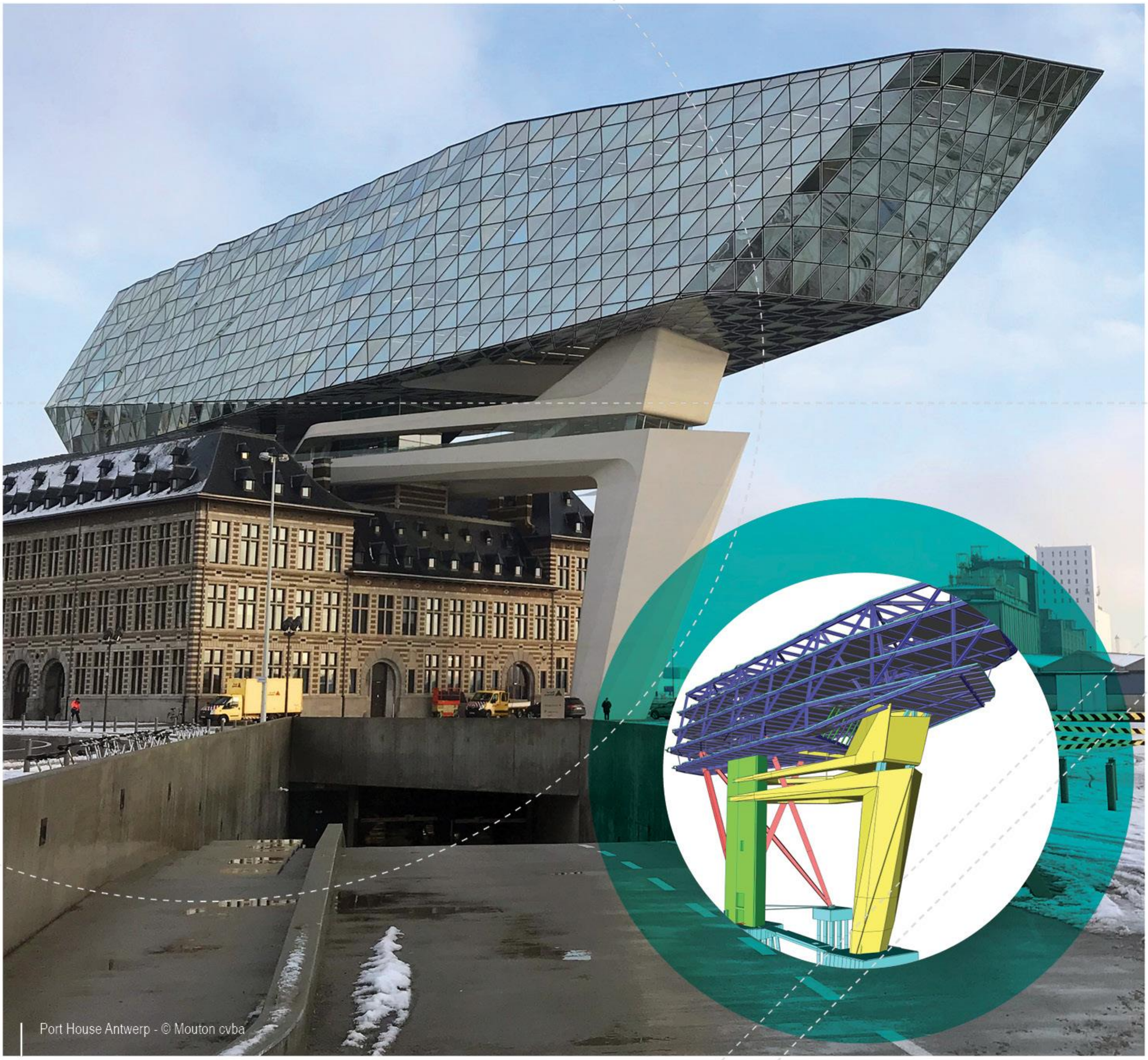


ADVANCED CONCEPT TRAINING

REINFORCED CONCRETE (EN1992) – 2D MEMBERS

Table of Contents

Introduction

The examples in this manual can be made in a full licensed as well as in a tryout or student version of SCIA Engineer.

Here follows an overview of the required SCIA Engineer modules / editions, per subject:

- **Theoretical reinforcement design**
  esacd.02 (2D members) **Concept edition**

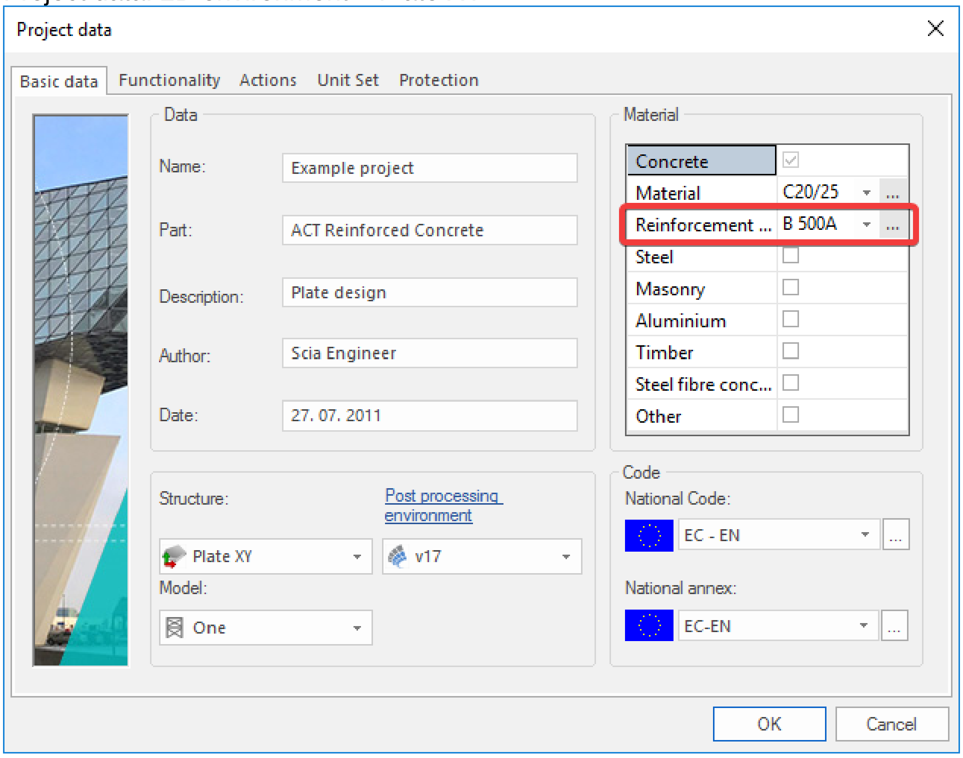
- **Practical reinforcement design**
  esacdt.03 (2D members) **Concept edition**

Plate design

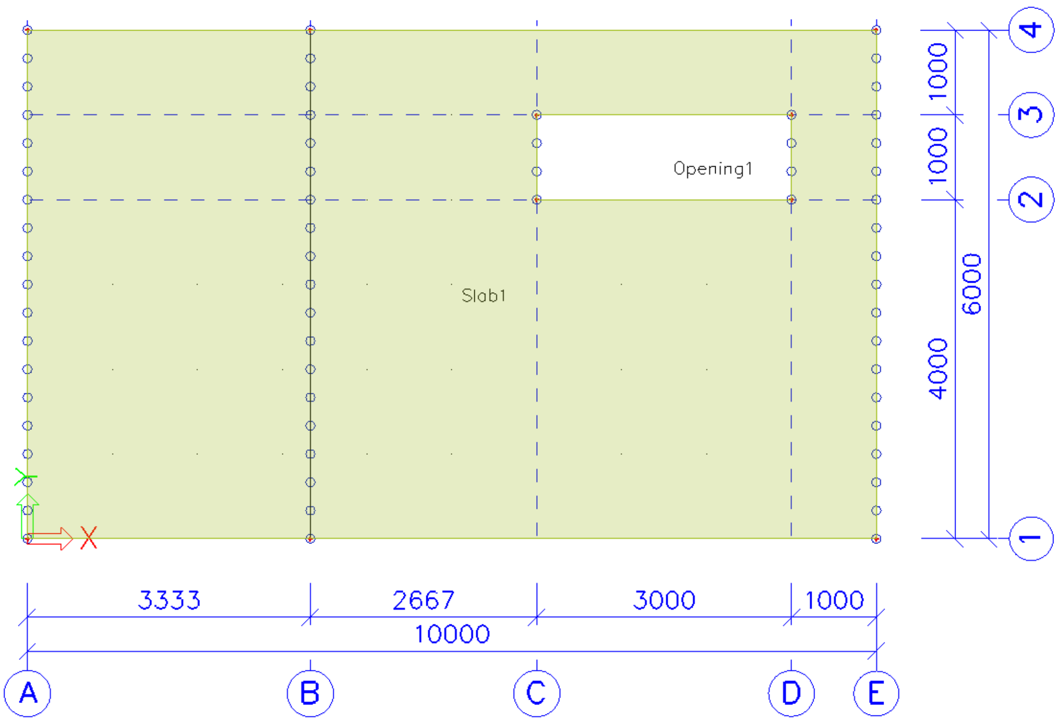
Model

1_Input of geometry

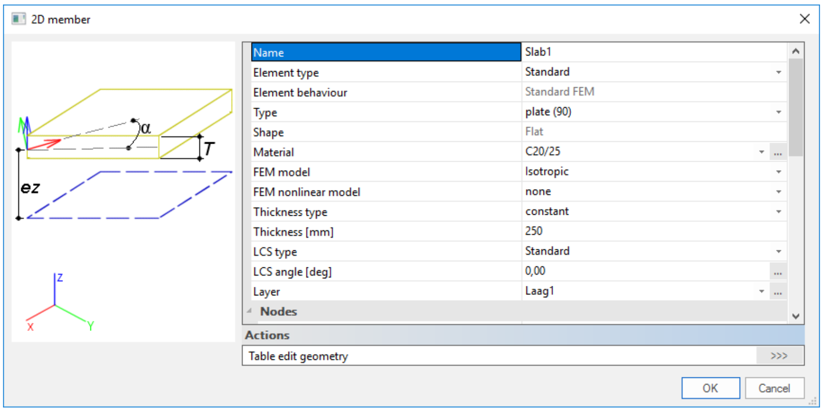
Project data: 2D environment = Plate XY

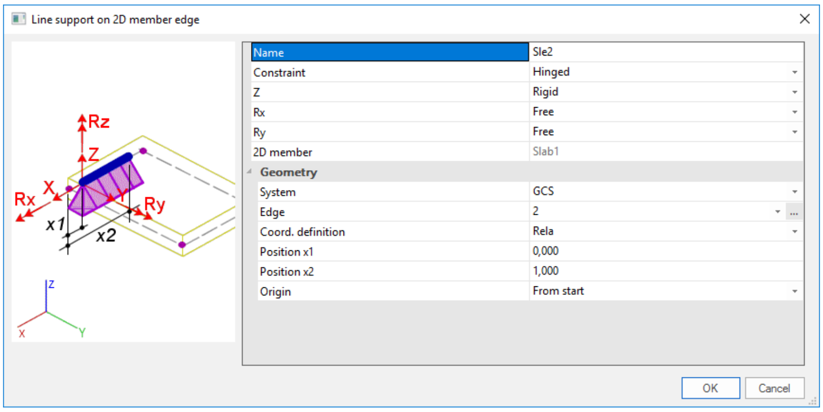


The Reinforcement material (e.g. B500A) chosen in the Project data window, will define the steel quality used for the theoretical reinforcement design.



Properties of the slab and the line supports:

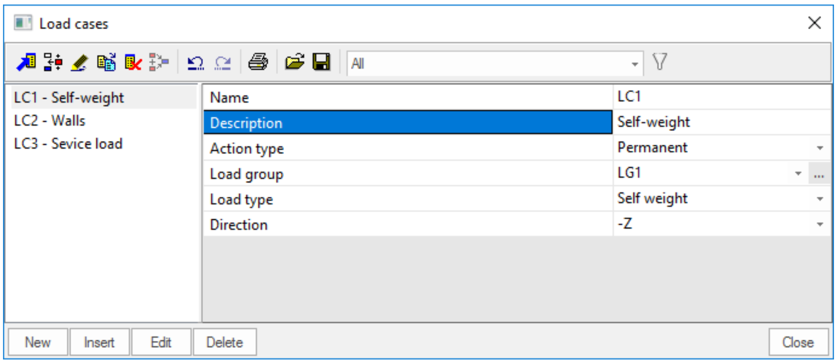


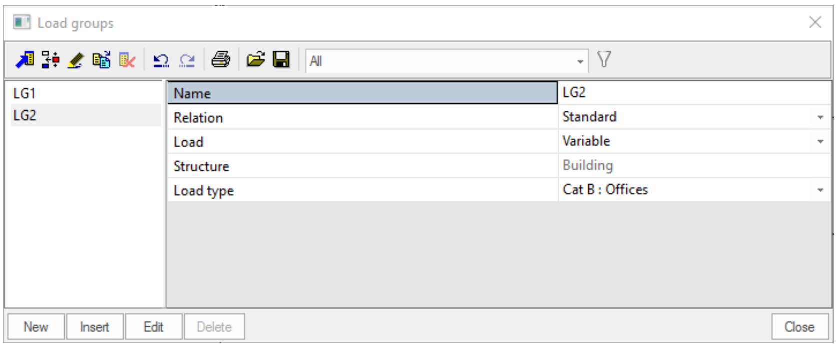


2_Loads

Load cases & Load groups

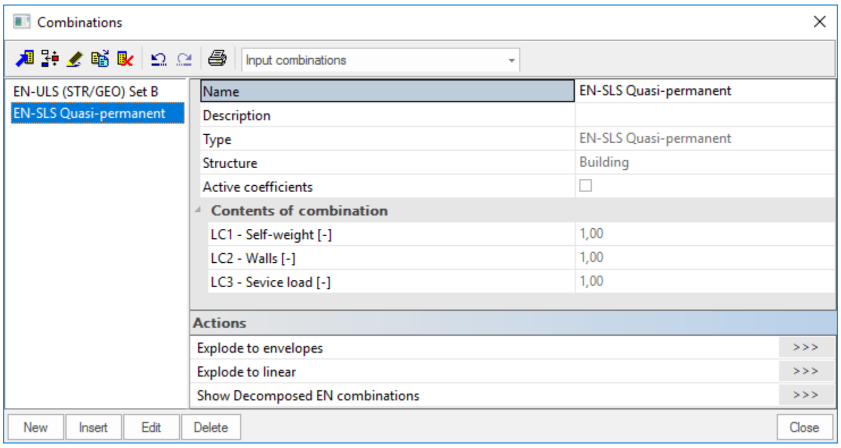
Load Case	***Action type***	***Load Group***	***Relation***	***EC1-Load type***
Self-weight	Permanent	LG1	/	/
Walls	Permanent	LG1	/	/
Service load	Variable	LG2	Standard	Cat B: Offices





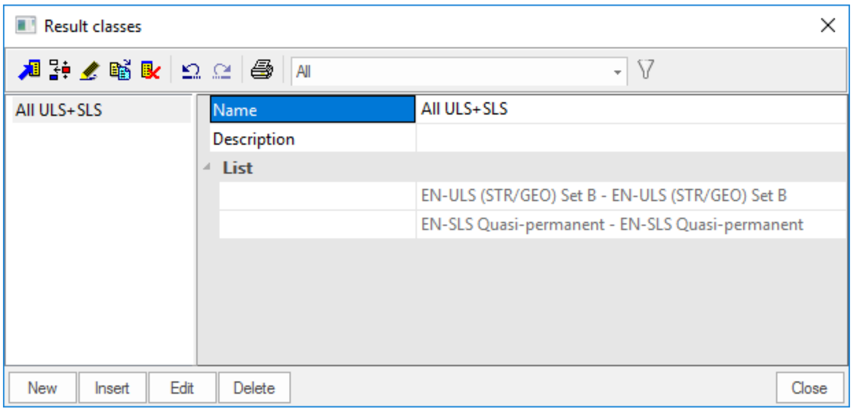
Load combinations

Type EN-ULS (STR/GEO) Set B
Type EN-SLS Quasi Permanent



Result classes

All ULS+SLS

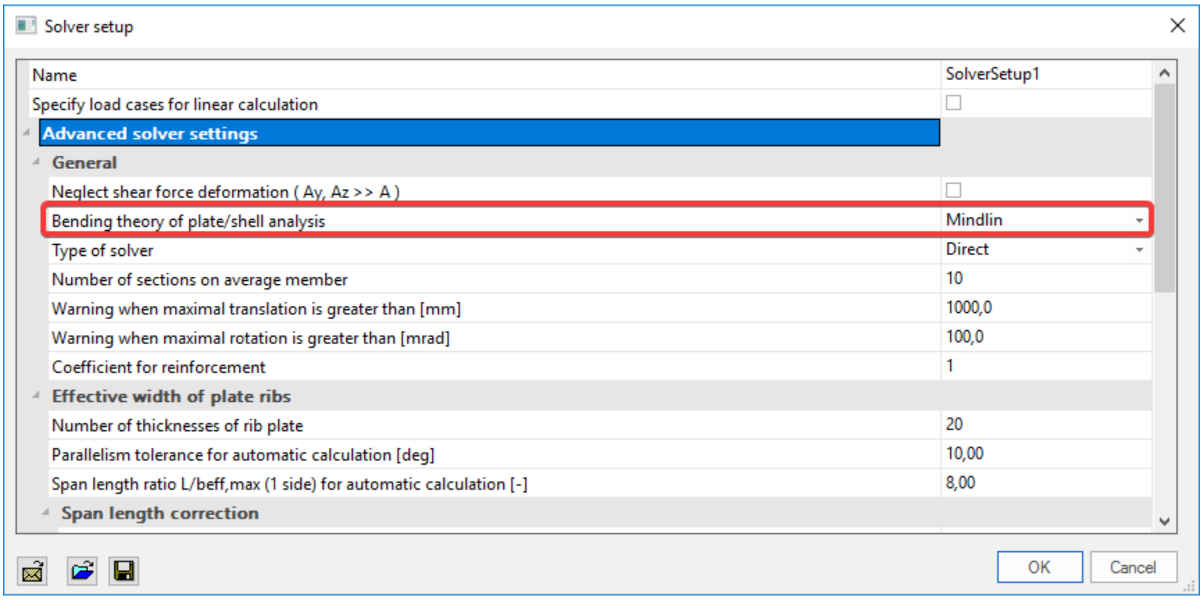


3_Finite element mesh

Introduction

2 types of finite elements are implemented in SCIA Engineer:
- The **Mindlin element** including shear force deformation, which is the standard in SCIA Engineer. The Mindlin theory is valid for the calculation of both thin and thick plates.
- The **Kirchhoff element** without shear force deformation, which can be used to calculate and design only thin plates.

The element type used for the current calculation is defined in the Setup menu > Solver:



Mesh generation

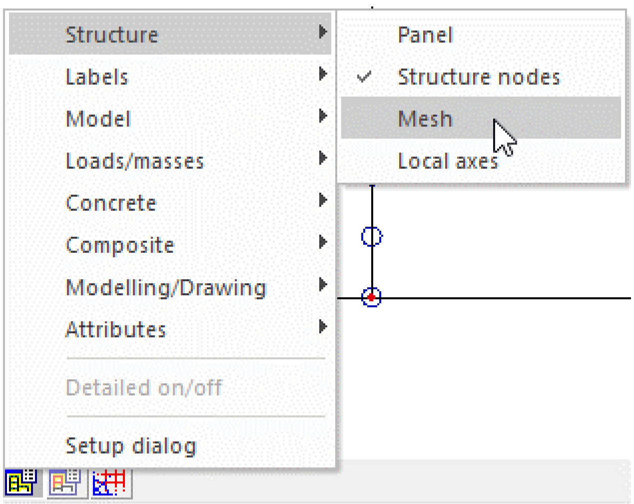
Via the Main menu → Calculation, mesh → Mesh generation, or ‘Project’ toolbar

Graphical display of the mesh

Set view parameters for all, via right mouse click in screen or Command line toolbar
- Structure tab → Mesh → Draw mesh
- Labels tab → Mesh → Display label

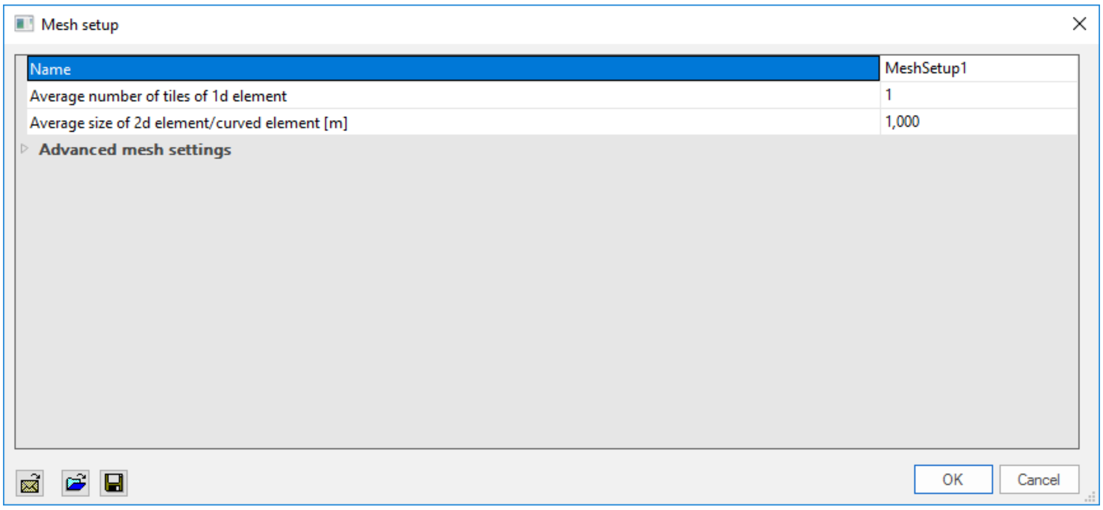
OR

Click on the ‘Fast adjustment of view parameters on whole model’ above the command line.



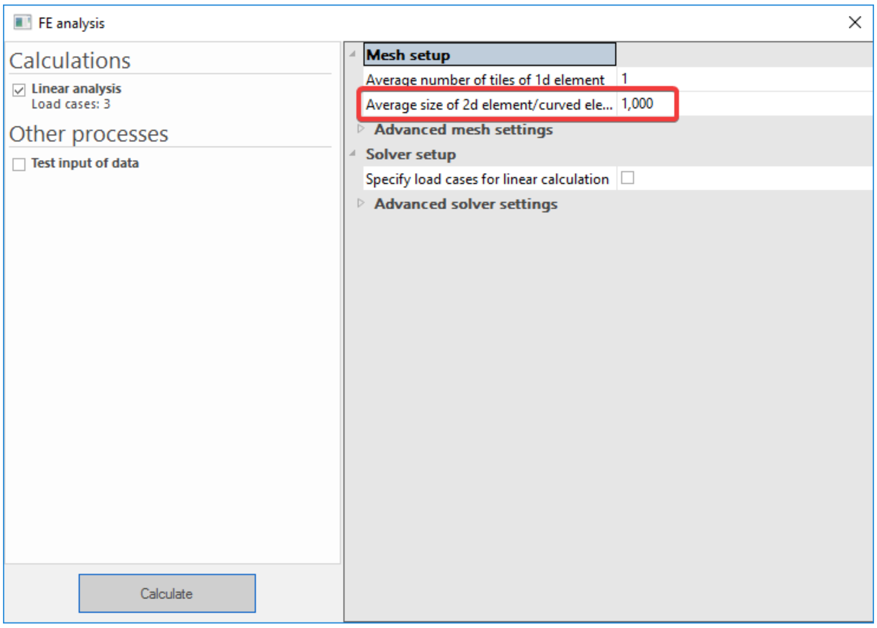
Mesh refinement

Via the Main menu → Calculation, mesh → Mesh setup, or Setup menu → Mesh
Average size of 2D (mesh) elements is by default = 1m.

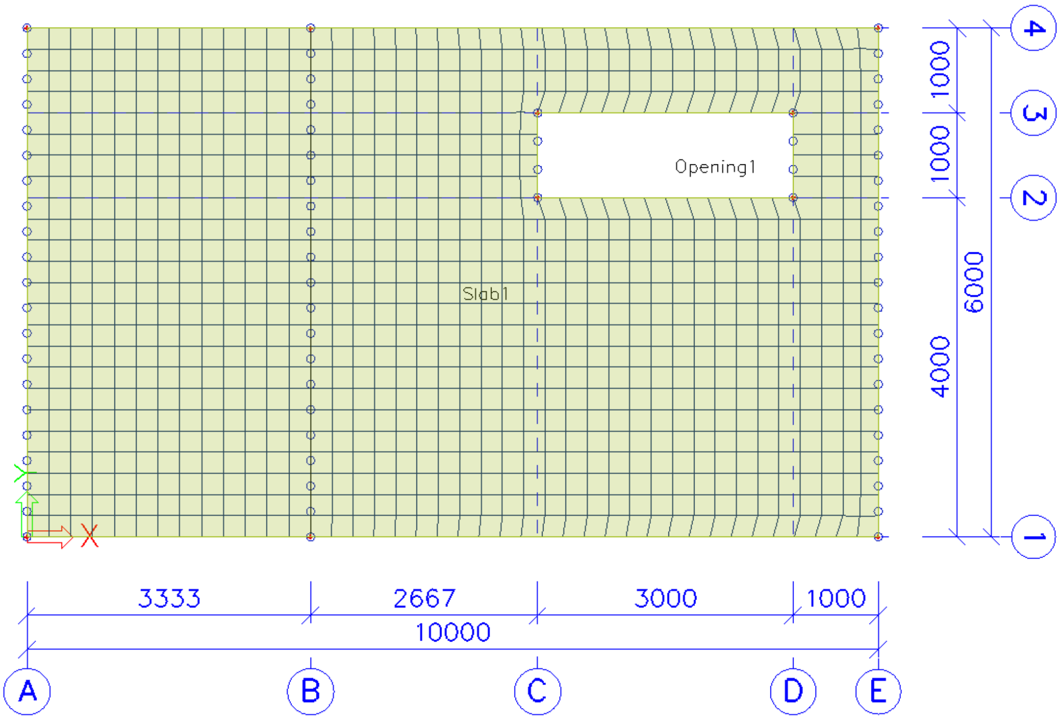


OR

The mesh size can be changed in the FE analysis window before running the calculation.



'Basic rule' for the size of 2D mesh elements: take 1 to 2 times the thickness of the plates in the project. For this example, take a mesh size of 0,25 m.

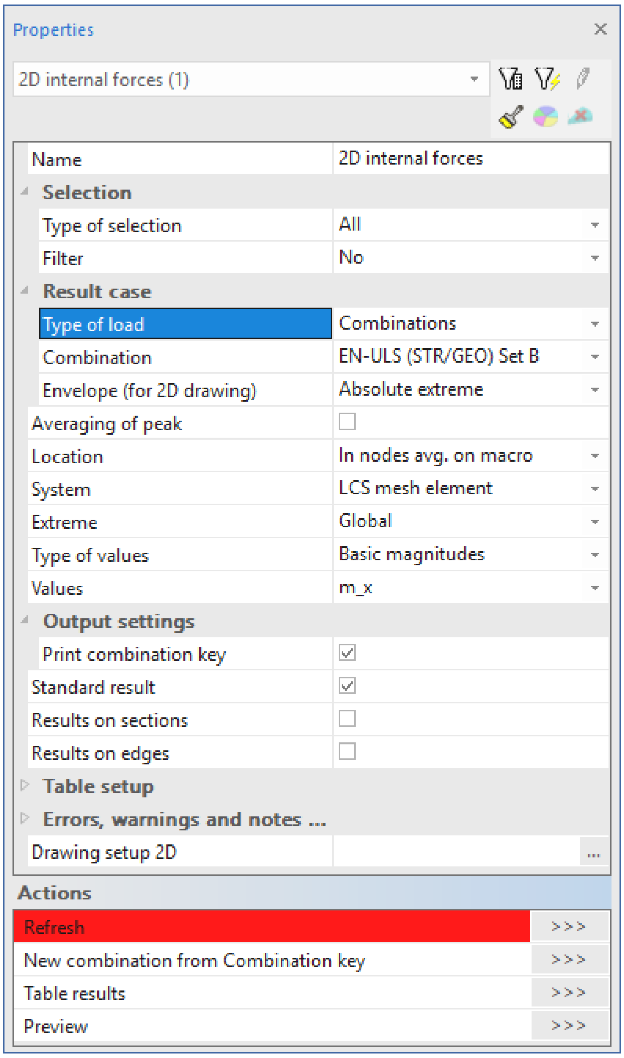


Opening1
Slab1
3333
2667
3000
1000
10000
A
B
C
D
E
1000
1000
4000
6000
4
3
2
1
X
Y

4_Results for the linear calculation

Specification of results

After running the linear calculation, go to the Main menu → Results → 2D Members → 2D Internal Forces.
Specify the desired result in the Properties menu:



System:

- LCS mesh element: according to the local axes of the *individual* mesh elements
- LCS - Member 2D: according to the LCS of the 2D member (Pay attention when working with shell elements!)

Location: 4 different ways to ask for the results, see Annex 2
Type forces: Basic, Principal or Design magnitudes, see Annex 1

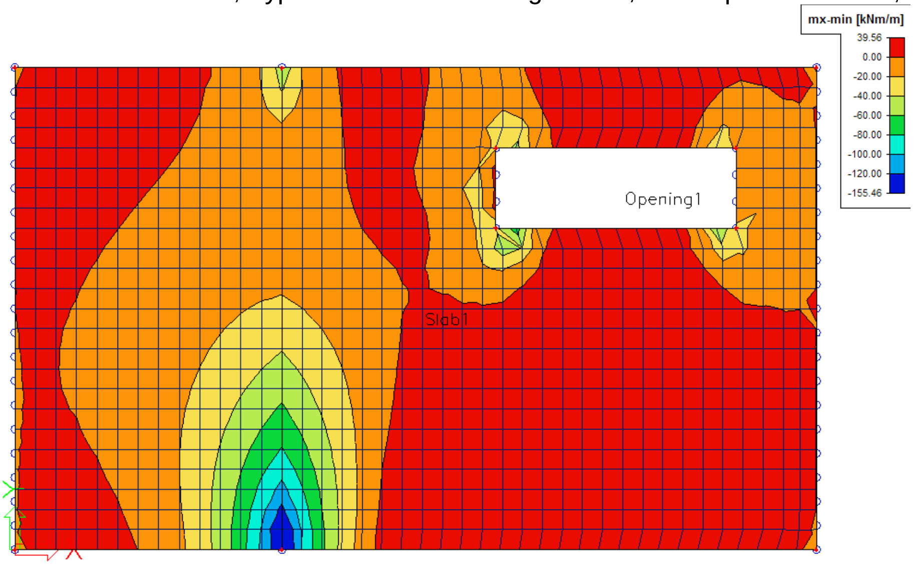
Drawing setup 2D: Click on the button ... . Here you can modify the display of 2D results (Isobands / Isolines / Numerical results / ...), modify the minimum and maximum settings, ...

After making changes in the Properties menu, you always have to execute the 'Refresh' action.

Types of results

Basic magnitudes

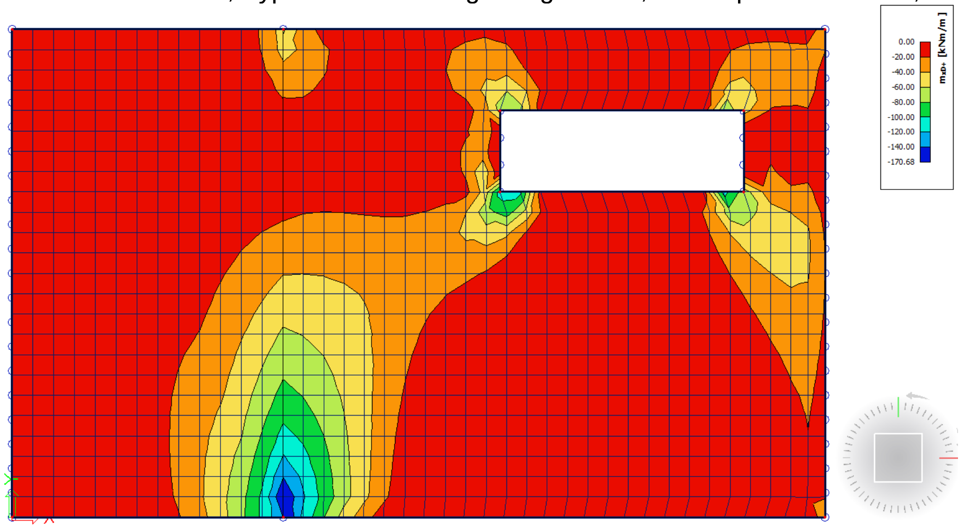
Combination = ULS; Type forces = Basic magnitudes; Envelope = Minimum; Values = **m_x**



These are the characteristic values coming from de FE-analysis in the center of the plate.

Elementary design magnitudes

Combination = ULS; Type forces = Design magnitudes; Envelope = Maximum; Values = **m_xD+**



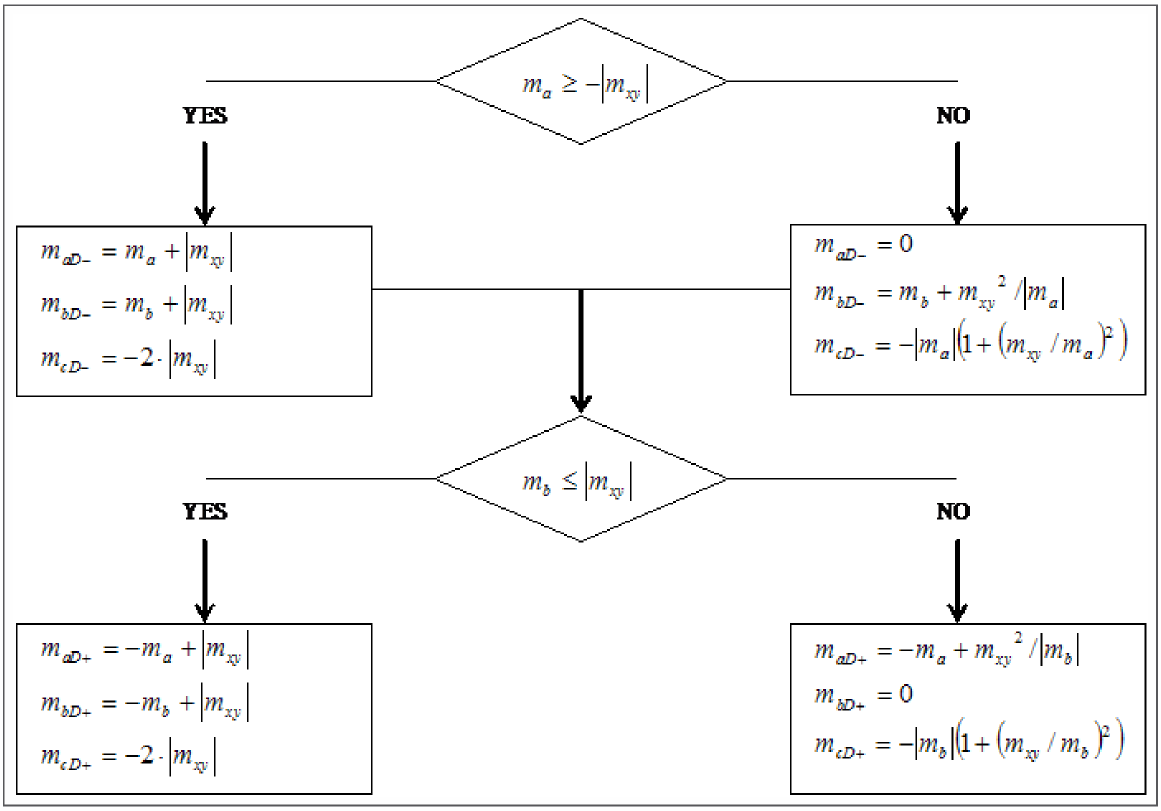
The convention for the sign of the design moments has been changed since the v17 post-processor. Now a moment is positive when it causes a tensile force on the bottom of the plate and negative when it causes tensile force at the top of the plate.

In the v16 post-processor a design moment is positive when you should reinforce for this moment. This means that for a positive value for m_xD+ there is a tensile force at the top of the plate and that for a positive value for m_xD- there is a tensile force at the bottom of the plate.

The available values are mxD, myD and mcD, where ‘D’ stands for design. The ‘+’ and ‘-‘ respectively stand for the values at the positive and negative side of the local z axis of the 2D member.
So for instance the value mxD+ is the moment that will be used for the design of the upper reinforcement in the local x-direction of the 2D member.

The calculation of design moments for *plates* and *shells* according to the EC2 algorithm follows the chart from CSN P ENV 1992-1-1, Annex 2, paragraph A2.8.

$$m_y \geq m_x \quad : \quad a = x, b = y$$
$$m_y < m_x \quad : \quad a = y, b = x$$



What happens, is that for the 3 characteristic (bending and torsion) moments an equivalent set of 3 design moments is calculated:

mx		***mxD***
my	≈	myD
mxy		mcD

It is clear that mxD and myD are the moments to be used for the reinforcement design in the respective direction. The quantity mcD is the design moment that has to be taken by the concrete. The Eurocode does not mention any check for this value, but it is however available in SCIA Engineer for the reason of completeness.

The calculation of design forces for *walls* according to the EC2 algorithm follows the chart from CSN P ENV 1992-1-1, Annex 2, paragraph A2.9.

$n_y \geq n_x \quad : \quad a = x, b = y$

$n_y < n_x \quad : \quad a = y, b = x$

$$n_a \geq -|n_{xy}|$$

YES

$$n_{aD} = n_a + |n_{xy}|$$
$$n_{bD} = n_b + |n_{xy}|$$
$$n_{cD} = -2 \cdot |n_{xy}|$$

NO

$$n_{aD} = 0$$
$$n_{bD} = n_b + n_{xy}^2 / |n_a|$$
$$n_{cD} = -|n_a|\left(1 + \left(n_{xy} / n_a\right)^2\right)$$

Analogously, if membrane effects are present, for the 3 characteristic membrane forces an equivalent set of 3 design forces is calculated:

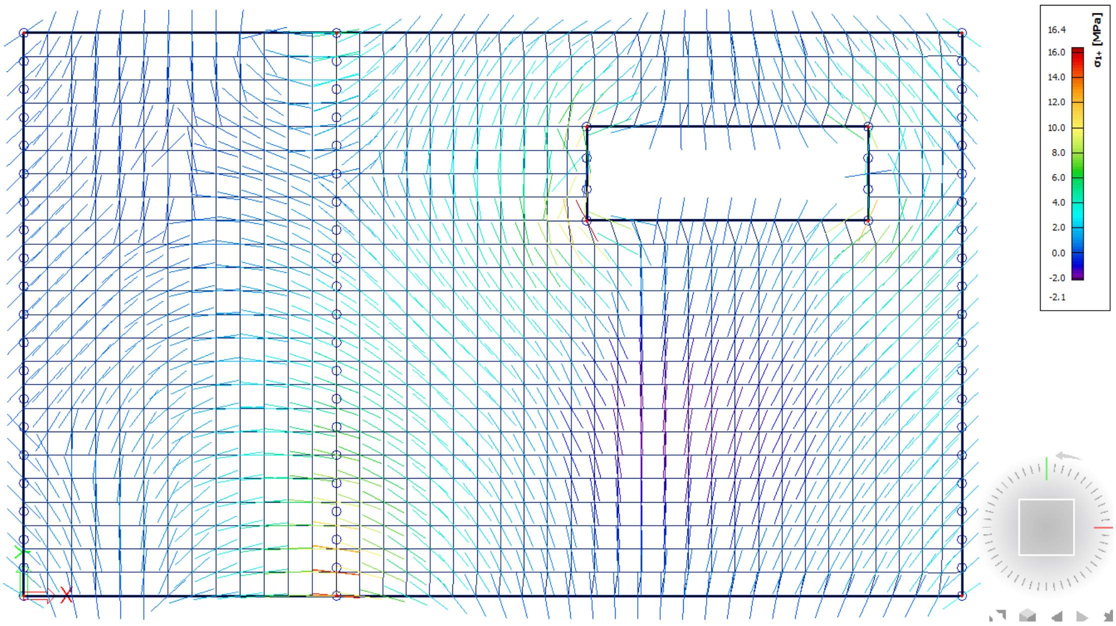
nx		***nxD***
ny	≈	nyD
nxy		ncD

Here, the quantity ncD does have a clear meaning: it is the compression force that has to be taken by the concrete compression struts. Therefore, to make sure that concrete crushing will not occur, the value ncD should be checked to be ≤ fcd.

<u>**Attention**</u>**:** These design magnitudes are not the ones used by SCIA Engineer for the reinforcement design in the Concrete menu. A much more refined transformation procedure is implemented there to calculate the design magnitudes from the basic magnitudes.

Principal magnitudes

Results menu → 2D Members → 2D stress/strain
Combination = ULS; Type forces = Principal magnitudes; Envelope = Maximum; Values = **sig1+**
Drawing = Trajectories



'1' and '2' refer to the principal directions, calculated based on Mohr's circle.

The first direction is the direction of maximum tension (or minimum compression). The second direction is the direction of maximum compression (or minimum tension).

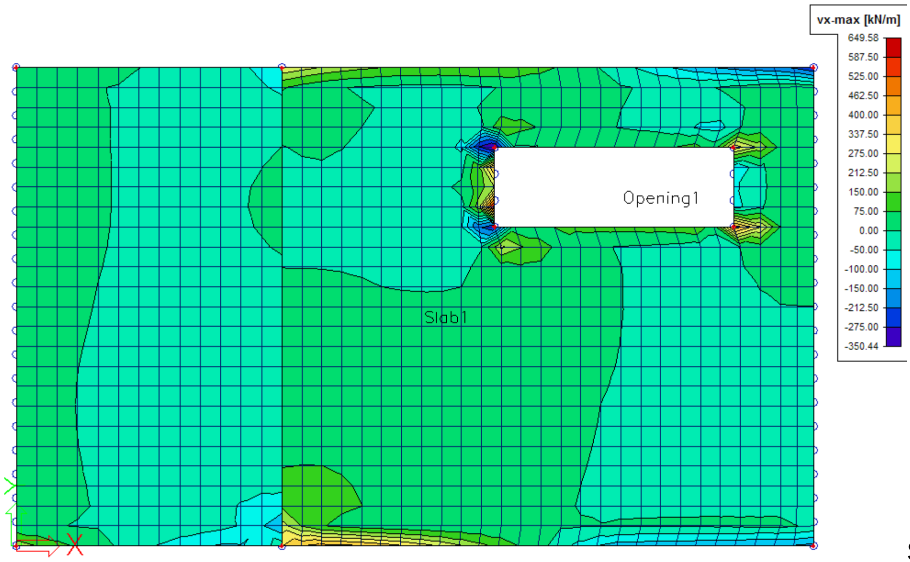
Keep in mind that the most economic reinforcement paths are the ones that follow the trajectories of the principal directions!

Comparison Mindlin ↔ Kirchhoff

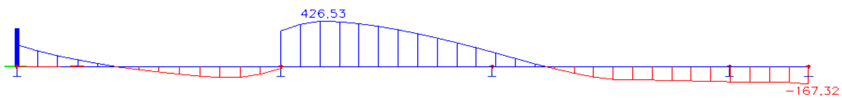
Shear force vx

Combination = ULS; Type forces = Basic magnitudes; Envelope = Maximum; Values = **v_x**

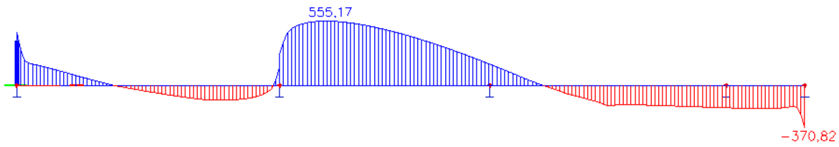
Mindlin



Section at lower edge

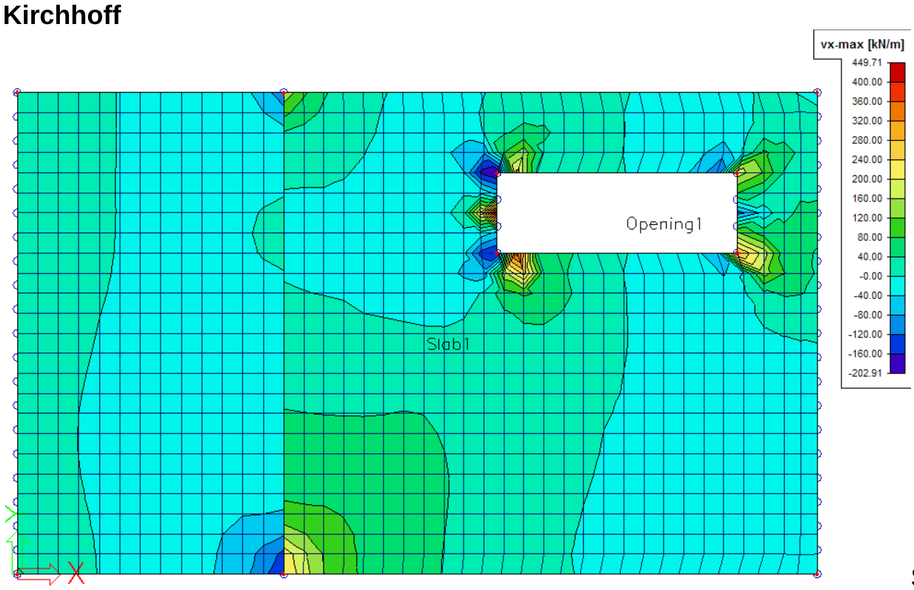


Mesh size = 0,25 m

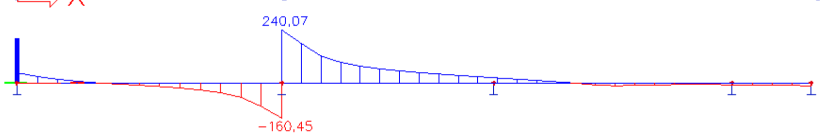


Mesh size = 0,05 m

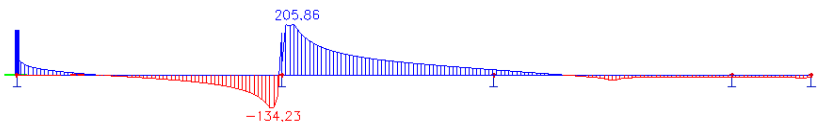
Kirchhoff



Section at lower edge



Mesh size = 0,25 m

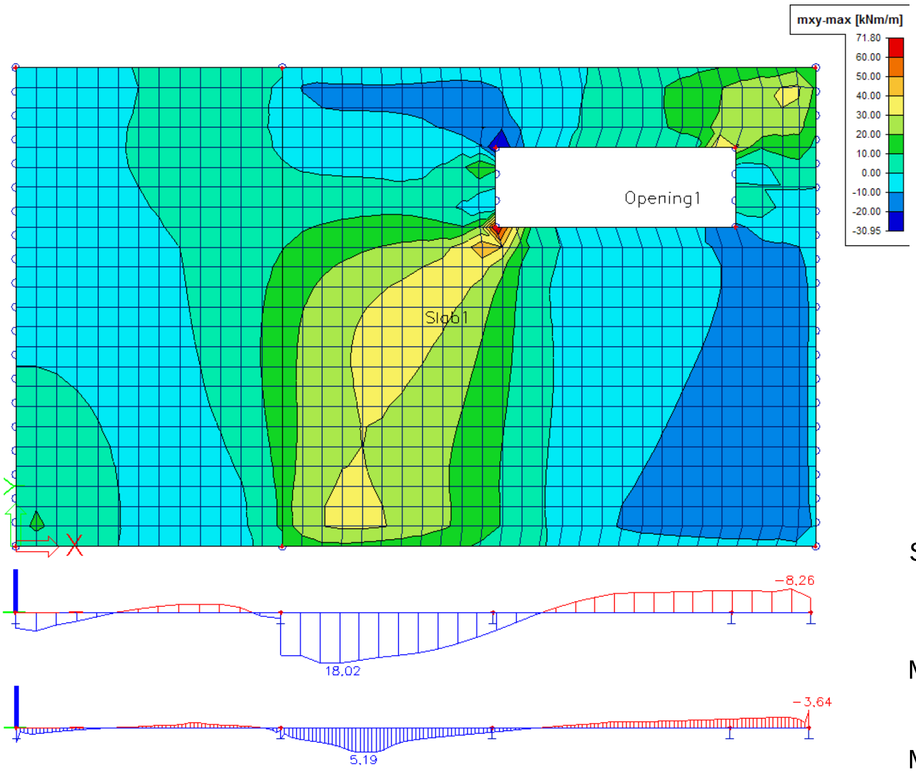


Mesh size = 0,05 m

Torsion moment mxy

Combination = ULS; Type forces = Basic magnitudes; Envelope = Maximum; Values = **m_xy**

Mindlin

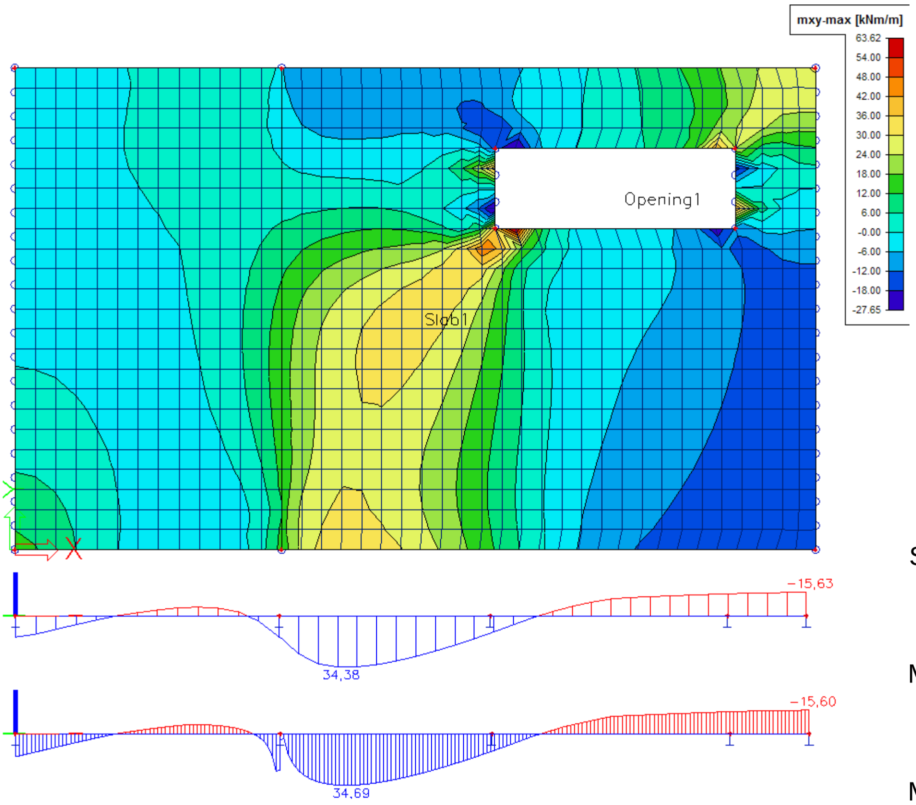


Section at lower edge

Mesh size = 0,25 m

Mesh size = 0,05 m

Kirchhoff



Section at lower edge

Mesh size = 0,25 m

Mesh size = 0,05 m

Conclusion: Kirchhoff gives the expected shear force values, Mindlin gives the expected torsion moments.

Concrete setups

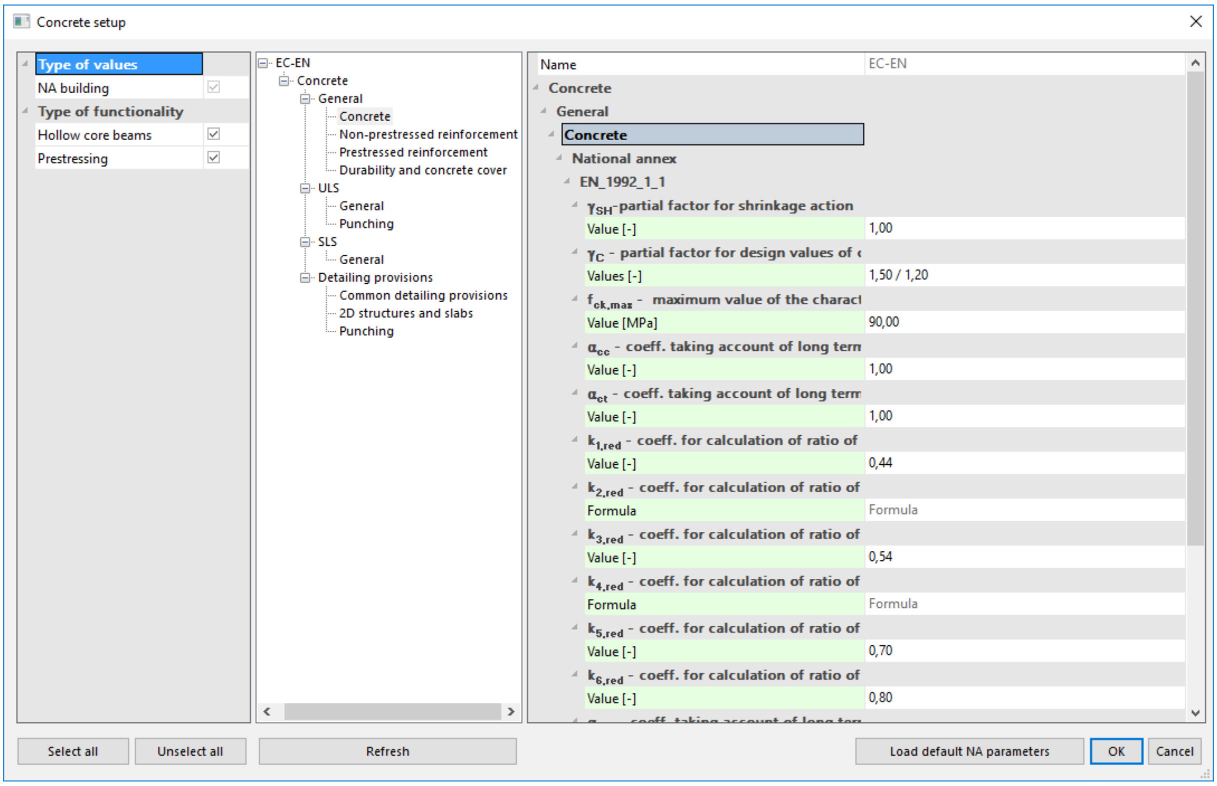
1_General setups

Setup 1: National Determined Parameters

Main menu → Project data → National annex […] → EN 1992-1-1 […]

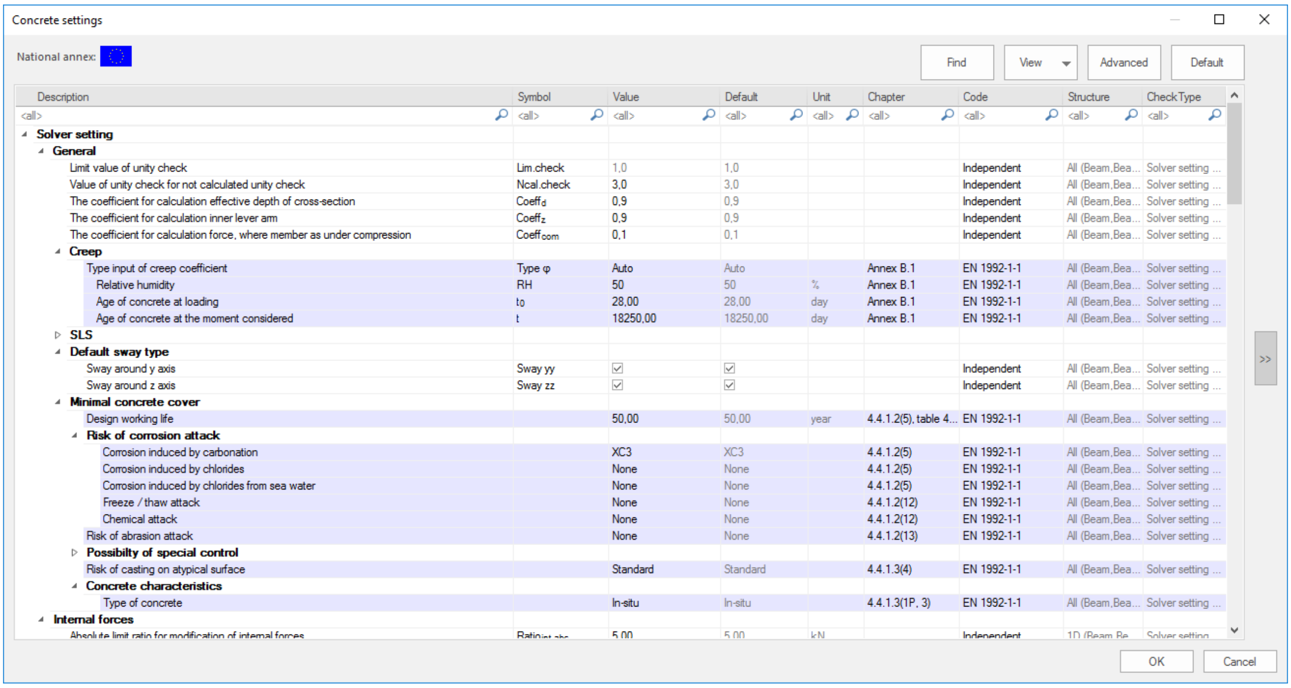
OR

Click on the flag at the bottom right of SCIA Engineer.



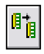
Setup 2: Concrete settings

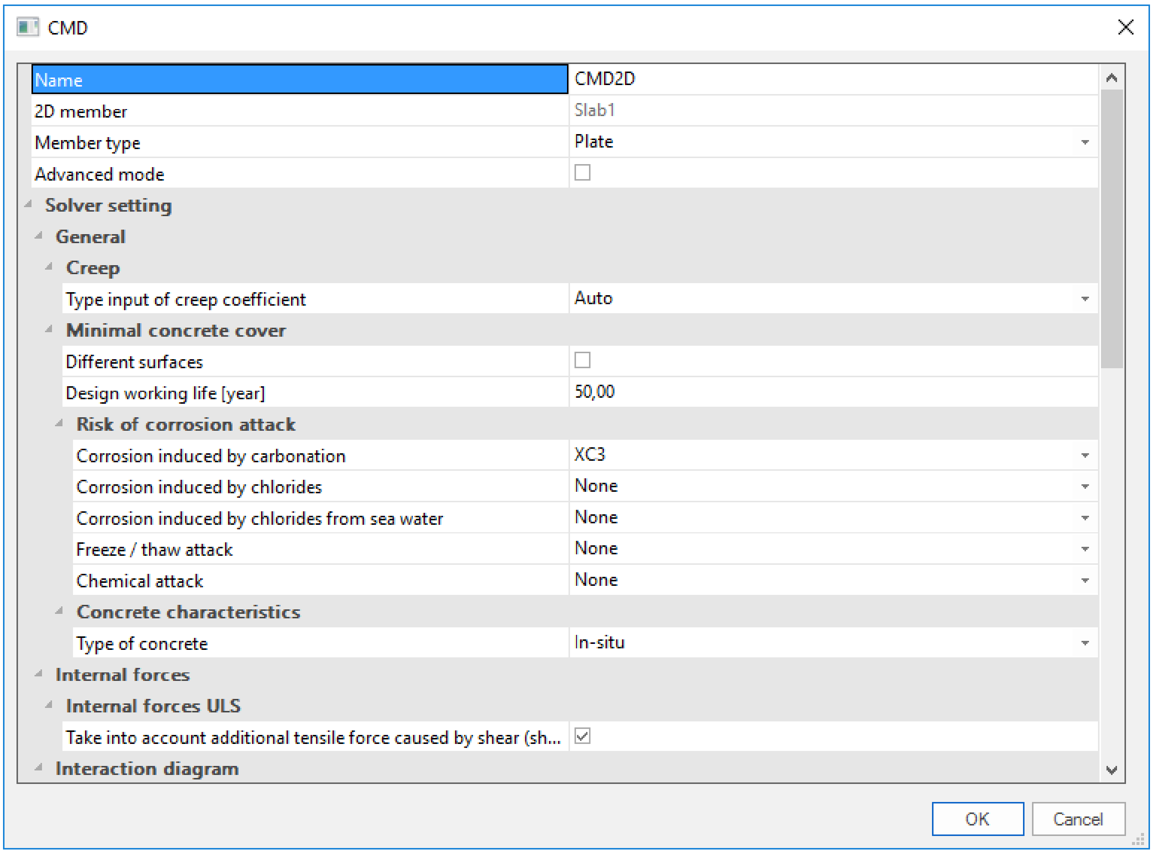
Concrete menu → Concrete settings



All of the adjustments made in one of the two general setups are valid for the **whole project**, except for the members to which ‘Member data’ are added.

2_ Member data

It is possible to **overwrite** the data from the general setups per 2D member, namely by means of Member data; see Concrete menu Setting per member – 2D member data.
On a plate with Member data appears a label, e.g. CMD1 (= Concrete member data). This label can be selected at any time to view or to adapt the data via the Properties menu. Since Member data are additional data, it is possible to copy them to other plates, via ‘Geometry manipulations’ toolbar  or via a right mouse click.


CMD
Name
CMD2D
2D member
Slab1
Member type
Plate
Advanced mode
Solver setting
General
Creep
Type input of creep coefficient
Auto
Minimal concrete cover
Different surfaces
Design working life [year]
50,00
Risk of corrosion attack
Corrosion induced by carbonation
XC3
Corrosion induced by chlorides
None
Corrosion induced by chlorides from sea water
None
Freeze / thaw attack
None
Chemical attack
None
Concrete characteristics
Type of concrete
In-situ
Internal forces
Internal forces ULS
Take into account additional tensile force caused by shear (sh...
Interaction diagram
OK
Cancel

ULS design

1_Reinforcement design

Internal forces

Concrete menu → Reinforcement design → 2D members → Internal forces

Basic (centroid)

The values shown here are exactly the same as in the Results menu; they are calculated by the FEM solver.

Design (centroid)

The values shown here are different from those in the Results menu.
- The design magnitudes in the **Results** menu are calculated by the **FEM** solver according to some simple formulas specified in EC-ENV.
- The design magnitudes in the **Concrete** menu are calculated by the **NEDIM** solver, where a much finer transformation procedure is implemented, based on the theory of Baumann.
These are the values that will be used for the SCIA Engineer reinforcement design.

Theory of Baumann.

1) Calculation of the lever arm.

The lever arm is necessary for the calculation of surface forces. Value z will be calculated in the direction of the angle of the first principal moment. The forces will be recalculated and a cross-section set will be created in this direction. The reinforcement will be designed for these recalculated forces and from the designed reinforcement the inner lever arm will be calculated.

Principal stresses and directions at both surfaces

$\sigma_{I-} = 0.48$ MPa $\sigma_{II-} = 0.11$ MPa -> $\alpha_{z-} = -5.86 = -5.86\ °$

$\sigma_{I+} = -0.11$ MPa $\sigma_{II+} = -0.48$ MPa -> $\alpha_{z+} = -5.86\ °$

-> direction for calculation inner lever arm

$\alpha_z = -5.86$

Recalculated forces to direction of inner lever arm

$n_z = 0.0 \quad m_z = 4970.4$

$$f_{cd} = \frac{\alpha_{cc} \cdot f_{ck}}{\gamma_C} = \frac{1 \cdot 20 \cdot 10^6}{1.5} = 13.33 \text{ MPa}$$

$d = 210$ mm

$$\eta = 1 - 0.5 \cdot \frac{\varepsilon_{c2}}{\varepsilon_{cu2}} = 1 - 0.5 \cdot \frac{0.0018}{0.0035} = 0.75$$

$$\beta = 1 - \frac{\frac{\varepsilon_{cu2}^2}{2} - \frac{\varepsilon_{c2}^2}{6}}{\varepsilon_{cu2}^2 - \frac{\varepsilon_{cu2} \cdot \varepsilon_{c2}}{2}} = 1 - \frac{\frac{0.0035^2}{2} - \frac{0.0018^2}{6}}{0.0035^2 - \frac{0.0035 \cdot 0.0018}{2}} = 0.389$$

$$\xi_{bal} = \frac{\varepsilon_{cu2}}{\varepsilon_{cu2} + \frac{f_{yk}}{\gamma_S \cdot E_s}} = \frac{0.0035}{0.0035 + \frac{500}{1.15 \cdot 200000}} = 0.617$$

$x_{bal} = \xi_{bal} \cdot d = 0.617 \cdot 210 = 0.13$

$n_{c,bal} = -\xi_{bal} \cdot d \cdot b \cdot \eta \cdot f_{cd} = -0.617 \cdot 210 \cdot 1000 \cdot 0.75 \cdot 13.33 = -1295$ kN/m

$n_z = 0$ kN/m $> n_{c,bal} = -1295$ kN/m => predominant tension

$$x = \frac{d_-}{2 \cdot \beta} \cdot \left(1 - \sqrt{1 - 4 \cdot \beta \cdot \frac{abs(m_z) - n_z \cdot (d_- - 0.5 \cdot h)}{b \cdot d_-^2 \cdot \eta \cdot f_{cd}}}\right)$$

$$= \frac{0.21}{2 \cdot 0.389} \cdot \left(1 - \sqrt{1 - 4 \cdot 0.389 \cdot \frac{abs(4970) - 0 \cdot (0.21 - 0.5 \cdot 250)}{1000 \cdot 0.21^2 \cdot 0.75 \cdot 13.33}}\right) = 2 \text{ mm}$$

$z = d - \beta \cdot x = 210 - 0.389 \cdot 2 = 209$ mm

$z_+ = 124$ mm

$z_- = 85$ mm

If value z can not be calculated it will be calculated according to formula: z = 0,9 . d

2) Calculation of normal forces at the surfaces of 2D element.

The inputted internal forces will be recalculated to both surfaces according the following formulas:

Lower surface

$$n_{x-} = \frac{n_x}{2} + \frac{m_x}{z} = \frac{0}{2} + \frac{4.93}{0.209} = 23.6 \text{ kN/m}$$

$$n_{y-} = \frac{n_y}{2} + \frac{m_y}{z} = \frac{0}{2} + \frac{1.22}{0.209} = 5.8 \text{ kN/m}$$

$$n_{xy-} = \frac{n_{xy}}{2} + \frac{m_{xy}}{z} = \frac{0}{2} + \frac{-0.385}{0.209} = -1.8 \text{ kN/m}$$

Upper surface

$$n_{x+} = \frac{n_x}{2} - \frac{m_x}{z} = \frac{0}{2} - \frac{4.93}{0.209} = -23.6 \text{ kN/m}$$

$$n_{y+} = \frac{n_y}{2} - \frac{m_y}{z} = \frac{0}{2} - \frac{1.22}{0.209} = -5.8 \text{ kN/m}$$

$$n_{xy+} = \frac{n_{xy}}{2} - \frac{m_{xy}}{z} = \frac{0}{2} - \frac{-0.385}{0.209} = 1.8 \text{ kN/m}$$

3) Calculation of principal forces at surfaces of 2D element.

The principal forces at both surfaces and the direction of the first principal force will be calculated according to the following formulas:

Lower surface

Principal forces at lower surface:

$$n_{I-} = \frac{n_{x-} + n_{y-}}{2} + \frac{1}{2} \cdot \sqrt{\left(n_{x-} - n_{y-}\right)^2 + 4 \cdot n_{xy-}^{\ 2}}$$

$$= \frac{23.6 + 5.8}{2} + \frac{1}{2} \cdot \sqrt{\left(23.6 - 5.8\right)^2 + 4 \cdot -1.8^2} = 23.8 \text{ kN/m}$$

$$n_{II-} = \frac{n_{x-} + n_{y-}}{2} - \frac{1}{2} \cdot \sqrt{\left(n_{x-} - n_{y-}\right)^2 + 4 \cdot n_{xy-}^{\ 2}}$$

$$= \frac{23.6 + 5.8}{2} - \frac{1}{2} \cdot \sqrt{\left(23.6 - 5.8\right)^2 + 4 \cdot -1.8^2} = 5.7 \text{ kN/m}$$

Direction of principal forces:

$$\alpha_{I-} = 0.5 \cdot \mathrm{ArcTg}\left(\frac{2 \cdot n_{xy-}}{n_{x-} - n_{y-}}\right) = 0.5 \cdot \mathrm{ArcTg}\left(\frac{2 \cdot -1.8}{23.6 - 5.8}\right) = -6\ °$$

Upper surface

Principal forces at upper surface:

$$n_{I+} = \frac{n_{x+} + n_{y+}}{2} + \frac{1}{2} \cdot \sqrt{\left(n_{x+} - n_{y+}\right)^2 + 4 \cdot n_{xy+}^{\ 2}}$$

$$= \frac{-23.6 + -5.8}{2} + \frac{1}{2} \cdot \sqrt{\left(-23.6 - -5.8\right)^2 + 4 \cdot 1.8^2} = -5.7 \text{ kN/m}$$

$$n_{II+} = \frac{n_{x+} + n_{y+}}{2} - \frac{1}{2} \cdot \sqrt{\left(n_{x+} - n_{y+}\right)^2 + 4 \cdot n_{xy+}^{\ 2}}$$

$$= \frac{-23.6 + -5.8}{2} - \frac{1}{2} \cdot \sqrt{\left(-23.6 - -5.8\right)^2 + 4 \cdot 1.8^2} = -23.8 \text{ kN/m}$$

Direction of principal forces:

$$\alpha_{I+} = 0.5 \cdot \mathrm{ArcTg}\left(\frac{2 \cdot n_{xy+}}{n_{x+} - n_{y+}}\right) - 90 = 0.5 \cdot \mathrm{ArcTg}\left(\frac{2 \cdot 1.8}{-23.6 - -5.8}\right) - 90 = -96\ °$$

4) Recalculation of principal forces at both surfaces to inputted directions.

The recalculation of the principal forces to the inputted direction will be done separately for both surfaces by using Baumann's transformation formula.

Lower surface

Angles for Baumann's transformation formula

$\alpha_{1-} = \alpha_{inp,1-} - \alpha_{I-} = 0 - -6 = 6\ °$

$\alpha_{2-} = \alpha_{inp,2-} - \alpha_{I-} = 90 - -6 = 96\ °$

$\alpha_{3-} = \alpha_{con-} - \alpha_{I-} = 45 - -6 = 51\ °$

Recalculated dimensional forces at lower surface (acc. to Baumann)

$$n_{Eds1-} = \frac{n_{I-} \cdot \sin(\alpha_{2-}) \cdot \sin(\alpha_{3-}) + n_{II-} \cdot \cos(\alpha_{2-}) \cdot \cos(\alpha_{3-})}{\sin(\alpha_{2-} - \alpha_{1-}) \cdot \sin(\alpha_{3-} - \alpha_{1-})}$$

$$= \frac{23.8 \cdot \sin(96) \cdot \sin(51) + 5.7 \cdot \cos(96) \cdot \cos(51)}{\sin(96 - 6) \cdot \sin(51 - 6)} = 25.4\ \text{kN/m}$$

$$n_{Eds2-} = \frac{n_{I-} \cdot \sin(\alpha_{3-}) \cdot \sin(\alpha_{1-}) + n_{II-} \cdot \cos(\alpha_{3-}) \cdot \cos(\alpha_{1-})}{\sin(\alpha_{3-} - \alpha_{2-}) \cdot \sin(\alpha_{1-} - \alpha_{2-})}$$

$$= \frac{23.8 \cdot \sin(51) \cdot \sin(6) + 5.7 \cdot \cos(51) \cdot \cos(6)}{\sin(51 - 96) \cdot \sin(6 - 96)} = 7.7\ \text{kN/m}$$

$$n_{Eds3-} = \frac{n_{I-} \cdot \sin(\alpha_{1-}) \cdot \sin(\alpha_{2-}) + n_{II-} \cdot \cos(\alpha_{1-}) \cdot \cos(\alpha_{2-})}{\sin(\alpha_{1-} - \alpha_{3-}) \cdot \sin(\alpha_{2-} - \alpha_{3-})}$$

$$= \frac{23.8 \cdot \sin(6) \cdot \sin(96) + 5.7 \cdot \cos(6) \cdot \cos(96)}{\sin(6 - 51) \cdot \sin(96 - 51)} = -3.7\ \text{kN/m}$$

Upper surface

Angles for Baumann's transformation formula

$\alpha_{1+} = \alpha_{inp,1+} - \alpha_{I+} = 0 - -96 = 96\ °$

$\alpha_{2+} = \alpha_{inp,2+} - \alpha_{I+} = 90 - -96 = 186\ °$

$\alpha_{3+} = \alpha_{con+} - \alpha_{I+} = 135 - -96 = 231\ °$

Recalculated dimensional forces at upper surface (acc. to Baumann)

$$n_{Eds1+} = \frac{n_{I+} \cdot \sin(\alpha_{2+}) \cdot \sin(\alpha_{3+}) + n_{II+} \cdot \cos(\alpha_{2+}) \cdot \cos(\alpha_{3+})}{\sin(\alpha_{2+} - \alpha_{1+}) \cdot \sin(\alpha_{3+} - \alpha_{1+})}$$

$$= \frac{-5.7 \cdot \sin(186) \cdot \sin(231) + -23.8 \cdot \cos(186) \cdot \cos(231)}{\sin(186 - 96) \cdot \sin(231 - 96)} = -21.7\ \text{kN/m}$$

$$n_{Eds2+} = \frac{n_{I+} \cdot \sin(\alpha_{3+}) \cdot \sin(\alpha_{1+}) + n_{II+} \cdot \cos(\alpha_{3+}) \cdot \cos(\alpha_{1+})}{\sin(\alpha_{3+} - \alpha_{2+}) \cdot \sin(\alpha_{1+} - \alpha_{2+})}$$

$$= \frac{-5.7 \cdot \sin(231) \cdot \sin(96) + -23.8 \cdot \cos(231) \cdot \cos(96)}{\sin(231 - 186) \cdot \sin(96 - 186)} = -4.0\ \text{kN/m}$$

$$n_{Eds3+} = \frac{n_{I+} \cdot \sin(\alpha_{1+}) \cdot \sin(\alpha_{2+}) + n_{II+} \cdot \cos(\alpha_{1+}) \cdot \cos(\alpha_{2+})}{\sin(\alpha_{1+} - \alpha_{3+}) \cdot \sin(\alpha_{2+} - \alpha_{3+})}$$

$$= \frac{-5.7 \cdot \sin(96) \cdot \sin(186) + -23.8 \cdot \cos(96) \cdot \cos(186)}{\sin(96 - 231) \cdot \sin(186 - 231)} = -3.7\ \text{kN/m}$$

5) Calculation of virtual forces at both surfaces to inputted directions.

The virtual forces are necessary to convert the pressure/tensile forces at the surface back to the center of the plate. The virtual force represents the equivalent force at the other side of the plate.

Virtual forces at both surfaces

Lower surface

Angles for Baumann's transformation formula

$\alpha_{1+} = \alpha_{inp,1+} - \alpha_{I-} = 0 - -6 = 6\,°$

$\alpha_{2+} = \alpha_{inp,2+} - \alpha_{I-} = 90 - -6 = 96\,°$

$\alpha_{3+} = \alpha_{con-} - \alpha_{I-} = 45 - -6 = 51\,°$

Recalculated virtual forces at lower surface (acc. to Baumann)

$$n_{Edsvirt1-} = \frac{n_{I-} \cdot \sin(\alpha_{2+}) \cdot \sin(\alpha_{3+}) + n_{II-} \cdot \cos(\alpha_{2+}) \cdot \cos(\alpha_{3+})}{\sin(\alpha_{2+} - \alpha_{1+}) \cdot \sin(\alpha_{3+} - \alpha_{1+})}$$
$$= \frac{23.8 \cdot \sin(96) \cdot \sin(51) + 5.7 \cdot \cos(96) \cdot \cos(51)}{\sin(96 - 6) \cdot \sin(51 - 6)} = 25.4\ \text{kN/m}$$

$$n_{Edsvirt2-} = \frac{n_{I-} \cdot \sin(\alpha_{3+}) \cdot \sin(\alpha_{1+}) + n_{II-} \cdot \cos(\alpha_{3+}) \cdot \cos(\alpha_{1+})}{\sin(\alpha_{3+} - \alpha_{2+}) \cdot \sin(\alpha_{1+} - \alpha_{2+})}$$
$$= \frac{23.8 \cdot \sin(51) \cdot \sin(6) + 5.7 \cdot \cos(51) \cdot \cos(6)}{\sin(51 - 96) \cdot \sin(6 - 96)} = 7.7\ \text{kN/m}$$

$$n_{Edsvirt3-} = \frac{n_{I-} \cdot \sin(\alpha_{1+}) \cdot \sin(\alpha_{2+}) + n_{II-} \cdot \cos(\alpha_{1+}) \cdot \cos(\alpha_{2+})}{\sin(\alpha_{1+} - \alpha_{3+}) \cdot \sin(\alpha_{2+} - \alpha_{3+})}$$
$$= \frac{23.8 \cdot \sin(6) \cdot \sin(96) + 5.7 \cdot \cos(6) \cdot \cos(96)}{\sin(6 - 51) \cdot \sin(96 - 51)} = -3.7\ \text{kN/m}$$

Upper surface

Angles for Baumann's transformation formula

$\alpha_{1-} = \alpha_{inp,1-} - \alpha_{I+} = 0 - -96 = 96\,°$

$\alpha_{2-} = \alpha_{inp,2-} - \alpha_{I+} = 90 - -96 = 186\,°$

$\alpha_{3-} = \alpha_{con+} - \alpha_{I+} = 135 - -96 = 231\,°$

Recalculated virtual forces at upper surface (acc. to Baumann)

$$n_{Edsvirt1+} = \frac{n_{I+} \cdot \sin(\alpha_{2-}) \cdot \sin(\alpha_{3-}) + n_{II+} \cdot \cos(\alpha_{2-}) \cdot \cos(\alpha_{3-})}{\sin(\alpha_{2-} - \alpha_{1-}) \cdot \sin(\alpha_{3-} - \alpha_{1-})}$$
$$= \frac{-5.7 \cdot \sin(186) \cdot \sin(231) + -23.8 \cdot \cos(186) \cdot \cos(231)}{\sin(186 - 96) \cdot \sin(231 - 96)} = -21.7\ \text{kN/m}$$

$$n_{Edsvirt2+} = \frac{n_{I+} \cdot \sin(\alpha_{3-}) \cdot \sin(\alpha_{1-}) + n_{II+} \cdot \cos(\alpha_{3-}) \cdot \cos(\alpha_{1-})}{\sin(\alpha_{3-} - \alpha_{2-}) \cdot \sin(\alpha_{1-} - \alpha_{2-})}$$
$$= \frac{-5.7 \cdot \sin(231) \cdot \sin(96) + -23.8 \cdot \cos(231) \cdot \cos(96)}{\sin(231 - 186) \cdot \sin(96 - 186)} = -4.0\ \text{kN/m}$$

$$n_{Edsvirt3+} = \frac{n_{I+} \cdot \sin(\alpha_{1-}) \cdot \sin(\alpha_{2-}) + n_{II+} \cdot \cos(\alpha_{1-}) \cdot \cos(\alpha_{2-})}{\sin(\alpha_{1-} - \alpha_{3-}) \cdot \sin(\alpha_{2-} - \alpha_{3-})}$$
$$= \frac{-5.7 \cdot \sin(96) \cdot \sin(186) + -23.8 \cdot \cos(96) \cdot \cos(186)}{\sin(96 - 231) \cdot \sin(186 - 231)} = -3.7\ \text{kN/m}$$

6) Recalculation of forces at surfaces to center of gravity of cross-section.

Using the transformed dimensional forces and virtual forces the internal forces at the center of the plate can be calculated.

Lower surface

Dimensional forces of lower surface transformed to centroid

$$n_{Ed1-} = n_{Eds1-} + n_{Edsvirt1+} = 25.4 + -21.7 = 3.7 \text{ kN/m}$$

$$m_{Ed1-} = n_{Eds1-} \cdot z_- - n_{Edsvirt1+} \cdot z_+ = 25.4 \cdot 85 - -21.7 \cdot 124 = 4.9 \text{ kNm/m}$$

$$n_{Ed2-} = n_{Eds2-} + n_{Edsvirt2+} = 7.7 + -4.0 = 3.7 \text{ kN/m}$$

$$m_{Ed2-} = n_{Eds2-} \cdot z_- - n_{Edsvirt2+} \cdot z_+ = 7.7 \cdot 85 - -4.0 \cdot 124 = 1.2 \text{ kNm/m}$$

$$n_{Ed3-} = n_{Eds3-} + n_{Edsvirt3+} = -3.7 + -3.7 = -7.4 \text{ kN/m}$$

$$m_{Ed3-} = n_{Eds3-} \cdot z_- - n_{Edsvirt3+} \cdot z_+ = -3.7 \cdot 85 - -3.7 \cdot 124 = 0.1 \text{ kNm/m}$$

Upper surface

Dimensional forces of upper surface transformed to centroid

$$n_{Ed1+} = n_{Eds1+} + n_{Edsvirt1-} = -21.7 + 25.4 = 3.7 \text{ kN/m}$$

$$m_{Ed1+} = - n_{Eds1+} \cdot z_+ + n_{Edsvirt1-} \cdot z_- = - -21.7 \cdot 124 + 25.4 \cdot 85 = 4.9 \text{ kNm/m}$$

$$n_{Ed2+} = n_{Eds2+} + n_{Edsvirt2-} = -4.0 + 7.7 = 3.7 \text{ kN/m}$$

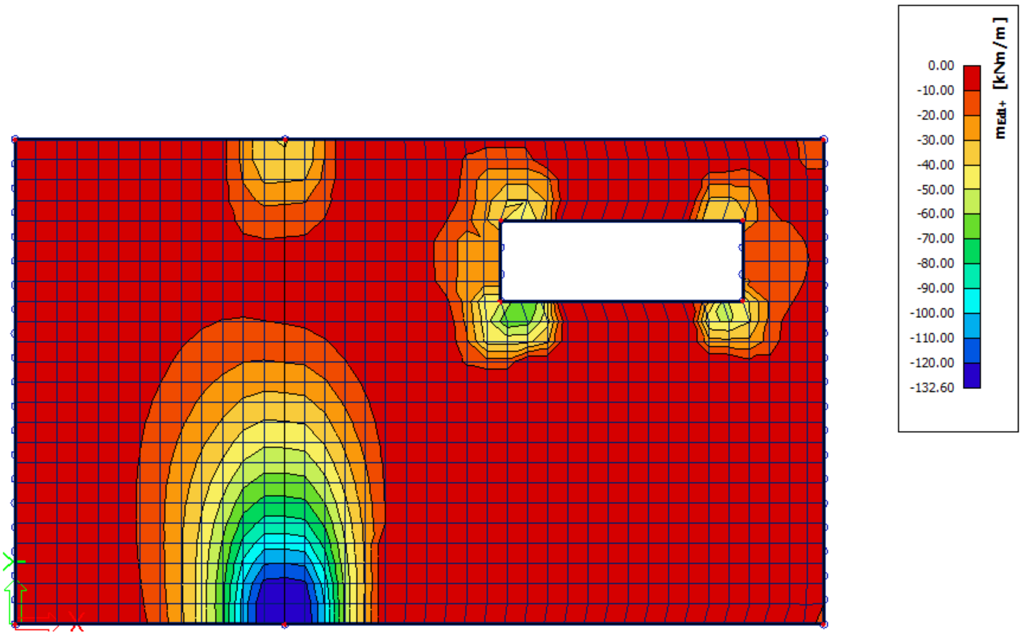
$$m_{Ed2+} = - n_{Eds2+} \cdot z_+ + n_{Edsvirt2-} \cdot z_- = - -4.0 \cdot 124 + 7.7 \cdot 85 = 1.2 \text{ kNm/m}$$

$$n_{Ed3+} = n_{Eds3+} + n_{Edsvirt3-} = -3.7 + -3.7 = -7.4 \text{ kN/m}$$

$$m_{Ed3+} = - n_{Eds3+} \cdot z_+ + n_{Edsvirt3-} \cdot z_- = - -3.7 \cdot 124 + -3.7 \cdot 85 = 0.1 \text{ kNm/m}$$

The available values are: mEd,1+, mEd,2+, mEd,c+, mEd,1-, mEd,2-, mEd,c-, nEd,1+, nEd,2+, nEd,c+, nEd,1-, nEd,2-, nEd,c- and vEd. "+" and "-" stand for the design values at respectively the positive and the negative side of the local z-axis of the 2D member. "1" and "2" stand for the reinforcement directions, which are by default respectively the local x- and y- direction of the 2D member. (mEd,c+ and mEd,c- are the design moments that would have to be taken by the concrete, but they have no real significance for the reinforcement design.)

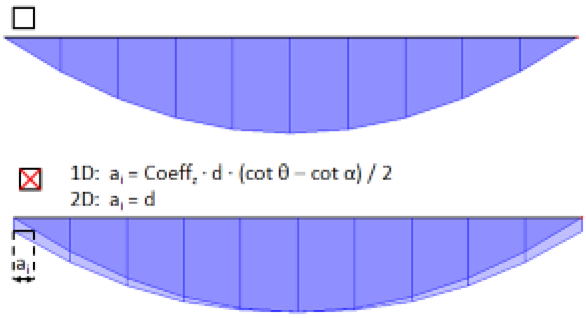
Combination = ULS; Type values = Design magnitudes; Value = **mEd,1+**



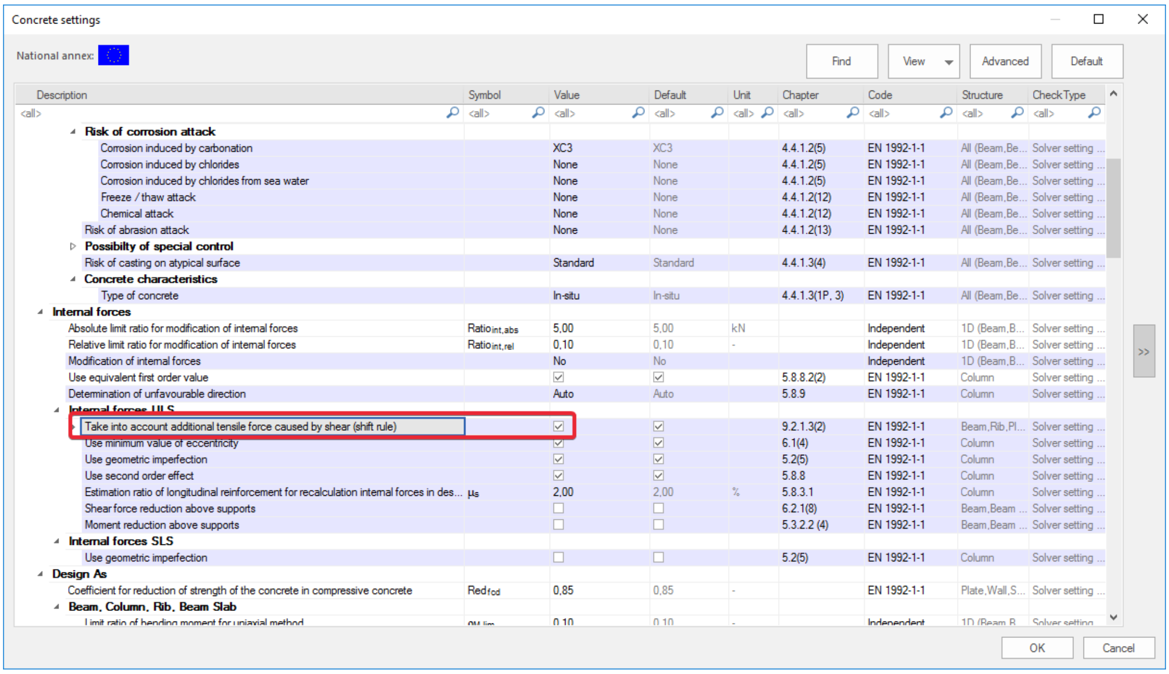
Compare the result for this value mEd,1+ (Concrete menu) with the result for the equivalent value mxD+ (Result menu) shown on p.10.
Despite the different transformation procedures, the general image of the results will be similar for *orthogonal* reinforcement directions (acc. to the local x and y axes). The largest difference is caused by the shift rule that is only taken into account in the design magnitudes calculated by the NEDIM solver (values mEd,1 and mEd,2).

Shift rule

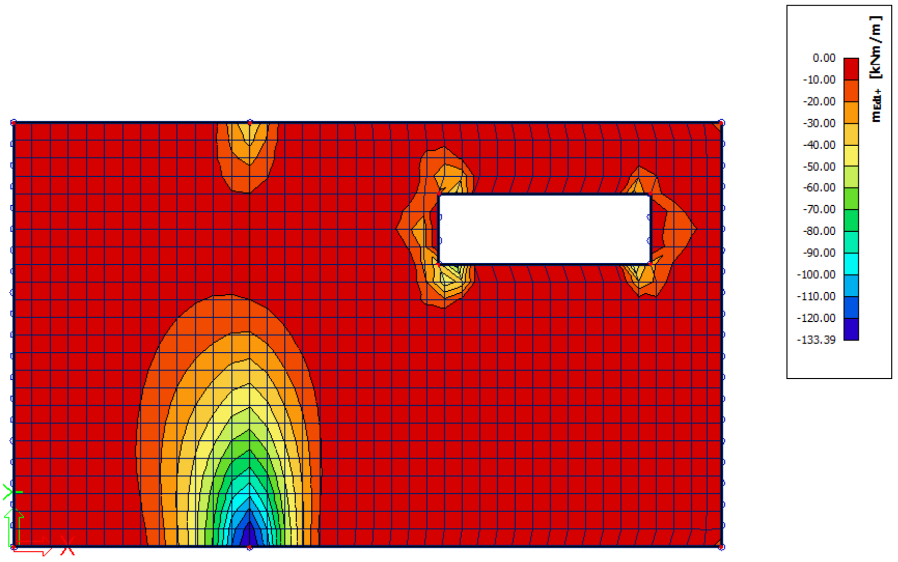
The shift rule takes into account the additional tensile force caused by the shear force by shifting the moment line by a distance a_i. a_i is determined as in the image below.



The shift rule is taken into account in the default concrete settings. You can deactivate this option in the concrete settings.



If we uncheck this option the general image of mEd,1+ is closer to the one obtained for mxD+ (page 10).



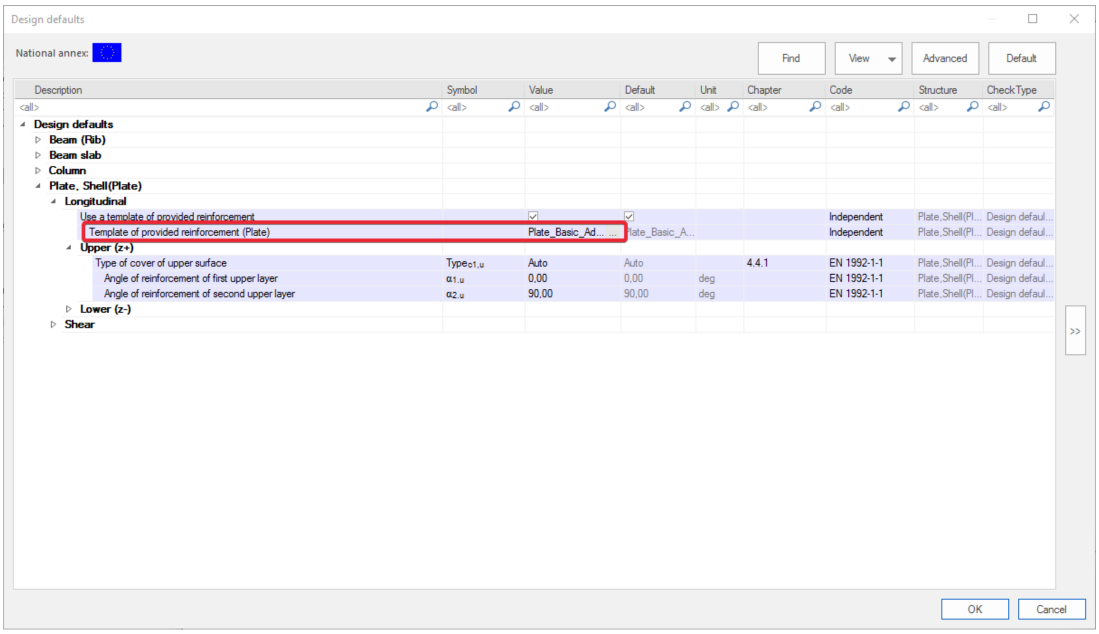
Provided reinforcement

Before calculating the theoretical reinforcement it is possible to add a template of reinforcement to your plate(s). This template can be used to:

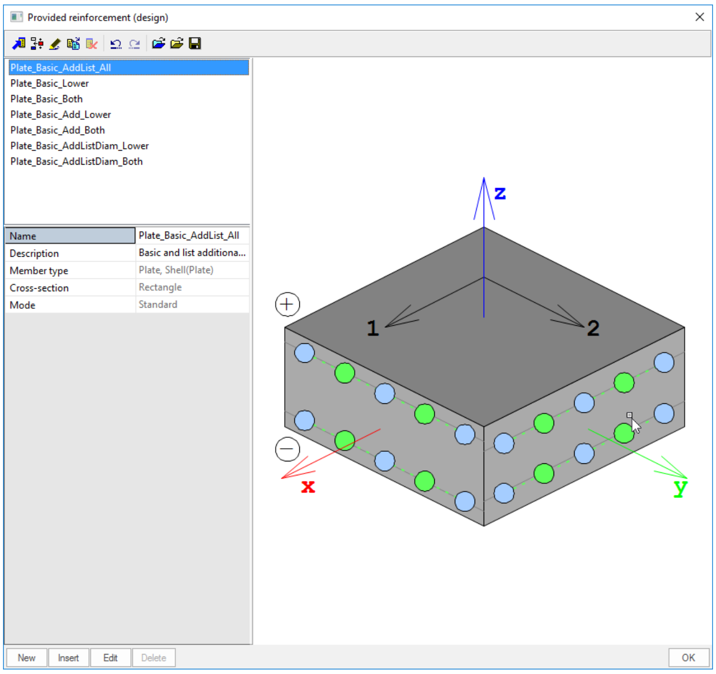
- *Compare the template with the calculated theoretical reinforcement. By doing this it is easy to see where this basic template is not sufficient.*
- *Perform the punching design, Crack width check and the code dependent deflections.*

*The reinforcement added by the template is called **Provided reinforcement**.*

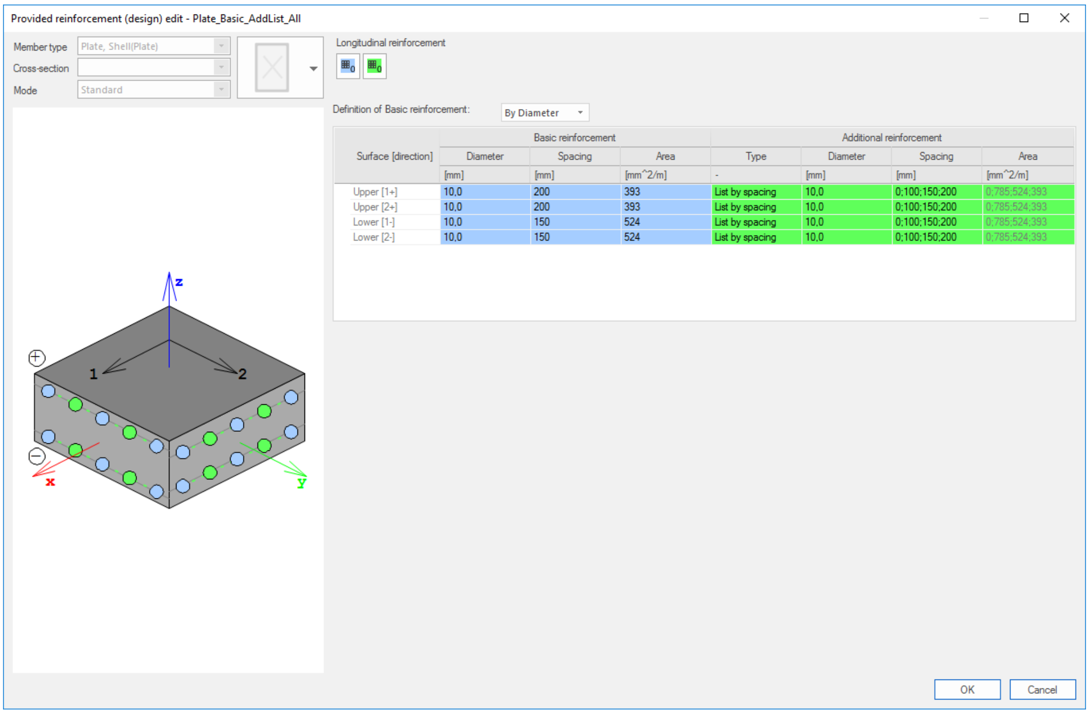
*To add **Provided reinforcement** go to Concrete → Reinforcement design → Design defaults*



Click on the 3 dots next to the 'Template of provided reinforcement (Plate)'. This opens a window with all the default templates.



You can select one of these templates, make a new one or edit one of the existing templates. Select the first template and click 'Edit'.



In this window the reinforcement can be defined. There are 2 types of reinforcement in templates:

- ***Basic reinforcement:*** *This type of reinforcement is added over the entire plate.*
- ***Additional reinforcement:*** *This type of reinforcement is only added in zones where, according to the calculated theoretical reinforcement , extra reinforcement is needed. You can define a single diameter and spacing as extra reinforcement. Or a list of reinforcement with either various diameters or various spacings.*

Note:

1) *The diameter used for the Additional reinforcement is used also to perform the calculation of the theoretical required reinforcement.*

2) *In the design defaults you can change the reinforcement directions. These directions are respected by as well the provided as the theoretical required reinforcement.*

Description	Symbol	Value	Default	Unit	Chapter	Code	Structure	CheckType
<all>	<all>	<all>	<all>	<all>	<all>	<all>	<all>	<all>
Design defaults								
Beam (Rib)								
Beam slab								
Column								
Plate, Shell(Plate)								
Longitudinal								
Use a template of provided reinforcement		☑	☑			Independent	Plate,Shell(Pl...	Design defaul...
Template of provided reinforcement (Plate)		Plate_Basic_Ad... ...	Plate_Basic_A...			Independent	Plate,Shell(Pl...	Design defaul...
Upper (z+)								
Type of cover of upper surface	$Type_{c1,u}$	Auto	Auto		4.4.1	EN 1992-1-1	Plate,Shell(Pl...	Design defaul...
Angle of reinforcement of first upper layer	$\alpha_{1,u}$	0,00	0,00	deg		EN 1992-1-1	Plate,Shell(Pl...	Design defaul...
Angle of reinforcement of second upper layer	$\alpha_{2,u}$	90,00	90,00	deg		EN 1992-1-1	Plate,Shell(Pl...	Design defaul...
Lower (z-)								
Type of cover of lower surface	$Type_{c1,l}$	Auto	Auto		4.4.1	EN 1992-1-1	Plate,Shell(Pl...	Design defaul...
Angle of reinforcement of first lower layer	$\alpha_{1,l}$	0,00	0,00	deg		EN 1992-1-1	Plate,Shell(Pl...	Design defaul...
Angle of reinforcement of second lower layer	$\alpha_{2,l}$	90,00	90,00	deg		EN 1992-1-1	Plate,Shell(Pl...	Design defaul...
Shear								

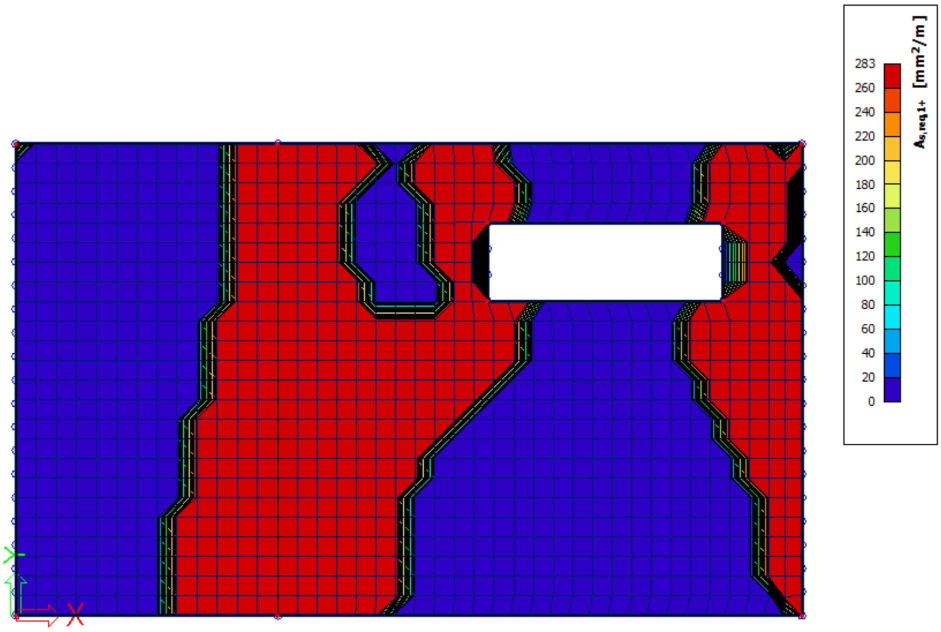
- ***Theoretically required reinforcement***

Concrete → Reinforcement design → 2D members → Reinforcement design (ULS)

In the menu Reinforcement design (ULS) you have 4 types of values:

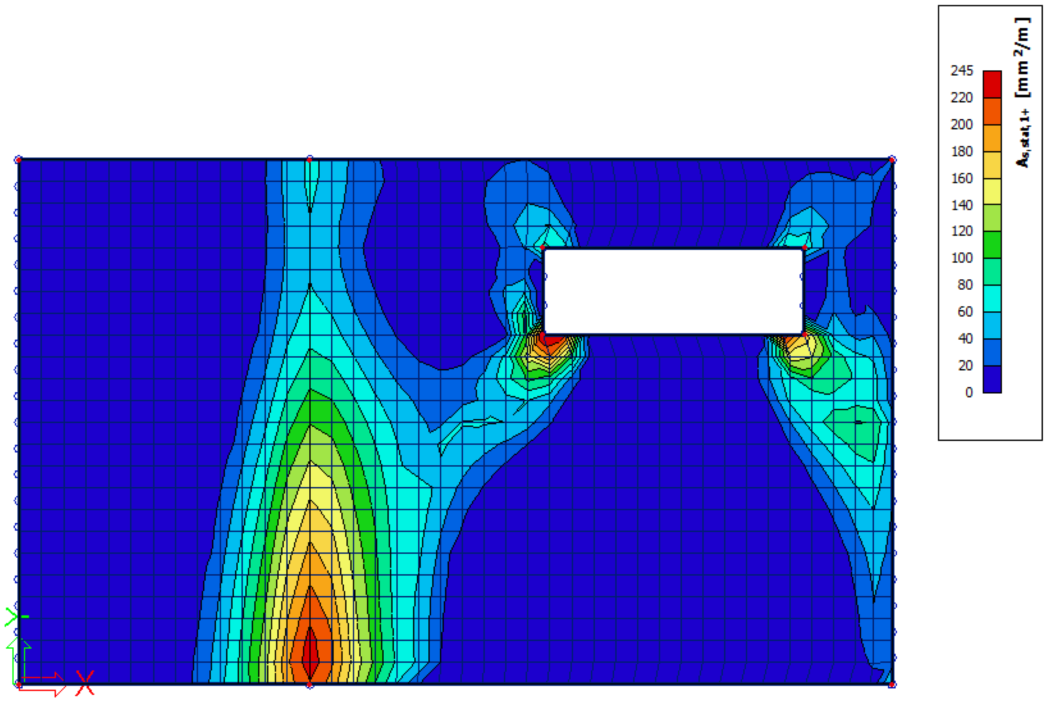
- **Required:** These values represent the theoretical reinforcement calculated by SCIA Engineer. This takes into account the detailing provisions.

Plate, Shell(Plate)		
Longitudinal		
Check min. ratio of principal reinforcement		☑
Type of the minimum tension principal reinforcement for the upper surface		Auto
Type of the minimum tension principal reinforcement for the lower surface		Auto
Check max. ratio of principal reinforcement		☑
Check min. transverse ratio of secondary reinforcement		☐
Check min. bar distance		☑
Minimal bar distance	slp.min	20
Check max.spacing of principal longitudinal reinforcement		☑
Check max.spacing of secondary longitudinal reinforcement		☑
Shear		
Check min. ratio of shear reinforcement		☑
Check min. thickness of member with shear reinforcement		☑
Min. thickness of member with shear reinforcement	h_{min}	200
Check max. spacing of shear links		☑
Max. spacing of shear links	$Coeff_{smax.p.s}$	0,8



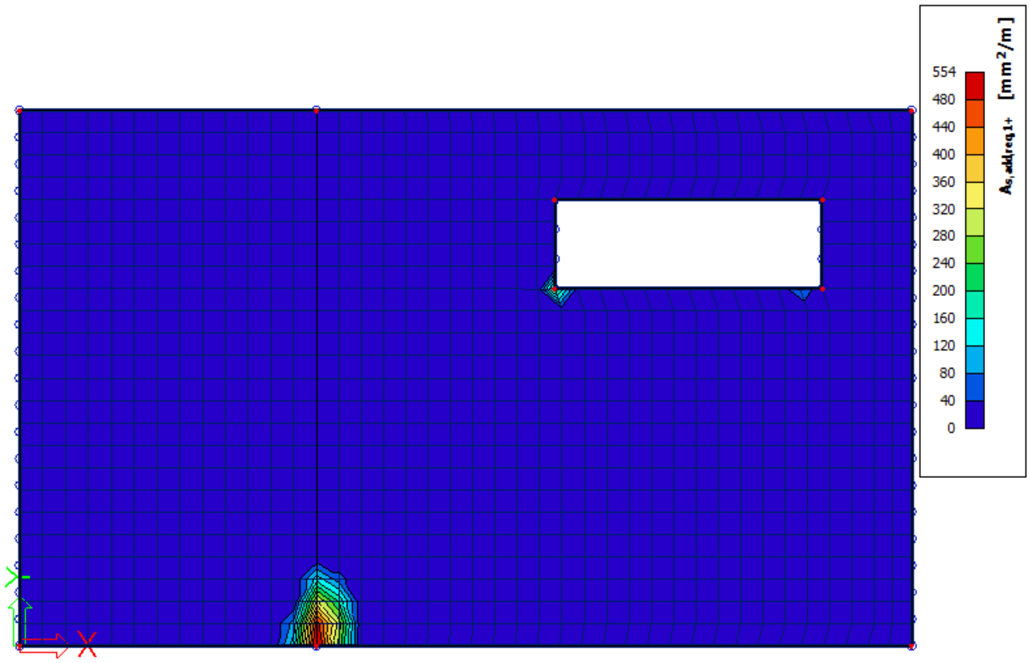
As,req1+: Theoretical required reinforcement on the top side of the plate (positive z direction) in the first reinforcement direction. Taking into account the detailing provisions.

- **Required (statically):** These values represent the theoretical reinforcement calculated by SCIA Engineer **without** the detailing provisions taken into account.



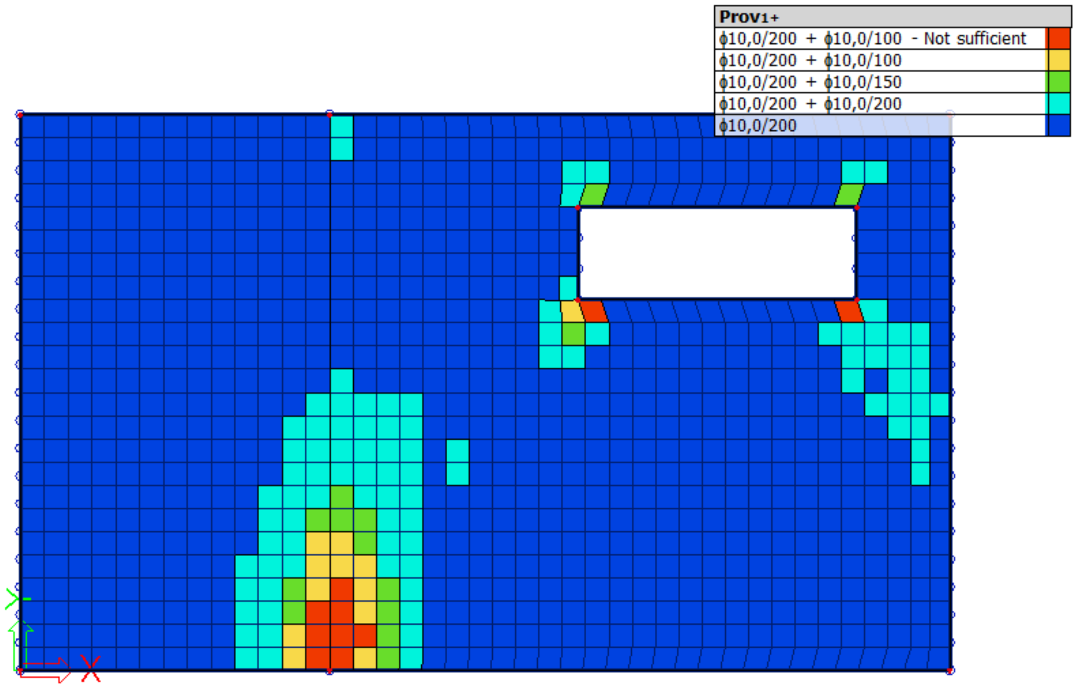
As,stat1+: Theoretical required reinforcement on the top side of the plate (positive z direction) in the first reinforcement direction. **Without** taking into account the detailing provisions.

- **Required (additional):** These values show if there is extra reinforcement needed on top of the provided reinforcement. Areas where this value is 0 are areas where no extra reinforcement is needed (compared to the provided reinforcement). Areas where these values are not 0 are areas where the provided reinforcement is not sufficient.



As,add,req1+: Theoretical additional required reinforcement on top of the provided reinforcement on the top side of the plate (positive z direction) in the first reinforcement direction.

- **Provided:** These values show you the provided reinforcement defined in the templates.



As,Prov1+: Provided reinforcement on the plate. If elements are red the additional reinforcement in the template is not sufficient.

Calculation of longitudinal reinforcement

The theoretical longitudinal reinforcement is calculated out of the design internal forces.

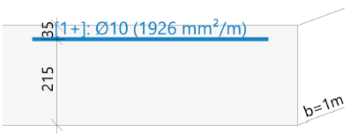
Design direction [α=90°]

[2+]: upper surface
n_{Ed} = 327 kN/m m_{Ed} = -21.7 kNm/m [EN-ULS (STR/GEO) Set B/2]
Ø10 mm (d_1=45 mm ->d=205 mm)
$A_{s,req}$=688 mm^2/m (tensile)
(ρ=0,336%)

[2-]: lower surface
n_{Ed} = 249 kN/m m_{Ed} = -15 kNm/m [EN-ULS (STR/GEO) Set B/3]
Ø10 mm (d_1=45 mm ->d=205 mm)
$A_{s,req}$=70 mm^2/m (tensile)
(ρ=0,034%)

Design direction [α=0°]

[1+]: upper surface
n_{Ed} = 327 kN/m m_{Ed} = -129 kNm/m [EN-ULS (STR/GEO) Set B/2]
Ø10 mm (d_1=35 mm ->d=215 mm)
$A_{s,req}$=1926 mm^2/m (tensile)
(ρ=0,896%)



Calculation of shear reinforcement

Before calculating the shear reinforcement two checks are done:

1) $V_{Ed} \leq V_{Rd,max}$: The design internal forces on the plate should be lower or equal to the maximum shear resistance of the plate.

$$V_{Rd,max} = \frac{\alpha_{cw} \cdot b_w \cdot z \cdot \nu_1 \cdot f_{cd}}{\left(\mathrm{cotg}(\theta) + \mathrm{tg}(\theta)\right)}$$

2) $V_{Ed} < V_{Rdc}$: If V_{Ed} is smaller than V_{Rdc} no shear reinforcement is required. If this is not the case punching shear reinforcement will be automatically calculated by SCIA Engineer.

$$v_{Rdc} = \max\left(v_{Rdc};v_{Rdcmin}\right)$$

$$v_{Rdc} = \max\left(10^{6}\cdot\left(C_{Rdc}\cdot k\cdot\left(100\cdot\rho_1\cdot f_{ck}\right)^{\frac{1}{3}}+k_1\cdot\sigma_{cp}\right)\cdot d;0\right)$$

$$v_{Rdcmin} = \max\left(10^{6}\cdot\left(v_{min}+k_1\cdot\sigma_{cp}\right)\cdot d;0\right)$$

Check shear capacity (without shear reinforcement)

Check $v_{Rd,max}$

v_{Ed} = 145 kN/m ≤ $v_{Rd,max}$ = 653 kN/m **(OK)**

Check v_{Rdc}

v_{Ed} = 145 kN/m > v_{Rdc} = 113 kN/m **(NOT OK, shear reinforcement is required)**

Statically required shear reinforcement

$$f_{ywd,req} = \frac{f_{ywk,req}}{\gamma_S} = \frac{500}{1.15} = 435\ \text{MPa}$$

$$A_{sw,req} = \frac{v_{Ed}}{z\cdot f_{ywd,req}\cdot \text{cotg}(\theta)} = \frac{145.5}{0.18\cdot 435\cdot \text{cotg}(40)} = 1559\ \text{mm}^2/\text{m}$$

Required shear reinforcement

32ϕ8/m² => $A_{sw,req}$ = 1559mm²/m²

Shear resistance (vRD,max) is not sufficient

When VEd > VRd,max the following error appears in the output of the reinforcement design.

Warning	Punching shear resistance at the column perimeter (vRd,max) is not sufficient acc. to §6.4.3(2).	Increase the column size or change plate properties (use higher grade of concrete material or increase the thickness).

This error message is found at locations with high peak values for the shear stress. Most of the time these peak values are singularities, and do not occur in reality. You have roughly 2 options: you can just ignore the peaks or average them, for example by means of Averaging strips.

Averaging strips

Concrete menu → Result tools 2D → Averaging strip

Add averaging strips to the short sides of the opening, where the line supports are located and above the line supports over the total width of the slab.

RS

Name	RS1
Type	Strip
Width [m]	0,700
Direction	both

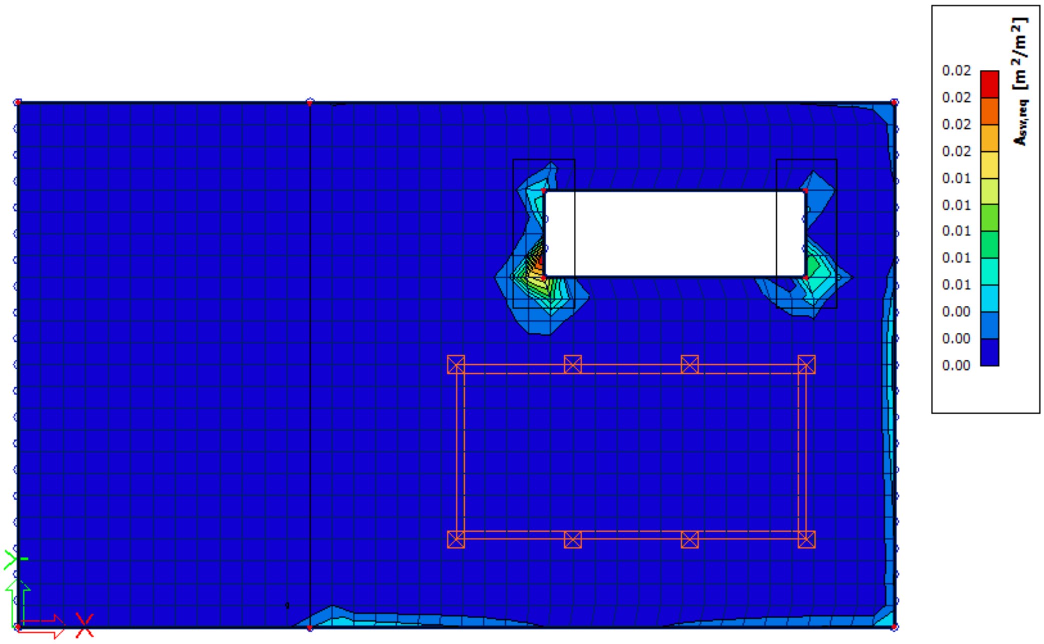
As a basic rule, take the width of the averaging strips equal to the width of the support + 1 to 2 times the thickness of the slab. You can find more information about the direction of the Averaging strip in Annex 4.

When asking results now, select the option ‘Averaging of peak’ in the Properties menu!

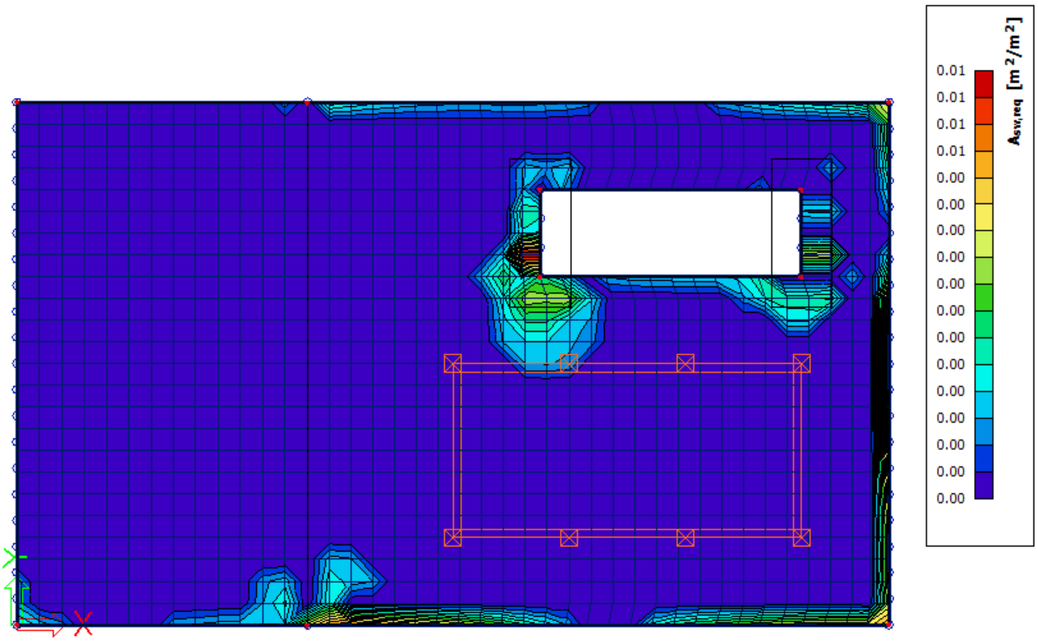
Result case	
Type of load	Combinations
Combination	EN-ULS (STR/GEO) Set B
Envelope (for 2D drawing)	Absolute extreme
Averaging of peak	☑

When asking the values for the theoretical reinforcement again, the peak should be lowered and the error should disappear.

Asw,req without Averaging of peaks

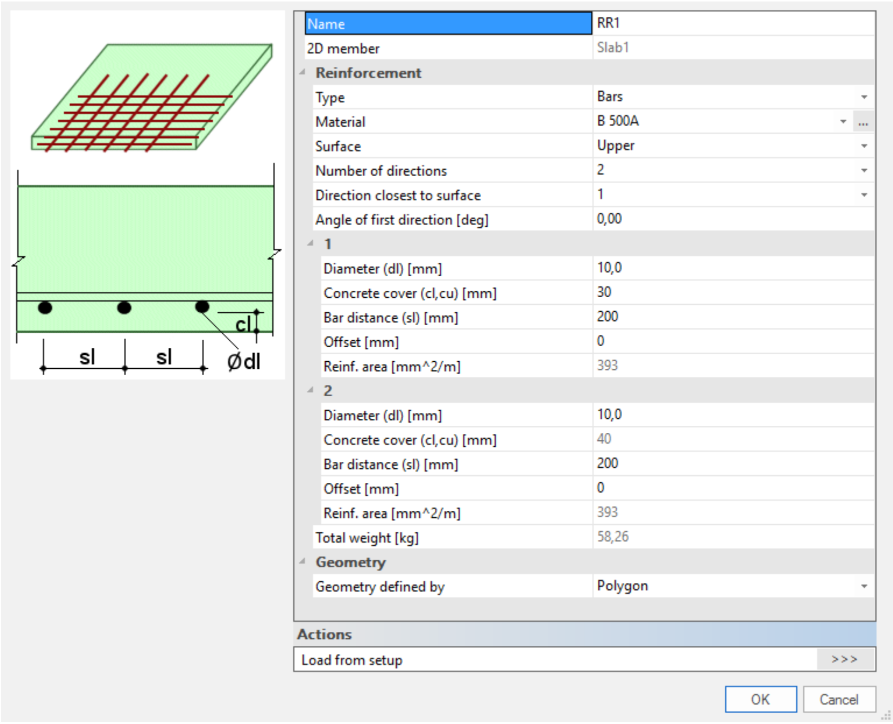


Asw,req with averaging of peaks

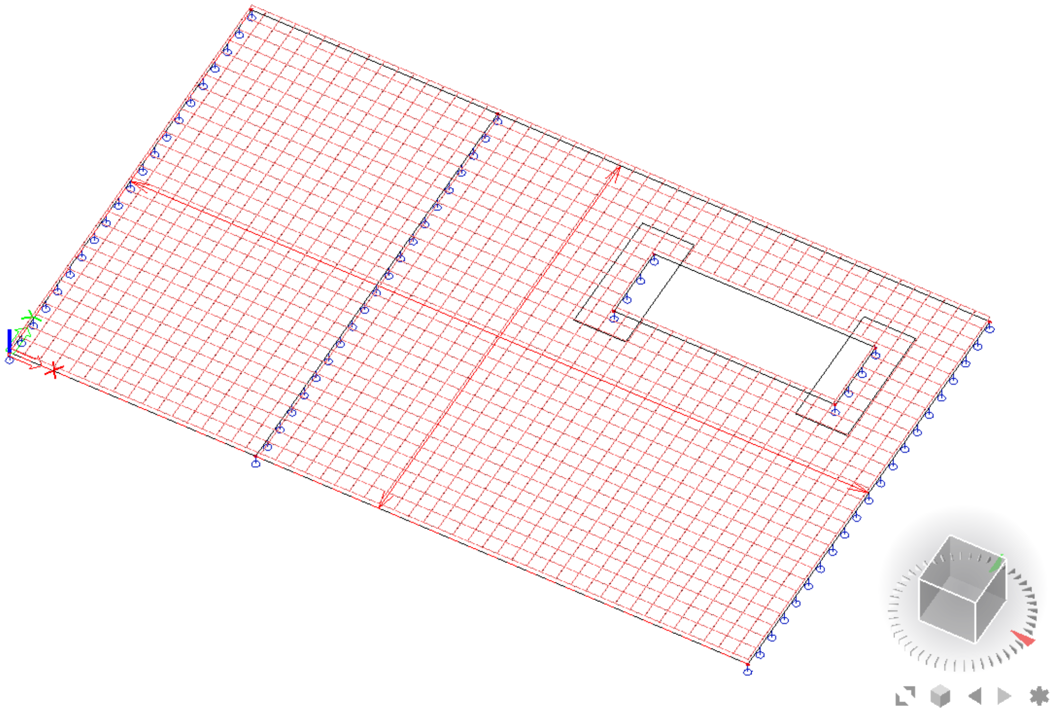


2_Practical reinforcement design

Next to theoretical required and provided reinforcement you have also practical or **User** reinforcement. This type of reinforcement can be added to the plate via Concrete → Reinforcement input + edit → Reinforcement 2D.



This reinforcement is to be added separately at the upper and lower side, and in the different reinforcement directions.



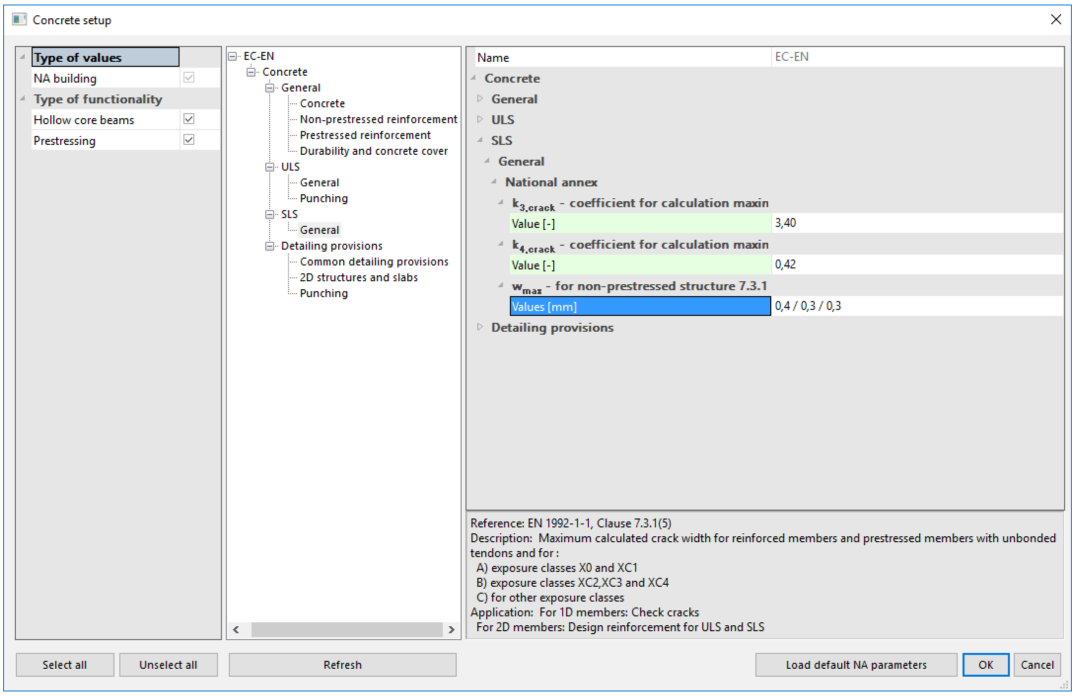
Note: You can add multiple layers of practical reinforcement on the same area. The reinforcement added to this area is the sum of all these layers.

Crack control

1_Input data for crack control

Maximum crack width

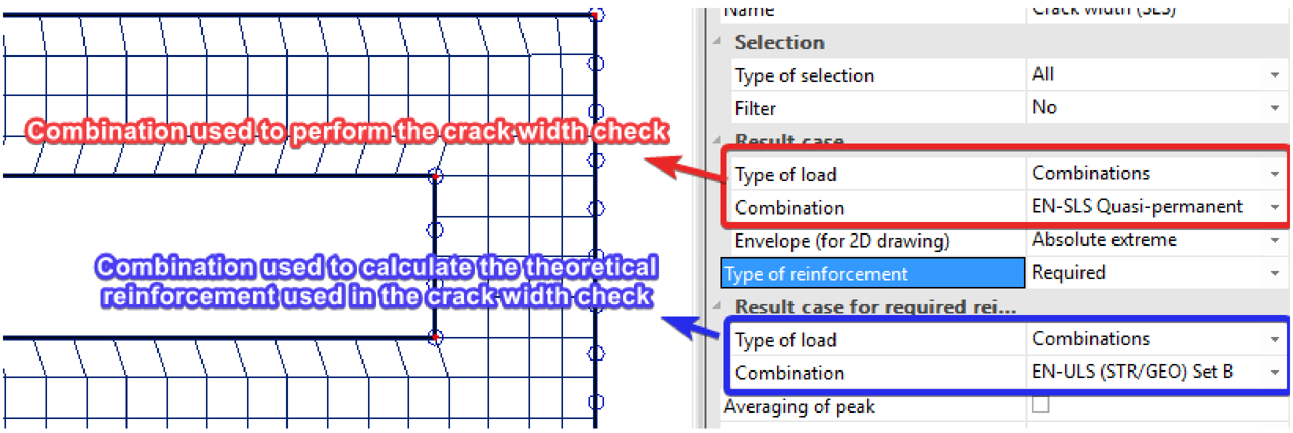
The values of the maximum crack width (w_{max}) are national determined parameters, dependent on the chosen exposure class. Therefore, this value can be found in the setup for National Determined Parameters, via the Main menu → Project data → National annex […] → EN 1992-1-1 […].



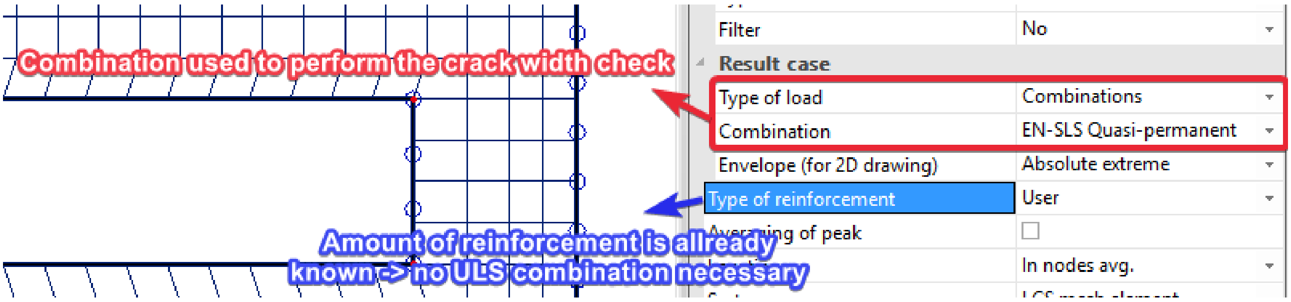
Type of used reinforcement

You can perform the Crack width check for all three types of reinforcement (Required, provided and user reinforcement). The crack width check is performed on a Quasi permanent SLS combination.
If the type of reinforcement used for the crack width check is either the provided or required reinforcement an ULS combination should be chosen as well. This is necessary because the required/provided reinforcement is calculated based on an ULS combination. After this reinforcement is calculated it can be used to perform the crack width check. All this is done automatically and can be set in the properties window of the crack width check.

Required/provided reinforcement



User reinforcement



Theoretical background

<u>Crack appearance</u>

If condition below is satisfied no cracks will appear in the concrete.

$$\sigma_{ct,max\pm} \leq f_{ct,eff}$$

With:

$\sigma_{ct,max\pm} = \frac{n_{i\pm}}{A_{i,i\pm}} + \frac{m_{i\pm}}{Ii,i\pm} \cdot z_{t,max,i\pm}$ = Normal concrete stress on un-cracked section at the most tensioned fiber of concrete cross-section

$f_{ct,eff}$ = The mean value of the tensile strength of the concrete effective at the time

<u>Calculation of crack width</u>

$$w = \varepsilon_{sm\_cm} \cdot s_{r,max}$$

With:

$$(\varepsilon_{sm} - \varepsilon_{cm})_{i\pm} = \max\left[\frac{\sigma_{s,i\pm} - k_t \cdot \frac{f_{ct,eff}}{\rho_{p,eff,i\pm}} \cdot \left(1 + \alpha_{e,i\pm} \cdot \rho_{p,eff,i\pm}\right)}{E_{s,i\pm}}; 0{,}6 \cdot \frac{\sigma_{s,i\pm}}{E_{s,i\pm}}\right]$$

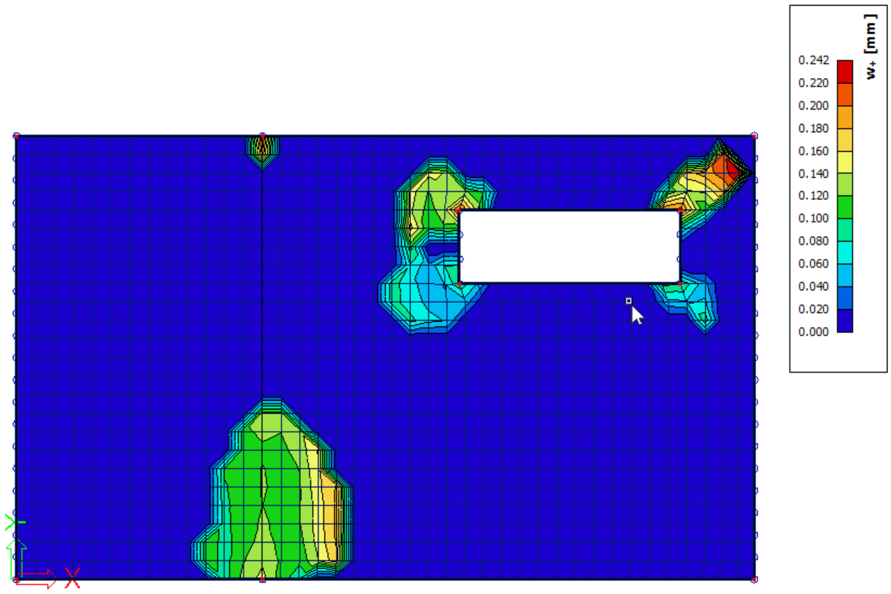
$$s_{r,max,i\pm} \begin{cases} \min\left(k_3 c_{i\pm} + \frac{k_{1,i\pm} k_{2,i\pm} k_4 d_{s,i\pm}}{\rho_{p,eff,i\pm}}; 1{,}3 \cdot (h - x_{i\pm})\right) & if\ s_{s,i\pm} \leq 5\left(c_{i\pm} + 0{,}5 d_{s,i\pm}\right) \\ 1{,}3 \cdot (h - x_{i\pm}) & if\ s_{s,i\pm} > 5\left(c_{i\pm} + 0{,}5 d_{s,i\pm}\right) \end{cases}$$

2_Results for required theoretical reinforcement

Concrete menu → Reinforcement check (ULS+SLS) → 2D members – Crack width (SLS)

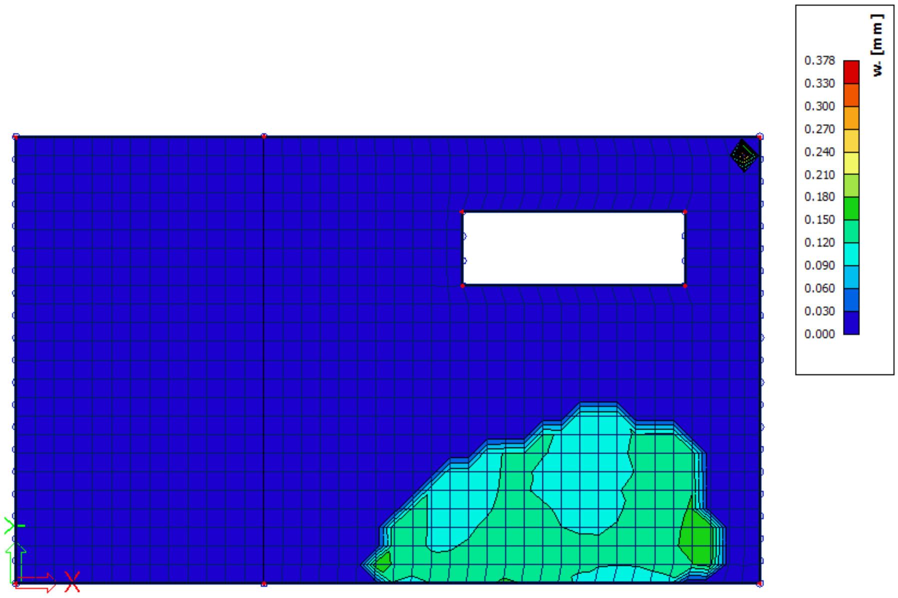
Crack width w+

Combination = SLS; Type of used reinforcement = Required; Value = **w+**



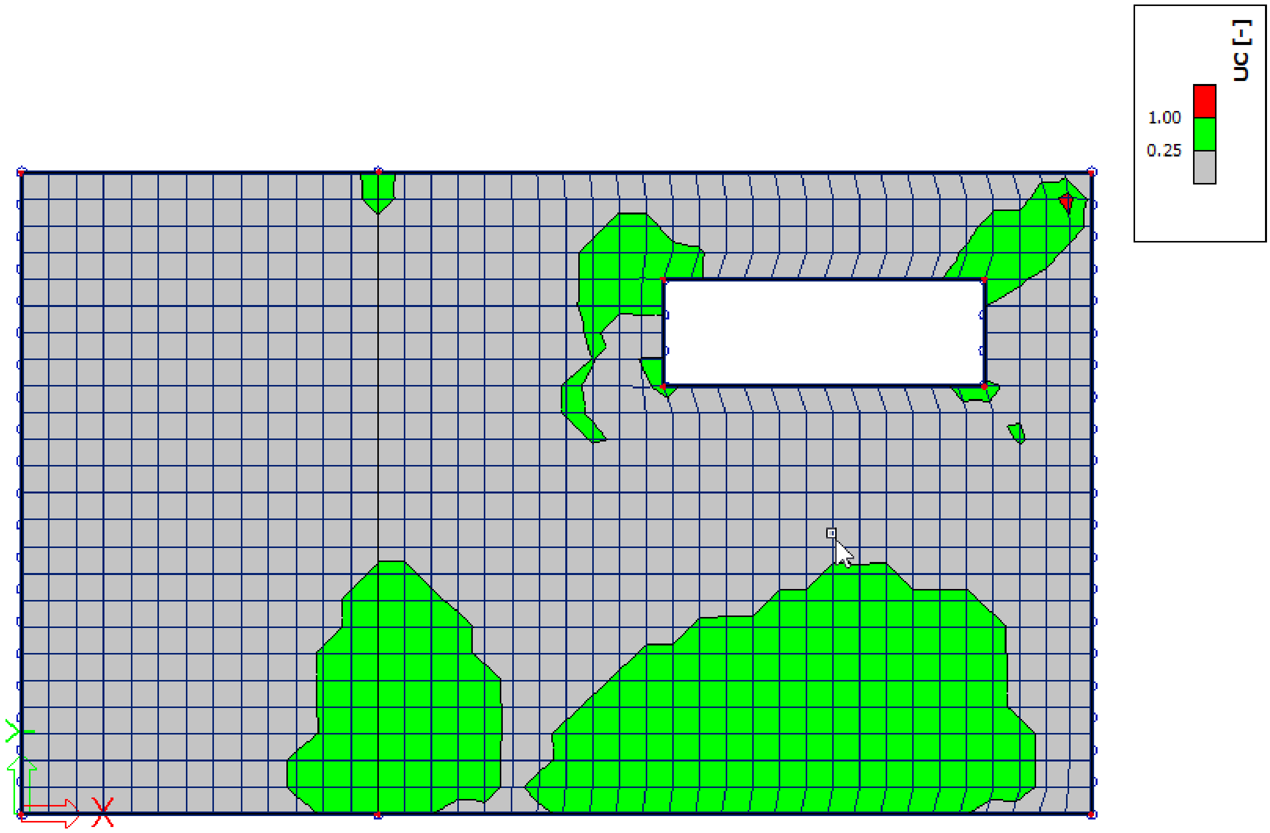
Crack width w-

Combination = SLS; Type of used reinforcement = Required; Value = **w-**



Unity check

Combination = SLS; Type of used reinforcement = Required; Value = **Check value**



A green value stands for a Unity check ≤ 1 (w_{calc} ≤ w_{max}), agrey value stands for Unity check ≤ 0.25 and a red value means that w_{max} is exceeded.

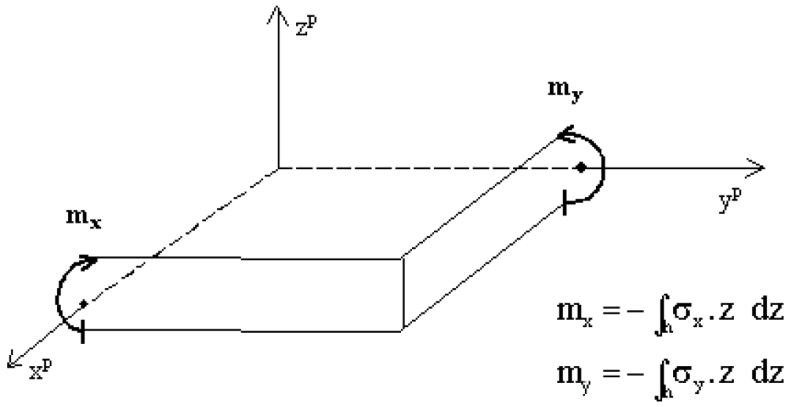
Annexes

Annex 1: Conventions for the results on 2D members

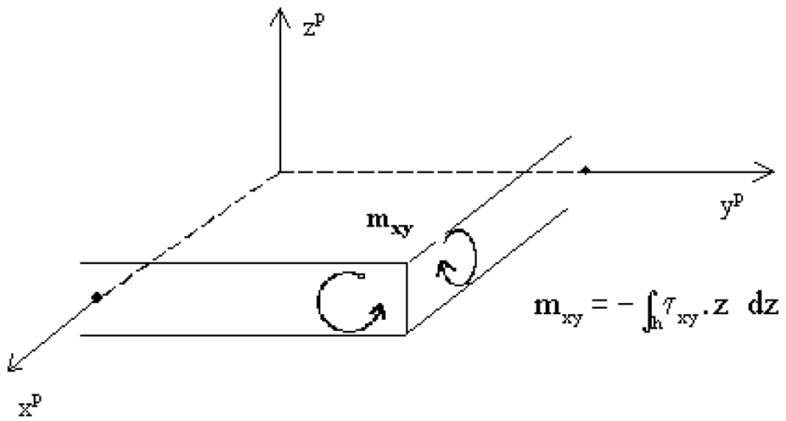
1_ Basic magnitudes = Characteristic values

Bending (plates, shells)

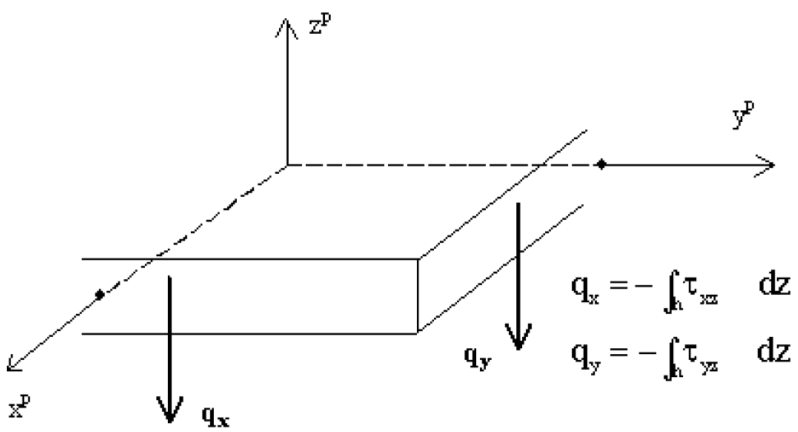
*Bending moments mx, my



*Torsion moment mxy

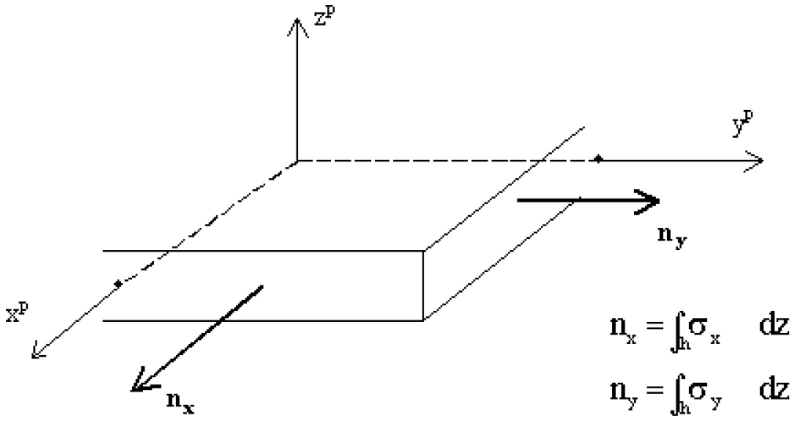


*Shear forces qx, qy (=vx, vy)

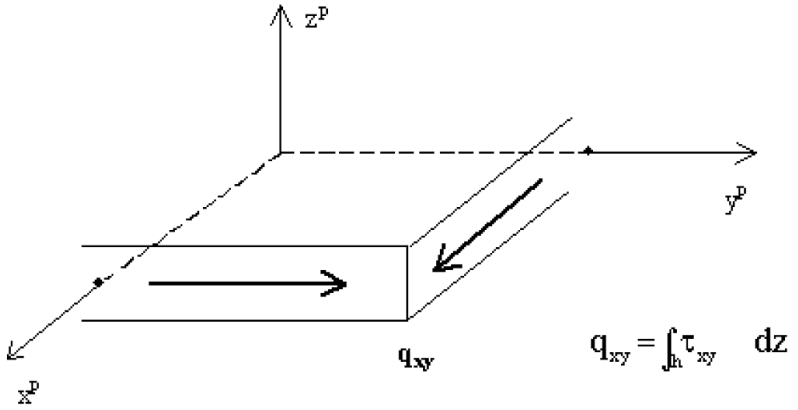


Membrane effects (walls, shells)

*Membrane forces nx, ny



*Shear forces qxy (=nxy)



2_Principal magnitudes

The principal magnitudes give the results according to the axes of the directions of the largest stresses (principal directions). These directions are defined with the help of the circle of Mohr.

3_Design magnitudes

To derive the design magnitudes from the basic magnitudes, formulas from the Eurocode EC-ENV are used.
See also the Help menu > Contents > Reference guide, for these formulas.

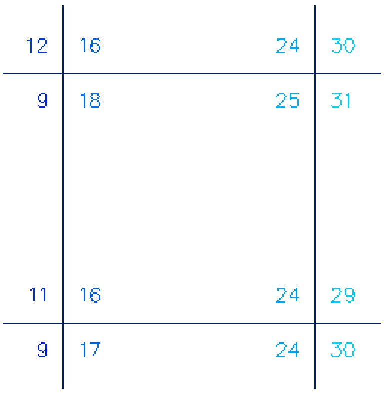
Annex 2: Results in Mesh elements and Mesh nodes → 4 Locations

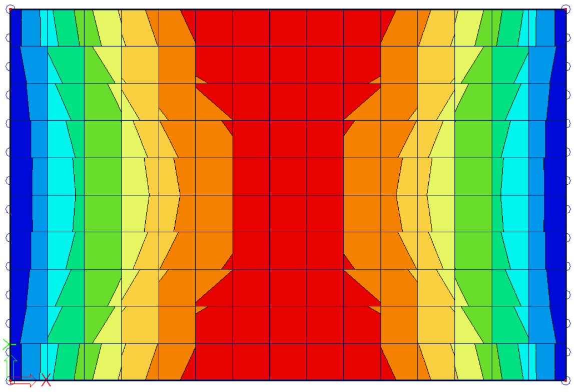
During a calculation in SCIA Engineer, the node deformations and the reactions are calculated exactly (by means of the displacement method). The stresses and internal forces are derived from these magnitudes by means of the assumed basic functions, and are therefore in the Finite Elements Method always less accurate.

The Finite Elements Mesh in SCIA Engineer exists of linear 3- and/or 4-angular elements. Per mesh element 3 or 4 results are calculated, one in each node. When asking the results on 2D members, the option ‘Location’ in the Properties window gives the possibility to display these results in 4 ways.

1_ In nodes, no average

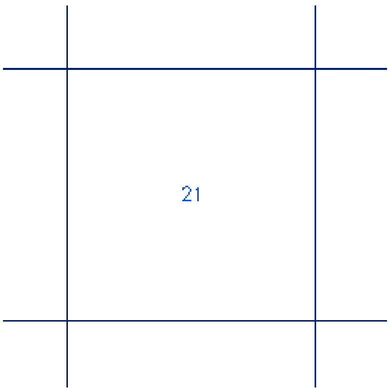
All of the values of the results are taken into account, there is no averaging. In each node are therefore the 4 values of the adjacent mesh elements shown. If these 4 results differ a lot from each other, it is an indication that the chosen mesh size is too large.
This display of results therefore gives a good idea of the discretisation error in the calculation model.

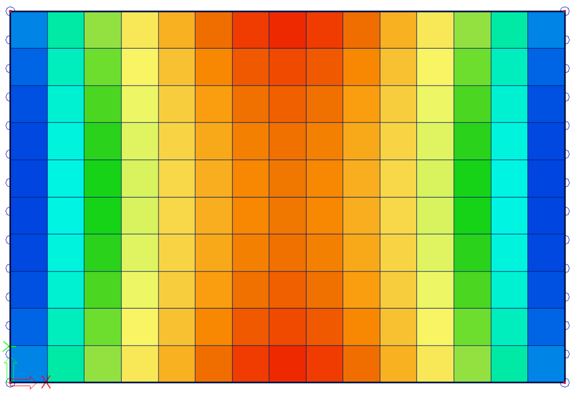




2_ In centres

Per finite element, the mean value of the results in the nodes of that element is calculated. Since there is only 1 result per element, the display of isobands becomes a mosaic. The course over a section is a curve with a constant step per mesh element.



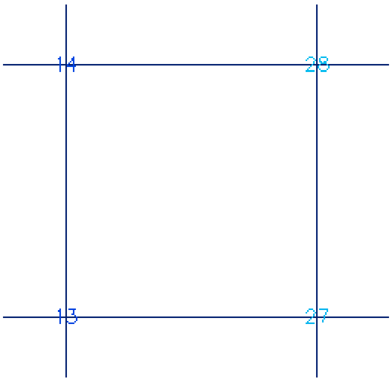


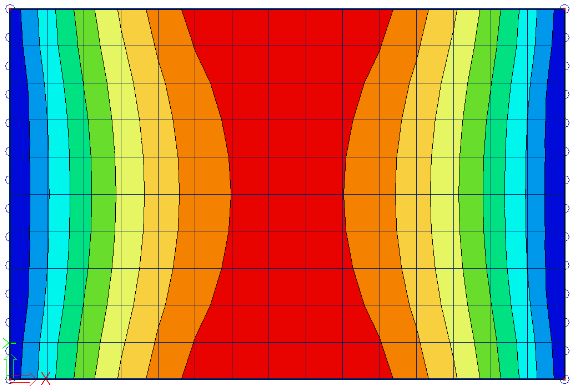
3_ In nodes, average

The values of the results of adjacent finite elements are averaged in the common node. Because of this, the graphical display is a smooth course of isobands.

In certain cases, it is not permissible to average the values of the results in the common node:
- At the transition between 2D members (plates, walls, shells) with different local axes.

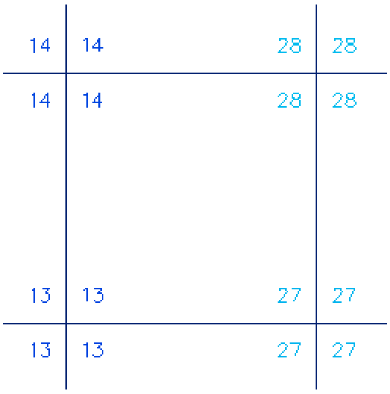
- If a result is really discontinuous, like the shear force at the place of a line support in a plate. The peaks will disappear completely by the averaging of positive and negative shear forces.

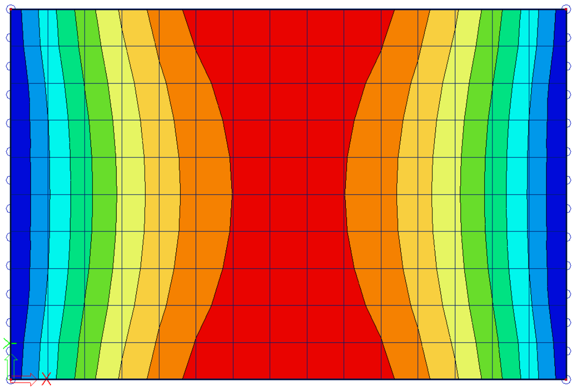




4_ In nodes, average on macro

The values of the results are averaged per node *only* over mesh elements which belong to the same 2D member ánd which have the same directions of their local axes. This resolves the problems mentioned at the option 'In nodes, average'.





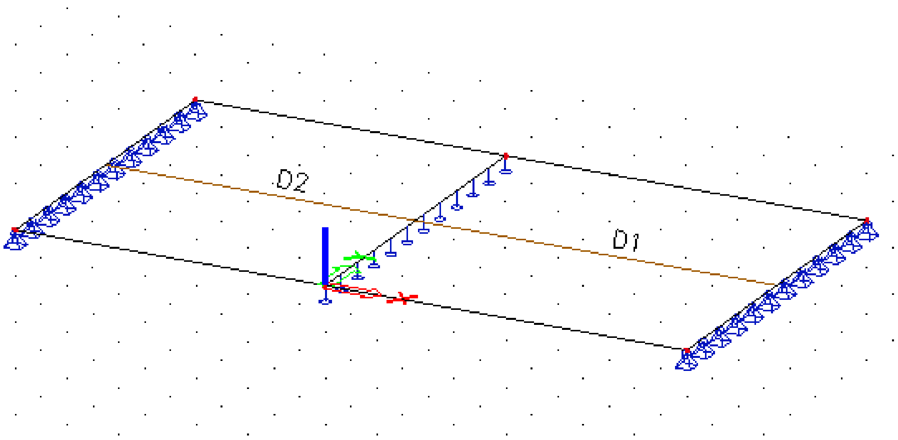
Accuracy of the results

If the results according to the 4 locations differ a lot, then the results are inaccurate and this means the finite element mesh has to be refined. A basic rule for a good size of the 2D mesh elements, is to take 1 to 2 times the thickness of the plates in the project.

Annex 3: Local Coordinate system for 2D members

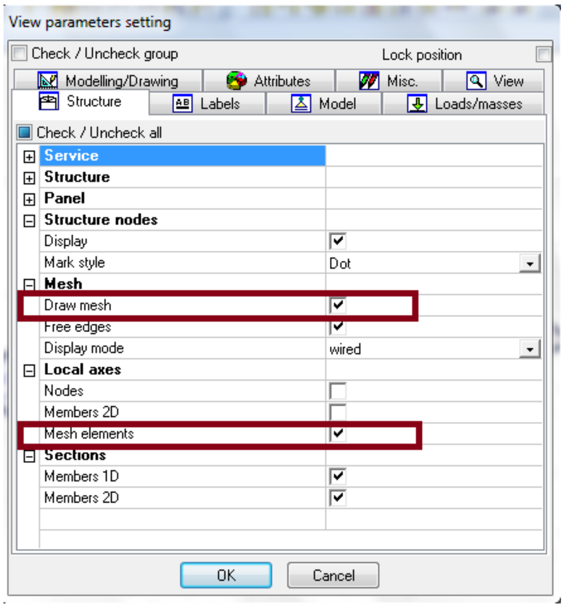
The results for a Finite Element analysis are computed according to the Local Coordinate System (LCS) of each mesh element. As a consequence, these results depend on the way the local axes for mesh elements are defined. A wrong definition of local axes can lead to very misleading results.

Let's consider the example below. A continuous plate is modeled as two elements D1 and D2:

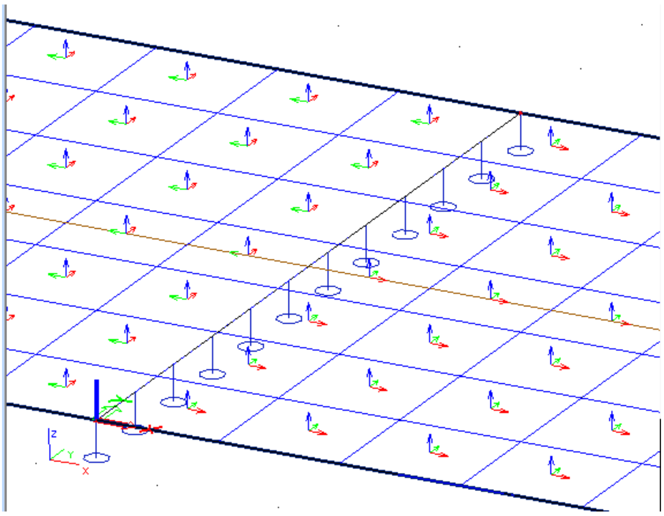


In order to display the mesh local axes, you need first to generate the mesh. You can use the button 'Mesh generation'  or **Main menu > Calculation, Mesh > Mesh generation**

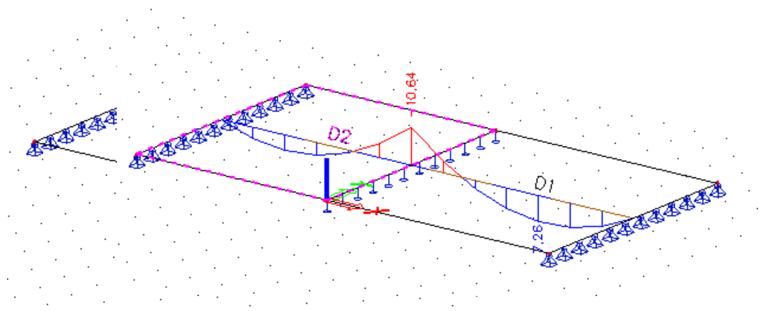
Afterwards, the mesh and local axes can be displayed from '**Set View parameters for all**': right click of the mouse on the screen



It is clear from the orientation of the axes that the continuity of the moments mx and my cannot be satisfied. The moment mx on D1 corresponds in this case to the moment my in D2.

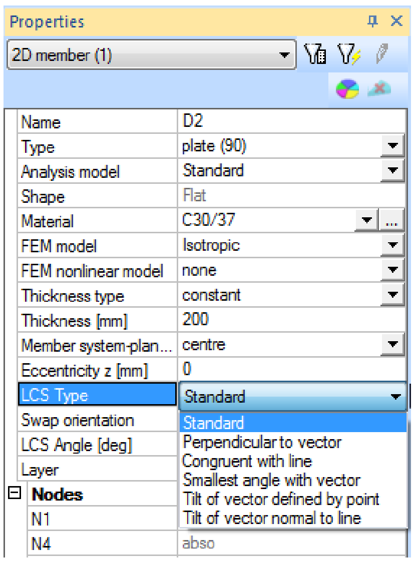


The moment mx on a section of the plate gives the Moment diagram1. After correction of the local axes orientation, Moment diagram2 is obtained.



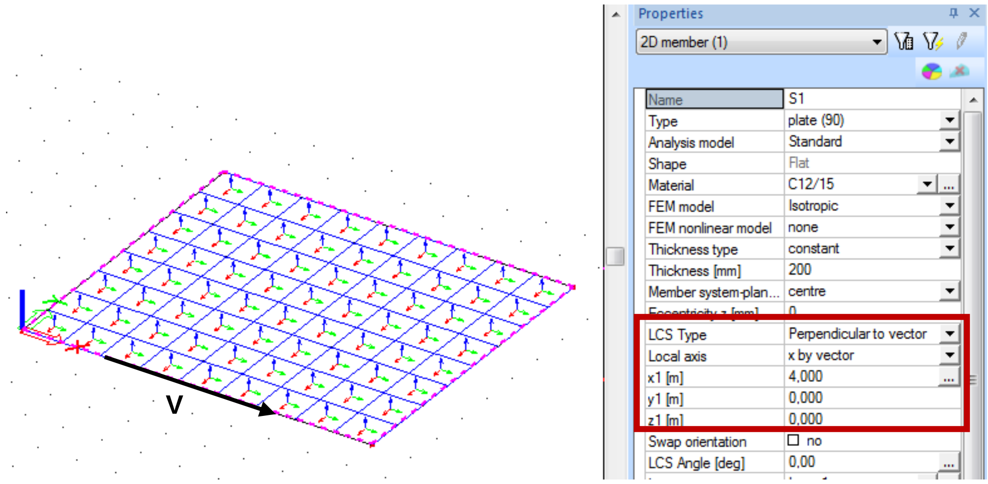
Moment diagram 1 **Moment diagram 2**

By default, the program computes the local axes automatically. The user can adapt the direction of the axes in several ways using the Properties menu:



1_ Perpendicular to vector

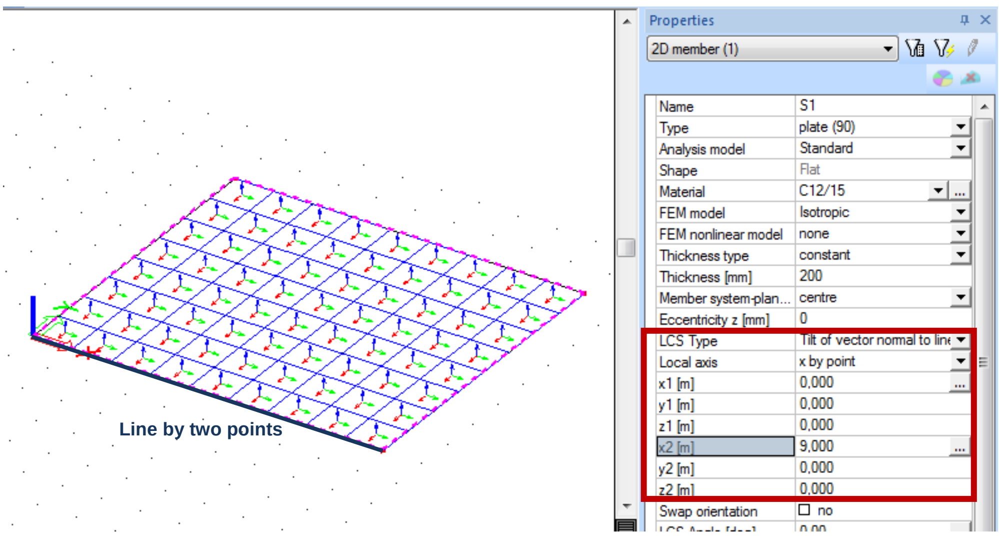
The local axis x(y) is perpendicular to a vector that is defined with its coordinates V(x1 y1 z1).



The coordinates of the vector can be introduced in the properties window. You can also use the ... button next to x1 and define the vector graphically with a start point and end point.

2_ Tilt of vector normal to line

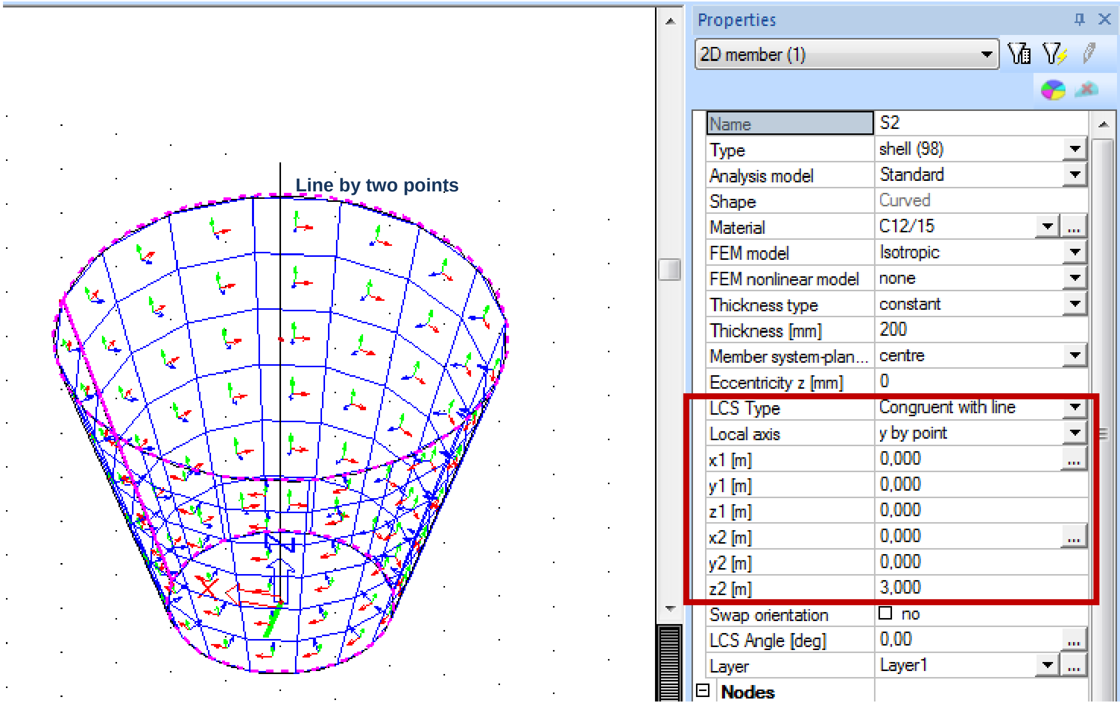
This method is similar to the one above. Instead of introducing a vector, a line is defined between two points. (x1 y1 z1) are the coordinates of the first point and (x2 y2 z2) are the coordinates of the second point. The x(y) axis is perpendicular to the introduce line and points towards it.



The coordinates of the points that define the line can be introduced in the properties window. You can also click on the ... button next to x1 and x2 and define these points graphically.

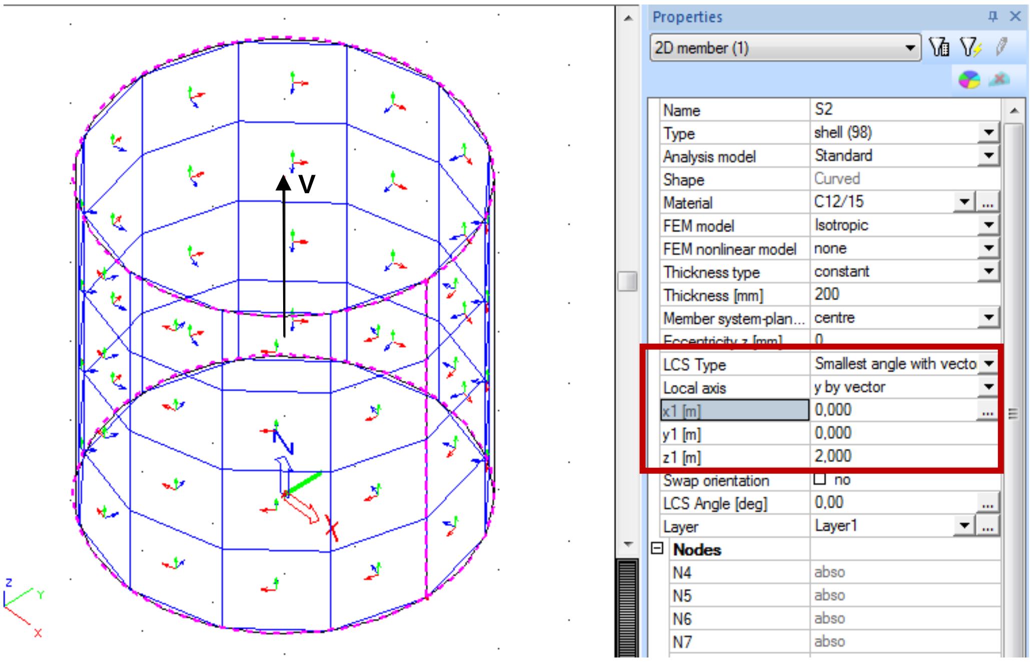
3_ Congruent with line

x(y) is oriented from the center of the element towards the intersection between the mesh element and the defined line :



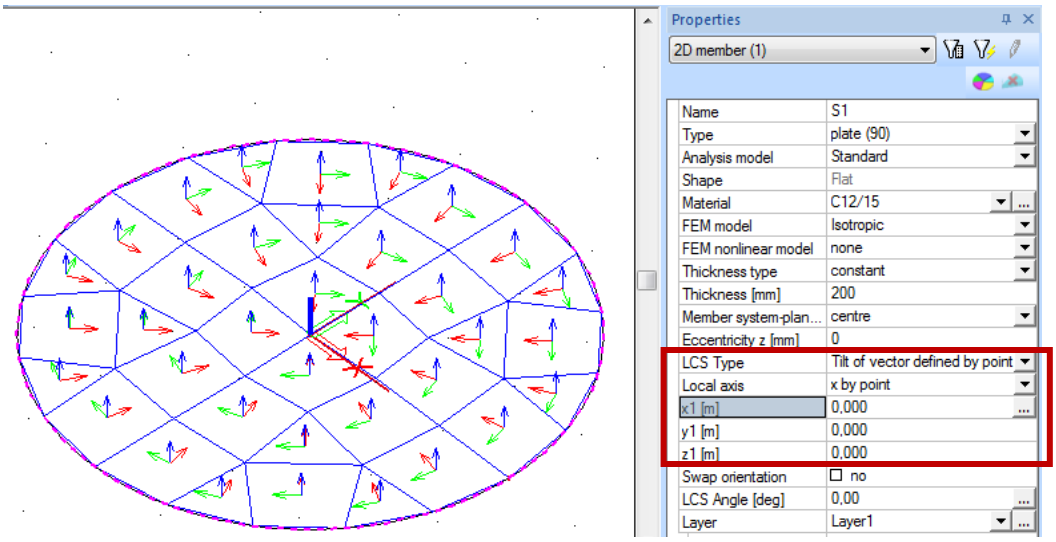
4_ Smallest angle with vector

x(y) is oriented such that it makes the smallest angle with the defined vector V(x1 y1 z1).

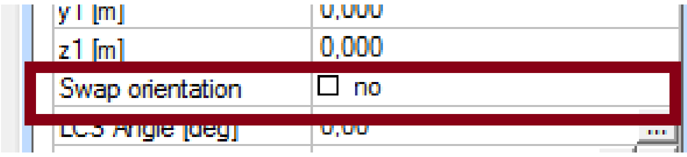


5_ Tilt of vector defined by point

x(y) is oriented towards a defined point (x1 y1 z1). This is suitable in case of a circular plates for example and allows the user to calculate radial reinforcement.



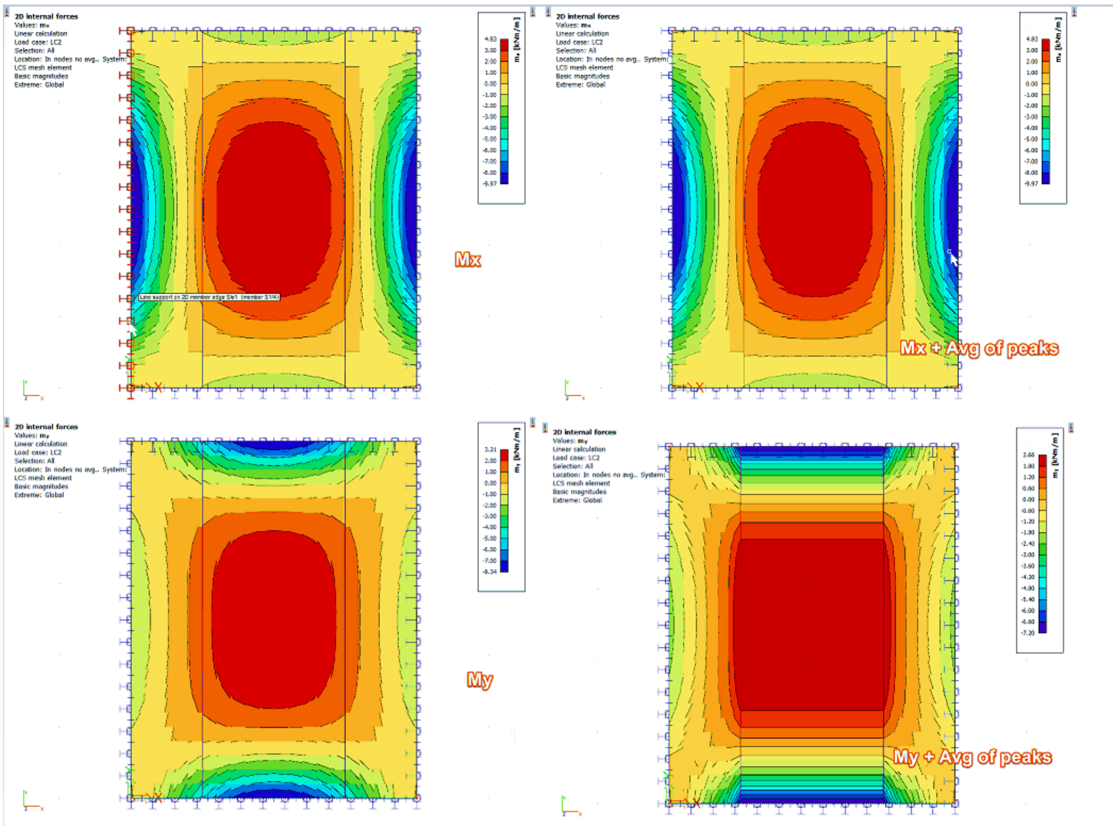
Remark: All the methods above are about how to adapt the x and y axes. The local z axis is defined automatically by the program but its orientation can be changed by ticking the box 'Swap orientation' in the Properties menu



Annex 4: Averaging strips

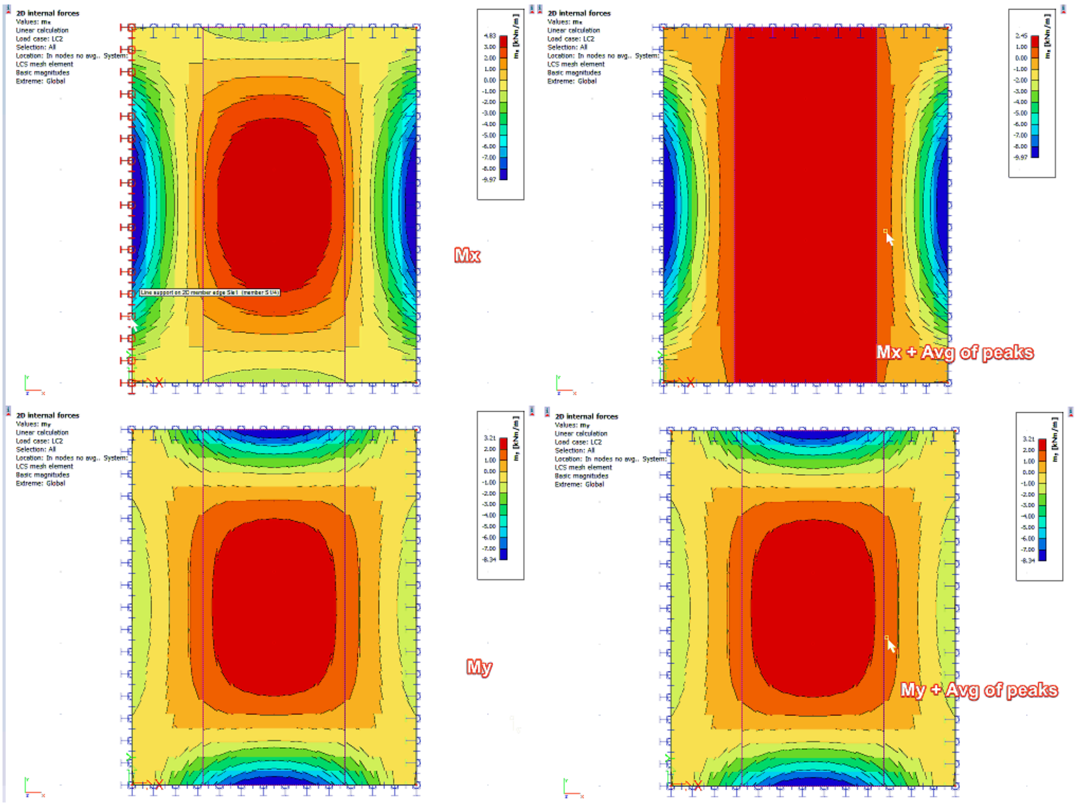
Averaging strips average the results perpendicular to the direction of the internal force. The option ‘Direction’ in the settings of the averaging strip is dependent on how the averaging strip is drawn.

1) Direction = Longitudinal



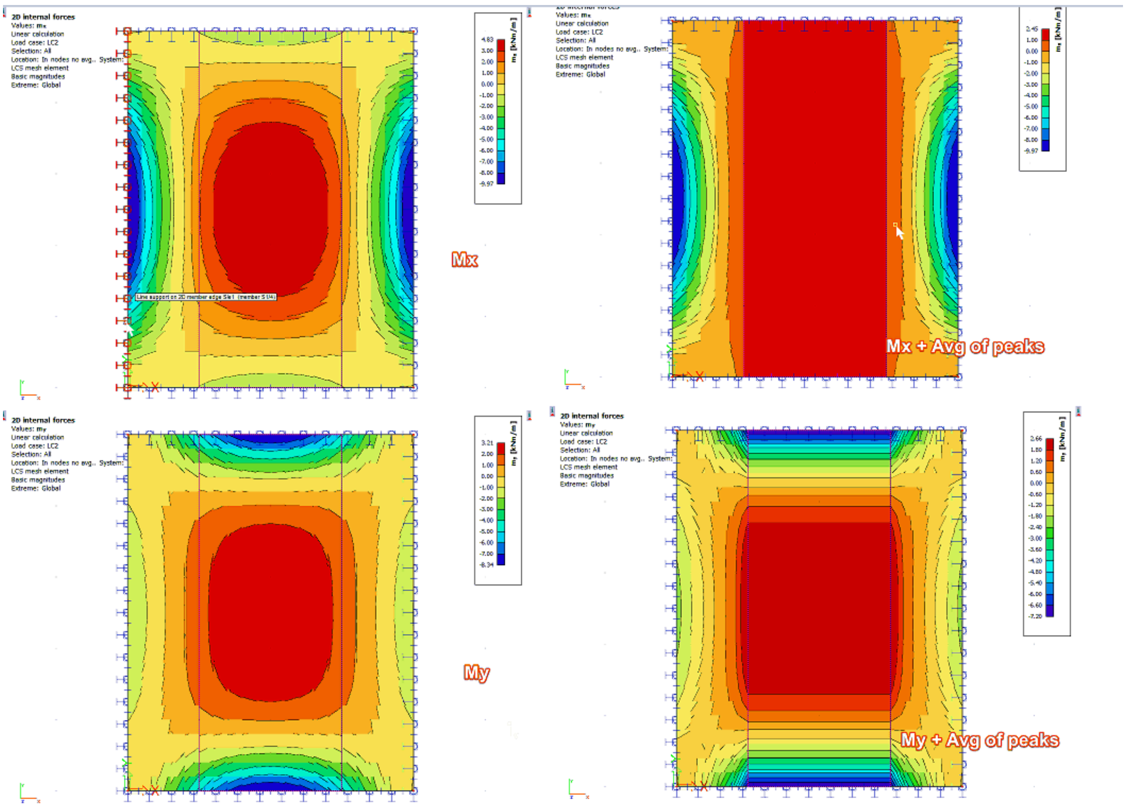
Longitudinal means that the averaging is done in the longitudinal direction of the strip. In the example above this is the y-direction. This means that the averaging is done for my. The values my are averaged in the x-direction.

2) Direction = perpendicular



Perpendicular means that the averaging is done perpendicular to the longitudinal direction of the strip. In the example above this is the x-direction. This means that the averaging is done for mx. The values mx are averaged in the y-direction.

3) Direction = Both



Both means that the averaging is done in both directions of the averaging strip. This means the values are averaged for as well mx as my in the direction perpendicular to mx and my.