{cccc}             2 & y < 0, x = 0 \\             2 & y = 0, x = 0 \end{array} $				
	Equivalent stress according to VON MISES, HUBER, HENCKY (Shape modification hypothesis)				
	$\sigma_{\rm eqv} = \sqrt{\sigma_x^2 + \sigma_y^2 - \sigma_x \sigma_y + 3\tau_{xy}^2}$				
	Equivalent stress according to TRESCA (Maximum shear stress criterion)				
~	$\sigma_{\text{eqv}} = \max\left[\sqrt{\left(\sigma_x - \sigma_y\right)^2 + 4\tau_{xy}^2}, \frac{\left \sigma_x + \sigma_y\right  + \sqrt{\left(\sigma_x - \sigma_y\right)^2 + 4\tau_{xy}^2}}{2}\right]$				
$\sigma_{ m eqv}$	Equivalent stress according to RANKINE, LAMÉ (Maximum principal stress criterion)				
	$\sigma_{\text{eqv}} = \frac{\left \sigma_x + \sigma_y\right  + \sqrt{\left(\sigma_x - \sigma_y\right)^2 + 4\tau_{xy}^2}}{2}$				
	Equivalent stress according to BACH, NAVIER, ST. VENANT, PONCELET (Principal strain criterion)				
	$\sigma_{\text{eqv}} = \max\left[\frac{1-\nu}{2}\left \sigma_{x} + \sigma_{y}\right  + \frac{1+\nu}{2}\sqrt{\left(\sigma_{x} - \sigma_{y}\right)^{2} + 4\tau_{xy}^{2}}, \nu\left \sigma_{x} + \sigma_{y}\right \right]$				
Table 4.4: Stre	esses				



#### **Equivalent stresses**

You can determine the equivalent stresses in four different ways.

#### Von Mises, Huber, Hencky (shape modification hypothesis)

The shape modification hypothesis is also known as HMH (HUBER, VON MISES, HENCKY). The equivalent stresses are calculated as follows:

$$\sigma_{\rm eqv} = \sqrt{\sigma_x^2 + \sigma_y^2 - \sigma_x \sigma_y + 3\tau_{xy}^2}$$
(4.4)

#### Tresca (maximum shear stress criterion)

This equivalent stress is generally defined by using the relation:

$$\sigma_{\text{eqv}} = \max(|\sigma_1 - \sigma_2|, |\sigma_1 - \sigma_3|, |\sigma_2 - \sigma_3|)$$
(4.5)

which on the condition  $\sigma_3 = 0$  can be simplified to:

$$\sigma_{\text{eqv}} = \max(|\sigma_1 - \sigma_2|, |\sigma_1|, |\sigma_2|) \tag{4.6}$$

This results in the following equation:

$$\sigma_{\text{eqv}} = \max\left[\sqrt{\left(\sigma_x - \sigma_y\right)^2 + 4\tau_{xy}^2}, \frac{\left|\sigma_x + \sigma_y\right| + \sqrt{\left(\sigma_x - \sigma_y\right)^2 + 4\tau_{xy}^2}}{2}\right]$$
(4.7)

#### 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.

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

which on the condition  $\sigma_3 = 0$  can be simplified to:

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

This results in the following equation:

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

#### Bach, Navier, St. Venant, Poncelet (principal strain criterion)

The principal deformation hypothesis is also known as the equivalent stress according to Bach. It is assumed that the failure occurs in the direction of the greatest strain. The equivalent stress is determined as follows:

$$\sigma_{\mathsf{eqv}} = \max\left(\left|\sigma_1 - \nu(\sigma_2 + \sigma_3)\right|, \left|\sigma_2 - \nu(\sigma_1 + \sigma_3)\right|, \left|\sigma_3 - \nu(\sigma_1 + \sigma_2)\right|\right)$$
(4.11)

For  $\sigma_3 = 0$ , we can simplify:

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

This results in the following equation:

$$\sigma_{\text{eqv}} = \max\left[\frac{1-\nu}{2}\left|\sigma_x + \sigma_y\right| + \frac{1+\nu}{2}\sqrt{\left(\sigma_x - \sigma_y\right)^2 + 4\tau_{xy}^2}, \nu\left|\sigma_x + \sigma_y\right|\right]$$
(4.13)



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

4.1.2	Results
-------	---------

ails		
Stresses Results		
Display Result Tables	Results in	
2.1 Max Stress/Ratio by Loading	Image: FE mesh points	
2.2 Max Stress/Ratio by Surface	Grid points	
2.3 Stresses in All Points		
2.4 Line Support Reactions		
2.5 Nodal Support Reactions		
3.1 Max Displacements		
3.2 Gas Pressure		
4.1 Parts List		
Only for surfaces to be designed		
Of all surfaces		
2 🐻 🕥 🖷 🖷		OK Cancel

Figure 4.3: Dialog box Details, tab Results

#### **Display Result Tables**

In this section, you can specify which result tables you want to display (stresses, reactions, displacements, gas pressure, parts lists).

The result windows are described in Chapter 5 Results, page 55.

#### **Results in**

Stresses and displacements are displayed in all FE mesh points by default. The results can also be shown in the grid points (see RFEM manual, Chapter 8.12). The grid points can be defined in RFEM as property of a surface.

For small surfaces, the default grid point spacing of 0.5 m can result in only a small number of grid points (or even just one result grid point in the origin). In this case, the spacing of grid points should be adapted to the surface dimensions in RFEM in order to create more grid points.



## 4.2 Details of Composition

alculation / Modeling	
Iethod of Analysis         Linear static analysis         Large deformation analysis (nonlinear)         Newton-Raphson with constant stiffness matrix         Newton-Raphson         Number of load increments:	Stiffness Reduction Factors For shear stiffness elements K44 : 1.00 (m) [] K55 : 1.00 (m) [] Plate Bending Theory (a) Mindlin
Iodeling of Laminated Glass	Kirchhoff
<ul> <li>3D if ratio (G t / G<sub>f</sub> t<sub>f</sub>) is greater than:</li> <li>3D</li> <li>2D</li> </ul>	Insulating Glass Unit
calculation Options	Modulus of elasticity E : [MPa]
Save created temporary models Consider coupling	Shear modulus     G :     Image: [mpa]       Poisson's ratio     v:     Image: [mp]       Width     b :     Image: [mm]
Target FE length: [m] Change standard settings Precison of convergence criteria of	b.
nonlinear calculation:	Secondary seal
	Number of finite element layers in gas layers:

Figure 4.4: Dialog box Details, Calculation / Modeling for Local calculation type

To open the *Details of Composition* dialog box, click the [Edit Composition Details] button available in the upper part of Windows 1.2 through 1.6.

#### **Method of Analysis**

15

This section controls whether a calculation is carried out according to the *Linear static analysis* or the *Large deformation analysis (nonlinear)*. The linear static analysis is preset.

However, if you create an insulating glass (with an intermediate gas layer), the program automatically switches to the large deformation analysis: The intermediate gas layer introduces a non-linearity to this model, which causes differences between the linear static and large deformation analysis even for small load values. Here, calculation according to the large deformation analysis gives more precise results. The iterative calculation of the solid elements is done according to the methods *Newton-Raphson with constant stiffness matrix* or *Newton-Raphson*. The differences between these methods are described in the RFEM manual, Chapter 7.3.1.1.

#### **Modeling of Laminated Glass**

As described in Chapter 2.2.1, the standard theory may give incorrect results. If the ratio  $G \cdot t / G_f \cdot t_f$  is greater than the defined limit value, the calculation proceeds in *3D*. In this term, *G* is the shear modulus of glass, *t* is the thickness of the glass layer,  $G_f$  is the shear modulus of the foil, and  $t_f$  is the foil thickness.

You can also set the type of calculation manually to 2D or 3D. Of course, 3D calculations are more accurate, but also more time-consuming.



If you create an insulating glass (with an intermediate gas layer), the 3D calculation is set automatically.

#### **Calculation Options**

In this section, you can specify general settings for the RF-GLASS calculation. The first check box of this section allows you to Save created temporary models: As already mentioned in Chapter 3.1 General Data on page 17, for the Local calculation type the supports and boundary members are entered directly in RF-GLASS without influencing the rest of the RFEM model. If the check box is selected, these models are saved as new RFEM files when you save the entered data in RF-GLASS. They can be found in the same project folder as the original file and are marked by the addition RF\_GLASS in the file name. After opening such a file, you can graphically check the RF-GLASS models in RFEM with all supports, members, solids, etc.

The Consider coupling check box is automatically selected in the case of laminated glass with foils so that the shear resistance of the laminating foil is considered. Shear coupling of layers was already mentioned in Chapter 2.2.1 Laminated Glass on page 9.

If you Activate FE mesh refinement, you can manually define the Target FE length.

If you select the Change standard settings check box, you can influence the precision of the convergence criteria for the nonlinear calculation. The value 1.0 is set as default here. The minimum allowable value is 0.01, the maximum value is 100.

#### **Stiffness Reduction Factors**

In the section Stiffness Reduction Factors, you can reduce shear stiffness matrix elements D<sub>44</sub> and  $D_{55}$  by using reduction factors  $K_{44}$  and  $K_{55}$ . The correction is possible only for 2D calculation.

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

n	1 <sub>x</sub>		$\left[ D_{11} \right]$	D <sub>12</sub>	0	0	0	0	0	0 ]	$\kappa_{\chi}$	
n	n <sub>y</sub>			D <sub>22</sub>	0	0	0	0	0	0	ĸy	
n	n <sub>xy</sub>				D <sub>33</sub>	0	0	0	0	0	κ <sub>xy</sub>	
ļv	x					K <sub>44</sub> D <sub>44</sub>	0	0	0	0	γ <sub>xz</sub>	(4.14)
Ìv	y	> =					$K_{55}D_{55}$	0	0	0	γ <sub>yz</sub>	> (4.14)
n	x				sym.			D <sub>66</sub>	D <sub>67</sub>	0	Ex	
n	y								D <sub>77</sub>	0	$\varepsilon_y$	
ln	xy J		L							D <sub>88</sub> ]	γ <sub>xy</sub>	

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

#### Plate Bending Theory

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

- Mindlin or •
- Kirchhoff •

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

$$\tau_{xz,\max} = \frac{3}{2} \frac{v_x}{t} = 1.5 \frac{v_x}{t}$$
(4.15)

### **4** Calculation



$$\tau_{yz,\max} = \frac{3}{2} \frac{v_y}{t} = 1.5 \frac{v_y}{t}$$
(4.16)

#### **Insulating Glass Unit**

This dialog section is accessible only for insulating glass. After selecting the *Consider secondary seal* check box, you can enter the material properties and the width of a secondary seal.

The following figure illustrates the individual components of insulating glass.

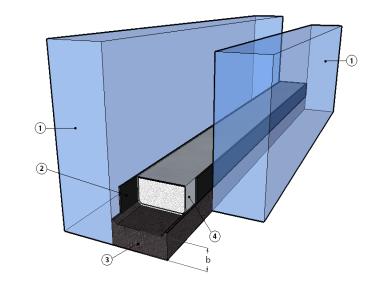


Figure 4.5: Insulating glass with 1) glass pane, 2) primary seal, 3) secondary seal, and 4) spacer

If necessary, you can adjust the Number of finite element layers in gas layers for insulating glass.



## 4.3 Standard

Standard - DIN 18008:2010-12				×
General				
Partial Factors for Material Properties		Serviceability Limits (Deflection	ons)	
Design situation:		Type of combination:		Cantilevers
- Persistent and Transient		- Characteristic	L/ 100 🜩 🕨	Lc/ 50 🔹 🕨
- For thermally toughened glass	ум: 1.50 ⇒ [-]	- Frequent	L/ 100 🜩 🕨	Lc/ 50 🜩
- For other glass	γм: 1.80 ⇒ [-]	- Quasi-permanent	L/ 100 🜩 🕨	Lc/ 50 📚
- Accidental	γм: 1.00 ★ [-]			
Construction Factor		Insulating Glass Unit		
For thermally toughened glass	kc: 1.00 🗭 [·]	LC Factor for climatic load:	Summer	Winter
For other glass	kc: 1.00 🗭 [-]	- Temperature:	1.00 🕀 📔 [-]	1.00
		- Atmospheric Pressure:	1.00 🔃 [-]	1.00 🗭 [-]
Modification Factor		- Altitude:	1.00 🜩 [-]	1.00 🗭 [-]
Load duration class (LDC):	kmod : 0.25 ⇒ [-]			
- Permanent				
- Middle	kmod: 0.40 ↔ [-]			
- Short	k <sub>mod</sub> : 0.70 ↓ [-]			
🔎 🔤 🕥 📭 🕅 🗙				OK Cancel

Standard

Figure 4.6: Dialog box Standard

To open the *Standard* dialog box, click [Standard].

In the upper right corner of Window 1.1 *General Data*, you select the standard from which the parameters will be applied for the design and the limit values of the deflection.

The following standards can be selected:

- DIN 18008:2010-12
- TRLV:2006-08
- None

### 4.3.1 Standard – DIN 18008:2010-12

For standard *DIN 18008:2010-12* the design value (limit stress) dependents on the type of the glass. Therefore in the 1.2 *Layers* window you have to set whether glass is thermally toughened or not.

For thermally toughened glass the design stress value ( $\sigma_{\text{limit,d}}$ ) is calculated from the characteristic limit stress value ( $\sigma_{\text{limit,k}}$ ), according to the following relation:

$$\sigma_{\text{limit,d}} = \frac{k_c \cdot \sigma_{\text{limit,k}}}{\gamma_M}$$

where  $k_c$ : is the construction factor

 $\sigma_{{\rm limit},k}$  : is the characteristic limit stress value set in the 1.2 Layers window

 $\gamma_M$ : is the partial factor for thermally toughened glass

#### 4 Calculation

The design stress value for glass which is not thermally toughened is calculated according to the following relation:

$$\sigma_{\text{limit,d}} = \frac{k_{\text{mod}} \cdot k_c \cdot \sigma_{\text{limit,k}}}{\gamma_M}$$

where  $k_{mod}$ : is the modification factor

 $k_c$ : is the construction factor

 $\sigma_{\text{limit}\,\text{k}}$  : is the characteristic limit stress value set in the 1.2 Layers window

 $\gamma_M$ : is the partial factor for other glass

andard - DIN 18008:2010-12				
General				
Partial Factors for Material Properties Design situation: - Persistent and Transient		Serviceability Limits (Deflection Type of combination:		Cantilevers
- Persistent and Transent - For thermally toughened glass - For other glass - Accidental	γм: 1.50 + F] γм: 1.80 + F] γм: 1.00 + F]	- Characteristic - Frequent - Quasi-permanent	L/ 100 (m/h) L/ 100 (m/h) L/ 100 (m/h)	Lc/ 50 + h Lc/ 50 + h Lc/ 50 + h
Construction Factor For thermally toughened glass For other glass	k₀: 1.00 ★ k [·] k₀: 1.00 ★ k [·]	Insulating Glass Unit LC Factor for climatic load: - Temperature:	Summer 1.00 m [-]	Winter
Modification Factor Load duration class (LDC): - Permanent - Middle - Short	kmod:     0.25 ***     [-]       kmod:     0.40 **     [-]       kmod:     0.70 **     [-]	- Atmospheric Pressure: - Atitude:	1.00 + F.	1.00 mm [-]
			(	OK Cancel

Figure 4.7: Dialog box Standard – DIN 18008:2010-12

Individual sections of this dialog are described in the following sections.

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

In this section, you can check the partial factors of the material properties  $\gamma_M$  for the possible design situations. The design situations are to be assigned to individual load cases and combinations in the *Ultimate Limit State* tab of the 1.1 *General Data* window (see Chapter 3.1.1, page 21).

#### **Construction Factor**

In this section, you can define the factor  $k_c$  manually.

#### **Modification Factor**

The values of the modification factor  $k_{mod}$  are shown in this section for every load duration class. The values follow the standard DIN 18008-2:2010-12, Table 6. The modification factor  $k_{mod}$  is assigned to the load cases according to the corresponding load duration class in the 1.7 *Load Duration* window, (see Chapter 3.7, page 37).



### Serviceability Limits (Deflections)

The limit values of allowable deflections can be set in the six available text boxes. In this way, you can enter specific data for various action combinations (*Characteristic, Frequent,* and *Quasipermanent*) and for surfaces supported on both sides or one side only.

The classification of the load cases is to be done in the *Serviceability Limit State* tab of the 1.1 *General Data* window (see Chapter 3.1.2, page 22). In the 1.8 *Serviceability Data* window, you enter the reference lengths *L* for individual surfaces (see Chapter 3.8, page 38).

### **Insulating Glass Unit**

In this section, you can set factors, by which the climatic load is to be multiplied. The climatic load is then considered with this factor for each action that is selected in the 1.1 *General Data* window for design. Climatic loads are set in the 1.6 *Climatic Load Parameters* window.

The buttons in the bottom left corner of the Standard dialog box allow you to save modified

**)** (16) (16)

X

values as the default setting. Furthermore, you can use the buttons to import saved parameters or to restore the default settings of the program.

A user-defined standard can be deleted by using the [Delete] button.

### 4.3.2 Standard – TRLV:2006-08

For the standard *TRLV:2006-08* the design value (limit stress) is the same value as the characteristic limit stress set in the 1.2 *Layers* window.

 $\sigma_{\text{limit,d}} = \sigma_{\text{limit,k}}$ 

ieneral			
Serviceability Limits (Defle	ections)		
Type of combination: - Characteristic	L/ 100	Cantilevers Le / 50 +	
- Frequent	L/ 100	Lc/ 50	
- Quasi-permanent	L/ 100 🖈	Lc/ 50 -	
) 🔤 🕥 📭			OK Cano

Figure 4.8: Dialog box Standard – TRLV:2006-08



### Serviceability Limits (Deflections)

The limit values of allowable deflections can be set in the six available text boxes. In this way, you can enter specific data for various action combinations (*Characteristic, Frequent,* and *Quasipermanent*) and for surfaces supported on both sides or one side only.

The classification of the load cases is to be done in the *Serviceability Limit State* tab of the 1.1 *General Data* window (see Chapter 3.1.2, page 22). In the 1.8 *Serviceability Data* window, you enter the reference lengths *L* for individual surfaces (see Chapter 3.8, page 38).

### 4.3.3 Standard – None

By selecting *None* for the standard, the design stress value ( $\sigma_{\text{limit,d}}$ ) is calculated from the characteristic limit stress value ( $\sigma_{\text{limit,k}}$ ), according to the following relation:

 $\sigma_{\text{limit,d}} = \frac{\sigma_{\text{limit,k}}}{\gamma_{M}}$ 

where  $\sigma_{\text{limit,k}}$  : is the characteristic limit stress value set in the 1.2 Layers window

 $\gamma_M$ : is the partial factor

If the partial factor is not used, then the module assumes  $\gamma_M = 1$ .

ndard - None				
General Partial Factors for Material Properties		Serviceability Limits (Defi	ections)	
Activate		Type of combination:		Cantilevers
Design situation:		- Characteristic	L/ 100 🜩 🕨	Lc/ 50+
Persistent and Transient	ум:	- Frequent	L/ 100 🕀	Lc/ 50
Accidental	ум: ÷		L/ 100 🔶	Lc/ 50 🕀
ACCIDENTIAL	TM -	- wuasepermanent		50
) 🔤 🐧 🖪 🗳 🗙				OK Cancel

Figure 4.9: Dialog box Standard – None

Individual sections of this dialog are described in the following sections.

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

In this section, you can activate and then rewrite the partial factors of the material properties  $\gamma_M$  for the possible design situations. The design situations are to be assigned to individual load cases and combinations in the *Ultimate Limit State* tab of the 1.1 *General Data* window (see Chapter 3.1.1, page 21).



### Serviceability Limits (Deflections)

The limit values of allowable deflections can be set in the six available text boxes. In this way, you can enter specific data for various action combinations (*Characteristic, Frequent,* and *Quasipermanent*) and for surfaces supported on both sides or one side only.

The classification of the load cases is to be done in the *Serviceability Limit State* tab of the 1.1 *General Data* window (see Chapter 3.1.2, page 22). In the 1.8 *Serviceability Data* window, you enter the reference lengths L for individual surfaces (see Chapter 3.8, page 38).



### 4.4 Start calculation

Calculation

You can start the [Calculation] in all input windows of the module by using the button with the same name.

You can also start the calculation from the RFEM user interface. The *To Calculate* dialog box (menu *Calculate*  $\rightarrow$  *To Calculate*) lists the design cases of the add-on modules like load cases or load combinations.

t Calculate	d		:	Selected for C	Calculation	
o. ^	Description	*	[	No.	Description	
LC1 LC3	Self-Weigth			CA1	RF-GLASS - Design of glass surfaces	
LC2						
			>			
			>>			
		E				
			~			
All					1	

Figure 4.10: Dialog box To Calculate in RFEM

If the RF-GLASS design case is missing in the *Not Calculated* list, you have to select *All* or *Add-on Modules* in the drop-down list located in bottom part of the dialog box.



۹

By using the  $[\blacktriangleright]$  button, you transfer the selected RF-GLASS case to the list on the right. To start the calculation, click [OK].

You can also start the RF-GLASS calculation directly from the toolbar. To do this, set the RF-GLASS case in the list, and then click [Show Results].

RF-GLASS - Design of glass surface	s		٥	>	P	2 3	- -	X.XX		60	<b>a</b> 1	864 1
RF-GLASS - Design of glass surface	Ŧ	 5	e	f (	× 1	đ	P	X	Ŷ	Īz	-X	<b>;</b> -

Figure 4.11: Direct calculation of the RF-GLASS case in RFEM



# 5. Results

Details...

OK

nediately after the calculation, the 2.1 *Max Stress/Ratio* 

Immediately after the calculation, the 2.1 *Max Stress/Ratio by Loading* window appears (see Figure 5.1). To select other result windows, you can click the corresponding item in the navigator. To browse through the module windows, you can use the buttons [<] and [>], or press the function keys [F2] and [F3].

In the *Results* tab of the *Details* dialog box, you can specify which result windows you want to display (see Chapter 4.1.2 on page 45).

To save the results, click [OK]. Thus, you exit RF-GLASS and return to the main program.

Below the tables, you find a number of buttons that are useful for the evaluation of the results:

Button	Name	Function
۲	View mode	Allows you to jump to the RFEM work window to change the view
1	Selection	Allows for the graphical selection of a surface or a point to display these results in the table
9	Result diagrams	Displays results from the current row in the RFEM back- ground graphic
7,1	Exceeding	Displays only the rows with the design ratio >1 in tables, that is, the design is not satisfied
	Relation scale	Displays or hides the color bars in the results windows
	Excel export	Opens the <i>Export table</i> dialog box → Chapter 7.2, page 69

Table 5.1: Buttons in results windows

Chapter 5 Results presents the results windows in their order.





### 5.1 Max Stress/Ratio by Loading

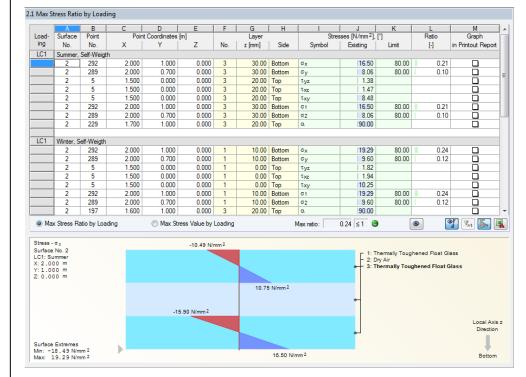


Figure 5.1: Window 2.1 Max Stress/Ratio by Loading

The window lists the maximum stress values or ratios for each load case and each load and result combination selected in the *Ultimate Limit State* tab of the 1.1 *General Data* window. The numbers of load cases are shown in the heading of each section.

#### Surface No.

This column shows the numbers of the surfaces with the governing points.

#### **Point No.**

In this FE or grid points, the maximum ratio was determined. The stress type is shown in column I *Symbol*.

The FE mesh points are created automatically. The grid points, on the other hand, can be controlled in RFEM, because user-defined result grids are possible for surfaces. The function is described in Chapter 8.12 of the RFEM manual. In the *Results* tab of the *Details* dialog box, you can specify whether you want to evaluate the results in FE mesh points or grid points (see Chapter 4.1.2, page 45). If you decide to change the settings, the results are deleted.

#### **Point Coordinates**

The three columns show the coordinates of the respective governing FE or grid points.

#### Layer

Columns F, G, and H list the numbers, z –coordinates, and sides of the layers, where maximum stress values occur, respectively.

Details.



#### Stresses

#### Symbol / Existing

These two columns show the stresses selected in the *Stresses* tab of the *Details* dialog box (see Chapter 4.1.1). They show the respective stress type with the maximum value.

#### Limit

The limit values (limit stresses)  $\sigma_{\text{limit,d}}$  are based on the materials specified in the 1.2 *Layers* window and in the selected standard. The calculation of limit value is described in Chapter 4.3, page 49.

#### Ratio

Max ratio: 0.24 ≤ 1 🥹

For the tension stress components  $\sigma_x$ ,  $\sigma_y$ ,  $\sigma_1$  and  $\sigma_2$ , the ratio of the design is determined with regard to the limit stress. If the limit stress is not exceeded, the ratio is less than or equal to 1, and the stress analysis is satisfied. Thus, the values in column L allow you to quickly evaluate the design economy.

The ratio is calculated only for positive (tension) stress values  $\sigma_x$ ,  $\sigma_y$ ,  $\sigma_1$  a  $\sigma_2$ , because the tension stiffness  $\sigma_{\text{limit,d}}$  is governing for glass.

The following table describes the calculation of the ratios.

Stresses [Pa]	Ratio [-]
$\sigma_{x}$	$=\frac{\sigma_x}{\sigma_{\text{limit,d}}}$
$\sigma_y$	$=\frac{\sigma_y}{\sigma_{\text{limit,d}}}$
$\sigma_1$	$=\frac{\sigma_1}{\sigma_{\text{limit,d}}}$
$\sigma_2$	$=\frac{\sigma_2}{\sigma_{\text{limit,d}}}$

Table 5.2: Ratio

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

In the lower part of the window, the stress distribution in the layers is shown graphically for the current point (that is, for the table row, in which the cursor is placed).

By selecting the check boxes in this column, you can include stress graphs in the results chapter of the printout report (see Chapter 6.1, page 65).



2214-0



## 5.2 Max Stress/Ratio by Surface

	A	B	C	D	E	F	G	H		J	K	L	M
Surface	Point	Point	Coordinates	[m]	Load-		Layer		Stresse	es [N/mm²],	[']	Ratio	Graph
No.	No.	X	Y	Z	ing	No.	z [mm]	Side	Symbol	Existing	Limit	•	in Printout Repo
1	88	0.500	0.700	0.000	LC1	1	0.00	Тор	σx	-4.43			
	88	0.500	0.700	0.000	LC1	1	0.00	Тор	σγ	-2.70			
	96	0.500	1.500	0.000	LC1	3	10.38	Тор	τ <sub>yz</sub>	-0.29			
	168	1.000	0.700	0.000	LC1	3	10.38		τ <sub>xz</sub>	-0.35			
	2	0.000	1.500	0.000	LC1	1	0.00	Тор	τ <sub>xy</sub>	-2.37			
	88	0.500	0.700	0.000	LC1	3		Bottom	G1	4.43	80.00	0.06	
	88	0.500	0.700	0.000	LC1	1	0.00		σ2	-4.43			
	89	0.500	0.800	0.000	LC1	3	10.38	Тор	α	-89.97			
2	292	2.000	1.000	0.000	LC1: Winter	1	10.00	Bottom	σx	19.29	80.00	0.24	
	289	2.000	0.700	0.000	LC1: Winter	1		Bottom	σγ	9.60	80.00	0.12	ū
	5	1.500	0.000	0.000	LC1: Winter	1	0.00		τ <sub>yz</sub>	1.82			ū
	5	1.500	0.000	0.000	LC1; Winter	1	0.00		τ <sub>xz</sub>	1.94			ā
	5	1.500	0.000	0.000	LC1; Winter	1	0.00		τ <sub>XV</sub>	10.25			
	292	2.000	1.000	0.000	LC1: Winter	1		Bottom	<b>0</b> 1	19.29	80.00	0.24	Ē
	292	2.000	1.000	0.000	LC1; Winter	1	0.00	Тор	σ <u>2</u>	-18.49		-	
	229	1.700	1.000	0.000	LC1; Summer	3	20.00	Тор	α	90.00			
⊚ Max	Stress Rat	io by Surface		Max Stress	Value by Surface			Max ratio:	0.24 ≤ 1	۲	۲	<b>i</b>	N 🛃
Stress -	σx			-4 43 N/	mm 2								
Surface LC1 X: 0.50 Y: 0.70 Z: 0.00	0 m 0 m	,								2: PVB 22	°C loading ur	d Float Glas ntil 3 min 1 Float Glass	s Local Axis Direction

Figure 5.2: Window 2.2 Max Stress/Ratio by Surface

This results window lists the maximum stress values or ratios for every designed surface. The columns of this table are described in Chapter 5.1.





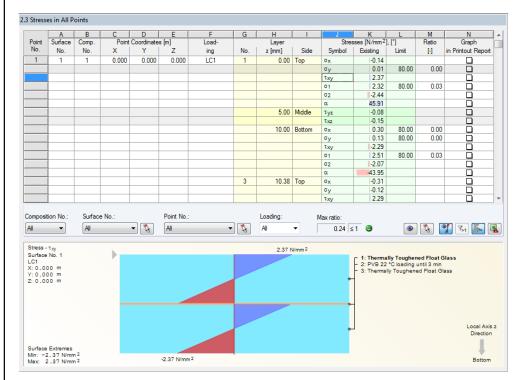


Figure 5.3: Window 2.3 Stresses in All Points

This table shows the stresses and stress ratios for each FE mesh or grid point of the designed surfaces. In the Results tab of the Details dialog box (see Chapter 4.1.2, page 45) you can set whether you want to display the results in FE mesh points or grid points.

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

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

#### **Filtering result columns**

For a clearer overview, you can filter the table by composition, surface and point number as well as by loading. The drop-down lists allows you to make a selection by object number. You can also specify the points and surfaces graphically in the RFEM work window after clicking [<sup>5</sup>].









	A	В	C	D	E	F	G	H (		J	K
Line	Surface	Load-	Packet/	Support	Location	Suppo	rt Forces [k]	l/m]	Support I	Moments [k]	Vm/m]
No.	No.	ing	Layer No.	Location	x [m]	рх	PY	pz	mx	my	mz
1	1	LC1	1	Middle	0.000	0.20	0.20	-0.04	0.00	0.00	0.0
					0.100	7.97	2.87	0.38	0.00	0.00	-0.0
					0.200	10.35	3.14	0.38	0.00	0.00	-0.0
					0.300	13.05	3.17	0.39	0.00	0.00	-0.0
					0.400	15.19	2.79	0.40	0.00	0.00	-0.0
					0.500	16.81	2.14	0.40	0.00	0.00	-0.0
					0.600	17.92	1.34	0.40	0.00	0.00	0.0
					0.700	18.47	0.46	0.40	0.00	0.00	0.0
					0.800	18.47	-0.46	0.40	0.00	0.00	0.0
					0.900	17.92	-1.34	0.40	0.00	0.00	0.0
					1.000	16.81	-2.14	0.40	0.00	0.00	0.0
					1.100	15.19	-2.79	0.40	0.00	0.00	0.0
					1.200	13.05	-3.17	0.39	0.00	0.00	0.0
					1.300	10.35	-3.14	0.38	0.00	0.00	0.0
					1.400	7.97	-2.87	0.38	0.00	0.00	0.0
					1.500	2.24	-2.22	-0.44	0.00	0.00	0.0
1	1	LC1	3	Middle	0.000	-0.20	-0.20	-0.08	0.00	0.00	0.0
					0.100	-7.98	-2.88	0.52	0.00	0.00	0.0
					0.200	-10.36	-3.15	0.93	0.00	0.00	0.0
					0.300	-13.07	-3.18	1.27	0.00	0.00	0.0
					0.400	-15.20	-2.80	1.46	0.00	0.00	0.0
					0.500	-16.83	-2.15	1.60	0.00	0.00	0.0
					0.600	-17.93	-1.34	1.68	0.00	0.00	0.0
					0.700	-18.49	-0.46	1.72	0.00	0.00	0.0
					0.800	-18.49	0.46	1.72	0.00	0.00	0.0
					0.900	-17.93	1.34	1.68	0.00	0.00	0.0
					1.000	-16.83	2.15	1.60	0.00	0.00	-0.0
					1.100	-15.20	2.80	1.46	0.00	0.00	-0.0
					1.200	-13.07	3.18	1.27	0.00	0.00	-0.0
Related		Cocal axis sy Layer No		Global - Support Location:	axis system X, Y, Z Loading:						
Julie INC		Layer No		Support Location:	Loading:						

Figure 5.4: Window 2.4 Line Support Reactions

This results window is shown only if you select *Local* calculation type in Window 1.1. The window shows the support forces and support moments for each line support defined in the 1.3 *Line Supports* window. The table is sorted by line numbers.

#### Surface No.

This column shows the numbers of the surfaces with the governing lines.

#### Loading

In the column loading, you can see the load case numbers or numbers of load and result combinations.

#### Packet/Layer No.

This column snows the numbers of packets or glass type layers on which the line supports are defined.

#### **Support Location**

The column shows the exact support location.

#### Location x

The x-locations shown in the table column represent the spacing of FE nodes along the line. The surface grid is not relevant for line support reactions.

#### Support forces p<sub>x</sub>/p<sub>y</sub>/p<sub>z</sub>

The support forces are listed in three table columns. The forces can be related to the global axes X, Y and Z or the local axes x, y and z of the line supports. The table shows the forces which are introduced into the support.



#### Support moments m<sub>x</sub>/m<sub>y</sub>/m<sub>z</sub>

The support moments are listed in three table columns. They are related to the global axis system XYZ or the local axis system of the line support xyz. The table shows the moments which are introduced into the support.

#### **Filtering result columns**

For a clearer overview, you can filter the table by line and layer number, support location as well as by loading. The drop-down lists allow you to make a selection by object number. You can also specify the line graphically in the RFEM work window after clicking [<sup>\*</sup>].

## 5.5 Nodal Support Reactions

	A	В	C	D	E	F	G	H		J
Node	Surface	Load-	Packet/	Support	Sup	oport Forces [	kN]	Suppo	ort Moments [	kNm]
No.	No.	ing	Layer No.	Location	Px	Py	Pz	Mx	MY	Mz
1	1	LC1	1	Middle	0.20	0.20	-0.04	0.00	0.00	0.00
1	1	LC1	3	Middle	-0.20	-0.20	-0.08	0.00	0.00	0.00
1	1	LC2	1	Middle	0.54	0.54	-0.11	0.00	0.00	0.00
1	1	LC2	3	Middle	-0.54	-0.54	-0.22	0.00	0.00	0.00
5	2	LC1; Summer	1	Тор	1.15	1.11	-0.05	0.00	0.00	0.00
5	2	LC1; Summer	3	Тор	1.78	1.68	-0.07	0.00	0.00	0.00
5	2	LC1; Summer	3	Middle	-0.31	-0.09	0.19	0.00	0.00	0.00
5	2	LC1; Summer	3	Bottom	-1.31	-1.45	-0.03	0.00	0.00	0.00
5	2	LC1; Winter	1	Тор	2.23	2.09	-0.09	0.00	0.00	0.00
5	2	LC1; Winter	3	Тор	0.77	0.75	-0.03	0.00	0.00	0.00
5	2	LC1; Winter	3	Middle	-0.06	-0.01	0.09	0.00	0.00	0.00
5	2	LC1; Winter	3	Bottom	-0.67	-0.71	-0.02	0.00	0.00	0.00
5	2	LC2; Summer	1	Тор	3.78	3.43	-0.15	0.00	0.00	0.00
5	2	LC2; Summer	3	Тор	4.11	3.69	-0.16	0.00	0.00	0.00
5	2	LC2; Summer	3	Middle	-1.18	-0.36	0.36	-0.01	0.01	0.00
5	2	LC2; Summer	3	Bottom	-2.30	-2.77	0.01	0.00	0.00	0.00
5	2	LC2; Winter	1	Тор	5.17	4.62	-0.20	-0.01	0.01	0.00
5	2	LC2; Winter	3	Тор	2.78	2.55	-0.11	0.00	0.00	0.00
5	2	LC2; Winter	3	Middle	-0.62	-0.16	0.27	0.00	0.00	0.00
5	2	LC2; Winter	3	Bottom	-1.83	-2.09	-0.01	0.00	0.00	0.00
6	2	LC1; Summer	1	Тор	1.15	-1.11	-0.05	0.00	0.00	0.00
6	2	LC1; Summer	3	Тор	1.78	-1.68	-0.07	0.00	0.00	0.00
6	2	LC1; Summer	3	Middle	-0.31	0.09	0.19	0.00	0.00	0.00
6	2	LC1; Summer	3	Bottom	-1.31	1.44	-0.03	0.00	0.00	0.00
6	2	LC1; Winter	1	Тор	2.23	-2.09	-0.09	0.00	0.00	0.00
6	2	LC1; Winter	3	Тор	0.77	-0.75	-0.03	0.00	0.00	0.00
6	2	LC1; Winter	3	Middle	-0.06	0.01	0.09	0.00	0.00	0.00
6	2	LC1; Winter	3	Bottom	-0.67	0.71	-0.02	0.00	0.00	0.00
6	2	LC2; Summer	1	Тор	3.78	-3.43	-0.15	0.00	0.00	0.00
6	2	LC2; Summer	3	Тор	4.11	-3.69	-0.16	0.00	0.00	0.00
Related Node N		Cocal axis system Layer No.:	em X', Y', Z'	Global axis Support Location:	system X, Y, Z Loading:					

Figure 5.5: Window 2.5 Nodal Support Reactions

This results window is shown only if you select the *Local* calculation type in Window 1.1. The window shows the support forces and support moments for each nodal support defined in the 1.4 *Nodal Supports* window. The table is sorted by node numbers.

The columns of this table are described in Chapter 5.4.



•

All

1 2 3



	A	B	C	D	E	F	G	H (	1 [	J	
Surface	Point		t Coordinates		Load-	Type of	Packet	Displaceme	ents [mm]	Ratio	
No.	No.	X	Y	Z	ing	Comb.	No.	Uz	Limit uz	uz [·]	
1	89	0.500	0.800	0.000	LC1	СН	1	0.864	15.000	0.06	
	89	0.500	0.800	0.000	LC2	CH	1	2.301	15.000	0.15	
2	292	2.000	1.000	0.000	LC1; Summer	CH	1	2.754	10.000	0.28	
	292	2.000	1.000	0.000			2	4.213	10.000	0.42	
	292	2.000	1.000	0.000	LC1; Winter	CH	1	4.919	10.000	0.49	
	292	2.000	1.000	0.000			2	1.991	10.000	0.20	
	292	2.000	1.000	0.000	LC2; Summer	CH	1	7.486	10.000	0.75	
	292	2.000	1.000	0.000			2	8.162	10.000	0.82	
	292	2.000	1.000	0.000	LC2; Winter	CH	1	9.430	10.000	0.94	
	292	2.000	1.000	0.000			2	6.052	10.000	0.61	
	Maximum	Displacement	/ Maximum Di	isplacement F	atio						
2	292	2.000	1.000	0.000	LC2; Winter	CH	1	9.430	10.000	0.94	
	292	2.000	1.000	0.000	LC2; Winter	CH	1	9.430	10.000	0.94	
									4 ≤1 ❷	•	) 🌒 🏹 🖺 🛙

### 5.6 Max Displacements

Figure 5.6: Window 3.1 Max Displacements

The deformation analyses are carried out only if you selected at least one load case for design in the *Serviceability Limit State* tab of the 1.1 *General Data* dialog box (see Chapter 3.1.2, page 22).

Window 3.1 shows the maximum displacements from load cases or load combinations for the serviceability limit state and compares them with the allowable deformations. The table is sorted by surface numbers.

The columns A to D are described in Chapter 5.1 on page 56.

#### Type of Comb.

The column shows the design combinations assigned in the *Serviceability Limit State* tab of the 1.1 *General Data* window: Characteristic (*CH*), Frequent (*FR*), or Quasi-permanent (*QP*).

#### Displacements

In the column  $u_z$ , you can see the displacements in the direction of the local surface axis z, which are governing for the deformation analysis. You can display the axes of the surfaces by using the *Display* navigator of RFEM or by using the context menu of the surfaces.

The values in column I show the limit deformations in the direction of the *z*-axis of each surface. These values are determined from the reference lengths *L*, which are specified in the 1.8 *Serviceability Data* window (see Chapter 3.8, page 38), and from the limit values defined for the serviceability design in the *Standard* dialog box (see Chapter 4.3).

#### Ratio

Max ratio: 0.94 ≤ 1 🥹

The last column shows the quotients from the resulting displacement  $u_z$  (column H) and the limit displacement (column I). If the limit deformations are not exceeded, then the ratio is less than or equal to 1 and the deformation design is satisfied.



## 5.7 Gas Pressure

	A	B	С	
ad-	Surface	Laver	Gas Pressure	
g	No.	No.	p [kN/m <sup>2</sup> ]	
.C1	Summer,			
	2	2	103.49	
	1			
LC1	Winter, S	elf-Weigth		
	2	2	104.01	
LC2	Summer			
	2	2	106.80	
LC2	Winter			
	2	2	107.08	

Figure 5.7: Window 3.2 Gas Pressure

This results window is shown only if you defined at least one gas layer in the 1.2 *Layers* window. The *Gas Pressure* results for the gas layers are listed by load case.

## 5.8 Parts List

	А	B	С	D	E	F	G		
uface	Material	Thickness	No. of	Area	Coating	Volume	Weight		
No.	Description	t [mm]	Layers	[m <sup>2</sup> ]	[m <sup>2</sup> ]	[m <sup>3</sup> ]	[t]		
1	Thermally Toughened Float Glass	10.00	2	1.500	3.000	0.030	0.075		
	PVB 22 °C loading until 3 min	0.38	1	1.500	0.000	0.001	0.001		
Σ		20.38	3	1.500	3.000	0.031	0.076		
2	Thermally Toughened Float Glass	10.00	2	2.000	4.000	0.040	0.100		
	Dry Air	10.00	1	2.000	0.000	0.020	0.000		
Σ		30.00	3	2.000	4.000	0.060	0.100		
Total				3.500	7.000	0.091	0.176		
							۲	1	F

Program RF-GLASS© 2013 Dlubal Software GmbH



The last results window shows an overall review of the designed surfaces. By default, the data in this list refers only to designed surfaces. If you wish to display the parts list for all surfaces in the model, you can set this in the *Results* tab of the *Details* dialog box (Chapter 4.1.2, page 45).

#### Surface No.

This column contains the numbers of individual surfaces.

#### **Material Description**

The data is listed by materials.

#### Thickness t

Column B shows the thickness of the layers. You can find this input data in the 1.2 *Layers* window.

#### No. of Layers

This column shows the number of layers with the same material and the same thickness.

#### Area

The column provides information on the surface areas of the individual layers.

#### Coating

The total surface coating is determined from the top and bottom sides of a surface. The side surfaces of the thin-walled surfaces are neglected.

#### Volume

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

#### Weight

The last column displays the weight of every layer. This entry is determined as product of the volume and specific weight of the used material.

#### ΣTotal

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-GLASS, to which you can add graphics and explanations. In the global selection, you can select input and output data of RF-GLASS that you want to include in the printout report.

5

The printout report is described in detail in the RFEM manual. Chapter 10.1.3.4 *Selecting Data of Add-on Modules* describes how to prepare the input and output data of add-on modules for the printout.

The printout report shows only the stress types that appear in the RF-GLASS output windows. Therefore, if you want to print for example the maximum shear stresses, you must activate the display of the stresses  $\tau_{max}$  in RF-GLASS. The selection of stresses is described in Chapter 4.1.1, page 40.

For complicated structural systems with a great number of design cases, it is recommended to divide the data into several printout reports to get a better overview.

## 6.2 Printing RF-GLASS Graphics

### 6.2.1 Results on the RFEM Model

In RFEM, you can transfer every image shown in the work window to the printout report or send it directly to a printer. In the same way, you can also prepare the stresses, ratios, and sections shown on the RFEM model for the printout report.



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

You can print the current graphic of the RF-GLASS results in the RFEM work window by using the command from the main menu

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

or clicking the corresponding button in the toolbar.



Figure 6.1: Button Print Graphic in the RFEM toolbar

The following dialog box opens.



General Options Color Scale Factors	Margins and Stretch Factors	
Graphic Picture	Window To Print	Graphic Size
O Directly to a printer	<ul> <li>Current only</li> </ul>	As screen view
To a printout report:     PR1	More	🔍 🔘 Window filling
To the Clipboard	Mass print	○ To scale 1: 100 ▼
Graphic Picture Size and Rotation	Options	
Use whole page width	Show results for selected	ed x-location in result
🗇 Use whole page height	Lock graphic picture (w	vithout update)
Height: 51 → [% of page]		
	Show printout report on	[OK]
Rotation: 0 🚔 [°]		
Header of Graphic Picture		
RF-GLASS - Displacements for LC1 u-z, C	A1. Isometric	

Figure 6.2: Dialog box Graphic Printout, tab General

This dialog box is described in Chapter 10.2 of the RFEM manual. The other tabs *Options* and *Color Scale* are described there as well.

You can move a graphic to another place in the printout report by using the drag-and-drop operation as usual.

To adjust an inserted graphic in the printout report subsequently, right-click the appropriate item in the navigator of the report. By using the *Properties* option in the context menu, you reopen the *Graphic Printout*, where you can modify the settings.

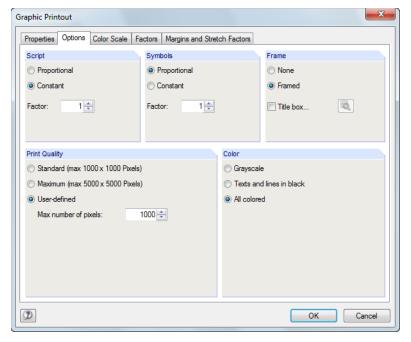


Figure 6.3: Dialog box Graphic Printout, tab Options





### 6.2.2 Results in Layers

Results Windows 2.1 through 2.3 show the stress distribution in the layers. The stress graphics can be added to the printout report by selecting the *Graph in Printout Report* check boxes in column M.

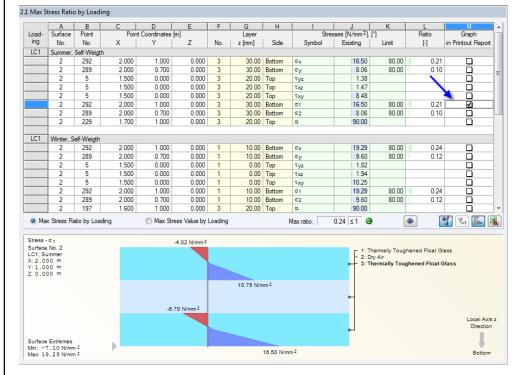
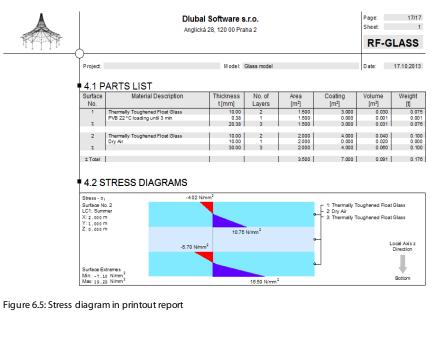


Figure 6.4: Specifying Diagram in Printout Report

The graphics then appear in the 4.2 Stress Diagrams sections of the printout report.





# 7. General Functions

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

## 7.1 Units and Decimal Places

The units and decimal places are managed in one dialog box for RFEM and the add-on modules. In RF-GLASS, you open the dialog box for adjusting the units by using the command from the menu

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

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

Program / Module     Input Data     Results       RF-CONCRETE Surfac      Coordinates       RF-CONCRETE Columi     Unit     Dec. places       RF-PUNCH     Unit     Dec. places       RF-TIMBER AWC     mm     3im       RF-TIMBER R     Thicknesses:     mm       RF-TIMBER     Specific weights:     kl/m^3	Dec. places 3 $\sqrt[]{2}$ 2 $\sqrt[]{2}$ 3 $\sqrt[]{2}$ 1 $\sqrt[]{2}$								
-RF-CONCRETE Memby     Coordinates     Layers       -RF-CONCRETE Column     -RF-PUNCH     Lengths:     Imit     Dec. places     Unit     D       -RF-TIMBER Pro     -RF-TIMBER AWC     Thicknesses:     mm     2 Imit     Specific weights:     kN/m^3 ▼       -RF-TIMBER     Surface weights:     kN/m^2 ▼	3 2 2 3 7 1								
Unit     Dec. places     Unit     D       RF-FUNCH     Lengths:     m     3   →       RF-TIMBER AWC     Thicknesses:     mm     2   →       RF-TIMBER     Specific weights:     kN/m^3 ▼	3 2 2 3 7 1								
RF-PUNCH     Lengths:     m     3 ↓     E-Modules, pressures:     MPa       RF-TIMBER AWC     Thicknesses:     mm     2 ↓     Specific weights:     kN/m^3 ▼       RF-TIMBER     Surface weights:     kN/m^2 ▼	3 2 2 3 7 1								
RF-TIMBER Pro     RF-TIMBER AWC     Thicknesses:     mm     Z     Specific weights:     kN/m^3     Surface weights:     kN/m^2									
RF-TIMBER AWC     Thicknesses: mm      Z     Specific weights: kN/m^3      Surface weights: kN/m^2	3 🜩								
- RF-TIMBER Surface weights: kN/m <sup>*</sup> 2	3 🜩								
Surace weights: KIV/m 2 V	1								
RFJOINTS     Supports     Thermal expansion coef.: 1/K	Local I								
RF-END-PLATE Forces: kN V 2 Thermal conductivity: W/m/K V	2 🌩								
m RF-FRAME-JOINT Pro Lengths: m 3 Poisson's ratios:	3 ≑								
RF-DSTV Lengths for moment: m V 3									
RF-DOWEL Climatic Load Properties									
Angles: 2 Atmosph. pressures: N/mm^2	3 🚔 🖣								
	1 🚔 🖣								
- RF-DEFORM = Factors: Z	1 🚔 🖣								
···· RF-IMP Stiffness Matrix Elements									
- RF-SOILIN Bending, torsion: kNm									
Bending, torsion:	1 🚔								
RF-LAMINATE Shear, membrane: kN/m -	1 🚔								
RF-TOWER Structure	1								
RF-TOWER Equipment									
RF-TOWER Loading	1 🚔								
- RF-TOWER Effective L									
H RF-10WER Design									
ОК (	Cancel								

Figure 7.1: Dialog box Units and Decimal Places

In the figure above, some units are marked by a red triangle (section *ClimaticLoad Properties*). The dialog box was opened from the 1.6 *ClimaticLoad Parameters* window. These marks allow you to find the relevant units of this window more easily.



You can save the settings as a user profile and reuse them in other models. This functions are described in Chapter 11.1.3 of the RFEM manual.



## 7.2 Export of Results

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

#### Clipboard

You can copy the marked rows of a results window to the Clipboard by using the buttons [Ctrl]+[C], and then transfer them, for example, to a word processor by using [Ctrl]+[V]. The headings of table columns are not exported.

#### **Printout Report**

RF-GLASS data can be printed in the printout report (Chapter 6.1, page 65), and then exported by using the command from the menu

 $\textbf{File} \rightarrow \textbf{Export to RTF}.$ 

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

#### Excel / OpenOffice

RF-GLASS allows for a direct export of data to MS Excel, OpenOffice.org Calc, or the CSV format. You call up this function from the RF-GLASS menu

#### $\textbf{File} \rightarrow \textbf{Export Tables}.$

The following dialog box for the data export opens:

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

Figure 7.2: Dialog box Export – MS Excel

Having selected the required parameters, you can start the export by clicking [OK]. Excel or OpenOffice are started automatically, that is, the programs do not need to run in the background.

X	<b>9</b> •	(Ci + )	-			List1 - Micr	oson e	xcei				
Fi	le	Home	Insert	Page Layo	out Form	iulas Dat	ta f	Review \	/iew A	dd-Ins	^ (?) ⊂	3 🖞 🗆
Pas Clipl	l 🔓.	Calibr B		A A	≡ ≡ <mark>≡</mark> ≣ ≣ ⊒ ⊈ ≇	······································	√ % .00 mber	* A Styles	Pelet Pelet Form Cells	e • 💽 •	Sort & Find & Filter Y Select Y Editing	
A1 • fx Load-												
	А	В	С	D	E	F	G	Н	I.	J	K	L
1	Load-	Surface	Point	Poir	nt Coordinates	[m]		Layer		St	resses [N/mm²], [°	
2	ing	No.	No.	x	Y	z	No.	z [mm]	Side	Symbol	Existing	Lim <sup>=</sup>
3	LC1	Summer,	Gelf-Weigt	h								
4		2	292	2,000	1,000	0,000	з	30,00	Bottom	σ,	16,50	
5		2	289	2,000	0,700	0,000	з	30,00	Bottom	σ,	8,06	6
6		2	5	1,500	0,000	0,000	з	20,00	Тор	τ <sub>va</sub>	1,38	
7		2	5	1,500	0,000	0,000	з	20,00	Тор	τ <sub>sa</sub>	1,47	
8		2	5	1,500	0,000	0,000	з	20,00	Тор	τ <sub>xy</sub>	8,48	_
9		2	292	2,000	1,000	0,000	з	30,00	Bottom	σ	16,50	1
10		2	289	2,000	0,700	0,000	з	30,00	Bottom	σ2	8,06	
11		2	229	1,700	1,000	0,000	З	20,00	Тор	α	90,00	<b>_</b> _
H + + H 2.1 Max Stress Ratio by Loading / I + H												

Figure 7.3: Results in Excel - window 2.1 Max Stress by Load Case

## 7.3 Connection to RFEM

If a surface of the type *Glass* is created in RFEM, this surface has to be defined in RF-GLASS. When the *Local* calculation type is selected in the 1.1 *General Data* window, the data defined in RF-GLASS is used only for this module. It does not influence RFEM specifications. In the main program, each surface is described by the specification of the *Material* and the *Thickness*. When the *Global* calculation type is selected in the 1.1 *General Data* window, the stiffness of the surfaces defined in RF-GLASS is transferred to RFEM.

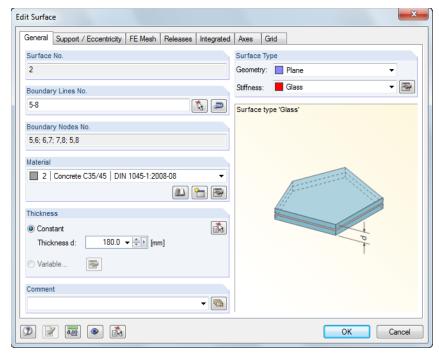


Figure 7.4: Dialog box Edit Surface

# 8. Examples

Various examples are introduced in the following chapter.

## 8.1 Determination of Stiffness Matrix Elements

Consider a plate consisting of the following layers: A thermally toughened glass pane with a thickness of 12 mm, a PVB foil with a thickness of 1.14 mm, and of a thermally toughened glass pane with a thickness of 10 mm.

Shear coupling of layers is taken into account.

Furthermore, consider the calculation according to the 2D plate theory.

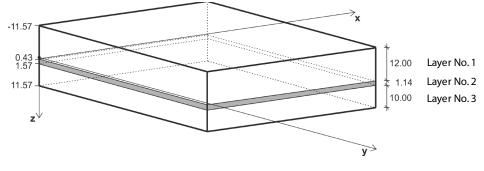


Figure 8.1: Layer composition

Layers									
	A	B	C	D	E	F	G	Н	
Layer	Layer	Material	Thickness	Modulus of Elast.	Shear Modulus	Poisson's Ratio	Specific Weight	Coeff. of Th. Exp.	Limit Stress
No.	Туре	Description	t [mm]	E [N/mm <sup>2</sup> ]	G [N/mm <sup>2</sup> ]	v [-]	γ [kN/m <sup>3</sup> ]	ατ [1/Κ]	σlimit [N/mm <sup>2</sup> ]
1	Glass	Themally Toughened Float Glass	12.00	70000.000	28455.300	0.230	25.00	9.0E-06	120.000
2	<u>F</u> oil	PVB 22 °C loading until 10 sec	1.14	12.000	4.003	0.499	10.70	8.0E-05	
3	Glass	Themally Toughened Float Glass	10.00	70000.000	28455.300	0.230	25.00	9.0E-06	120.000

Figure 8.2: Window 1.2 Layers

The stiffness matrix of the individual layers is determined as follows:

$$d_{i} = \begin{bmatrix} d_{11,i} & d_{12,i} & 0 \\ d_{22,i} & 0 \\ \text{sym.} & d_{33,i} \end{bmatrix} = \begin{bmatrix} \frac{E_{i}}{1-v_{i}^{2}} & \frac{v_{i}E_{i}}{1-v_{i}^{2}} & 0 \\ \frac{E_{i}}{1-v_{i}^{2}} & 0 \\ \text{sym.} & G_{i} \end{bmatrix}, \quad G_{i} = \frac{E_{i}}{2 \cdot (1+v_{i})} \quad i = 1, ..., n \quad (8.1)$$

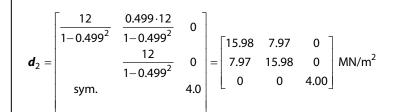
$$d_{1} = \begin{bmatrix} \frac{70000}{1-0.23^{2}} & \frac{0.23 \cdot 70000}{1-0.23^{2}} & 0 \\ \frac{70000}{1-0.23^{2}} & 0 \\ \text{sym.} & 28455.3 \end{bmatrix} = \begin{bmatrix} 73909.8 & 16999.3 & 0 \\ 16999.3 & 73909.8 & 0 \\ 0 & 0 & 28455.3 \end{bmatrix} MN/m^{2}$$

$$Matrix Elements$$

$$d_{11} = \begin{bmatrix} 73909.8 & 16999.3 & 0 \\ 0 & 0 & 28455.3 \end{bmatrix} MN/m^{2}$$

$$d_{12} = \begin{bmatrix} 16999.3 & [MN/m^{2}] \\ d_{22} & 73909.8 & [MN/m^{2}] \\ d_{22} & 73909.8 & [MN/m^{2}] \end{bmatrix}$$

Figure 8.3: Stiffness matrix of layer No. 1



#### Matrix Elements

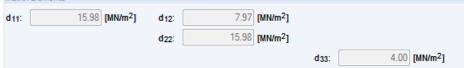


Figure 8.4: Stiffness matrix of layer No. 2

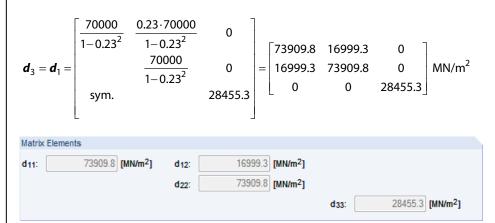


Figure 8.5: Stiffness matrix of layer No. 3

Then, the global stiffness matrix has the following shape:

	D <sub>11</sub>	D <sub>12</sub>	0	0	0	D <sub>16</sub>		
<b>D</b> =		D <sub>22</sub>	0	0	0	sym.	D <sub>27</sub>	0
			D <sub>33</sub>	0	0	sym.	sym.	D <sub>38</sub>
				D <sub>44</sub>	0	0	0	0
	-				D <sub>55</sub>	0	0	0
			sym.			D <sub>66</sub>	D <sub>67</sub>	0
							D <sub>77</sub>	0
								D <sub>88</sub>

(8.2)

The determination of the stiffness matrix elements is described on the following pages.

Stiffness matrix elements (bending and torsion) [N/m]

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

$$\begin{split} D_{11} &= \frac{(0.43 \cdot 10^{-3})^3 - (-11.57 \cdot 10^{-3})^3}{3} 73909.8 \cdot 10^3 + \frac{(1.57 \cdot 10^{-3})^3 - (0.43 \cdot 10^{-3})^3}{3} 15.98 \cdot 10^3 + \\ &+ \frac{(11.57 \cdot 10^{-3})^3 - (1.57 \cdot 10^{-3})^3}{3} 73909.8 \cdot 10^3 = 76.2 \text{ Nm} \end{split}$$

$$\begin{split} D_{12} &= \frac{(0.43 \cdot 10^{-3})^3 - (-11.57 \cdot 10^{-3})^3}{3} 16999.3 \cdot 10^3 + \frac{(1.57 \cdot 10^{-3})^3 - (0.43 \cdot 10^{-3})^3}{3} 7.97 \cdot 10^3 + \\ &+ \frac{(11.57 \cdot 10^{-3})^3 - (-11.57 \cdot 10^{-3})^3}{3} 16999.3 \cdot 10^3 = 17.5 \text{ Nm} \end{split}$$

$$\begin{split} D_{22} &= \frac{(0.43 \cdot 10^{-3})^3 - (-11.57 \cdot 10^{-3})^3}{3} 73909.8 \cdot 10^3 + \frac{(1.57 \cdot 10^{-3})^3 - (0.43 \cdot 10^{-3})^3}{3} 15.98 \cdot 10^3 + \\ &+ \frac{(11.57 \cdot 10^{-3})^3 - (-11.57 \cdot 10^{-3})^3}{3} 73909.8 \cdot 10^3 = 76.2 \text{ Nm} \end{split}$$

$$\begin{split} D_{33} &= \frac{(0.43 \cdot 10^{-3})^3 - (-11.57 \cdot 10^{-3})^3}{3} 28455.3 \cdot 10^3 + \frac{(1.57 \cdot 10^{-3})^3 - (0.43 \cdot 10^{-3})^3}{3} 4.0 \cdot 10^3 + \\ &+ \frac{(11.57 \cdot 10^{-3})^3 - (1.57 \cdot 10^{-3})^3}{3} 28455.3 \cdot 10^3 = 29.3 \text{ Nm} \end{split}$$

Dlubal

Stiffness matrix elements (eccentric effects) [N/m]

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

$$\begin{split} D_{16} &= \frac{(0.43 \cdot 10^{-3})^2 - (-11.57 \cdot 10^{-3})^2}{2} 73909.8 \cdot 10^3 + \frac{(1.57 \cdot 10^{-3})^2 - (0.43 \cdot 10^{-3})^2}{2} 15.98 \cdot 10^3 + \\ &+ \frac{(11.57 \cdot 10^{-3})^2 - (1.57 \cdot 10^{-3})^2}{2} 73909.8 \cdot 10^3 = -84.2 \text{ Nm/m} \\ D_{17} &= \frac{(0.43 \cdot 10^{-3})^2 - (-11.57 \cdot 10^{-3})^2}{2} 16999.3 \cdot 10^3 + \frac{(1.57 \cdot 10^{-3})^2 - (0.43 \cdot 10^{-3})^2}{2} 7.97 \cdot 10^3 + \\ &+ \frac{(11.57 \cdot 10^{-3})^2 - (1.57 \cdot 10^{-3})^2}{2} 16999.3 \cdot 10^3 = -19.4 \text{ Nm/m} \\ D_{27} &= \frac{(0.43 \cdot 10^{-3})^2 - (-11.57 \cdot 10^{-3})^2}{2} 73909.8 \cdot 10^3 + \frac{(1.57 \cdot 10^{-3})^2 - (0.43 \cdot 10^{-3})^2}{2} 15.98 \cdot 10^3 + \\ &+ \frac{(11.57 \cdot 10^{-3})^2 - (-11.57 \cdot 10^{-3})^2}{2} 73909.8 \cdot 10^3 = -84.2 \text{ Nm/m} \\ D_{38} &= \frac{(0.43 \cdot 10^{-3})^2 - (-11.57 \cdot 10^{-3})^2}{2} 28455.3 \cdot 10^3 + \frac{(1.57 \cdot 10^{-3})^2 - (0.43 \cdot 10^{-3})^2}{2} 4.0 \cdot 10^3 + \\ &+ \frac{(11.57 \cdot 10^{-3})^2 - (-11.57 \cdot 10^{-3})^2}{2} 28455.3 \cdot 10^3 = -32.4 \text{ Nm/m} \end{split}$$

**Dlubal** 

Stiffness matrix elements (membrane) [N/m]

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

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

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

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

$$D_{66} = 12 \cdot 10^{-3} \cdot 73909.8 \cdot 10^3 + 1.14 \cdot 10^{-3} \cdot 15.98 \cdot 10^3 + 10 \cdot 10^{-3} \cdot 73909.8 \cdot 10^3 = 16260$$

$$\begin{split} D_{66} &= 12 \cdot 10^{-3} \cdot 73909.8 \cdot 10^3 + 1.14 \cdot 10^{-3} \cdot 15.98 \cdot 10^3 + 10 \cdot 10^{-3} \cdot 73909.8 \cdot 10^3 = 1626030 \text{ N/m} \\ D_{67} &= 12 \cdot 10^{-3} \cdot 16999.3 \cdot 10^3 + 1.14 \cdot 10^{-3} \cdot 7.97 \cdot 10^3 + 10 \cdot 10^{-3} \cdot 16999.3 \cdot 10^3 = 373993 \text{ N/m} \\ D_{77} &= 12 \cdot 10^{-3} \cdot 73909.8 \cdot 10^3 + 1.14 \cdot 10^{-3} \cdot 15.98 \cdot 10^3 + 10 \cdot 10^{-3} \cdot 73909.8 \cdot 10^3 = 1626030 \text{ N/m} \\ D_{88} &= 12 \cdot 10^{-3} \cdot 28455.3 \cdot 10^3 + 1.14 \cdot 10^{-3} \cdot 4.0 \cdot 10^3 + 10 \cdot 10^{-3} \cdot 28455.3 \cdot 10^3 = 626021 \text{ N/m} \end{split}$$

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

$$D_{44/55,\text{calc}} = \frac{1}{\int_{-t/2}^{t/2} \frac{1}{G(z)} \left( \int_{-t/2}^{t/2} \frac{d_{11}(\overline{z})(\overline{z} - z_0) d\overline{z}}{\int_{-t/2}^{t/2} \frac{1}{G(z)} (\overline{z} - z_0)^2 d\overline{z}} \right)^2} dz, \quad z_0 = \frac{\int_{-t/2}^{t/2} d_{11}(\overline{z}) \overline{z} d\overline{z}}{\int_{-t/2}^{t/2} d_{11}(\overline{z}) d\overline{z}}}$$
(8.3)

 $D_{44,calc} = D_{55,calc} = 850.32 \text{ kN/m}$ 

$$D_{44} = D_{55} = \max\left(D_{44/55, \text{calc}}, \frac{48}{5l^2} \frac{1}{\frac{1}{\sum_{i=1}^{n} E_i \frac{t_i^3}{12}} - \frac{1}{\sum_{i=1}^{n} E_i \frac{z_{\text{max},i}^3 - z_{\text{min},i}^3}{3}}\right)$$
(8.4)  
$$\sum_{i=1}^{n} E_i \frac{t_i^3}{12} = 70000 \cdot 10^3 \frac{(12 \cdot 10^{-3})^3}{12} + 12 \cdot 10^3 \frac{(1.14 \cdot 10^{-3})^3}{12} + 70000 \cdot 10^3 \frac{(10 \cdot 10^{-3})^3}{12} = 15.913 \text{ kNm}$$
$$\sum_{i=1}^{n} E_i \frac{z_{\text{max},i}^3 - z_{\text{min},i}^3}{3} = 70000 \cdot 10^3 \frac{(0.43 \cdot 10^{-3})^3 - (-11.57 \cdot 10^{-3})^3}{3} + 12 \cdot 10^3 \frac{(1.57 \cdot 10^{-3})^3 - (0.43 \cdot 10^{-3})^3}{3} + 70000 \cdot 10^3 \frac{(11.57 \cdot 10^{-3})^3 - (1.57 \cdot 10^{-3})^3}{3} = 72.190 \text{ kNm}$$
$$D_{44} = D_{55} = \max\left(850.32, \frac{48}{5 \cdot 1^2} \frac{1}{\frac{1}{15.913}} - \frac{1}{72.190}\right) = \max(850.32, 195.97) = 850.32 \text{ kN/m}$$

#### **Global stiffness matrix**

	76.2	17.5 76.2	0 0	0 0	0 0	84.2 19.4	-19.4 -84.2	0 0
			29.3	0	0	0	0	-32.4
~				850.32	0	0	0	0
<b>D</b> =					850.32	0	0	0
			sym.			1626030	373993	0
							1626030	0
								626021

Surface No. Pack	et No.	Matrix Type										
1 • 🖏 1	T	Standard	-									
Stiffness Matrix Elements (Bending a	and Torsion)											
D11: 76.2 [kNm]	D12:	17.5 [kNm]										
	D22:	76.2 [kNm]			$D_{11}$	$D_{12}$						
			D33:	29.3 [kNm]		$D_{22}$	0					
								0		0		
Stiffness Matrix Elements (Shear)								$D_{44}$			0	1
D44: 850.3 [kN/m]									$D_{55}$		0 D	
	D55:	850.3 [kN/m]					sym.			$D_{66}$		
											$D_{77}$	
Stiffness Matrix Elements (Membran	e)											L
D66: 1626030.0 [kN/m]	D67: 37	73993.0 [kN/m]			D <sub>11</sub>	$1 \dots D$	33 [N	m]				
	D77: 162	26030.0 [kN/m]				<i>D</i>	INT	/1				
			D88:	626021.0 [kN/m]	$D_{44}$	<i>D</i>	88 [IN	/mj				
					$D_{10}$	$\dots D$	38 [N	m/m	]			
Stiffness Matrix Elements (Eccentric												
D16: -84.2 [kNm/m]	D17:	-19.4 [kNm/m]										
	D27:	-84.2 [kNm/m]										
			D38:	-32.4 [kNm/m]								

Figure 8.6: Dialog box Extended Stiffness Matrix Elements



# 8.2 Insulating glass

Consider an insulating glass with the dimensions 1.0 x 1.5 m, with a hinged support, and a composition according to Figure 8.7 as well as the following parameters:

Glass pane dim	ension in x-ax	is direction	<i>a</i> =	1.0 m
Glass pane dim	ension in $y$ -ax	ris direction	b =	1.5 m
Thickness of ex	ternal glass lay	er	$t_1 =$	8 mm
Thickness of air layer			$t_2 =$	12 mm
Thickness of internal glass layer			$t_3 =$	12 mm
	Modulus of e	lasticity	E =	70000 MPa
Glass parameters	Shear modul	us	G =	28455 MPa
	Poisson's rati	0	<i>v</i> =	0.23
		Temperature	$T_p =$	0°C
	Manufac- turing	Atmospheric pressure	$p_{p,\text{met}} =$	101 kPa
		Sealevel	$H_1 =$	0 m
Climatic load		Temperature (external = gas = internal)	$T_1 =$	25 °C
	Mount	Atmospheric pressure	p <sub>out,met</sub> =	97 kPa
		Sealevel	$H_2 =$	100 m

Table 8.1: Parameters of insulating glass

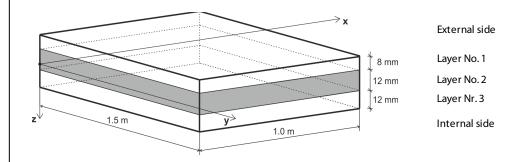


Figure 8.7: Layer composition

The length of the finite elements is 50 mm.

**D** 



# 8.2.1 Calculation in RF-GLASS

First, create a New Model in RFEM.

General History			
Model Name	Description		
Insulating glass			
Project Name	Description		
Examples	-		
Folder:			<b>b</b>
C:\Users\Public\Documents\D	lubal\Projects\Exam	ples	
Type of Model		Classification of Load Cases and Combinations	
3D		According to Standard:	
) 2D - XY (uz/ox/oy)	×	None	
© 2D - XZ (ux/uz/φγ)	Ž		
2D - XY (ux/uy/qz)		Create combinations automatically	
		Load combinations	
		<ul> <li>Result combinations (for linear analysis only)</li> </ul>	
Positive Orientation of Global Ax	cis Z	Template	
🔿 Upward		Open template model:	
Downward			
Comment			
			-

۴.

Having entered the general data, you then create a *New Rectangular Surface*: Select *Glass* as surface type, and then define a surface with the dimensions 1.0 m x 1.5 m.

New Rectangular Surface	×
Surface No.	Surface Type
1	Geometry: Plane
Material	Stiffness: Standard
1   Concrete C30/37   DIN 1045-1:2008-08	Surface thic Without tension Orthotropic Glass
Thickness	Laminate レオ
Ocnstant     Thickness d: 180.0 ▼ ♠ [mm]	Membrane Membrane - Orthotropic
🔘 Variable 📷	
Comment	
	OK Cancel

Figure 8.9: Dialog box New Rectangular Surface

Because you want to carry out the *Local* calculation type in RF-GLASS, the supports are defined directly in the module. Therefore, it is not necessary to define any supports in RFEM.

Figure 8.8: Dialog box New Model - General Data

### 8 Examples



<u>\*</u>

Although there is no external loading on the model, you have to create a load case to start the calculation in RF-GLASS. The self-weight should **not** be *Active*.

Edit Load Cases and Combinations		×
Load Cases Load Combinations Result Combinations		
Existing Load Cases	LC No.	To Solve
	General Calculation Parameters Action Category Without	
	Cial Imposed	
	Self-Weight	
	Factor in direction:           X:	
	Y: () Z: () () ()	
· · · · · · · · · · · · · · · · · · ·	Comment	
		OK Cancel

Figure 8.10: Dialog box Edit Load Cases and Combinations

In the FE Mesh Settings dialog box, set the required length of finite elements to **0.05 m**.

eneral		Surfaces	
arget length of finite lements	IFE: 0.050 km [m]	Maximum ratio of FE rectangle diagonals         ΔD:         1.800 ★ [-]	,∼IFE,
laximum distance between a ode and a line to integrate it into ne line	ε: 0.001 <b>€</b> [m]	Maximum out of plane inclination of two finite elements α.: 0.50 (*) [*]	3
laximum number of mesh nodes n thousands)	max: 500	FE mesh refinement along lines (with type 'Plate XY' only)	0 *** 03
embers		Relationship ∆ <sub>b</sub> : [·]	
lumber of divisions for special type embers able, elastic foundation, taper, onlinearity):	es of	Integrate unutilized objects into surfaces	4
<ul> <li>Activate divisions for straight m in surfaces, with concrete mate for nonlinear calculation)</li> </ul>	embers, which are not integrate	<ul> <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 🜩	Same squares where possible	Option
Activate member divisions for la post-critical analysis	arge deformation or	Mapped mesh preferred	Regenerate FE mesh on [OK]
Use division for straight membe integrated in surfaces, with	rs, which are not	Solids	
Target length I <sub>FE</sub> of finite el	ements	Refinement of FE mesh on solids containing close nodes	
○ Set length I <sub>FE</sub> :	(m)	Maximum number of elements (in thousands): 200	
Minimum number of member divisions:			
Use division for members with r	nodes lying on them		

Figure 8.11: Dialog box FE Mesh Settings



Next, start the RF-GLASS module (see Chapter 1.4).

Because you want to create insulating glass, you should select the *Local – Individual glass surfaces* calculation type in Window 1.1 *General Data*. You cannot select a load case now, because it does not contain any load data. This is indicated by an asterisk (\*) (see Chapter 3.1.1).

ata eral Data						
eral Data	1.1 General Data					
ers	Design of			Standard		
Supports al Supports	Surfaces No.:			None	- 🔁 🗟	
ai Supports ndary Members	1		🍾 🗙 🔳 🗛			
	Libburgha Lingth Charles Inc.	1.00.10.00.0				
	Ultimate Limit State Service Existing Load Cases	ceability Limit State	Selected for Des			1.1.1
	Qi LC1		Selected for Des	gn		S
						<b>RF-GLASS</b>
			>			4
			>>			
						(5)
						<b>Y</b>
						1.1
						Calculation and design of glass surfaces
	All (1)	- 2/ 22				
	Comment			Calculation Type		
				<ul> <li>Ocal - Individual glass</li> </ul>	surfaces	
				- O Global - Whole model		

Figure 8.12: Window 1.1 General Data

Therefore, you have to go to the 1.2 *Layers* window to define an insulating glass with a gas layer first. Then, you can return to Window 1.1 to select a load case.

urrent	Composition				List of Surfac	es		C	omposition No
1   Com	p. 1	▼ 4	› 🎦 👺		1				1
ayers									
	A	B	C	D	E		F	G	Н
Layer	Layer	Material	Thickness	Modulus of Elast.	Shear Mod		Poisson's Ratio	Specific Weight	Coeff. of Th
No.	Туре	Description	t [mm]	E [N/mm <sup>2</sup> ]	G [N/mm	2]	v [-]	γ <b>[</b> kN/m <sup>3</sup> ]	ατ [1/Η
1	Glass	Thermally Toughened Float Glass	8.00	70000.000	2845	5.300	0.230	25.00	9
2	Gas	Dry Air	12.00					0.01	
3	Glass	Thermally Toughened Float Glass	12.00	70000.000	2845	5.300	0.230	25.00	9
4									
5									
6									
7									
8									
9									
			1						
)						Info		00	
) [		External Side				Info		00	
)		External Side	Toughened Float	Glass			No.: 1		
		External Side	Toughened Float			Layer	No.: 1 ce weight:	0.200 [ktV/m <sup>2</sup> ]	
		External Side	-			Layer Surfac			
		External Side	-			Layer Surfac Σ Thic	ce weight:	0.200 [kN/m <sup>2</sup> ] 32.00 [mm]	
		External Side	-			Layer Surfac Σ Thic	ce weight:	0.200 [kN/m <sup>2</sup> ]	
		External Side	-			Layer Surfac Σ Thic	ce weight:	0.200 [kN/m <sup>2</sup> ] 32.00 [mm]	
		External Side External Side 1: Thermally 2: Dry Air 3: Thermally T	-			Layer Surfac Σ Thic	ce weight:	0.200 [kN/m <sup>2</sup> ] 32.00 [mm]	
		External Side External Side 1: Thermally 2: Dry Air 3: Thermally T	-			Layer Surfac Σ Thic	ce weight:	0.200 [kN/m <sup>2</sup> ] 32.00 [mm]	
		External Side External Side 1: Thermally 2: Dry Air 3: Thermally T	-			Layer Surfac Σ Thic	ce weight:	0.200 [kN/m <sup>2</sup> ] 32.00 [mm]	
		External Side External Side 1: Thermally 2: Dry Air 3: Thermally T	-			Layer Surfac Σ Thic	ce weight:	0.200 [kN/m <sup>2</sup> ] 32.00 [mm]	
		External Side External Side 1: Thermally 2: Dry Air 3: Thermally T	-			Layer Surfac Σ Thic	ce weight:	0.200 [kN/m <sup>2</sup> ] 32.00 [mm]	
		External Side External Side 1: Thermally 2: Dry Air 3: Thermally T	-		Local Axis z	Layer Surfac Σ Thic	ce weight:	0.200 [kN/m <sup>2</sup> ] 32.00 [mm]	
		External Side External Side 1: Thermally 2: Dry Air 3: Thermally T	-		Local Axis z Direction	Layer Surfac Σ Thic	ce weight:	0.200 [kN/m <sup>2</sup> ] 32.00 [mm]	
		External Side External Side 1: Thermally 2: Dry Air 3: Thermally T	-			Layer Surfac Σ Thic	ce weight:	0.200 [kN/m <sup>2</sup> ] 32.00 [mm]	

Figure 8.13: Window 1.2 Layers

## 8 Examples



Because you are interested only in the deformation of the model right now, select LC1 <u>only</u> in the *Serviceability Limit State* tab of the 1.1 *General Data* window.

.1 General Data				
Design of		Standard		
Surfaces No.:		None	- 🎦 🗟	13
1	🚯 🗙 🗖 Ali			
Ultimate Limit State Serviceability Limit State				
Existing Load Cases	Selected for Design			
		Qharacteri	SIC .	RF-GLASS
Al (1)			87 89	Calculation and design of glass surfaces
Comment		Calculation Type Calculation Type Calculation Type Calculation Type Calculation Type Calculation Type Calculation Type	es	

In the 1.3 *Line Supports* window, you then select the *Hinged - type 1* support type for the lines 1 to 4.

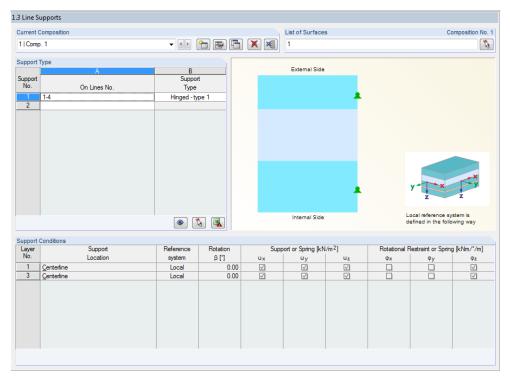


Figure 8.15: Window 1.3 Line Supports

Keep the Windows 1.4 Nodal Supports and 1.5 Boundary empty.

Figure 8.14: Window 1.1 General Data, tab Serviceability Limit State



Proceed to Window 1.6 *Climatic Load Parameters for Insulating Glass* to make the following entries:

.6 Climatic Load Param	neters for Insulating Glass				
Current Composition			List of Sur	faces	Composition No. 1
1   Comp. 1		- • 🎦 📴 📑	1		\$
Climatic Load Parameters	s - Summer				
V Use					
	Manufacturing		Mount	Difference	
Temperature:	0.0 🚔 [°C]	Temperature External:	25.0 🚔 [°C]	25.0 ÷ [°C]	
		Gas:	25.0 🚔 [°C]	25.0 🔶 [°C]	
	Manufacturing	Internal:	25.0 🚔 [°C]	25.0 🔶 [°C]	
Atmospheric pressure:	101.00 🚔 [kPa]	Atmospheric pressure:	97.00 🚔 [kPa]	-4.00 🔶 [kPa]	
Altitude:	0.0 🌲 [m]	Altitude:	100.0 🚔 [m]	100.0 🔶 <sup>[m]</sup>	
Climatic Load Parameters	s - Winter				
	Manufacturing		Mount	Difference	
Temperature:	["C]	Temperature External:	[°C]	(°C)	
		Gas:	(°C]	[°C]	
	Manufacturing	Internal:	(°C]	(°C]	
Atmospheric pressure:	(kPa)	Atmospheric pressure:	[kPa]	[kPa]	
Altitude:	(m)	Altitude:	[m]	(m)	
	v [m]	Philipping.	v [m]	↓ 11	
Force Load Distribution					
No.	Description	Load Part [%] on Glass Si External Interna			
LC1		100.0	0.0		
					<b>P P </b>

Figure 8.16: Window 1.6 Climatic Load Parameters for Insulating Glass

In the 1.8 *Serviceability Data* window, you add surface No. 1 to the *List of Surfaces*. Because the *Maximum border line* reference length type is selected, the *Reference LengthL* is specified automatically.

	А	B	C	DÍ	E	F F
lo.		Reference Len	gth	Canti-		
	List of Surfaces	Туре	L [m]	lever	Deformation Related to	Comment
1	1	Maximum border line	1.500		Undeformed system	
2						
						۵ 🔇

#### 8 Examples

-



Finally, check the settings in the Details of Composition dialog box.

Calculation / Modeling		
Method of Analysis	Stiffness Reduction Factors	
🔿 Linear static analysis	For shear stiffness elements	
<ul> <li>Large deformation analysis (nonlinear)</li> </ul>	K44:	
Newton-Raphson with constant stiffness matrix	K66:	
Newton-Raphson		_
Number of load increments: 5	Plate Bending Theory	
Modeling of Laminated Glass		
◯ 3D if ratio (G t / G f t f) is greater than:		
• 3 <u>D</u>	Insulating Glass Unit	
© <u>2</u> D	Consider secondary seal	
	Modulus of elasticity E : [N/nm <sup>2</sup> ]	
Calculation Options	Shear modulus G : [N/nm²]	
Save created temporary models	Poisson's ratio v:	
Consider coupling	Width b: [mm]	
CActivate FE mesh refinement		
Target FE length:		
Change standard settings	b	
Precison of convergence criteria of	÷	
nonlinear calculation:		
(Lower factor -> More exact)	Secondary seal	
	- Jeonitary sea	
	Number of finite element layers in gas	
	layers: 2	

Figure 8.18: Dialog box Details of Composition, tab Calculation / Modeling

#### Calculation

Start the [Calculation]. You can notice that due to insulating glass, the calculation proceeds in 3D, thus analyzing individual layers as solids.

	Running RFEM - Calculation by FEM		
SOLVER	RF-GLASS		
	Partial Steps		
	Load Incremental Step 4 / 5 Iteration 6	Maximum Displacement (including gas) [mm]	
	- Processing Input Data	31.3073	
	Creating 3D-Element Stiffness Matrices		
	- Creating 2D-Element Stiffness Matrices		
S	- Creating 1D-Element Stiffness Matrices		
	- Creating Global Stiffness Matrix	5/4	
	- Solving Equation System, Left Hand Side	Number of 3D Elements 3	3600
	Solving Equation System, Right Hand Side		1200
	- Determining 2D-Element Internal Forces	Number of 1D Elements	0
	Determining 1D-Element Internal Forces		1006 1036
		Number of Equations 24	1030
		cel 🗸 Gra	aph

Figure 8.19: Calculation

During the calculation, the *FE-Calculation* dialog box shows the maximum displacement, including the displacement in gas elements. As the displacements are considerable (see figure), the large deformation analysis is always used for insulating glass. This analysis considers the model geometry correctly.



After the calculation, the 3.1 Max Displacements shows the displacements of the glass panes.

	A	BÍ	С	D	EÍ	F	G	H (		J	
Surface	Point	Poir	nt Coordinates	[m]	Load-	Type of	Packet	Displaceme	ents [mm]	Ratio	
No.	No.	X	Y	Z	ing	Comb.	No.	Uz	Limit uz	uz [·]	
1	326	0.500	0.750	0.000	LC1; Summer	CH	1	-2.803	15.000	0.19	
	326	0.500	0.750	0.000			2	0.980	15.000	0.07	
	Maximum	Displacement	/ Maximum D	isplacement F	atio						
1	326	0.500	0.750	0.000	LC1; Summer	CH	1	-2.803	15.000	0.19	
	326	0.500	0.750	0.000	LC1; Summer	CH	1	-2.803	15.000	0.19	
							Marc	ratio: 0.19	9 ≤1 🥹		3 🛐 🏹 🖺 🛛

Figure 8.20: Window 3.1 Max Displacements

### 8.2.2 Check of Calculation

The check calculation of this example is carried out in RFEM. Because the calculation of insulating glass proceeds in 3D, you must adjust the RFEM model to the modifications in RF-GLASS. To this end, you selected the *Save created temporary models* check box in the *Details of Composition* dialog box (see Figure 8.18).

Open the generated model in RFEM (it can be found in the same project folder as the original file). Remove the gas solid in this model. Then, assign a surface load p to the solids of the glass panes. The surface load can be calculated from the thermal state equation for ideal gases:

$$\frac{pV}{T} = \text{const.}$$
(8.5)

$$\frac{p_p V_{01}}{T_p} = \frac{p_1 V_1}{T_1} = \frac{p_1 [V_{01} + C_v (p_1 - p_{out})]}{T_1}$$
(8.6)

where  $C_v$ : is the ductility of glass plates defined by the relation

$$C_{\nu}(p) = \frac{V(p)}{p} \quad \text{m}^3/\text{Pa}$$
(8.7)

where V(p): is the volume between the undeformed and deformed position of a given glass layer due to the pressure p. The ductility value depends on the instantaneous pressure value.



#### 8 Examples

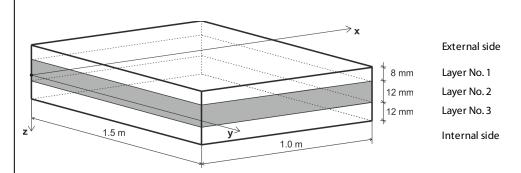


Figure 8.21: Layer composition

The initial gas volume in this example is:

 $V_{01} = a \cdot b \cdot t_2 = 1.0 \cdot 1.5 \cdot 0.012 = 0.018 \text{ m}^3$ 

The external gas pressure during at mount is determined as follows:

$$p_{\text{out}} = p_{\text{out,met}} - c_2 \cdot H_2 = p_{p,\text{met}} + \Delta p_{\text{met}} - c_2 \cdot H_2 = p_p + \Delta p_{\text{met}} - c_2 \Delta H$$
(8.8)

By substitution, you get:

$$\frac{p_p v_{01} r_1}{T_p} = p_1 \Big[ V_{01} + C_v (p_1 - p_p - \Delta p_{met} + c_2 \Delta H) \Big]$$
$$C_v p_1^2 + \Big[ V_{01} - C_v (p_p + \Delta p_{met} - c_2 \Delta H) \Big] p_1 - \frac{p_p V_{01} T_1}{T_p} = 0$$

The internal gas pressure at mount is then

$$p_{1} = \frac{C_{v}(p_{p} + \Delta p_{met} - c_{2} \cdot \Delta H) - V_{01} + \sqrt{\left[V_{01} - C_{v}(p_{p} + \Delta p_{met} - c_{2} \cdot \Delta H)\right]^{2} + 4C_{v}\frac{p_{p}V_{01}T_{1}}{T_{p}}}{2C_{v}}$$

 $p_p = p_{p,\text{met}} - c_2 \cdot H_1 = 101000 - 12 \cdot 0 = 101000 \text{ Pa}$ 

 $\Delta p_{met} = p_{out,met} - p_{p,met} = 97000 - 101000 = -4000 \text{ Pa}$ 

 $p_{out} = p_p + \Delta p_{met} - c_2 \cdot \Delta H = 101000 - 4000 - 12 \cdot (100 - 0) = 95800 \text{ Pa}$ 

The factor  $C_v$  depends on the support type, dimensions, and stiffness of the glass panes. It is calculated by using the following relations:

$$C_{v} = C_{v1} + C_{v2} \tag{8.9}$$

$$C_{v1}(p) = \frac{V_1}{p} = \frac{1}{p} \int_{0}^{ab} \int_{0}^{b} w_1(x, y) \, dx \, dy \tag{8.10}$$

$$C_{v2}(p) = \frac{V_2}{p} = \frac{1}{p} \int_{0}^{ab} \int_{0}^{b} w_2(x, y) dxdy$$
(8.11)

where  $C_{v1}$ : is the ductility of layer 1  $C_{v2}$ : is the ductility of layer 3

Because this factor depends on the pressure  $p = p_1 - p_{out}$ , the calculation is iterative.

#### 1. Iteration step

For  $p_1 = 100800 \text{ Pa}$ , you obtain

 $p = p_1 - p_{out} = 100800 - 95800 = 5000 \text{ Pa}$ 

In a nonlinear analysis with an FE mesh element length of 50 mm and a loading of p = 5000 Pa, RFEM determines the maximum deflections

 $w_1 = -6.132$  mm and  $w_2 = 3.207$  mm.

By using the RF-IMP module, you obtain the deformations of individual points in the model. Then, you can calculate the volume between the deformed surface and the surface before the deformation, that is,  $V_1 = 4.058 \cdot 10^{-3} \text{ m}^3$  and  $V_2 = 2.047 \cdot 10^{-3} \text{ m}^3$ .

$$C_{v1}(p) = \frac{V_1}{p} = \frac{4.058 \cdot 10^{-3}}{5000} = 8.116 \cdot 10^{-7} \text{ m}^3/\text{Pa}$$

$$C_{v2}(p) = \frac{V_2}{p} = \frac{2.047 \cdot 10^{-3}}{5000} = 4.094 \cdot 10^{-7} \text{ m}^3/\text{Pa}$$

$$C_v = C_{v1} + C_{v2} = 8.116 \cdot 10^{-7} + 4.094 \cdot 10^{-7} = 1.221 \cdot 10^{-6} \text{ m}^3/\text{Pa}$$

$$C_v \left( p_p + \Delta p_{\text{met}} - c_2 \cdot \Delta H \right) - V_{01} + \sqrt{\left[ V_{01} - C_v \left( p_p + \Delta p_{\text{met}} - c_2 \cdot \Delta H \right) \right]^2 + 4C_v}$$

$$P_1 - 2C_v$$

Recalculate the root from the previous formula:

$$\sqrt{\left[V_{01} - C_v \left(p_p + \Delta p_{\text{met}} - c_2 \cdot \Delta H\right)\right]^2 + 4C_v \frac{p_p V_{01} T_1}{T_p}} = \sqrt{\left[0.018 - 1.221 \cdot 10^{-6} \cdot 95800\right]^2 + 4 \cdot 1.221 \cdot 10^{-6} \frac{101000 \cdot 0.018 \cdot 298.15}{273.15}} = 0.1396 \text{ m}^3$$

The gas pressure is then equal to:

$$p_{1} = \frac{C_{v} \left( p_{p} + \Delta p_{met} - c_{2} \cdot \Delta H \right) - V_{01} + \sqrt{\left[ V_{01} - C_{v} \left( p_{p} + \Delta p_{met} - c_{2} \cdot \Delta H \right) \right]^{2} + 4C_{v} \frac{p_{p} V_{01} T_{1}}{T_{p}}}{2C_{v}} = \frac{1.221 \cdot 10^{-6} \cdot 95800 - 0.018 + 0.1396}{2 \cdot 1.221 \cdot 10^{-6}} = 97694 \text{ Pa}$$

#### 2. Iteration step

For  $p_1 = 97694 \text{ Pa}$ , you obtain

 $p = p_1 - p_{out} = 97694 - 95800 = 1894$  Pa

With this loading of p = 1894 Pa, RFEM determines the maximum deflections

 $w_1 = -3.497$  mm and  $w_2 = 1.323$  mm.

Redetermine the deformations of the points with RF-IMP. With  $V_1 = 2.253 \cdot 10^{-3} \text{ m}^3$  and  $V_2 = 0.840 \cdot 10^{-3} \text{ m}^3$ , you can determine the volumes of the deformations.

 $\frac{p_p V_{01} T_1}{T_1}$ 

$$C_{v1}(p) = \frac{V_1}{p} = \frac{2.253 \cdot 10^{-3}}{1894} = 1.190 \cdot 10^{-6} \text{ m}^3/\text{Pa}$$

$$C_{v2}(p) = \frac{V_2}{p} = \frac{0.840 \cdot 10^{-3}}{1894} = 4.435 \cdot 10^{-7} \text{ m}^3/\text{Pa}$$

$$C_v = C_{v1} + C_{v2} = 1.190 \cdot 10^{-6} + 4.435 \cdot 10^{-7} = 1.633 \cdot 10^{-6} \text{ m}^3/\text{Pa}$$

$$P_1 = \frac{C_v \left(p_p + \Delta p_{\text{met}} - c_2 \cdot \Delta H\right) - V_{01} + \sqrt{\left[V_{01} - C_v \left(p_p + \Delta p_{\text{met}} - c_2 \cdot \Delta H\right)\right]^2 + 4C_v \frac{p_p V_{01} T_1}{T_p}}{2C_v}}$$

Recalculate the root from the previous formula:

$$\sqrt{\left[V_{01} - C_v \left(p_p + \Delta p_{\text{met}} - c_2 \cdot \Delta H\right)\right]^2 + 4C_v \frac{p_p V_{01} T_1}{T_p}} = \sqrt{\left[0.018 - 1.633 \cdot 10^{-6} \cdot 95800\right]^2 + 4 \cdot 1.633 \cdot 10^{-6} \frac{101000 \cdot 0.018 \cdot 298.15}{273.15}} = 0.1792 \text{ m}^3$$

The gas pressure is then equal to:

$$p_{1} = \frac{C_{v}(p_{p} + \Delta p_{met} - c_{2} \cdot \Delta H) - V_{01} + \sqrt{\left[V_{01} - C_{v}\left(p_{p} + \Delta p_{met} - c_{2} \cdot \Delta H\right)\right]^{2} + 4C_{v}\frac{p_{p}V_{01}T_{1}}{T_{p}}}{2C_{v}} = \frac{1.633 \cdot 10^{-6} \cdot 95800 - 0.018 + 0.1792}{2 \cdot 1.633 \cdot 10^{-6}} = 97270 \text{ Pa}$$

#### 3. Iteration step

The procedure of the next steps is the same. Therefore, only the most important values are quoted.

 $p = p_1 - p_{out} = 97270 - 95800 = 1470 \text{ Pa}$   $w_1 = -2.920 \text{ mm}, w_2 = 1.034 \text{ mm}$   $V_1 = 1.873 \cdot 10^{-3} \text{ m}^3, V_2 = 0.656 \cdot 10^{-3} \text{ m}^3$   $C_v = C_{v1} + C_{v2} = 1.274 \cdot 10^{-6} + 4.464 \cdot 10^{-7} = 1.720 \cdot 10^{-6} \text{ m}^3/\text{Pa}$  $p_1 = 97204 \text{ Pa}$ 

#### 4. Iteration step

 $p = p_1 - p_{out} = 97204 - 95800 = 1404 \text{ Pa}$   $w_1 = -2.821 \text{ mm}, w_2 = 0.988 \text{ mm}$   $V_1 = 1.808 \cdot 10^{-3} \text{ m}^3, V_2 = 0.627 \cdot 10^{-3} \text{ m}^3$   $C_v = C_{v1} + C_{v2} = 1.288 \cdot 10^{-6} + 4.468 \cdot 10^{-7} = 1.735 \cdot 10^{-6} \text{ m}^3/\text{Pa}$  $p_1 = 97193 \text{ Pa}$ 



#### 5. Iteration step

 $p = p_1 - p_{out} = 97193 - 95800 = 1393 \text{ Pa}$   $w_1 = -2.805 \text{ mm}, w_2 = 0.981 \text{ mm}$   $V_1 = 1.800 \cdot 10^{-3} \text{ m}^3, V_2 = 0.623 \cdot 10^{-3} \text{ m}^3$   $C_v = C_{v1} + C_{v2} = 1.290 \cdot 10^{-6} + 4.469 \cdot 10^{-7} = 1.737 \cdot 10^{-6} \text{ m}^3/\text{Pa}$  $p_1 = 97192 \text{ Pa}$ 

#### 6. Iteration step

 $p = p_1 - p_{out} = 97192 - 95800 = 1392 \text{ Pa}$   $w_1 = -2.803 \text{ mm}, w_2 = 0.980 \text{ mm}$   $V_1 = 1.796 \cdot 10^{-3} \text{ m}^3, V_2 = 0.622 \cdot 10^{-3} \text{ m}^3$   $C_v = C_{v1} + C_{v2} = 1.290 \cdot 10^{-6} + 4.469 \cdot 10^{-7} = 1.737 \cdot 10^{-6} \text{ m}^3/\text{Pa}$  $p_1 = 97192 \text{ Pa}$ 

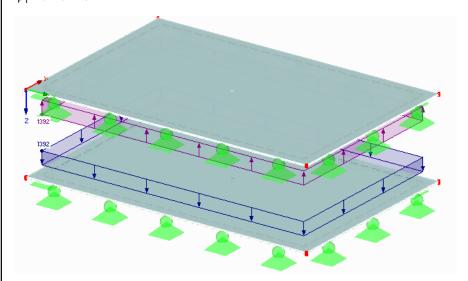


Figure 8.22: RFEM model

#### 7. Iteration step

 $p = p_1 - p_{out} = 97192 - 95800 = 1392 \text{ Pa}$   $w_1 = -2.802 \text{ mm}, w_2 = 0.980 \text{ mm}$   $V_1 = 1.796 \cdot 10^{-3} \text{ m}^3, V_2 = 0.622 \cdot 10^{-3} \text{ m}^3$   $C_v = C_{v1} + C_{v2} = 1.290 \cdot 10^{-6} + 4.469 \cdot 10^{-7} = 1.737 \cdot 10^{-6} \text{ m}^3/\text{Pa}$   $p_1 = 97191 \text{ Pa}$ 8. Iteration step  $p = p_1 - p_{out} = 97191 - 95800 = 1391 \text{ Pa}$  $w_1 = -2.802 \text{ mm}, w_2 = 0.980 \text{ mm}$ 

 $V_1 = 1.796 \cdot 10^{-3} \text{ m}^3$ ,  $V_2 = 0.622 \cdot 10^{-3} \text{ m}^3$ 



$$C_{v} = C_{v1} + C_{v2} = 1.290 \cdot 10^{-6} + 4.469 \cdot 10^{-7} = 1.737 \cdot 10^{-6} \text{ m}^3/\text{Pa}$$

*p*<sub>1</sub> = 97191 Pa

Because the results are identical in the seventh and eighth iteration steps, the iteration process terminates. Therefore, you obtain the maximum deflections

 $w_1 = -2.802 \text{ mm}, w_2 = 0.980 \text{ mm}.$ 

The result values in RF-GLASS are  $w_1 = -2.803$  mm and  $w_2 = 0.980$  mm, thus confirming the results.



# 8.3 Insulating Glass According to TRLV, Annex A

Consider the example presented in Chapter 8.2 according to TRLV, Annex A (or DIN 18008-2: 2010-12, Annex A).

For this, the model has to satisfy the following conditions:

- Rectangular surface without opening
- Exactly one gas layer
- Line support of the type Hinged type 7 on all boundary lines
- -Loading only by surface load

The calculation is carried out according to Kirchhoff's bending theory and the linear static analysis ([1], [2], [5]).

Glass pane dim	ension in x-axi	a =	1.0 m	
Glass pane dime	ension in y-axi	<i>b</i> =	1.5 m	
Thickness of ext	ternal glass lay	er	$t_1 =$	8 mm
Thickness of air	layer		$t_2 =$	12 mm
Thickness of int	ernal glass laye	er	$t_3 =$	12 mm
	modulus of elasticity			70000 MPa
Glass parameters	shear modul	us	G =	28455 MPa
	Poisson's rati	io	<i>v</i> =	0.23
	manufac- turing	temperature	$T_p =$	0°C
		atmospheric pressure	p <sub>p,met</sub> =	101 kPa
Climatic lass d		sealevel	$H_1 =$	0 m
Climatic load	mount	temperature (external = gas = internal)	T =	25 ℃
		atmospheric pressure	p <sub>out,met</sub> =	97 kPa
		sealevel	$H_2 =$	100 m

Table 8.2: Insulating glass parameters

In contrast to the previous example, the external glass pane is additionally loaded by the external loading  $w_1 = 1 \text{ kN/m}^2$ , that is to be entered in RFEM.

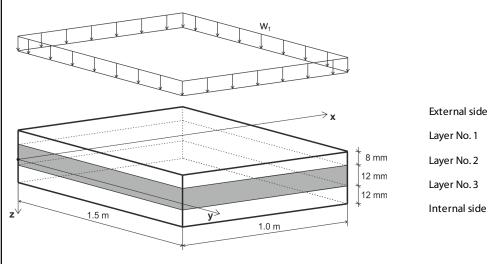


Figure 8.23: Loading scheme



The rate of the external glass pane in the total bending stiffness is:

$$\delta_1 = \frac{t_1^3}{t_1^3 + t_2^3} = \frac{8^3}{8^3 + 12^3} = 0.2286$$
 TRLV, Annex A, Equation A1

The rate of the internal glass pane in the total bending stiffness is:

$$\delta_2 = \frac{t_2^3}{t_1^3 + t_2^3} = \frac{12^3}{8^3 + 12^3} = 0.7714$$
 TRLV, Annex A, Equation A2

The characteristic edge length is then:

$$a^* = 28.9 \cdot \sqrt[4]{\frac{t_3 \cdot t_1^3 \cdot t_2^3}{(t_1^3 + t_2^3)B_V}}$$
 TRLV, Annex A, Equation A3

where  $B_v$ : is determined from the ratio a/b = 1000/1500 = 0.667

a/b	1.0	0.9	0.8	0.7	0.6	0.5	0.4	0.3	0.2	0.1
B <sub>v</sub>	0.0194	0.0237	0.0288	0.0350	0.0421	0.0501	0.0587	0.0676	0.0767	0.0857

TRLV, Annex A, Table A1

By linear interpolation, you obtain:

 $B_v = 0.0373$ 

$$a^* = 28.9 \cdot 4 \frac{t_3 \cdot t_1^3 \cdot t_2^3}{(t_1^3 + t_2^3)B_V} = 28.9 \cdot 4 \frac{12 \cdot 8^3 \cdot 12^3}{(8^3 + 12^3)0.0373} = 546 \text{ mm}$$

The factor  $\varphi$  is determined as:

$$\varphi = \frac{1}{1 + \left(\frac{a}{a^*}\right)^4} = \frac{1}{1 + \left(\frac{1000}{546}\right)^4} = 0.081$$
TRLV, Annex A, Equation A4

The pressure inside the insulating glass due to climatic changes is:

$$p_0 = c_1 \Delta T - \Delta p_{met} + c_2 \Delta H = 340 \cdot 25 - (97000 - 101000) + 12 \cdot 100 = 13700 \text{ Pa} = 13.7 \text{ kN/m}^2$$

TRLV, Annex A, Equation A5

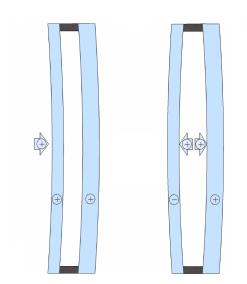


Figure 8.24: Wind load  $w_1$  (left) and load due to climatic changes  $p_0$  (right)



The following table shows the load distribution for the individual glass panes:

Loading on		Load part for		
		External pane	Internal pane	
External pane	wind w <sub>1</sub>	$(\delta_1 + \varphi \delta_2) \cdot w_1$	$(1-\varphi)\delta_2\cdot w_1$	
	snow s	$(\delta_1 + \varphi \delta_2) \cdot s$	$(1-\varphi)\delta_2\cdot s$	
Internal pane	wind w <sub>2</sub>	$(1-\varphi)\delta_1\cdot w_2$	$(\varphi \delta_1 + \delta_2) \cdot w_2$	
Both panes	internal pressure $p_0$	$-\varphi \cdot p_0$	$+ \varphi \cdot p_0$	

For this example:

TRLV, Annex A, Table A2

Loading on		Load p	part for
		External pane	Internal pane
External pane	W <sub>1</sub>	$(0.2286 + 0.081 \cdot 0.7714) \cdot 1.0 =$	(1-0.081)·0.7714·1.0 =
pane		$= 0.29 \text{ kN/m}^2$	$= 0.71  \text{kN/m}^2$
Both panes	$p_0$	$-0.081 \cdot 13.7 = -1.11  \text{kN/m}^2$	$0.081 \cdot 13.7 = 1.11  kN/m^2$

Table 8.3: Load for glass panes

The total loading is therefore:

- External glass pane  $f_1 = 0.29 + (-1.11) = -0.82 \text{ kN/m}^2$
- Internal glass pane  $f_2 = 0.71 + 1.11 = 1.82 \text{ kN/m}^2$

The input in RF-GLASS is done as described in Chapter 8.2. In the 1.1 *General Data* window, select the TRLV standard. In the 1.6 *Climatic Load Parameters* window, select the option *Calculation according to Appendix A*.

```
Options

Calculation according to Appendix A

-help model for rectangular surfaces supported by line support Hinged - type 7
```

Figure 8.25: Calculation according to TRLV, Annex A

created temporary models check box.

Check in the *Force Load Distribution* section of this window if 100 % of the load acts on the *External* glass side.

In the Details of Composition dialog box, select the Linear static analysis option and the Save



Calculation

The [Calculation] gives maximum deflections of  $w_1 = -2.015$  mm,  $w_2 = 1.323$  mm.

If we open the generated model in RFEM, we can display and check the loads of the glass surfaces.

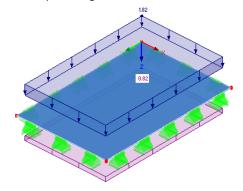


Figure 8.26: RFEM model with loading

Program RF-GLASS© 2013 Dlubal Software GmbH



# 8.4 Curved Insulating Glass

Consider a fixed curved insulating glass, with a composition displayed in Figure 8.27. The following parameters are used:

Glass pane dimens	а	=1.0 m		
Internal radius of c	R	=3.0 m		
Central angle			α	=30°
Thickness of glass	pane 1		<i>t</i> <sub>1</sub>	=5 mm
Thickness of air lay	rer		t <sub>SZR</sub>	=12 mm
Thickness of glass	pane 2		<i>t</i> <sub>2</sub>	=5 mm
Thickness of foil			t <sub>f</sub>	=0.76 mm
Thickness of glass	pane 3		t <sub>3</sub>	=5 mm
	modulus of elas	Ε	=70000 MPa	
Glass parameters	shear modulus	G	=28455 MPa	
	Poisson's ratio	v	=0.23	
	modulus of elas	Ε	=3 MPa	
Foil parameters	shear modulus	G	=1 MPa	
	Poisson's ratio	v	=0.499	
		temperature	$T_p$	=0 °C
	manufacturing	atmospheric pressure	$p_{p,\rm met}$	=101 kPa
		sealevel	H <sub>1</sub>	=0 m
Climatic load		temperature (external = gas = internal)	<i>T</i> <sub>1</sub>	=25 °C
	mount	atmospheric pressure	p <sub>out,met</sub>	=101 kPa
		sealevel	H <sub>2</sub>	=0 m

Table 8.4: Parameters of fixed curved insulating glass

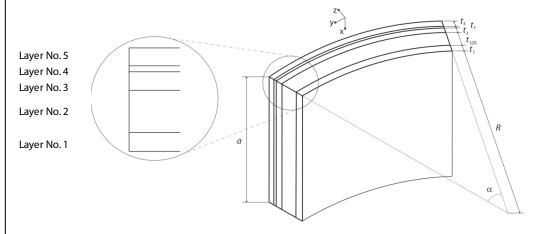


Figure 8.27: Curved glass

The length of the finite elements is 50 mm.

n

Ð



## 8.4.1 Calculation in RF-GLASS

First, we create a *New Model* in RFEM.

New Model - General Data			×
General Options History			
Model Name	Description		
Insulating glass unit - curved			
Project Name	Description		
Examples 💌			
Folder:			<b>b</b>
C:\Users\Public\Documents\Dlubal\	Projects\Examp	les	
Type of Model		Classification of Load Cases and Combinations	
③ 3D		According to Standard:	
© 2D - <u>X</u> Y (uz/φx/φy)	*X	None	
© <u>2</u> D - XZ (uχ/uz/φγ)		Create combinations automatically	
© 2D - X <u>Y</u> (ux/uy/φz)		Load combinations	
		$\bigcirc$ Result combinations (for linear analysis only)	
Positive Orientation of Global Axis Z		Template	
Upward     Downward		Open template model:	-
Comment			
			-
🦻 📝 🚾 🖪 喝		ОК	Cancel

Figure 8.28: Dialog box New Model - General Data

After entering the general data, you define an *Arc via Center Node, Edge Node and Angle*. The distance between the center and edge node is 3 m, the included angle is 30°.

Then, open the *Extrude Line into Surface* dialog box. Define the height h = 1 m and the offset as e = 0 m. Next, check if the surface axis system is oriented as shown in the following figure. If this is not the case, rotate it.

Because you want to carry out the *Local* calculation type in RF-GLASS, the supports are defined directly in the module. Therefore, it is not necessary to define any supports in RFEM.

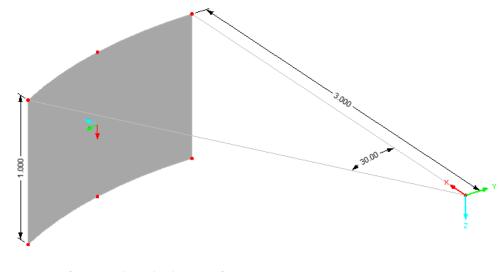


Figure 8.29: Defining arc and extruding line into surface

### 8 Examples



<u>\*</u>

Although there is no external loading on the model, you have to create a load case to start the calculation in RF-GLASS. The self-weight should **not** be *Active*.

Edit Load Cases and Combinations		×
Load Cases Load Combinations Result Combinations		
Existing Load Cases	LC No.	To Solve
	General Calculation Parameters Action Category Without	
	Cial Imposed	
	Self-Weight	
	Factor in direction:           X:	
	Y: () Z: () () ()	
· · · · · · · · · · · · · · · · · · ·	Comment	
		OK Cancel

Figure 8.30: Dialog box Edit Load Cases and Combinations

In the FE Mesh Settings dialog box, you specify **0.05 m** as target length of finite elements.

eneral		Surfaces	
arget length of finite lements	IFE: 0.050 🗭 [m]	Maximum ratio of FE rectangle diagonals ΔD: 1.800	H INTE
faximum distance between a ode and a line to integrate it into ne line	ε: 0.001 <u>★</u> [m]	Maximum out-of-plane inclination of two finite elements α: 0.50 (*)	
faximum number of mesh nodes n thousands)	max: 500	FE mesh refinement along lines (with type 'Plate XY' only)	01/22 3
lembers		Relationship 👍:	e   < 🔀 > 📲
lumber of divisions for special type tembers cable, elastic foundation, taper,		Integrate unutilized objects into surfaces	
onlinearity):	10 🛬	Shape of finite Ouadrangles only	
Activate divisions for straight me in surfaces, with concrete mate		elements:  Triangles only	$\Delta_{D} = \frac{D_{I}}{D_{D}} \qquad D_{I} \ge D_{D}$
for nonlinear calculation)		<ul> <li>Triangles and quadrangles</li> <li>Same squares where</li> </ul>	52
Minimum number of member divisions:	10 🜩	possible	Option
Activate member divisions for la post-critical analysis	arge 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 IFF of finite eligible	amanta	Refinement of FE mesh on solids containing close nodes	
Set length IFF :	[m]	Maximum number of elements	
Minimum number of member divisions:		(in thousands): 200	
Use division for members with n	odes lying on them		
2 📝 📭 📭 🚃			OK Cancel

Figure 8.31: Dialog box FE Mesh Settings



Now, start the RF-GLASS module.

Because you want to create insulating glass, you should select the *Local – Individual glass surfaces* calculation type in Window 1.1 *General Data*. You cannot select a load case now, because LC1 does not contain any load data. This is indicated by an asterisk (\*) (see Chapter 3.1.1).

Settings Help								
ıt Data	1.1 General Data							
General Data Layers	Design of		Standard					
Line Supports Nodal Supports Boundary Members	Surfaces No.: 1	🚯 🗙 🗉 Al	None	- 1				
	Ultimate Limit State Serviceability Limit	State						
	Existing Load Cases	Selected for De	sign					
	G · LC1	8 8			<b>RF-GLASS</b>			
					Calculation and design of glass surfaces			
	All (1)							
	Comment		Calculation Type					
			Local - Individual glass su     Global - Whole model in R					
<b>FB</b>	Calculation Details Stand	dard Graphics			OK Cano			

Figure 8.32: Window 1.1 General Data

Therefore, go to the 1.2 *Layers* window, where you select an insulating glass with gas layer. Now it is possible to select a load case in Window 1.1.

Intent	Composition				List of Surfac	es		C	omposition No
Com	p. 1	▼ 1	• 🛅 🗃		1				
ayers									
	A	В	С	D	E		F	G	Н
ayer	Layer	Material	Thickness	Modulus of Elast.	Shear Mod		Poisson's Ratio	Specific Weight	Coeff. of Th
No.	Туре	Description	t [mm]	E [N/mm <sup>2</sup> ]	G [N/mm		v [-]	γ <b>[</b> kN/m <sup>3</sup> ]	αт [1/
1	Glass	Thermally Toughened Float Glass	5.00	70000.000	2845	5.300	0.230	25.00	
2	Gas	Dry Air	12.00					0.01	
3	Glass	Thermally Toughened Float Glass	5.00	70000.000	2845	55.300	0.230	25.00	9
4	<u>F</u> oil	PVB 22 °C loading until 3 min	0.76	3.000		1.001	0.499	10.70	1
5	Glass	Thermally Toughened Float Glass	5.00	70000.000	2845	55.300	0.230	25.00	
6									
7									
8									
9									
) [								00	
						Info			
	E	xternal Side					No · 1		
		- 1: Thermally 1	oughened Float	Glass		Layer	No.: 1	0.125 mm 2	
	E	1: Thermally T	-			Layer	No.: 1 ce weight:	0.125 [kN/m <sup>2</sup> ]	
		1: Thermally 1 2: Dry Air 3: Thermally T 4: PVB 22 °C I	oughened Float G oading until 3 min	Blass		Layer Surfa	ce weight:		
		1: Thermally 1 2: Dry Air 3: Thermally T 4: PVB 22 °C I	- oughened Float G	Blass		Layer Surfa		0.125 [kN/m <sup>2</sup> ] 27.76 [mm]	
		1: Thermally 1 2: Dry Air 3: Thermally T 4: PVB 22 °C I	oughened Float G oading until 3 min	Blass		Layer Surfac Σ Thie	ce weight:	27.76 [mm]	
		- 1: Thermally 1 - 2: Dry kir - 3: Thermally T - 4: PVB 22 * C1 - 5: Thermally Tr	oughened Float G oading until 3 min	Blass		Layer Surfac Σ Thie	ce weight:		
		1: Thermally 1 2: Dry Air 3: Thermally T 4: PVB 22 °C I	oughened Float G oading until 3 min	Blass		Layer Surfac Σ Thie	ce weight:	27.76 [mm]	
		- 1: Thermally 1 - 2: Dry kir - 3: Thermally T - 4: PVB 22 * C1 - 5: Thermally Tr	oughened Float G oading until 3 min	Blass		Layer Surfac Σ Thie	ce weight:	27.76 [mm]	
		- 1: Thermally 1 - 2: Dry kir - 3: Thermally T - 4: PVB 22 * C1 - 5: Thermally Tr	oughened Float G oading until 3 min	Blass		Layer Surfac Σ Thie	ce weight:	27.76 [mm]	
		- 1: Thermally 1 - 2: Dry kir - 3: Thermally T - 4: PVB 22 * C1 - 5: Thermally Tr	oughened Float G oading until 3 min	Blass		Layer Surfac Σ Thie	ce weight:	27.76 [mm]	
		- 1: Thermally 1 - 2: Dry kir - 3: Thermally T - 4: PVB 22 * C1 - 5: Thermally Tr	oughened Float G oading until 3 min	Blass		Layer Surfac Σ Thie	ce weight:	27.76 [mm]	
		- 1: Thermally 1 - 2: Dry kir - 3: Thermally T - 4: PVB 22 * C1 - 5: Thermally Tr	oughened Float G oading until 3 min	slass I Ilass		Layer Surfac Σ Thie	ce weight:	27.76 [mm]	
		- 1: Thermally 1 - 2: Dry kir - 3: Thermally T - 4: PVB 22 * C1 - 5: Thermally Tr	oughened Float G oading until 3 min	slass I Ilass	Local Axis z	Layer Surfac Σ Thie	ce weight:	27.76 [mm]	
		- 1: Thermally 1 - 2: Dry kir - 3: Thermally T - 4: PVB 22 * C1 - 5: Thermally Tr	oughened Float G oading until 3 min	slass I Ilass	Local Axis z Direction	Layer Surfac Σ Thie	ce weight:	27.76 [mm]	
		- 1: Thermally 1 - 2: Dry kir - 3: Thermally T - 4: PVB 22 * C1 - 5: Thermally Tr	oughened Float G oading until 3 min	slass I Ilass		Layer Surfac Σ Thie	ce weight:	27.76 [mm]	

Figure 8.33: Window 1.2 Layers

## 8 Examples



Because you are currently only interested in the deformation of the model, select LC1 <u>only</u> in the *Serviceability Limit State* tab of the 1.1 *General Data* window.

1 General Data			
Design of		Standard	
Surfaces No.:		None	- 🖻 🖻 🛛
1	🍾 🗙 📃 Ali		
Ultimate Limit State Serviceability Limit State			
Existing Load Cases	Selected for Design		
	0         LC1           >         >            >	<u>Characteristic</u>	RF-GLASS
Al (1)			Calculation and design of glass surfaces
Comment		Calculation Type	
	*	Local - Individual glass surfaces	
	-	Global - Whole model in RFEM	

Figure 8.34: Window 1.1 General Data, tab Serviceability Limit State

In the 1.3 Line Supports window, you select the support type Rigid for the lines 1 to 4.

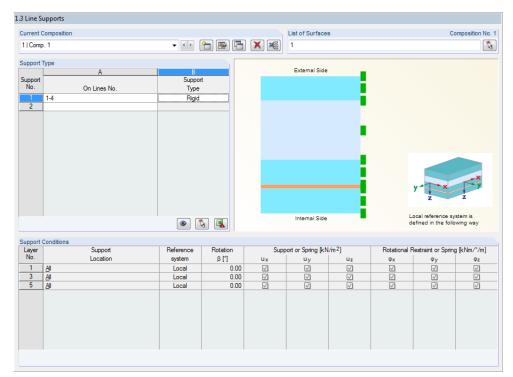


Figure 8.35: Window 1.3 Line Supports

Leave Windows 1.4 Nodal Supports and 1.5 Boundary empty.



Proceed to Window 1.6 *Climatic Load Parameters for Insulating Glass* to make the following entries:

	neters for Insulating Glass				
Current Composition			List of Surfa	ces	Composition No. 1
1   Comp. 1		- + 🎦 🗃 🗄			3
Climatic Load Parameter	rs - Summer				
V Use					
	Manufacturing		Mount	Difference	
Temperature:	0.0 ≑ [°C]	Temperature External:	25.0 🚔 [°C]	25.0 ÷ [°C]	
		Gas:	25.0 ≑ [°C]	25.0 ÷ [°C]	
	Manufacturing	Internal:	25.0 🔶 [°C]	25.0 ♀ [°C]	
Atmospheric pressure:	0.101 (N/mm <sup>2</sup> )	Atmospheric pressure:	0.101 🚔 [N/mm <sup>2</sup> ]	0.000 - [N/mm <sup>2</sup> ]	
Altitude:	0.0 🌲 [m]	Altitude:	0.0 ≑ [m]	0.0 🌲 [m]	
Climatic Load Parameter	rs - Winter				
🗾 Use					
	Manufacturing		Mount	Difference	
Temperature:	(°C]	Temperature External:	÷["C]	["C]	
		Gas:	- [°C]	[*C]	
	Manufacturing	Internal:	÷ ["C]	["C]	
Atmospheric pressure:	[N/mm <sup>2</sup> ]	Atmospheric pressure:	[N/mm <sup>2</sup> ]	[N/mm <sup>2</sup> ]	
Altitude:	(m)	Altitude:	(m)	(m)	
Hadde.	<b>v</b> [11]	Alutode.	<b>v</b> [11]	Tul	
Force Load Distribution					
No.	Description	Load Part [%] on Glass			
LC1	Description	External Intern 100.0	0.0		
		100.0	0.0		
					( <b>m</b> ) ( <b>m</b> ) ( <b>m</b> )
					<b>P P </b>

Figure 8.36: Window 1.6 Climatic Load Parameters for Insulating Glass

In the 1.8 *Serviceability Data* window, you add surface No. 1 to the *List of Surfaces*. Because the *Maximum border line* reference length type is selected, the *Reference Length L* is automatically completed.

A	B	C	D	E	F
	Reference Len	igth	Canti-		
List of Surfaces	Туре	L [m]	lever	Deformation Related to	Comment
1	Maximum border line	1.571		Undeformed system	
					۲



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

etails of Composition No. 1	
Calculation / Modeling	
Method of Analysis	Stiffness Reduction Factors
Linear static analysis	For shear stiffness elements
<ul> <li>Large deformation analysis (nonlinear)</li> </ul>	K44:
Newton-Raphson with constant stiffness matrix	K55:
Newton-Raphson	
Number of load increments: 5	Plate Bending Theory O Mindlin
Modeling of Laminated Glass	Kirchhoff
◯ <u>3</u> D if ratio (G t / G f t f) is greater than:	Insulating Glass Unit
© 3 <u>D</u>	
○ <u>2</u> D	Consider secondary seal
Calculation Options	Modulus of elasticity E : [Wmm <sup>2</sup> ]
Save created temporary models	Shear modulus G : [N/mm <sup>2</sup> ]
Consider coupling	Poisson's ratio v:
Activate FE mesh refinement	Width b: [mm]
Target FE length: [m]	
Change standard settings	b
Precison of convergence criteria of	**
nonlinear calculation:	
(Lower factor -> More exact)	Secondary seal
	— Secondary sear
	Number of finite element layers in gas
	layers: 2
	OK Cancel
	OK Cancel

Figure 8.38: Dialog box Details of Composition

Start the [Calculation]. Because of the insulating glass, the calculation runs in 3D in which all layers are analyzed as solids.

The 3.1 Max Displacements window shows the displacements of the glass panes.

	A	B	C	D	EÍ	F	G	H (	1 1	J	
urface	Point	Poin	t Coordinates	[m]	Load-	Type of	Packet	Displacemen	nts (mm)	Ratio	
No.	No.	X	Y	Z	ing	Comb.	No.	Uz	Limit uz	uz [·]	
1	421	2.772	-1.148	0.600	LC1; Summer	CH	1	-0.265	15.708	0.02	
	454	2.772	-1.148	0.650			2	0.123	15.708	0.01	
	Maximum	Displacement	/ Maximum D	isplacement R	atio						
1	421	2.772	-1.148		LC1; Summer	CH	1	-0.265	15.708	0.02	
	421	2.772	-1.148	0.600	LC1; Summer	CH	1	-0.265	15.708	0.02	
							Maxr		≤1 ☺		a 9 % E I

Figure 8.39: Window 3.1 Max Displacements





### 8.4.2 Check of Calculation

The check calculation of this example is carried out in RFEM. Because the calculation of insulating glass proceeds in 3D, you must adjust the RFEM model to the modifications in RF-GLASS. To this end, you selected the *Save created temporary models* check box in the *Details of Composition* dialog box (see Figure 8.38).

Open the generated model in RFEM (it can be found in the same project folder as the original file). Remove the gas solid in this model. Then, assign a surface load *p* to the solids of the glass panes. The surface load can be calculated from the thermal state equation for ideal gases as follows:

$$\frac{pV}{T} = \text{const.}$$
(8.12)

$$\frac{p_p V_{01}}{T_p} = \frac{p_1 V_1}{T_1} = \frac{p_1 [V_{01} + C_v (p_1 - p_{out})]}{T_1}$$
(8.13)

where  $C_v$ : is the ductility of the glass plates, defined as

$$C_{v}(p) = \frac{V(p)}{p} \quad \text{m}^{3}/\text{Pa}$$
(8.14)

where V(p): is the volume between the undeformed and deformed position of a given glass layer due to the pressure p. The ductility value depends on the instantaneous pressure value.

The initial volume in this example is:

$$V_{01} = a \cdot b \cdot t_2 = 1.0 \cdot 1.5 \cdot 0.012 = 189.153 \cdot 10^{-4} \text{ m}^3$$

The external gas pressure at mount is determined as follows:

$$p_{\text{out}} = p_{\text{out,met}} - c_2 \cdot H_2 = p_{p,\text{met}} + \Delta p_{\text{met}} - c_2 \cdot H_2 = p_p + \Delta p_{\text{met}} - c_2 \Delta H$$
(8.15)

By substitution, you obtain:

$$\frac{p_p V_{01} T_1}{T_p} = p_1 \Big[ V_{01} + C_v (p_1 - p_p - \Delta p_{met} + c_2 \Delta H) \Big]$$
$$C_v p_1^2 + \Big[ V_{01} - C_v (p_p + \Delta p_{met} - c_2 \Delta H) \Big] p_1 - \frac{p_p V_{01} T_1}{T_p} = 0$$

The internal gas pressure at mount is then

$$p_{1} = \frac{C_{v} \left( p_{p} + \Delta p_{met} - c_{2} \cdot \Delta H \right) - V_{01} + \sqrt{\left[ V_{01} - C_{v} \left( p_{p} + \Delta p_{met} - c_{2} \cdot \Delta H \right) \right]^{2} + 4C_{v} \frac{p_{p} V_{01} T_{1}}{T_{p}}}{2C_{v}}$$

 $p_p = p_{p,\text{met}} - c_2 \cdot H_1 = 101000 - 12 \cdot 0 = 101000 \text{ Pa}$ 

 $\Delta p_{\rm met} = p_{\rm out,met} - p_{p,\rm met} = 101000 - 101000 = 0$  Pa

 $p_{\text{out}} = p_p + \Delta p_{\text{met}} - c_2 \cdot \Delta H = 101000 - 0 - 12 \cdot (100 - 0) = 101000 \text{ Pa}$ 

The factor  $C_v$  depends on the support type, dimensions, and stiffness of the glass panes. You can calculate them by using the following formula:

$$C_{v} = C_{v1} + C_{v2} \tag{8.16}$$

$$C_{v1}(p) = \frac{V_1}{p} = \frac{1}{p} \int_{0}^{ab} w_1(x, y) \, dx \, dy \tag{8.17}$$

$$C_{v2}(p) = \frac{V_2}{p} = \frac{1}{p} \int_{0}^{ab} \int_{0}^{b} w_2(x, y) dx dy$$
(8.18)

where  $C_{v1}$ : is the ductility of layer 1

 $C_{\nu 2}$ : is the ductility of layer 3

Because this factor depends on the pressure  $p = p_1 - p_{out}$ , the calculation is iterative.

#### 1. Iteration step

For  $p_1 = 106000 \text{ Pa}$ , you obtain

 $p = p_1 - p_{out} = 106000 - 10100 = 5000 \text{ Pa}$ 

In a nonlinear analysis with an FE mesh element length of 50 mm and a loading of p = 5000 Pa, RFEM determines the maximum deflections

 $w_1 = -0.178 \text{ mm}$  and  $w_2 = 0.084 \text{ mm}$ .

By using the RF-IMP module, you obtain the deformations of individual points in the model. Then, you can calculate the volume between the deformed surface and the surface before the deformation, that is,  $V_1 = 14.997 \cdot 10^{-5} \text{ m}^3$  and  $V_2 = 7.478 \cdot 10^{-5} \text{ m}^3$ .

$$C_{v1}(p) = \frac{V_1}{p} = \frac{14.997 \cdot 10^{-5}}{5000} = 2.999 \cdot 10^{-8} \text{ m}^3/\text{Pa}$$

$$C_{v2}(p) = \frac{V_2}{p} = \frac{7.478 \cdot 10^{-5}}{5000} = 1.496 \cdot 10^{-8} \text{ m}^3/\text{Pa}$$

$$C_v = C_{v1} + C_{v2} = 2.999 \cdot 10^{-8} + 1.496 \cdot 10^{-8} = 4.495 \cdot 10^{-8} \text{ m}^3/\text{Pa}$$

$$C_v \left(p_p + \Delta p_{\text{met}} - c_2 \cdot \Delta H\right) - V_{01} + \sqrt{\left[V_{01} - C_v \left(p_p + \Delta p_{\text{met}} - c_2 \cdot \Delta H\right)\right]^2 + 4C_v \frac{p_p V_{01} T_v}{T_p}}{T_p}$$

 $p_1 =$ 

Recalculate the root from the previous formula:

$$\sqrt{\left[V_{01} - C_v \left(p_p + \Delta p_{met} - c_2 \cdot \Delta H\right)\right]^2 + 4C_v \frac{p_p V_{01} T_1}{T_p}} = \sqrt{\left[189.153 \cdot 10^{-4} - 4.495 \cdot 10^{-8} \cdot 101000\right]^2 + 4 \cdot 4.495 \cdot 10^{-8} \frac{101000 \cdot 189.153 \cdot 10^{-4} \cdot 298.15}{273.15}} = 0.02412 \text{ m}^3$$

 $2C_{v}$ 

The gas pressure is then equal to:

$$p_{1} = \frac{C_{v} \left(p_{p} + \Delta p_{met} - c_{2} \cdot \Delta H\right) - V_{01} + \sqrt{\left[V_{01} - C_{v} \left(p_{p} + \Delta p_{met} - c_{2} \cdot \Delta H\right)\right]^{2} + 4C_{v} \frac{p_{p}V_{01}T_{1}}{T_{p}}}{2C_{v}}}{2C_{v}} = \frac{4.495 \cdot 10^{-8} \cdot 101000 - 189.153 \cdot 10^{-4} + 0.02412}{2 \cdot 4.495 \cdot 10^{-8}} = 108351 \,\mathrm{Pa}$$

#### 2. Iteration step

For  $p_1 = 108351 \, \text{Pa}$  , we obtain

 $p = p_1 - p_{out} = 108351 - 101000 = 7351 \, Pa$ 

With this loading of  $p = 7351 \,\mathrm{Pa}$ , RFEM determines the maximum deflections

 $w_1 = -0.265 \text{ mm}$  and  $w_2 = 0.123 \text{ mm}$ .

The deformations of the points are recalculated with RF-IMP. Then, you can determine the volume of the deformations with  $V_1 = 2.212 \cdot 10^{-4} \text{ m}^3$  and  $V_2 = 1.099 \cdot 10^{-4} \text{ m}^3$ .

$$C_{v1}(p) = \frac{V_1}{p} = \frac{2.212 \cdot 10^{-4}}{7351} = 3.008 \cdot 10^{-8} \text{ m}^3/\text{Pa}$$

$$C_{v2}(p) = \frac{V_2}{p} = \frac{1.099 \cdot 10^{-4}}{7351} = 1.494 \cdot 10^{-8} \text{ m}^3/\text{Pa}$$

$$C_v = C_{v1} + C_{v2} = 3.008 \cdot 10^{-8} + 1.494 \cdot 10^{-8} = 4.503 \cdot 10^{-8} \text{ m}^3/\text{Pa}$$

$$C_v \left(p_p + \Delta p_{\text{met}} - c_2 \cdot \Delta H\right) - V_{01} + \sqrt{\left[V_{01} - C_v \left(p_p + \Delta p_{\text{met}} - c_2 \cdot \Delta H\right)\right]^2 + 4C_v \frac{p_p V_{01} T_1}{T}}{T}$$

$$p_{1} = \frac{C_{v}(p_{p} + \Delta p_{met} - c_{2} \cdot \Delta H) - V_{01} + \sqrt{\left\lfloor V_{01} - C_{v}(p_{p} + \Delta p_{met} - c_{2} \cdot \Delta H) \right\rfloor} + 4C_{v} \frac{P_{p} \cdot 0^{v+1}}{T_{p}}}{2C_{v}} = 108349 \text{ Pa}$$

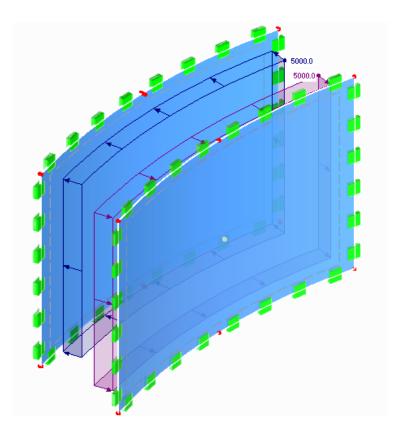


Figure 8.40: RFEM model



#### 3. Iteration step

The procedure of the next steps is the same. Therefore, only the most important values are quoted.

 $p = p_1 - p_{out} = 108349 - 101000 = 7349 \text{ Pa}$   $w_1 = -0.265 \text{ mm}, w_2 = 0.123 \text{ mm}$   $V_1 = 2.210 \cdot 10^{-4} \text{ m}^3, V_2 = 1.098 \cdot 10^{-4} \text{ m}^3$   $C_v = C_{v1} + C_{v2} = 3.008 \cdot 10^{-8} + 1.494 \cdot 10^{-8} = 4.502 \cdot 10^{-8} \text{ m}^3/\text{Pa}$  $p_1 = 108349 \text{ Pa}$ 

Because the results in the second and third iteration steps are identical, the iteration process is terminated. Thus, the maximum deflections are

 $w_1 = -0.265 \text{ mm}, w_2 = 0.123 \text{ mm}.$ 

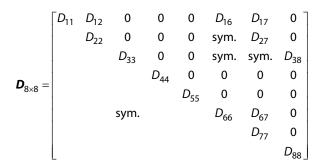
The result values in RF-GLASS are  $w_1 = -0.265$  mm and  $w_2 = 0.123$  mm, thus confirming the results.



# 9. Appendix A

# 9.1 Stiffness Matrix Check for Positive Definiteness

The stiffness matrix which is given in the form:



is checked for the following conditions:

- Matrix **D** is positive definite (i.e., all of the leading principal minors must be positive. As a consequence, there are positive values on the diagonal:  $D_{ii} > 0$ , i = 1, ..., n.)
- Moreover, it is required that (positive definiteness in a more restrictive sense):

$$\det \begin{bmatrix} D_{11} & D_{12} \\ D_{12} & D_{22} \end{bmatrix} \ge cD_{11}D_{22}$$
$$\det \begin{bmatrix} D_{66} & D_{67} \\ D_{67} & D_{77} \end{bmatrix} \ge cD_{66}D_{77}$$
$$c = 1 - 0.999^2 = 0.001999$$



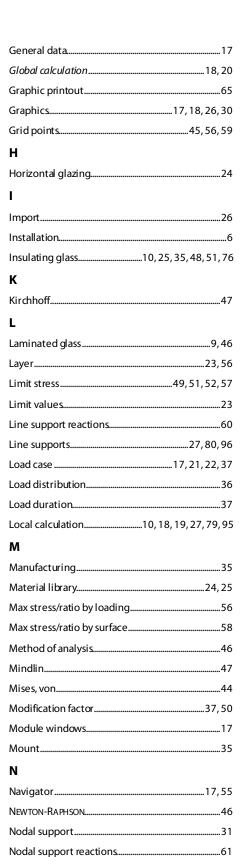
# **A** Literature

- [1] TRLV. Technische Regeln für die Verwendung von linienförmig gelagerten Verglasungen. August 2006.9 p.
- [2] WELLER, Bernhard, et al. *Glasbau-Praxis : Konstruktion und Bemessung*. Berlin : Bauwerk, 2009. 263 p. ISBN 978-3-89932-190-6.
- [3] WELLER, Bernhard, et al. *Glasbau-Praxis in Beispielen : Konstruktion und Berechnung*. Berlin : Bauwerk, 2005. 150 p. ISBN 3-934369-47-2.
- [4] DIN 18008-1. Glas im Bauwesen Bemessungs- und Konstruktionsregeln Teil 1: Begriffe und allgemeine Grundlagen. Dezember 2010. 14 p.
- [5] DIN 18008-2. Glas im Bauwesen Bemessungs- und Konstruktionsregeln Teil 2: Linienförmig gelagerte Verglasungen. Dezember 2010. 13 p.
- [6] OFNER, Robert. *Leichtbau und Glasbau: Bemessung von Glaskonstruktionen,* Vorlesungsskript. Technische Universität Graz, Institut für Stahlbau, Holzbau und Flächentragwerke, Wintersemester 2003/2004.
- [7] David R. LIDE, ed. CRC Handbook of Chemistry and Physics, 89th Edition (Internetversion 2009), CRC Press/Taylor and Francis, Boca Raton, FL.

A



Accidental22
Action combination51, 52, 53
Altitude35
Atmospheric pressure35
В
Bach
Boundary members
Browsing through module windows
c
Calculation17,39
Calculation type18
Characteristic23,62
Climatic load parameter
Comment21
Construction factor50
Coordinates56
Curved insulating glass92
D
Decimal places
Deflection23, 51, 52, 53
Design of18
Design situation
Details17, 39
Details of composition24, 46, 82
Diagram57,67
DIN 1800849
DIN 18008-2, Annex A36, 89
Displacements62
E
Equivalent stress44
Exit RF-GLASS17,55
Export26, 30
Export results
F
FE mesh points
Filter24, 59, 61
Foil9
Frequent23,62
G
Gas pressure





# Ρ

Panel7
Partial safety factor22, 50, 52
Parts list63
Persistent and transient22
Plate bending theory47
Positive definiteness103
Printout report
Q
Quasi-permanent23, 62
R
Rankine44
Ratio57, 62
Reference length62
Relation scale55
Results45, 55
S
Save temporary model47
Secondary seal10
Serviceability51, 52, 53
Serviceability data
Shear coupling between layers

Standard18, 49
Start calculation54
Start RF-GLASS6
Stiffness matrix12
Stiffness matrix elements13,71
Stiffness reduction47
Stress distribution67
Stresses40, 57
Stresses in all points59
Symbols8
т
Temperature35
Tresca44
TRLV51
TRLV, Annex A24, 36, 89
U
Units
User profile68
V
Vertical glazing24
View mode55