Version
November 2016

Program

RFEM 5

Orthotropic Surfaces

Program Description

Am Zellweg 2
93464 Tiefenbach
Germany

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

Contents

1 Introduction

1.1 Orthotropic Surfaces in RFEM 5

RFEM 5 Orthotropic Surfaces, as the name says, is an integrated part of RFEM 5 which deals with the analysis of surfaces, whose material and/or geometrical properties reveal elastic orthotropy in geometrical, material or combined sense. The new feature expands the previous orthotropic surfaces, integrated in RFEM 4 and introduces a series of new, in practice highly usable and from the academical point of view highly desirable capabilities with regards to static and dynamic analysis of two-dimensional isotropic/orthotropic plates and membranes.

Orthotropic surfaces are nowadays widely used in building industries-—from reinforced and voided concrete slabs to thin or thick-walled plates, reinforced by ribs in one or two directions. In the mechanical engineering field there is also a high applicability and demand for such surfaces. RFEM 5 offers a robust dialog, by means of which a 3D solid of a complicated geometry could be easily transformed into an equivalent 2D Mindlin or a Kirchoff surface, whose elastic proper-ties are both a function of the geometry and/or the material orthotropic constants and direc-tions. Thus the laborious task of creating solids and supporting them is easily circumvented with a high level of efficiency and accuracy. The user could either use one of the eight available geometries in a combination with a chosen material, or simply input his own stiffness matrix coefficients, provided that these are known apriori. In any case, the resulting elastic stiffness matrices are dispalyed, as well as the ones, resulting from the orthotropic rotation and any optional user modifications, accompanied by all necessary checks on positive definiteness.

RFEM 5 orthotropic surfaces resemble the previous feature of RFEM 4 in terms of modeling and combining surfaces with other features such as elastic foundations, plasticity, nonlinearities and second-order effects in static and dynamic analysis. However, there are certain material, geometrical, and analytical restrictions to each plate type which are clearly outlined in the proceeding chapters. The graphical output remains the same as the one the user has already familiarized himself with in RFEM 4.

We wish you lots of success in using this specific feature of RFEM.

Your team from DLUBAL SOFTWARE GMBH

1.2 RFEM Development Team

The following people were involved in the development of orthotropic surfaces:

Program coordination

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

Programming

Ph.D. Gabriel Pathó
M.Sc. Evgeni Pirianov
Ing. Toni Sakiqi

Database

Ph.D. Jan Rybín

Program design, dialog figures, icons

Dipl.-Ing. Georg Dlubal
MgA. Robert Kolouch

Blocks

Ing. Dmitry Bystroc
Ing. Tommy Brtek
Ing. Evžen Haluzík

Program supervision

Ing. Ondřej Čížek
M.Sc. Evgeni Pirianov
Ph.D. Vítězslav Štembera

Manual, examples

Ing. Toomas Kaaljas
Ph.D. Gabriel Pathó
M.Sc. Evgeni Pirianov
Ph.D. Vítězslav Štembera
Dipl.-Ing. (FH) Robert Vogl

Technical support, quality management

Dipl.-Ing. (BA) Markus Baumgärtel
M.Sc. Sonja von Bloh
Dipl.-Ing. Frank Faulstich
Dipl.-Ing. (FH) René Flori
Dipl.-Ing. (FH) Stefan Frenzel
Dipl.-Ing. (FH) Walter Fröhlich
Dipl.-Ing. (FH) Paul Kieloch
Dipl.-Ing. (FH) Bastian Kuhn
Dipl.-Ing. (FH) Ulrich Lex
Dipl.-Ing. (BA) Sandy Matula
M.Eng. Dipl.-Ing. (BA) Andreas Niemeier
Dipl.-Ing. (FH) Gerhard Rehm
M.Sc. Dipl.-Ing. (FH) Frank Sonntag
Dipl.-Ing. (FH) Lukas Sühnel

1.3 Using the Manual

General information on the RFEM program is given in the main RFEM 5 Manual and, therefore, is not included in this additional documentation. The main purpose of this manual is to focus on the specific features of the various types of orthotropic surfaces.

The text of the manual shows the described **buttons** in square brackets, for example [Apply]. At the same time, they are shown in the left margin. In addition, **expressions** used in dialog boxes, tables, and menus are set in *italics* to clarify the explanations.

The index at the end of the manual helps you to find specific terms and subjects. However, if you do not 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 Using Orthotropic Surfaces in RFEM

The following description provides the user with a brief explanation on how to run RFEM 5 orthotropic surfaces within the interface of the program. At this point we expect that the user should have already familiarized himself with the definition type surface, its properties and general capabilities which are examined in detail with several accompanying examples in the main RFEM 5 Manual. We consider this step important at this stage, as it would give the user a general idea as to which part of the program he is in and how easy it is to operate with this new feature. We would later present a complete step-by-step example, as well as make several references to tabs and menu icons throughout the process of examining each orthotropic surface in the proceeding sections.

We assume that the user is already acquainted with setting up a general planar or quadrangular surface in RFEM (please refer to the main RFEM 5 Manual). The one illustrated below, in Figure 1.1 is a default planar surface of constant thickness and user-defined material. For the time being, we assume that an isotropic material model is applicable to all surface types and geometries (planar or quadrangular).

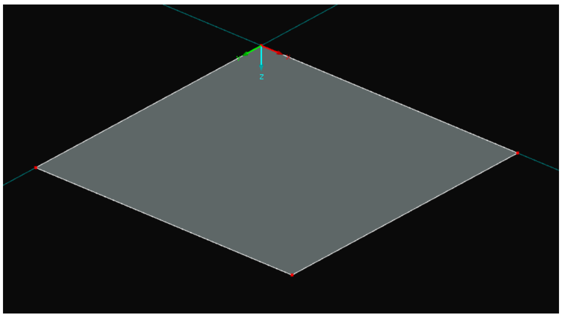


Figure 1.1: View of a default surface of geometry type plane

When done, the created planar surface is automatically reflected in Table 1.4 entitled *Surfaces*—which is a default RFEM object and should be visible to the user without any special settings, see Figure 1.2 below.

1.4 Surfaces

Surface No.	A Surface Type Geometry	B Surface Type Stiffness	C Boundary Lines No.	D Material No.	E Thickness Type	F Thickness d [mm]	G Eccentricity e_z [mm]
1	Plane	Standard	1-4	1	Constant	180.0	0.0
2							

Figure 1.2: RFEM Table 1.4 regarding surface properties

It is very important to mention at this point that orthotropy of a surface is a feature which is attributed to both its material and geometrical characteristics, and these are the main factors by which we will differentiate between various orthotropic surfaces later on in this manual. The two terms combine most conveniently in the definition of *Stiffness*. Therefore, RFEM 5 uses the definition stiffness to distinguish between various types of surfaces as being *Orthotropic*, *Laminate*, *Glass*, etc., whereas the term geometry remains strictly reserved for the global spacial classification of a surface—plane, spline, pipe, etc.—based on the trajectory the surface traced in the Euclidian coordinate system. In that sense, a surface of geometry pipe would represent a curvilinear closed spacious "pipe" surface, whereas changing its stiffness to orthotropic would make that surface obtain orthotropic properties on the local material and geometrical level.

1.4.1 Accessing Dialog Box to Define Orthotropic Surface

Having defined a simple planar (or quadrangular) surface geometry and assigned some material model to it (please, refer to the main RFEM 5 Manual), changing its stiffness to orthotropic and accessing the main orthotropic surface dialog is a straightforward task. There are three different ways to do that.

Access via double-clicking the selected surface in the GUI

Double clicking on the surface, the *Edit Surface* dialog box should automatically open, see Figure 1.3. In the stiffness combo box on the right, selecting the *Orthotropic* label and clicking on the button right next to it (boxed in red below), the main orthotropic dialog *Edit Surface Stiffness – Orthotropic* (see Figure 1.6, page 6) will appear.

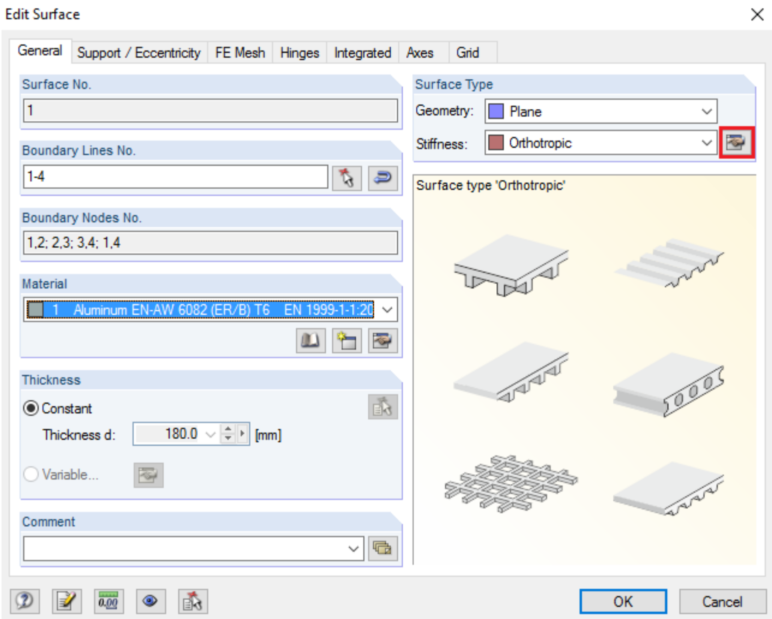


Figure 1.3: *Edit Surface* dialog box

Access via Table 1.4 *Surfaces*

Here, one could easily set the current plate, designated by a number, to an orthotropic stiffness type by executing a single left click on the mask combo box under *Surface Type* and selecting *Orthotropic*. Finally, clicking on the button positioned right next to it, boxed in red in Figure 1.4 below, takes us to the *Edit Surface – Orthotropic* dialog box (see Figure 1.6, page 6).

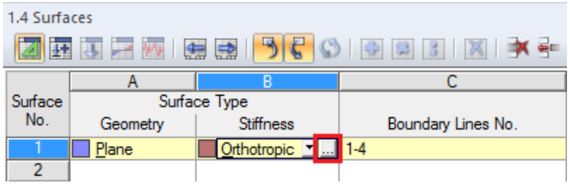


Figure 1.4: Table 1.4 *Surfaces*

Access via the *Edit* menu in Table 1.4 *Surfaces*

Having defined the surface stiffness as orthotropic in Table 1.4 *Surfaces* (see above), right-clicking on the number of the desired surface in Table 1.4 and then choosing *Edit via Dialog Box* (boxed in red in Figure 1.5 below) also results in displaying the *Edit Surface – Orthotropic* dialog box (see Figure 1.6, page 6).

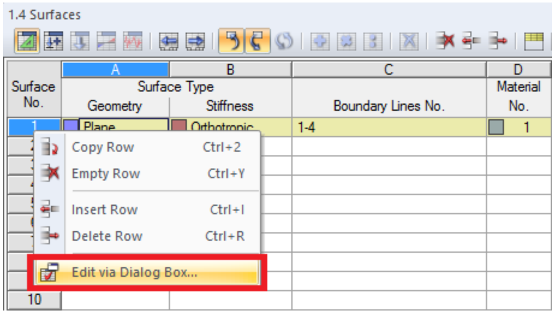


Figure 1.5: *Edit via Dialog Box* command

1.4.2 Structure of Dialog Box *Edit Surface - Orthotropic*

This is the main dialog where all orthotropic surfaces available in this module shall be generated, modified and reviewed. This dialog and all its included features shall be the focal point of our ongoing discussion. There are a few key elements regarding this dialog which deserve to be pointed out to the attention of the user prior to their individual analysis. Each of them is boxed and given a number in Figure 1.6 below, followed by a brief description.

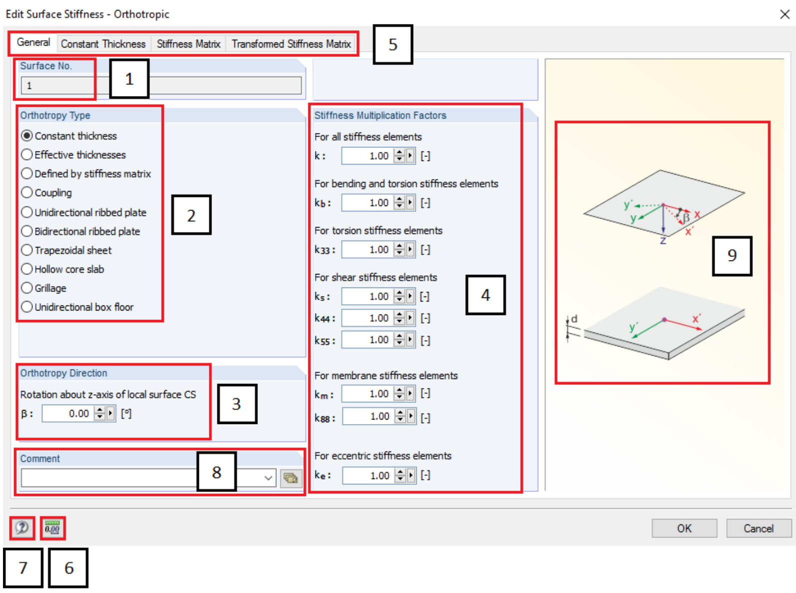


Figure 1.6: *Edit Surface Stiffness – Orthotropic* dialog box, *General* tab

1 • Number of the surface or a series of surfaces selected by the user which are about to be assigned the orthotropic characteristics. This is a read-in window, so no changes could be done straight via it, but rather through surface selection procedures outside of the dialog, described in the main RFEM 5 Manual. The surface numbers coincide with the enumeration structure visible in Table 1.4, accessible to the user via the main program window and GUI. It is only those surfaces that are selected and reflected in the read-in window to whom the current dialog settings would introduce changes to.

2 • A list of altogether nine available RFEM 5 orthotropic surface types to choose from via a radio button. Having chosen one, all enumerated surfaces from Point 1 shall adopt the prescribed

characteristics and become assigned to that chosen type. An image of each surface type is provided on the right along with its geometrical dimensions and further notations necessary for its understanding.

3 • A user-input window for the angle of orthotropy which is globally set in the global coordinate system for and applicable to all surface types and surface numbers mentioned in the previous two points.

4 • Various user definable modification factors, applicable to generated or manually input by the user stiffness matrix coefficients (and groups), belonging to a chosen plate type from Point 2 and applicable to all surfaces from Point 1.

5 • Tabs, entitled from left to right:

  — **General**: when pressed, the items shown on Figure 1.6—and referred to as Points 1, 2, 3, 4, 6, 7, 8 above—become visible. As the name itself suggests, general (or global) features and properties common to a surface or a group of surfaces are being set up here. We will explore each of these items in the foregoing chapters.

  — **Orthotropy Type**: Having already chosen a particular surface type and having set up its global parameters under *General*, this tab enables the user to actually input individually the local surface geometry, review the surface material properties and review or redefine the equivalent surface thicknesses that have initially been computed by the dialog (via 2 options). For more information on the latter, the user could refer to a specially dedicated section in the manual. In the forgoing chapters we will look at each orthotropy type, its geometrical configuration and specific parameters.

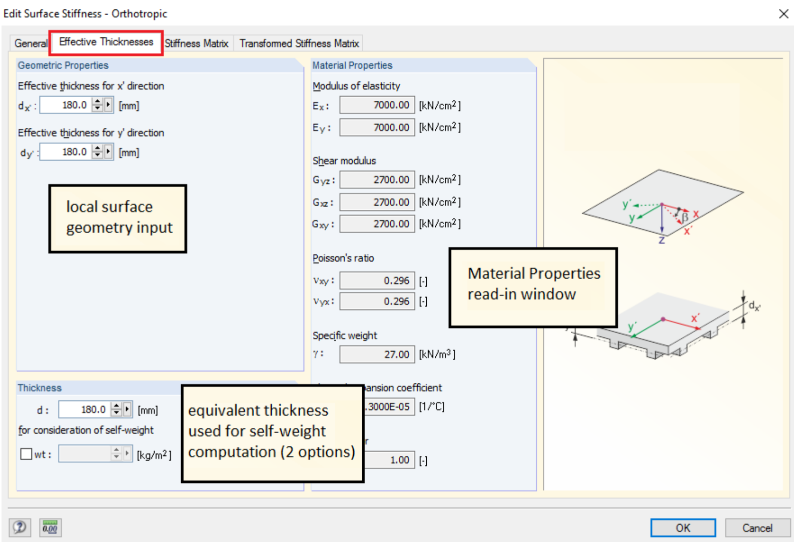


Figure 1.7: Tab of the selected orthotropy type (here: *Effective Thickness*)

  — **Stiffness Matrix**: Having defined a surface type and set up its global and local characteristics under *General* and *Orthotropy Type*—the first two tabs, under *Stiffness Matrix* one could view the already generated stiffness matrix coefficients and modify them if necessary upon changing the initially defined orthotropy model to the *Defined by stiffness matrix* one. This option will be explained later in greater details.

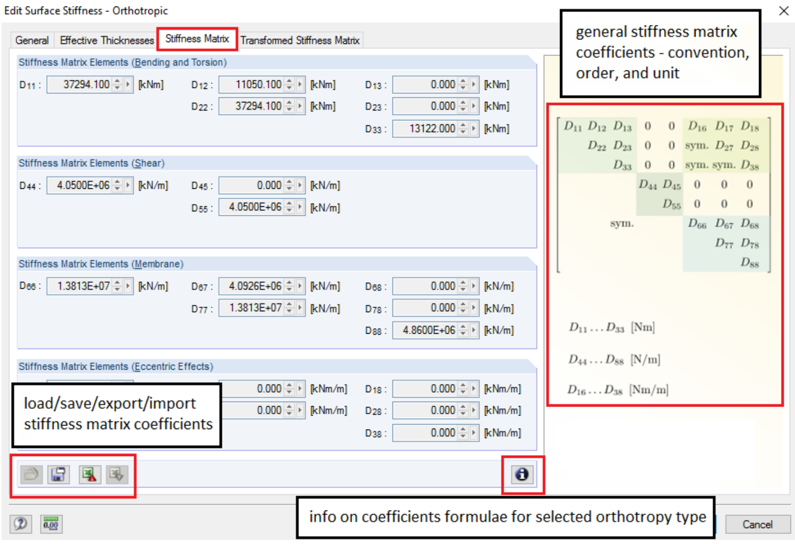


Figure 1.8: Tab *Stiffness Matrix*

— **Transformed Stiffness Matrix**: under this tab one can view the already modified stiffness matrix coefficients as a result of any applied modification factors. This option will be explained later in greater details.

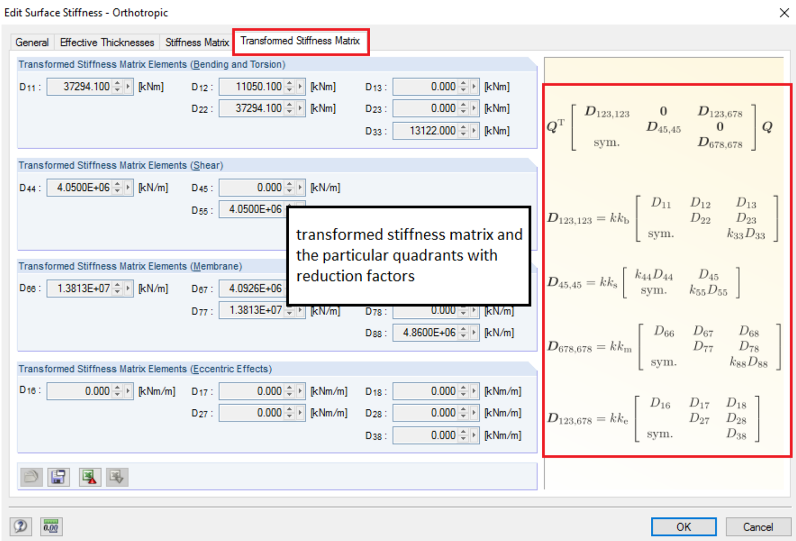


Figure 1.9: Tab *Transformed Stiffness Matrix*

6 • RFEM 5 orthotropic surfaces use the units and decimal places globally set for RFEM. Changing all units and decimal places can be done via the main RFEM menu, but also easily achieved via the *Edit Surface Stiffness – Orthotropic* dialog upon clicking on the button circled in red in the bottom left corner of the dialog. Any changes to the global RFEM units and decimal places shall be automatically reflected for all orthotropic surfaces (for more information, please refer to the main RFEM 5 Manual).

7 • A button that calls RFEM Topic Help (for more information, please refer to the main RFEM 5 Manual).

8 • A general user input comment (for more information, please refer to the main RFEM 5 Manual).

9 • An image of the selected orthotropy type from Point 2 showing its geometry and geometrical parameters.

2 Orthotropy – Definitions

2.1 Nomenclature

Symbol	Description	Unit
E	Young's modulus for isotropic material	[Pa]
E_x	Young's modulus in x-direction	[Pa]
E_y	Young's modulus in y-direction	[Pa]
E_z	Young's modulus in z-direction	[Pa]
E_p	strain-hardening modulus	[Pa]
G	shear modulus for isotropic material	[Pa]
G_{yz}	shear modulus in yz plane	[Pa]
G_{xz}	shear modulus in xz plane	[Pa]
G_{xy}	shear modulus in xy plane	[Pa]
ν	Poission's ratio for isotropic material	[-]
ν_{xy}, ν_{yx}	Poission's ratio in xy plate	[-]
α_T	coefficient of thermal expansion	[1/K]
γ	material self-weight	[N/m^3]
w_t	angle of orthotropy	[°]

Table 2.1: Nomenclature

2.2 Orthotropy in 2D

2.2.1 Pure Material Orthotropy

An orthotropic elastic material is defined as a material which has clearly distinguishable directional elastic properties. In the 2D case, it is defined by its two Young moduli E_x, E_y in the two mutually orthogonal directions x, y respectively, three shear moduli G_{yz}, G_{xz}, G_{xy} corresponding to the planes yz, xz, xy, and two Poisson coefficients defined as ν_{xy}, ν_{yx} defined in the planes xy and yx. The final two material parameters are its specific weight γ and coefficient of thermal expansion α.

$$\begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{xz} \end{bmatrix} = \begin{bmatrix} 1/E_x & -\nu_{xy}/E_x & & & \\ -\nu_{yx}/E_y & 1/E_y & & & \\ & & 1/G_{xy} & & \\ & & & 1/G_{yz} & \\ & & & & 1/G_{xy} \end{bmatrix} \begin{bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{xz} \end{bmatrix} \tag{2.1}$$

In total have seven material parameters and one condition defining the full elastic matrix symmetry via the following relationship between major ν_{xy} and minor ν_{yx} Poisson ratii

$$\frac{\nu_{yx}}{E_y} = \frac{\nu_{xy}}{E_x} \tag{2.2}$$

The latter can be understood as a definition of the minor Poisson's ratio

$$\nu_{yx} = \nu_{xy}\frac{E_y}{E_x} \tag{2.3}$$

Thus, in general we have six independent parameters defining the material in 2D

$$E_x, \quad E_y, \quad \nu_{xy}, \quad G_{yz}, \quad G_{xz}, \quad G_{xy} \tag{2.4}$$

Material restrictions

In order to yield a stable, positive definite stiffness matrix, the following relationships need to be satisfied (positive definiteness of the compliance matrix is equivalent to positive definiteness of the stiffness matrix)

$$E_x > 0, \quad E_y > 0, \quad G_{yz} > 0, \quad G_{xz} > 0, \quad G_{xy} > 0$$

$$|\nu_{xy}| \leq 0.999\sqrt{\frac{E_x}{E_y}} \tag{2.5}$$

which is the result of positive definiteness constraint on the compliance matrix.

Any user-defined *Orthotropic Elastic 2D* material to be assigned to a surface should necessarily satisfy these two conditions (2.5) and if that is not the case, an error is thrown upon data input entry in the **Edit material** → **Material Model** → **Orthotropic Elastic 2D** menu shown in Figure 2.1 below. For more information regarding restrictions, material model specifities, accessing the *Edit Material* dialog box and assigning an *Orthotropic Elastic 2D* material to a general surface, please refer to the main RFEM 5 Manual.

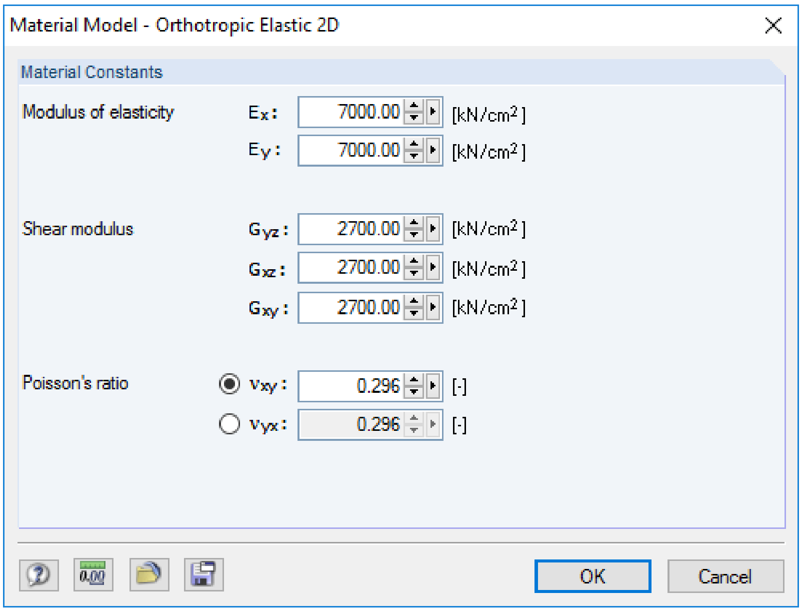


Figure 2.1: *Orthotropic Elastic 2D* material

Examples of pure material orthotropy

Pure material orthotropy is only present in homogenous, uniform thickness surfaces the material of which exhibits orthotropic features (the material model assigned to the surface must be *Orthotropic Elastic 2D*). An example for such a plate is illustrated below

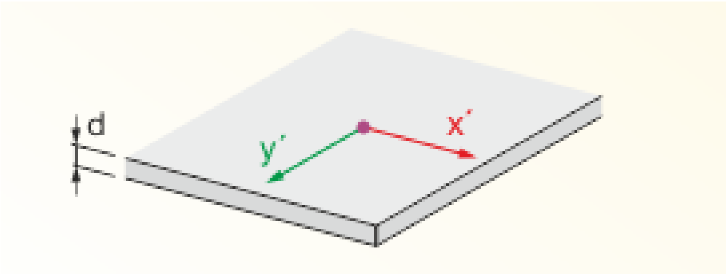


Figure 2.2: Homogeneous orthotropic plate of constant thickness

2.2.2 Pure Material Isotropy as Special Case of Pure Material Orthotropy

When isotropy is present under the same assumptions, the material is said to have the same elastic characteristics in all three dimensions, suggesting that

$$\begin{aligned} E_x &= E_y = E \\ \nu_{xy} &= \nu_{yx} = \nu \\ G_{yz} &= G_{xz} = G_{xy} = G \end{aligned} \tag{2.6}$$

where E, G, ν represent the isotropic Young's modulus, shear modulus, and Poisson's ratio of the material. Moreover, the following well-known relation from the theory of elasticity between E, G, ν holds

$$G = \frac{E}{2(1+\nu)} \tag{2.7}$$

In this case, the fully symmetric compliance matrix reduces simply to

$$\begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{xz} \end{bmatrix} = \begin{bmatrix} 1/E & -\nu/E & & & \\ -\nu/E & 1/E & & & \\ & & 1/G & & \\ & & & 1/G & \\ & & & & 1/G \end{bmatrix} \begin{bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{xz} \end{bmatrix} \tag{2.8}$$

Material Restrictions

In order to yield a stable, positive definite matrix, the following relations must apply

$$E > 0, \quad G > 0, \quad \text{and} \quad -0.999 \leq \nu < 0.5 \quad \text{(for 2D)} \tag{2.9}$$

as a result of the postive definiteness of the compliance matrix and a compressible material. Any user defined *Isotropic Linear Elastic* material which is to be assigned to a surface should necessarily satisfy the conditions (2.9) and if that is not the case, an error is thrown upon the entry data input in the **Edit material** → **Material Model** → **Isotropic Linear Elastic** menu, shown in Figure 2.3 below. For more information regarding restrictions, material model specifities, accessing the *Edit Material* dialog box and assigning an *Isotropic Linear Elastic* material to a general surface, please refer to the main RFEM 5 Manual.

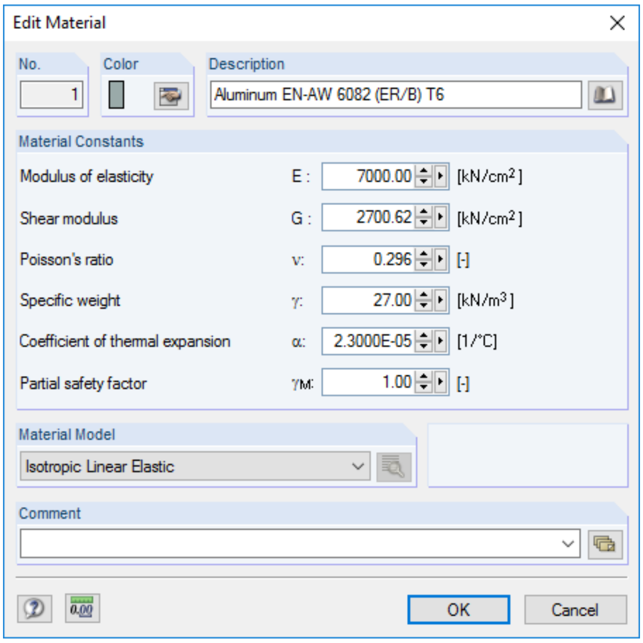


Figure 2.3: *Isotropic Linear Elastic* material model

Examples of pure material isotropy

Pure material isotropy is only present in homogeneous, uniform thickness surfaces whose material reveals isotropic features (the material model assigned to the surface must be *Isotropic Linear Elastic*). An example for such a plate is illustrated below

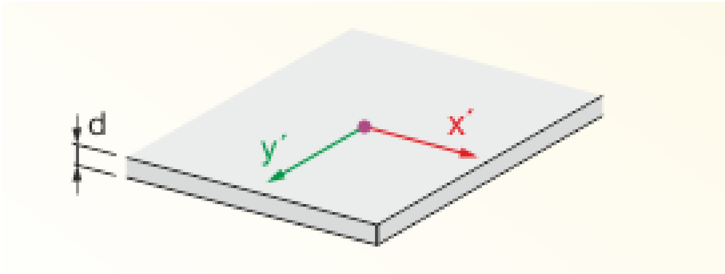
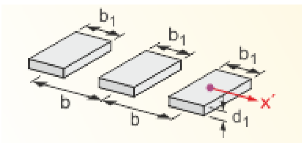


Figure 2.4: Homogeneous isotropic plate of constant thickness

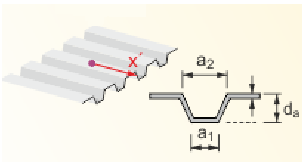
2.2.3 Pure Geometrical Orthotropy in 2D

Pure geometrical orthotropy is a special type of orthotropy, exhibited by surfaces the material properties of which remain the same in all directions, but geometrically their configuration is such that in at least two non-coinciding plane directions have different stiffness properties.

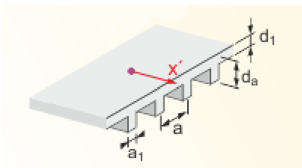
Typical examples of pure geometrical orthotropy



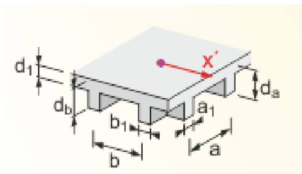
Coupling



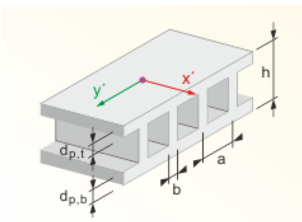
Trapezoidal sheet



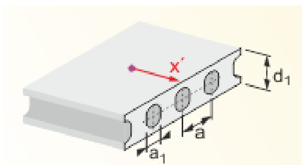
Unidirectional ribbed plate



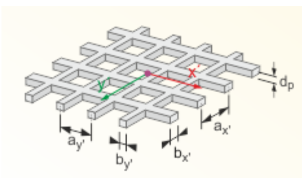
Bidirectional ribbed plate



Unidirectional boxed floor



Hollow core slab



Grillage

Figure 2.5: Examples of geometrically orthotropic surfaces

In all these examples, the material remains isotropic and homogeneous in all planes and direction, whereas the geometry defines two strictly independent – in terms of behavior, orthogonal – stiffness directions x′ and y′.

Bearing in mind that the material contribution to the overall stiffness is equal in all seven cases above, it is the geometrical configuration that distinguishes between pure material and pure geometrical orthotropy. Note that in some exceptional cases, as for the bidirectionally ribbed plate in Figure ?? when the ribs are equally spaced and of equal geometry, the geometrical stiffness might be the same in both x′ and y′ directions.

2.2.4 Combined Geometrical and Material Orthotropy in 2D

Combined geometrical and material orthotropy can be often found in geometrically irregular plates like the ones shown in Section ??, made out of purely orthotropic material. Figure 2.6 below shows an example of a ribbed plate in the *x* direction, thus geometrical orthotropy is already present. If the material itself is 2D orthotropic, as indicated below, then the surface is automatically considered as a combined model of geometrical and material orthotropy. Quite often in practice the material orthotropy is due to the fact that each of the two orthogonal directions receives a different amount of reinforcement, even though the surface material itself might be of isotropic nature, somewhat like non-reinforced concrete.

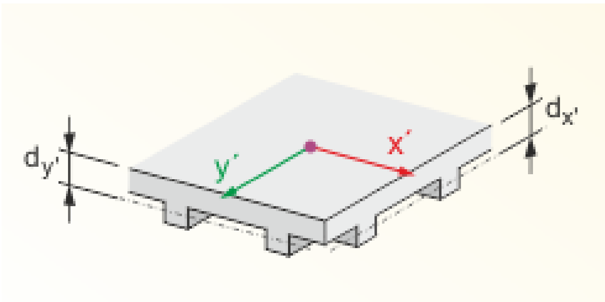


Figure 2.6: *Effective thickness* model – exhibiting both geometrical and material orthotropy

A similar model exists in RFEM, which is termed *Effective thickness*. We will discuss that in greater details in Chapter ??, but for the time being it is worth mentioning that RFEM 5 does indeed possess a combined model allowing the modeling of both geometrically and materially orthotropic surfaces, as well as the possibility to provide every user with completely user-definable stiffness matrix coefficients. The latter is the most general model, termed *Defined by stiffness matrix* in RFEM 5 and it could represent any possible type of orthotropy, provided the stiffness coefficients are known a-priori by the engineer. We shall also discuss its properties further in Chapter ??.

$$\boldsymbol{D} = \begin{bmatrix} D_{11} & D_{12} & \\ & D_{22} & \\ \text{sym.} & & \ddots \end{bmatrix}$$

Figure 2.7: User-defined stiffness matrix approach to model general orthotropic surfaces

2.2.5 Common Features of All Orthotropic Surfaces in RFEM

Stiffness Matrix Coefficients

Regardless of the orthotropy type of a surface, all orthotropic surfaces in RFEM 4 and 5 are always described by the generalized stiffness matrix which is subsequently used in FE formulations in the following way

$$\begin{bmatrix} m_x \\ m_y \\ m_{xy} \\ v_x \\ v_y \\ n_x \\ n_y \\ n_{xy} \end{bmatrix} = \begin{bmatrix} D_{11} & D_{12} & D_{13} & 0 & 0 & D_{16} & D_{17} & D_{18} \\ & D_{22} & D_{23} & 0 & 0 & D_{26} & D_{27} & D_{38} \\ & & D_{33} & 0 & 0 & D_{36} & D_{37} & D_{38} \\ & & & D_{44} & 0 & 0 & 0 & 0 \\ & & & & D_{55} & 0 & 0 & 0 \\ & & \text{sym} & & & D_{66} & D_{67} & D_{68} \\ & & & & & & D_{77} & D_{78} \\ & & & & & & & D_{88} \end{bmatrix} \begin{bmatrix} \kappa_x \\ \kappa_y \\ \kappa_{xy} \\ \gamma_{xz} \\ \gamma_{yz} \\ \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{bmatrix} \tag{2.10}$$

and

$$\begin{aligned} \kappa_x &= \frac{\partial \varphi_y}{\partial x}, & \kappa_y &= \frac{\partial \varphi_x}{\partial y}, & \kappa_{xy} &= \frac{\partial \varphi_y}{\partial y} - \frac{\partial \varphi_x}{\partial x}, \\ \gamma_{xz} &= \frac{\partial w}{\partial x} + \varphi_y, & \gamma_{yz} &= \frac{\partial w}{\partial y} - \varphi_x, & & \\ \varepsilon_x &= \frac{\partial u}{\partial x}, & \varepsilon_y &= \frac{\partial v}{\partial y}, & \gamma_{xy} &= \frac{\partial u}{\partial y} - \frac{\partial v}{\partial x} \end{aligned} \tag{2.11}$$

where the left-hand side vector represents all internal FE nodal forces and the right-hand side one the generalized strains at all nodal points, here formulated only for thick Mindlin plate elements. Both are coupled through the 8×8 stiffness matrix $\boldsymbol{D}$, which is fully symmetric about its main diagonal.

The following figure shows the general stiffness matrix of an orthotropic surface in RFEM.

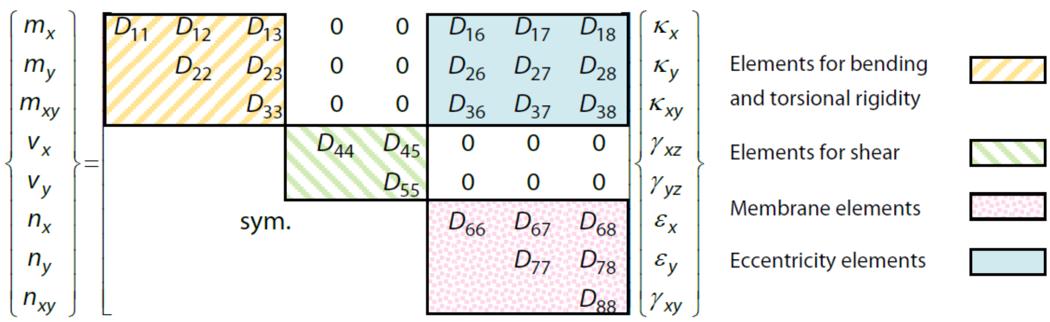


Figure 2.8: Stiffness matrix exhibited by any RFEM 5 surface element

The corresponding bending, shear, membrane and eccentric terms are indicated by different colors in Figure 2.8 above. One might also notice that the eccentric terms quadrant (the upper right 3×3 matrix) is always zero, and RFEM 5 indeed nullifies this block for all surface types. Another observation is the complete matrix symmetry. Therefore, on the basis of computational efficiency, only diagonal and upper diagonal terms are saved and if any modifications are applied to $\boldsymbol{D}$ by the user, those will only be assigned to the upper triangle of the matrix, since the lower one is a direct result of matrix symmetry.

Stiffness Matrix Units

In SI units system, the dimensions of the stiffness coefficients are given as follows.

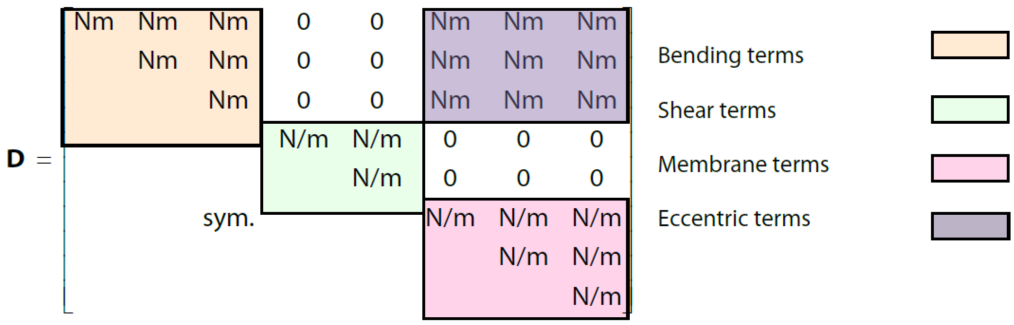


Figure 2.9: Stiffness matrix coefficients units

The dimensions of those are by default in SI units, but could be easily switched to any user–preferred units via the [Units and Decimal Places] button under *Edit Surface* dialog and then choosing RFEM as the main program module.

Orthotropy Direction and Angle of Orthotropy

The orthotropy direction and its inferred angle of orthotropy are highly important features of all orthotropic surfaces. The setup box for them is loacated in the bottom left corner of the *Edit Surface – Orthotropic* dialog box, shown in Figure 2.10.

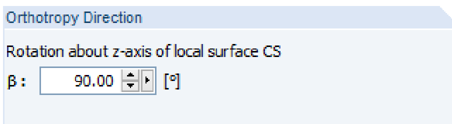


Figure 2.10: Dialog section *Orthotropy Direction*

Since orthotropy – either material, geometric, or combined – is always directional, the orthotropy directions must be clearly defined prior to the analysis. We already explained that orthoropy is usually referred to in its local xy coordinate system, which is in all RFEM orthotropic surfaces by default set to the global xy plane of the Euclidean coordinate system. This can be seen below in Figure 2.11 where a simple rectangular planar surface has just been defined and its local axes automatically orientated by RFEM to coincide with the global ones. In many practical cases the local orthotropy directions of the plate will coincide with the globally defined x and y axis of the model. in these cases the angle of orthotropy is said to be equal to zero degrees. In RFEM this is the default value for β, the latter also denoted as the *Rotation about z axis of local surface CS*.

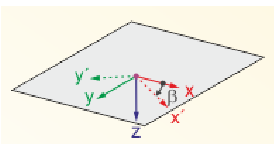


Figure 2.11: Angle of *Orthotropy Direction*

Quite often in practice, however, the orthotropic surface needs to be rotated or positioned at an angle different from zero degrees to the main global axis of the model. In this particular case, a new set of rotated local axis needs to be defined, entitled x′y′ rotated by an angle β around the z axis with respect to the global xy system. The purpose of such an action is to obtain the orthotropic stiffness contribution in those two global directions along which the entire structural model is about to be analyzed. The clockwise rotation of the local surface axes is indicated above in Figure 2.11, and a clockwise rotation is by convention positive. The value of β is user-defined.

In order to graphically review the rotated local x′y′ coordinate system, the main *Surfaces* checkbox needs to be activated (in the *Project Navigator* list) and then additionally the *Orthotropic directions* one, as shown on the left in Figure 2.12. This would graphically draw the new axis onto the surface and become visible upon selecting it or simply running with the mouse cursor over it (shown on the right in Figure 2.12).

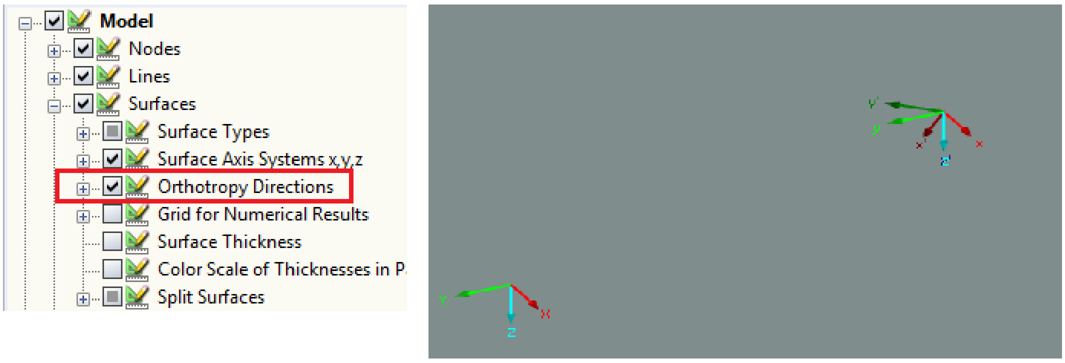


Figure 2.12: Displaying *Orthotropy Directions*

There are cases when the local coordinate system of a surface is not displayed and needs to be switched on. In this case, the user should apply a left button mouse click on the surface and check the option *Local Axis Systems on/off*. This would make the local coordinate system visible.

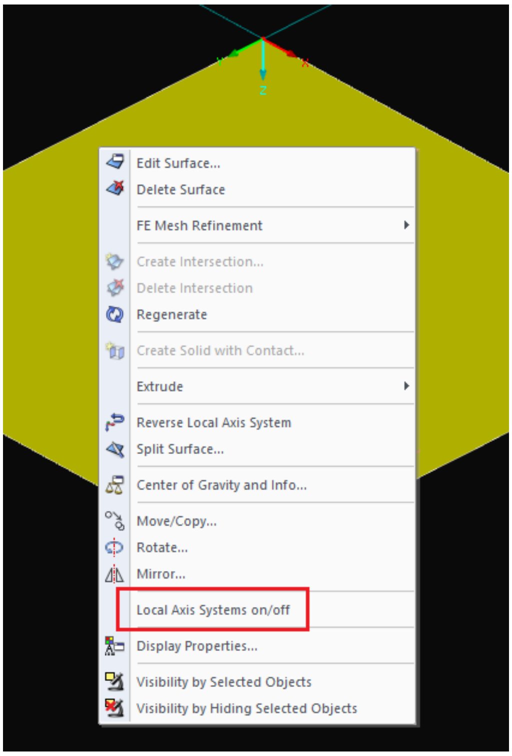


Figure 2.13: Switching the visibility of the local coordinate system

If there is more than one plate selected and on display in the *Edit - Surface Orthotropic* dialog under *Surface No.*, then all of these enumerated plates shall be modified equally by the specified β value.

Default value for β: $\beta = 0°$.

Restrictions on β: $-360° \leq \beta \leq 360°$.

Applicability: applicable to all surface types without exception.

Stiffness Multiplication Factors

In RFEM 5 stiffness reduction factors can be applied to various stiffness matrix terms. The modifiable stiffness matrix terms are D_{11}, D_{12}, D_{22}, D_{33} (bending terms), D_{44}, D_{45}, D_{55} (shear terms), D_{66}, D_{67}, D_{77}, D_{88} (membrane terms), D_{16}, D_{17}, D_{27}, D_{38} (eccentric terms), and their corresponding symmetric lower-diagonal terms, respectively. All stiffness reduction factors are to be found in the *Edit Surface – Orthotropic* dialog under the *General* tab. There are several possibilities to modify individual or group values.

- *Apply the same reduction factor to all matrix coefficients*

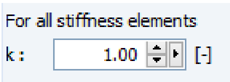


Figure 2.14: Modification factor for all stiffness elements

A single value must be defined by the user which consequently multiplies all stiffness matrix terms in the following manner

$$\boldsymbol{D}_K = \begin{bmatrix} kD_{11} & kD_{12} & & & & kD_{16} & kD_{17} & \\ & kD_{22} & & & & \text{sym} & kD_{27} & \\ & & kD_{33} & & & & & kD_{38} \\ & & & kD_{44} & kD_{45} & & & \\ & & & & kD_{55} & & & \\ & & \text{sym} & & & kD_{66} & kD_{67} & \\ & & & & & & kD_{77} & \\ & & & & & & & kD_{88} \end{bmatrix} \tag{2.12}$$

- *Modify the bending and torsional stiffness coefficients*

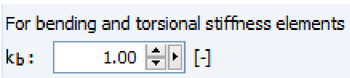


Figure 2.15: Modification factor for bending and torsional stiffness elements

A single value must be defined by the user which consequently multiplies all bending stiffness matrix terms in the following manner

$$\boldsymbol{D}_K = \begin{bmatrix} k_bD_{11} & k_bD_{12} & & & & D_{16} & D_{17} & \\ & k_bD_{22} & & & & \text{sym} & D_{27} & \\ & & k_bD_{33} & & & & & D_{38} \\ & & & D_{44} & D_{45} & & & \\ & & & & D_{55} & & & \\ & & \text{sym} & & & D_{66} & D_{67} & \\ & & & & & & D_{77} & \\ & & & & & & & D_{88} \end{bmatrix} \tag{2.13}$$

- *Modify the torsional stiffness coefficient*

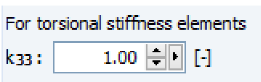


Figure 2.16: Modification factor for torsional stiffness element

A single value k_{33} must be defined by the user which consequently multiplies the tosional stiffness matrix term D_{33} in the following manner

$$\boldsymbol{D}_K = \begin{bmatrix} D_{11} & D_{12} & & & & D_{16} & D_{17} & \\ & D_{22} & & & & \text{sym} & D_{27} & \\ & & k_{33}D_{33} & & & & & D_{38} \\ & & & D_{44} & D_{45} & & & \\ & & & & D_{55} & & & \\ & & \text{sym} & & & D_{66} & D_{67} & \\ & & & & & & D_{77} & \\ & & & & & & & D_{88} \end{bmatrix} \qquad (2.14)$$

- *Modify the shear stiffness coefficients*

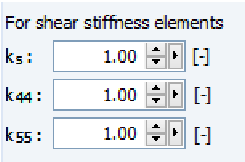


Figure 2.17: Modification factor for shear stiffness elements

A single value k_s must be defined by the user which consequently multiplies all shear stiffness matrix terms in the following manner

$$\boldsymbol{D}_K = \begin{bmatrix} D_{11} & D_{12} & & & & D_{16} & D_{17} & \\ & D_{22} & & & & \text{sym} & D_{27} & \\ & & D_{33} & & & & & D_{38} \\ & & & k_s k_{44} D_{44} & k_s D_{45} & & & \\ & & & & k_s k_{55} D_{55} & & & \\ & & \text{sym} & & & D_{66} & D_{67} & \\ & & & & & & D_{77} & \\ & & & & & & & D_{88} \end{bmatrix} \qquad (2.15)$$

Two additional values k_{44} and k_{55} are provided to modify the leading terms.

- *Modify the membrane stiffness coefficients*

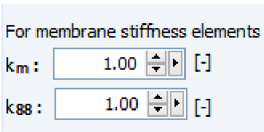


Figure 2.18: Modification factor for membrane stiffness elements

A single value k_m must be defined by the user which consequently multiplies all membrane stiffness matrix terms in the following manner

$$\boldsymbol{D}_K = \begin{bmatrix} D_{11} & D_{12} & & & & D_{16} & D_{17} & \\ & D_{22} & & & & \text{sym} & D_{27} & \\ & & D_{33} & & & & & D_{38} \\ & & & D_{44} & D_{45} & & & \\ & & & & D_{55} & & & \\ & & \text{sym} & & & k_m D_{66} & k_m D_{67} & \\ & & & & & & k_m D_{77} & \\ & & & & & & & k_m k_{88} D_{88} \end{bmatrix} \qquad (2.16)$$

An additional value k_{88} is provided to modify the coefficient D_{88}.

- *Modify the eccentric stiffness coefficients*

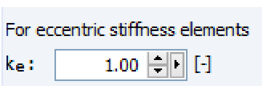


Figure 2.19: Modification factor for eccentric stiffness elements

A single value k_e must be defined by the user which consequently multiplies all eccentric stiffness matrix terms in the following manner

$$\boldsymbol{D}_K = \begin{bmatrix} D_{11} & D_{12} & & & & k_e D_{16} & k_e D_{17} & \\ & D_{22} & & & & \text{sym} & k_e D_{27} & \\ & & D_{33} & & & & & k_e D_{38} \\ & & & D_{44} & D_{45} & & & \\ & & & & D_{55} & & & \\ & & \text{sym} & & & D_{66} & D_{67} & \\ & & & & & & D_{77} & \\ & & & & & & & D_{88} \end{bmatrix} \tag{2.17}$$

Only diagonal and upper diagonal terms of the stiffness matrix are modified and automatically being copied over to the lower triangular ones due to computational efficiency. Therefore, symmetric lower diagonal entries need not be defined explicitely thanks to the symmetricity of the stiffness matrix.

The user could also apply any combination of the above mentioned modifications at the same time simply upon setting the corresponding dialog entries to a value different from 1 which is the default. If there is more than one plate selected and on display in the *Edit - Surface Orthotropic* dialog under *Surface No.*, then all of these enumerated plates shall be affected by the simultaneous action of all modifying factors.

Another way to modify a given number of stiffness matrix coefficients is by going to the *Defined by stiffness matrix* tab in the main *Edit Surface – Orthotropic* dialog, and revising the RFEM already generated stiffness coefficient values. A prerequisite is that the user should have already created a type of orthotropic surface and input its global charactersistics and local geometry, as mentioned with regards to tabs *General* and selected *Orthotropy Type* in Chapter 1. When a surface is generated and its coefficients computed, they become immediately available for various modifications under the tab *Stiffness Matrix* in the main *Edit Surface – Orthotropic* dialog. This concept shall be clearly illustrated in Chapter 6 where we present a complete example regarding techniques such as modeling, analysis and modifications.

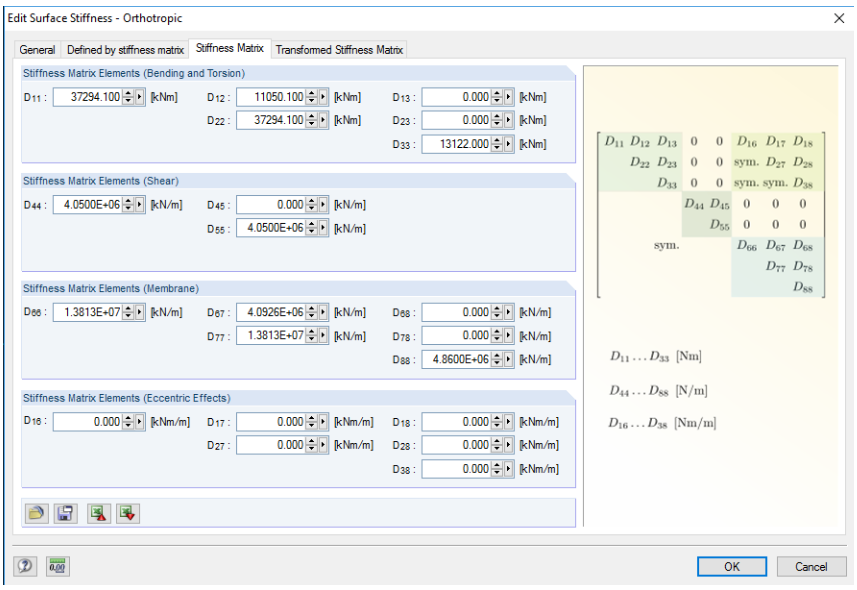


Figure 2.20: *Stiffness matrix* tab

Default value for k_i: $k_i = 1$.

Applicability: applicable to all surface types without restriction.

Symmetry: symmetry is preserved, since $\boldsymbol{D}_K^\top = \boldsymbol{D}_K$.

Stiffness Matrix – Orthogonal Rotation

Before the final stiffness matrix of an element is passed onto the solver, an orthogonal transformation needs to take place as to transform the initial orthotropic xy plane into the new x′y′ plane, rotated by the angle β around the z-axis.

The performed transformation $\boldsymbol{D}_{\mathrm{K}} \rightarrow \boldsymbol{D}_{\mathrm{K}\beta}$ is given by

$$\boldsymbol{D}_{K\beta} = \tilde{\boldsymbol{Q}}^{\top} \boldsymbol{D}_{K} \tilde{\boldsymbol{Q}} \tag{2.18}$$

Where $\tilde{\boldsymbol{Q}}$ is a global transformation matrix containing the sines and cosines of the rotation angle β. Due to the large number of zeros and for the sake of computational efficiency, the above transfromation is carried out in RFEM in a blockwise manner in such a way that each of the bending, shear and membrane stiffness quadrants is transformed separately and then assembled back into a global 8×8 stiffness matrix. The process is described below.

- Bending stiffness transformation

$$\boldsymbol{D}_{K\beta_{1..3,1..3}} = \boldsymbol{Q}_{3,3}^{\top} \boldsymbol{D}_{K_{1..3,1..3}} \boldsymbol{Q}_{3,3} \tag{2.19}$$

- Shear stiffness transformation

$$\boldsymbol{D}_{K\beta_{4..5,4..5}} = \boldsymbol{Q}_{2,2}^{\top} \boldsymbol{D}_{K_{4..5,4..5}} \boldsymbol{Q}_{2,2} \tag{2.20}$$

- Membrane stiffness transformation

$$\boldsymbol{D}_{K\beta_{6..8,6..8}} = \boldsymbol{Q}_{3,3}^{\top} \boldsymbol{D}_{K_{6..8,6..8}} \boldsymbol{Q}_{3,3} \tag{2.21}$$

- Eccentric stiffness transformation

$$\boldsymbol{D}_{K\beta_{1..3,6..8}} = \boldsymbol{Q}_{3,3}^{\top} \boldsymbol{D}_{K_{1..3,6..8}} \boldsymbol{Q}_{3,3} \tag{2.22}$$

where

$$\boldsymbol{Q}_{2,2} = \begin{bmatrix} \cos\beta & \sin\beta \\ -\sin\beta & \cos\beta \end{bmatrix} \tag{2.23}$$

$$\boldsymbol{Q}_{3,3} = \begin{bmatrix} \cos^2\beta & \sin^2\beta & \sin\beta\cos\beta \\ \sin^2\beta & \cos^2\beta & -\sin\beta\cos\beta \\ -2\sin\beta\cos\beta & 2\sin\beta\cos\beta & \cos^2\beta - \sin^2\beta \end{bmatrix} \tag{2.24}$$

If $\beta = 0$, then $\boldsymbol{D}_{\mathrm{K}\beta} = \boldsymbol{D}_{\mathrm{K}}$, since $\boldsymbol{Q}_{3,3} = \boldsymbol{I}_{3,3}$, $\boldsymbol{Q}_{2,2} = \boldsymbol{I}_{2,2}$. The latter suggests that no effect having transformation between global and orthogonal axis has been performed, resulting in the original xy stiffness being reobtained.

If $\beta \neq 0$, the transformations result in a stiffness matrix with nonzero coupled terms in both lower and upper triangular parts, with some of the entries being negative.

Applicability: applicable to all surface types without exception.

Symmetry: symmetry is preserved, as $\boldsymbol{D}_{\mathrm{K}\beta}^{\top} = \boldsymbol{D}_{\mathrm{K}\beta}$.

Stiffness Matrix – Positive Definiteness

The positive definiteness of the stiffness matrix is a check which ensures that

- the geometrical and material configuration of the orthotropic surface is reasonable and realsitic;
- the givene analysis will be stable and convergent, regardless of the analysis order and the nonlinearities involved.

The stiffness matrix $\boldsymbol{D}$ – generated on the basis of geometry and material model – is in general positive definite. However, upon user modifications and application of various stiffness reduction factors, it might become positive semi-definite, negative semi-negative, or negative definite in the most general case. In fact, only a positive definite matrix makes sense from a structural point of view and should be therefore allowed, whereas all other intermediate cases would immediately result in unstable orthotropic surface model that should in all cases be avoided.

Therefore, RFEM has an integrated preliminary subroutine-check prior to any FE computation, which checks for a positive definite matrix and if that is not the case, the following message is returned

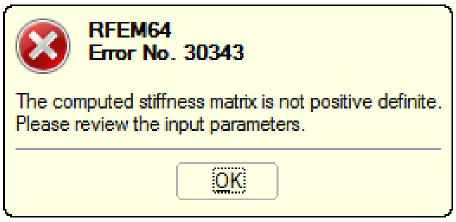


Figure 2.21: Error message in RFEM

Any further process is being put on hold until the user revises the model.

The matrix used for the check is the one obtained in the final modification stage, the sequence of which is given below.

1. *Construction of the matrix **D** based on material and geometrical considerations.*

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

2. *Application of the user-defined stiffness modification factors to obtain the matrix $\boldsymbol{D}_{\text{K}}$.*

$$\boldsymbol{D}_K = \begin{bmatrix} k_b D_{11} & k_b D_{12} & & & & k_e D_{16} & k_e D_{17} & \\ & k_b D_{22} & & & & \text{sym} & k_e D_{27} & \\ & & k_b k_{33} D_{33} & & & & & k_e D_{38} \\ & & & k_s k_{44} D_{44} & k_s D_{45} & & & \\ & & & & k_s k_{55} D_{55} & & & \\ & & \text{sym} & & & k_m D_{66} & k_m D_{67} & \\ & & & & & & k_m D_{77} & \\ & & & & & & & k_m k_{88} D_{88} \end{bmatrix}$$

3. *Application of the rotation by angle β to obtain $\boldsymbol{D}_{\text{K}\beta}$ (the last term utilizing its block form).*

$$\boldsymbol{D}_{K\beta} = \tilde{\boldsymbol{Q}}^{\top} \boldsymbol{D}_K \tilde{\boldsymbol{Q}} = \begin{bmatrix} \boldsymbol{D}_{3\times3}^{\text{bending}} & \boldsymbol{0} & \boldsymbol{D}_{3\times3}^{\text{eccentric}} \\ & \boldsymbol{D}_{2\times2}^{\text{shear}} & \boldsymbol{0} \\ \text{sym} & & \boldsymbol{D}_{3\times3}^{\text{membrane}} \end{bmatrix}$$

4. *Application of a positive definiteness check to $\boldsymbol{D}_{\text{K}\beta}$.*

The positive definiteness check implemented in RFEM 5 is based on the Sylvester criterion suggesting that any matrix is positive definite as long as its leading principal minors are positive. The complete – due to numerical considerations more restrictive – procedure goes as follows.

1. *Positivity check of the submatrix determinants (minors)*

$$\det \left[\boldsymbol{D}_{K\beta}\right]_{i,j=1}^{\ell} > 0, \quad \text{for } \ell = 1, \ldots, 8 \qquad (2.25)$$

2. *Special-condition check for the blocks $\boldsymbol{D}_{3\times3}^{\text{bending}}$, $\boldsymbol{D}_{2\times2}^{\text{shear}}$, $\boldsymbol{D}_{3\times3}^{\text{membrane}}$*

$$\det \begin{bmatrix} D_{11} & D_{12} \\ D_{12} & D_{22} \end{bmatrix} \geq c D_{11} D_{22},$$

$$\det\begin{bmatrix} D_{44} & D_{45} \\ D_{45} & D_{55} \end{bmatrix} \geq cD_{44}D_{55}, \qquad (2.26)$$

$$\det\begin{bmatrix} D_{66} & D_{67} \\ D_{67} & D_{77} \end{bmatrix} \geq cD_{66}D_{77},$$

where $c = 1 - 0.999^2 = 0.001999$.

If even one of these two conditions is not fulfilled, the error message Figure 2.21 is displayed. Computations do not proceed further unless corrected.

Equivalent Thickness for Self-Weight Computation

The equivalent thickness for self-weight computation is the thickness used by RFEM 5 for the self-weight computation of a finite element and evaluating its equivalent nodal forces in the case of it being included as an extra dead load via the load case manager. Self-weight is usually considered in static analysis as a separate gravity load and often needs to be taken into account from practical point of view, as well as in dynamic applications where the distributed mass matrix of an element is to be constructed. The material parameter associated with it is the material specific weight γ [N/m^3]. The latter is entirely a material parameter and can either be manually set by the user, or simply read in from the material library during the definition of the 2D material model.

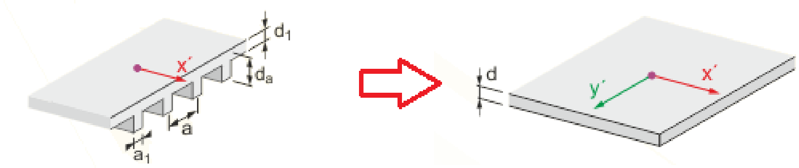


Figure 2.22: Equivalent thickness for self-weight computation

For a constant thickness plate or assembly – consisting of equal and constant thicknesses – the equivalent thickness is indeed the plate thickness d. Due to irregular geometries and non-constant assembly height – as a result of ribs or voids integration, see, e.g., Figure 2.22 – it is required to smear the varying thickness (height) into a single parameter, globally denoted as d. It is then easy to obtain the FE surface weight as

$$W_A = \gamma d \text{ [N/m}^2\text{]} \qquad (2.27)$$

Note that d is used in the finite element formulation as an additional external distributed load onto the element surface area.

The value of d is computed for each plate on the basis of the presumption that the volume occupied by a unit surface area of the considered orthotropic plate should equal the volume occupied by a unit surface area of an equivalent plate of thickness d. As we go through the various types of orthotropic surfaces, we will show how to calculate d for each case via simple formulae.

In RFEM, the equivalent thickness is computed in the read-in window under the tab for the chosen orthotropy type in *Edit Surface Stiffness - Orthotropic* main dialog. The window is located on the bottom left side of the dialog, as shown below.

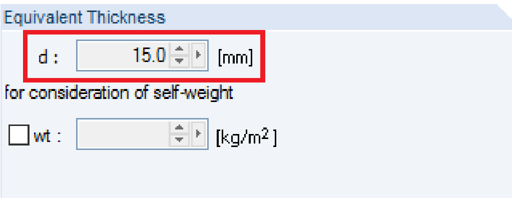


Figure 2.23: Dialog section *Equivalent thickness*

Under the read-in value of d [mm] there is an extra checkbox button wt, which is by default masked and disabled. When enabled, it activates the input field for a user–defined self-weight (in [kg/m2]). The value overwrites the initial equivalent thickness d computed by RFEM.

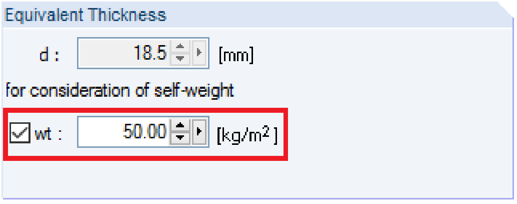


Figure 2.24: Dialog section *Equivalent thickness*

The equivalent thickness is in this case computed as

$$d = \frac{w_t g}{\gamma}, \tag{2.28}$$

where g is the gravitational constant, initially set in RFEM to g $=$ 10 m/s^2.

Both self-weight measures become visible, but if wt is activated, it shall overwrite d which is then displayed in the dialog as read–in value for user information. Upon confirming the dialog settings, the value of d is transferred over to the main *Edit Surface* dialog, where it is revealed as a read–in, uneditable value.

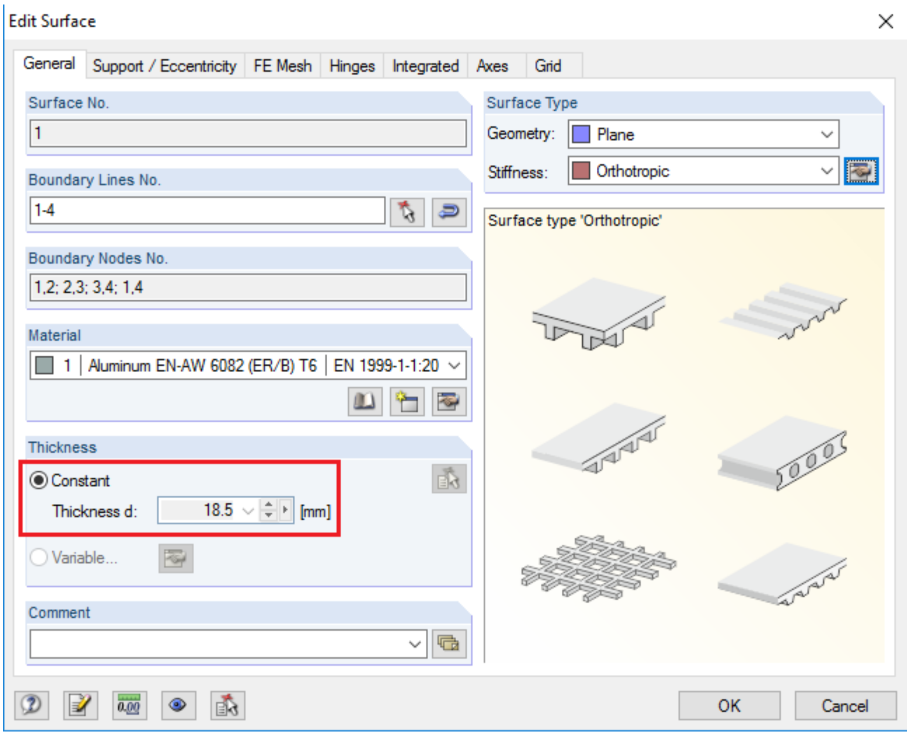


Figure 2.25: Thickness d in dialog box *Edit Surface*

This applies to all orthotropy models except for *Constant thickness*, *Effective thickness* and *Defined by stiffness matrix*. For the latter three models, RFEM does not generate any equivalent thickness, but expects the user to input it manually via any of the two possible ways. There is a default editable value 180 mm.

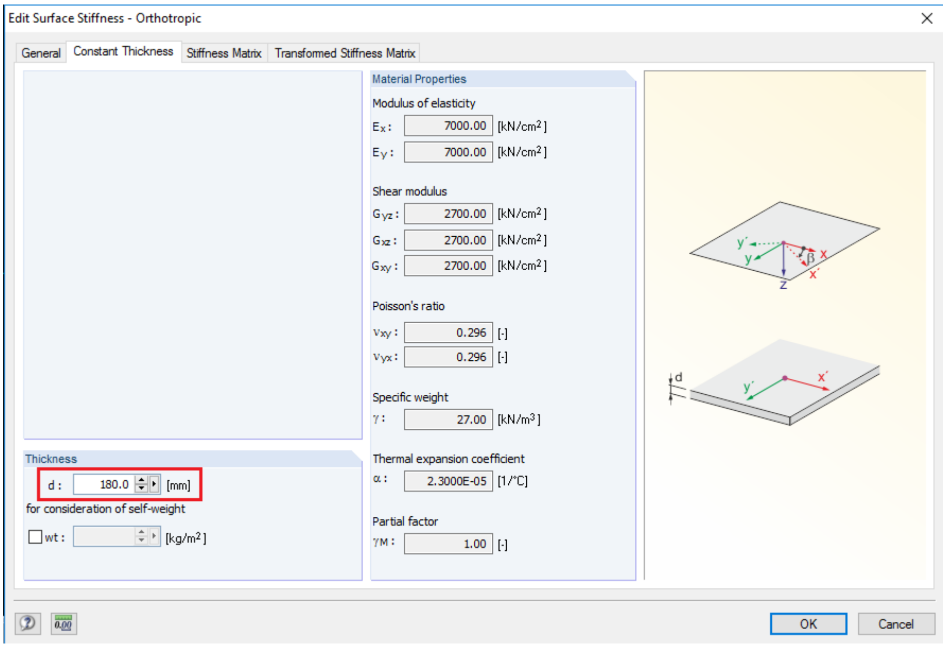


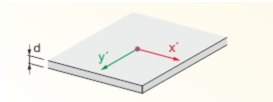
Figure 2.26: Default thickness d of an orthotropic surface *Constant thickness*

In some cases, the material specific weight γ might be explicitely set by the user to zero. If the alternative self-weight definition option via wt is enabled and the material has indeed zero self-weight, an error message is displayed indicating that the user must assign some value to γ if the equivalent thickness is to be computed on the basis of user–defined wt. This is due to the fact that in the conversion formula above γ appears in the denominator.

3 Orthotropy in RFEM 5

3.1 Material Orthotropy in 2D

3.1.1 Constant Thickness



Access

Accessing this type of orthotropy model can be done from the *Edit Surface Stiffness - Orthotropic* dialog and choosing *Constant thickness* under Orthotropy Type on the left, see Figure 3.1.

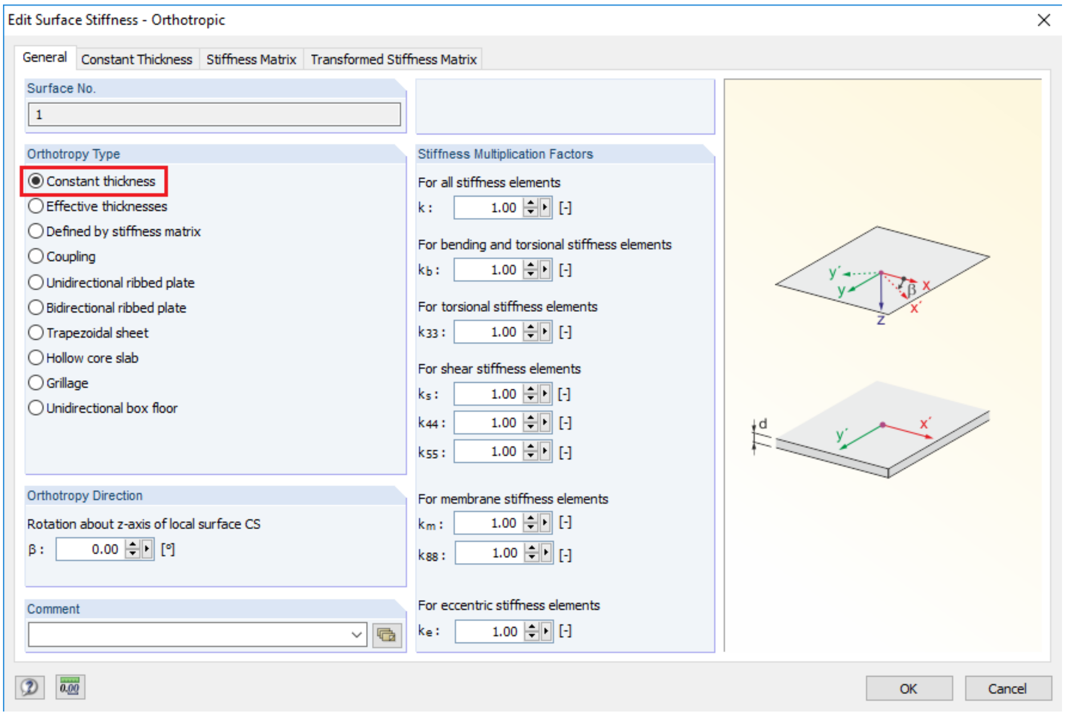


Figure 3.1: Accessing *Constant thickness* material model via *Edit Surface Stiffness - Orthotropy* dialog box

Description

This is the most common type of a pure material isotropy/orthotropy, exhibited by a homogeneous constant thickness plate.

Stiffness matrix coefficients

In the case of pure material orthotropy, the stiffness matrix coefficients generated by RFEM 5 are given as

Bending Components	Shear Components	Membrane Components
$D_{11} = \dfrac{E_x d^3}{12(1 - \nu_{xy}\nu_{yx})}$	$D_{44} = \dfrac{5}{6} G_{xz} d$	$D_{66} = \dfrac{E_x d}{1 - \nu_{xy}\nu_{yx}}$
$D_{22} = \dfrac{E_y d^3}{12(1 - \nu_{xy}\nu_{yx})}$	$D_{55} = \dfrac{5}{6} G_{yz} d$	$D_{77} = \dfrac{E_y d}{1 - \nu_{xy}\nu_{yx}}$
$D_{12} = \sqrt{\nu_{xy}\nu_{yx} D_{11} D_{22}}$		$D_{67} = \sqrt{\nu_{xy}\nu_{yx} D_{66} D_{77}}$
$D_{33} = \dfrac{G_{xy} d^3}{12}$		$D_{88} = G_{xy} d$

Table 3.1: Stiffness coefficients of orthotropy type *Constant thickness*

where E_x, E_y, ν_{xy}, ν_{yx}, G_{yz}, G_{xz}, G_{xy} are the elastic material properties of the *Orthotropic Elastic 2D* material selected from the **Edit Material** → **Material Model** combobox (cf. Chapter 1).

In case of pure material isotropy (*Isotropic Linear Elastic* material model being selected in the **Edit Material** → **Material Model** combobox and assigned to the plate via **Edit Surface** → **Material**), the material properties become

$$E_x = E_y = E, \quad \nu_{xy} = \nu_{yx} = \nu, \quad G_{xy} = G_{xz} = G_{yz} = G = \frac{E}{2(1+\nu)} \tag{3.1}$$

The stiffness coefficients generated by RFEM are then

Bending Components	Shear Components	Membrane Components
$D_{11} = \frac{Ed^3}{12(1-\nu^2)}$	$D_{44} = \frac{5}{6}Gd$	$D_{66} = \frac{Ed}{1-\nu^2}$
$D_{22} = \frac{Ed^3}{12(1-\nu^2)}$	$D_{55} = \frac{5}{6}Gd$	$D_{77} = \frac{Ed}{1-\nu^2}$
$D_{12} = \nu\sqrt{D_{11}D_{22}} = \frac{Ed^3\nu}{12(1-\nu^2)}$		$D_{67} = \nu\sqrt{D_{66}D_{77}} = \frac{Ed\nu}{1-\nu^2}$
$D_{33} = \frac{G_{xy}d^3}{12} = \frac{Ed^3}{24(1+\nu)}$		$D_{88} = Gd = \frac{Ed}{2(1+\nu)}$

Table 3.2: Stiffness coefficients of orthotropy type *Constant thickness*

All eccentric terms of this model are zero.

Model limitations

The model is based on exactly derived theoretical coefficients suggesting that accuracy and stability of the solution are implicitly inherited. The model has no limitations.

Plate theory applicable – both Kirchhoff and Mindlin.

Analysis type applicable – I., II., and III. order in combination with geometrical nonlinearities only. Linear bending elastic static analysis uses the herein generated stiffness matrix coefficients. Membranes, second and third order analyses require as an additional parameter the equivalent thickness d for self-weight computation in order to form the geometrical stiffness matrrix (also used in stability analysis), large displacement and rotations stiffnesses, membrane effects and soil–structure interaction. Material nonlinearity, such as plasticity, is not possible.

Material type applicable – both *Isotropic* and *Orthotropic Elastic 2D/3D* are possible and set via the *Edit Material* dialog.

Orthotropic direction and angle β – can be applied and manually set by the user via the **Edit Surface - Orthotropic** → **General** path.

Stiffness reduction factors – all types of stiffness reduction factors, cf. Section 2.2.5, are applicable and editable via **Edit Surface - Orthotropic** → **Defined by stiffness matrix**.

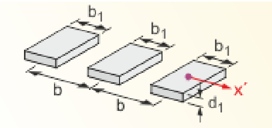
Equivalent thickness for self-weight computation – only user-definable via d or wt in the *Edit Surface* dialog.

User recommendations

This model belongs to one of the most commonly encountered ones in practice. It provides excellent results with plate mesh refinement and accurate representation of higher order nonlinear problems.

3.2 Geometrical Orthotropy in 2D

3.2.1 Coupling



Access

Accessing this model can be done from the *Edit Surface - Orthotropic* dialog and choosing *Coupling* under Orthotropy Type on the left, see Figure 3.2.

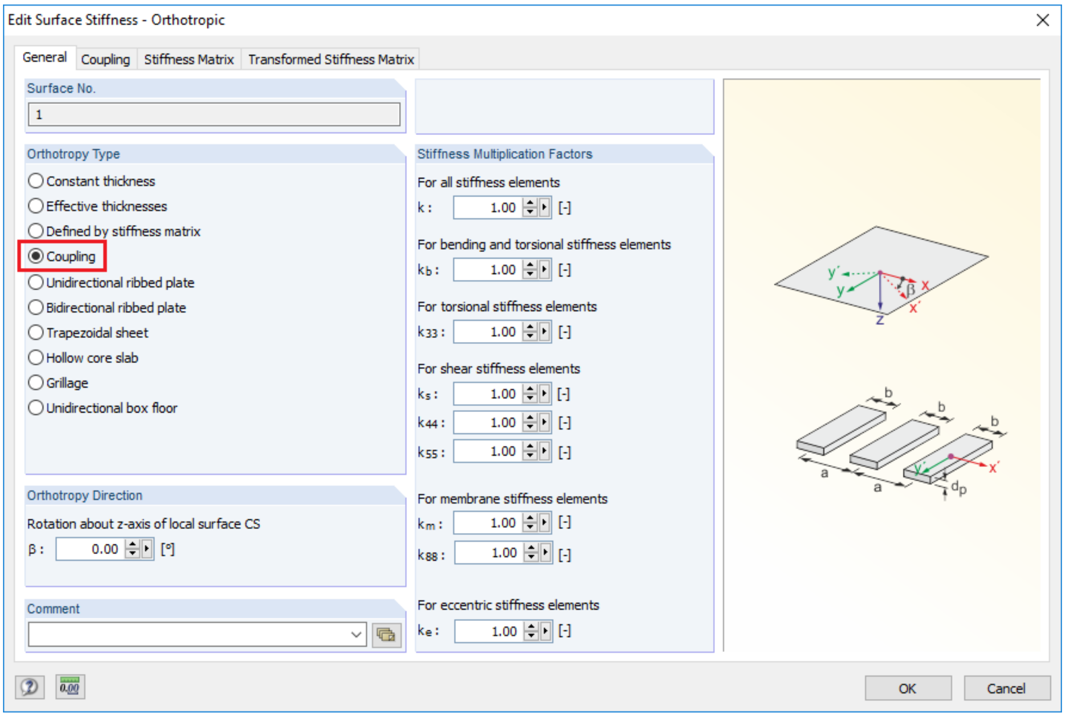


Figure 3.2: Accessing *Coupling* material model via *Edit Surface Stiffness - Orthotropy* dialog box

Description

Coupling is designed to represent the connection of two plates or two beams by connectors made of isotropic material and equivalent thickness d. The equivalent thickness represents a thickness which takes into account the spacing and the dimensions of two connecting plate or beam elements. If the spacing between the connectors is a, the lenght of the connector in x-direction is b and thickness is d_p, then the equivalent thickness d is computed as

$$d = d_p \frac{b}{a} \tag{3.2}$$

The geometric properties for this model, which are to be found under the *Coupling* tab in the *Edit Surface - Orthotropic* material dialog, have the following format

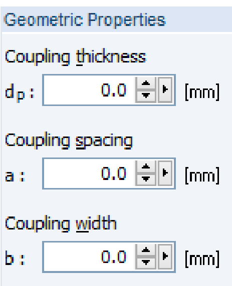


Figure 3.3: Dialog section *Geometric Properties*

Depending upon the user input, two possibilities exist

1. a > b, then d < d_p, which is a realistic coupling model
2. a = b, then d = d_p, and the model approaches the special case of a uniform continuous constant thickness plate model

However, RFEM 5 imposes the geometrical restrictions as to distinguish between both models

1. $d_p > 0, \quad a > 0, \quad b > 0$
2. $b < a$

These restrictions are necessary as they allow us to model a realistic coupling model.

Stiffness matrix coefficients

The stiffness matrix coefficients are given as

Bending Components	Shear Components	Membrane Components
$D_{11} = 10^{-6} D_{22}$	$D_{44} = 10^{6} D_{55}$	$D_{66} = 10^{6} D_{77}$
$D_{22} = \frac{Ed^3}{12(1-\nu^2)}$	$D_{55} = \frac{5}{6} Gd$	$D_{77} = \frac{Ed}{1-\nu^2}$
$D_{12} = \nu\sqrt{D_{11}D_{22}}$		$D_{67} = \nu\sqrt{D_{66}D_{77}}$
$D_{33} = G\frac{k_1 b d_p^3}{4a}$		$D_{88} = Gd$

Table 3.3: Stiffness coefficients of orthotropy type *Coupling*

These coefficients are indeed the same as those obtained in the case of a constant thickness isotropic plate, see Section 3.1.1, with the only difference being that now the plate thickness is replaced by the effective plate thickness d and the x′-direction shear, bending and membrane stiffnesses are practically zero, $D_{11} \approx 0$, $D_{44} \approx 0$, $D_{77} \approx 0$. If the user chooses $a \to b$ or $a = b$, the model tends to be in fact a constant thickness isotropic plate with practically no stiffness in the x′ orthotropic directio and no coupling effects, $D_{12} \to 0$, $D_{67} \to 0$

$$\lim_{a\to b} D_{22} = \lim_{a\to b}\left(\frac{Ed^3}{12(1-\nu^2)}\right) = \lim_{a\to b}\left(\frac{Ed_p^3}{12(1-\nu^2)}\left(\frac{b}{a}\right)^3\right) = \frac{Ed_p^3}{12(1-\nu^2)}$$

$$\lim_{a\to b} D_{11} = 10^{-6}\lim_{a\to b} D_{22} \approx 0$$

$$\lim_{a\to b} D_{12} = \lim_{a\to b}\left(\nu\sqrt{D_{11}D_{22}}\right) \approx 0$$

$$\lim_{a\to b} D_{33} = \lim_{a\to b, k_1\to\frac{1}{3}} G\frac{k_1 b d_p^3}{4a} = G\frac{d_p^3}{12}$$

$$\lim_{a\to b} D_{55} = \frac{5}{6}\lim_{a\to b} Gd_p\frac{b}{a} = \frac{5}{6}Gd_p$$

$$\lim_{a\to b} D_{44} = 10^{-6}\lim_{a\to b} D_{55} \approx 0$$

$$\lim_{a\to b} D_{77} = \lim_{a\to b}\left(\frac{Ed}{1-\nu^2}\right) = \lim_{a\to b}\left(\frac{Ed_p}{1-\nu^2}\left(\frac{b}{a}\right)\right) = \frac{Ed_p}{1-\nu^2}$$

$$\lim_{a\to b} D_{66} = 10^{-6}\lim_{a\to b} D_{77} \approx 0$$

$$\lim_{a\to b} D_{88} = \lim a \to b\left(Gd_p\frac{b}{a}\right) = Gd_p$$

All eccentric terms of this model are zero.

Model limitations

- The stiffness in the x-direction is implicitly decreased to a negligibly small value, the purpose of which is to avoid numerical instability and a singular stiffness matrix. In reality all x-direction stiffnesses are indeed very small, especially when compared to those in the y-direction, nevertheless non-zero. That is the reason why RFEM 5 introduces a set of ``approximate" reductions of the order 10^{-6}, which have porven to be satisfactory in the most practical cases.
- Orthotropic material is not allowed to be used.
- The plate of beam members being coupled need to be of uniform constant thickness.

Plate theory applicable – both Kirchhoff and Mindlin.

Analysis type applicable – I., II., and III. order in combination with geometrical nonlinearities only. Linear bending elastic static analysis uses the herein generated stiffness matrix coeddicients. Membranes, second and third order analysis require as an additional parameter the equivalent thickness d for self-weight computation in order to form the geometrical stiffness matrix (also used in instability analysis), large displacement and rotations stiffnesses, membrane effects and soil–structure interaction. Material nonlinearity, such as plasticity, is not possible.

Material type applicable – *Isotropic Linear Elastic* material only set via the *Edit Material* dialog.

Orthotropic direction and angle β – can be applied and manually set by the user via the **Edit Surface - Orthotropic** $\rightarrow$ **General** tab.

Stiffness reduction factors – all types of stiffness reduction factors, cf. Section 2.2.5, are applicable and editable via **Edit Surface - Orthotropic** $\rightarrow$ **Defined by stiffness matrix**.

Equivalent thickness for self-weight computation – the value of d or wt is automatically computed by RFEM 5 as

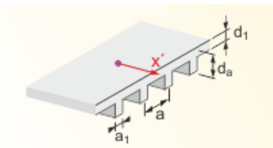
$$d = d_p \tag{3.3}$$

It can be also user edited via the d or wt options in the *Edit Surface* dialog.

User recommedations

This model is recommended in cases where plate or beam members remain stiff and flexible in the transverse y′-direction, but behave as ideally disconnected and not interacting in the longitudinal direction. Also, since the x′-direction stiffness is practically set to zero, a reasonable physical representation of this model would suggest short breadth (narrow), but long in the y′-direction, members of strip-like behavior the y′-direction stiffness of which must be several magnitudes greater than the one in the x′-direction. The x′-direction spacing between the members could in fact be quite large, since members are to be treated as loose in it. In fact, the coupling model could be used to effectively transfer forces in the transverse y′-direction from one plate model to another.

3.2.2 Unidirectional Ribbed Plate



Access

Accessing this model can be done from the *Edit Surface - Orthotropic* dialog and choosing *Unidirectional Ribbed Plate* under Orthotropy Type on the left, see Figure 3.4.

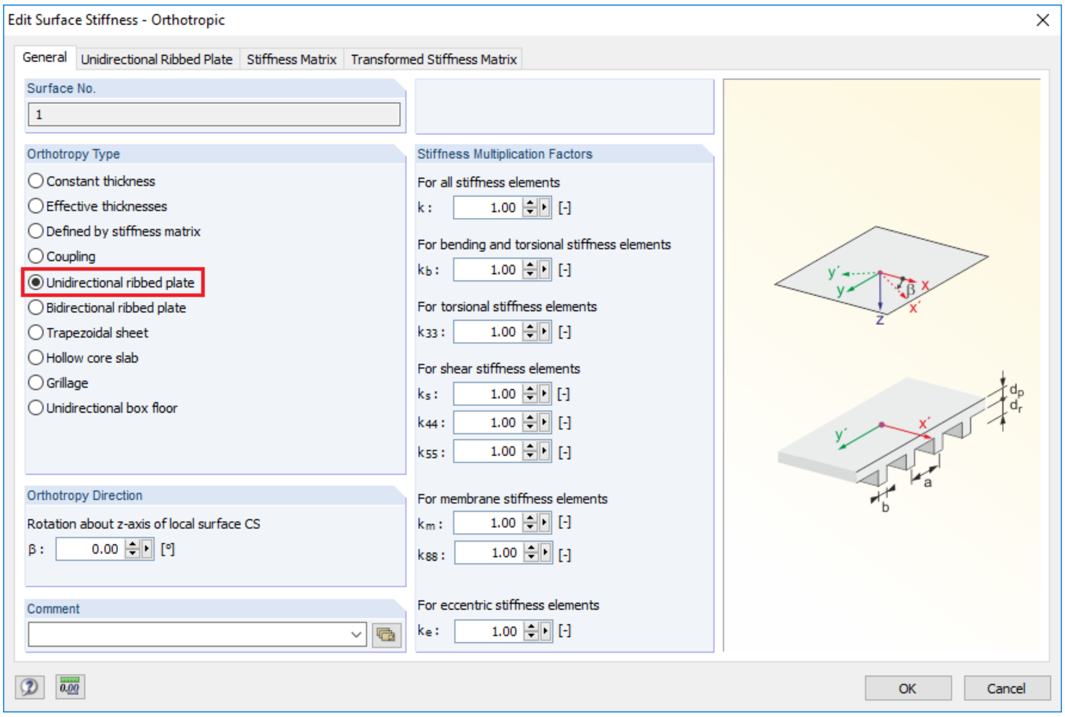


Figure 3.4: Accessing *Unidirectional ribbed plate* material model via *Edit Surface Stiffness - Orthotropy* dialog

Description

The unidirectional ribbed plate approximates a unidirectionally spanning slab, reinforced by rectangular ribs in the longitudinal direction. The transverse direction is not reinforced and is represented by a constant thickness plate. The requested geometrical dimensions are as follows

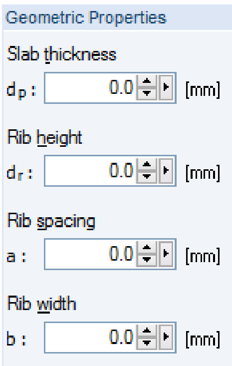


Figure 3.5: Geometrical input parameters

The following geometrical restrictions apply

1. $d_p > 0$, $d_r > 0$, $a > 0$, $b > 0$
2. $b < a$

Stiffness matrix coefficients

The stiffness matrix coefficients are given as

Bending Components	Shear Components	Membrane Components
$D_{11} = \frac{EI^*_{xx'}}{a}$	$D_{44} = \frac{GA_{x'}}{a\beta_{x'}}$	$D_{66} = \frac{EA^*_{x'}}{a}$
$D_{22} = EI^*_{yy'}$	$D_{55} = \frac{GA_{y'}}{\beta_{y'}}$	$D_{77} = EA^*_{y'}$
$D_{12} = \nu D_{22}$		$D_{67} = \nu D_{77}$

$D_{33} = GI_k$			$D_{88} = Gd_p$

Table 3.4: Stiffness coefficients of orthotropy type *Coupling*

The stiffness matrix coefficients are based on the simplified theory of Timoshenko and Woinowsky-Krieger, see [1], the interested users are referred to [1] and the references therein for full derivation and further assumptions.

The focus here shall not lie on the theoretical background presented in [1], but rather the definition of the required terms and parameters appearing in the stiffness coefficients.

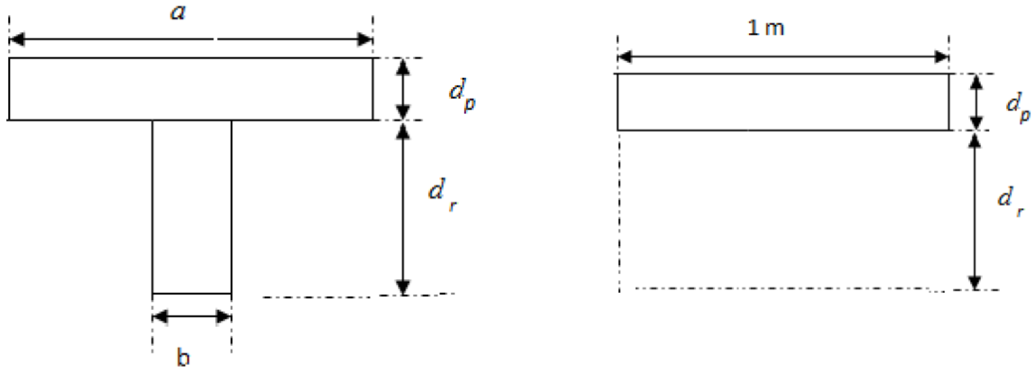


Figure 3.6: Cross-section in the longitudinal x−x direction and unit cross-section in y−y transverse direction

- $I^{\star}_{xx'}$ = second moment of area of a unit T cross-section of flange width $\frac{a}{1-\nu^2}$ and other dimensions, as shown in Figure 3.6. The introduction of the Poisson ratio is due to the anticlastic effect provided by the plate, as the T-beam flange remains continuous throughout the entire surface. Thus it can be considered to have a stiffening effect, allowing for plane strain conditions in the case when the ribbed surface is subjected to curvature in one direction only.
- $A_{x'}$ = fully uncrackled gross cross-sectional area of a T cross-section of width a equal to the rib spacing, and other dimensions as shown in Figure 3.6.
- $A^{\star}_{x'}$ = cross-sectional area resisting the x-directional membrane force

$$A^{\star}_{x'} = \frac{Ead_p}{1-\nu^2} + Ebd_r \tag{3.4}$$

- $\beta_{x'}$ = Jurawski–Grashof coefficient of a unit T cross-section of width a

$$\beta_{x'} = \frac{A_{x'}}{I^2_{xx'}} S_{Qx'} \tag{3.5}$$

where $S_{Qx'}$ is defined as

$$S_{Qx'} = \iint\limits_{A} \frac{S^2_{x'}(z)}{\eta^2_{x'}(z)} \mathrm{d}A \tag{3.6}$$

for the statical moment of area $S^2_{x'}(z)$ of the cross-section at a location z, and the thickness $\eta_{x'}(z)$ of the cross-sectional cut above the neutral axis, and dA means that the integration is carried out over the whole $x - x$ cross-section.

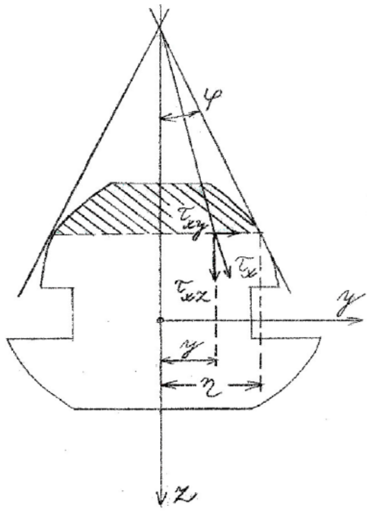


Figure 3.7: Definition of first statical moment of area

- $I^{\star}_{yy'}$ = second moment of area of a unit cross-section of width $1/(1-\nu^2)$ along the $y-y$ axis, but partially accounting for the discontinuously interacting ribs in the x-direction via

$$I^{\star}_{yy'} = \frac{ad_p^3}{12(1-\nu^2)\psi}, \quad \psi = a + b\left(\left(\frac{d_p}{d_r + d_p}\right)^3 - 1\right) \tag{3.7}$$

  The correction factor was proposed by Timoshenko & Woinowsky-Krieger [1]. Note that $\psi \to a$ for $d_r \to 0$, in which case $I^{\star}_{yy'} \to \frac{ad_p^3}{12(1-\nu^2)}$, i.e., the second moment of area of a constant-thickness plate of unit width.

  The introduction of the Poisson ratio is due to the anticlastic effect provided by the plate, as the T-beam flange remains continuous thorughout the whole surface. Thus it can be considered to have a stiffenning effect.

- $A^{\star}_{xx'}$ = cross-sectional area resisting the y-directional membrane force

$$A^{\star}_{xx'} = \frac{Ead_p}{1-\nu^2} \tag{3.8}$$

- $\beta_{y'}$ = Jurawski–Grashof coefficient of a unit cross-section of width 1 along the y-axis

$$\beta_{y'} = \frac{6}{5} = 1.2 \tag{3.9}$$

- I_k = torsional constant of the unidirectional ribbed plate, being the sum of the torsional rigidities of the slab $I_{k,slab} = \frac{d_p^3}{12}$ and the rib in the longitudinal x-direction $I_{k,rib} = \frac{k_1 C_{rib}}{4a}$, where C_{rib} is the x-directional torsional stiffness of the rib and k_1 its correction factor. The latter is the solution of the governing torsional equation for a rib, given by Timoshenko & Goodier [2], depending on the ratio of the rib height h and its width t, as

$$C_{rib} = k_1 h t^3, \quad k_1 = \frac{1}{3}\left(1 - \frac{192}{\pi^5}\frac{t}{h}\sum_{n=1,2,5,\ldots}\frac{1}{n^5}\tanh\frac{nh\pi}{2t}\right) \quad \text{if } h \geq t, \tag{3.10}$$

$$C_{rib} = k_1 t h^3, \quad k_1 = \frac{1}{3}\left(1 - \frac{192}{\pi^5}\frac{h}{t}\sum_{n=1,2,5,\ldots}\frac{1}{n^5}\tanh\frac{nt\pi}{2h}\right) \quad \text{if } h < t, \tag{3.11}$$

  RFEM calculates k_1 iteratively until a convergence in the sense that $\frac{k_{1,n}-k_{1,n-1}}{k_{1,n}} \leq \varepsilon = 10^{-5}$ is achieved, where n and $n-1$ denote two consecutive approximations. On the other hand, an approximate expression is was also given by Roark, see [3], and could be used for a quick hand-calculation

$$C_{rib} = ht^3\left[\frac{1}{3} - 0.21\frac{t}{h}\left(1 - \frac{t^4}{12h^4}\right)\right] \quad \text{for } h \geq t \tag{3.12}$$

Model limitations

- Orthotropic material is not allowed to be used. The model supports only linear elastic iso-tropic material.
- The material remains fully elastic and fully uncracked, i.e., both tension and compression zones remain fully active. If concrete is to be used, we should regard this state as concrete state I. Therefore plate material shall not yield or crack in dependence of the resulting stress. It is up to the designer to use a reasonable loading such as to prevent initiation of any post-elastic (plastic or cracked) behaviour. Reinforcement is not considered in this type of analysis.
- The model disregards the fact that the neutral axis in the x and y-directions change their positions which introduces a degree of nonlinearity, especially with increasing loading, more distinctive geometry and greater deflections.
- Rib spacing between two consective unit modules must be always constant and thus cannot be varying, i.e., $a = \text{const}$.
- Eccentric stiffness matrix terms are disregarded and set to zero.
- The bending stiffness in the orthogonal $y - y$ direction, originally proposed in [1], is slightly overestimated as shown and commented in [4]. In reality the stiffness takes into account the transverse contraction and is less than the one predicted by the theory employed herein.

Plate theory applicability – both Kirchhoff and Mindlin.

Analysis types applicable – I., II., and III. order in combination with geometrical nonlinearities only. Linear bending elastic static analysis uses the herein generated stiffness matrix coefficients. Membranes, second and third order analysis require as an additional parameter the equivalent thickness for self-weight computation d in order to form the geometrical sitffness matrix (also used in instability analysis), large displacement and rotation stiffnesses, membrane effects and soil–structure interaction. Material nonlinearity such as plasticity is not possible.

Material types applicable – Isotropic material only set via the *Edit Material* dialog and assigned to the plate via **Edit Surface** $\rightarrow$ **Material**.

Orthotropic direction and angle β – can be applied and manually set by the user via the *Edit Surface - Orthotropic* dialog.

Stiffness reduction factors – all types of stiffness reduction factors applicable, cf. Section 2.2.5, and also editable in the *Edit Surface - Orthotropic* dialog.

Equivalent thickness for self-weight computation – the value of d is computed according to

$$d = \frac{A_{x'}}{a} \tag{3.13}$$

and shown in the *Equivalent Thickness* read-in window under the *Unidirectional Ribbed Plate* tab in *Edit Surface - Orthotropic* dialog.

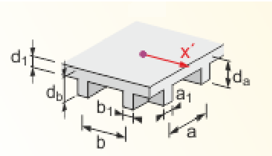
User recommendations

This model is generally recommended for a reasonably approximate modeling of concrete slab systems (T-beam floors), reinforced with unidirectionally spanning rectangular ribs. The smaller the rib spacing and the rib depth, the more accurate the analysis results, due to the fact that in the limit as $d_r \rightarrow 0$ and $b \rightarrow a$, the unidirectional ribbed model tends to revert back to the constant thickness isotropic material one, the analytical solution of which is indeed exact.

The flexural rigidity in the longitudinal direction, evaluated by means of standard methods for calculating the second moment of area about the centroid of the gross cross-section, have been proven to be accurate for this type of design. Therefore, no modification factors are here recommended and the default value of 1 should not be changed. The latter also applies to the torsional stiffness, which in light of the current theory used has been proven to yield realistic and accurate values.

Due to the fact that the transverse stiffness coefficient D_{22} tends to be slightly overestimated, a slight reduction by a factor of 0.9 might be manually applied by the user in order to improve the accuracy of the transverse stiffness, especially when there are wide ribs in the longitudinal direc-tion.

3.2.3 Bidirectional Ribbed Plate



Access

Accessing this model can be done from the *Edit Surface - Orthotropic* dialog and choosing *Bidirectional Ribbed Plate* under Orthotropy Type on the left, see Figure 3.8.

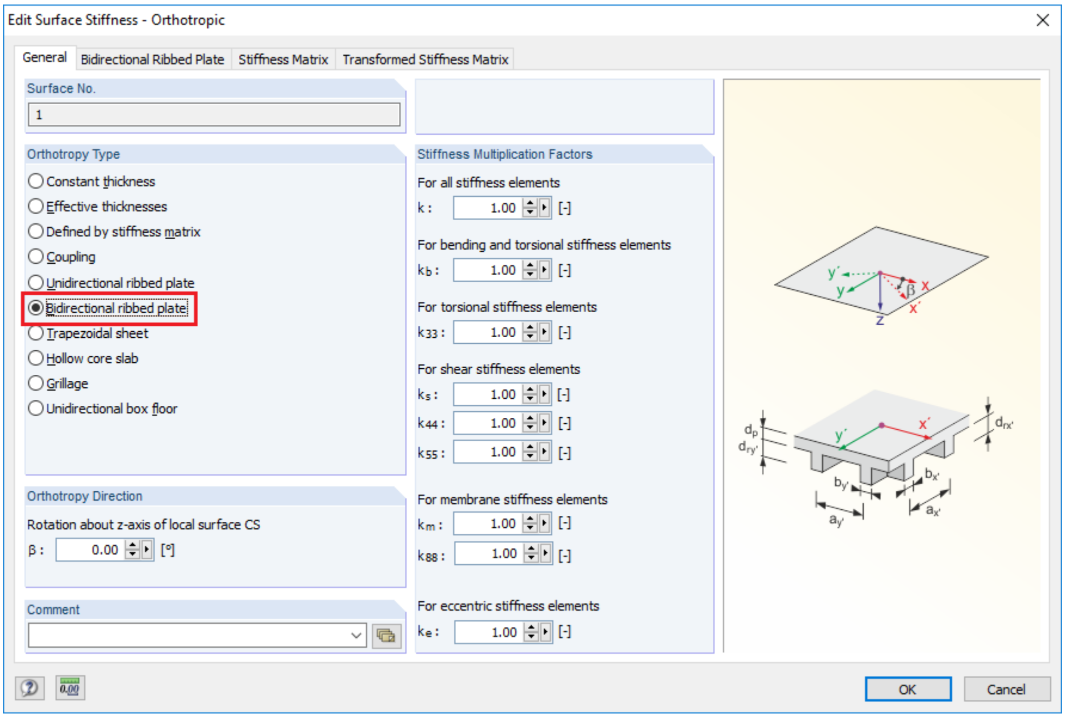


Figure 3.8: Accessing *Bidirectional ribbed plate* material model via *Edit Surface Stiffness - Orthotropy* dialog

Description

The model represents a common case of a constant-thickness slab, spanning along and ribbed in two orthogonal directions by eccentrical rectangular cross-section ribs.

The geometrical parameters are as follows

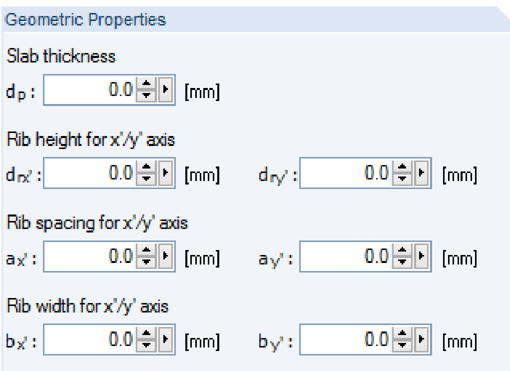


Figure 3.9: Geometrical input parameters

The following geometrical restrictions apply

1. $d_p > 0, \quad d_{rx'} > 0, \quad d_{ry'} > 0, \quad a_{x'} > 0, \quad a_{y'} > 0, \quad b_{x'} > 0, \quad b_{y'} > 0$
2. $a_{x'} > b_{x'}, \quad a_{y'} > b_{y'}$

Stiffness matrix coefficients

The stiffness matrix coefficients are given as

Bending Components	Shear Components	Membrane Components
$D_{11} = \frac{EI^{\star}_{xx'}}{a_{x'}}$	$D_{44} = \frac{GA_{x'}}{a_{x'}\beta_{x'}}$	$D_{66} = \frac{EA^{\star}_{x'}}{a_{x'}}$
$D_{22} = \frac{EI^{\star}_{yy'}}{a_{y'}}$	$D_{55} = \frac{GA_{y'}}{a_{y'}\beta_{y'}}$	$D_{77} = \frac{EA^{\star}_{y'}}{a_{y'}}$
$D_{12} = \frac{1}{2}(D_{12x'} + D_{12y'})$		$D_{67} = \frac{1}{2}(D_{67x'} + D_{67y'})$
$D_{33} = GI_k$		$D_{88} = Gd_p$

Table 3.5: Stiffness coefficients of orthotropy type *Coupling*

The stiffness matrix coefficients are on the simplified theory of Cusens et al. [4] as it is considered to be superior to other theories such as [1], [5], and [6] in terms of evaluationg flexural and torsional stiffnesses of bidirectionally ribbed systems. The interested users are referred to these articles, especially [4], and the references therein for full derivations and further assumptions.

The focus here shall not lie on the theoretical background presented in [4], but rather the definition of the required terms and parameters appearing in the stiffness coefficients.

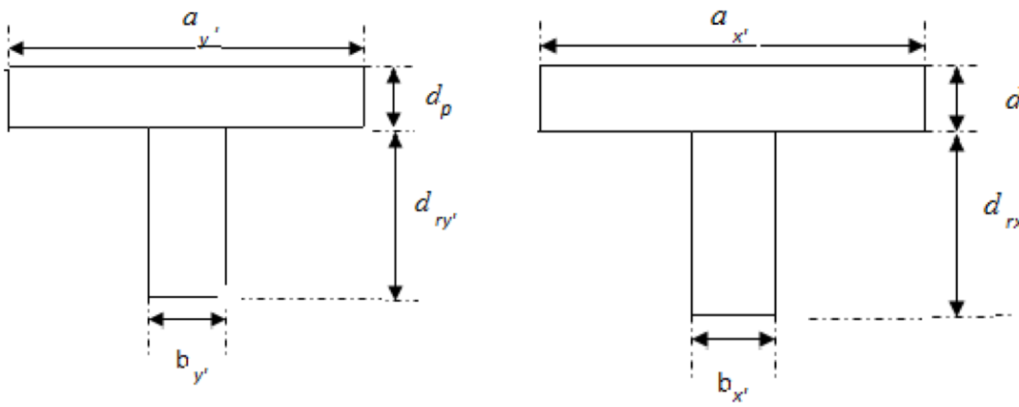


Figure 3.10: Cross-section in the y − y transverse direction and in x − x direction

- $I^{\star}_{xx'}$ = second moment of area of a unit T cross-section of flange width $\frac{a_{x'}}{1-\nu^2}$, rib width $\frac{b_{x'}}{1-\chi\nu^2}$, where $\frac{b_{x'}b_{y'}}{a_{x'}a_{y'}}$ (contraction ratio), and other dimensions, as shown in Figure 3.10. The latter is equivalent to the expression presented in [4] as

$$I^{\star}_{xx'} = \frac{a_{x'}d_p^3}{12(1-\nu^2)} + \frac{b_{x'}}{6(1-\chi\nu^2)}\left[\left(d_{rx'} - \left(e_x - \frac{d_p}{2}\right)\right)^2 (2d_{rx'} + e_x + d_p) - \left(e_x - \frac{d_p}{2}\right)^2 (e_x + d_p)\right] \tag{3.14}$$

The parameter e_x represents the position of the neutral axis of the cross-section measured from the slab centerline

$$e_x = \frac{b_{x'}d_{rx'}(d_{rx'} + d_p)}{2(\lambda a_{x'}d_p + b_{x'}d_{rx'})} \quad \text{for } \lambda = \frac{1-\chi\nu^2}{1-\nu^2} \tag{3.15}$$

The introduction of the Poisson ratio is due to the anticlastic effect provided by the plate, as the T-beam flange remains continuous throughout the entire surface. Thus it can be considered to have a stiffening effect, allowing for plane strain conditions in the case when the ribbed surface is subjected to curvature in one direction only.

- $A_{x'}$ = fully uncrackled gross cross-sectional area of a T cross-section of width $a_{x'}$ equal to the rib spacing, and other dimensions as shown in Figure 3.10.
- $A^{\star}_{x'}$ = cross-sectional area resisting the x-directional membrane force

$$A^{\star}_{x'} = \frac{\nu E a_{x'} d_p}{1-\nu^2} + \frac{E b_{x'} d_{rx'}}{1-\chi\nu^2} \tag{3.16}$$

- $\beta_{x'}$ = Jurawski–Grashof coefficient of a unit T cross-section of width $a_{x'}$ equal to the rib spacing, and other dimensions as shown in Figure 3.10, and defined in Section 3.2.2.
- $I^{\star}_{yy'}$ = second moment of area of a unit T cross-section of flange width $a_{y'}/(1-\nu^2)$, rib width $b_{y'}/(1-\chi\nu^2)$, for $\chi = b_{x'} b_{y'}/a_{x'} a_{y'}$. The latter is equivalent to the expression presented in [4] as

$$I^{\star}_{yy'} = \frac{a_{x'} d_p^3}{12(1-\nu^2)} + \frac{b_{y'}}{6(1-\chi\nu^2)} \left[\left(d_{ry'} - \left(e_y - \frac{d_p}{2} \right) \right)^2 (2d_{ry'} + e_y + d_p) - \left(e_y - \frac{d_p}{2} \right)^2 (e_y + d_p) \right] \tag{3.17}$$

The parameter e_y represents the position of the neutral axis of the cross-section measured from the slab centerline

$$e_y = \frac{b_{y'} d_{ry'} (d_{ry'} + d_p)}{2(\lambda a_{y'} d_p + b_{y'} d_{ry'})} \quad \text{for } \lambda = \frac{1-\chi\nu^2}{1-\nu^2} \tag{3.18}$$

The introduction of the Poisson ratio is due to the anticlastic effect provided by the plate, as the T-beam flange remains continuous throughout the entire surface. Thus it can be considered to have a stiffening effect, allowing for plane strain conditions in the case when the ribbed surface is subjected to curvature in one direction only.

- $A_{y'}$ = fully uncrackled gross cross-sectional area of a T cross-section of width $a_{y'}$ equal to the rib spacing, and other dimensions as shown in Figure 3.10.
- $A^{\star}_{y'}$ = cross-sectional area resisting the y-directional membrane force

$$A^{\star}_{y'} = \frac{\nu E a_{y'} d_p}{1-\nu^2} + \frac{E b_{y'} d_{ry'}}{1-\chi\nu^2} \tag{3.19}$$

- $\beta_{y'}$ = Jurawski–Grashof coefficient of a unit T cross-section of width $a_{y'}$ equal to the rib spacing, and other dimensions as shown in Figure 3.10, and defined analogously as $\beta_{x'}$.
- The beding coupling stiffnesses $D_{12} = D_{21}$ are determined as

$$D_{12} = D_{21} = \frac{1}{2}\left(D_{12x'} + D_{12y'}\right) \tag{3.20}$$

where the two, directionally different, coefficients $D_{12x'} \neq D_{12y'}$ are given as, see [4],

$$D_{12x'} = \frac{\nu E d_p^3}{12(1-\nu^2)} + \frac{\chi\nu E}{6(1-\chi\nu^2)} \left[\left(d_{ry'} - \left(e_y - \frac{d_p}{2} \right) \right)^2 (2d_{ry'} + e_y + d_p) - \left(e_y - \frac{d_p}{2} \right)^2 (e_y + d_p) \right] \tag{3.21}$$

$$D_{12y'} = \frac{\nu E d_p^3}{12(1-\nu^2)} + \frac{\chi\nu E}{6(1-\chi\nu^2)} \left[\left(d_{ry'} - \left(e_x - \frac{d_p}{2} \right) \right)^2 (2d_{ry'} + e_x + d_p) - \left(e_x - \frac{d_p}{2} \right)^2 (e_x + d_p) \right] \tag{3.22}$$

One could observe the presence of additional second term, taking into account the contribution of both ribs and their eccntricities to the overall coupling stiffness. Usually only the first term (solely the plate coupling stiffness) is suggested by authors such as Sawko and Cope [6]. The geometrical average of both is taken into account in order to ensure a single coupling component and a conservative value. If the full theory is to be used, one has to consider a special type of plate kinematics where $D_{12x'} \neq D_{12y'}$, resulting in an non-symmetrical problem.

- The membrane coupling stiffnesses $D_{67} = D_{76}$ are determined as

$$D_{67} = D_{76} = \frac{1}{2}\left(D_{67x'} + D_{67y'}\right) \tag{3.23}$$

where the two, directionally different, coefficients $D_{67x'} \neq D_{67y'}$ are given as, see [4],

$$D_{67x'} = \frac{\nu E d_p}{12(1-\nu^2)} + \frac{\nu\chi E d_{rx'}}{1-\chi\nu^2} \tag{3.24}$$

$$D_{67y'} = \frac{\nu E d_p}{12(1-\nu^2)} + \frac{\nu\chi E d_{ry'}}{1-\chi\nu^2} \tag{3.25}$$

- I_k = torsional constant of the bidirectional ribbed plate, being the sum of the torsional rigidities of the slab $I_{k,slab} = \frac{d_p^3}{12}$, the rib in the longitudinal x-direction $I_{k,rib,x} = \frac{k_{1,x} C_{rib,x}}{4}$, and the rib in the transversal y-direction $I_{k,rib,y} = \frac{k_{1,y} C_{rib,y}}{4}$, where $C_{rib,i}$ are the torsional rib stiffnesses of the rib and $k_{1,i}$ the corresponding correction factors. The latter are the solution of the governing torsional equation for a rib, given by Timoshenko & Goodier [2], depending on the ratio of the rib height h_i and its width t_i, as

$$C_{rib,i} = k_{1,i} h_i t_i^3, \quad k_{1,i} = \frac{1}{3}\left(1 - \frac{192}{\pi^5}\frac{t_i}{h_i}\sum_{n=1,2,5,\ldots}\frac{1}{n^5}\tanh\frac{n h_i \pi}{2 t_i}\right) \quad \text{if } h \geq t, \tag{3.26}$$

$$C_{rib,i} = k_{1,i} t_i h_i^3, \quad k_{1,i} = \frac{1}{3}\left(1 - \frac{192}{\pi^5}\frac{h_i}{t_i}\sum_{n=1,2,5,\ldots}\frac{1}{n^5}\tanh\frac{n t_i \pi}{2 h_i}\right) \quad \text{if } h < t, \tag{3.27}$$

RFEM calculates $k_{1,i}$ iteratively until a convergence in the sense that $\frac{k_{1,i,n} - k_{1,i,n-1}}{k_{1,i,n}} \leq \varepsilon = 10^{-5}$ is achieved, where n and $n-1$ denote two consecutive approximations. On the other hand, an approximate expression is was also given by Roark, see [3], and could be used for a quick hand-calculation

$$C_{rib,i} = h_i t_i^3\left[\frac{1}{3} - 0.21\frac{t_i}{h_i}\left(1 - \frac{t_i^4}{12 h_i^4}\right)\right] \quad \text{for } h_i \geq t_i \tag{3.28}$$

Note that the equivalent thickness d for self-weight computation is in this case calculated as

$$d = \frac{V}{a_{x'} a_{y'}} \tag{3.29}$$

where V is a volume segment shown in Figure 3.11 and computed as

$$V = a_{x'} a_{y'} d_p + d_{ry'} b_{y'} a_{x'} + d_{rx'} b_{x'} a_{y'} - b_{x'} b_{y'} \overline{d_{eff}} \tag{3.30}$$

where

$$\overline{d_{eff}} = \begin{cases} d_{ry'}, & \text{for } d_{rx'} \geq d_{ry'}; \\ d_{rx'}, & \text{for } d_{rx'} < d_{ry'} \end{cases} \tag{3.31}$$

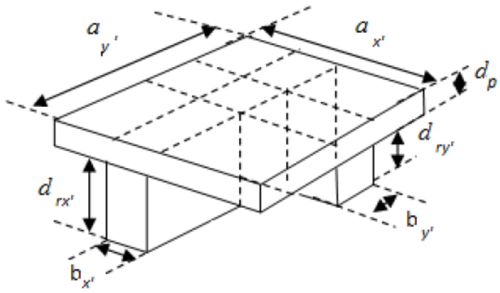


Figure 3.11: Volume section V for a bidirectional ribbed plate

Model limitations

- Orthotropic material is not allowed to be used. The model supports only linear elastic iso-tropic material.
- The material remains fully elastic and fully uncracked, i.e., both tension and compression zones remain fully active. If concrete is to be used, we should regard this state as concrete state I. Therefore plate material shall not yield or crack in dependence of the resulting stress. It is up to the designer to use a reasonable loading such as to prevent initiation of any post-elastic (plastic or cracked) behaviour. Reinforcement is not considered in this type of analysis.
- The model disregards the fact that the neutral axis in the x and y-directions change their positions which introduces a degree of nonlinearity, especially with increasing loading, more distinctive geometry and greater deflections.
- For very widely spaced ribs shear lag effects will occur which are in the current model disregarded. The model is also approximate and for certain geometries results might deviate from exact solutions (see User recommendations).
- Rib spacing between two consective unit modules must be always constant and thus cannot be varying, i.e., $a_{x'} = \text{const}$, $a_{y'} = \text{const}$.
- Eccentric stiffness matrix terms are disregarded and set to zero.

Plate theory applicability – both Kirchhoff and Mindlin.

Analysis types applicable – I., II., and III. order in combination with geometrical nonlinearities only. Linear bending elastic static analysis uses the herein generated stiffness matrix coefficients. Membranes, second and third order analysis require as an additional parameter the equivalent thickness for self-weight computation d in order to form the geometrical sitffness matrix (also used in instability analysis), large displacement and rotation stiffnesses, membrane effects and soil–structure interaction. Material nonlinearity such as plasticity is not possible.

Material types applicable – Isotropic material only set via the *Edit Material* dialog and assigned to the plate via **Edit Surface** $\rightarrow$ **Material**.

Orthotropic direction and angle β – can be applied and manually set by the user via the *Edit Surface - Orthotropic* dialog.

Stiffness reduction factors – all types of stiffness reduction factors applicable, cf. Section 2.2.5, and also editable in the *Edit Surface - Orthotropic* dialog.

Equivalent thickness for self-weight computation – the value of d is computed according to

$$d = \frac{V}{a_{x'} a_{y'}} \tag{3.32}$$

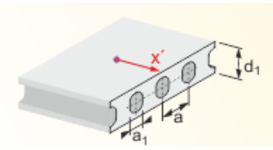
where V is defined in (3.30), and shown in the *Equivalent Thickness* read-in window under the *Unidirectional Ribbed Plate* tab in *Edit Surface - Orthotropic* dialog.

User recommendations

The bidirectionally ribbed slab has a common use in many structural applications, such as coffered slab floors, beam and slab bridge decks and other industrial platforms. Some recommendations include

- The model is generally recommended for a reasonably accurate modeling of concrete slab systems, reinforced eccentrically with an orthogonal system of intersecting ribs of rectangular cross-section. The ribs need not be geometrically equal, implying that the proposed model could also handle different x- and y-directional eccentricities.
- Due to the approximate stiffnesses, the magnitude of the error generally increases with rib spacing, rib depth and differences in eccentricity in both directions. The smaller the rib spacing and the rib depth, the more accurate the analysis results, due to the fact that in the limit as $d_{rx'} \to 0$, $d_{ry'} \to 0$, $b_{x'} \to a_{x'}$, and $b_{y'} \to a_{y'}$, the unidirectional ribbed model tends to revert back to the constant thickness isotropic material one, the analytical solution of which is indeed exact. General recommendations aiming at improving the analysis also include reducing the rib spacing in both directions, keeping the rib-to-plate height ratio low and choosing acceptable rib eccentricities which do not differ significantly in x' and y' directions.
- In cases when the rib spacing is greater than 13 times the top slab thickness, the effects of shear lag must necessarily be taken into account. In such cases, the RFEM model accuracy might deteriorate. Then a grillage model would suffice, or ideally a 3D model should reflect the latter accurately.
- If orthotropic material is to be necessarily used and provided that both ribs are relatively shallow and wide in comparison to the slab depth, the *Effective thickness* model is also possible to be used, cf. Section ??.
- Flexural rigidities in both directions, evaluated by means of standard methods for calculating the second moment of area about the centroid of the gross cross-section, have been proven to be accurate for this type of design and used by the theory employed. Therefore, no modification factors are here recommended and the default value of 1 should not be changed. The latter also applies to the torsional stiffness, which in light of the current theory used has been proven to yield realistic and accurate values.
- The employed theory is based on a mathematical model which considers the slab as a continuum system spanning over the ribs, recognizing the effects of the Poisson ratio in both slab and rib systems, and taking into account the limited contact area of intersection of the rib system. The theory, however, makes an assumption that the curvature in a direction orthogonal to one bending axis has as little as no influence to that bending stress, which is true in concrete structures, where the Poisson ratio is small and often negligible. Therefore, bidirectionally ribbed plates in RFEM are generally recommended to be used in combination with concrete materials.

3.2.4 Hollow Core Slab



Access

Accessing this model can be done from the *Edit Surface - Orthotropic* dialog and choosing *Hollow core slab* under Orthotropy Type on the left, see Figure 3.12.

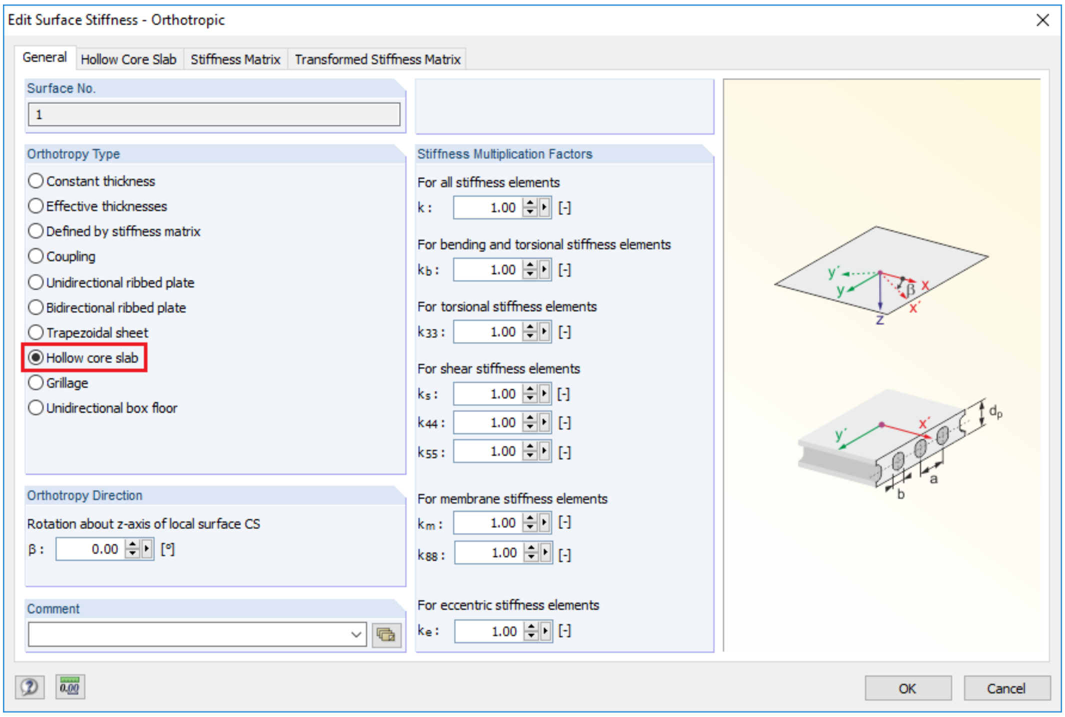


Figure 3.12: Accessing *Hollow core slab* material model via *Edit Surface Stiffness - Orthotropy* dialog

Description

The model represents a unidirectionally spanning constant thickness slab along the longitudinal direction of which circular, centreline symmetrical openings (voids) have been protruded as a measure of self-weight reduction.

The total slab depth is denoted by d_p, the void diameter is given by b and spacing, respectively, by a.

The following geometrical restrictions apply

1. $d_p > 0, b > 0, \quad a > 0$
2. $a > b, \quad d_p > b$

ensuring that a circular void is fully integrated within a unit module.

Stiffness matrix coefficients

The stiffness matrix coefficients are based on the theory Elliot and Clark [7], both of which came across empirical values, giving a good comparison with numerical and experimntal values. The interested users are referred to the articles in [8] for full derivations and further assumptions.

In here, we shall not focus on the theoretical background of the theory presented in [8], but rather define the required terms and parameters, appearing in our stiffness coefficients.

The stiffness matrix coefficients are given as

Bending Components	Shear Components	Membrane Components
$D_{11} = \dfrac{EI_{xx'}}{a(1-\nu^2)}$	$D_{44} = \dfrac{GA_x}{a\beta_{x'}}$	$D_{66} = \dfrac{EA_{x'}}{a(1-\nu^2)}$
$D_{22} = \dfrac{EI_{yy'}}{1-\nu^2}$	$D_{55} = \dfrac{GA_y}{\beta_{y'}}$	$D_{77} = \dfrac{EA_{y'}}{1-\nu^2}$
$D_{12} = \nu D_{22}$		$D_{67} = \nu D_{77}$
$D_{33} = GI_k$		$D_{88} = G(d_p - b)$

Table 3.6: Stiffness coefficients of orthotropy type *Coupling*

- $I_{xx'}$ = second moment of area of a unit x cross-section of void opening b and flange width equal to the voids spacing. Cross-section and dimensions are shown in Figure 3.13.

$$I_{xx'} = \frac{ad_p^3}{12} - \frac{\pi b^4}{64} \tag{3.33}$$

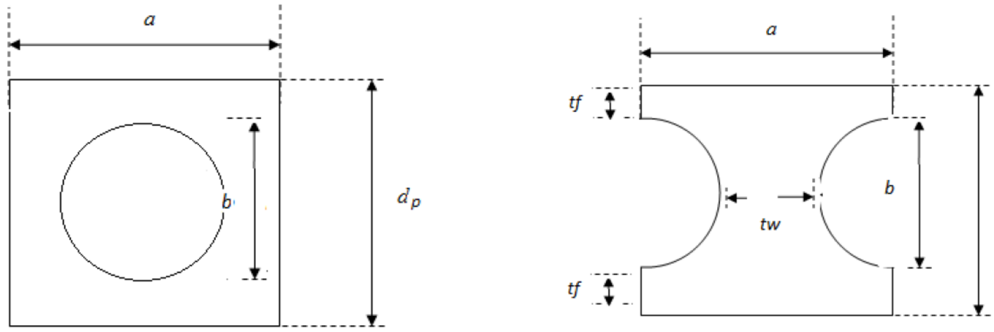


Figure 3.13: Cross-section in the longitudinal x − x direction

- $A_{x'}$ = fully uncrackled gross cross-sectional area of a unit cross-section of void opening b and flange width a, equal to the void spacing. Cross-section and other dimensions as shown in Figure 3.13.

$$A_{x'} = ad_p - \frac{\pi b^2}{4} \tag{3.34}$$

- $\beta_{x'}$ = Jurawski–Grashof coefficient of a unit x-cross-section of void openingb and flange width a, equal to the void spacing. Cross-section and other dimensions are shown in Figure 3.13.
- $I_{yy'}$ = second moment of area of an effective unit cross-section of width $1/(1-\nu^2)$ (unit width along the x-axis), taking into account the reduced inertia due to voids, expressed as

$$I_{yy'} = \frac{d_p^3}{12}\xi_b \tag{3.35}$$

  The parameter ξ_b is an empirical correction factor for bending proposed by Elliot and Clark, accounting for the continuously changing inertia along the y − y direction

$$\xi_b = \left[1 - 0.93\left(\frac{b}{d_p}\right)^{3.6}\right]\left[1 + 0.02\left(\frac{t_w}{t_f} - 2\right)\right] \tag{3.36}$$

  where $t_w = a - b$ and $t_f = (d_p - b)/2$, see Figure 3.13. Nevertheless, Elliot and Clark advice that the above formula could be further simplified on the basis that the web-to-flange thickness parameter, which reflects the ratio void-spacing-to-slab-depth, is not significant and a close approximation to the above equation could be further made, the range of error being maximum 4%, from the above formula as

$$\xi_b = \left[1 - \left(\frac{b}{d_p}\right)^4\right] \tag{3.37}$$

  This formula is used in RFEM to modify the transverse bending stiffnes, as already clarified.[1]

- $A_{y'}$ = area of the resulting unit cross-section in the y − y direction, see Figure 3.14,

[1] Practice reveals that any void opening for which $b \geq 0.6d_p$ needs to be definitely considered and accounted for, whereas for values less than that, ξ_b is allowed to be taken as unity. In cases when $b < 0.6d_p$ the cell distortion could be considered as too small as to affect the results significantly. RFEM does not follows this rule of thumb strictly and calculates ξ_b regardless how negligible the value becomes. As per Elliot and Clark, the empirical coefficients prove to yield satisfactory results for the b/d_p ratio between $0.47 < b/d_p \leq 0.81$, which covers most of the practical applications.

$$A_{y'} = 2\left(\frac{d_p - b}{2}\right) = d_p - b \tag{3.38}$$

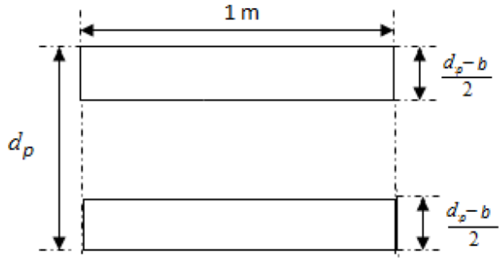


Figure 3.14: Unit cross-section in the y — y transverse direction

- $\beta_{y'}$ = Jurawski–Grashof coefficient of a unit cross-section along the y — y direction, see Figure 3.14,

$$\beta_{y'} = \frac{6}{5} = 1.2 \tag{3.39}$$

- I_k = torsional stiffness of the voided slab, represented by the stiffness of a solid slab

$$I_k = \frac{d_p^3}{12}\xi_T \tag{3.40}$$

modified by an empirical factor ξ_T, proposed by Elliot and Clark, accounting for the continuously changing torsional stiffness

$$\xi_T = \left[1 - 0.82\left(\frac{b}{d_p}\right)^{3.9}\right]\left[1 + 0.026\left(\frac{t_w}{t_f} - 2\right)\right] \tag{3.41}$$

Nevertheless, Elliot and Clark advice that the above formula could be further simplified on the basis that the web-to-flange thickness parameter, which reflects the ratio void-spacing-to-slab-depth, is not significant and a close approximation to the above equation could be further made, the range of error being maximum 4%, from the above formula, as

$$\xi_T = \left[1 - 0.85\left(\frac{b}{d_p}\right)^4\right] \tag{3.42}$$

This formula is used in RFEM to modify the transverse torsional stiffness, as already clarified.

Model limitations

- Orthotropic material is not allowed to be used. This model allows the use of only linear elastic isotropic material. If concrete is the representative slab material, a Poisson ratio of 0.2 is recommended.
- Material remains fully elastic and fully uncracked, i.e., both tension and compression zones remain fully active. If concrete is to be used, we should regard this state as Concrete State I. Reinforcement effects are ignored, but could be approximately taken into account via the stiffness matrix modification factors.
- Only circular voids, symmetrically located about the statical neutral axis of the slab are allowed.
- Variable void spacing is not allowed. It is recommended that users should aim at limiting void diameter to plate depth ratios to $0.47 < b/d_p \leq 0.81$ in order to achieve a reasonable level of accuracy. If, however, greater openings are present, the hollow core slab model turns into a cellular slab model, where cell bending and distortion must be taken into account. Such a model shall be introduced in later versions of RFEM.

- Transverse shear D_{55} and transverse membrane components D_{77} are calculated on the assumption that top and bottom slabs will resist all shear and in-plane effects, respectively. Any cell wall effects have been disregarded in those stiffness matrix coefficients. This is due to the fact that there are integrated voids, making geometry discontinuous and not fully efficiently contributing towards resisting the applied in-plane forces.
- Please note that the anticlastic plate effect has been considered in the membrane terms D_{66}, D_{67} and D_{77} as the voided slab is likely to exibit it due to its two continuous top and bottom plates. The internal walls, however, and their contribution to that have been disregarded. Even though they have a strenghtening effect, due to their discontinuous nature the effect is likely to be very small. This approach is accurate enough for design purposes and also conservative, which guarantees safe design values.

Plate theory applicability – both Kirchhoff and Mindlin.

Analysis types applicable – I., II., and III. order in combination with geometrical nonlinearities only. Linear bending elastic static analysis uses the herein generated stiffness matrix coefficients. Membranes, second and third order analysis require as an additional parameter the equivalent thickness for self-weight computation d in order to form the geometrical sitffness matrix (also used in instability analysis), large displacement and rotation stiffnesses, membrane effects and soil–structure interaction. Material nonlinearity such as plasticity is not possible.

Material types applicable – Isotropic material only.

Orthotropic direction and angle β – can be applied and manually set by the user via the *Edit Surface - Orthotropic* dialog.

Stiffness reduction factors – all types of stiffness reduction factors applicable, cf. Section 2.2.5, and also editable in the *Edit Surface - Orthotropic* dialog.

Equivalent thickness for self-weight computation – the value of d is automatically computed by RFEM as

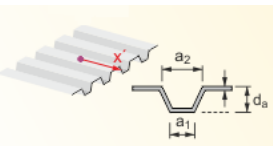
$$d = \frac{A_{x'}}{a} \tag{3.43}$$

It can be also user-editable via the d or wt options in the *Edit Surface* dialog box.

User recommendations

The voided slab model is quite often used in orthotropic bridge deck analyses or industrial applications, where reasonable strength at expense of weight is required. We recommend using this model with caution as it is important that all circular voids are suffieciently closely spaced in order to ensure correct numerical results.

3.2.5 Trapezoidal Sheet



Access

Accessing this model can be done from the *Edit Surface - Orthotropic* dialog and choosing *Trapezoidal sheet* under Orthotropy Type on the left, see Figure 3.15.

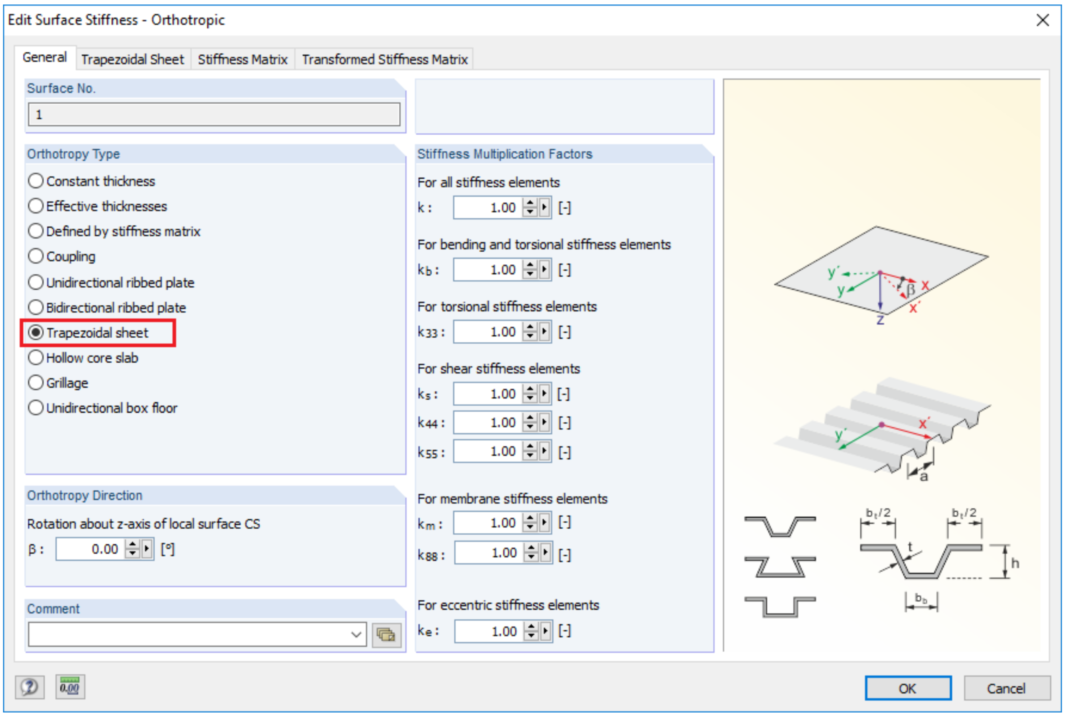


Figure 3.15: Accessing *Trapezoidal sheet* material model via *Edit Surface Stiffness - Orthotropy* dialog

Description

This model enables us to approximate the stiffnesses of a unidirectionally spanning corrugated sheeting with trapezoidal shaped ribs. The geometrical parameters are given below

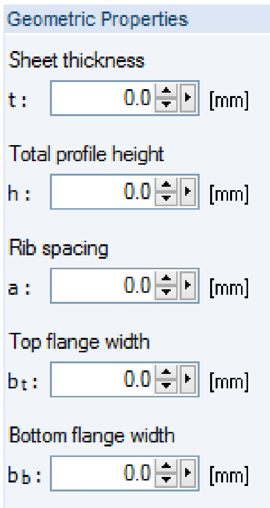


Figure 3.16: Geometrical input parameters

The following geometrical restrictions apply to the profile geometry

1. $t > 0, h > 0, \quad a > 0, \quad b_t > 0, \quad b_b > 0$
2. $h > 2t$
3. $a \geq \max\{b_t, b_b\}$

Note that upon setting the geometrical dimensions, 3 types of structurally meaningful cross-sections may exist, necessarily applying to the above imposed restrictions

1.

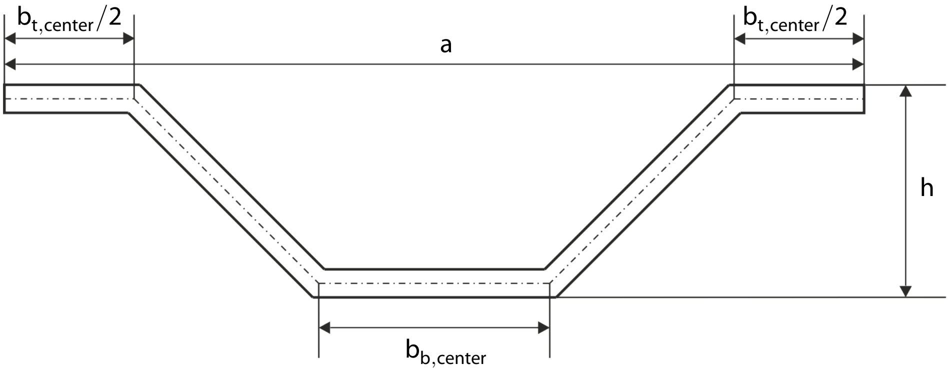


Figure 3.17: Shape of the trapezoidal sheet when $a > b_{t,center} + b_{b,center}$

2.

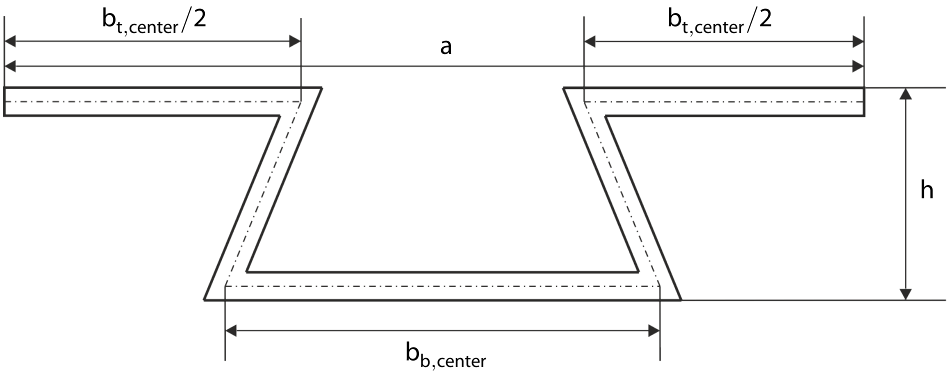


Figure 3.18: Shape of the trapezoidal sheet when $a < b_{t,center} + b_{b,center}$

3.

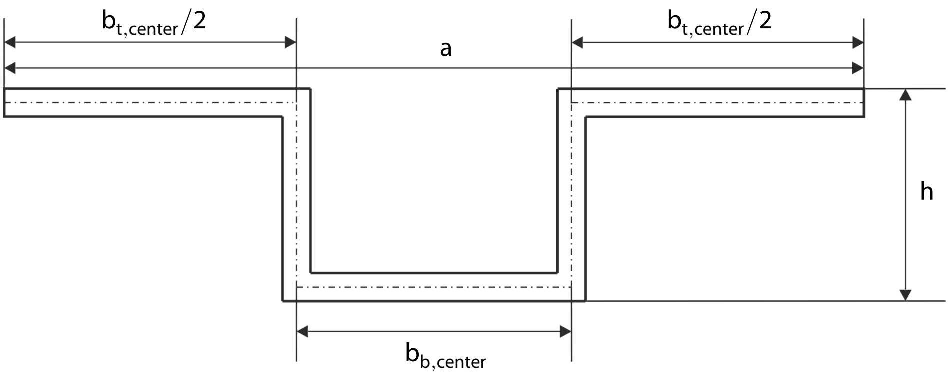


Figure 3.19: Shape of the trapezoidal sheet when $a = b_{t,center} + b_{b,center}$

Stiffness matrix coefficients

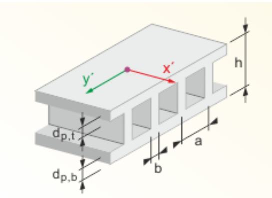
The Kirchhoff-plate coefficients of the stiffness matrix are based on the Equivalent (Strain) Energy and Equivalent Force Method, cf. [9], namely, that the strain energy, or the internal forces and moments, respectively, of the resulting orthotropic plate should be equal to their average counterparts of a representative volume element of the original trapezoidal sheet – see Figures 3.17, 3.18, 3.19, i.e., the profile that is assumed to be periodically extented. The shear stiffness coefficients are approximated by the stiffness of a planar sheet.

The stiffness matrix coefficients are given as

Bending Components	Shear Components	Membrane Components
$D_{11} = \dfrac{EI_{xx'}}{a(1-\nu^2)}$	$D_{44} = \dfrac{5}{6} G \dfrac{2I_x}{a} t$	$D_{66} = \dfrac{EA_{x'}}{a(1-\nu^2)}$
$D_{22} = \dfrac{EI_{yy'}}{1-\nu^2}$	$D_{55} = \dfrac{5}{6} Gt$	$D_{77} = \dfrac{EA_{y'}}{1-\nu^2}$
$D_{12} = \nu D_{22}$		$D_{67} = \nu D_{77}$
$D_{33} = GI_k$		$D_{88} = G(d_p - b)$

Table 3.7: Stiffness coefficients of orthotropy type *Coupling*

3.2.6 Unidirectional Box Floor



Access

Accessing this model can be done from the *Edit Surface - Orthotropic* dialog and choosing *Unidirectional box floor* under Orthotropy Type on the left, see Figure 3.20.

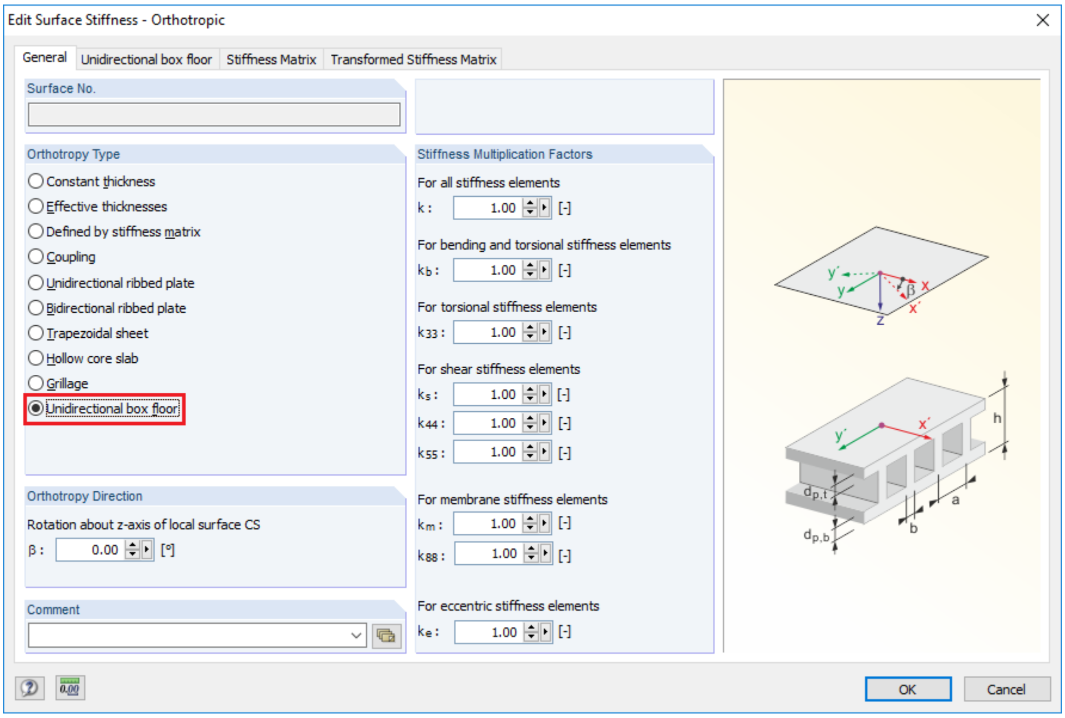


Figure 3.20: Accessing *Unidirectional box floor* material model via *Edit Surface Stiffness - Orthotropy* dialog

Description

The model represents a unidirectionally spanning constant thickness slab along the longitudinal direction of which rectangular openings (voids) have been protruded as a measure of self-weight reduction.

The geometric properties for this model, which are to be found under the *Unidirectional box floor* tab in the *Edit Surface - Orthotropic material* dialog, have the following format

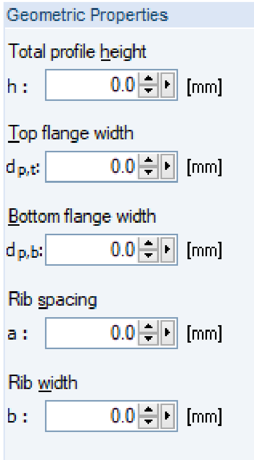


Figure 3.21: Dialog section *Geometric Properties*

The following geometrical restrictions apply

1. $\mathrm{h} > 0, \quad \mathrm{d}_{\mathrm{p,t}} > 0, \quad \mathrm{d}_{\mathrm{p,b}} > 0, \quad \mathrm{b} > 0, \quad \mathrm{a} > 0$
2. $\mathrm{d}_{\mathrm{p,t}} + \mathrm{d}_{\mathrm{p,b}} < \mathrm{h}$

ensuring that a rectangular void is fully integrated within a unit cell (I-beam cross-section).

Stiffness matrix coefficients

The stiffness matrix coefficients are given as

Bending Components	Shear Components	Membrane Components
$D_{11} = \dfrac{EI_{xx'}}{a(1-\nu^2)}$	$D_{44} = \dfrac{GA_{x'}}{a\beta_{x'}}$	$D_{66} = \dfrac{EA_{x'}}{a(1-\nu^2)}$
$D_{22} = \dfrac{EI_{yy'}}{1-\nu^2}$	$D_{55} = \dfrac{GA_{y'}}{\beta_{y'}}$	$D_{77} = \dfrac{EA_{y'}}{1-\nu^2}$
$D_{12} = \nu D_{22}$		$D_{67} = \nu D_{77}$
$D_{33} = GI_k$		$D_{88} = G(d_{p,t} + d_{p,b})$

Table 3.8: Stiffness coefficients of orthotropy type *Unidirectional box floor*

- $A_{x'}, A_{y'}$ = gross cross-sectional area in the correcponding direction, namely

$$A_{x'} = a \cdot (d_{p,t} + d_{p,b}) + b \cdot (h - d_{p,t} - d_{p,b}), \quad A_{y'} = 1 \cdot (d_{p,t} + d_{p,b}) \tag{3.44}$$

- $I_{xx'}, I_{yy'}$ = second moments of area of the corresponding unit I-cross-section
- I_k = torsional constant of the corresponding unit I-cross-section approximated as the sum of the torsional constant of the two flanges and the web, cf. (3.10)–(3.11)
- $\beta_{x'}, \beta_{y'}$ = Jurawski–Grashof shear coefficients of the corresponding unit I-cross-section

Model limitations

- Orthotropic material is not allowed to be used. This model allows the use of only linear elastic isotropic material. If concrete is the representative slab material, a Poisson ratio of 0.2 is recommended.
- Material remains fully elastic and fully uncracked, i.e., both tension and compression zones remain fully active. If concrete is to be used, we should regard this state as Concrete State I. Reinforcement effects are ignored, but could be approximately taken into account via the stiffness matrix modification factors.

- Only circular voids, symmetrically located about the statical neutral axis of the slab are allowed.
- Variable void spacing is not allowed.

Plate theory applicability – both Kirchhoff and Mindlin.

Analysis types applicable – I., II., and III. order in combination with geometrical nonlinearities only. Linear bending elastic static analysis uses the herein generated stiffness matrix coefficients. Membranes, second and third order analysis require as an additional parameter the equivalent thickness for self-weight computation d in order to form the geometrical sitffness matrix (also used in instability analysis), large displacement and rotation stiffnesses, membrane effects and soil–structure interaction. Material nonlinearity such as plasticity is not possible.

Material types applicable – Isotropic material only.

Orthotropic direction and angle β – can be applied and manually set by the user via the *Edit Surface - Orthotropic* dialog.

Stiffness reduction factors – all types of stiffness reduction factors applicable, cf. Section 2.2.5, and also editable in the *Edit Surface - Orthotropic* dialog.

Equivalent thickness for self-weight computation – the value of d is automatically computed by RFEM as

$$d = \frac{A_{x'}}{a} \tag{3.45}$$

It can be also user-editable via the d or wt options in the *Edit Surface* dialog box.

4 Membranes in RFEM 5

Lorem ipsum dolor sit amet, consectetur adipiscing elit. Quisque eget lectus nec diam euismod eleifend. Praesent tincidunt iaculis nulla, sed laoreet metus bibendum a. In cursus eros nec eros sodales dictum. Ut rutrum convallis turpis, a luctus arcu iaculis eu. In hac habitasse platea dictumst. Sed maximus turpis maximus, hendrerit mi nec, consectetur nulla. Donec pellentesque diam ultricies, porttitor ex nec, hendrerit nulla. Nulla eget eleifend turpis. Suspendisse sagittis auctor neque, et porta sapien ultrices a. Praesent congue arcu leo, eu maximus dolor aliquam auctor. Vestibulum ac blandit tellus. Suspendisse dapibus a leo a ultricies. Sed ac viverra leo. Pellentesque blandit nibh quis hendrerit cursus. Nullam sit amet sodales leo, eu congue metus. In at turpis fermentum, interdum sapien vitae, accumsan sapien.

Fusce vulputate augue at nunc posuere vestibulum. In at lectus sed purus interdum rhoncus. In lobortis cursus nisi. Curabitur sit amet pharetra diam. Pellentesque eu ligula sagittis, posuere libero eget, rutrum metus. Suspendisse potenti. Aenean pulvinar bibendum tellus, id rhoncus quam ultricies mattis. Cras mi augue, hendrerit sed orci at, fermentum molestie arcu.

Etiam mattis, tortor vel dignissim eleifend, turpis metus sollicitudin odio, in suscipit erat lacus sed mi. In hendrerit ante non velit dignissim, non placerat sapien ornare. Mauris at tortor pretium, euismod odio non, viverra enim. Nam id odio mi. Phasellus facilisis vel ligula et maximus. Cras convallis et nulla eget mollis. Aliquam tempor placerat enim, eu fringilla sem blandit eget. Nulla ultricies vestibulum velit quis imperdiet. Maecenas porttitor tellus eu venenatis sagittis. In porta faucibus sollicitudin.

Donec in magna consequat orci vulputate hendrerit quis vel nibh. Duis non massa et ex aliquet pulvinar ut quis nibh. Nulla efficitur viverra placerat. Etiam scelerisque iaculis imperdiet. Donec dignissim ipsum at accumsan fermentum. Vivamus tincidunt convallis suscipit. Mauris vitae est tellus. In nec eros porta, feugiat massa sed, malesuada dui. Etiam sit amet efficitur lectus.

Etiam laoreet commodo pulvinar. Phasellus vehicula purus in libero rhoncus commodo. Integer non aliquam urna. Proin nec posuere eros. Fusce at sapien rhoncus, venenatis tortor vel, viverra justo. Duis urna enim, accumsan eget orci non, laoreet accumsan orci. Interdum et malesuada fames ac ante ipsum primis in faucibus. Pellentesque nec nisl odio. Proin quis efficitur risus. Curabitur nunc neque, aliquet eget congue vel, blandit non ipsum. Maecenas lobortis nulla eget nisl efficitur, id dapibus magna aliquet. Quisque malesuada dolor at felis accumsan rutrum. Pellentesque porttitor, dui ac rutrum commodo, mauris libero luctus lacus, nec scelerisque tortor nisl ut mi. Donec molestie dapibus elit ut vehicula.

Literature

[1] S. Timoshenko and S. Woinowski-Krieger. *Theory of Plates and Shells*. McGraw-Hill, New York, 2. edition, 1959.

[2] S. Timoshenko and J. N. Goodier. *Theory of Elasticity*. McGraw-Hill Book Company, Inc., New York, 2. edition, 1951.

[3] W. C. Young and R. G. Boudynas. *Roark's Formulas for Stress and Strain*. McGraw-Hill, New York, 7. edition, 2002.

[4] A. R. Cusens, M. A. Zeidan and R. P. Pama. Elastic Rigidities of Ribbed Plates. *Building Science*, 7:23–32, 1972.

[5] E. Giencke. Die Grundgleichungen für die orthotrope Platte mit excentrischen Streifen. *Der Stahlbau*, 24:128, 1955.

[6] F. Sawko and R. J. Cope. Analysis of Multi-Cell Bridges Without Transverse Diaphragms - a Finite Element Approach. *The Structural Engineer*, 47:55–60, 1969.

[7] G. Elliot and L. A. Clark. Circular Voided Concrete Slab Stiffnesses. *Journal of the Structural Division*, 108:2379–2393, 1982.

[8] S. G. Lekhnitskii. *Anisotropic Plates*. Gordon and Breach Science Publishers, New York, 1968.

[9] Y. Xia, M. I. Friswell and E. I. Saavedra Flores. Equivalent models of corrugated panels. *Int. J. Solids Struct.*, 49:1453–1462, 2012.

Index