SCIAENGINEER

Barco One Campus - © Image Jaspers-Eyers Architects – photography by Marc Detiffe

Advanced Concept Training
FEM

All information in this document is subject to modification without prior notice. No part of this manual may be reproduced, stored in a database or retrieval system or published, in any form or in any way, electronically, mechanically, by print, photo print, microfilm or any other means without prior written permission from the publisher. SCIA is not responsible for any direct or indirect damage because of imperfections in the documentation and/or the software.

Table of contents

Introduction

All discussed topics are available in the **Concept Edition** of SCIA Engineer, unless it is explicitly mentioned for a certain specific topic.

As an introduction, some basic rules for good use of fem software are given:

- Do not start too complex. It is better to draw up a coarse model first and to refine it afterwards. From the coarse model many primary conclusions can be already drawn to simplify the rest of the course of the modelling.

- In many cases the Finite Element mesh is too coarse in a specific detail area to obtain exact results. Instead of trying to refine the mesh in such an area, it is mostly advisable to draw up a sub model of the detail.

- Drawing up a sub model is based on the Saint-Venant principle that indicates that if the real force distribution is replaced by a static equivalent system, the stress distribution is only influenced in the direct environment of the point of application of the forces. More specific: this means that if the edges of the sub model are removed far enough from the stress concentrations that you want to examine, the sub model gives reliable results.

- Restrict the structure type to the necessary. It is not always necessary to model a 3D structure. A 2D environment can provide just as good results in a quicker and simpler way. Especially the restriction of the number of degrees of freedom can lead to fewer problems with the calculation.

- If possible, use symmetry to restrict the calculation model in size.

- Always apply/test new functionalities or special techniques to a small project, and apply it only afterwards on the real complex project.

- Always calculate the structure after modelling, loaded with the self-weight. The other loads can only be imported when no problems were encountered.

- Always consider the compliances of the complete construction when having an instability/singularity. If the degrees of freedom are obstructed for the entire structure according to the construction type, only then have a look at the members.

- After calculation:
    - Checking if Σ(loads) equals Σ(reactions)
    - Checking the reaction forces
    - Checking if the moment diagram progresses as expected
    - Checking if the structure is deformed as expected

- If possible, always perform a short manual calculation to verify or estimate the order of magnitude of the results.

The elements in SCIA Engineer

Type of Element – Solver

The solver of SCIA Engineer uses the same element for plates as for the bending behaviour of shell members. Analogously, the same element is used for walls and for their functioning, namely the wall inner forces.

In a General XYZ environment there are 6 degrees of freedom for each node. Physically, these 6 degrees of freedom represent the following: the displacements u, v, w and the rotations φ_x, φ_y, φ_z. The components of displacement are given in the local axis of the element. So u, v, φ_z represent the plane stress/strain state, w, φ_x, φ_y indicate bending/shear force.

The element used in SCIA Engineer for the calculation of membrane forces includes a 3-nodes triangle and a 4-nodes quadrilateral with 3 degrees of freedom per node.
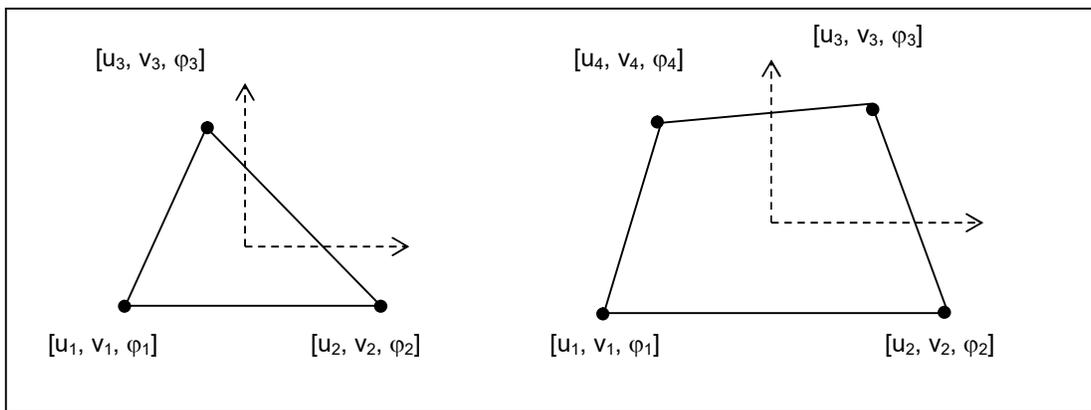
Regarding the application of these elements, the following advantages can be given:

- The structure is calculated more accurately.

- The 2D elements can be connected to the 1D elements in the nodes thanks to their rotational degree of freedom.

For more information see ref. [5].
A detailed description of the element used for bending/shear force is given in ref. [7].

Because of the following properties of the solver in SCIA Engineer, an optimal result is obtained for the finite elements calculation (ref. [6]):

- Implementation of the algorithm for interpolation on edges.
  This provides an accurate approach of the shear stresses on an edge.

- Great precision concerning the calculation of internal forces in ribs.

- Use of the model for line supports on 2D member edges: this method introduces a good representation of the intensity with the guarantee of continuity of the stresses on the intersections of crossing line supports.

- By optimizing of the algorithms and by rewriting some parts of the code with respect to earlier versions (before version 5.0), there is a gain of speed of approximately 15%.

For the bending behaviour there are 2 types of elements implemented:

- The **Mindlin element** including shear force deformation

- The **Kirchhoff element** without shear force deformation

The difference between a Kirchhoff and a Mindlin calculation will be discussed in Chapter 9.
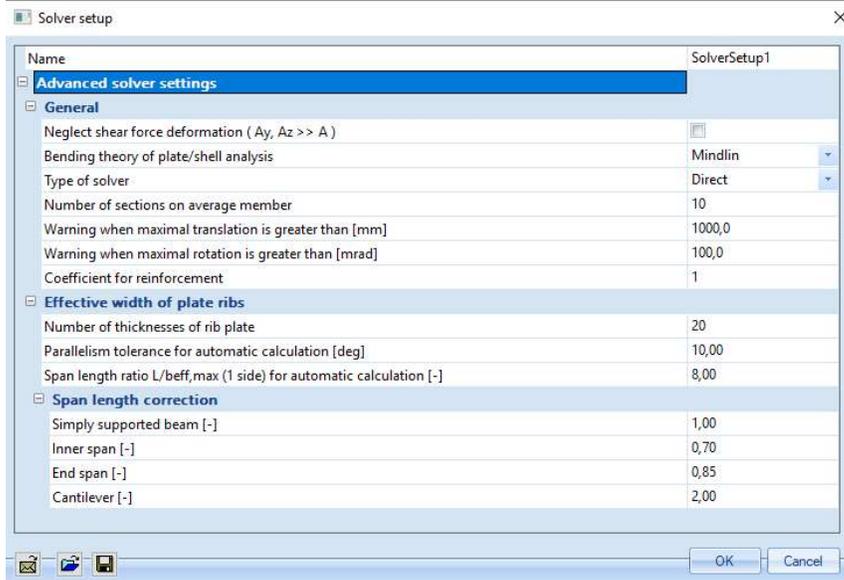
Local axis of a 2D element

Each 2D finite element has a local axis. The results (internal forces, stresses) are represented according to these axes and the loads can be introduced according to them.

The local coordinate system can be drawn on each element:
Set view parameters for all > Structure > Local axes > Members 2D

For projects with **Plate XY** and **Wall XY**, the local axis is identical to the global axis.
The following explanation relates to **General XYZ** projects.

The local coordinate system is a right-handed Cartesian coordinate system. The origin of the axis lies in the starting point I of the 2D mesh element. The results are independent of the choice of starting point I. The generator of the finite element mesh defines the begin node of each mesh element automatically.

The z-axis is perpendicular to the plane of the 2D mesh element and the orientation is defined as follows:

When an observer looks in the direction (resp. in opposite direction) of the z-axis, the description of the nodes I, J, K, L of a mesh element is clockwise (resp. counter clockwise).

The direction of rotation of a 2D finite element corresponds to that of the 2D member to which the finite element takes belongs. The direction of rotation of a 2D member depends on the direction in which the nodes are established while introducing the geometry.

The user can modify the direction of the local z-axis after input of the geometry. This can be useful to simplify the input of the loads. Select the 2D member and check the option **LCS Z axis Swap orientation.** Pay attention to the local axes of the mesh elements, that they will not rotate with this option.

The local x- and y-axes are located in the reference plane of the 2D mesh element (the middle plane) and since the coordinate system is always orthogonal, it is sufficient to define the local x-axis:

The local x-axis is the intersection curve of a horizontal surface through the origin of the global axis and the surface of the 2D mesh element. Its positive orientation is defined in such a way that the angle between the +x and +X-axis is acute.

There are two cases for which above-mentioned definition is inadequate:

1. If the local x-axis is perpendicular to the X-axis, the criterion of the acute angle is not valid anymore.
2. When the element plane and the XY surface are parallel, a cross-section cannot be defined.

The rule for the definition of the local x-axis in case of these exceptions is as follows:
1. The orientation of the local +x-axis corresponds to the orientation of the +Y-axis. This criterion replaces the criterion of the acute angle.
2. The +x and +X axes are identical.

EXAMPLE: 01_01_Axis.esa

In this example, a sloping plate is introduced as a shell element to indicate the position of the local axis according to the above-mentioned rules.

The numeration of the nodes is given clockwise when you look in the direction of the local z-axis.

> The intersection line of a horizontal plane through the origin with the plane of the 2D member represents a straight line according to line **K1** and **K2.** This means that the local x-axis is perpendicular to the global X-axis so the criterion of the acute angle is not valid. As a result, the orientation of the local +x-axis corresponds to the orientation of the +Y-axis.
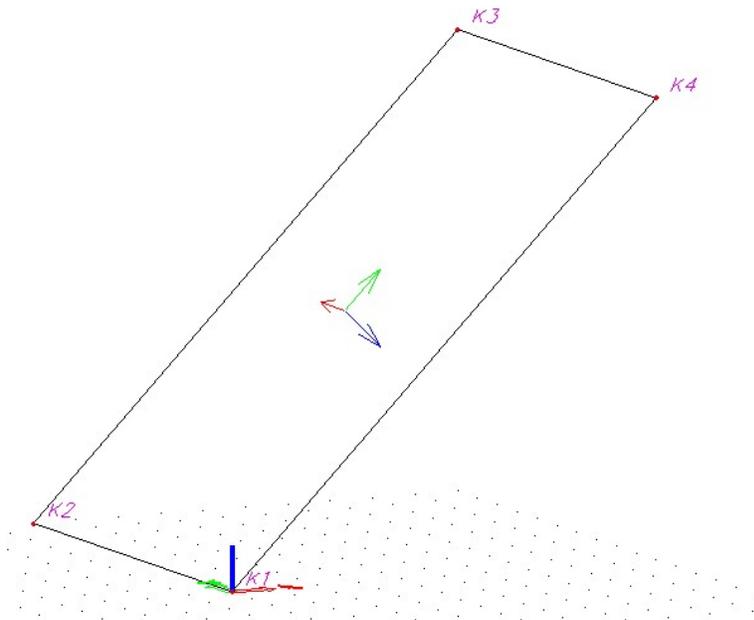
Since version SCIA Engineer 2011 the user has the possibility to set the direction of the local system axis in following ways:

0. x(y) automatic (same as in SCIA Engineer 2010.1)

1. x(y) parallel to a given plane

2. x(y) concurrent with a given line.
It leads from the center of the element to the cross of the element plane and the line.

3. x(y) as a tilt of a vector defined by a vector.
It is a vector, lying in the element plane, which has the smallest angle with the given vector)

4. x(y) as a tilt of a vector defined by a given point.
It is a vector, oriented from the element center to the given point, will be tilted to the element plane.

5. x(y) as a tilt of a vector defined by a given line.
It is a vector, starting at the element center and perpendicular to the given line will be tilted to the element plane.

The user can interact and swap the local z direction on all 6 methods.
Points and vectors required to define the new types of local system axis are set by the user in the properties of the 2D member.
- The local system of the type 1 and 3 is defined by one vector.
- The local system of type 4 is defined by one point.
- The local system of the type 2 and 5 is defined by two points.

EXAMPLE: 01_02_User defined LCS.esa

In this example the use of a user defined LCS with local x-axis pointing to a given point is illustrated.
There are two cones. The cone at the left has standard definition of local axis. On the other cone, the LCS is defined by a tilt of a vector defined by the top of the cone.
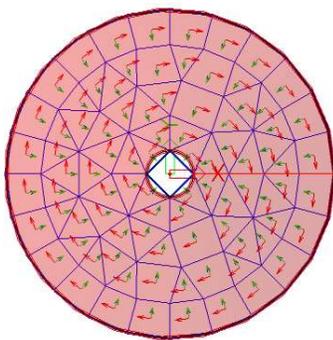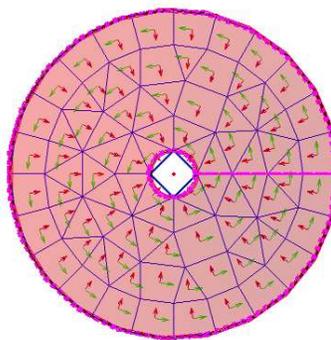
Standard method   tilt of vector defined by point
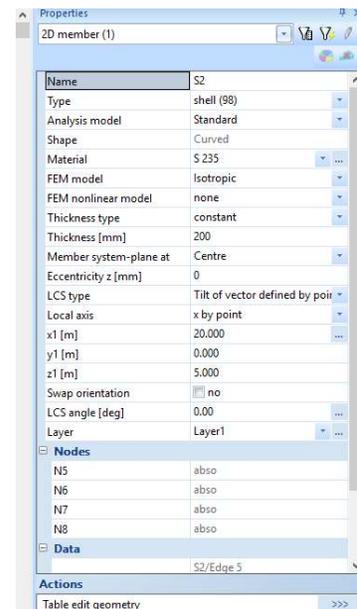
Mesh generator and mesh refinements

With **Calculation, Mesh > Mesh setup** the size of the mesh elements and the general setup of the mesh generator can be changed. The adopted dimension of the mesh for 2D members is valid for the entire structure.

By means of **Calculation, Mesh > Local mesh refinement** a local mesh refinement can be introduced on a specific node, 2D member edge or on a surface. That way for example, a coarse distribution can be adopted on the global structure and a more refined mesh for a specific 2D element.

General setup with **Calculation, Mesh > Mesh setup:**

| Mesh setup	☒
Name	MeshSetup1
Average number of tiles of 1d element	1
Average size of 2d element/curved element [m]	2,000
Advanced mesh settings	
General mesh settings	
Minimal distance between two points [m]	0,001
Definition of mesh element size for panels	Automatic
Average size of panel element [m]	1,000
Hanging nodes for prestressing	✓
1D elements	
Minimal length of beam element [m]	0,100
Maximal length of beam element [m]	100,000
Average size of cables, tendons, elements on subsoil, nonlinear soil spring [m]	1,000
Generation of nodes in connections of beam elements	✓
Generation of nodes under concentrated loads on beam elements	✓
Generation of eccentric elements on members with variable height	☐
Division on haunches and arbitrary members	5
Division for 2D-1D upgrade	50
Mesh refinement following the beam type	None
2D elements	
Maximal out of plane angle of a quadrilateral [mrad]	30,0
Predefined mesh ratio	1,5

Average number of tiles of 1d element
If necessary, more than one finite element may be generated on a single beam.
The value here specifies how many finite elements should be created on the beam.

When dividing a beam into several tiles, then also the value for **Minimal length of beam element** will be considered.

Average size of 2D element/curved element [m]
The average size of an edge of a 2D element. This size may be changing when refining the mesh in specified points.
This option also defines the magnitude of finite elements generated on curved 1d (!) elements.

General mesh settings

Minimal distance between two points [m]
If the distance between two mesh nodes is less than the value specified here, then these two points are automatically merged into one single point. This option applies for both 1D and 2D elements.

Definition of mesh element size for panels
If the mesh size for load panels is set to Automatic, then SCIA Engineer will use a mesh size of 1 meter. If this value is set to Manual, then the user can specify the average mesh size in the next field.

Average size of panel elements [m]
Defines the average size of mesh elements for load panels (when the definition is set to manual).

Hanging nodes for prestressing
If this option is ON, then this allows to connect a prestress cable to a slab when the prestress cable is drawn above or below the central plane of the slab without having to define an internal line in the slab.
If this option is OFF, then the prestress cable must always be defined on an internal line of the slab.

1D Elements

Minimal length of beam element [m]
A 1D element cannot become shorter than this value. This value supersedes the parameter above (**Average number of tiles of 1D element**).

Maximal length of beam element [m]
A 1D element cannot become longer that this value. If a beam of the structure is longer than this value, then this beam will be divided into multiple mesh elements to satisfy the condition of maximal length.

Average size of cables, tendons, elements on subsoil, nonlinear soil springs
To obtain precise results, it is necessary to generate a sufficient dense element mesh on cables, tendons (prestressed concrete), 1d elements on subsoil and when using nonlinear soil springs on 1d elements.

Generation of nodes in connections of beam elements
If this option is ON, a check for "touching" beams/FE node and FE node/FE node is performed.
This means:

-    if a FE node of one beam "touches" another beam in a point where there is no node, then the two beams are connected by a FE node.

-    If an FE node of one beam coincides with an FE node of another beam, then these 2 FE nodes become 1 FE nodes, thus connecting both beams

If the option is OFF, such a situation remains unsolved and the beams are not connected to each other.

The function has the same effect as performing the function **Check of data**.

Be aware: when you mesh 2 wind bracings with an even number of tiles, then they will automatically connect because they will have a common FE node. It is always recommended to use an odd number of tiles when meshing 1D members.

Generation of nodes under concentrated loads on beam elements
If this option is ON, mesh nodes are generated in points where the concentrated load is acting.

Generation of eccentric elements on members with variable height
If this option is ON, eccentric elements are generated on beams with a haunch on beams with variable height. The finite element axis is no longer in the axis of the original beam, but it follows the centre of the generated cross-sections along the haunch. (example)

Division on haunches and arbitrary members
This specifies the number of mesh elements generated on a haunch. This option prescribes the precision of the modelling. The larger the number, the better the model approaches the reality. (example).

Division for 2D-1D upgrade
This specifies the number of mesh elements generated on the (resulting) 1D member when doing a 2D-1D upgrade.

Mesh refinement following the beam type
This specifies if the nodal mesh refinements should also be applied to beam members.

The nodal mesh refinement is represented by a volumetric sphere. As a consequent, the mesh of all the structure elements situated in this sphere will be refined taking the following possibilities into account:
None
The refinement is not applied to 1D members (so, it will only be applied to 2D members).
Beams and Columns
The refinement is applied only to 1D members with Type set to 'Beam (80)' or 'Column (100)'.
All members
The refinement is applied to all 1D members.

2D elements

Maximal out of plane angle of a quadrilateral element
This value determines whether a spatial quadrilateral element whose nodes are not in one plane will be replaced by two triangular elements. This parameter is only meaningful for out-of-plane surfaces – shells. The assessed angle is measured between the plane made by the first three nodes of the quadrilateral and the remaining node of this quadrilateral.

Predefined mesh ratio
Defines the relative distance between the predefined mesh formed by regular quadrilateral elements and the nearest edge. The edge may consist of an internal edge, external edge or border of refined area. The final distance is calculated as a multiple of the defined ratio and adjusted average element size for 2D elements.

EXAMPLE: 02_01_SUBSOIL.esa

In this example a HEA200 with steel quality S235 is calculated according to the EC. The beam has a length of 10 m and is supported by means of a foundation strip with a stiffness of 50MN/ m². At the ends and in the middle the beam is loaded by point forces of 50 kN.

Successively, the number of elements of a beam on subsoil has increased to evaluate the influence of the mesh refinement on the results. The results are considered in the middle of the beam.

Average Size [m]	# 1D Elements	Uz [mm]	My [kNm]
10	2	-0,20	-0,18
5	3	-0,08	-0,37
1	11	-0,48	3,31
0,5	21	-0,57	5,98
0,25	41	-0,61	8.07
0,1	101	-0,63	9,63

It shows clearly that a mesh refinement has a significant influence on the results.

In this case, it is important to realize that a standard mesh, satisfactory for normal beams, is insufficient for beams on a subsoil. For this reason, SCIA Engineer allows to refine the mesh particularly for this type of beams.

EXAMPLE: 02_02_HAUNCH.esa

A haunch imagines a structural element with a variable height. However, a 1D mesh element has a constant height. Because of this, it is necessary to design these beams with a variable height by means of a finer mesh.

In this example a haunch with a length of half the beam is considered. This beam has a width of 100 mm, a length of 4 m and concrete quality C25/30 according to the EC. The height of the haunch varies between 1000 mm and 200 mm. The whole is loaded in the middle by a point force of 250 kN.

If for example the number of finite elements per haunch is putted on 2, then the haunch is divided into two segments during the mesh generation. Subsequently the section in the middle of each segment is determined. This segment will be used for the finite element.
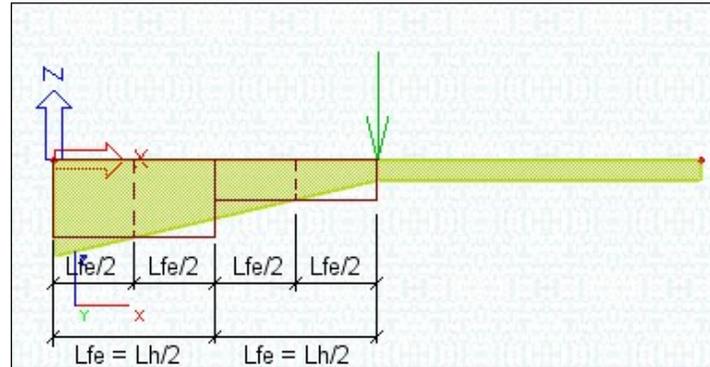

The number of finite elements per haunch is increased and the influence on the global maximal deflection of the beam is analyzed:

| # EE per haunch	Uz [mm]
2	-45,89
5	-48,40
10	-49,69
20	-50,15
50	-50,29

The results show that a higher number of elements will approach the real haunch better. On the other hand, the number of elements is not directly in proportion to the precision. This example shows also that using about 10 mesh elements is already sufficient to receive an accurate result.
So, it is not necessary to use an extremely high number of elements.

EXAMPLE: 02_03_ECCENTRIC_ELEMENTS.esa

In the following example a frame XZ is constructed by means of the catalogue blocks. Next, an eccentricity is given to the left column by changing the **member system-line**, namely by choosing the type '**under**'.
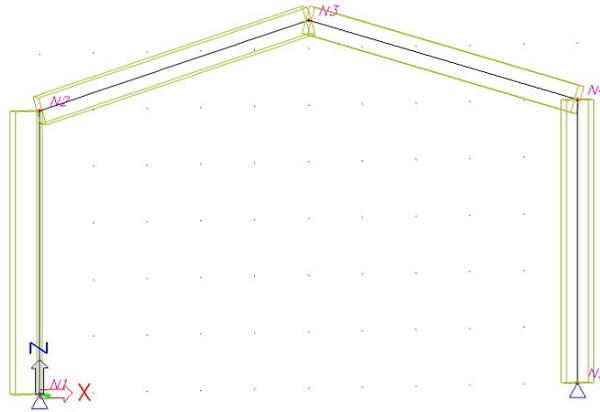
Both columns are hinged, so that you do not expect moments in the end nodes. When the project is calculated, the results proved to be asymmetrical. What's more, the moment in node N1 differs from 0.
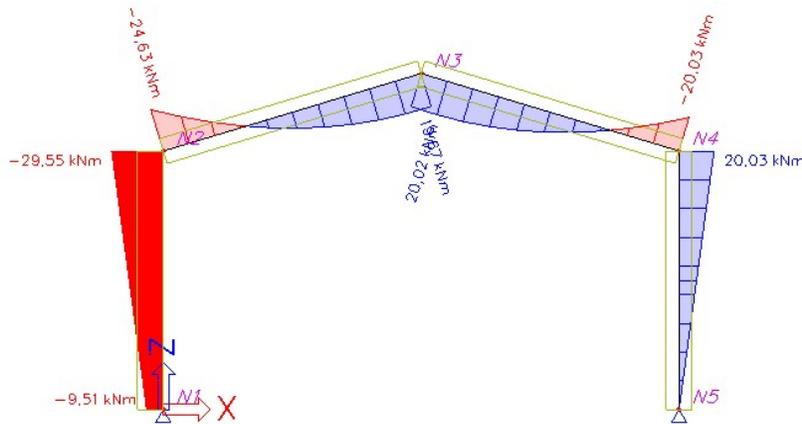
This result is caused by the eccentric position of the column. The normal force in node N1 takes part in the middle of the section. This means that in the case of a column with dimensions 500x300mm, this internal force takes part at a distance of 250mm of the support.
Please note that the supports are placed on the member system line and not on the reference line of the beam itself.

From this follows that a reaction of –38,06kN generates a moment of:

-38,06 kN x 0,250 m = -9,515 kNm

This can also be verified when claiming the internal forces for column B1:

	Name	dx [m]	Case	N [kN]	Vz [kN]	My [kNm]
1	B1	0,000	LC1	-38.06	-4,01	-9,51
2	B1	5,000	LC1	-19.66	-4,01	-29,55

Warning Elements with angle < 5°

If there are mesh elements with an angle < 5° the program gives a warning during the mesh generation with the numbers of these elements. These elements give inaccurate results. Usually it is about an error in the geometry input. The inaccuracy of results reveals itself in the neighborhood of this mesh element.
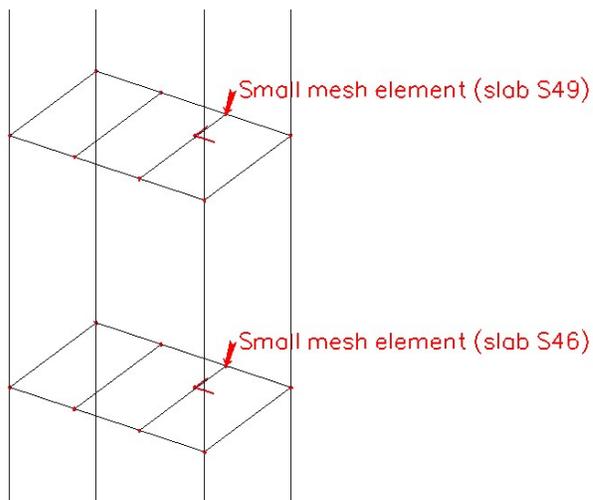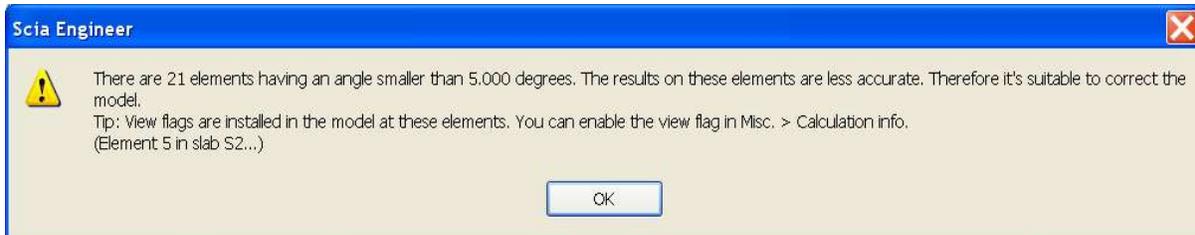
The arrows indicate the position of the less accurate mesh elements and they give the name of the 2D element to which these mesh elements belong.

Reproduction Mesh nodes and Mesh elements

When using **Set view parameters for all > Labels > Mesh** the numbers of the mesh nodes, 1D and 2D elements, can be displayed. This makes it possible to recover the elements of the above-mentioned warning or to have a look at the results in a specific node of the deformed mesh.
Further, this option also allows verifying for example the number of 1D elements on a beam.

Representing these labels is only possible after generating the Finite Elements mesh.

It is also possible to select a mesh-node by typing the command SELMN in the Command line.

Distribution of the loads

The solver of SCIA Engineer knows 3 types of loads on the finite element mesh:

- a constant surface load on a 2D element.
  This load works according to a local axis of the element, or according to the global axis

- a point force in a node of the element mesh

- a point moment in a node of the element mesh

The loads that are inputted in SCIA Engineer, are recalculated to loads on the finite element mesh as follows:

Introduced load	Is converted to:
2D surface load	Surface load on 2D elements
Free load – Rectangular or polygon	Surface load on 2D elements those are located completely into the charged surface. If 2D elements are charged partially, this load is converted into point forces on the nodes. The load on each 2D element is constant. A linear load is translated in a load that goes off gradually.
Free load – Line load or point force	Point forces on nodes
Free load – Point force (moment)	Point moments on nodes
1D line force or 1D point force	Point forces on nodes
1D line force or 1D point moment	Point moments on nodes

A point force is translated into forces on nodes of the mesh – without moments. The error created by this is generally smaller than the error caused by a finite division in elements. The error converges to zero when refining the mesh.

After the calculation, the loads on the finite element mesh can be looked at with **Calculation, mesh > 2D data viewer> surface loads.**

For larger projects, it is not necessary to calculate the entire structure in order to examine the division of the loads: start the calculation and choose the option **Test of input data** instead of **Linear calculation** in the dialogue box for the calculation. The loads are spread over the mesh, but the structure is not calculated.

Beside the forces and the moments there are also deformation loads: curving **epsilon** and expansion/shrinkage **k**. These are shown under **Calculation, Mesh > 2D data viewer > Temperature load.**

If a free 2D load is not shown in the menu 2D data viewer, it can be ascribed to the fact that the chosen mesh is too coarse. The 2D load is then displayed by means of point forces on the mesh nodes and not as a surface load on the element.

EXAMPLE: 04_01_LOAD DISTRIBUTION.esa

In this example, a plate is loaded gradually with load cases out from above-mentioned table. By way of **2D data viewer** it can be examined which way these load cases are spread over the mesh nodes and mesh elements.

Calculation – Solution methods

During the calculation, the solver has to formulate and solve the following system of equations:

$$F = K . \Delta$$

In this equation, F represents the force matrix, K the stiffness matrix of the structure and Δ the displacement matrix. For large projects, this involves a system that solves ten thousand equations (e.g. 40.000 nodes = 240.000 degrees of freedom = 240.000 equations).

To solve this system, two solution methods have been implemented in SCIA Engineer: the Direct Solution and the Iterative Solution.

Direct Solver

This is a standard Cholesky solution, based on a decomposition of the matrix of the system. Using this method, the system is solved directly into the displacements.

The advantage of this method consists of several right sides that can be solved simultaneously. This means that all linear load cases can be solved collectively. This type of solution is especially effective for little and medium-sized problems. The limit depends on the size of the problem and consequently the size of the available RAM memory.

Generally, it can be said that this solution is more convenient for most of the problems.

The disadvantage of this solution may emerge with extremely large problems. The calculation time may rise considerably when the RAM size is unsatisfactory. What's more, the problem cannot be solved at all when the available disk space (for memory swapping) is inadequate.

If the problem is big and of poor numerical condition, the rounding error may be so big that it exceeds the acceptable limit. This may result in an imbalance between results of load and reactions. The difference between the total sums of loads and reactions should not be bigger than about 0.5%. But even a value of 0.1% suggests that the results may be suspicious.

The total action and reaction per load case can be found in the **Calculation protocol** under **Results.**

If this imbalance appears, the program gives a warning and you should choose for the iterative solver. A precise value of the number of elements when this might occur cannot be defined, because the rounding errors also depends greatly on the ratio of the stiffness's of the elements.

Iterative Solver

The Incomplete Cholesky conjugate gradient method is applied. With this method, an assumed value for the displacements is introduced in the system. Subsequently the forces F are calculated and compared to the inputted forces. From this, new displacements are deduced. In that way, the displacements are calculated iteratively.

The advantage is a minimal demand on RAM and disk size (a project with 150.000 nodes requires about 250 Mb RAM memory). Therefore, the solution is convenient especially for extremely large problems that cannot be solved by means of direct solution or whose calculation time would be enormous for that kind of solution due to excessive disk operations.

Another advantage is that due to the ability of continuous improvement of the accuracy, the method is able to find a technically accurate solution even for equation systems that would be numerically unstable in the direct solution.

The disadvantage is that the method can only employ one right side at a time and this increases the time demands for equation systems with several right sides.
The desired solution method can be established under **Calculation, mesh > Setup solver**

Results

Convention of terminology and sign of the results

The following internal forces are given in relation to the finite elements. The indicated directions of the internal forces declare the positive directions. The action on the 2D mesh elements is drawn.

**FOR BENDING (2D plates, 3D shells) :**

Bending moments m_x , m_y

$$m_x = - \int_h \sigma_x . z \ dz$$
$$m_y = - \int_h \sigma_y . z \ dz$$
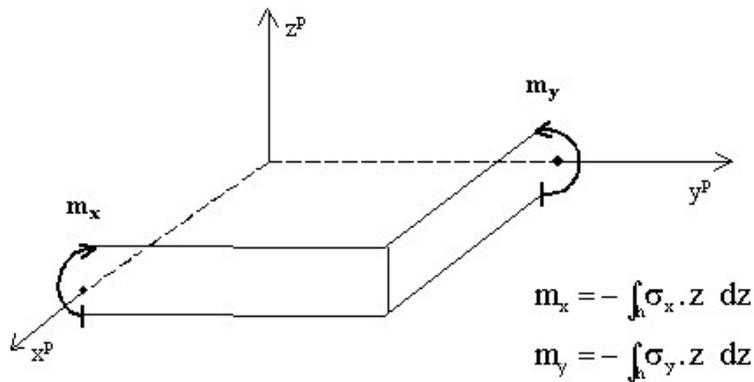
In case of homogeneous plates, the stress on the positive side z = +h/2 is:

σ_x = -m_x / **W**, σ_y = -m_y / **W**, with **W = h² / 6**.

Torsional moment m_{xy}

$$m_{xy} = - \int_h \tau_{xy} . z \ dz$$
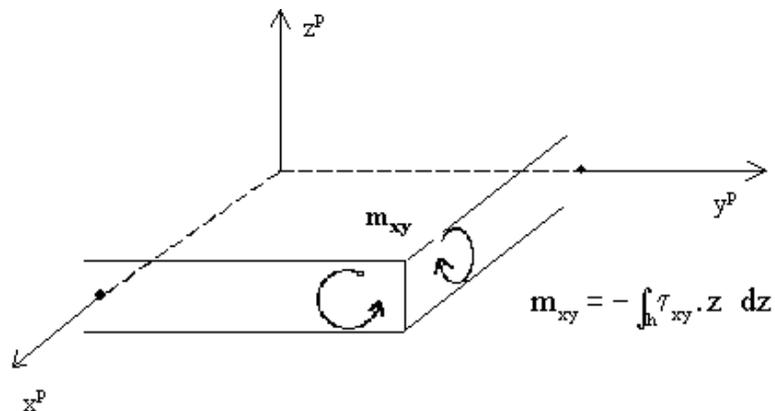
In case of homogeneous plates, the stress on the positive side z = +h/2 is:

σ_{xy} = **-m_{xy} / W**,   with **W = h² / 6**.

Shear forces q_x, q_y

$$q_x = - \int_h \tau_{xz} \ dz$$

$$q_y = - \int_h \tau_{yz} \ dz$$
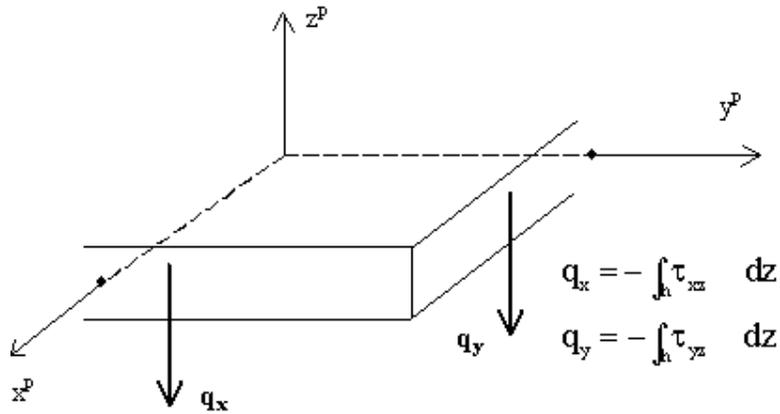
The vertical components of shear stress are τ_{xz} = **-q_x / h**, τ_{yz} = **-q_y / h**, constant across the thickness (Mindlin). More general, they can be understood as mean values for the interval   -h/2 < z < h/2.

FOR MEMBRANE EFFECTS (2D walls, 3D shells) :

Membrane forces n_x, n_y

$$n_x = \int_h \sigma_x \ dz$$

$$n_y = \int_h \sigma_y \ dz$$
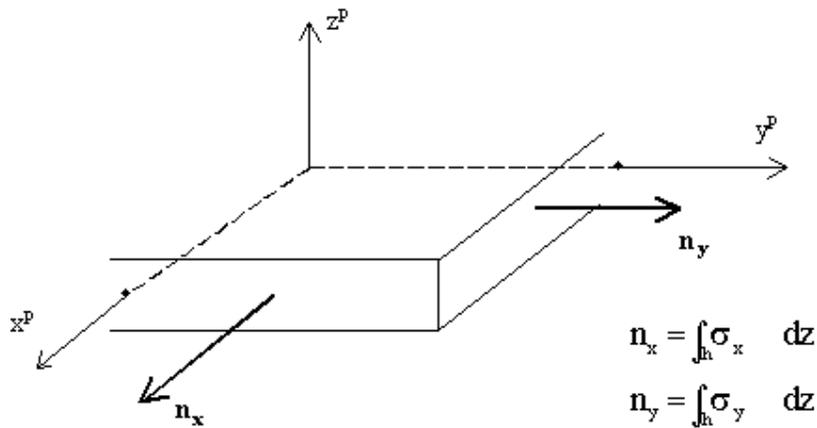
The normal forces are σ_x = **n_x / h**, σ_y = **n_y / h**.

Shear forces qxy

$$q_{xy} = \int_h \tau_{xy} \ dz$$
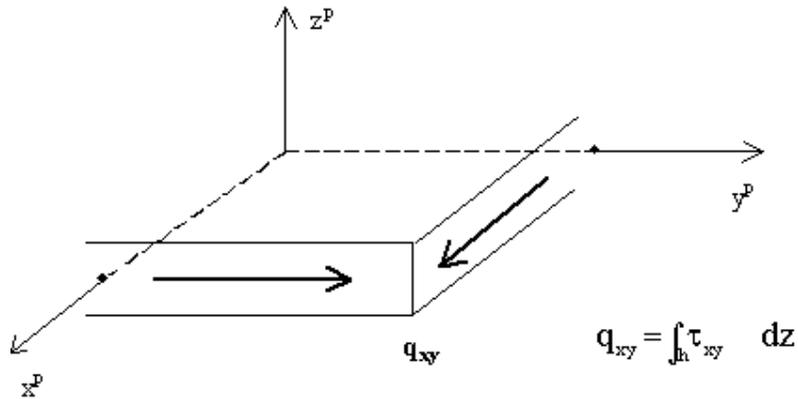
The horizontal component of the shear stress is $\tau_{xy} = q_{xy} / h$.

Comparison between a membrane and a standard element

When you compare a membrane element with a general 2D-element, then you can conclude that the bending moment mx equals 0 in this structure.
The figure below represents the stress σ_x in both elements.
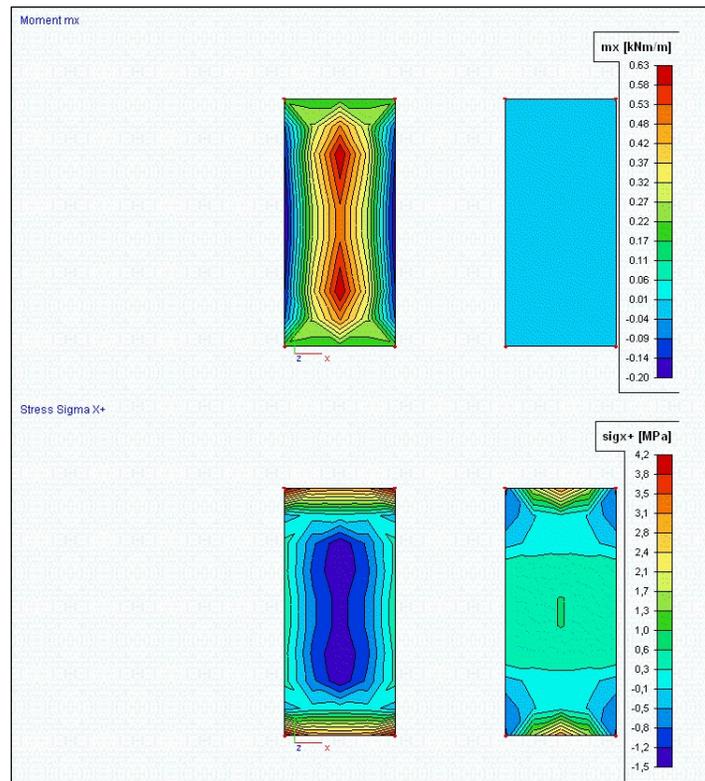
A plate element that is modeled in Plate XY environment will only take bending forces.
A wall element that is modeled in Wall XY environment will only take membrane forces.
It is however important to note that every 2D element that is modeled in a General XYZ environment can take both bending and membrane effects. This means not only shell elements, but also plate and wall

elements modeled in General XYZ can take both bending and membrane effects. Unless of course it is defined as a membrane element, because then the elements have only axial stiffness and will therefore take only membrane forces.

To define an element as a membrane element a nonlinear analysis should be run.
Also, the module esas.37 – membrane elements is needed. This module is only part of the EXPERT Edition. By making use of membrane elements, it is possible to neglect bending effects.

LCS mesh element and LCS-Member 2D

The x and y axes in which the internal forces are defined, as drawn above, may be the local axes of the mesh element (**LCS mesh element** option) or the local axis of the 2D element (**LCS-Member 2D** option).

Detailed results in mesh node

The internal forces of 2D-elements can be claimed for each mesh node or element. This is illustrated through following example.

EXAMPLE: 05_01_Results_mesh.esa

A plate supported on two edges is loaded by its self weight and a permanent surface load. Out of these two load cases an ULS and SLS combination are generated. Subsequently, the mesh is activated and by way of **Set view parameters for all > labels > mesh** the numbering of the mesh elements and the mesh nodes are displayed graphically.
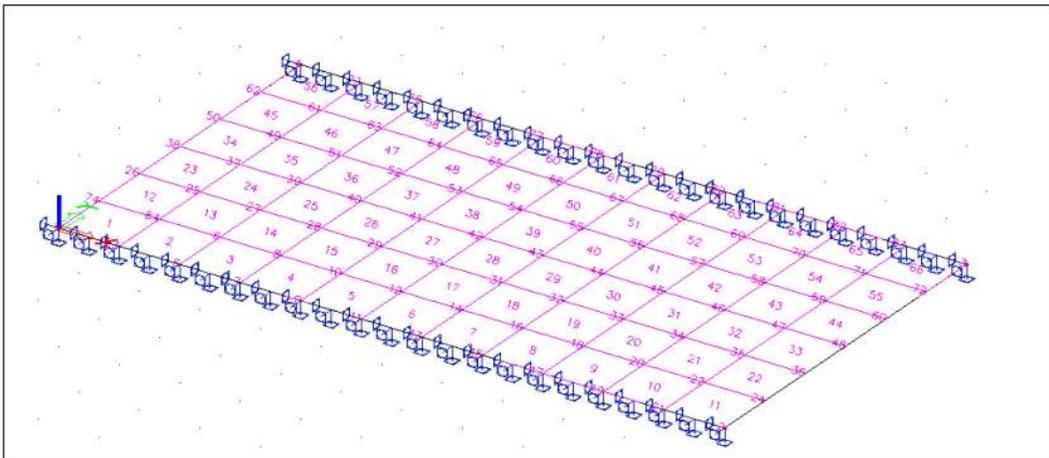
Then the values for mx and the ULS-combination are claimed by means of **Results > Member 2D – internal forces.**

22
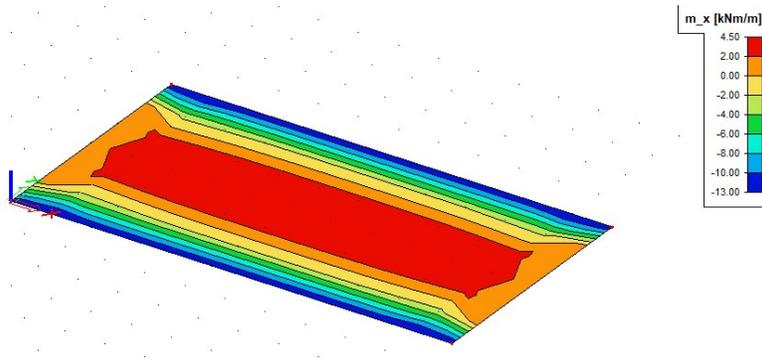
When drawing the results for a combination or Result class, then the property **Envelope (for 2D drawing)** should be set.

Since combinations are normally generated in SCIA Engineer, this means that one combination definition contains the results of all generated combinations.

The value "Minimum" will show the minimum values of these generated combinations.

The value "Maximum" will show the maximum values of these generated combinations.

The value "Absolute extreme" will show the value   max [abs (minimum value), abs (maximum value)].

Results in elements and mesh nodes

After a calculation of the structure, the deformation of the nodes and the reactions are calculated exactly (displacement method). These are represented in the menu **Results > 2D Members > Deformation of nodes** and **Results > Reactions.**

Deformations of nodes and reactions are the most accurate. The internal forces (and stresses) are derived from these quantities by means of the adopted basic functions and are always less accurate using the Finite Elements Method.

The internal forces of 2D elements are given in the menu **Results > 2D Members > Member 2D- Internal forces.** Here are 4 possibilities:

- **In centers**

- **In nodes no avg.**

- **In nodes avg.**

- **In nodes avg. on macro**

In centers

The average of the results, calculated in the four nodes of the quadrilateral finite element, is shown in the center of gravity of the element.

There is just 1 result per element, so the picture with isobands is a 'mosaic'. The development in a section is multi-stage curve.
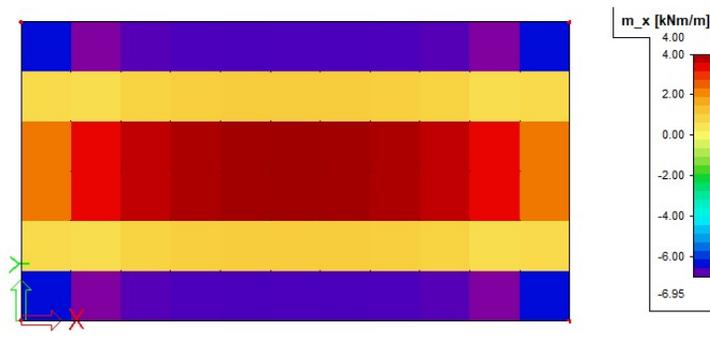
In nodes no avg.

The result value in four nodes of the quadrilateral finite element is considered. The distribution of a particular result quantity is considered linear over the element. No averaging between adjacent finite elements is performed. This provides that the four values of the adjacent elements are shown in a node.

If these 4 results differ considerably, this indicates that the applied mesh is too coarse. In such cases, a mesh refinement should be applied on those positions. This reproduction of the results gives a good idea of the discretization error in the calculation model.
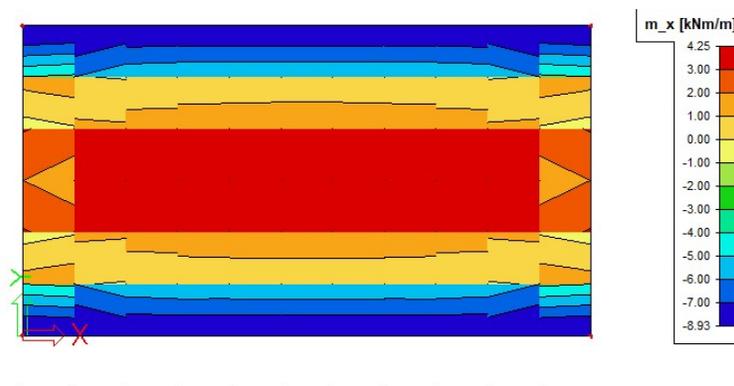

In nodes avg

Like the option above, but the result values from adjacent finite elements are averaged to smooth the distribution of the result from one element to another.

In some cases, it is not allowed to take the mean value of the internal forces in a node:

- 3D shells if the local x and y-axes of the 2 members not coincide. (example)
- If a result is discontinuous, for example the shear force at the line support of a plate: the peaks disappear completely when averaging the positive and negative shear forces. (example)
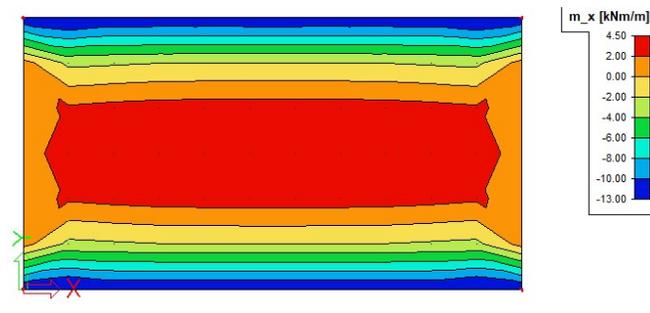
In nodes avg on macro

Like above, but the average is performed only for the elements of the same slab. On the connection of two slabs, the results from different slabs are NOT averaged. This solves the problems mentioned above.
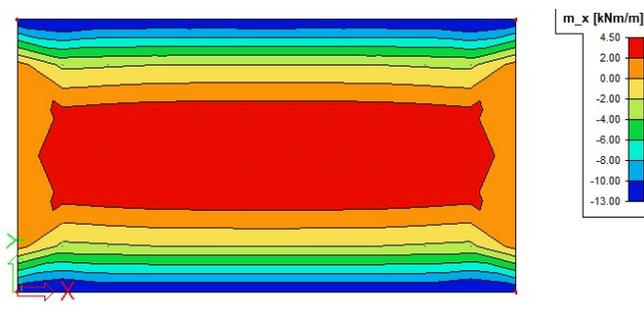
EXAMPLE: 06_01_RESULTS.esa

In this example, a plate of 5m x 5m with fixed translation in Z-direction for the 4 edges is calculated. The plate is made of concrete C25/30 according to EC, with a thickness of 100 mm and loaded by a point force of 100 kN in the middle.

Results in Nodes no avg.
The solver internally calculates the internal forces in integration points and at the end the calculated values are extrapolated from the integration points to the nodes of the 2D element using the formulas for a hyperbolic paraboloid.

Results in Centers
The result in the center of the mesh element is the (simple) average of the results in nodes no avg. of all nodes of this element.
For mesh element 60:

Nodes no avg.   Center
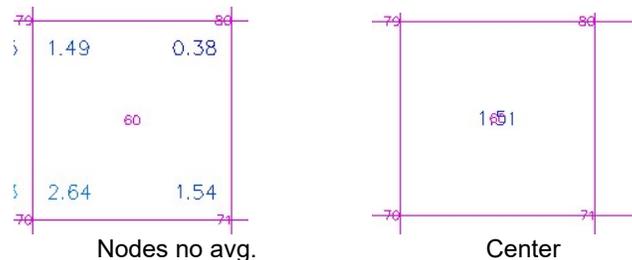
Results in Nodes avg. / Nodes avg. on macro

The averaged results of the moments are determined by first averaging and then smoothening.
The other averaged results are determined by only averaging.

While the values of moments in the integration points correspond to the theory, their extrapolation to nodes using the hyperbolic paraboloid leads to some loss of accuracy. The reason is that the hyperbolic paraboloid does not replace the distribution of the moments over the element with a satisfactory precision.
The calculation of the values of moments in nodes in SCIA Engineer uses an improved algorithm that replaces the hyperbolic paraboloid extrapolation with an advanced shear-force-integral method. As the distribution of shear forces over the surface of the element is assumed to follow the shape of the hyperbolic paraboloid (second order surface), the integral of this surface represents a third order surface that approximates the moment distribution with a higher accuracy. This is based on the formulas for the calculation of shear forces using the derivative of moments.

Moments are then written from these formulas:

$$m_x = \int \left(v_x - \frac{\partial m_{xy}}{\partial y} \right) dx + m_{x,0}$$

$$m_y = \int \left(v_y - \frac{\partial m_{xy}}{\partial x} \right) dy + m_{y,0}$$

Integration constants mx,0, my,0 are calculated from the condition of the value in the center of the element

$$m_{x,C} = \frac{\int_s m_x dS}{S}$$

$$m_{y,C} = \frac{\int_s m_y dS}{S}$$

With S the area of the finite element.

See also:
http://help.scia.net/17.0/en/#rb/calculation/smoothening_of_non-averaged_values_.htm

---

EXAMPLE: 06_02_Average1.esa

In this example, a steel structure consisting of 2 plates of 5x5m with thickness of 50 mm is considered. The plates are made of S235 according to EC and are fixed along two edges. The horizontal plate is loaded by a line load of 2kN/m.

The structure is calculated with a mesh size of **1m** and the results for **my** are claimed according to **System LCS – Member 2D.**

Results in Nodes Avg. versus in Nodes Avg. on macro.
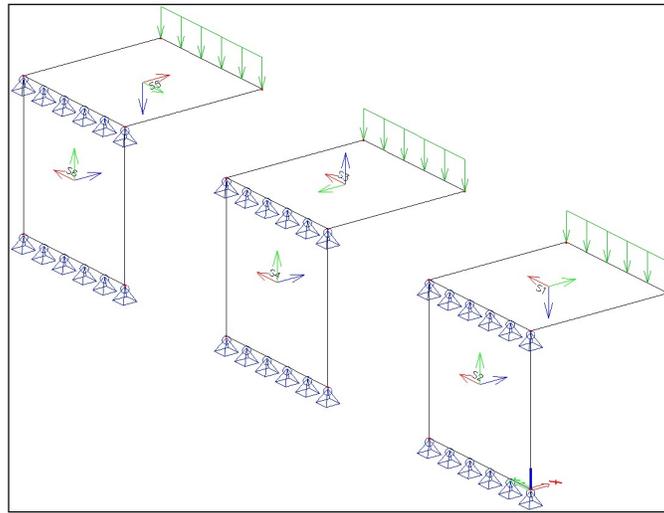
Concerning the plates **S1** and **S2**, the local x and y-axes have the same direction. Also the orientation of the local z-axis corresponds. The maximal moment in the section of the vertical plate **In nodes, averaging** is **3.39 kNm/m**.

For the plates **S3** and **S4,** the local x and y-axes correspond. But in this case, the orientation of the local z-axis does not correspond. On the edge, the average of 2 equal moments with an opposite sign is taken. The result is zero. The maximal moment in the section of the vertical plate **in nodes, averaging** is **2.47 kNm/m.** The result **In nodes, averaging on macro** gives **3.39 kNm/m**.

For plates **S5** and **S6**, the y-axis of plate S6 corresponds to the x-axis of plate S5. On the edge, the average of mx and my is taken, which has no significance. The maximal moment in the section of the vertical plate **in nodes, averaging** is **6.83 kNm/m**. The result **In nodes, averaging on macro** gives the correct value of **3.39** kNm/m.

**EXAMPLE: 06_03_Average2.esa**

A plate consisting of 2 fields of 7 m x 4 m (total 14m x 4m) and supported on its 3 shortest edges. Concrete quality C30/37 according to EC and a thickness 200mm. The plate is loaded by a uniform surface load of – 50 kN/m². The average mesh size amounts to **0,5m** and no averaging strips are used.

The results are claimed for the maximal shear force vx according to the carrying axle on a section in the middle of the plate.

In centers: 203,93 kN/m
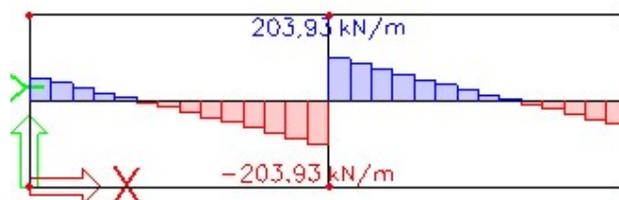
**In nodes, no averaging**: 199.28 kN/m
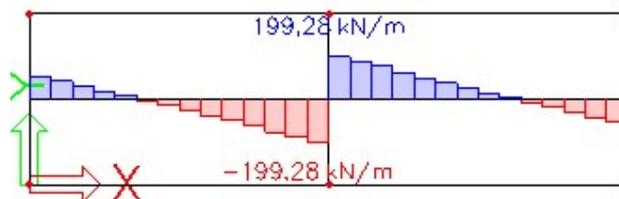
In nodes, averaging: 188.84 kN/m
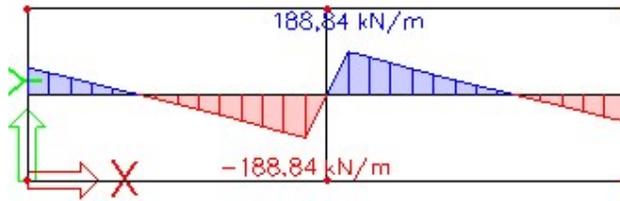
the average of the positive and negative shear force is taken on the edge.
Subsequently the result is zero so a lower maximal is obtained.
In nodes, averaging on macro: 209.72 kN/m
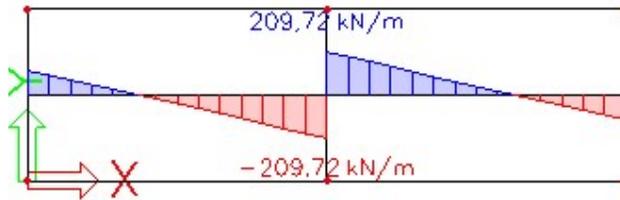

Averaging strip

An average result for a defined surface can be made using an averaging strip. This is illustrated in the following example.

EXAMPLE: 06_04_AveragingStrips.esa

A square slab is inputted with dimensions 2m x 2m. The mesh size is set to 0,5m and a surface load of 5kN/m² is inserted.
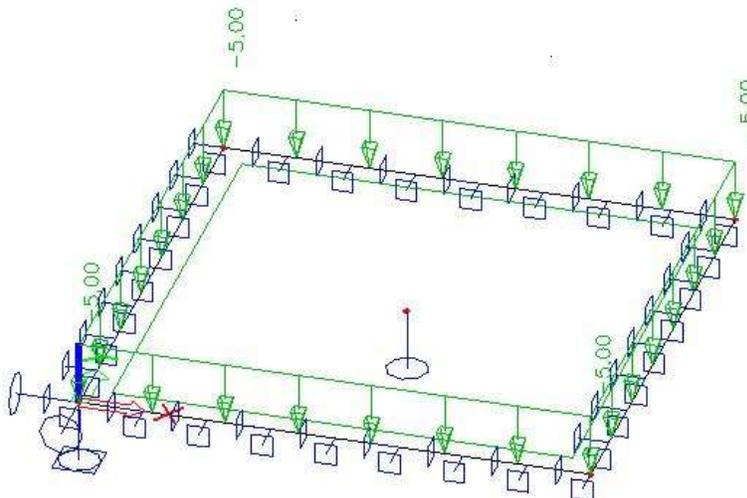
After the calculation, the following results for mx in nodes not averaged are obtained:
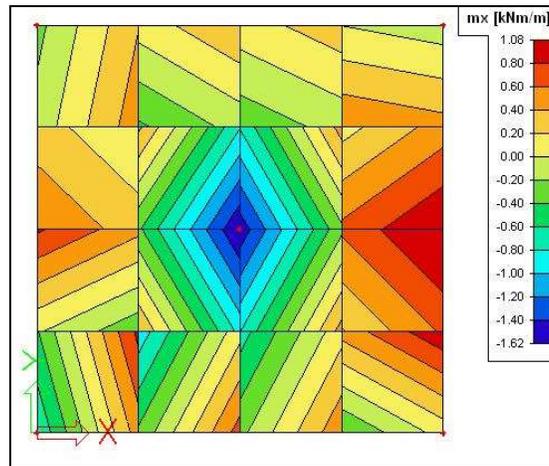
An averaging strip was inputted in the Y-direction with "Direction" set to "Perpendicular" and a width of "1m":
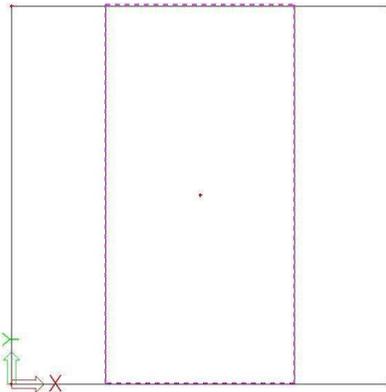
Results of mx in nodes not averaged with averaging strip:

29

When you look at the results, drawn with numbers, a manual verification can be made:

| −0.26	0.34	−0.19	−0.47	−0.73	0.17	0.28	−0.11
−0.12	0.34	−0.19	−0.47	−0.73	0.17	0.28	0.48
0.29	0.34	−0.19	−0.47	−0.73	0.17	0.28	0.50
0.59	0.34	−0.19	−0.47	−0.73	0.17	0.28	1.02
0.83	0.34	−0.19	−0.47	−0.73	0.17	0.28	1.08
0.03	0.34	−0.19	−0.47	−0.73	0.17	0.28	0.63
−0.42	0.34	−0.19	−0.47	−0.73	0.17	0.28	0.97
−0.77	0.34	−0.19	−0.47	−0.73	0.17	0.28	0.08

0,34   −0,19   −0,47
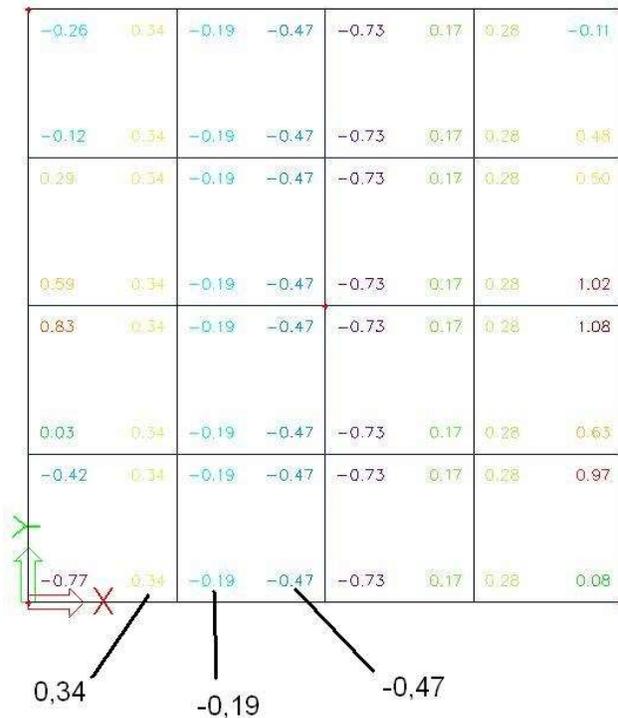
For the same Y-coordinate, in each element the same value will be obtained. Looking at the results in numbers without the averaging strip, the same value can be calculated taking the average of one line with the same Y-coordinate:

−0.26	0.39	0.09	0.40	−0.11	0.32	−0.00	−0.11
−0.12	0.53	−0.43	−0.13	−0.34	−0.13	0.59	0.48
0.29	0.01	0.44	−0.79	−0.78	0.49	0.12	0.50
0.59	0.30	−0.37	−1.60	−1.52	−0.24	0.65	1.02
0.83	0.46	−0.36	−1.62	−1.53	−0.24	0.63	1.08
0.03	−0.35	0.40	−0.87	−0.81	0.48	0.19	0.63
−0.42	0.86	−0.92	0.15	−0.69	0.15	0.48	0.97
−0.77	0.50	−0.36	0.70	−0.26	0.56	−0.41	0.08
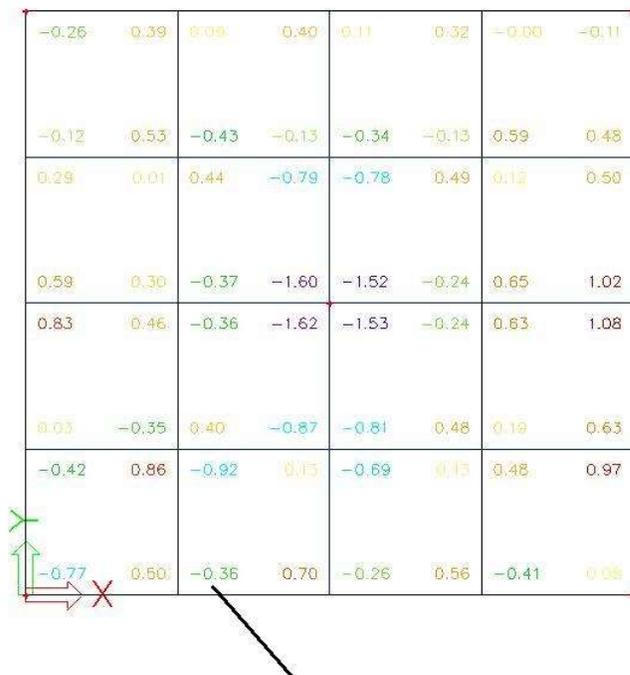
$$\frac{0,09 - 0,43 + 0,44 - 0,37 - 0,36 + 0,40 - 0,92 - 0,36}{8} = -0,18875 \approx -0,19$$

This averaging strip was defined as "Perpendicular" and inputted in the Y-direction. Looking at mx (perpendicular to the Y-direction) an average will be made.

When we look at my (parallel with the Y-direction) no average will be made:
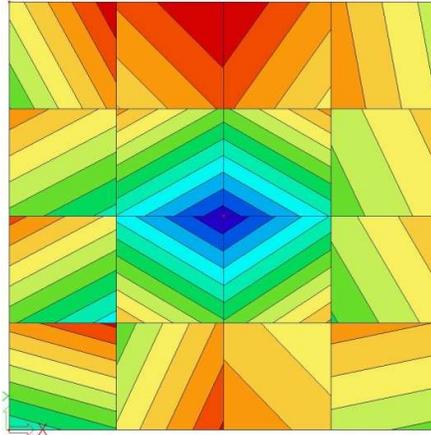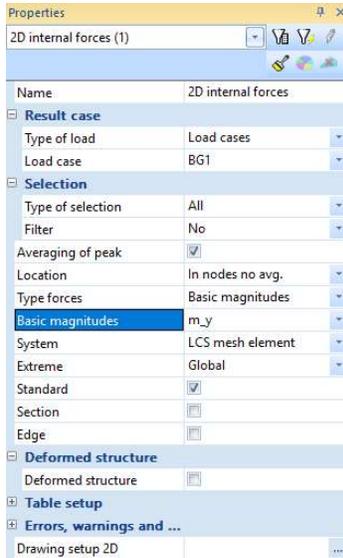
When changing this average strip from perpendicular to longitudinal:
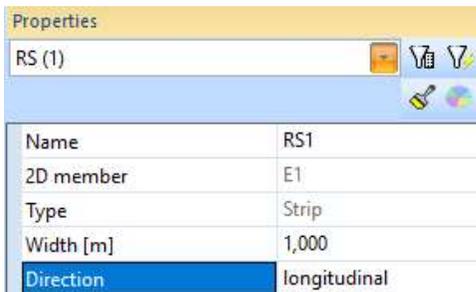
An average will be made for my but not anymore for mx:
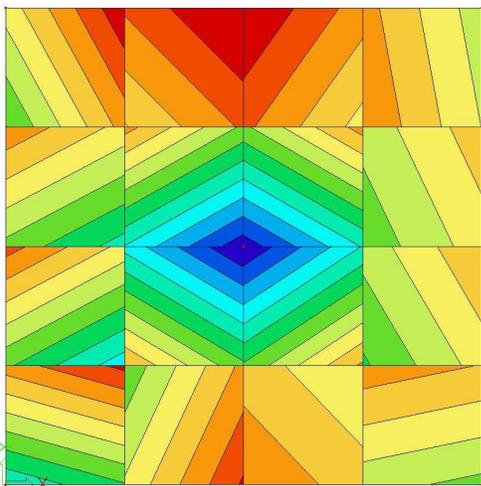
| *mx, average longitudinal* | *my, average longitudinal* |
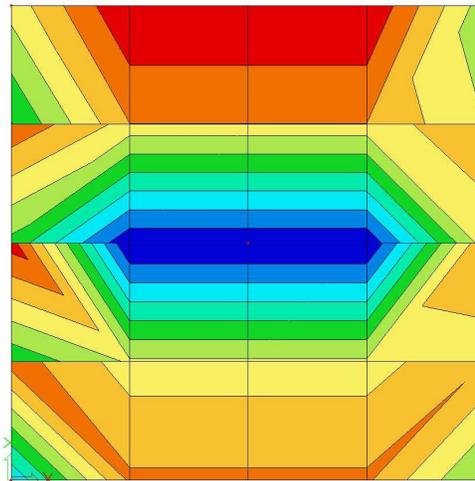
When choosing for "Both" an average will be made for the two directions.

Note:

The averaging algorithm uses only the finite elements that are located inside the averaging strip. This may cause certain inaccuracies especially in combination with larger finite elements. Therefore, it is recommended to define internal edges along the averaging strips. This ensures that finite element nodes are generated along the edge of the averaging strip, which may significantly improve the accuracy.

- The recommended procedure is thus:

    - Define the model of the structure
    - Perform the calculation
    - Review the results
    - Define averaging strips
    - Review the averaged results
    - Decide the final location and number of averaging strips
    - Define internal edges along the averaging strips
    - Repeat the calculation to obtain the improved results

Torsion – connection between plate and column

When a structure exists of a plate with a column on top of it, the user should pay extra attention to this when there is a question of torsion.

If the plate is subject to forces or moments, which cause torsion, very large deformations may occur. The reason for this is the lack of a degree of freedom in SCIA Engineer, namely the rotation around the z-axis. In other words, the moment mz is not present when asking for the internal forces of a 2D element.

The solution for this matter is the application of 'dummy-members' at the location of the connection between column and plate.

This is clarified with the following example.

EXAMPLE: 07_01_Column_plate.esa

Columns with a dimension of 500x500mm and a length of 4m are attached to a plate of 4x4m with a thickness of 500 mm.

As load case, two point forces of respectively –1 kN and 1 kN are applied on the edge nodes of the plate. These forces are lying according to the global X-axis. In this way, the plate will be subjected to a rotation in his own surface without any transformation of the geometry.

When the value U_total (Results > Displacements of nodes) in the plate is examined then very large deformations seem to appear. This is especially the case at the location of the edges.
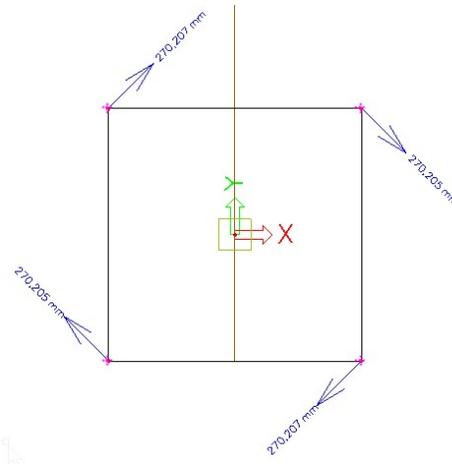
This phenomenon can be ascribed to the fact that the plate has no rotational stiffness around the Z-axis. 'Energy less' deformation occurs, which means that the plate does not know any resistance against the deformation Fiz.

However, in the middle of the plate, it is remarkable that this deformation will be much smaller and nearly equal to the deformations of the plates on which the dummy-members are fixed. This means that an infinite rigidity is ascribed to the connection plate-column. You can verify this by comparing the deformation of this node in the plate with the deformation fix of the column:
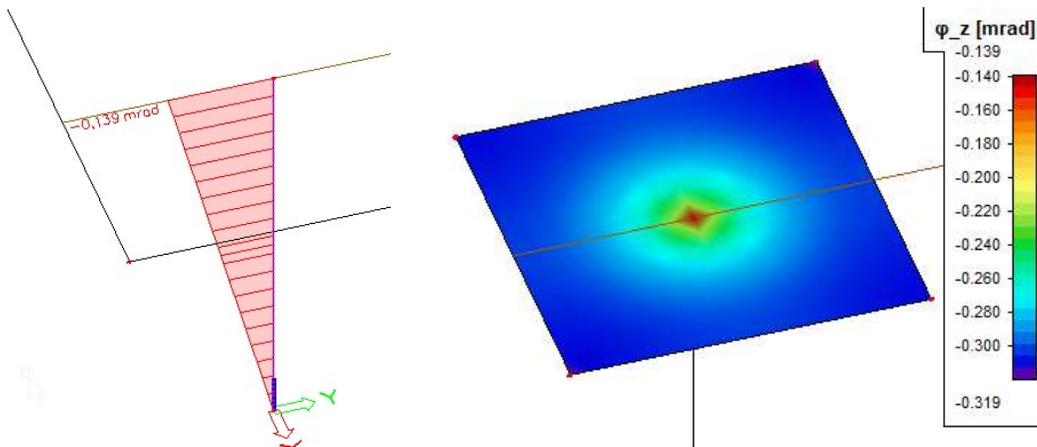
Applying two crossing dummy-members at the connections is a way to get a correct approach of the reality. These are attached to the plate by means of internal edges. This way, the small beams will take the rotation of the plate on themselves, so the plate has a stiffness around the Z-axis. In this case, the large deformations at the edges will be gone.

In the example, a variation of the length of the beams is applied to verify the influence on the deformations. With this you receive the following results with a constant mesh of 0.25 m:

Length of dummy [m]	nxy max [kN/m]	Fiz max [mrad]	U_total [mm]
0	57.54	-0.302	270.206
0,05	145.75	-0.130	0.564
0,1	27.35	-0.141	0.449
0,15	23.23	-0.137	0.428
0,2	13.35	-0.139	0.419
0,25	10.63	-0.140	0.414
0,35	6.21	-0.141	0.409
0,5	3.35	-0.141	0.407
0,75	1.74	-0.142	0.406
1	1.77	-0.143	0.406

Several conclusions can be drawn:

- When applying members of a very short length, this will affect the rotation and deformation sufficiently.

- Increasing the length of such a dummy-member will only have a small influence on the deformation and rotation.

- Nxy, on the other hand, has a larger influence when increasing the length: the larger the beams, the smaller the shear stress in the plate.

- The shear force varies little when a length of approximately half the section of the column is taken

- When using a length of the same dimensions as the section of the column, plausible results can be expected.

- The section of the beams has a significant influence on the shear stress: a greater section gives rise to a smaller shear stress and reverse.

  → Preparatory to an analysis, a width equal to the dimension of the column and a height equal to the thickness of the plate can be considered.

Subsequently the size of the mesh is varied when using a constant length of the dummy-beams, namely 0,25 m. The following results can be summarized in a table:

| Mesh Size [m]	qxy max [kN/m]	Fiz max [mrad]	U_total [mm]
1	4.13	-0,140	0,411
0,5	7,61	-0,139	0,412
0,25	10,80	-0,140	0,414
0,125	15,05	-0,140	0,415
0,1	14,44	-0,143	0,416
0,05	26.20	-0,140	0,416
0,025	39.55	-0,140	0,417

Also here following conclusions can be drawn:

- The deformation and rotation are only influenced with the size of the mesh to a very limited extent.

- The shear stress has a larger influence: it increases as the size of the mesh decreases.

→ Preparatory to an analysis, a mesh equal to the length of the beam or the double of the length can be taken, depending on the thickness of the plate.

EXAMPLE: 07_02_Moment mz Walls.esa

When two walls are connected with a beam, a similar phenomenon appears. Also, here a solution can be found by means of the application of dummy-members as in the following figure:
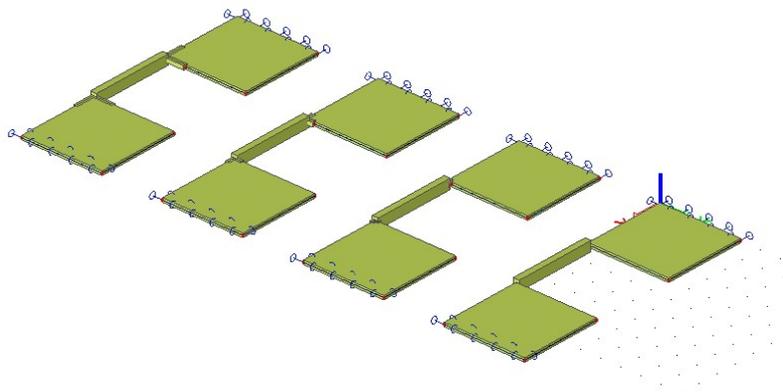
Two walls with a dimension of 4x4 m are connected with each other by means of a beam with a length of 4m. This member is loaded in the middle through a point force of 10kN.

When applying dummy-members with a variable length, the following results are obtained:

Length dummy-beam (m)	Field moment Mz (kNm)	Moment at the ends Mz (kNm)	Fiz (mrad)
0	9.99	0	0,277
0,2	6,41	-3,59	0,078
0,4	6,20	-3,80	0,067
1	6,18	-3,82	0,065

Even though the beam is fixed on both walls, it seems that this one has a moment of zero at the connections. In other words, it looks like there are hinged connections. When using dummy-beams, these moments rise slowly.

The walls show a much lower resistance against rotation around the Z-axis when there are no dummy-beams. This can be verified for the rotations Fiz. As in the previous example, there is a lack of rotational stiffness around the vertical axis.

Also in this case it can be solved by applying little beams to take care of this rotational stiffness. This effect becomes clear when the deformation of the connecting beam is verified:
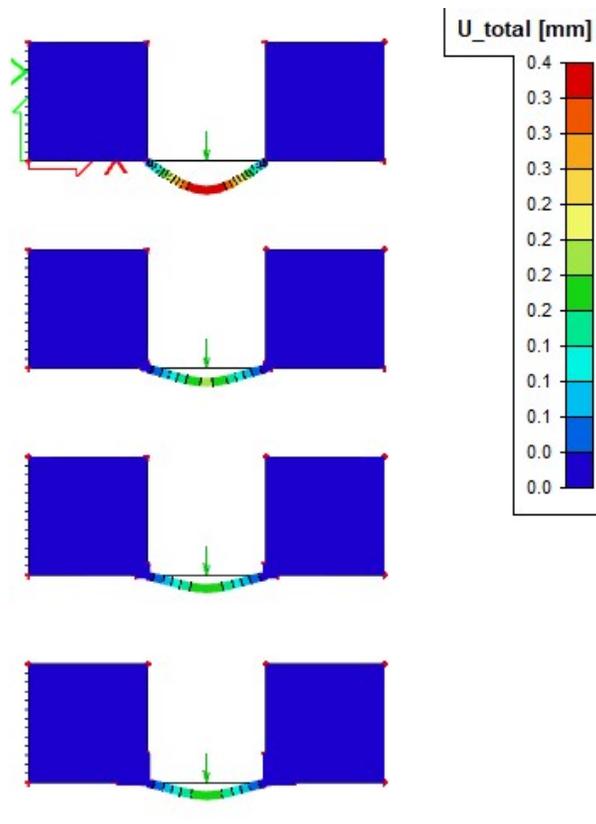
In figure a) there are no dummy-members. In this case, the beam has the largest deformation because there is no transition of the moment at the location of the connections. In figures b) up to d) the rotational stiffness is taken on the dummy-members and the beams will bend less than in other cases. Because of this, the moments at the ends will differ from zero. This way there is a better approach of the reality.

Choice of the element size

Introduction

There are no general methods to define the element size that has to be used in the calculation of certain structures. However, there are some basic rules, which will be explained below:

- a simple method to define the correct element size experimentally

- experience of the calculation of practical examples with this element type

- understanding into the singularities if the result does not converge to the sought solution.
  A refinement of the mesh does not offer any outcome in this case.

With a lack of experience with similar structures the experimental procedure- as mentioned below- will be in most cases the only method to define the right element size:

1. Calculate the structure with an element size L.

2. Calculate the structure with an element size L/2.
   The element size can be altered for the entire structure or just around the location of the sought result.

3. Compare the results and verify the differences (maximal field moment, maximal moment at the edges, maximal stress, theoretical needed reinforcement…).
   If this result is larger than the intended accuracy, it will be necessary to continue to refine the mesh.

The first rule for an initial choice of the element size exists of equating L with the thickness of the plate. Obviously, this is not applicable for thick plates. A second rule says that there has to be 5 to 10 elements per edge.

As mentioned above the option 'in nodes, no averaging' gives a good indication of the accuracy of the used mesh.

Further in this chapter, a few examples of 'patch tests' are given, in which the influence of the mesh refinement on the results can be verified. A finer mesh will lead to more accurate results but has a larger calculation time as a consequence.

However, in some cases the results of the finite elements calculation does not converge to the expected result. In such cases an increased mesh refinement does not offer a good solution.

Three examples of such singularities that occur in real terms are discussed in the paragraph "Singularities".

Patch test plate – Comparison with beam element

A beam with rectangular section (width 1m x height 0,3 m) is compared with a plate. The span is 8m, the beam is fixed at both sides and loaded with a line load of 100 kN/m. The material is C25/30 according to EC. The coefficient of Poisson is equal to 0 to eliminate the transverse deformation of the plate.

The result of a beam calculation in SCIA Engineer is taken as reference. The reason for this is that SCIA Engineer takes the shear force deformation into account in the standard way.
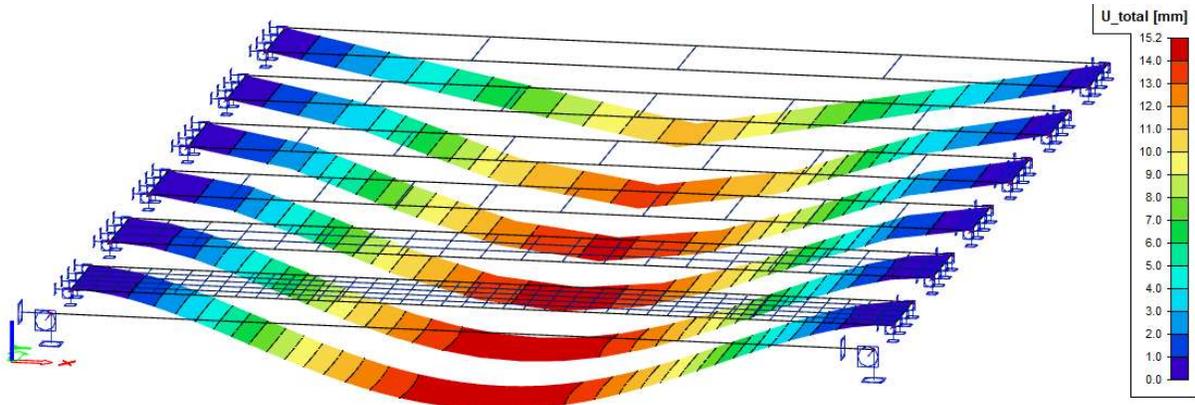
The results for the internal forces are claimed in **nodes, averaging.**

The maximal deformation U_total (reference result: **15,3 mm**)

| Element size [m]	# elements along the length	Uz [mm]
2	4	-11.6
1,3	6	-13.6
1	8	-14.4
0,8	10	-14.7
0,4	20	-15.1
0,2	40	-15,2

The maximal field moment mx (reference result: **266,67 kNm**)

Element size [m]	# elements along the length	mx [kNm/m]
2	4	256.62
1,3	6	265.83
1	8	265.12
0,8	10	265.28
0,4	20	266.35
0,2	40	265.99

The maximal moment at the ends (reference result: **-533,33 kNm**)

Element size [m]	# elements along the length	mx [kNm/m]
2	4	-473.71
1,3	6	-484.31
1	8	-494.03
0,8	10	-504,00
0,4	20	-545,51
0,2	40	-548.89

Next, the results are grouped in a convergence curve:
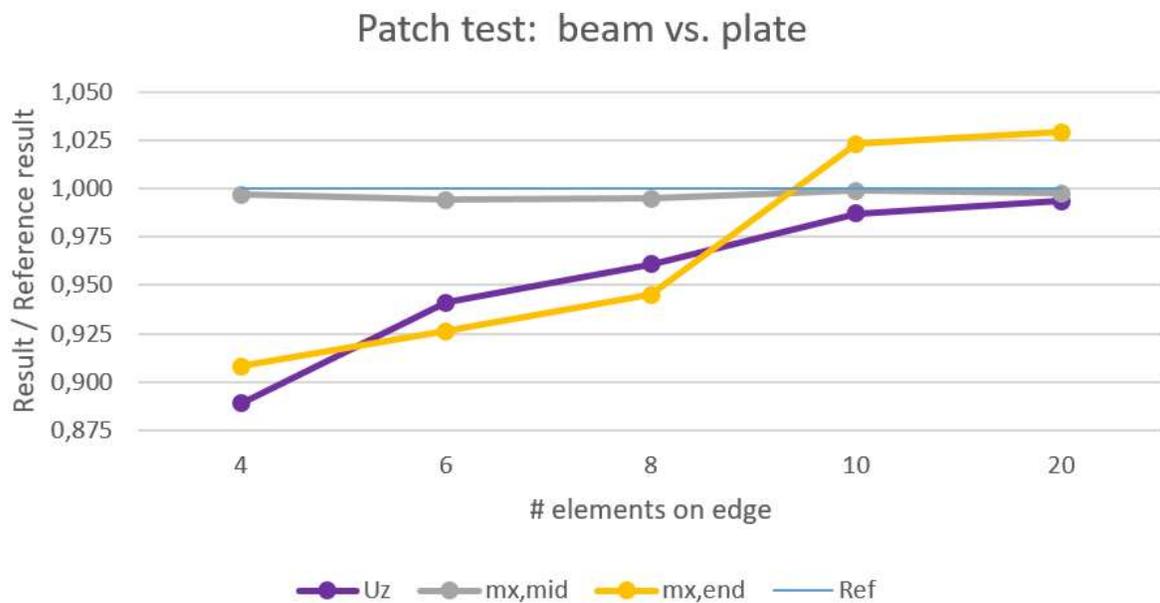
Conclusions:

- The results converge to the exact solution
- The field moment is always precise
- The moment at the edges (peak value) and the deformation require about 10 elements.

Patch test plate – comparison with analytic results

The maximal deflection, the maximal field moment and the maximal moment at the edges of a plate fixed on the four edges are compared with analytic results.

Project Data

Length: a = 5m
Thickness: t = 0,02m
Surface load: q = -10kN/m²
Steel: S235 according to EC
E = 210000 N/mm²
$\nu = 0,3$
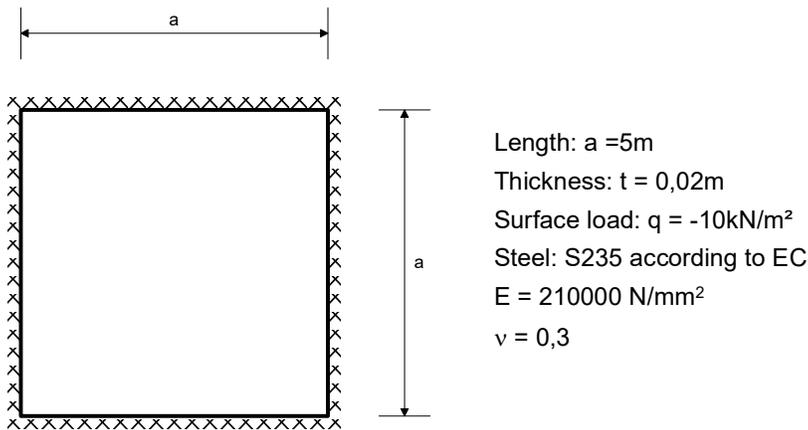
The plate is calculated in SCIA Engineer as well with Mindlin elements (Patchtest_2a) as Kirchhoff elements (Patchtest_2b).

Reference

S.P. Timoshenko and S. Wotnowsky-Krieger, *Theory of Plates and Shells*, 2nd edition, Mc Graw Hill, New York, 1987.

Analytic results

Deflection in the middle of the plate:
    $f = 0.00126 \; q \; a^4 / D$, with $D = E \; t^3 / (12 (1 - \nu^2))$ → **f = -51,187 mm**

Moment in the middle of an edge:
    $M = 0,0513 \; q \; a^2$ → **M = 12,8825 kNm/m**

Moment in the middle of the plate:
 $M = 0,0231 \; q \; a^2$ → **M = 5,775 kNm/m**
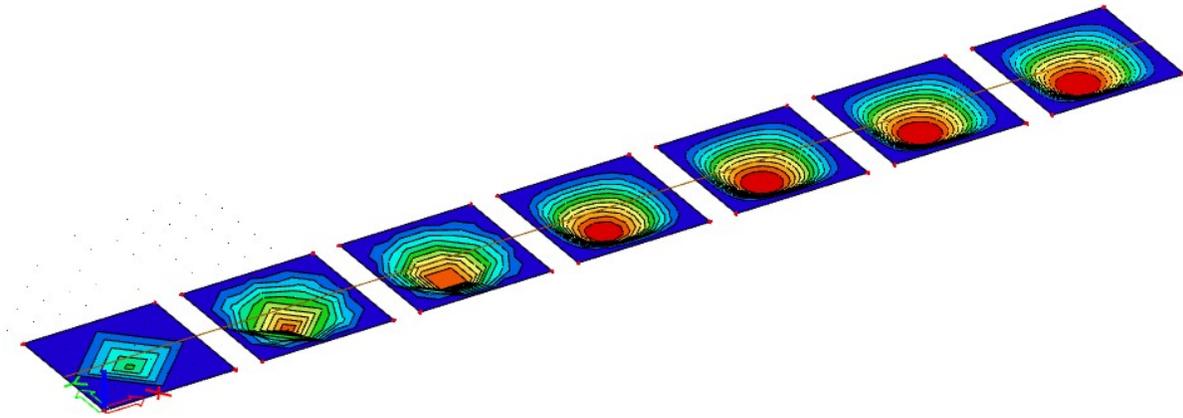
The results for the internal forces are claimed in **nodes, averaging.**
The graph shows the Result / Reference result.

The maximal deflection Uz in the middle of the plate (reference result: **51,187 mm**)

# Elements	Mindlin element	Kirchhoff element
4	-21,700	-56,324
16	-44,758	-52,591
25	-40,239	-45,349
100	-50,196	-51,610
400	-51,098	-51,460
2500	-51,355	-51,413
10000	-51,404	-51,406
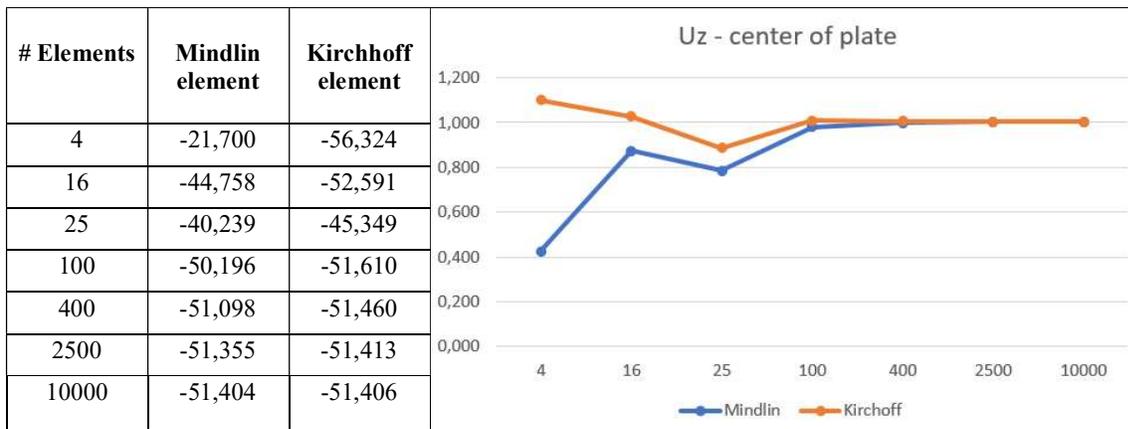
The maximal moment in the middle of an edge (reference result: **12,8825 kNm/m**)

# Elements	Mindlin element	Kirchhoff element
4	7,806	4,718
16	11,823	9,306
25	11,788	10,096
100	12,920	12,426
400	12,851	12,737
2500	12,835	12,818
10000	-12,833	12,830
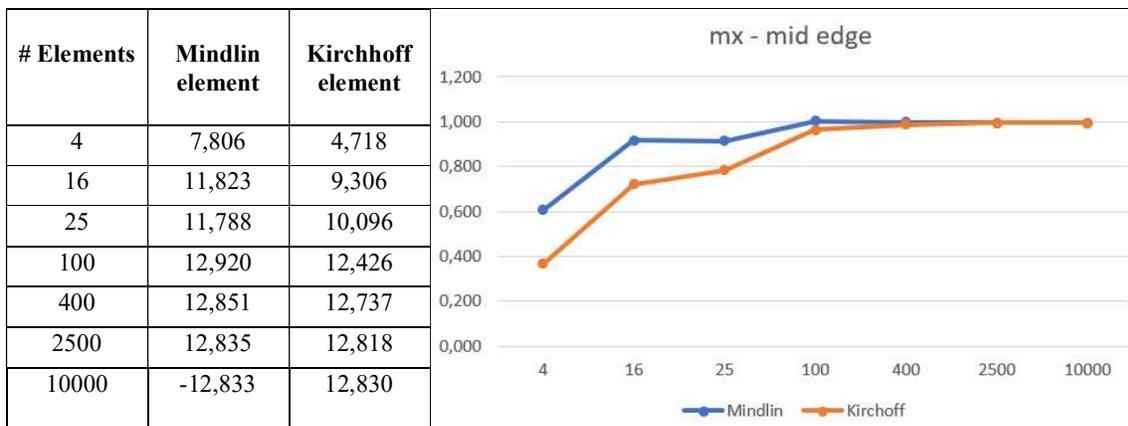
The maximal moment mx in the middle of the plate (reference result: **5,775 kNm/m**)

# Elements	Mindlin element	Kirchhoff element
4	3,299	4,025
16	8,026	5,712
25	4,999	4,933
100	5,681	5,706
400	5,713	5,719
2500	5,724	5,725
10000	5,725	5,725
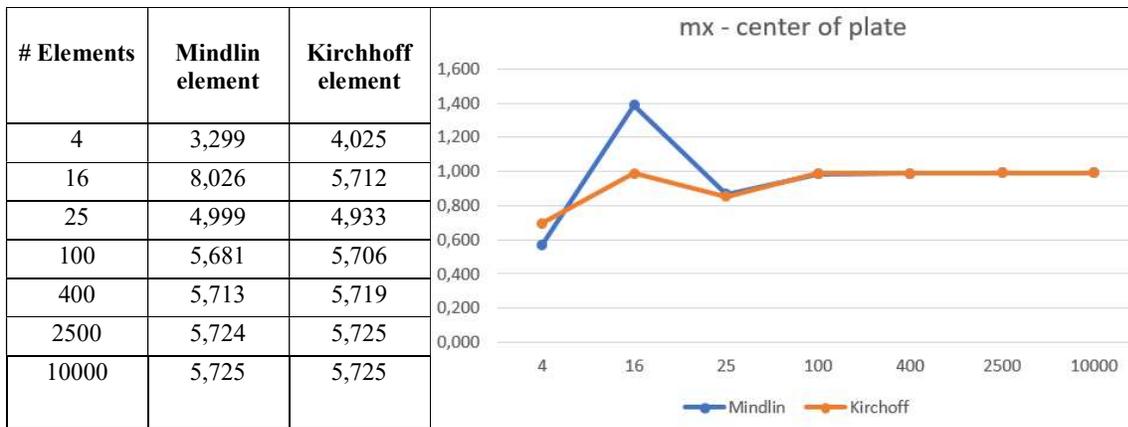
The conclusions:

- The results converge to an exact solution

- 5 elements along the field give approaching values, 10 elements along the field give accurate values

- The Kirchhoff element in this case gives a slightly better approach than the Mindlin element.
  Reason: the reference result does not take the shear force deformation into account.

Singularities

Equivalent plate ⇔ beam

A frequently occurring misunderstanding is the fact that the user thinks that a simple plate supported on 2 edges behaves as a beam. This is only the case without transverse contraction (if $\nu = 0$). With normal values of the Poisson coefficient ($\nu = 0,2$ or $\nu = 0,3$) very high peaks of the reactions appear near the angles.

Mesh refinement does not offer a satisfactory solution in this case and will even increases the peak value.

This peak value is correct and converges to the theoretical value infinity by increasing the mesh refinement. This can be explained as follows:

Consider the plate as different beams which lie next to each other. With $\nu = 0,2$, the bottom of the beam becomes smaller, the top on the other hand becomes broader. The plate is going to bend in a direction parallel to the supported edges, with the round side upwards (saddle forming: the plate deforms in the bearing direction with the round side upwards). This bending is prevented by the line supports.

In a continuous plate, this will cause bending moments my in the transverse direction, approximately with a size of 0,2 mx. If this moment my occurred along the entire width of the plate, the reaction would be constant. However, the moment must be zero on the free edges. So, it seems that an opposite moment 0,2 mx exists on this edge, which leads to great reactions in the corners. In other words: at the end of the plate the saddle forming is not prevented any more by the moments in the plate. The plate wants to deform downwards at the end, which is prevented by the rigid supports. Because of this, very large reactions appear.

EXAMPLE: 09_01_PLATEBEAM.esa

In this example, a plate of 3mx10m is calculated according to EC. The material is made of concrete C25/30. The thickness of the plate amounts to 200 mm. The plate is supported on the long edges and is loaded by a uniform load of 100 kN/m2.

Without the plate action, a uniform line load of 150 kN/m is expected along each border.

The plate is calculated with an increasingly finer mesh. The maximal reaction in the corner increases more and more:
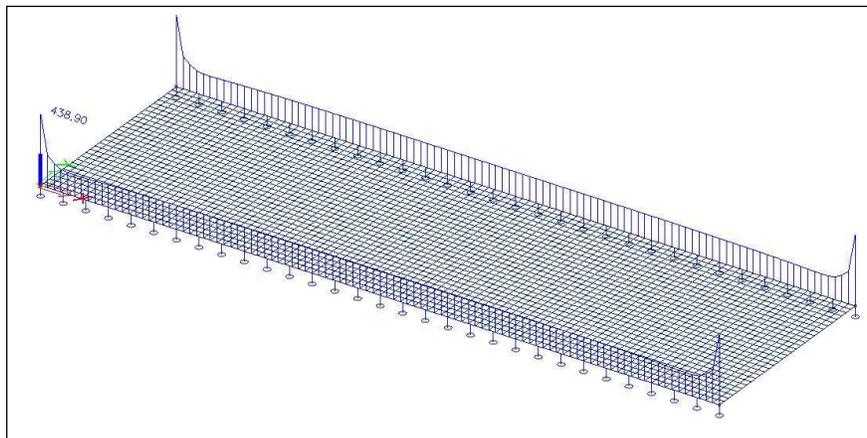
| Element size [m]	max. reaction rigid support [kN/m]	max. Reaction flexible support [kN/m]	Reduction peak value %
0,8	179,62	175,19	2,5 %
0,4	232,84	204,51	12 %
0,2	326,45	224,34	31 %
0,1	438,92	230,96	47 %
0,05	549,11	232,62	57 %

The peak in the reaction can be attributed to the infinite stiffness of the support. A realistic stiffness reduces the peak value considerably.

In this example, the rigid supports are replaced by the stiffness of a concrete wall with E-modulus 32.000 MPa, a thickness of 0,1m and a height of 4m.

$$k = \frac{E.t}{h} = \frac{32000 \frac{N}{mm^2} .100mm}{4000mm} = 800 \frac{N}{mm^2} = 800 \frac{MN}{m^2}$$

The results clearly indicate that the flexible supports have a decreasing influence on the peak value.

This effect is given in the following diagram:


Plate beam - flexible vs. rigid supports
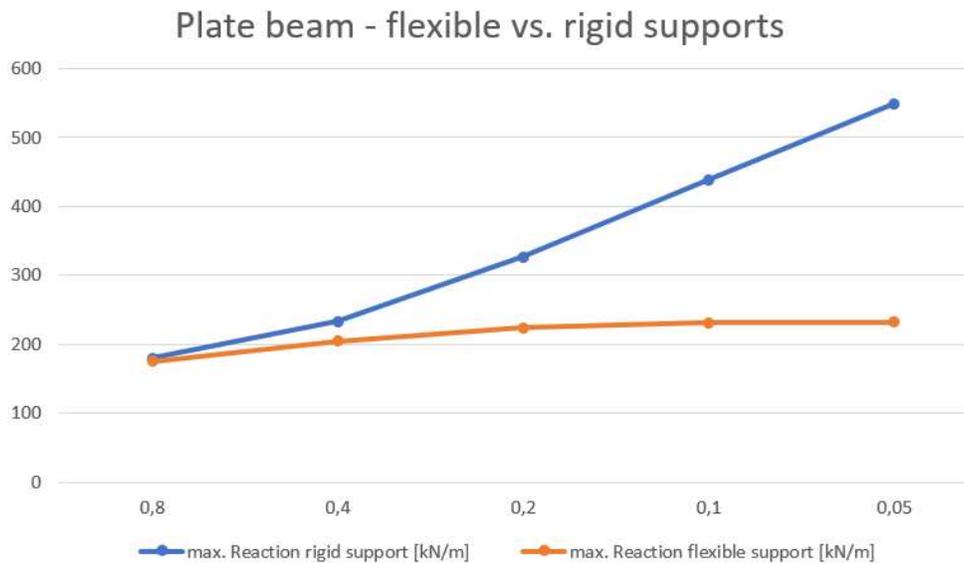
Free edge

With a free edge, a 2D plate cannot satisfy all the boundary conditions when using Kirchhoff elements. This effect is not a property of certain elements or the finite element method, but of the simplifications of the plate theory.

The explanation of this effect is explained in detail in reference [4].

The figure below shows the border of a plate that is free supported (hinged supports). In a section perpendicular to the support, a torsion moment mxy exists. The created shear stresses have to go round at the end of the plate, which occurs along a width equal to the thickness of the plate. In this part of the plate the shear stresses produce a resulting vertical force V.
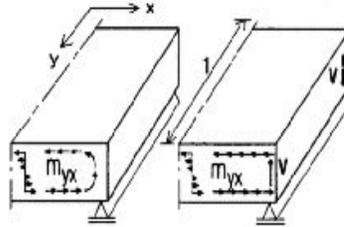

The dimension of the concentrated force V can be calculated by means of the part of the plate that is displayed in the following picture.
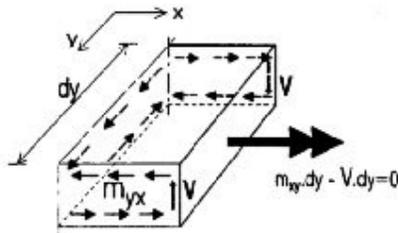
In the sections parallel to the x-axis, a moment myx prevails and in the section parallel to the y-axis a moment mxy prevails. Then the couple equilibrium around the x-axis is considered. In this equilibrium mxy and the forces V play a role from which follows:

$$V = m_{xy}$$

This is the case for a constant torsion moment. The vertical equilibrium of the forces with a varying torsion moment is drawn on the picture below.

$$r.dy = v_x.dy + m_{xy,y}.dy$$
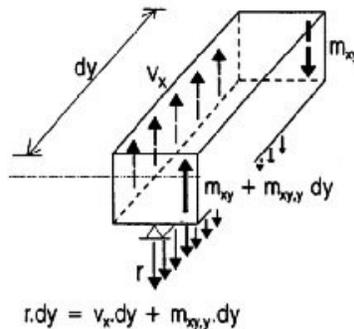
Over a distance dy, the concentrated vertical force has increased with the value dmxy/dy. On the basis of the vertical force equilibrium, the following formula is now valid:

$$r = v_x + dm_{xy}/dy$$

r is also called the shear force of Kirchhoff. The word shear force is not correct, since r is a reaction force of the line support.
Analogously, concerning the reaction of the edge parallel to the x-axis:

$$r = v_y + dm_{xy}/dy$$

The torsion moment is transformed under the form of additional reactions. Only the increase of the torsion moment seems to play an important role.
This effect is known at the end of the edge, in the corner of the plate. On the picture below, the corner of a plate has been drawn in which two free supported edges come together. The dimension of the plate e is very small. On the two edges outside the plate, there is no possibility for the appearance of a vertical force with a value of mxy or myx. In the sections trough the plate, they may occur. To ensure the vertical load of this corner, there has to be a concentrated reaction with a value of mxy+myx. A familiar example is the rectangular plate with a surface load and supported on all the edges. In here, tensile forces occur in the corners.
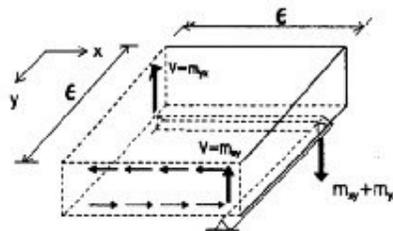
On the free edge the reaction equals zero:

$$r = v_x + dm_{xy}/dy = 0$$

This does not necessarily imply that v_x or m_{xy} equal 0.

Specifically, this means that when using the Kirchhoff elements in SCIA Engineer at a free edge, a torsion moment m_{xy} may occur.

When using Mindlin elements, there is an additional degree of freedom (rotation of the normal on the axis of the plate with regard to the plate axis in deformed state) and the condition of mxy = 0 can be met. There may occur a moment mxy in the plate and the gradient of mxy may give very huge shear forces because of the condition of the vertical equilibrium. These shear forces occur entirely in the first row of elements along the corner. With increasing mesh refinement this shear force converges to infinity. It can be neglected during the design.

EXAMPLE: 10_01_Free edge.esa

In this example a plate of dimension 5m x 5m is calculated with concrete quality C25/30 according to the EC. The thickness of the plate is 0,2m. The plate is hinged on two edges and loaded by a free surface load of -50 kN/m².

The plate is calculated with different dimensions of the Finite Element mesh. The results for the torsion moment on the free edge and the shear force qx in a corner are claimed in **nodes, averaging.**
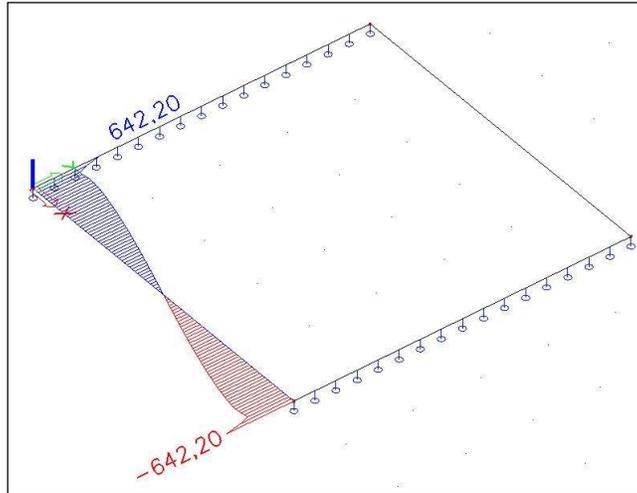

Mindlin element:

# Elements	mxy edge [kNm/m]	qx corner [kN/m]
25	13,75	168,61
100	16,13	221,57
400	12,81	312,72
1600	7,77	442,02
2500	6,98	489,20
10000	4,46	642,24
40000	2,26	792,51

Kirchhoff element:

# Elements	mxy edge [kNm/m]	qx corner [kN/m]
25	19,23	108,86
100	20,60	108,09
400	20,85	107,41
1600	20,90	106,95
2500	20,90	106,85
10000	20,90	106,61
40000	20,90	106,77

The results are shown in following convergence curves:

Torsional moment mxy in free edge

Shear force vx in edge
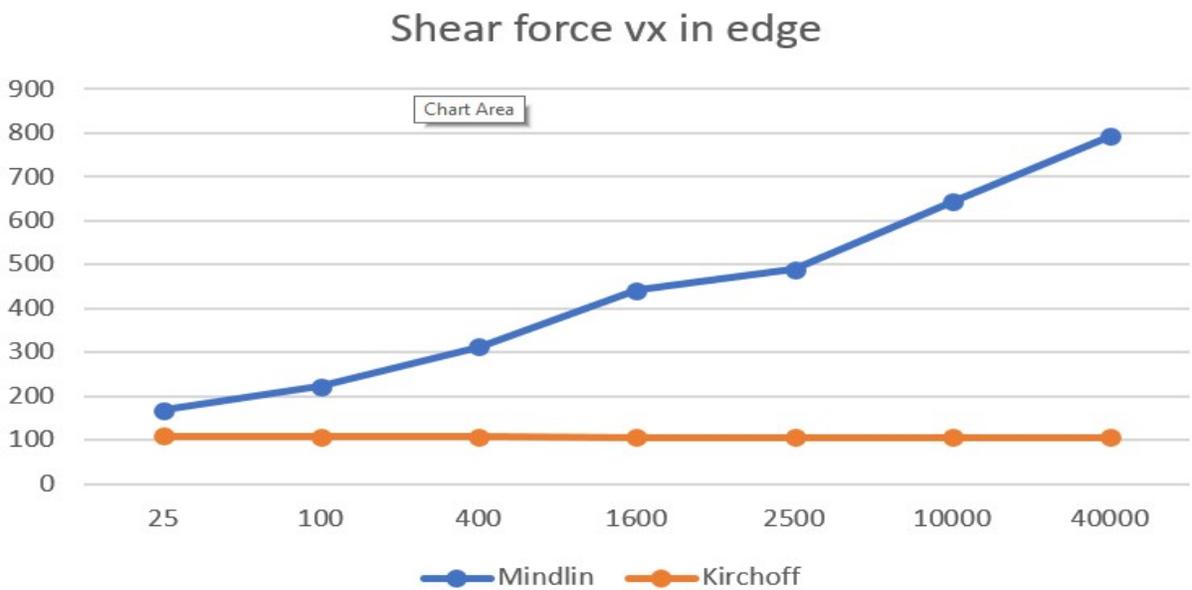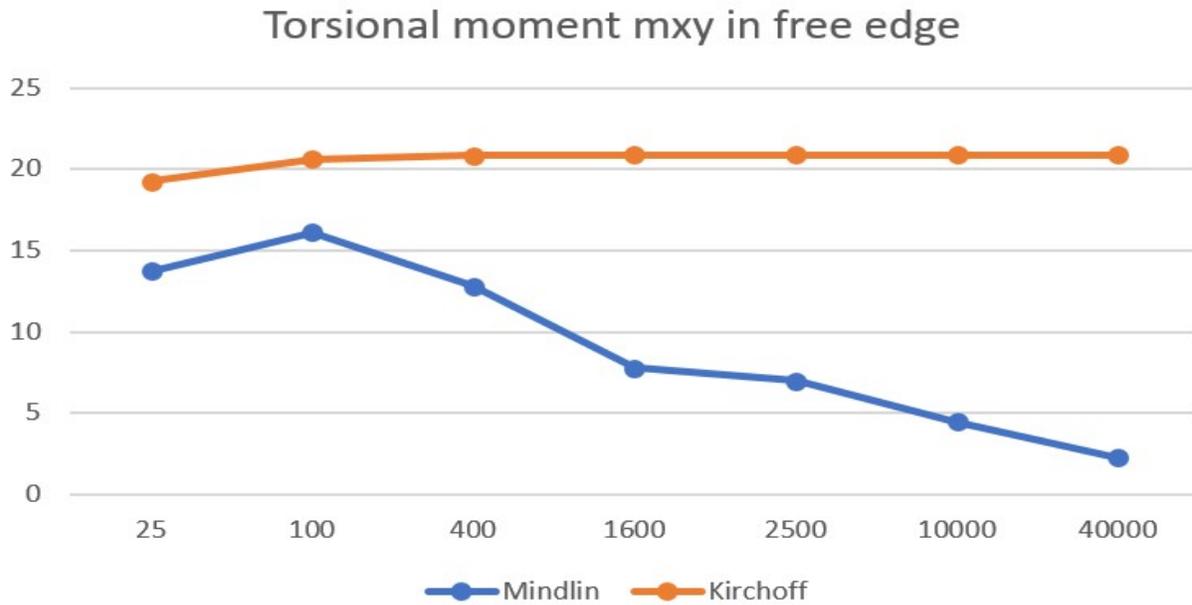
The above-mentioned behavior of both elements is clearly noticeable:
for the Mindlin element:

      mxy converges to the correct value zero, and

      on the other hand, the shear forces increase.

for the Kirchhoff element:

      the condition mxy = 0 cannot be satisfied, even with a mesh refinement, and

      the shear force converges to a constant value.

Nodal support

In most cases, a column or pole is introduced as a nodal support. The real dimensions of the support are neglected. In the Finite Element Method this is a singular node and the bending moment above this support is theoretically infinite. The moment will also converge to this infinite value with increasing mesh refinement.

Refining of the mesh does not lead to the desired results in this case since the moment does not converge to the real value.

A possibility to calculate this moment correctly is to introduce the column not as a nodal support but as a flexible supported sub region. The dimensions of the sub region are the dimensions of the column. The flexible support can be calculated out of the stiffness of the column. The results of such an approach are compared to the results of a nodal support in the example below.

With an element mesh of half the dimension of the column, the model with a sub region gives a good value of the occurring moment. The value is a little bit higher than the real occurring moment. An even finer mesh gives unreal values. An element size equal to the dimension of the column is too coarse and gives an underestimation of the real occurring moment.

**EXAMPLE: 11_01_NODAL SUPPORT.esa**

In this example a floor is analyzed. It is supported by columns. The plate has a thickness of 0,2 m and is made of concrete C25/30 according to the EC. The whole floor is loaded with a surface load of 100 kN/m².

For the calculation one field of 6mx6m is considered. In the middle of this field a nodal support is inserted to represent the column. At the edges the rotation of the plate is prevented in both directions since the plate is stuck 'on itself'.

In the first case the column is introduced by means of a nodal support. Secondly, the column is made as a sub region supported by a flexible foundation. And in the last case, an averaging strip is used with the dimensions of the column.

For the calculation of the stiffness a concrete column has been taken with a E-modulus of 32.000 MPa, height 4m and cross-section 0,5m x 0,5m.

$$k = \frac{E}{h} = \frac{32000\ N/mm^2}{4000mm} = 8\ N/mm^3 = 8000\ MN/m^3$$
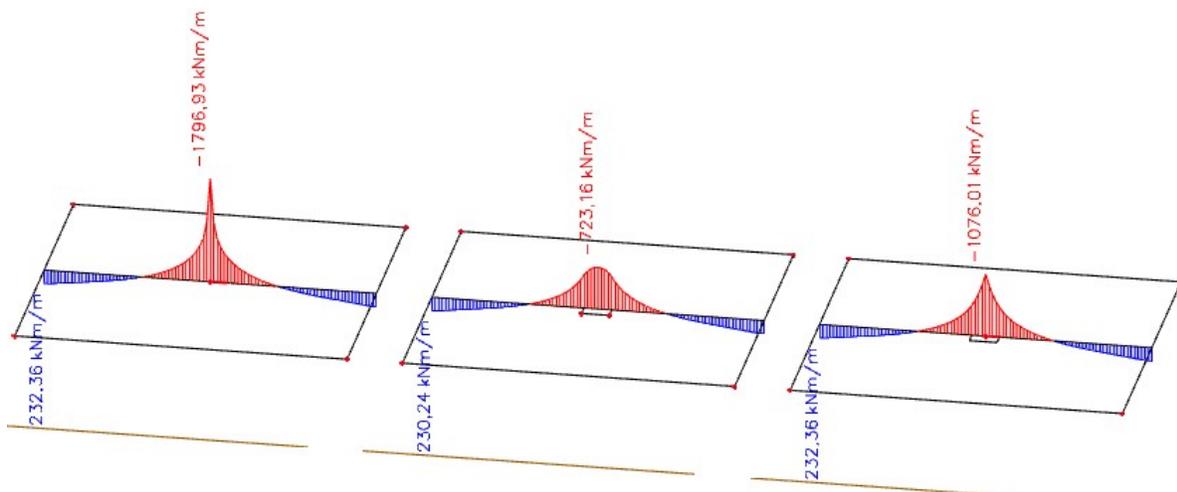
The structure is calculated with Mindlin elements. The results are claimed **in nodes, averaging.**

Element size [m]	max. mx nodal support [kNm/m]	max. mx elastic foundation [kNm/m]	max. mx Averaging strip [kNm/m]
1	840	459	-767
0,5	1078	491	932
0,35	1552	505	813
0,25	1316	693	1031
0,125	1556	722	1066
0,0625	1797	723	1076

The results are shown in following convergence curve:
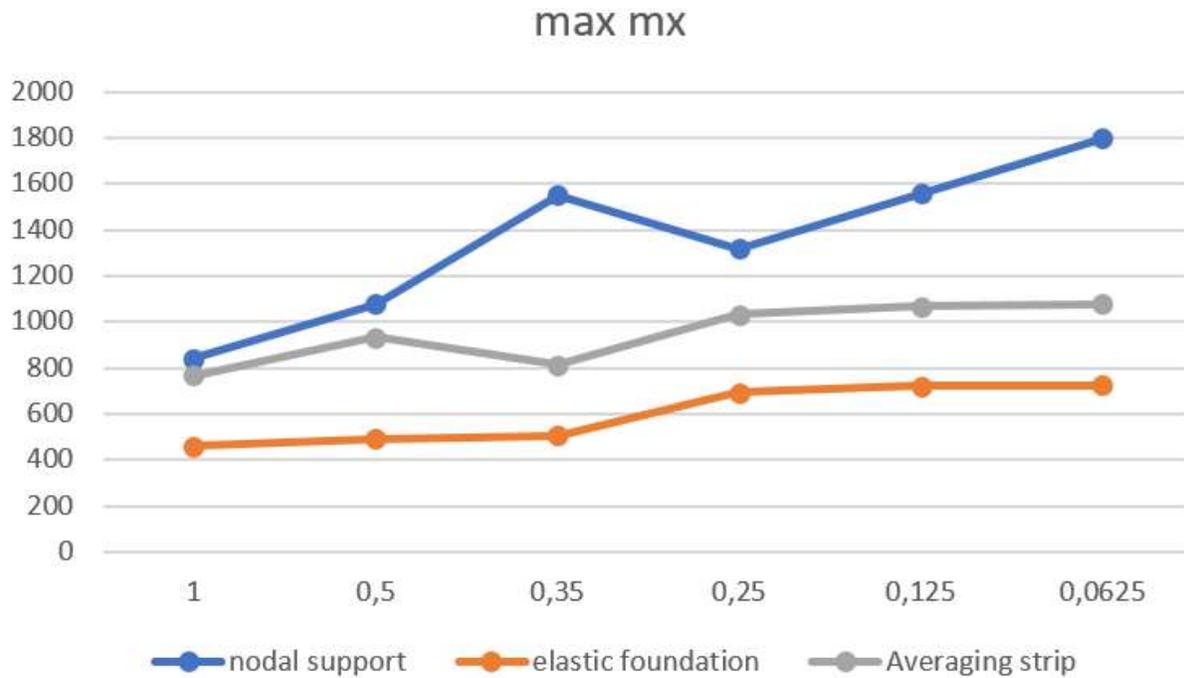
The buffing effect of the subsoil on the result is clearly noticeable.

From this, you can conclude that the subsoil will approach the reality most accurately.

Mindlin versus Kirchhoff

Shear force deformation

For the bending behaviour there are 2 types of elements implemented:

- The **Mindlin element** including shear force deformation

- The **Kirchhoff element** without shear force deformation

With the Kirchhoff element a plane section of the plate remains perpendicular to the deformed axis of the plate in the deformed state. This traditional bending theory is applied for thin plates and is supported by following assumptions (ref .[8]):

- The middle plane is free of strains and stresses

- The stress component perpendicular to the surface (σ_z) is negligible ($\sigma_z \cong 0$)

- A normal vector on the middle plane also remain perpendicular to the reference surface after the deformation (hypothesis of Bernoulli)

For this theory the following conditions have to be satisfied:

- The thickness t of the plate is small with regard to the span L (t/L < 1/5)

- The deflections w remain small in comparison to the thickness of the plate t (w/t < 1/5)

On the other hand, the Mindlin theory doesn't have one of the above-mentioned assumptions, namely: the normals on the middle plane remain straight but not necessarily perpendicular to the middle plane after deformation. As a consequence, additional strains γ_{xz} and γ_{yz} arise in case of a Mindlin element.

This is shown on the picture below.

Figure a) represents the used symbols.

Figure b) shows the Kirchhoff element.

Figure c) the Mindlin element.

On figure d) a Navier balk is demonstrated, which corresponds to the Kirchhoff element.
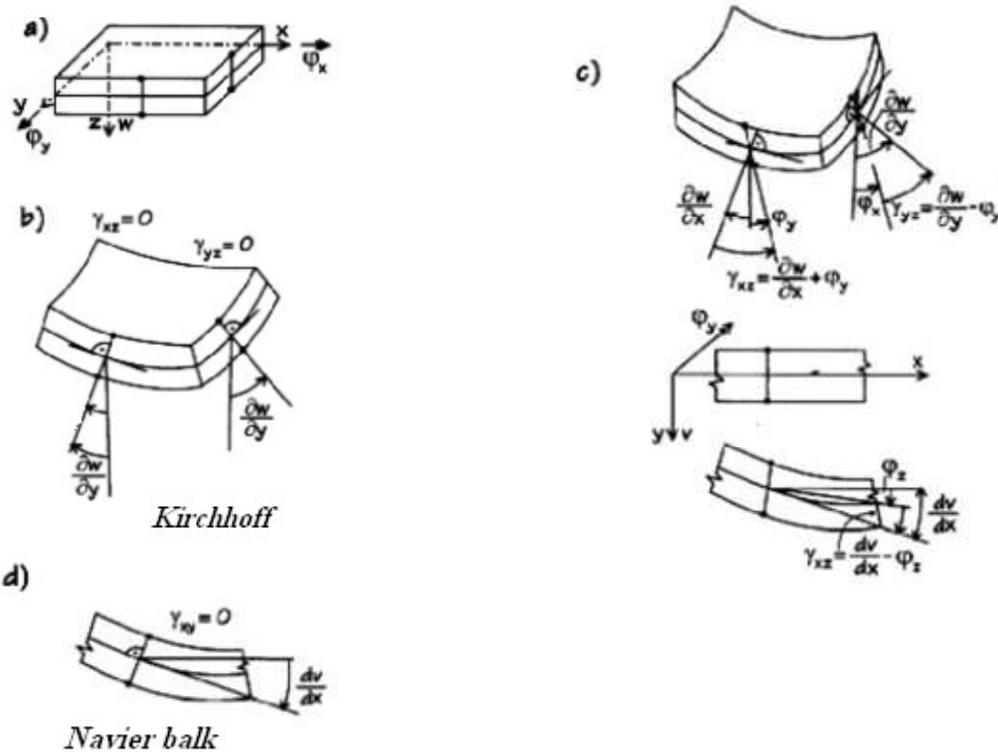
The choice between these two elements can be made using the menu function **Calculation, mesh > Solver setup.** Default the Mindlin theory is used and because of this, special attention should be paid to the use of thin plates.
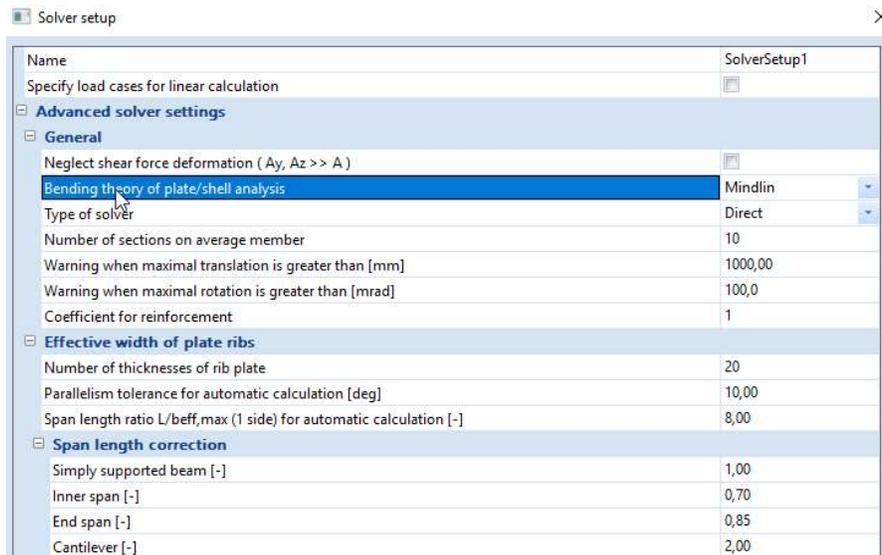

This option is only in relation with 2D elements. Specifically for beams, the shear force deformation can be taken into account or not by means of the option **Neglect shear force deformation (Ay, Az >> A)**.

The influence of the shear force deformation is especially important with thick plates with a small span.

EXAMPLE: 12_01_Shear force deformation.esa

In this example 3 plates of 2m x 5m, supported at the shortest edges and made of concrete C25/30 according to EC, are calculated. The thickness is successively 300mm, 600mm and 1200mm. A surface load of -150 kN/m2, -1200 kN/m2 and -9600 kN/m2 is applied. The mean dimension of the element is **0,5m**.
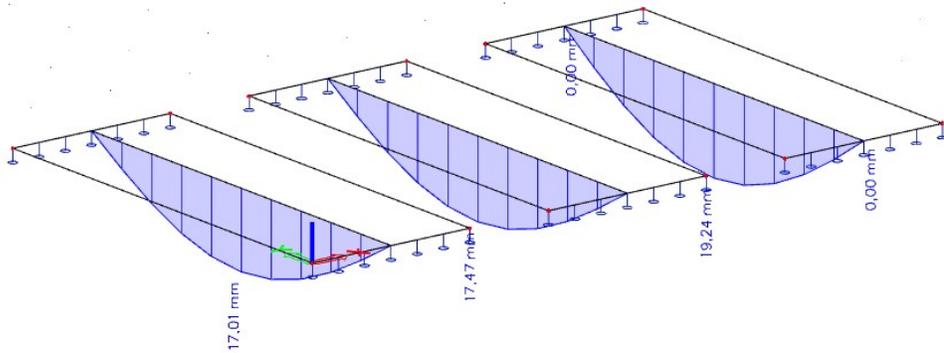

The deflection in the middle of the plate:

	Kirchhoff element	**Mindlin element**
Plate 300 mm	-16,94 mm	-17,01 mm
Plate 600 mm	idem	-17,47 mm
Plate 1200 mm	idem	-19,24 mm

## Kirchhoff versus Mindlin on the edge of an element

In the theory of Mindlin three degrees of freedom are available on the edge of a plate element:
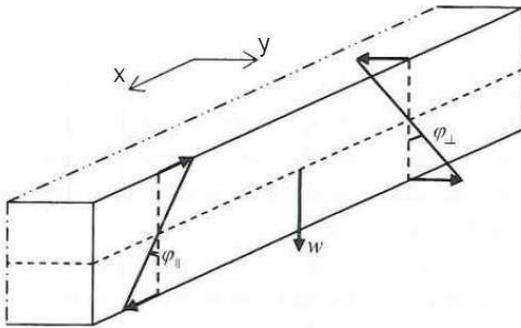
- w = deformation in the local z-direction of the plate

- $\varphi_{||}$ = rotation around ny (rotation parallel with the edge)

- $\varphi_{\perp}$ = rotation around nx (rotation perpendicular on the edge)

In Kirchhoff's theory only two variables are needed: the variable $\varphi_{||}$ does not exist, because shear deformation is not taking into account in Kirchhoff's theory.

On the edge, the following forces will be taking into account for Kirchhoff and Mindlin:
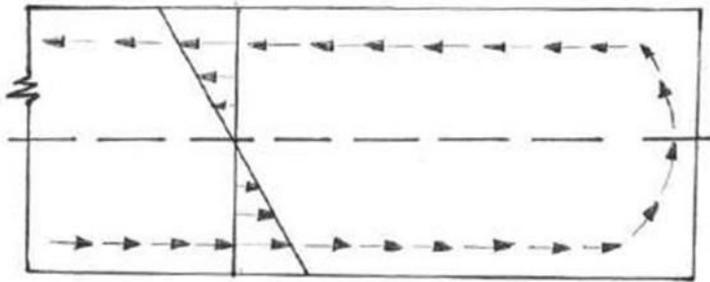
Kirchhoff	Mindlin
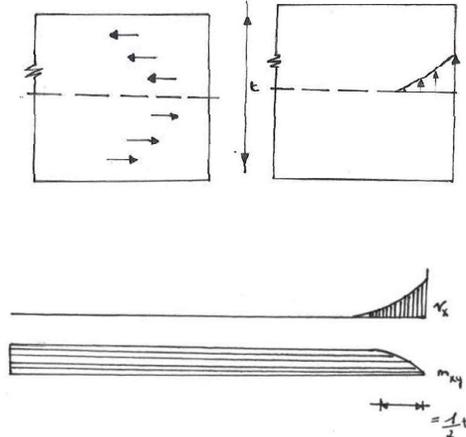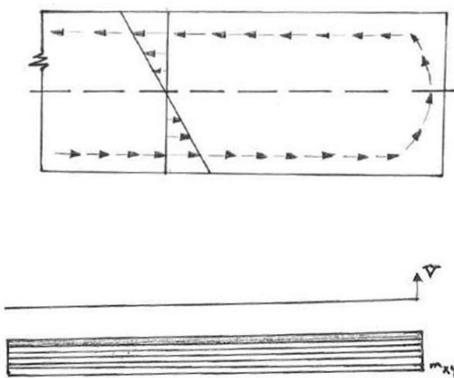
Kirchhoff assumes a constant torsional moment on the end of the plate.

Mindlin assumes that the torsional moment mxy will become zero on the edge, but this results in high values for vx and vy.

In Mindlin's theory the torsional moment will go from its maximum to zero over a distance of t/2 (t = the plate thickness). For thin plates, this is a very small area, so when using Mindlin's theory for thin plates a lot of finite elements will be necessary on the edges.

This is shown in the following example.

EXAMPLE: 13_01_Kirchhoff vs Mindlin.esa

This next example shows two plates with a different thickness (200mm and 2250mm).

The mesh of this plate is 0,5m, but on the edges a denser mesh has been inserted:
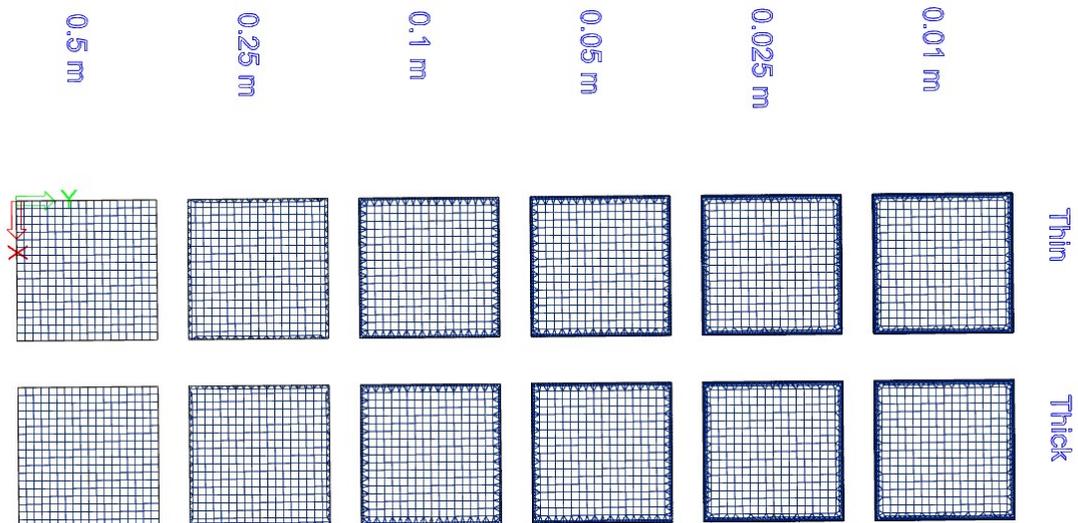
Thin plate:

| Element size edge [m]	Uz [mm]	Max |mxy| edge [kNm/m]	Max |vx| edge [kN/m]	Uz [mm]	Max |mxy| edge [kNm/m]	Max |vy| edge [kN/m]
	Mindlin			**Kirchhoff**		
0.50	6.2578	10.16	81.23	6.1911	15.08	15.14
0.25	6.2823	12.50	212.58	6.1907	15.03	16.96
0.10	6.3013	7.11	265.54	6.1907	14.97	16.36
0.05	6.3188	4.65	236.80	6.1908	14.98	21.91
0.025	6.3256	1.52	243.01	6.1908	14.99	19.35
0.01	6.3269	0.80	240.30	6.1907	14.99	19.61
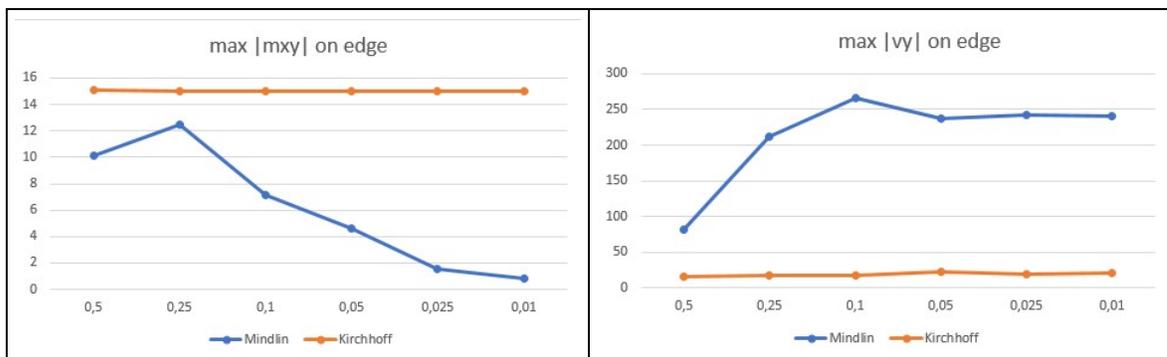
The deformation Uz for Mindlin and Kirchhoff (in the middle of the plate) are similar and do not depend on the edge mesh size.

As explained before will Mindlin results in a zero mxy using small elements.
The comparison between Mindlin and Kirchhoff is made in the diagram below:

When inserting a lot of elements, Mindlin will get mxy very small for thin plates.
In this example a mesh of minimum 0,025m is needed.
This is an unreasonable small mesh to calculate with.

The small value for vy at Kirchhoff's calculation is clearly noticeable, even with a small number of elements. But when using Mindlin, then vy will reach high values.

In the case of thin plates, calculating with Kirchhoff is a better option, because Mindlin does not get satisfactory results, unless when inserting an unreasonable small mesh on the plate borders.

Thick plate:

Element size edge [m]	Uz [mm]	Max |mxy| edge [kNm/m]	Max |vx| edge [kN/m]	Uz [mm]	Max |mxy| edge [kNm/m]	Max |vy| edge [kN/m]
	Mindlin			**Kirchhoff**		
0.50	0.0065	2.41	16.20	0.0043	15.08	15.14
0.25	0.0065	1.70	17.44	0.0043	15.03	16.96
0.10	0.0065	0.91	17.14	0.0043	14.97	16.36
0.05	0.0065	0.37	16.95	0.0043	14.98	21.91
0.025	0.0066	0.15	16.62	0.0043	14.99	19.35
0.01	0.0066	0.08	16.52	0.0043	14.99	19.61
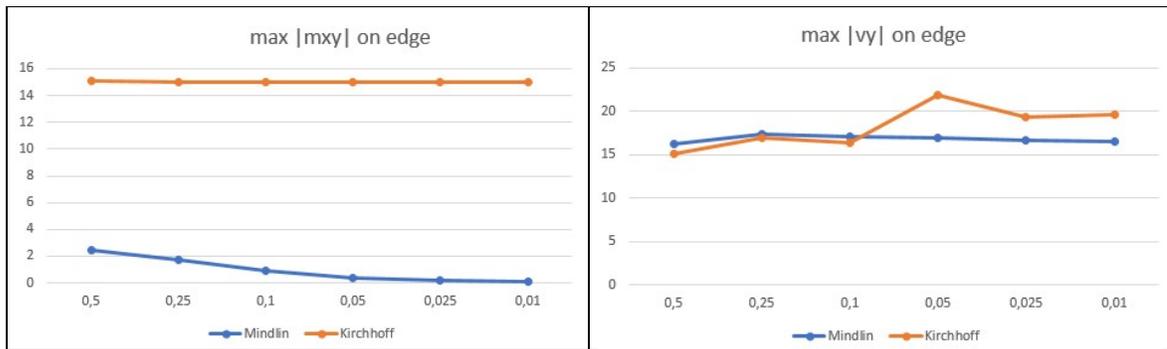
Deformations in the middle of the plate are different. In the case of Mindlin the shear force deformation is taken into account, and this leads to more accurate results for thick plates.

When calculation with Mindlin mxy reaches low values, even with a small number of elements at the borders (a mesh of 0,5m).

The values for vy are similar between Mindlin and Kirchhoff.

So, for thick plates, calculating with Mindlin will give the best results, because shear force deformation will have more importance and mxy will go to zero, even for a small number of elements.

Conclusion

Thin plates

- Calculating with Kirchhoff gives the best results for thin plates
- Using Mindlin dense mesh will be necessary on the edges to obtain good results.
- Using Kirchhoff, the size of the elements does not have to be smaller than the plate thickness.

Thick plates

- Calculating an isotropic, homogeneous plate, Mindlin will be necessary
- On the edge, a denser mesh will be necessary (more than 5 elements over the half of the plate thickness)
- Mindlin will also give reliable results for thin orthotropic plates with a small shear stiffness

Ribs

Introduction

By means of the menu **Structure > 2D element components> Plate rib** a plate can be stiffened with members.

A rib is calculated as a beam with eccentricity with regard to the axis of the plate. The member elements are connected to the plate at the height of the mesh nodes.

In a 2D plate project, a rib can only be inputted in the middle of the plate. In a 3D General project, the rib can also be placed below or above the plate. After all, a rib that lies below or above the plate causes membrane forces in the plate. Membrane forces are not present in 2D plates, only in 3D shells.

In SCIA Engineer a rib below a plate is always shear resistant connected to the plate.

The total rigidity is according to the rule of Steiner:

Rigidity beam + Rigidity plate + Surface beam x (axis-distance beam-plate)^2.

So, it is important to realize that also in reality the beam and the plate must be connected shear resistant to each other. When analyzing a prefab construction where the plate is on top of the beam, then the beam should be placed in the middle of the plate in the calculation model.

The collaborating width of the rib is calculated implicitly by the behavior of the finite elements under membrane forces during the Finite Elements Calculation. In the following view of the membrane forces nx in the longitudinal direction of the beam, the effective width is clearly noticeable.
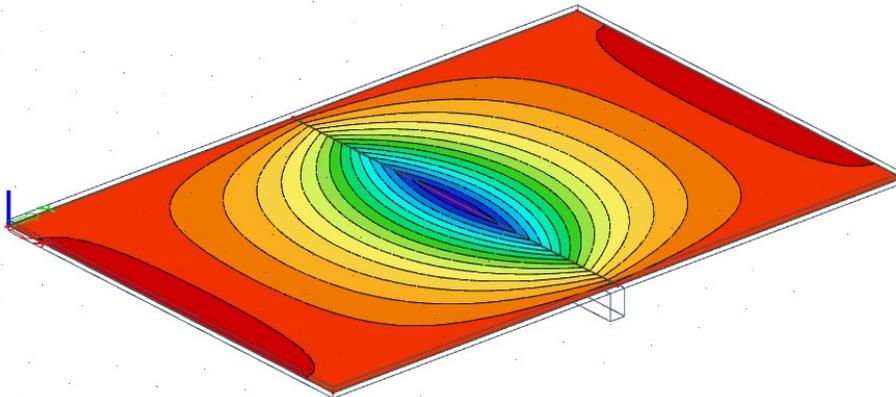
The section of the rib can be shown graphically, in that way you can see if the effective widths overlap each other or not.

This can be done by means of view parameters, namely 'Set view parameters for all > Structure > Draw cross-section'.
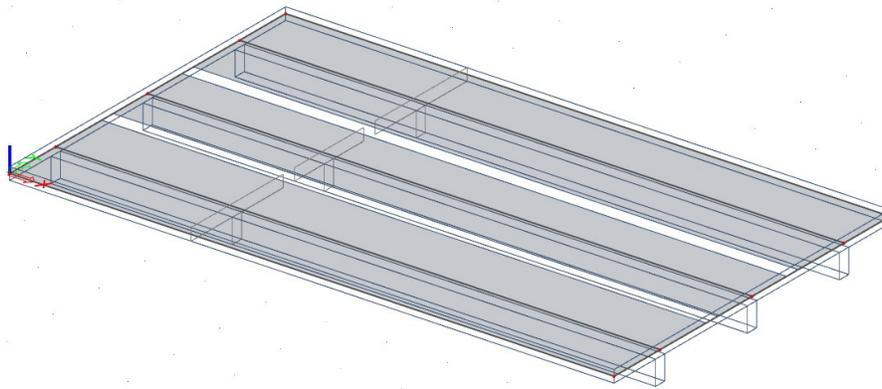
Replacement T-section

What is explained in the previous paragraph is also valid for a member that is connected to an **Internal edge** of the plate and has been aligned eccentrically. The difference with a plate rib is that for a rib an **Effective Width** can be inserted too.

The Effective Width was specifically implemented to follow the code concerning the calculation of the theoretical reinforcement. Because when the option **Rib** is marked with the results, a replacement T-section is used to calculate the results. The height of the T-section is determined by the height of the beam + the height of the plate. The flange width of the T-section equals the entered Effective Width.

The internal forces for the replacement T-beam are calculated as follows:

T	the heart of the entire replacement T-section
T1	the heart of the left part of the effective width
T2	the heart of the right part of the effective width
T3	the heart of the original rib
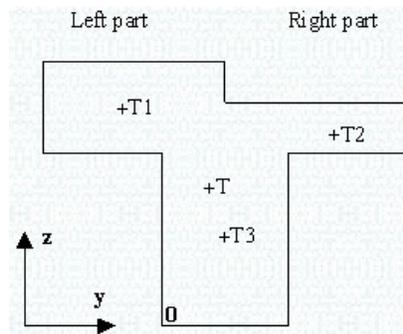
The coordinates of the hearts are used as lever arms in the Y and Z direction:

Lever arm $Z1 = T1z - Tz$	Lever arm $Y1 = T1y - Ty$
Lever arm $Z2 = T2z - Tz$	Lever arm $Y2 = T2y - Ty$
Lever arm $Z3 = T3z - Tz$	Lever arm $Y3 = T3y - Ty$
Lever arm $Z = Tz - 0z$	Lever arm $Y = Ty - 0y$

- **N** = N beam + N plate, left + N plate, right

- **Vy** = Vy beam + Vy plate, left + Vy plate, right

- **Vz** = Vz beam + Vz plate, left + Vz plate, right

- **Mx** = Mx beam + Mx plate, left + Mx plate, right

- **My** = My beam + My plate, left + My plate, right
  + N plate, left * (Lever arm Z1)
  + N plate, right * (Lever arm Z2)
  + N beam * (Lever arm Z3)

- **Mz** = Mz beam + Mz plate, left + Mz plate, right
  + N plate, left * (Lever arm Y1)
 + N plate, right * (Lever arm Y2)
  + N beam * (Lever arm Y3)

If the option **Rib** is activated when visualizing the plate forces, then the internal forces in the cooperating width of the rib are set to zero. This counts for the internal forces in the longitudinal direction of the rib. The forces perpendicular to the rib remain unchanged.

These internal forces can be set to zero for the reinforcement calculation because they are already considered in the reinforcement calculation of the rib. Thus, the whole plate-beam is replaced by a T-beam.

However, note that when using several ribs below a plate element, the cooperating widths should not overlap each other. If this does happen, then the values of the internal forces are counted double on the areas of the overlapping parts.

In this example, a beam is calculated with a length of 10m and concrete quality C25/30 according to EC. The beam is supported at the extremities, loaded with a distributed load of 200kN/m and has following section:
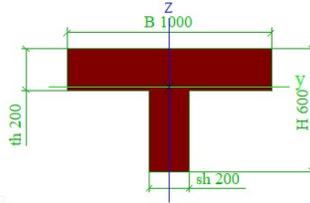
The beam is modeled in 3 separate ways:

- As member element
- As plate with a thickness of 200mm with a rib of 200mm x 400mm below the plate
- Entirely with Finite Elements
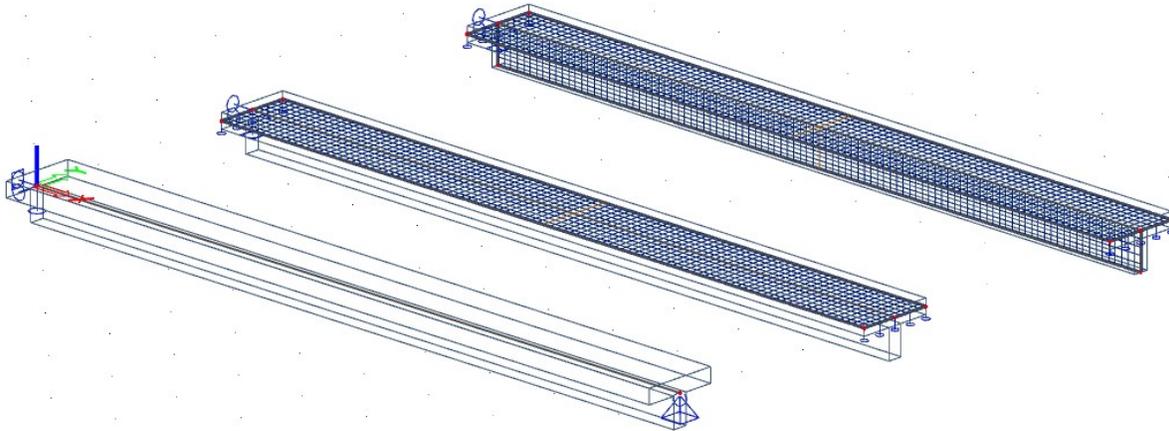
The whole is calculated with an average element size of **0,1m**. The results are visualized **in nodes, averaging on macro.**

	Beam	Plate + Rib	Difference [%]	Finite Elements	Difference [%]
Maximal bending in the middle [mm]	120,9	122,6	1,4 %	122,4	1,2 %
Maximal moment in the middle [kNm]	2500	2499,85	0,0 %	---	---
Maximal shear force at the end [kN]	1000	992.56	0,7 %	990	1,0 %

The results indicate that the calculation model with rib and the Finite Elements model approach the beam well. With the Finite Elements model however, a higher deviation occurs concerning the shear force at the end, but when refining the mesh more, this difference becomes minimal.

However, ribs that are modeled this way have a few restrictions. These are explained in the next paragraphs.

Boundary conditions

As already said at the introduction of this chapter, membrane forces in the plane of the plate (compression forces in the field) arise because of the rib below the plate. The plate is here shorter in the longitudinal direction.

The supports entered on a plate + rib are located at the axis height of the plate. This is not the axis of the T-section formed by the plate and the beam.

Concretely this means that if supports are input in the longitudinal direction of the plate, these supports will prevent the reduction of the plate. This lead to large reaction forces and a smaller bending since the rib will be more rigid.

EXAMPLE: 14_02_BOUNDARY CONDITIONS.esa

In this example, the same beam from the example 14_01_Rib.esa is considered. However, the plate with rib is now tied up to both ends in the X, Y and Z direction.
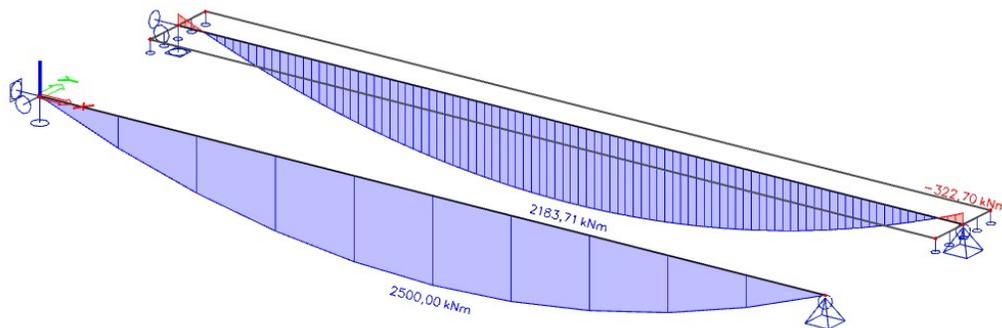
	Beam	Plate + Rib	% Difference
Maximal bending in the middle [mm]	120,9	104,0	13,98 %
Maximal moment in the middle [kNm]	2500	2183,7	12,65 %
Maximal shear force at the end [kN]	1000	999,17	0,08 %

The effect on the bending mentioned above is clearly noticeable. The reduction of the field moment can be ascribed to the moment that occurs at the ends. This moment is caused by the eccentricity of the central axis of the T-section in relation to the support.

The moment can be easily calculated:

The normal force in the T-section amounts to 3688 kN

The levers arm Z1 = 86 mm

$\Rightarrow$My = 3688 kN x 0,086 mm = **317,16 kNm** $\approx$ 322,70 kNm

Torsion

The calculation of a rib as a beam with eccentricity regarding the plate provides less accurate results for torsion.
The reason for this is that the torsional stiffness of the rectangular beam and the torsional stiffness of the plate added up do not give the torsional stiffness of the T-beam.

EXAMPLE: 14_03_TORSION.esa

In this example, the same beam from the example 14_01_Rib.esa is considered. The beam is clamped at the beginning and is free at the end. The beam is loaded with a distributed torsional moment of 50 kN/m.

To obtain an accurate reference result, the torsional moment of inertia of the T-section is calculated exactly by way of a Finite Elements calculation with element size **5mm.**
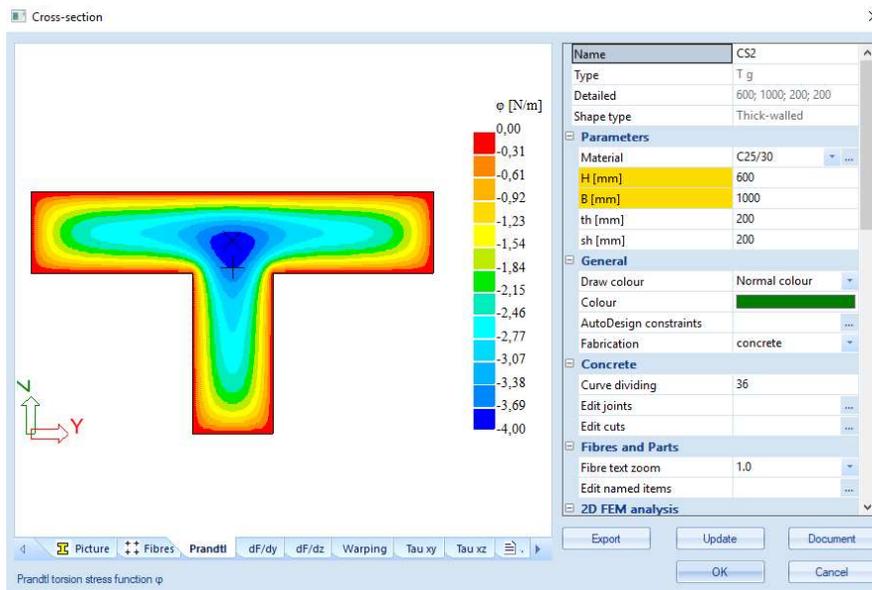
	Beam	Plate + Rib	% Difference	Finite Elements	% Difference
Maximal rotation Fix at the end [mrad]	52.1	59.3	13,8 %	55.4	6,3 %

The results show that the deviation is quite significant.
If torsion is important, it is advisable to switch to a Finite Elements model.

Shear force deformation

The shear force deformation in a beam is charged automatically in SCIA Engineer. The shear area Az is qualifying for the shear force deformation. This quantity is calculated accurately for the T-section.

With a rib, Az is calculated for the rectangular cross-section. When using the Mindlin element, the shear force deformation of the plate is also taken along. The two separate calculations do not have the same result as a T-section.

Shear force deformation only has an influence with high beams with short spans and here the influence remains restricted. This deviation is in most cases of less importance.

Line support

A plate with ribs is supported by those ribs at the height of the internal edge that connects both parts. However, the plate is supported over the width of the beam.

This distinction gives a difference in the moments in the plate in the direction perpendicular to the rib. This should be considered with ribs that have a large width or with ribs that consist of a reversed U cross-section (as with a tubular bridge).

EXAMPLE: 14_04_TUBULAR BRIDGE.esa

In this example, the tubular bridge is modeled. The different elements have a thickness of 22mm and are made from concrete C25/30 according to EC. The bridge deck has a length of 20m, a width of 5 m and is supported by 3 ribs with a height of 1 m and a width of 1m. The bridge is single supported at both sides. On the bridge deck, a uniform distributed load of 50kN/m² happens.

The whole is calculated with an average element size of **0,25m.** The results are visualized **in nodes, avg. on macro** in two sections in the middle of the bridge.
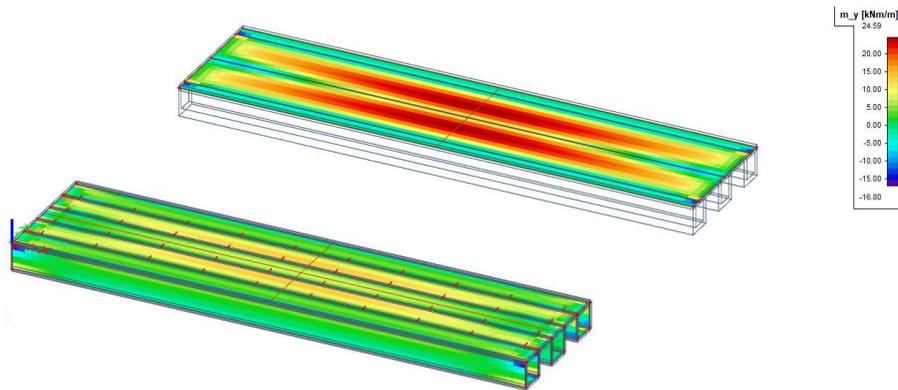
	Finite Elements	Plate + Ribs	% Difference
Maximal bending Uz in the longitudinal direction [mm]	35.0	39.5	12,9 %
Maximal longitudinal moment mx in the plate [kNm/m]	20.0	24,5	22,5 %
Maximal shear moment my in the plate [kNm/m]	13,3	24,6	85,0 %

In the Finite Elements model the supports are attached to the bridge deck at the correct position. With the model with ribs, the connection line is situated between rib and plate in the middle of the U-section, which does not correspond to the reality.

The results clearly show that the bridge deck is better support by the Finite Elements model, with lower shear moments as a result. In the model with ribs, the headway of the supports is much bigger with bigger shear moment as a result.

Orthotropy

Theory

In a 3D model, the following components of deformations appear in each point (respectively the deformations according to the x-, y- and z-axes):

$$u(x, y, z)$$

$$v(x, y, z)$$

$$w(x, y, z)$$

From these deformations, the following strains can be calculated:

$$\varepsilon = [\varepsilon_x, \varepsilon_y, \varepsilon_z, \varepsilon_{xy}, \varepsilon_{xz}, \varepsilon_{yz}]^T$$

$$\varepsilon_x = du/dx$$

$$\varepsilon_y = dv/dy$$

$$\varepsilon_z = dw/dz$$

$$\varepsilon_{xy} = 1/2\, \gamma_{xy} = 1/2\,(dv/dx + du/dy)$$

$$\varepsilon_{xz} = 1/2\, \gamma_{xz} = 1/2\,(du/dz + dw/dx)$$

$$\varepsilon_{yz} = 1/2\, \gamma_{yz} = 1/2\,(dw/dy + dv/dz)$$

The stresses in each point are:

$$\sigma = [\sigma_x, \sigma_y, \sigma_z, \sigma_{xy}, \sigma_{xz}, \sigma_{yz}]^T$$

The stresses and strains are related to each other, in the simplest case this relation is linear (Hooke's law):

$$\sigma = D\, \varepsilon$$

D is a 6x6 matrix. The connection between stresses and strains is not based on assumptions, but describes the real physical behavior of the material. For that reason, this matrix is called the "constitutive" matrix.

With the reduction to a 2D plate, the stresses are replaced by internal forces **s**. These internal forces **s** are known as the results of SCIA Engineer:

$$s = [s_m^T, s_b^T]$$

$$s_m = [n_x, n_y, q_{xy}]^T \text{ for membrane forces}$$
$$s_b = [m_x, m_y, m_{xy}, q_x, q_y]^T \text{ for bending}$$

The components of the deformations that are used with a 2D plate are the deformation of the axis of the plate, the rotation on the x-axis and the rotation on the y-axis:

$$w(x, y) = w(x, y, 0)$$

$$\phi_x(x, y)$$

$$\phi_y(x, y)$$

With the Kirchhoff element, the normal on the plate axis remains perpendicular to the plate axis. So, there is a double connection between w and ϕ:

$$\phi_y = -dw/dx$$

$$\phi_x = dw/dy$$

With the Mindlin element the shear force deformations γ_{xz} and γ_{yz} also occur:

$$\phi_y = -dw/dx + \gamma_{xz}$$

$$\phi_x = dw/dy + \gamma_{yz}$$
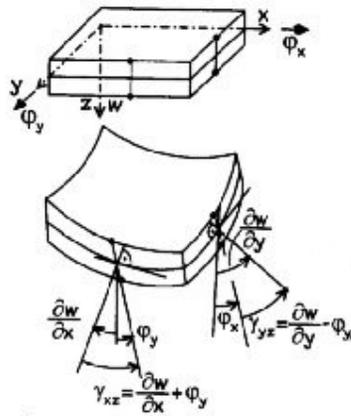
From these 3 components of the deformation the strain can be calculated in each point of the plate (with the usual assumption that an even cross-section remains plane). From this strain, the stress can be calculated in each point of the plate by means of the constitutive matrix. Through integration of these stresses over the thickness of the plate, the internal forces that belong to the deformation can be calculated (for the full calculation is referred to ref. [2]).

This gives the following connection for the <u>membrane forces</u> and deformations in the plane:

$$\begin{bmatrix} n_x \\ n_y \\ q_{xy} \end{bmatrix} = \begin{bmatrix} d_{11} & d_{12} & d_{13} \\ d_{12} & d_{22} & d_{23} \\ d_{13} & d_{23} & d_{33} \end{bmatrix} * \begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{bmatrix}$$

For <u>bending components</u> and deformations from the plane:

$$
\begin{bmatrix} m_x \\ m_y \\ m_{xy} \\ q_x \\ q_y \end{bmatrix} = \begin{bmatrix} D_{11} & D_{12} & D_{13} & 0 & 0 \\ D_{12} & D_{22} & D_{23} & 0 & 0 \\ D_{13} & D_{23} & D_{33} & 0 & 0 \\ 0 & 0 & D_{44} & D_{45} \\ 0 & 0 & D_{45} & D_{55} \end{bmatrix} * \begin{bmatrix} \varphi'_y \\ -\varphi^{\bullet}_x \\ (\varphi^{\bullet}_y - \varphi'_x) \\ \gamma_{xz} \\ \gamma_{yz} \end{bmatrix}
$$

'means the derivative to x, •, means the derivative to y. φ'_y en $-\varphi^{\bullet}_x$ are curves.

In most textbooks, the shear force deformation is neglected. Then:

$$\varphi'_y = -d^2w/dx^2 = \text{curve } \kappa_{xx}$$

$$-\varphi^{\bullet}_x = -d^2w/dy^2 = \text{curve } \kappa_{yy}$$

$$\varphi^{\bullet}_y - \varphi'_x = -d^2w/dxdy - d^2w/dxdy = -2\,d^2w/dxdy = \text{curve } 2\,\kappa_{xy}$$

The matrix for the bending effects is subsequently written as:

$$
\begin{bmatrix} m_x \\ m_y \\ m_{xy} \end{bmatrix} = \begin{bmatrix} D_{11} & D_{12} & D_{13} \\ D_{12} & D_{22} & D_{23} \\ D_{13} & D_{23} & D_{33} \end{bmatrix} * \begin{bmatrix} \kappa_{xx} \\ \kappa_{yy} \\ 2\kappa_{xy} \end{bmatrix}
$$

By dividing the membrane force components and the bending components, it is implicitly assumed that these components do not mutually influence each other.

These stiffness matrixes do not only describe the physical behavior of the material, but also the stiffness of a plate element. This is specified by the material, possibly different materials over the thickness (reinforced concrete, laminated plates) and by the geometry (ribs, …).

In SCIA Engineer the following components are entered in this matrix:

$$d_{11}, d_{22}, d_{33} \text{ and } d_{12}$$
$$D_{11}, D_{22}, D_{33}, D_{44}, D_{55}, D_{12}$$

D_{44} and D_{55} are added because Mindlin elements with shear force deformations are used. In many cases there are no simple formulas to calculate these stiffness's.

The orthotropic parameters can be calculated by means of following formulas:

for *plate elements*:

$$D_{11} = \frac{E_1 \cdot h^3}{(12(1 - \nu_{12} \cdot \nu_{21}))}$$

$$D_{22} = \frac{E_2 \cdot h^3}{(12(1 - \nu_{12} \cdot \nu_{21}))}$$

$$D_{12} = D_{21} = \nu_{21} \cdot D_{11} = \nu_{12} \cdot D_{22}$$

$$D_{33} = \frac{G_{12} \cdot h^3}{12}$$

$$D_{44} = G_{13} \cdot h$$

$$D_{55} = G_{23} \cdot h$$

G_{13} and G_{23} are used for the calculation of the stiffness's D_{44} and D_{55}. These are the stiffness's for shear force deformation. In some cases, they cannot be calculated exactly. In that case it is advised to enter D_{44} and D_{55} much larger (1000 times larger) than the other stiffness's.
In this way, you will neglect the shear force deformation. The influence of the shear force deformation is restricted with normal plate thicknesses/stresses.
The best method to have a better approach for G13 and G23 is to calculate with following formulas:

$$G_{13} = \frac{E_1}{2 \cdot (1 + \nu_{12})}$$

$$G_{23} = \frac{E_2}{2 \cdot (1 + \nu_{21})}$$

for *"wall" elements:*

$$d_{11} = \frac{E_1 \cdot h}{(1 - \nu_{12} \cdot \nu_{21})}$$

$$d_{22} = \frac{E_2 \cdot h}{(1 - \nu_{12} \cdot \nu_{21})}$$

$$d_{33} = G_{12} \cdot h$$

$$d_{12} = d_{21} = \nu_{21} \cdot d_{11} = \nu_{12} \cdot d_{22}$$

Shell elements have both characteristics of a plate element as from a "wall" element. That way all physical constants, as described above, need to be applied.

A real example is the use of floor plates that wear out in only one direction. With this, you can use orthotropic parameters. In the two directions, several stiffness's need to be applied, to which you can attribute a quasi-neglected stiffness to the shear direction.

Another method to model this real example can be done as follows: you reduce the measurements of the plate a bit so they just fail to hit the non-supporting beams. What's more, you attribute a Poisson coefficient of 0 to the plate material.

A plate that is respectively torn and not torn in the X and the Y direction can also be modeled as a plate with orthotropic parameters. This way a different E-module can be applied in both directions.

Orthotropic parameters: isotropic plate

The stiffness of an isotropic plate is entirely defined by the following parameters:

E (modulus of elasticity), **ν** (Poisson coefficient) and **h** (thickness)

For the membrane effects in an isotropic plate applies:

$$\sigma_x = E(\varepsilon_x + \nu\varepsilon_y)/(1-\nu^2)$$
$$\sigma_y = E(\varepsilon_y + \nu\varepsilon_y)/(1-\nu^2)$$
$$\tau_{xy} = G\gamma_{xy}$$

By integration over the thickness is n_x, n_y en q_{xy} in function as ε_x, ε_y and γ_{xy} obtained and so d_{11}, d_{22}, d_{33} and d_{12}:

$$d_{11} = d_{22} = Eh/(1-\nu^2), \qquad d_{33} = Gh,$$
$$d_{12} = \nu d_{11}, \qquad\qquad d_{13} = d_{23} = 0.$$

For bending:

$$D_{11} = D_{22} = Eh^3/(12-12\nu^2), \qquad D_{33} = Gh^3/12,$$
$$D_{44} = D_{55} = Gh/\beta, \qquad\qquad D_{12} = \nu D_{11},$$
$$D_{13} = D_{23} = D_{45} = 0$$

For a constant course of the shear stress τ_{xz}, τ_{yz} over the height h is $\beta = 1$. From the analogy with a rectangular cross-section follows the assumption that from a parabolic stress course and a coefficient $\beta = 1,2$.

EXAMPLE: 15_01_ISOTROPIC.esa

In this example, the stiffness parameters are calculated for an isotropic plate. The results of an orthotropic plate with these properties are compared to the results of an isotropic plate.

Project data
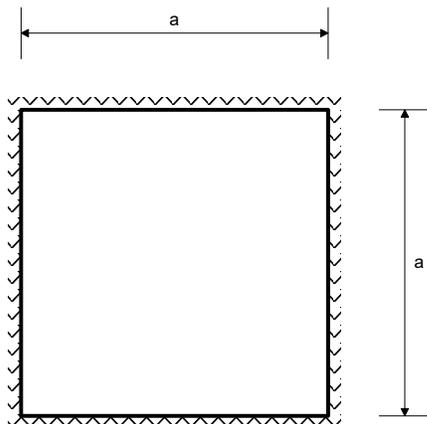
Length: a = 10m
Thickness: h = 400mm
Distributed load: q = -10 kN/m^2
Concrete: C25/30 according to EC
E = 31500 N/mm^2
ν = 0,2

Reference

S.P. Timoshenko and S. Wotnowsky-Krieger, *Theory of Plates and Shells*, 2nd edition, Mc Graw Hill, New York, 1987.

Calculation Stiffness parameters

$D_{11} = D_{22} = E\, h^3 / (12\, (1-v^2)) = 31500 \times 400^3 / (12 \times (1 - 0.2^2)) = 1.75\ \text{E+11 Nmm} = 175\ \text{MNm}$

$D_{12} = D_{21} = v\, D_{11} = \ldots = 35\ \text{MNm}$

$D_{33} = E\, h^3 / (24\, (1+v)) = \ldots = 70\ \text{MNm}$

$G = E / ((2\, (1+ v)) = \ldots = 13125\ \text{MPa}$

$D_{44} = D_{55} = G\, h / 1.2 = \ldots = 4375\ \text{MN/m}$

The plate is simply supported on four sides and calculated with an average element size of **1m** without averaging strips on the edges.
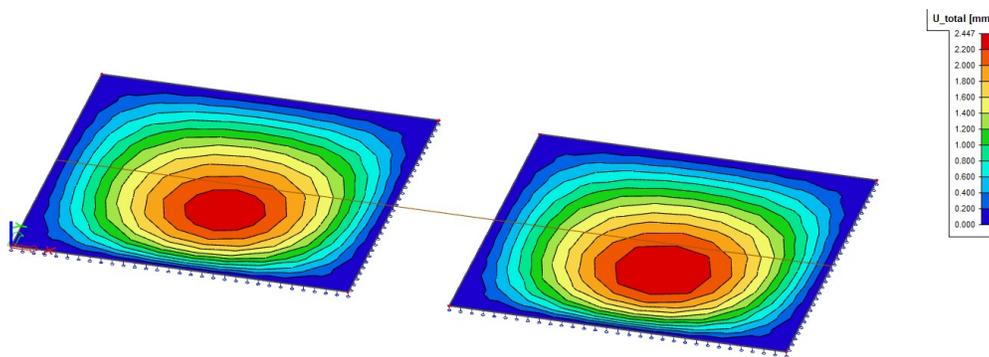
The results are claimed **in nodes, avg.** for a section in the middle of the plate.

	Isotropic	Orthotropic	% Difference
Deformation Uz [mm]	-2,370	-2,369	0,00 %
Moment mx [kNm/m]	44,67	44,67	0,00 %

# Orthotropic parameters: Practical cases

In the literature, there are numerous examples available for the calculation of these coefficients in practical cases. In this paragraph, a few specific cases are explained. For more information see ref. [1].

EXAMPLE: 15_02_FLOORWITHRIBSa.esa and 15_02_FLOORWITHRIBSb.esa

In this example, a floor with ribs is modelled. The plate has a thickness of 200mm and dimensions of 10m by 6m. In the longitudinal direction, the plate is stiffened with ribs of 400mm height and 200mm width.

The headway of the ribs is 2m. The whole is made of concrete C25/30 according to EC. The floor is loaded with a consistent distributed load of 5 kN/m².

Project Data
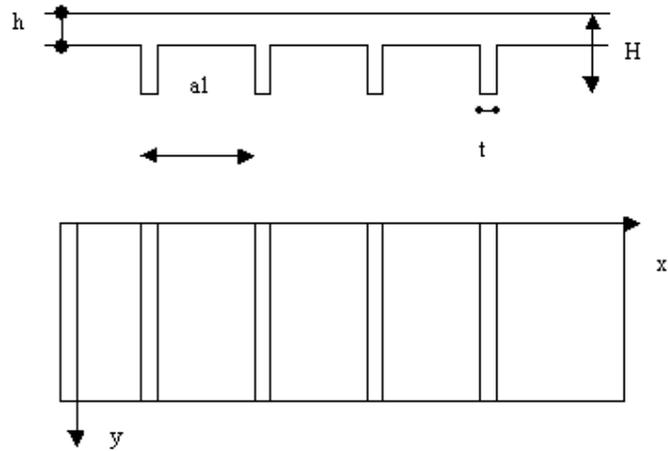
h = 200mm   t = 200mm

a1 = 2000mm   H = 600mm

Calculation stiffness parameters

$$D_x = \frac{E * a1 * h^3}{12(a1 - t + \alpha^3 * t)}$$

$$D_{44} = \frac{G * Az}{a1}$$

$$D_y = \frac{E * I}{a1}$$

$$D_{55} = \frac{G * h}{1,2}$$

$$D_1 = 0$$

$$D_{xy} = D'_{xy} + \frac{C}{2 * a1}$$

$$D'_{xy} = \frac{E * h^3}{12(1 + v)}$$

With:

     D'_{xy} = torsional stiffness of the plate without rib

     Az = shear area. T-section with width a1

     C = torsional stiffness one rib = $G * I_t$

     α = h / H

To calculate I, Az and I_t exactly, a Finite Elements calculation is performed of such a T-section with element size of **5mm.**

Slope dφ/dz of the torsion stress function
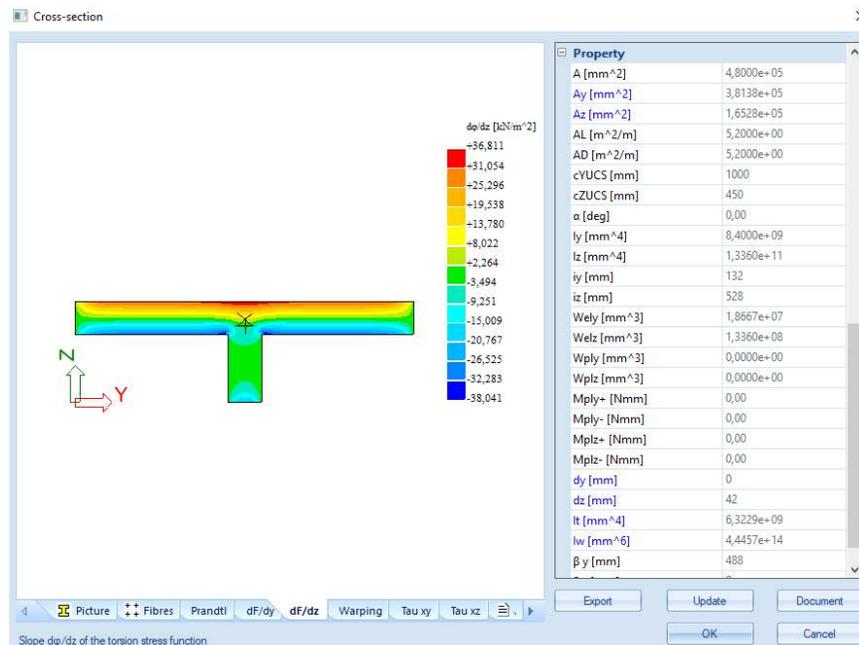
From this follows:

I = 8400000000 mm⁴			Material: C25/30	

I = 8400000000 mm⁴   Material: C25/30

Az = 165280 mm²   =>   E = 31500 N/mm²

It = 6322900000 mm⁴ ν = 0,2

C = G * It = 8,2988 e^{13} Nmm²   G = 13125 N/mm²

The formulae can be filled in now. According to the figure, the Y-axis lies in the longitudinal direction of the ribs. The ribs are entered in SCIA Engineer according to the X-axis. Specifically, this means that the formula for D_y matches in this case with D_{11}.

D11 = Dy = 31500 x 8400000000 / 2000 = 132.3 MNm

D22 = Dx = … = 23.24 MNm

$$D_{22} = D_x = \frac{30500 * 2000 * 200^3}{12\left(2000 - 200 + (200/600)^3 * 200\right)} = 22500000000 \, Nmm = 22,5 MNm$$

$$D_{12} = D1 = 0 MNm \quad \text{in SCIA ENGINEER a very small value is entered.}$$

$$D_{33} = D_{xy} = \frac{30500 * 200^3}{12(1 + 0,2)} + \frac{7,8678 e^{13}}{2 * 2000} = 36613980225,6 \, Nmm = 36,61 MNm$$

$$D_{44} = \frac{12708,33 * 169930}{2000} = 1079763,54 \, N\!\!\Big/\!mm = 1079,76 \, MN\!\!\Big/\!m$$

73

$$D_{55} = \frac{12708,33 * 200}{1,2} = 2118055,56 \, {N}\!/\!{mm} = 2118,06 \, {MN}\!/\!{m}$$

The floor with ribs is entered is SCIA Engineer in two ways: as orthotropic plate (Floorwithribsa) and as a plate with ribs (Floorwithribsb). The whole is calculated with the average element size of **0,25m** without averaging strips.

Maximal bending Uz in a section in the middle of the plate:

- Plate with ribs: **-5,110 mm**

- Orthotropic plate: **-5,111 mm**

Moment mx in a section in the middle of an orthotropic plate, claimed **in nodes, averaging**:

mx = 62,55 kNm/m, over a cooperative width of 2m gives this: **125,1 kNm**

Moment My in the middle rib of the plate with ribs:

My = **122,94 kNm**

The results show that the orthotropic parameters describe the behaviour of this stiffened plate well.

EXAMPLE: 15_03_GRID.esa

In this example, a grid is modeled. The frame consists of steel IPE 200 profiles and has dimensions of 14m by 14m. The headway between the beam amounts to 1m in both directions. The whole is made of S235 according to EC and is loaded with a surface load of 1kN/m².

Project Data
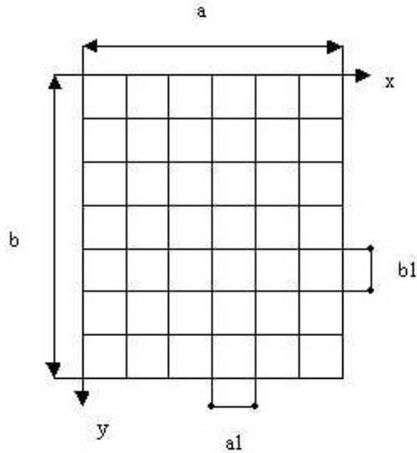
a = 14000mm
a1 = 1000mm
b = 14000mm
b1 = 1000mm

Calculation stiffness parameters

$$D_x = \frac{E_1 * I_1}{b1}$$

$$D_y = \frac{E_2 * I_2}{a1}$$

$$D_1 = 0$$

$$D_{xy} = \frac{1}{4}\left(\frac{C_1}{b1} + \frac{C_2}{a1}\right)$$

with:

   $E_1 = E_2 = 210000$ N/mm²

   $I_1 = I_2 = 19430000$ mm⁴

   $C_1 = C_2 = G * It = 5637692254$ Nmm²

   $G = 80769,23$ N/mm²

   $It = 69800$ mm⁴

From this follows:

$$D_{11} = D_x = \frac{210000*19430000}{1000} = 4080300000 Nmm = 4,0803 MNm$$

$$D_{22} = D_y = \frac{210000 * 19430000}{1000} = 4080300000 Nmm = 4,0803 MNm$$

$$D_{33} = D_{xy} = \frac{1}{4}\left(\frac{80769,23 * 69800}{1000} + \frac{80769,23 * 69800}{1000}\right) = 281884,615 Nmm = 0,002819 MNm$$

$D_{12} = D1 = 0 MNm$ in SCIA Engineer a very small value is entered for this.

For D44 and D55 no formulae are shown, so large values of this can be entered to neglect the effect of the shear force deformation.

The grid is entered in SCIA Engineer in two ways: as orthotropic plate and as real grid. The whole is calculated with the average element size of **1m** without averaging strips.

Maximal deflection Uz in a section in the middle of the orthotropic plate: **-77,017mm**.

Deflection uz in the middle of beam **B8: -77,042mm.**

Moment mx in a section in the middle of the orthotropic plate, claimed **in nodes, averaging: 14,93 kNm/m**.

Moment My in the middle of beam **B8: 15,05 kNm.**

Also here the results show that the behavior of grid is very good approachable through a 2D element with orthotropic properties.

Orthotropic parameters: predefined cases in SCIA Engineer

In SCIA Engineer there are different standard Types of orthotropy implemented.

Standard

This is the standard case of orthotropy where the user has access to all available orthotropy parameters.

The user must input all parameters himself: D11, D22, D12, D33, D44, D55, d11, d22, d12 and d33.

Name	OT1
Type of orthotropy	Standard
Thickness of Plate/Wall [mm]	200
Material	C12/15
D11 [MNm]	1.8819e+01
D22 [MNm]	1.8819e+01
D12 [MNm]	3.7639e+00
D33 [MNm]	7.5278e+00
D44 [MN/m]	1.8819e+03
D55 [MN/m]	1.8819e+03
d11 [MN/m]	5.6458e+03
d22 [MN/m]	5.6458e+03
d12 [MN/m]	1.1292e+03
d33 [MN/m]	2.2583e+03

When you change the "Thickness of the Plate/Wall" or the "Material Type", then values are inserted in the orthotropy parameters D11, D22, … These are the values for a standard isotropic plate with this thickness and this material.

$$D_{11} = \frac{E_1 \cdot h^3}{12 \cdot (1 - v_{12} \cdot v_{21})}$$

$$D_{22} = \frac{E_2 \cdot h^3}{12 \cdot (1 - v_{12} \cdot v_{21})}$$

$$d_{11} = \frac{E_1 \cdot h}{(1 - v_{12} \cdot v_{21})}$$

$$D_{12} = D_{21} = v_{21} \cdot D_{11} = v_{12} \cdot D_{22}$$

$$d_{22} = \frac{E_2 \cdot h}{(1 - v_{12} \cdot v_{21})}$$

$$D_{33} = \frac{G_{12} \cdot h^3}{12}$$

$$d_{33} = G_{12} \cdot h$$

$$D_{44} = G_{12} \cdot h / 1,2$$

$$d_{12} = d_{21} = v_{21} \cdot d_{11} = v_{12} \cdot d_{22}$$

$$D_{55} = G_{23} \cdot h / 1,2 \quad (1)$$

Two heights

Simulation of a slab with a different thickness in local x and local y direction.
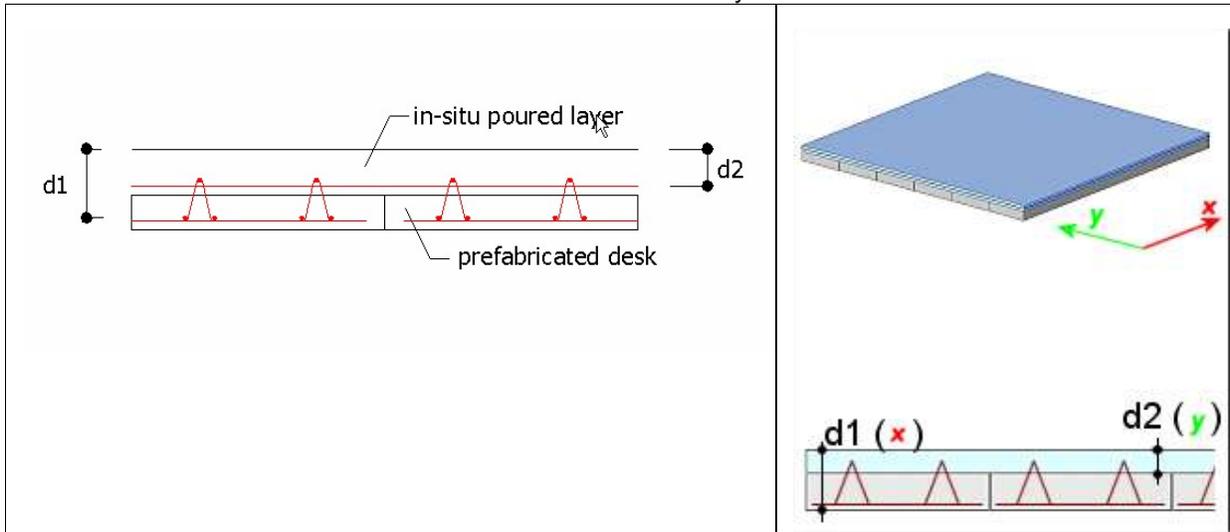
The user must input the effective heights and reduction coefficients:

Name	OT1
Type of orthotropy	Two_heights
Material	C12/15
Flexure	
Effective height (d1) [mm]	200
Effective height (d2) [mm]	200
Torsion reduction coeff	1
Shear reduction coeff	1.2
D11 [MNm]	1.8819e+01
D22 [MNm]	1.8819e+01
D12 [MNm]	3.7639e+00
D33 [MNm]	7.5278e+00
D44 [MN/m]	1.8819e+03
D55 [MN/m]	1.8819e+03
Membrane	
Effective height (h1) [mm]	100
Effective height (h2) [mm]	100
Shear reduction coeff	1
Material	C12/15
d11 [MN/m]	2.8229e+03
d22 [MN/m]	2.8229e+03
d12 [MN/m]	5.6458e+02
d33 [MN/m]	1.1292e+03

All orthotropic parameters are calculated:

$$D_{11} = \frac{E.d_1^3}{12.(1-v^2)}$$	$$d_{11} = \frac{E.h_1}{(1-v^2)}$$
$$D_{22} = \frac{E.d_2^3}{12.(1-v^2)}$$	$$d_{22} = \frac{E.h_2}{(1-v^2)}$$

78

$D_{12} = v.\sqrt{D_{11}.D_{22}}$	$d_{12} = v.\sqrt{d_{11}.d_{22}}$
$D_{33} = \gamma_{f1}.\dfrac{(1-v).\sqrt{D_{11}.D_{22}}}{2}$	$d_{33} = \gamma_{f2}.\dfrac{(1-v).\sqrt{d_{11}.d_{22}}}{2}$
$D_{44} = \dfrac{G.d_1}{\beta}$	
$D_{55} = \dfrac{G.d_2}{\beta}$	With:
	• $\gamma_{f2} = shear\ reduction\ coeff.$
With:	
• $\gamma_{f1} = torsion\ reduction\ coeff.$	
• $\beta = shear\ reduction\ coeff.$	

Value **β** = 1.2 (for rectangular sections) ; also value 1.0 can be used.

Value **γf1** : when setting this value to zero, then you get a 'torsion weak' plate, thus avoiding the lifting of the corner nodes. To get a numerically stable solution, use small values (0.05) instead of zero.

Value **γf2** : when setting this value to zero, then you get a 'shear weak' wall, thus avoiding horizontal (shear) stiffening of the construction. To get a numerically stable solution, use small values (0.05) instead of zero.

One direction slab

Simulation of a slab which carries it's load mainly in one direction:
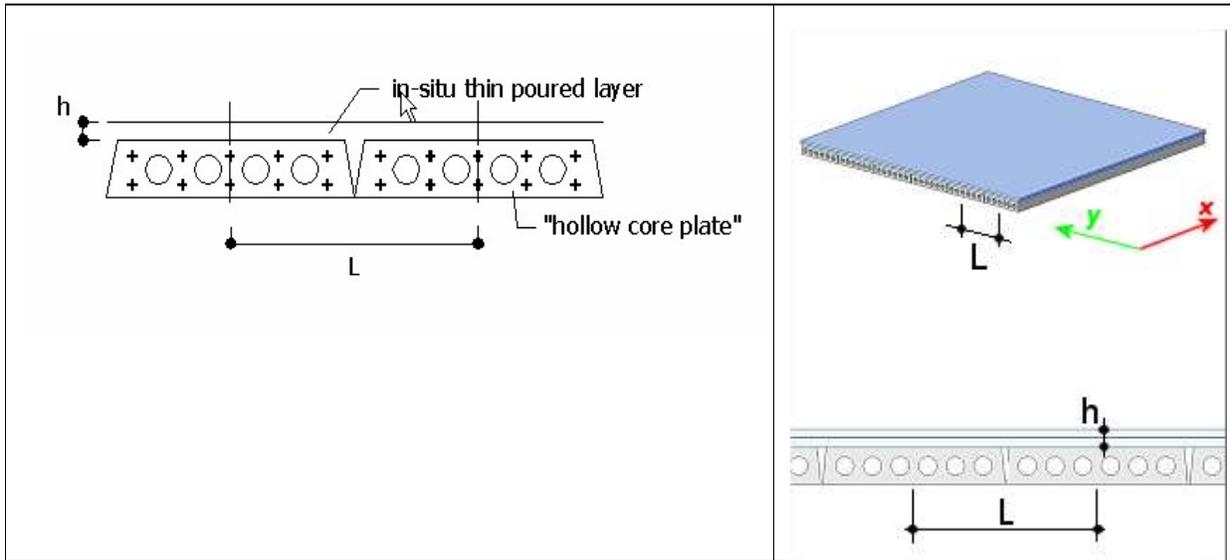
The rigidity in the main direction is calculated based upon the properties of a user defined cross-section. The user should define the cross-section (css) of these unidirectional prefab elements and then use this css to define the orthotropy.

Besides the css, the user must input the height of the topping h and the distance between the elements L :

Name	OT1
Type of orthotropy	One direction slab
Flexure	
CSS	CS1 - Rectangle (500; 300)
L [mm]	1000
Material	C12/15
h [mm]	200
D11 [MNm]	8.4688e+01
D22 [MNm]	1.8067e+01
D12 [MNm]	0.0000e+00
D33 [MNm]	7.7038e+00
D44 [MN/m]	1.4115e+03
D55 [MN/m]	1.8819e+03
Membrane	
Effective height (h1) [mm]	100
Effective height (h2) [mm]	100
Material	C12/15
d11 [MN/m]	2.8229e+03
d22 [MN/m]	2.8229e+03
d12 [MN/m]	5.6458e+02
d33 [MN/m]	1.1292e+03

All orthotropic parameters are calculated:

$$D_{11} = \frac{E_1 . I_1}{L}$$

$$d_{11} = \frac{E_2 . h_1}{(1 - v^2)}$$

$$D_{22} = \frac{E_2 . h^3}{12}$$

$$d_{22} = \frac{E_2 . h_2}{(1 - v^2)}$$

$$D_{12} = 0$$

$$d_{12} = v . \sqrt{d_{11} . d_{22}}$$

$$D_{33} = \frac{\left(\frac{G_1 . It_1}{L}\right) + \left(\frac{G_2 . h^3}{3}\right)}{8}$$

$$d_{33} = \frac{(1 - v) . \sqrt{d_{11} . d_{22}}}{2}$$

$$D_{44} = \frac{G_1 . Az_1}{L}$$

$$D_{55} = \frac{G_2 . h}{1.2}$$

Slab with ribs – rib inputted by the user
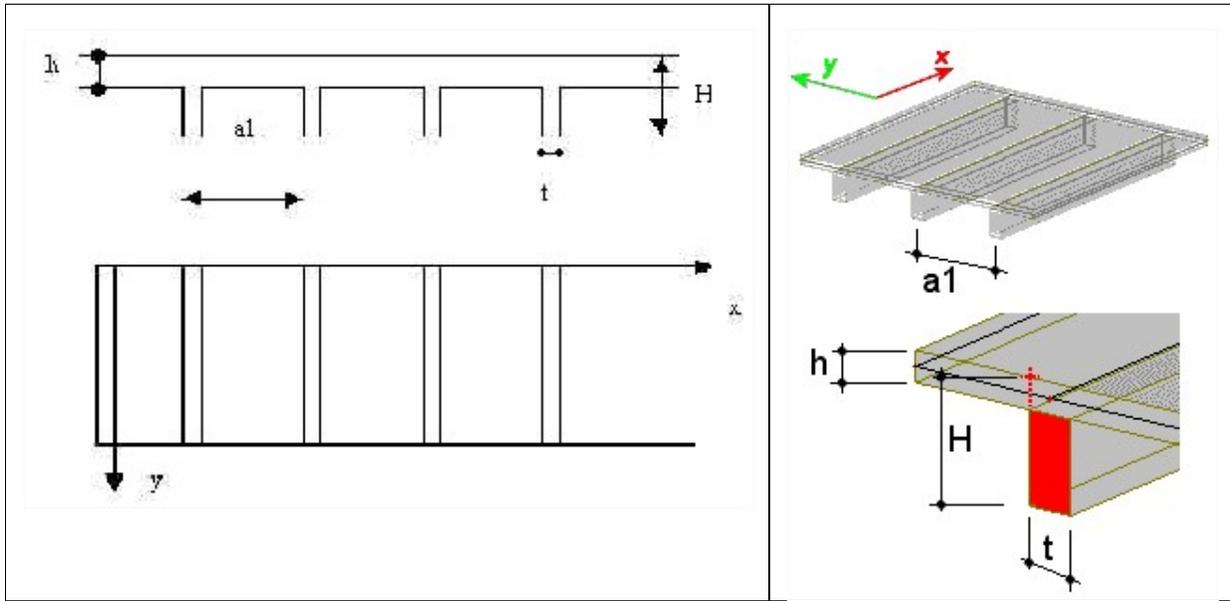
Simulation of a slab with ribs in one direction:

The user must input the rib dimensions, rib spacing and slab height:

Name	OT1
Type of orthotropy	Slab with ribs
⊟ **Flexure**	
⊟ **Rib**	
Rib	Input
Material 1	C12/15
Rib thickness,t [mm]	300
Rib depth,H-h [mm]	500
Spacing,a1 [m]	0.500
⊟ **Slab**	
Material	C12/15
Slab height, h [mm]	150
D11 [MNm]	4.6324e+02
D22 [MNm]	1.8710e+01
D12 [MNm]	0.0000e+00
D33 [MNm]	5.9579e+01
D44 [MN/m]	3.8781e+03
D55 [MN/m]	1.4115e+03
⊟ **Membrane**	
Effective height (H) [mm]	100
Effective height (h) [mm]	150
Material	C12/15
d11 [MN/m]	2.8229e+03
d22 [MN/m]	4.2344e+03
d12 [MN/m]	0.0000e+00
d33 [MN/m]	1.3829e+03

All orthotropic parameters are calculated:

$$D_{11} = \frac{E.I}{a1}$$

$$D_{22} = \frac{E.a1.h^3}{12.\left[a1 - t + \left(\left(\frac{h}{H}\right)^3.t\right)\right]}$$

$$D_{12} = 0$$

$$D_{33} = \frac{E.h^3}{12.(1-\nu)} + \frac{G.It}{2.a1}$$

$$D_{44} = \frac{G.h}{1.2}$$

$$D_{55} = \frac{G.Az}{a1}$$

$$d_{11} = E_1.d_1$$

$$d_{22} = E_2.d_2$$

$$d_{12} = \sqrt{d_{11}.d_{22}}$$

$$d_{33} = \frac{\sqrt{d_{11}.d_{22}}}{2}$$

Slab with ribs – rib selected from the cross-section library

Simulation of a slab with ribs in one direction:
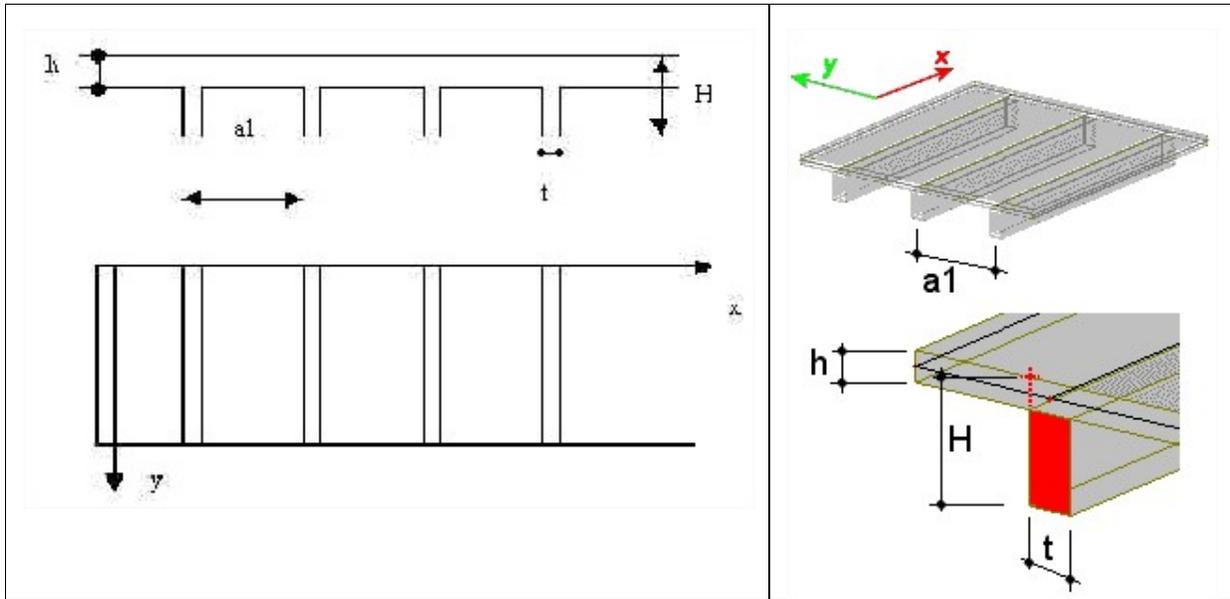
The user must select the rib from the library and input the rib spacing and slab height:

Name	OT1
Type of orthotropy	Slab with ribs
⊟ Flexure	
⊟ Rib	
Rib	CSS Lib
Cross Section	CS1 - Rectangle (500; 300)
Spacing,a1 [m]	0.500
⊟ Slab	
Material	C12/15
Slab height, h [mm]	150
D11 [MNm]	4.6324e+02
D22 [MNm]	7.6219e+00
D12 [MNm]	0.0000e+00
D33 [MNm]	2.9710e+01
D44 [MN/m]	2.8229e+03
D55 [MN/m]	1.4115e+03
⊟ Membrane	
Effective height (H) [mm]	100
Effective height (h) [mm]	150
Material	C12/15
d11 [MN/m]	2.8229e+03
d22 [MN/m]	4.2344e+03
d12 [MN/m]	0.0000e+00
d33 [MN/m]	1.3829e+03

All orthotropic parameters are calculated:

$$D_{11} = \frac{E_1 . I_1}{L}$$

$$d_{11} = \frac{E_2 . d_1}{(1 - v^2)}$$

$$D_{22} = \frac{E_2 . h^3}{12}$$

$$d_{22} = \frac{E_2 . d_2}{(1 - v^2)}$$

$$D_{12} = 0$$

$$d_{12} = v . \sqrt{d_{11} . d_{22}}$$

$$D_{33} = \frac{(G_1 . It_1) + \left(\frac{G_2 . h^3}{3}\right)}{8}$$

$$d_{33} = \frac{(1 - v) . \sqrt{d_{11} . d_{22}}}{2}$$

$$D_{44} = \frac{G_2 . h}{1.2}$$

$$D_{55} = \frac{G_1 . Az_1}{L}$$

With:

- index 1 – Cross-section properties

- index 2 - Slab properties

- following properties are taken from Cross-section and Material:

  o E modulus Ei

  o Moment of inertia I1

  o Torsional moment of inertia It1

  o Effective surface for shear Az1

- - G modulus Gi

Grid work – ribs inputted by the user
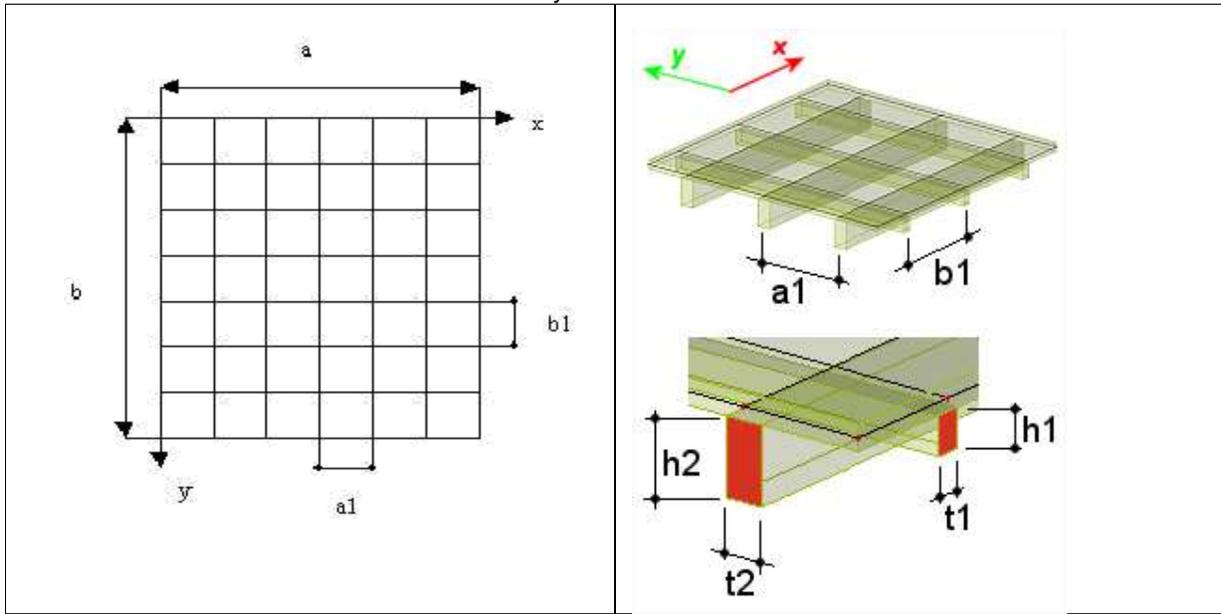
Simulation of a slab with ribs in local x and local y direction:

The user must input the rib dimensions, rib spacing and slab height:

Name	OT1
Type of orthotropy	Grid work
Flexure	
Beam	Input
beam,1	
Material1	C12/15
Width of beam, t1 [mm]	300
Depth of beam, h1 [mm]	450
spacing, a1 [m]	0.500
beam,2	
Material2	C12/15
Width of beam, t2 [mm]	300
Depth of beam, h2 [mm]	450
Spacing, b1 [m]	0.500
D11 [MNm]	1.2347e+02
D22 [MNm]	1.2347e+02
D12 [MNm]	0.0000e+00
D33 [MNm]	2.6840e+01
D44 [MN/m]	2.5406e+03
D55 [MN/m]	2.5406e+03
Membrane	
Effective height (h1) [mm]	100
Effective height (h2) [mm]	100
Material	C12/15
d11 [MN/m]	2.7100e+03
d22 [MN/m]	2.7100e+03
d12 [MN/m]	0.0000e+00
d33 [MN/m]	1.3550e+03

All orthotropic parameters are calculated:

$D_{11} = \dfrac{E_1.I_1}{b1}$	$d_{11} = E_1.h_1$
$D_{22} = \dfrac{E_2.I_2}{a1}$	$d_{22} = E_2.h_2$
$D_{12} = 0$	$d_{12} = \sqrt{d_{11}.d_{22}}$
$D_{33} = \dfrac{\left(\dfrac{G_1.It_1}{b1}\right) + \left(\dfrac{G_2.It_2}{a1}\right)}{4}$	$d_{33} = \dfrac{\sqrt{d_{11}.d_{22}}}{2}$
$D_{44} = \dfrac{G.Az1}{b1}$	
$D_{55} = \dfrac{G.Az2}{a1}$	

Grid work – ribs selected from the cross-section library

Simulation of a slab with ribs in local x and local y direction:
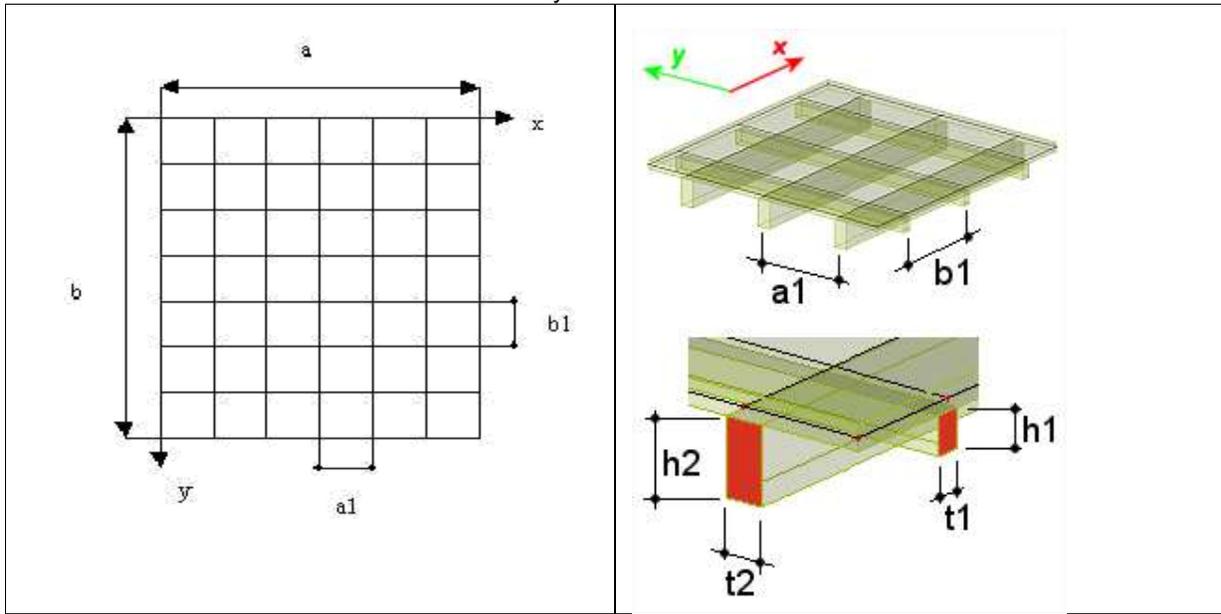
The user must select the ribs from the library and input the rib spacings and slab height:

Name	OT1
Type of orthotropy	Grid work
Flexure	
Beam	CSS Lib
beam,1	
Cross Section	CS1 - Rectangle (500; 300)
spacing, a1 [m]	0.500
beam,2	
Cross Section	CS1 - Rectangle (500; 300)
Spacing, b1 [m]	0.500
D11 [MNm]	1.6938e+02
D22 [MNm]	1.6938e+02
D12 [MNm]	0.0000e+00
D33 [MNm]	3.1519e+01
D44 [MN/m]	2.8229e+03
D55 [MN/m]	2.8229e+03
Membrane	
Effective height (h1) [mm]	100
Effective height (h2) [mm]	100
Material	C12/15
d11 [MN/m]	2.7100e+03
d22 [MN/m]	2.7100e+03
d12 [MN/m]	0.0000e+00
d33 [MN/m]	1.3550e+03

All orthotropic parameters are calculated:

$$D_{11} = \frac{E_1 . I_1}{b1}$$

$$D_{22} = \frac{E_2 . I_2}{a1}$$

$$D_{12} = 0$$

$$D_{33} = \frac{\left(\frac{G_1 . It_1}{b1}\right) + \left(\frac{G_2 . It_2}{a1}\right)}{4}$$

$$D_{44} = \frac{G . Az1}{b1}$$

$$D_{55} = \frac{G . Az2}{a1}$$

$$d_{11} = E_1 . h_1$$

$$d_{22} = E_2 . h_2$$

$$d_{12} = \sqrt{d_{11} . d_{22}}$$

$$d_{33} = \frac{\sqrt{d_{11} . d_{22}}}{2}$$

With:

- following properties are taken from Cross-section and Material:

    o E modulus Ei

    o Moment of inertia Ii

    o Torsion moment of inertia Iti

    o Effective surface for shear Azi

    o G modulus Gi

Property modifiers

This feature is available in the concept edition of SCIA Engineer, but the functionality for property modifiers should be activated.
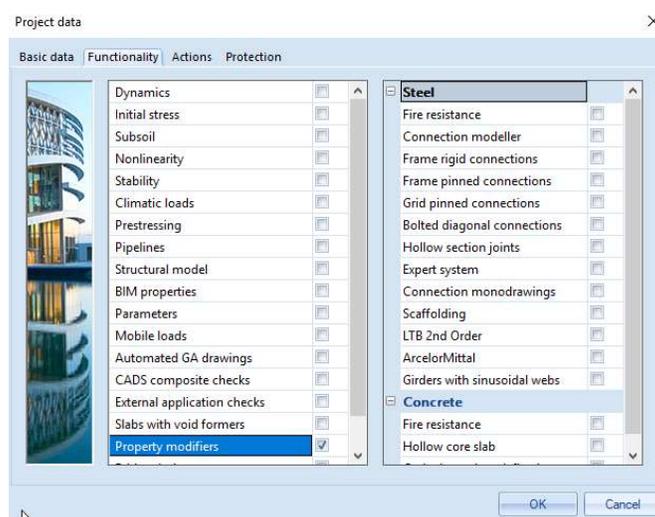
Property modifiers are used to change the stiffness's of both 1D or 2D elements.
They exist as additional data that can be inputted on 2D or 1D members.

EXAMPLE: 17_01_Property modifiers
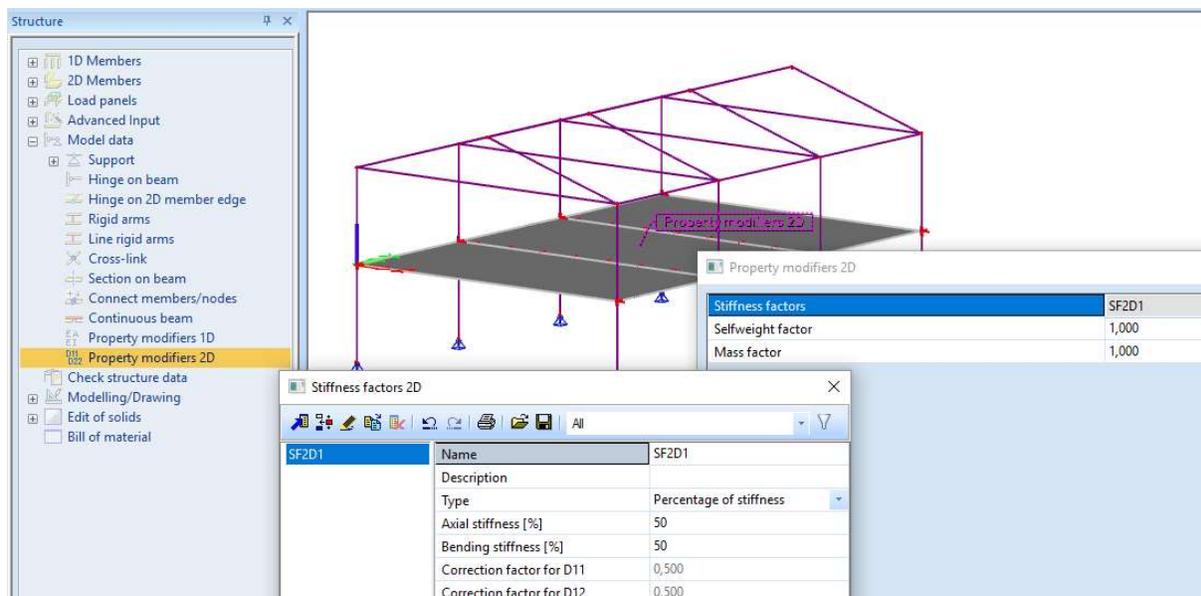
The advantage of property modifiers is that they are easy to apply and it is easy to understand how it works.
A lot of practical applications can be considered.

For example, the calculation of a concrete member that is assumed to be cracked.
By using property modifiers here, the user can keep on performing a linear calculation.
(By this a more complex PNL or CDD calculation (-> see advanced concrete workshop) is avoided.)

Pressure only

The topic 'pressure only' is not available in the **Concept Edition** of SCIA Engineer.
The license code is **esas.44** and it is only part of the **Professional or Expert Edition**.

When using pressure 2D elements, the functionality **Nonlinearity** and **Press only 2D members** must be activated.
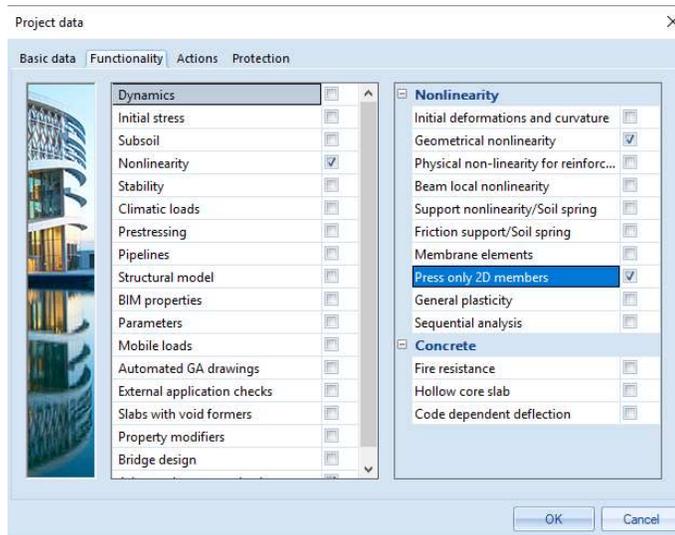
With this option, tension in 2D elements can be automatically eliminated. This is mostly used for masonry elements. In below some examples to clarify this option.

EXAMPLE: 18_01_Pressure only1

This project illustrates the use of pressure only elements.
In this project two 2D-elements are modelled. The first one is modelled as an isotropic element, the second one is a pressure only element:

When calculating those elements, for every mesh element a certain orthotropy will be inserted. At the first iteration step, all the pressure only elements will be calculated as isotropic elements. After the first calculation on all the elements in tension another stiffness will be inserted, so a certain orthotropy will be inserted. With this stiffness, the tension capacity of this element will decrease. After adapting the orthotropic parameters, a new calculation will be performed. After this second iteration step, again the elements in tension will get another stiffness. This process will be repeated until an equilibrium is reached.

The difference between the isotropic and orthotropic elements can be clearly view looking at the principal normal force n1 for these members:
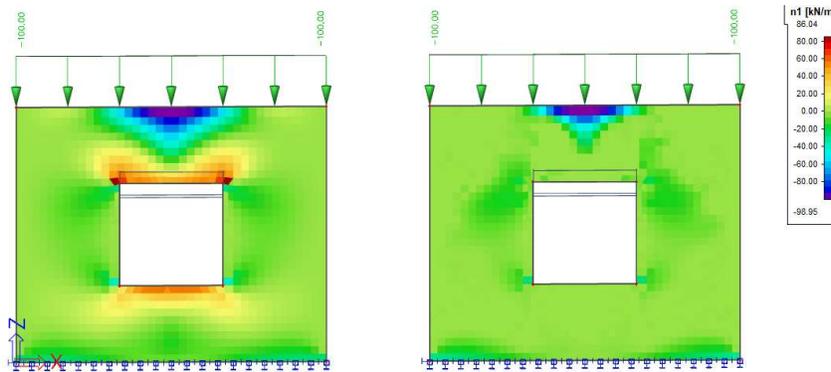

The results n1 shows the 'biggest' principal normal force (since n1 > n2). So, it is clear now that the right element only gets compression forces.

Looking at the trajectories of this normal force, the trajectory of the pressure force will be even more visible:
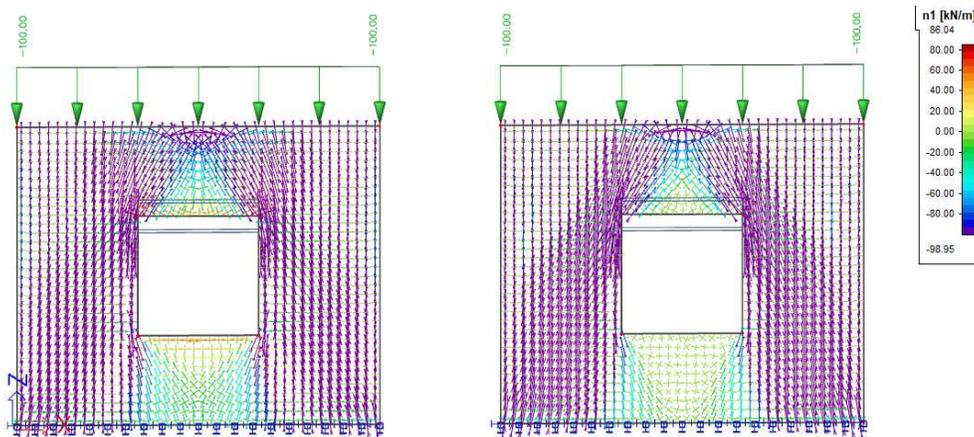
Also for the beam above the opening, there is a difference for the moment My:
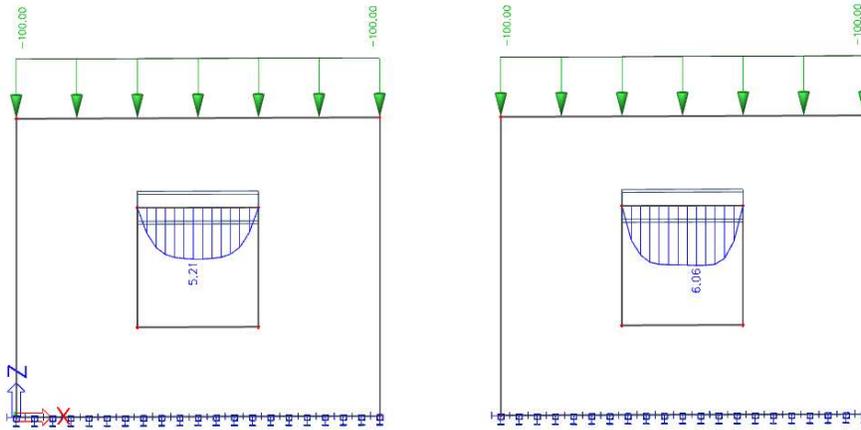
When looking at the pressure diagonals in a reinforced 2D concrete element, ribs can be used to simulate reinforcement.

In this example, a plate with a bearing support is inserted with two ribs acting as the reinforcement of the element:
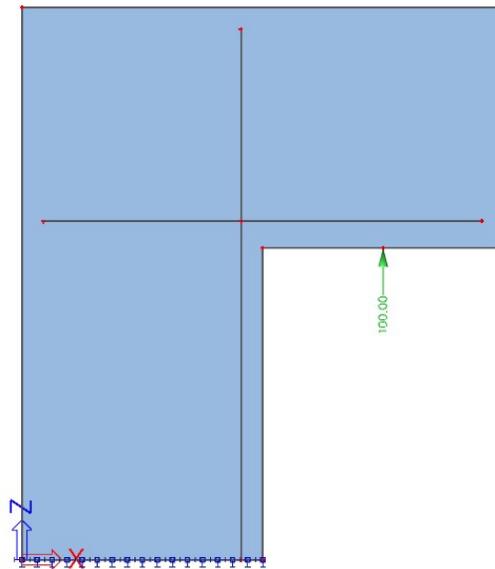
Looking at the results of this 2D element, the pressure diagonals inside this element are clearly visible:
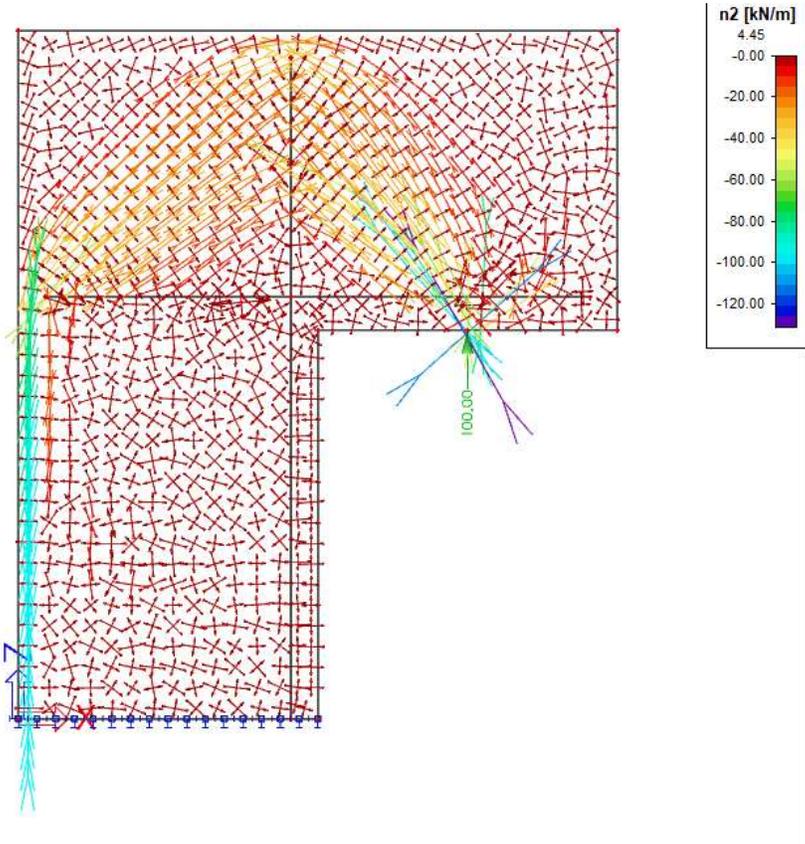
References

[1]	S.P. Timoshenko and S. Wotnowsky-Krieger, Theory of Plates and Shells, 2nd edition, Mc Graw Hill, New York, 1987.
[2]	Kolar, vl. et al.: FEM Principles and practice of the Finite Element Method (Czech language), Computer press, 1997.
[3]	Hobst, Ed.: Methode der finiten Elemente im Stahlbetonbau, Beton- und Stahlbetonbau 95, 2000, Heft 10.
[4]	Elementenmethode voor loodrecht op zijn vlak belaste platen, Vak toegepaste mechanica, Tweede jaar cursus constructeur BV/SG, september 1993.
[5]	2D Finite Element Membrane Element with rotational degrees of freedom, FEM-Consulting, 2004.
[6]	Nieuw in SCIA-ESA PT 5.0, SCIA-Group NV, 2004
[7]	Lynn-Dhillon, 2nd Conf. Matrix Meth. JAPAN - USA, Tokyo, 1971.
[8]	De Roeck, G.: De eindige-elementenmethode, Leuven, 1991.
[9]	Blaauwendraad, J.: Zin en onzin van Mindlin-Reisner, Delft, 2008