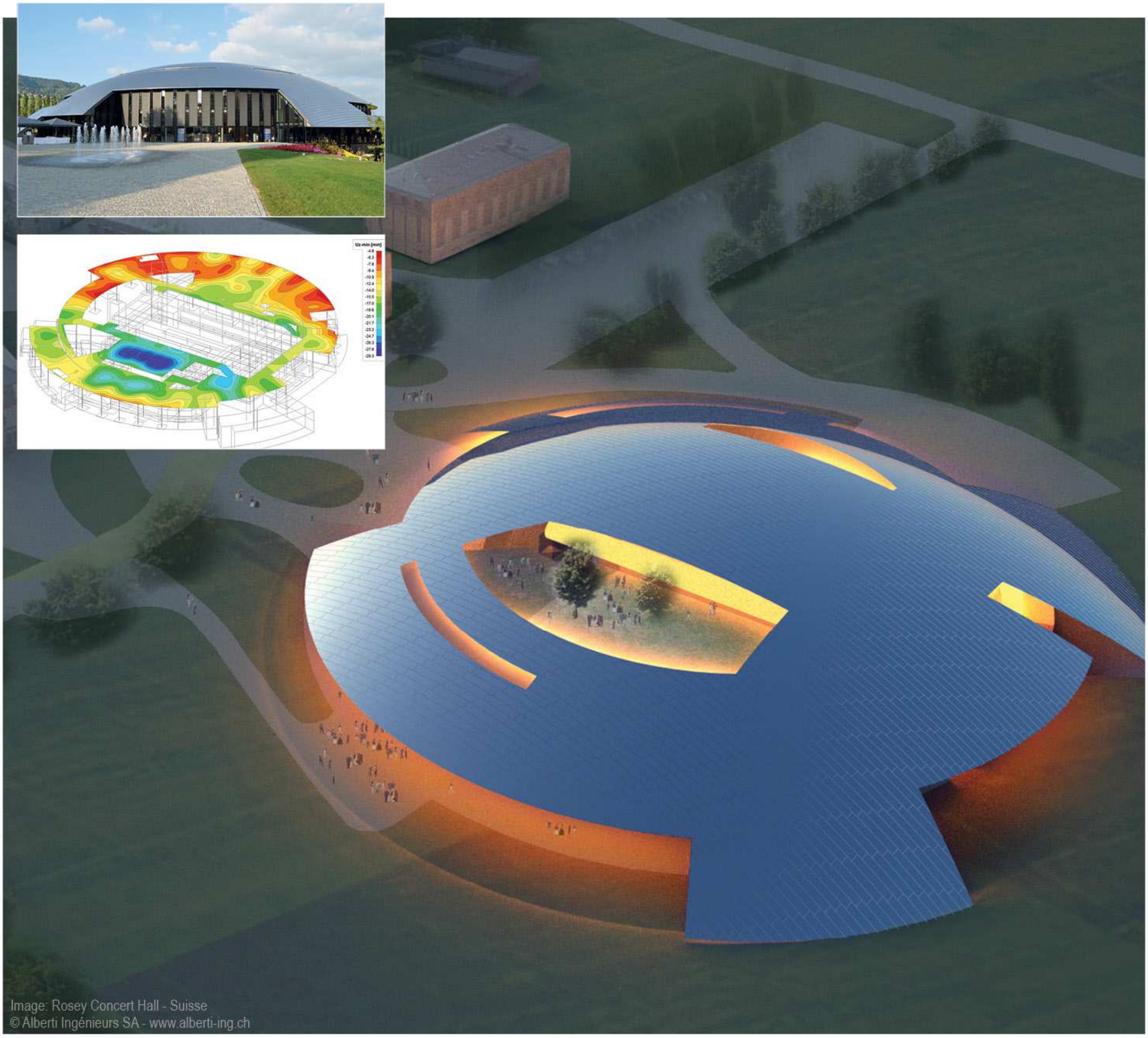
SCIA ENGINEER



Advanced Concept Training

Reinforced concrete (EN 1992) – 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**

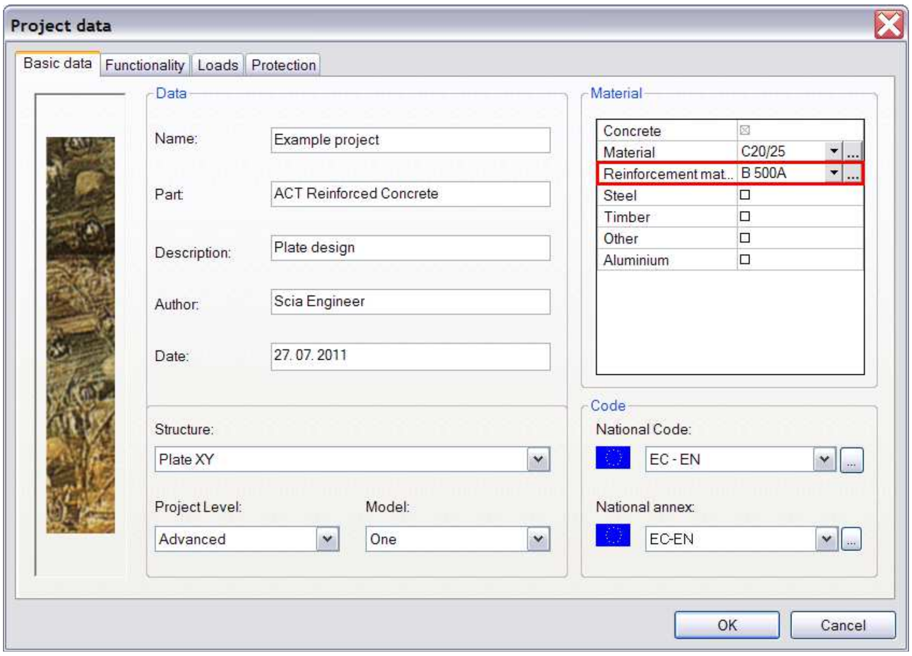
  esacdt.03 (2D members) **Concept edition**

Plate design

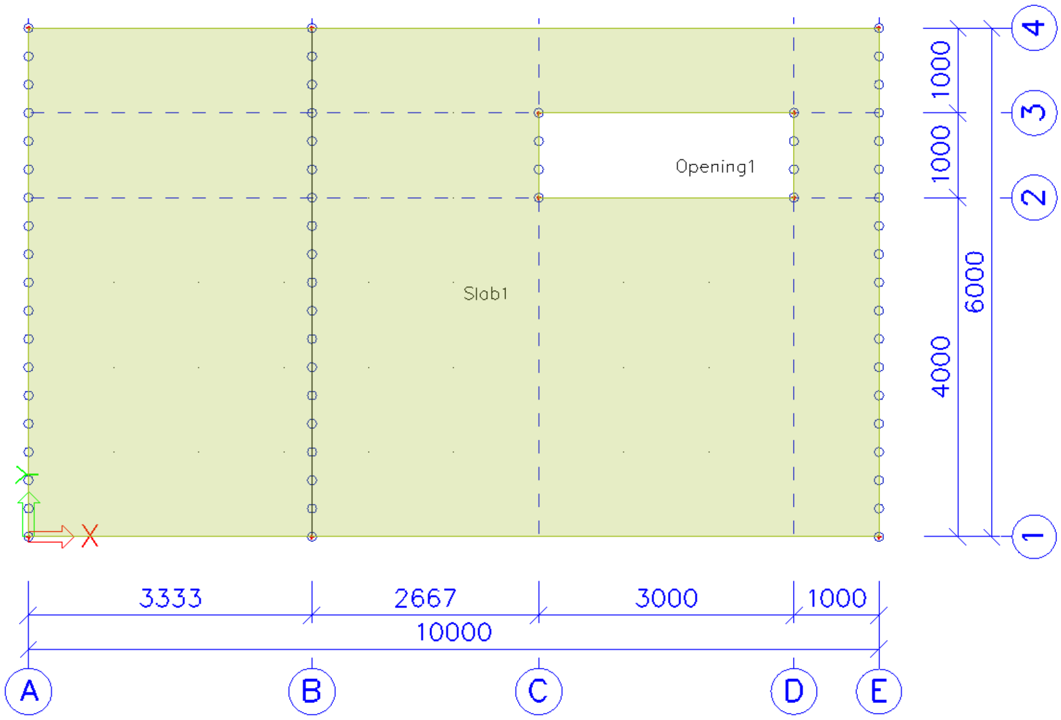
Model

1_Input of geometry

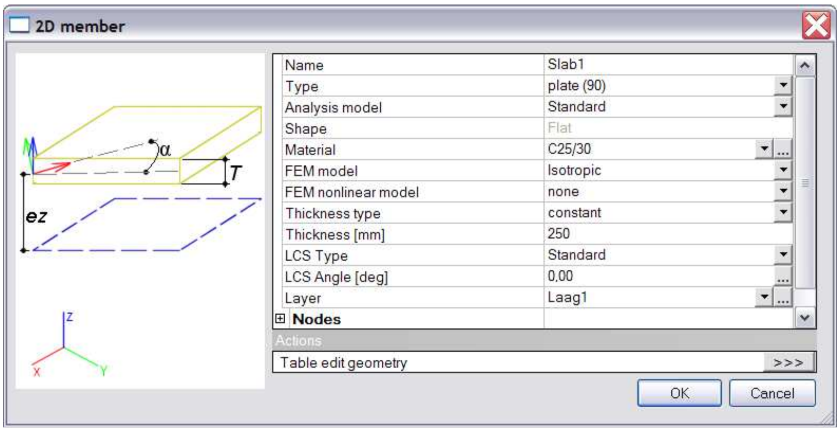
Project data: 2D environment = Plate XY

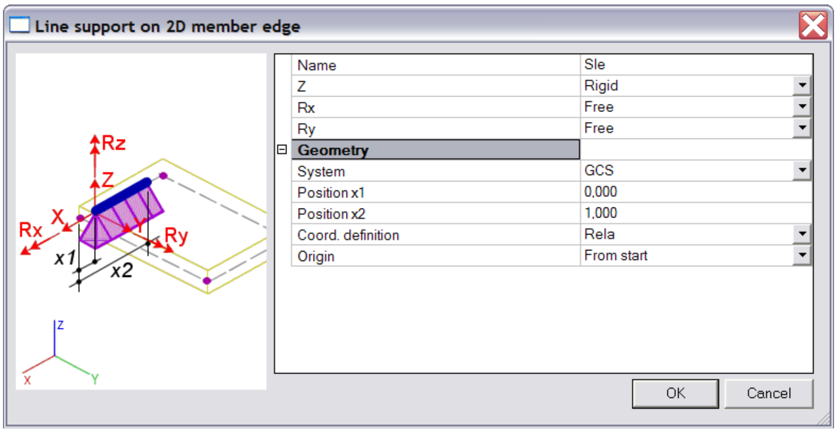


The Reinforcement material (e.g. B500A) chosen in the Project data window, will define the steel quality used for the theoretical reinforcement design.



Properties of the slab and the line supports:

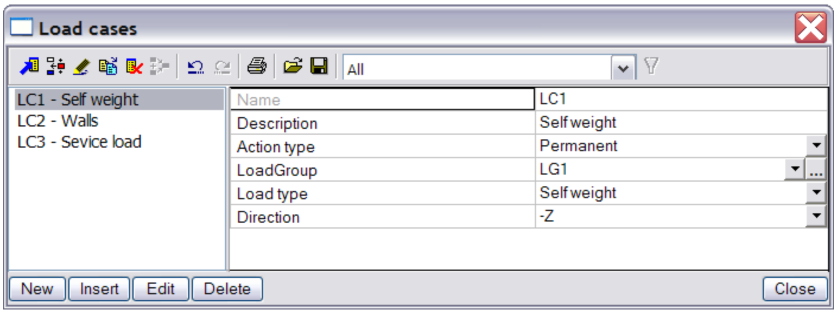


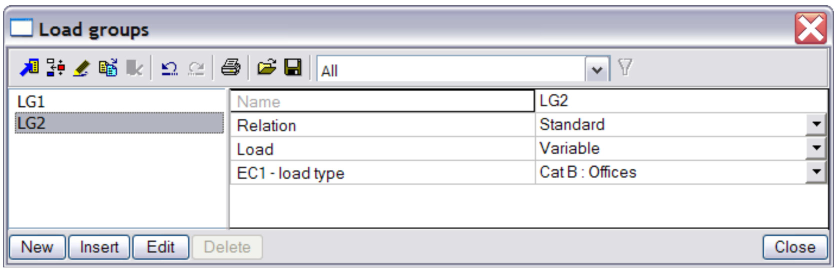


2_Loads

Load cases & Load groups

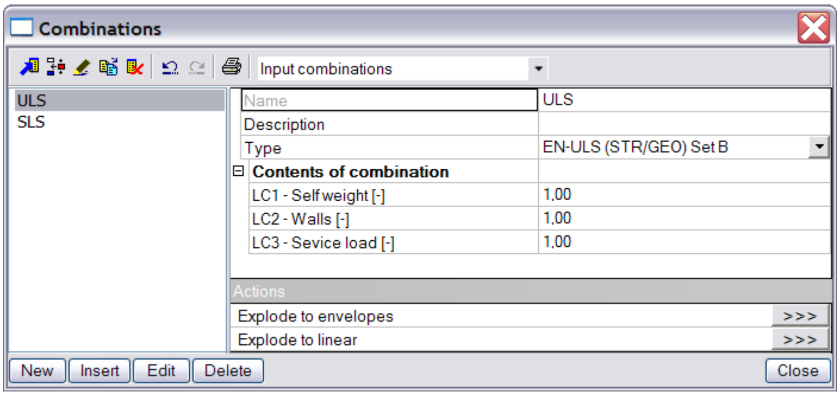
Load Case	*Action type*	*Load Group*	*Relation*	*EC1-Load type*
Self weight	Permanent	LG1	/	/
Walls	Permanent	LG1	/	/
Service load	Variable	LG2	Standard	Cat B: Offices





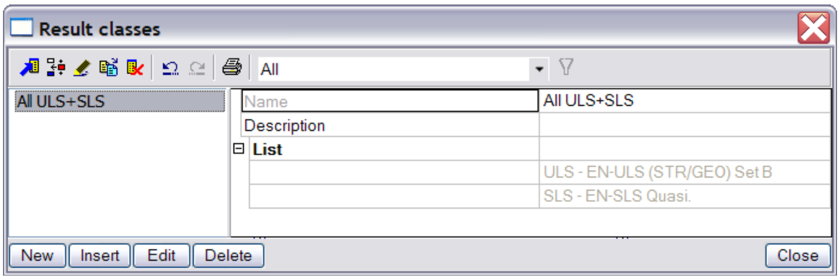
Load combinations

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



Result classes

All ULS+SLS

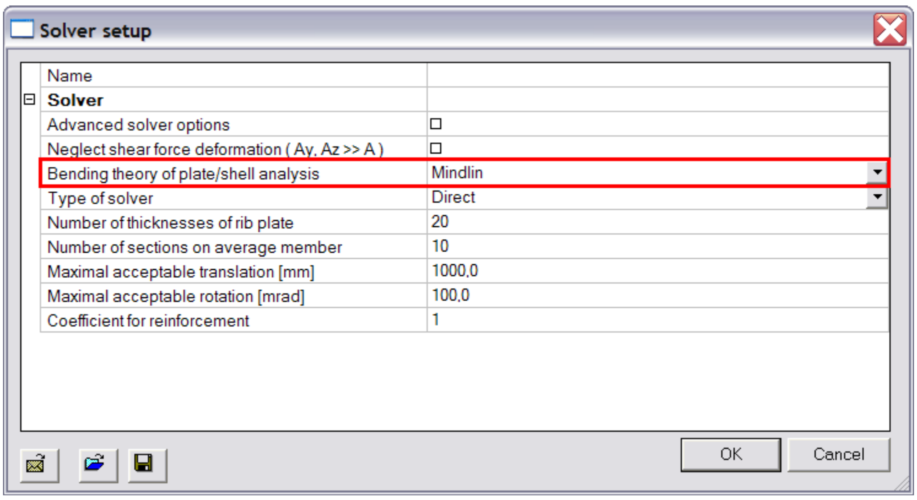


3_Finite element mesh

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

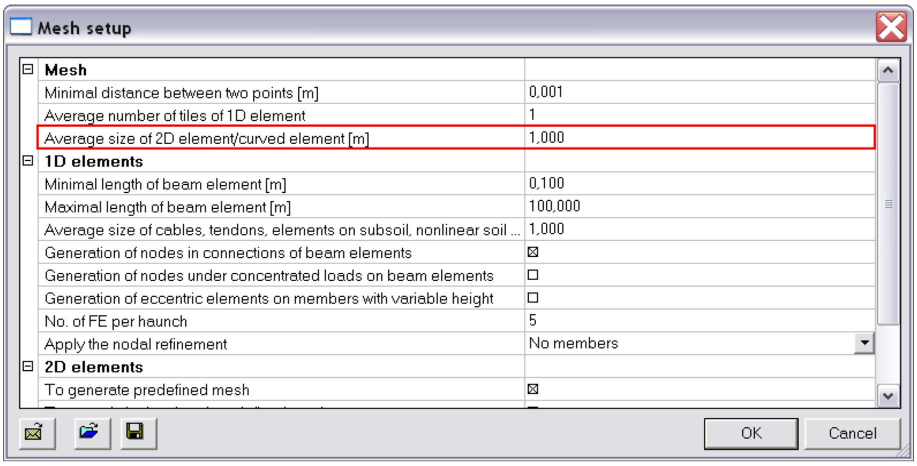
Via the Main menu > Calculation, mesh > Mesh generation, or ‘Project’ toolbar 

Graphical display of the mesh

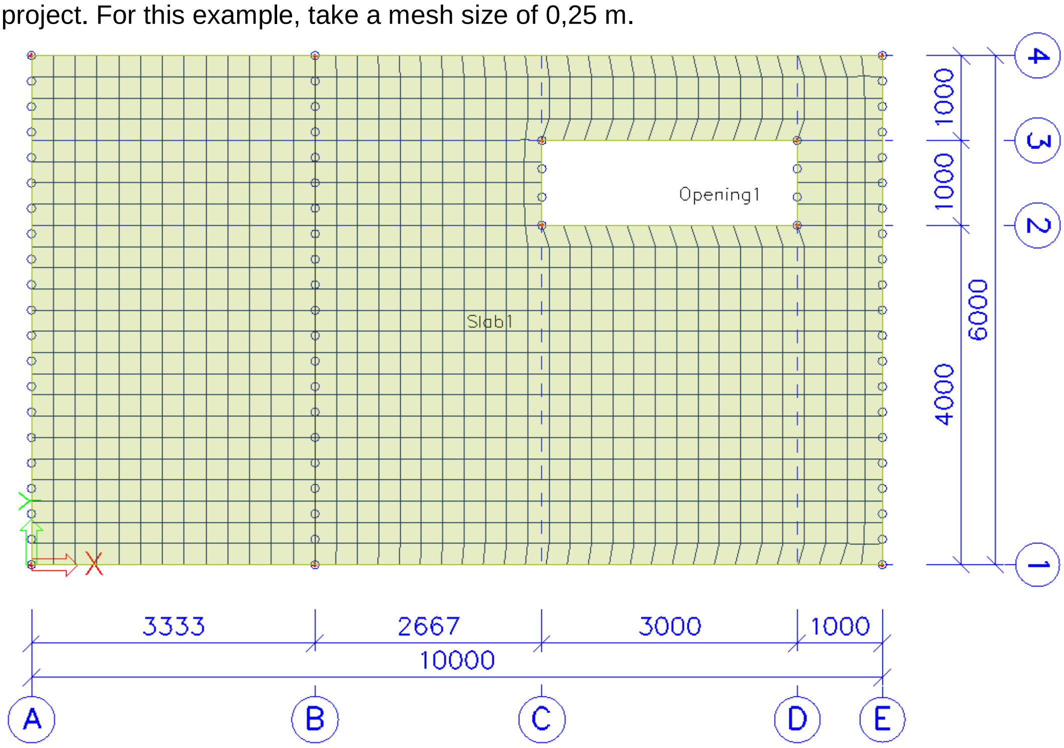
Set view parameters for all, via right mouse click in screen or Command line toolbar 
- Structure tab > Mesh > Draw mesh
- Labels tab > Mesh > Display label

Mesh refinement

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



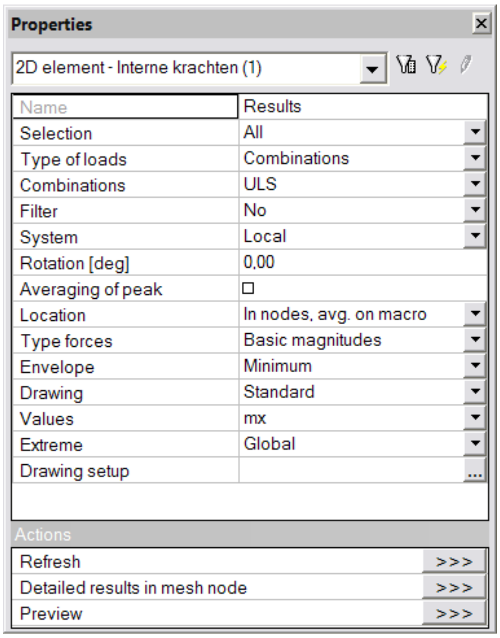
‘Basic rule’ for the size of 2D mesh elements: take 1 to 2 times the thickness of the plates in the project. For this example, take a mesh size of 0,25 m.



4_Results for the linear calculation

Specification of results

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



-System

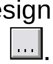
Local: according to the local axes of the *individual* mesh elements

LCS-Member 2D: according to the LCS of the 2D member

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

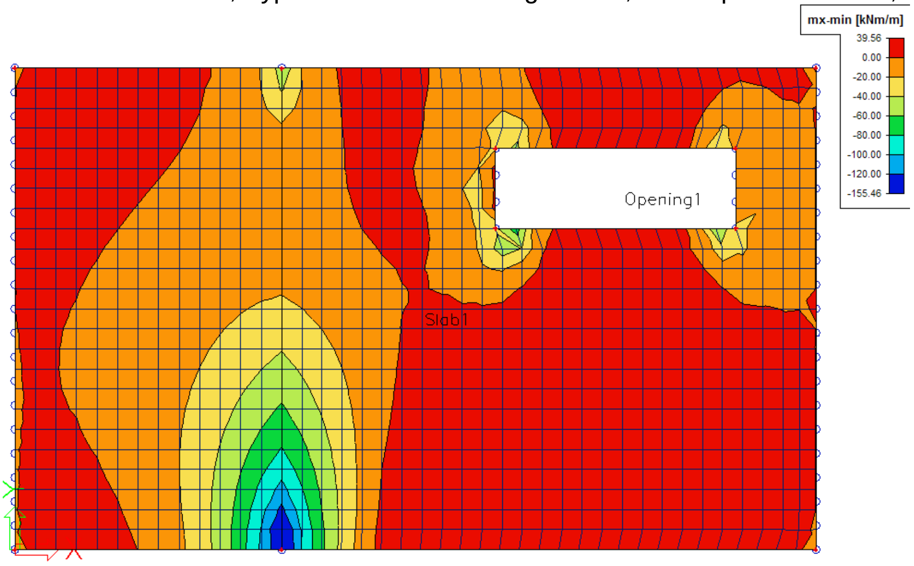
-Drawing setup: Click on the button . Here you can modify the display of 2D results (Isobands / Isolines / Numerical results / ...), modify the minimum and maximum settings, ...

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

Types of results

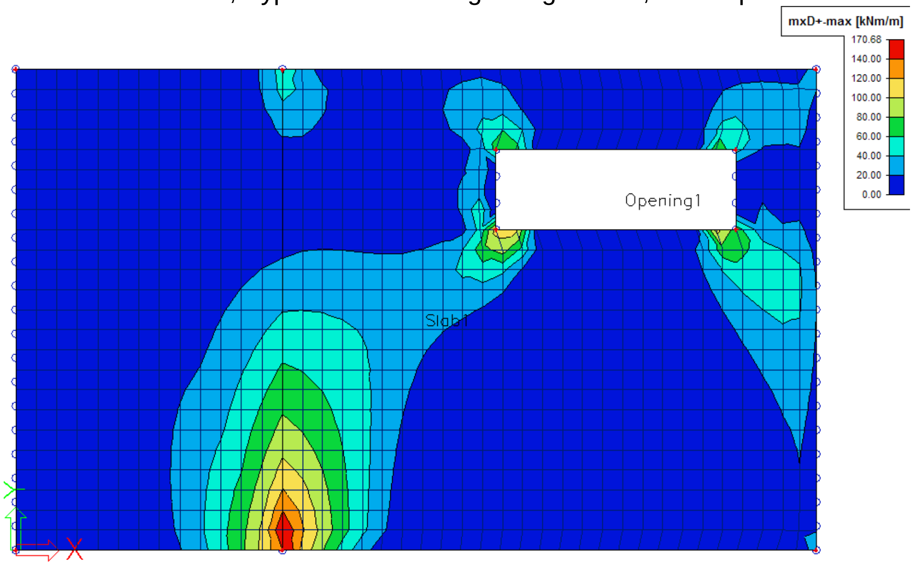
Basic magnitudes

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



Design magnitudes

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



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.
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.
An overview can be found in SCIA Engineer’s Help menu > Contents > Reference guide.

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.

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

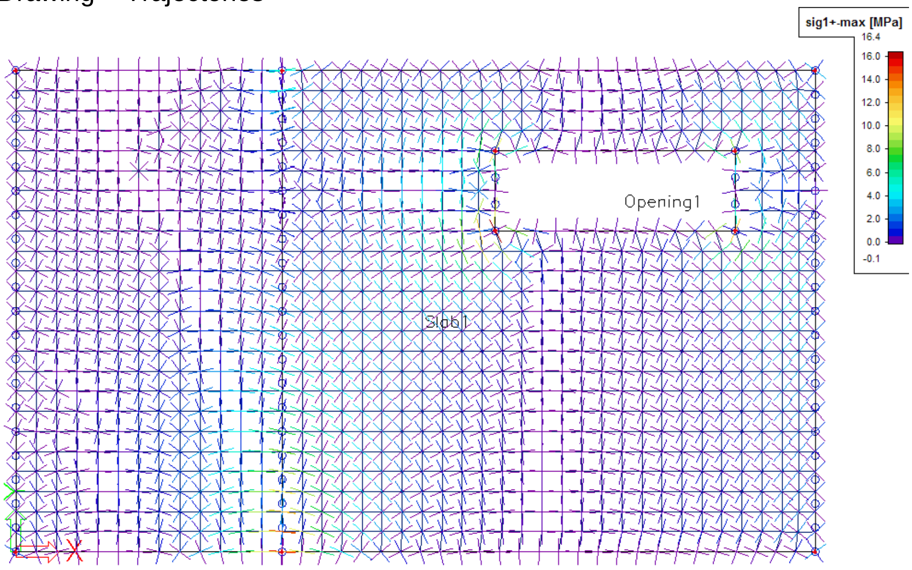
nx		***nxD***
ny	≈	nyD
nxy		ncD

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.

Attention: 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 > Member 2D – Stresses
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).

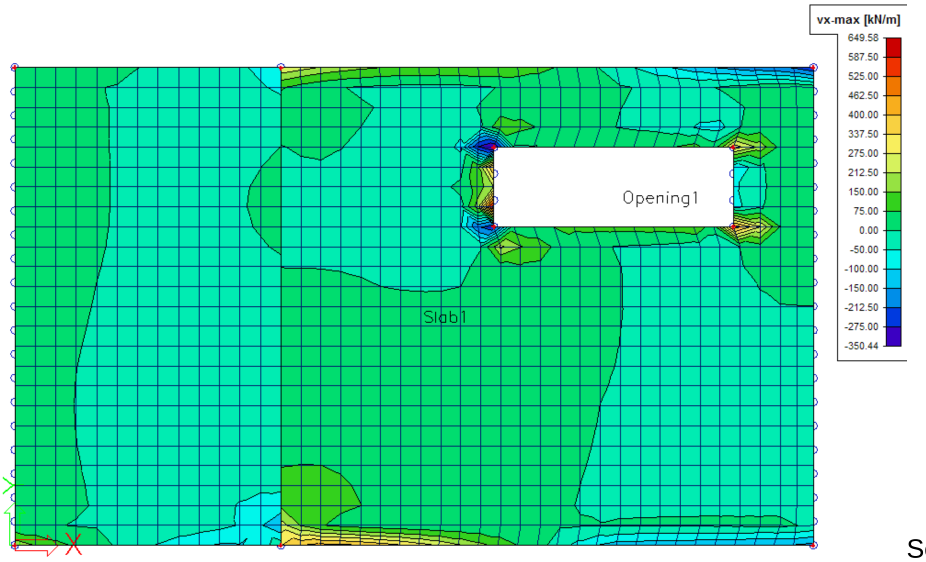
Keep in mind that the most economic reinforcement paths are the ones that follow the trajectories of the principal directions!

Comparison Mindlin ↔ Kirchhoff

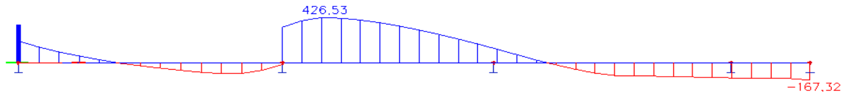
Shear force vx

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

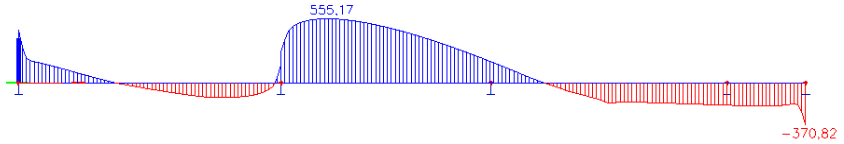
Mindlin



Section at lower edge

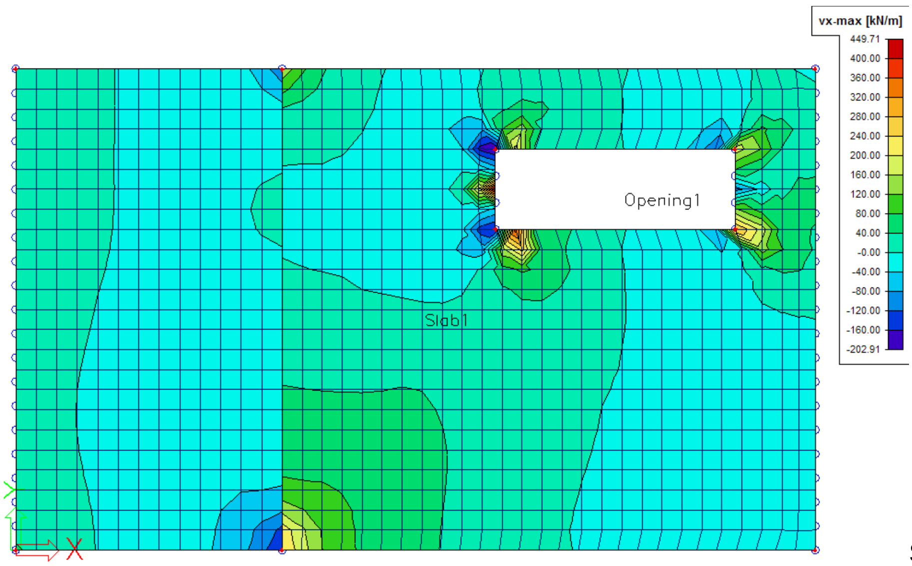


Mesh size = 0,25 m

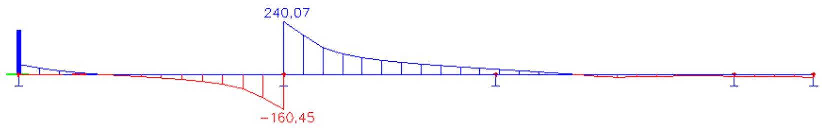


Mesh size = 0,05 m

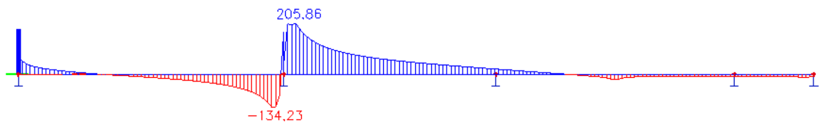
Kirchhoff



Section at lower edge



Mesh size = 0,25 m

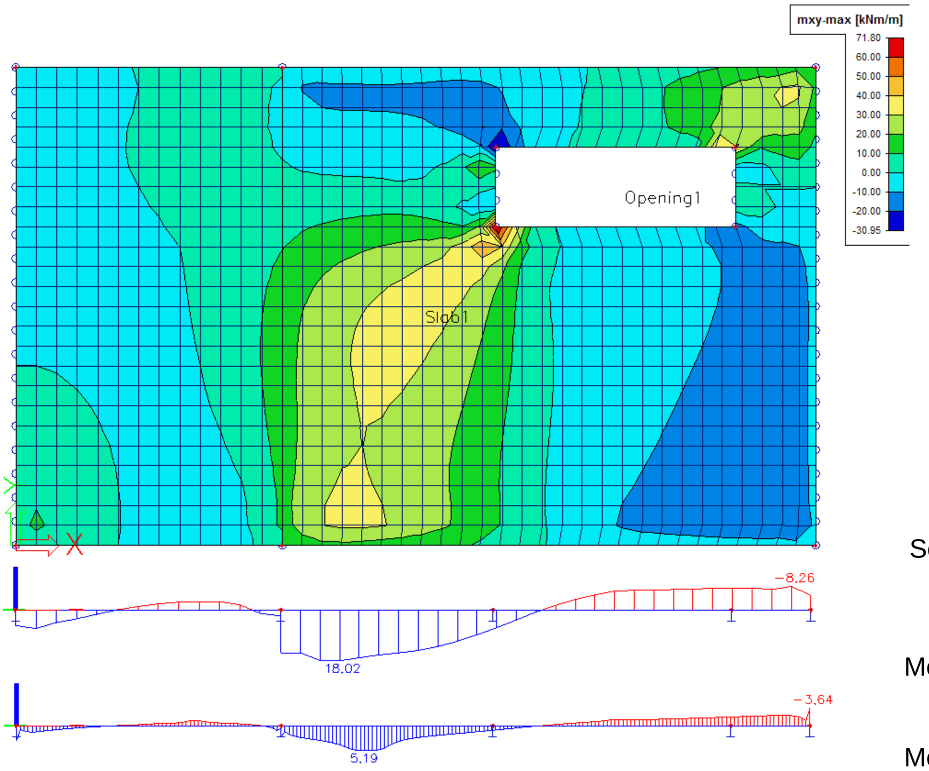


Mesh size = 0,05 m

Torsion moment mxy

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

Mindlin

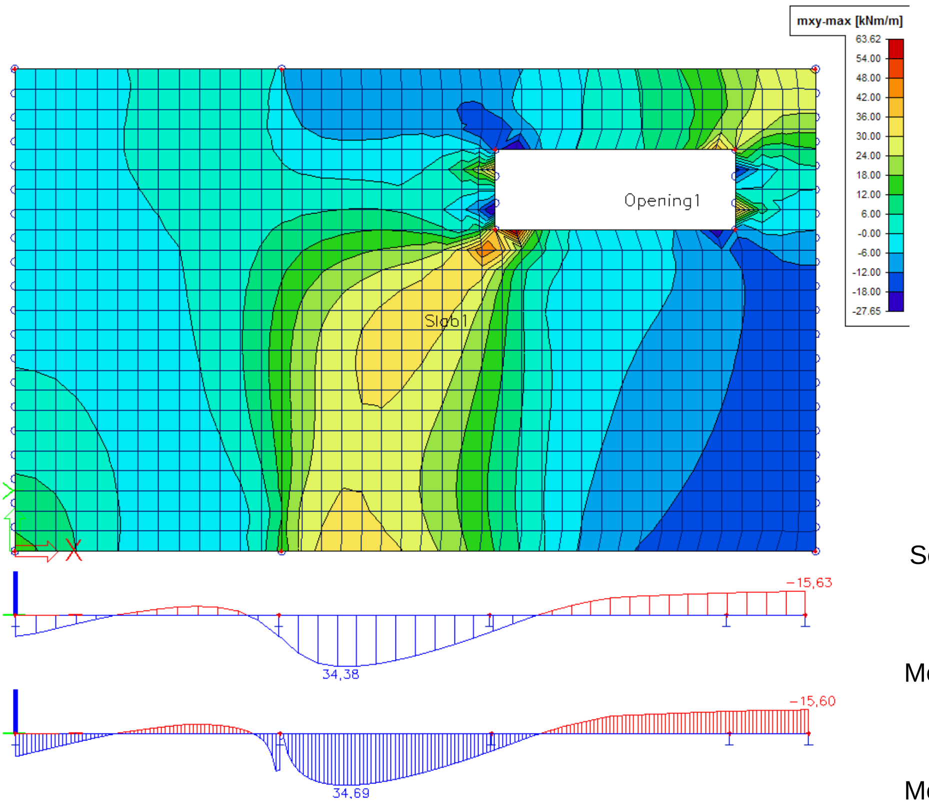


Section at lower edge

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

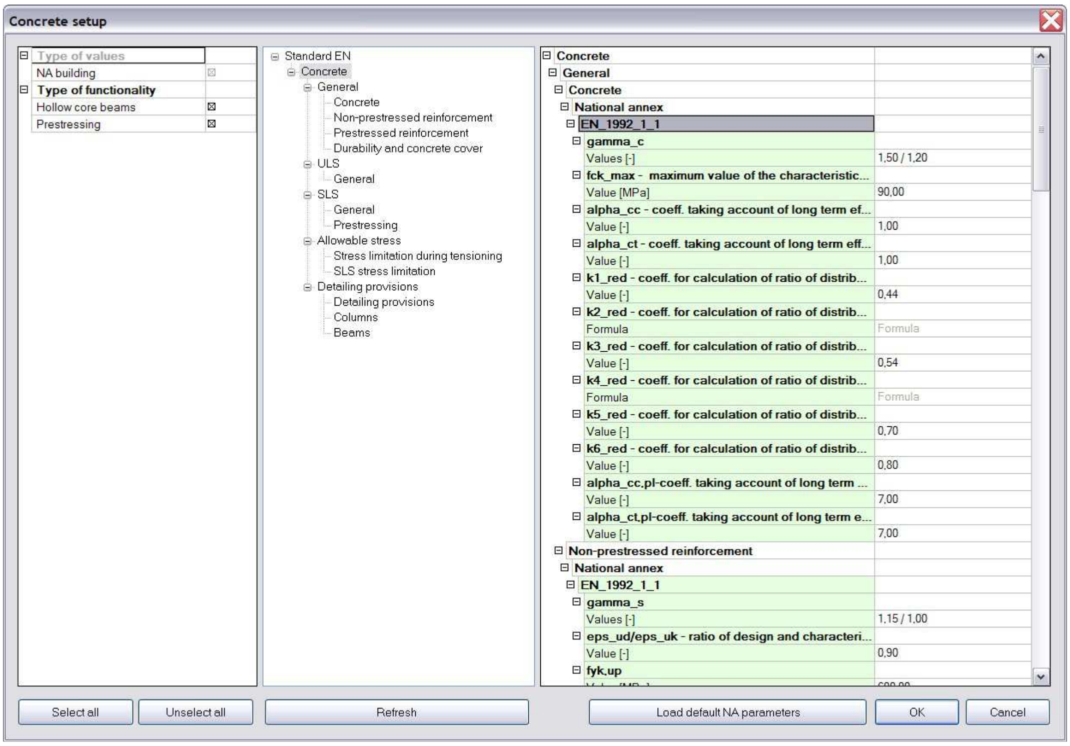
<u>Conclusion</u>: Kirchhoff gives the expected shear force values, Mindlin gives the expected torsion moments.

Concrete setups

1_General setups

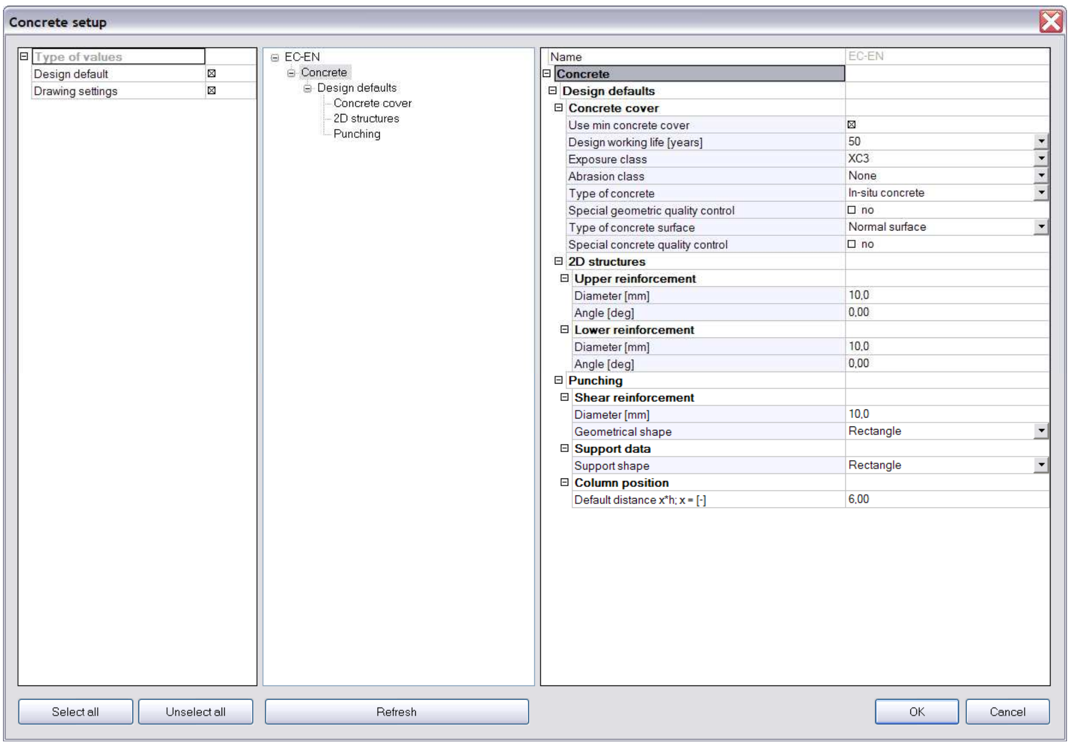
Setup 1: National Determined Parameters

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



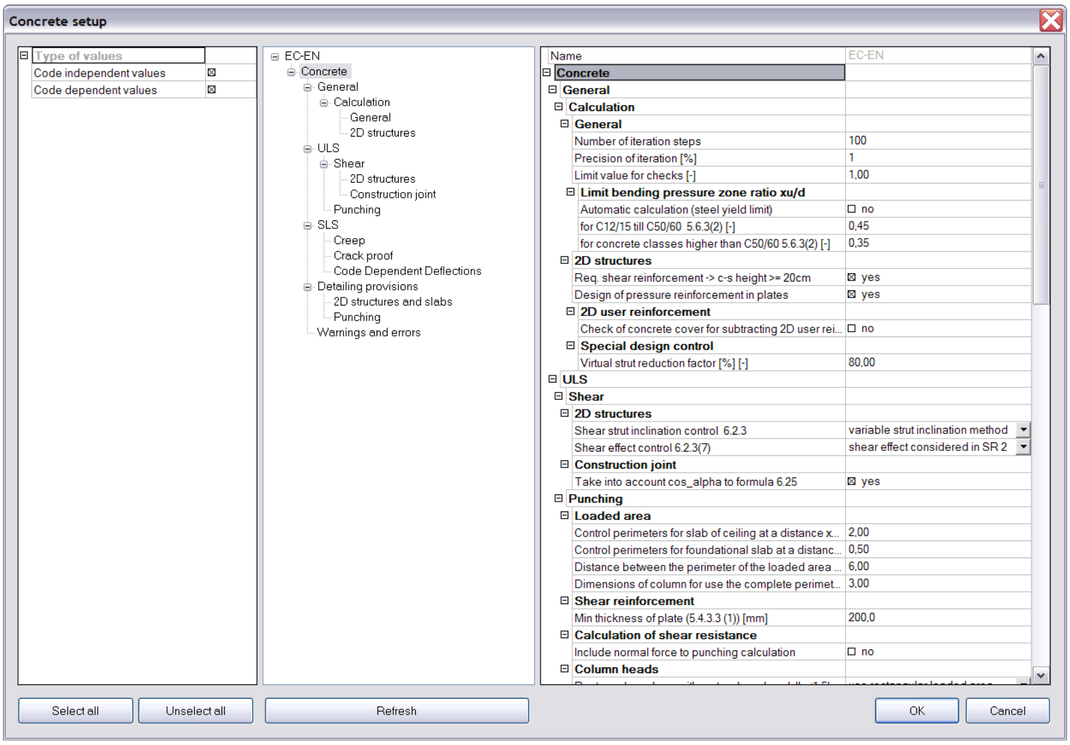
Setup 2: Basic settings

Concrete menu > Design defaults



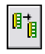
Setup 3: Advanced settings

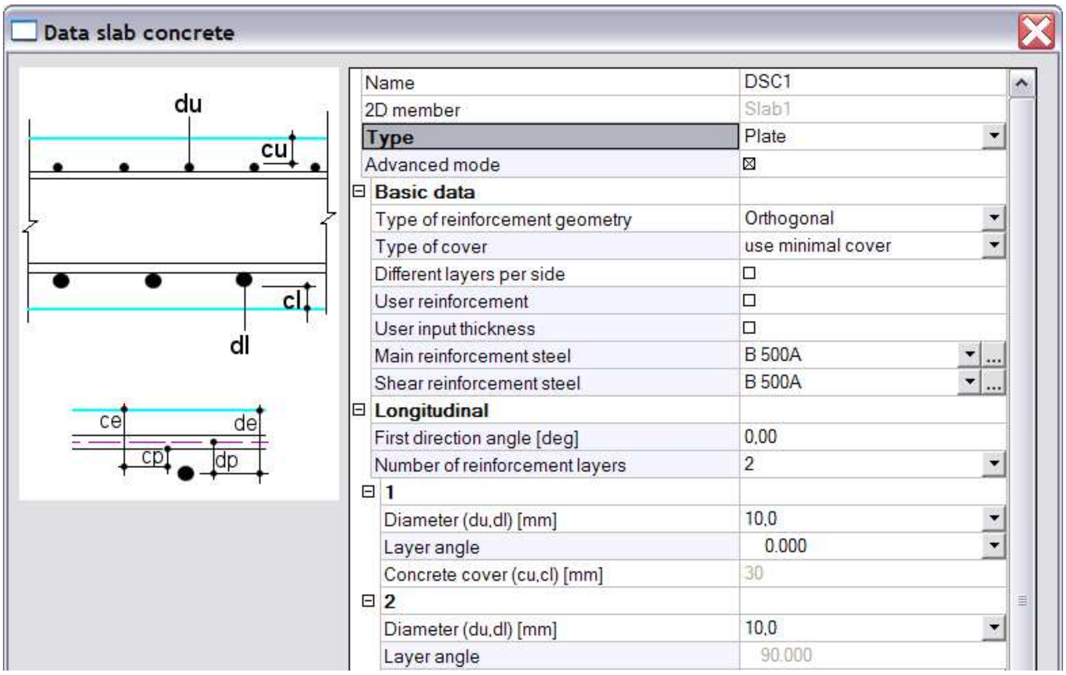
Setup menu > Concrete solver



All of the adjustments made in one of the three 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 > 2D member > Member data.
On a plate with Member data appears a label, e.g. DSC1 (= Data Slab Concrete). 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.


Data slab concrete
du
cu
cl
dl
ce
de
cp
dp
Name
DSC1
2D member
Slab1
Type
Plate
Advanced mode
Basic data
Type of reinforcement geometry
Orthogonal
Type of cover
use minimal cover
Different layers per side
User reinforcement
User input thickness
Main reinforcement steel
B 500A
Shear reinforcement steel
B 500A
Longitudinal
First direction angle [deg]
0,00
Number of reinforcement layers
2
1
Diameter (du,dl) [mm]
10,0
Layer angle
0.000
Concrete cover (cu,cl) [mm]
30
2
Diameter (du,dl) [mm]
10,0
Layer angle
90.000

ULS design

1_Theoretical reinforcement design

Internal forces

Concrete menu > 2D member > Member design > Internal forces ULS

Basic magnitudes

The values shown here are exactly the same as in the Results menu; they are calculated by the FEM solver.
The example is continued with the results based on the Mindlin theory.

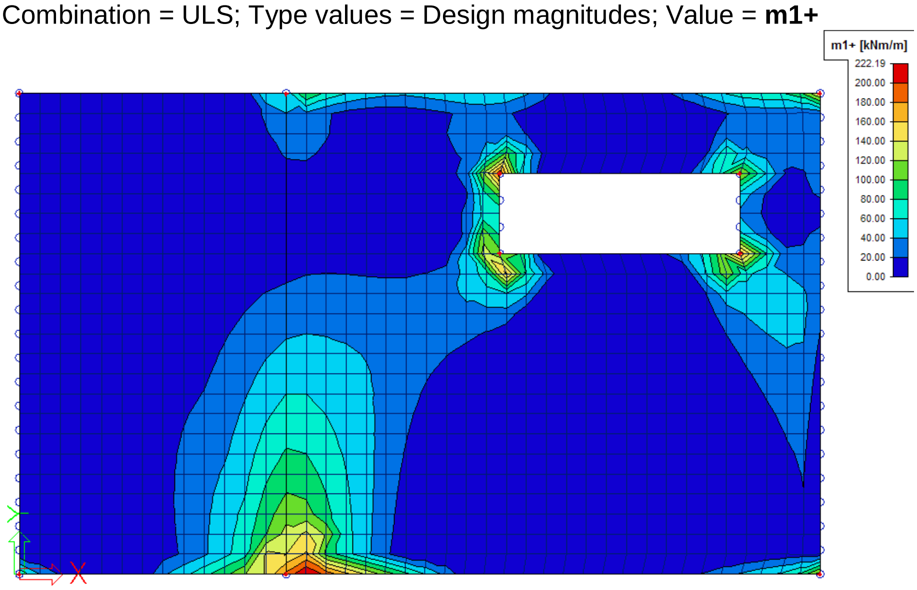
Design magnitudes

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.

For more detailed information, reference is made to the Benchmark examples added at the end of this manual.

Take a look at the available values: m1+, m1-, m2+, m2- (and mc+, mc-)
"+" 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.
(mc+ and mc- 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 = **m1+**



Compare the result for this value m1+ (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 'shear effect' / 'shift of the moment line' that is only taken into account in the design magnitudes calculated by the NEDIM solver (values m1 and m2).

Shear effect / Moment shift

The notion 'shear effect' is used for the additional tensile force in the longitudinal reinforcement, caused by shear.
This *additional tensile force* is nothing else than the natural force complement to the stirrups force, both representing, together with the concrete strut force, the resistance of the reinforced concrete continuum to shear force impact. The thus required *additional longitudinal reinforcement* is no increment to bending reinforcement, but an autonomic longitudinal shear reinforcement, complementary to the (lateral) stirrups. In low-height cross-sections, the longitudinal shear reinforcement is usually constructively merged with the tension reinforcement (due to m/n).In high cross-sections it has to be dispersed along the cross-section height.

In EN 1992, 2 approaches are described to take this additional tensile force into account in the reinforcement design:

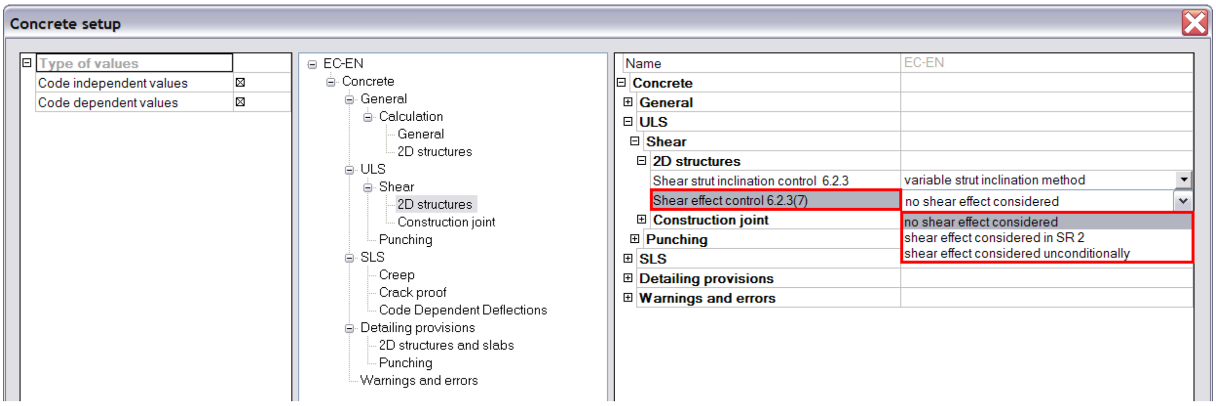
1) The 'shear effect' approach – EN §6.2.3(7)
This approach is meant for members with shear reinforcement.
The additional tensile force, ΔF_{td}, in the longitudinal reinforcement due to shear V_{Ed} is calculated from

$$\Delta F_{td} = 0{,}5\ V_{Ed}\ (\cot\theta - \cot\alpha) \qquad \text{(EN formula 6.18)}$$

and $(M_{Ed}/z) + \Delta F_{td} \leq M_{Ed,max}/z$, where $M_{Ed,max}$ is the maximum moment along the beam.

2) The classic 'moment shift' approach – EN §9.2.1.3(2) and §6.2.2(5)
→ For members without shear reinforcement, ΔF_{td} may be estimated by shifting the moment curve (in the region cracked in flexure) a distance $a_l = d$ in the unfavourable direction.
→ For members with shear reinforcement, this 'shift rule' may also be used as an alternative to the 'shear effect' approach, where $a_l = z\ (\cot\theta - \cot\alpha)\ /\ 2$. (EN formula 9.2)

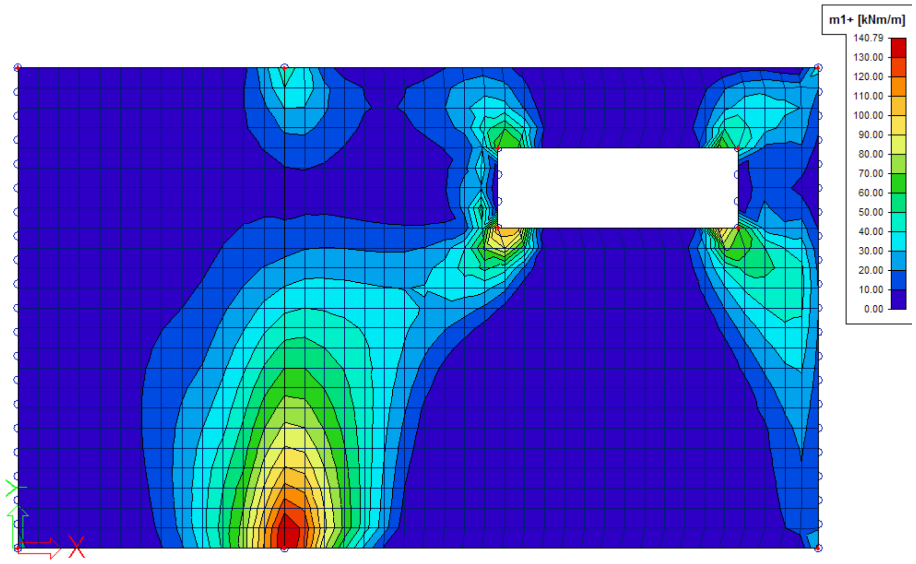
In SCIA Engineer there are 3 options to control the 'shear effect' in the 2D reinforcement design.
The choice can be made in the Setup menu > Concrete solver > ULS – Shear – 2D structures:



a) Shear effect considered in SR2
The value of ΔF_{td} is calculated directly (EN §6.2.3(7)), which is in fact the most accurate approach. 'SR2' stands for shear region 2, which is defined by the DIN 1045 terminology as the region where shear reinforcement is required to resist v_{Ed}.
This is the default setting in SCIA Engineer, which explains the design moments m1+ along the upper and lower edges of the plate (see image on previous page). Because of the high (singular) shear forces along the free edges, also a high value of ΔF_{td} is obtained, which is accounted for in the design moments.

b) No shear effect considered
The 'moment shift' approach (EN §9.2.1.3(2)) is applied.
If this option is activated, the result for m1+ is as follows.

Combination = ULS; Type values = Design magnitudes; Value = **m1+**



c) Shear effect considered unconditionally
The same as (a), but the value of ΔF_{td} is also calculated in the region SR1.
'SR1' stands for the region where no shear reinforcement is required; the cross-section resists v_{Ed} by the bearing capacity of plain concrete. It is, however non-standard, an option, because some norms, like NEN 6720 §8.1.1, require the 'shift' of the moment line also in SR1.
For design acc. to EN 1992, this is practically not an alternative.

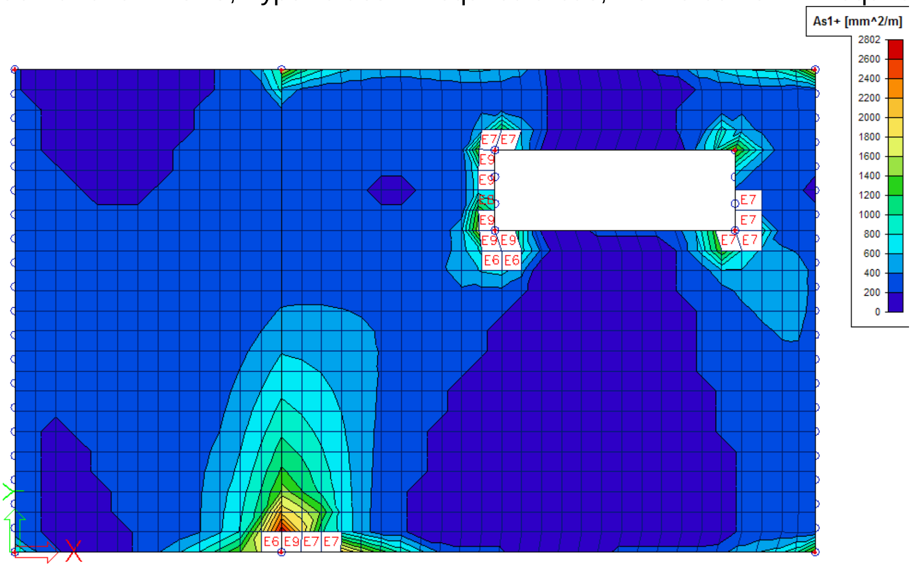
Theoretically required reinforcement

Concrete menu > 2D member > Member design > Member design ULS

Longitudinal reinforcement

Analogously to the design magnitudes, the available values here are: As1+, As1-, As2+, As2-

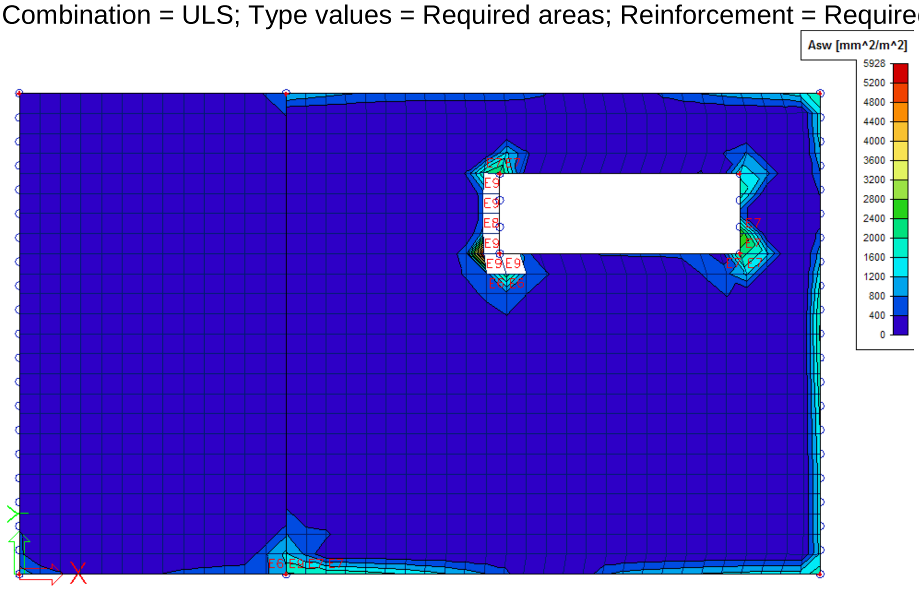
Combination = ULS; Type values = Required areas; Reinforcement = Required reinf.; Value = **As1+**



This is the longitudinal reinforcement, calculated based on the design magnitude **m1+**.
In the tension zones, a minimum reinforcement area is taken into account by default, according to EC-EN §9.2.1.1(1).

Shear force reinforcement

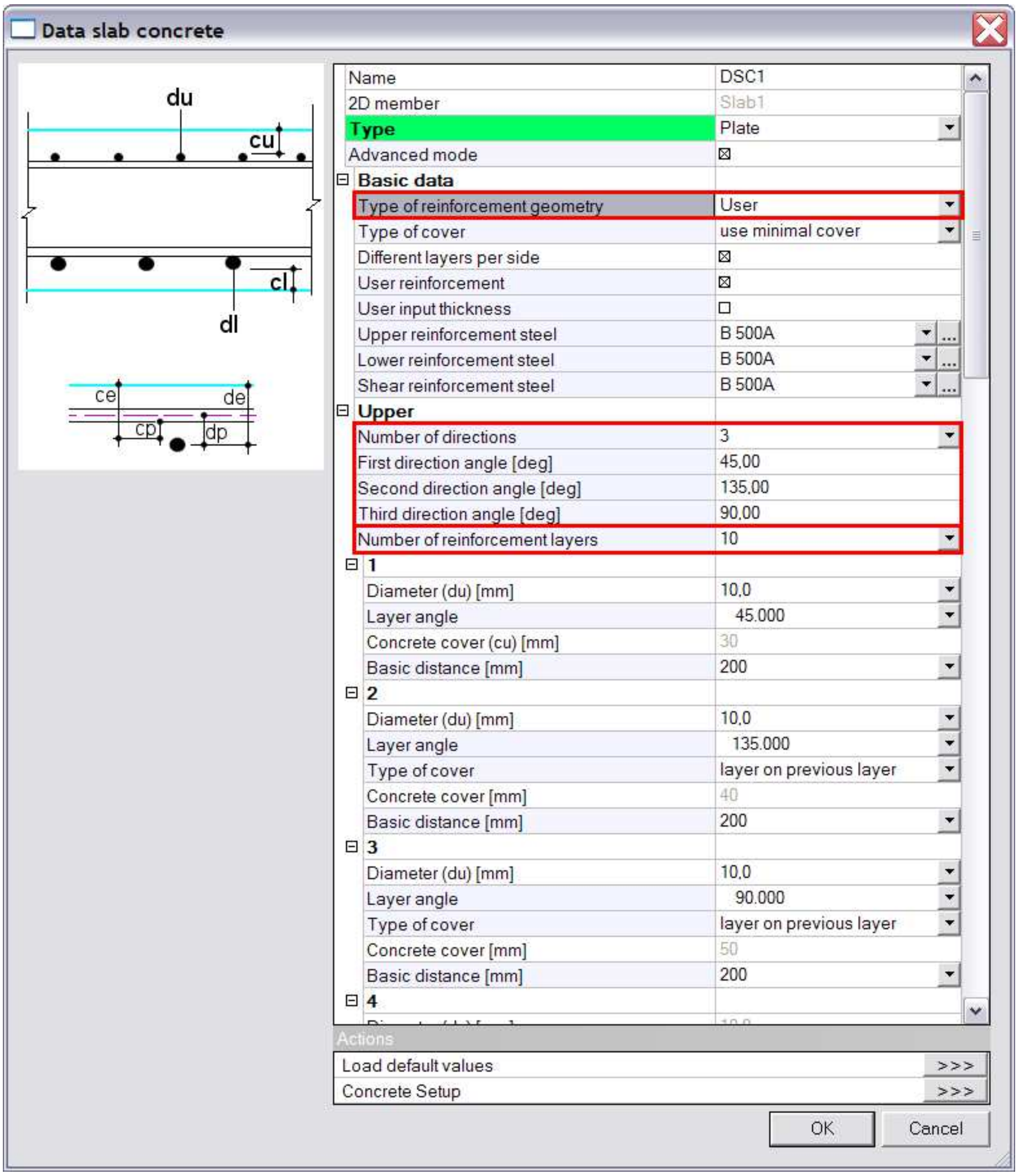
Combination = ULS; Type values = Required areas; Reinforcement = Required reinf.; Value = **Asw**



Reinforcement directions & layers

The convention for the reinforcement areas is obviously the same as for the design magnitudes: "+" and "-" stand for the reinforcement areas 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.

The user is free to change the default direction angles and, moreover, add a 3rd reinforcement direction. This is only possible via Member data:



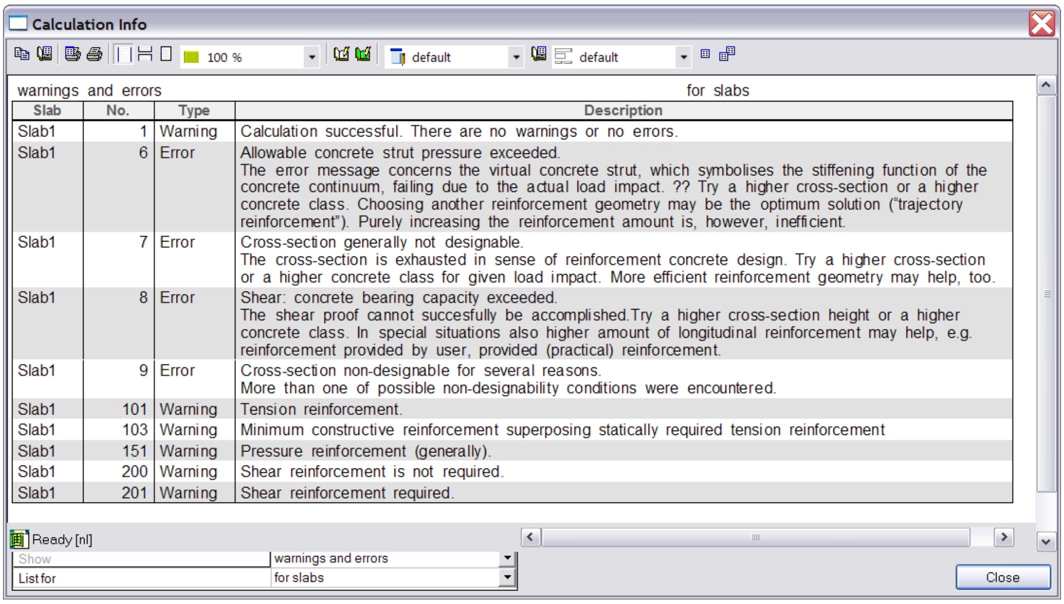
Up to 10 reinforcement layers per side (top / bottom) can be created; to each layer one of the 3 directions is assigned.

Attention: Layers are always numbered from the outside to the centre of the 2D member!

Errors & warnings

Concrete menu > 2D member > Member design > Member design ULS

At the bottom of the Properties menu, go to Actions > Calculation Info:



warnings and errors for slabs

Slab	No.	Type	Description
Slab1	1	Warning	Calculation successful. There are no warnings or no errors.
Slab1	6	Error	Allowable concrete strut pressure exceeded. The error message concerns the virtual concrete strut, which symbolises the stiffening function of the concrete continuum, failing due to the actual load impact. ?? Try a higher cross-section or a higher concrete class. Choosing another reinforcement geometry may be the optimum solution ("trajectory reinforcement"). Purely increasing the reinforcement amount is, however, inefficient.
Slab1	7	Error	Cross-section generally not designable. The cross-section is exhausted in sense of reinforcement concrete design. Try a higher cross-section or a higher concrete class for given load impact. More efficient reinforcement geometry may help, too.
Slab1	8	Error	Shear: concrete bearing capacity exceeded. The shear proof cannot succesfully be accomplished.Try a higher cross-section height or a higher concrete class. In special situations also higher amount of longitudinal reinforcement may help, e.g. reinforcement provided by user, provided (practical) reinforcement.
Slab1	9	Error	Cross-section non-designable for several reasons. More than one of possible non-designability conditions were encountered.
Slab1	101	Warning	Tension reinforcement.
Slab1	103	Warning	Minimum constructive reinforcement superposing statically required tension reinforcement
Slab1	151	Warning	Pressure reinforcement (generally).
Slab1	200	Warning	Shear reinforcement is not required.
Slab1	201	Warning	Shear reinforcement required.

This gives an overview of all the warnings and errors present in the project.

Warning = Information about the applied reinforcement.
Error = Real (theoretical) design problem: the result value cannot be calculated.

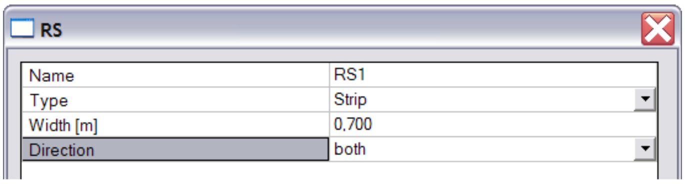
Most common errors

- Error E8: Shear: concrete bearing capacity exceeded
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.

- Error E6: Allowable concrete strut pressure exceeded
This means in fact that the resistance of the virtual concrete strut is exhausted, so the concrete strut gets 'crushed'. This is a way of failure that isn't considered in any other calculation software (according to the developer of our solver), but is a real issue. The virtual concrete strut symbolises the stiffening function of the concrete continuum. The pressure in the strut may not exceed 80% of the concrete strength fcd (this reduction coefficient can be adapted in the Concrete Setup), in order to provide for the effective diminuishing of the 'plain' concrete strength due to parallel cracks. Failure of the strut is generally caused by inefficient reinforcement geometry.
The solution is to try a thicker cross-section or a higher concrete class, this is generally the easiest way to improve the bearing capacity of the concrete strut. Choosing another reinforcement geometry may be the optimum solution ("trajectory reinforcement", following the principal directions), but is less practical. Purely augmenting the reinforcement amount is, however, inefficient. Also the use of Averaging strips won't help in most of the cases of E6 errors, because we are not dealing with a singularity here.

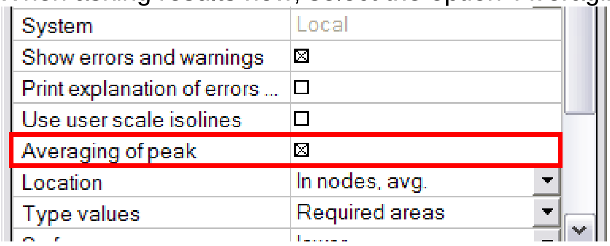
Measure 1: Averaging strips

Concrete menu > 2D member > Averaging strip
Add averaging strips to the short sides of the opening, where the line supports are located. Also above the line supports over the total width of the slab, an averaging strip can be added.



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.

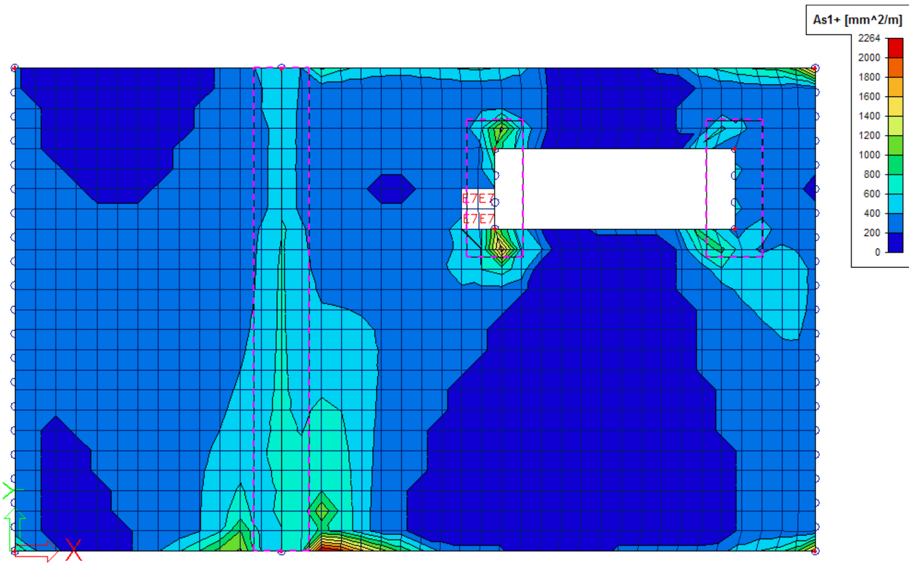
When asking results now, select the option ‘Averaging of peak’ in the Properties menu!



When asking the values for the theoretical reinforcement again, the errors E8 don’t appear anymore:

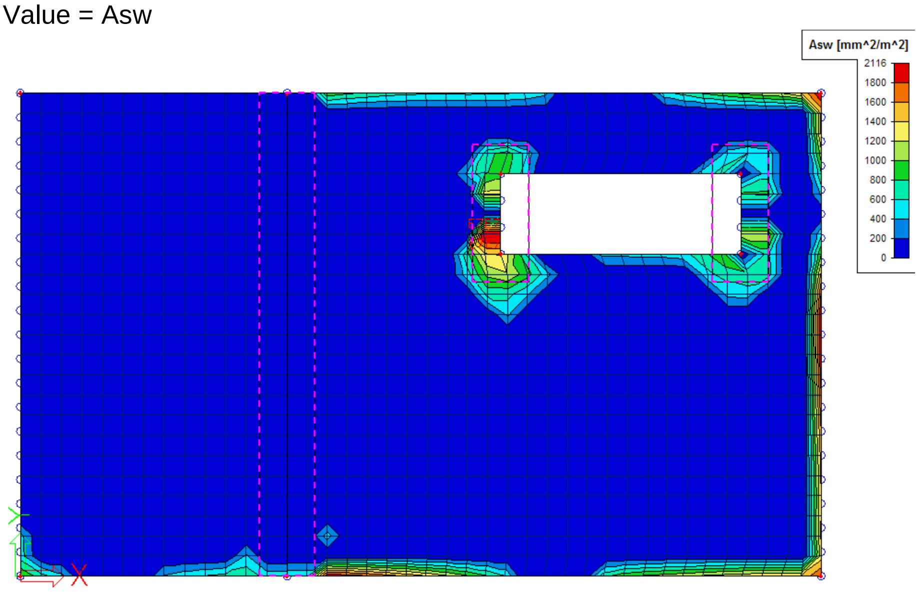
Longitudinal reinforcement

Value = As1+



Shear force reinforcement

Value = Asw



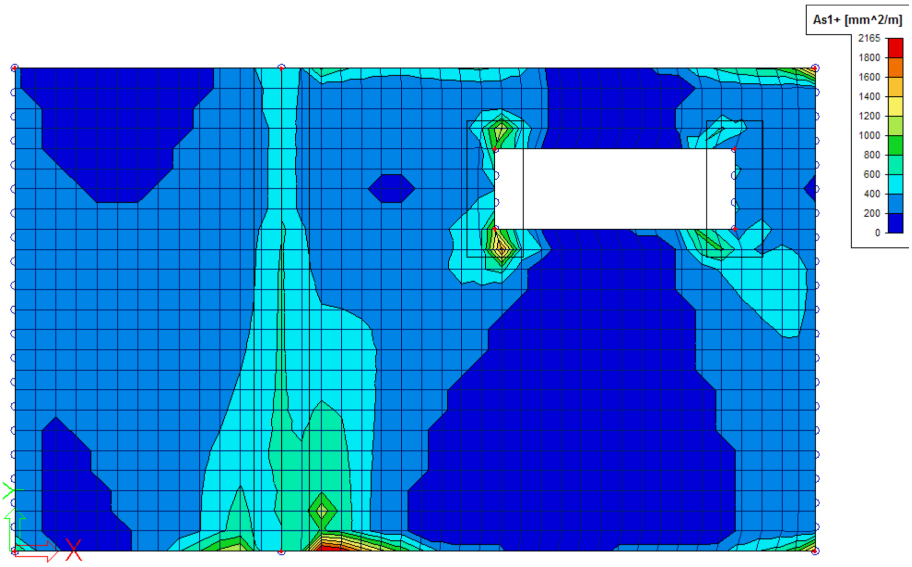
Measure 2: Increasing the concrete grade

In the Properties menu of the slab, choose for a higher concrete grade: C25/30 instead of C20/25.

After a new linear calculation, and when asking the values for the theoretical reinforcement again, the error E6 has also disappeared:

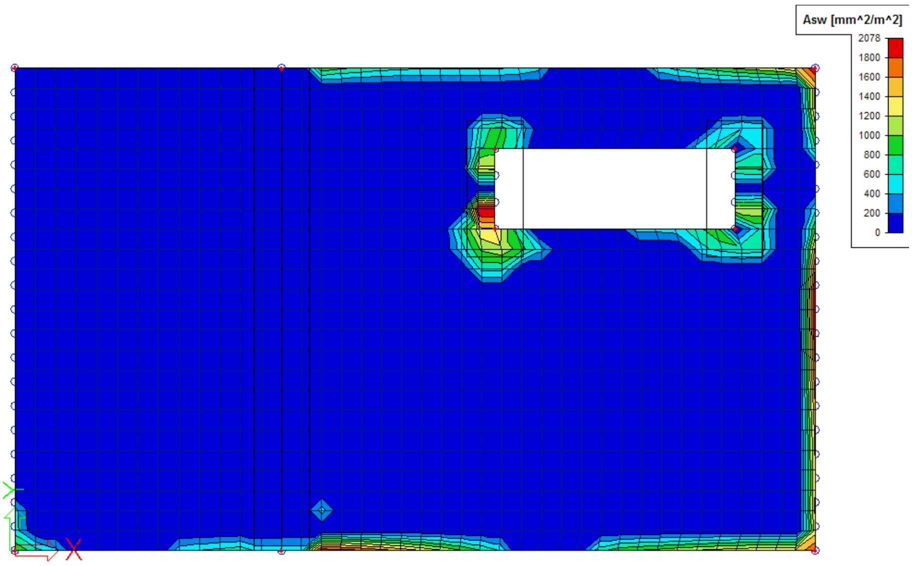
Longitudinal reinforcement

Value = As1+



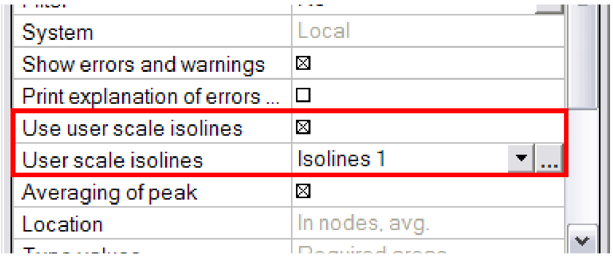
Shear force reinforcement

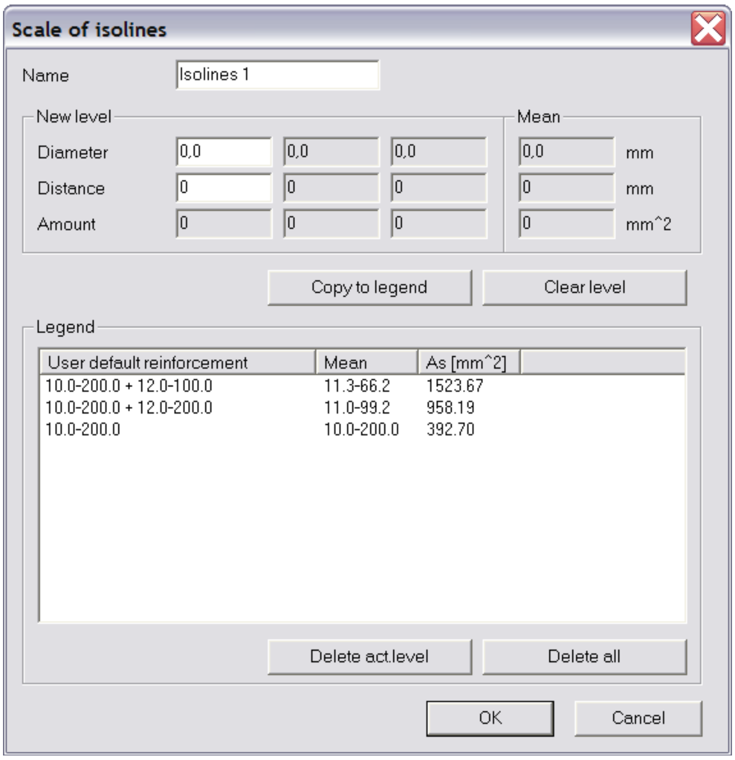
Value = Asw



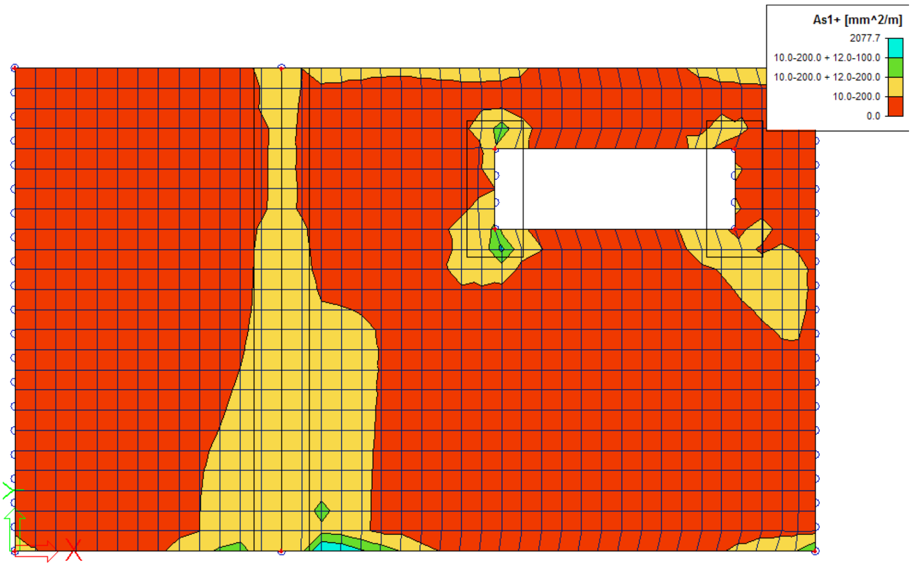
Extra feature: User scale isolines

Keep in mind that this feature is no basic reinforcement, it's only an adaptation of the graphical display for the results:





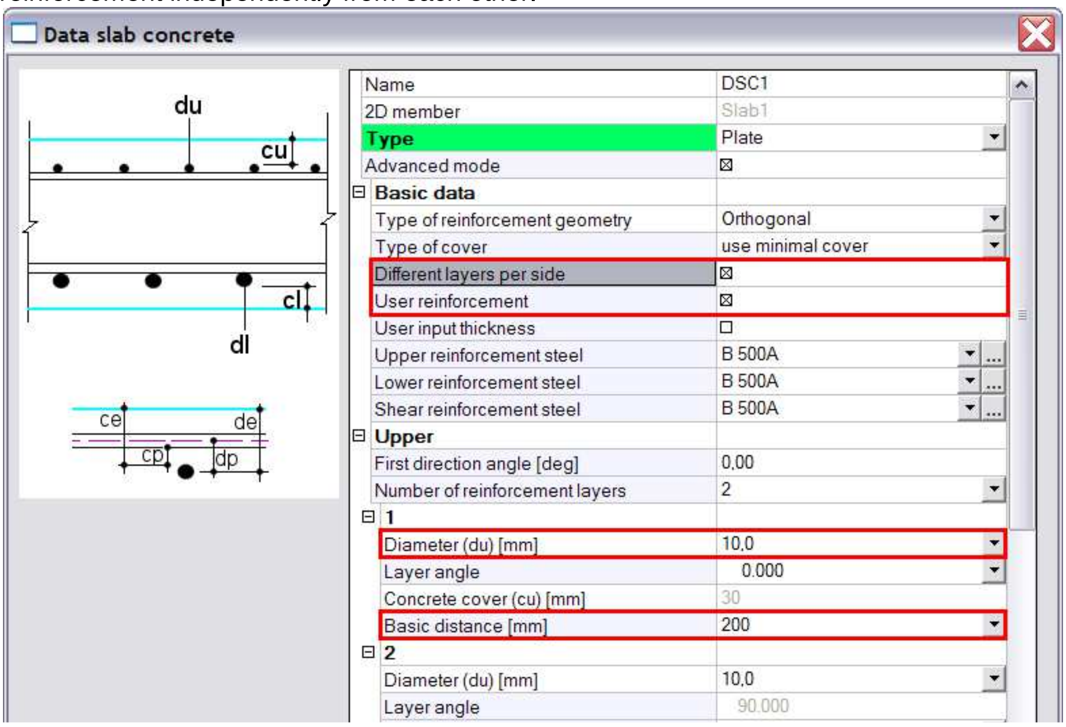
Longitudinal reinforcement
Value = As1+



Basic reinforcement

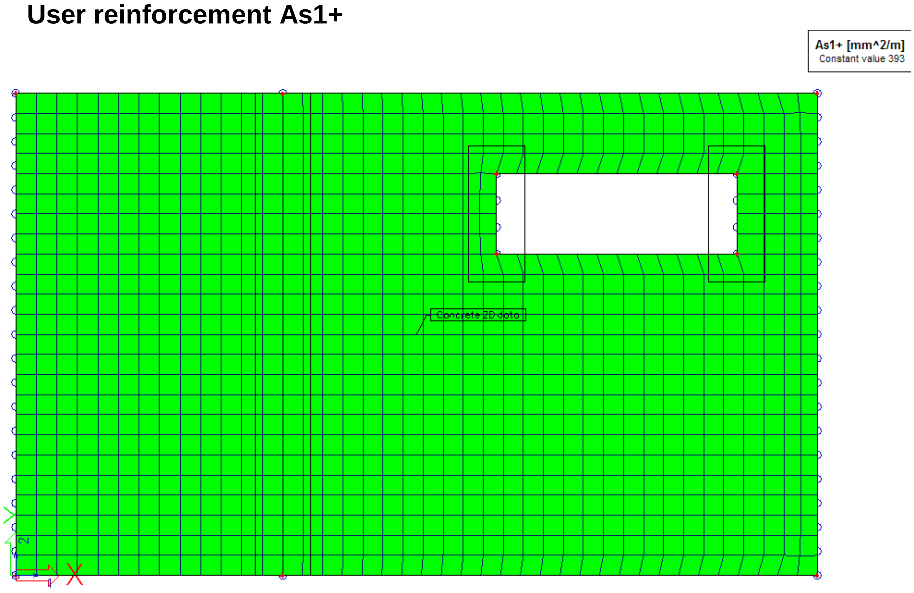
This is a reinforcement amount added to the whole plate. In SCIA Engineer, the *basic* reinforcement is referred to as *user* reinforcement

Defining (theoretical) basic reinforcement is only possible via Member data.
Select the plate, choose the option ‘User reinforcement’, and input a Diameter and Basic distance for directions 1 and 2.
If you select also the option ‘Different layers per side’, it is possible to define the upper and lower reinforcement independently from each other.

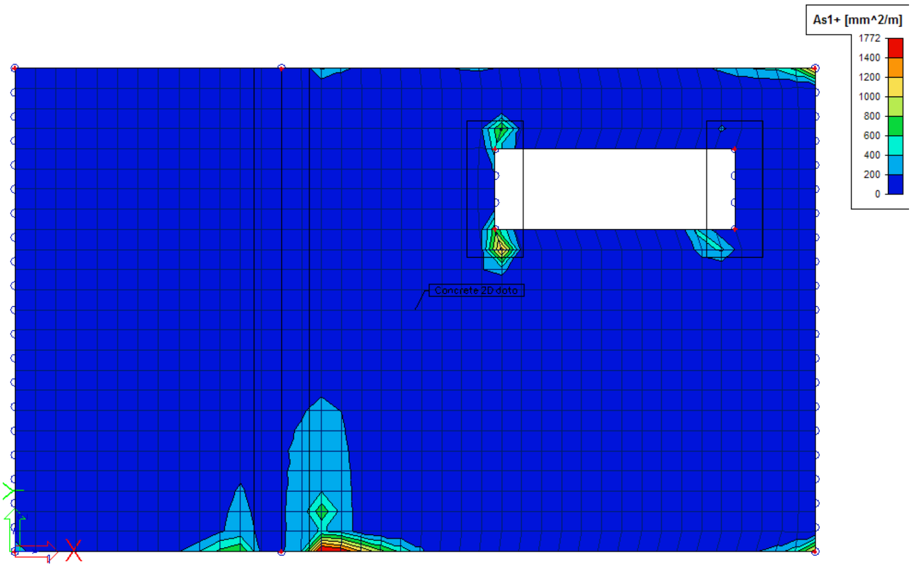


To view the modifications, go to 2D member > Member design – Design ULS. For Combination = ULS ask Reinforcement = User / Additional / Total reinforcement, with Value = As1+

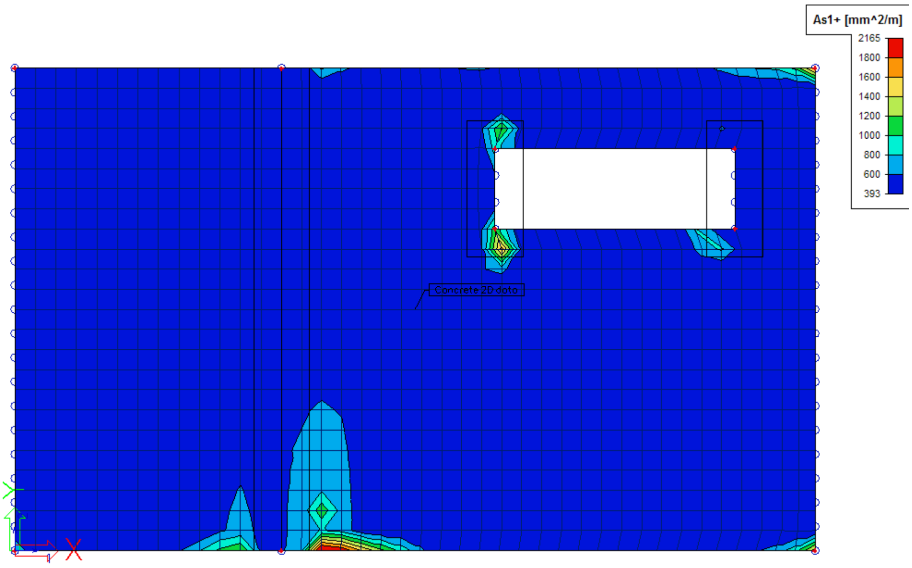
User reinforcement As1+



Additional reinforcement As1+



Total reinforcement As1+



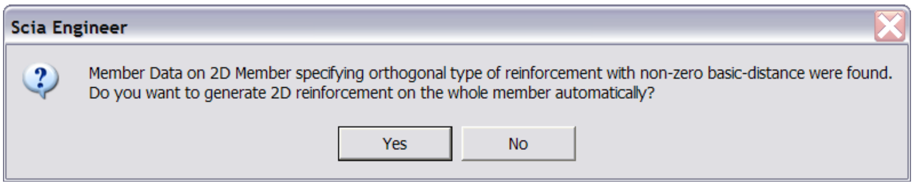
Take a look at the legend to see the difference in result values between additional and total reinforcement.

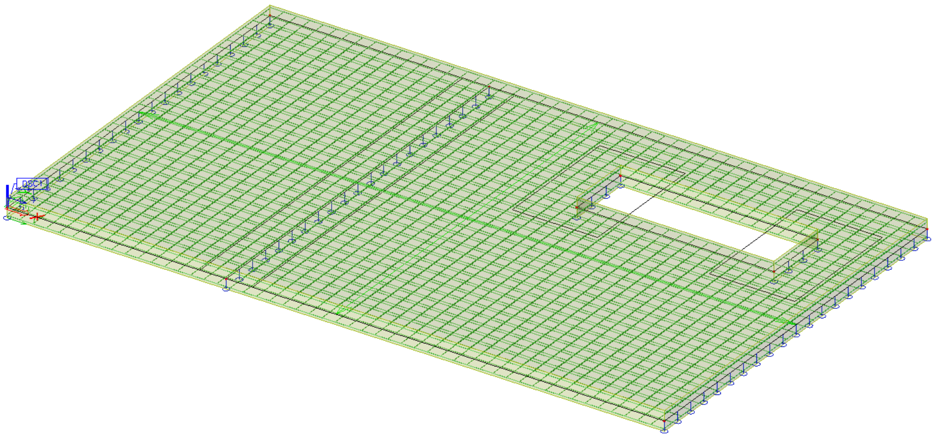
2_Practical reinforcement design

Basic reinforcement

It is logical to start with the definition of a practical reinforcement amount added to the whole plate. This can be done via the Concrete menu > 2D member > Reinforcement 2D.

A question appears, if the theoretical basic reinforcement (defined via Member data) should be transferred to practical reinforcement. Choose ‘Yes’.





<u>Attention</u>: From the moment practical reinforcement has been added to a member, the total amount of *practical* reinforcement is accounted for as the *user* reinforcement. This means that the theoretical basic reinforcement (defined via Member data) is overwritten!

Additional reinforcement

In a second step, additional reinforcement might be defined on s***pecific location(s) on the plate. This can be done via the same option -*** Concrete menu > 2D member > Reinforcement 2D.

The locations where additional reinforcement is required, can be asked for via the Concrete menu > 2D member > Member design – Design ULS. Since the present practical reinforcement is the User reinforcement, ask for Combination = ULS, Reinforcement = Additional reinforcement.

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

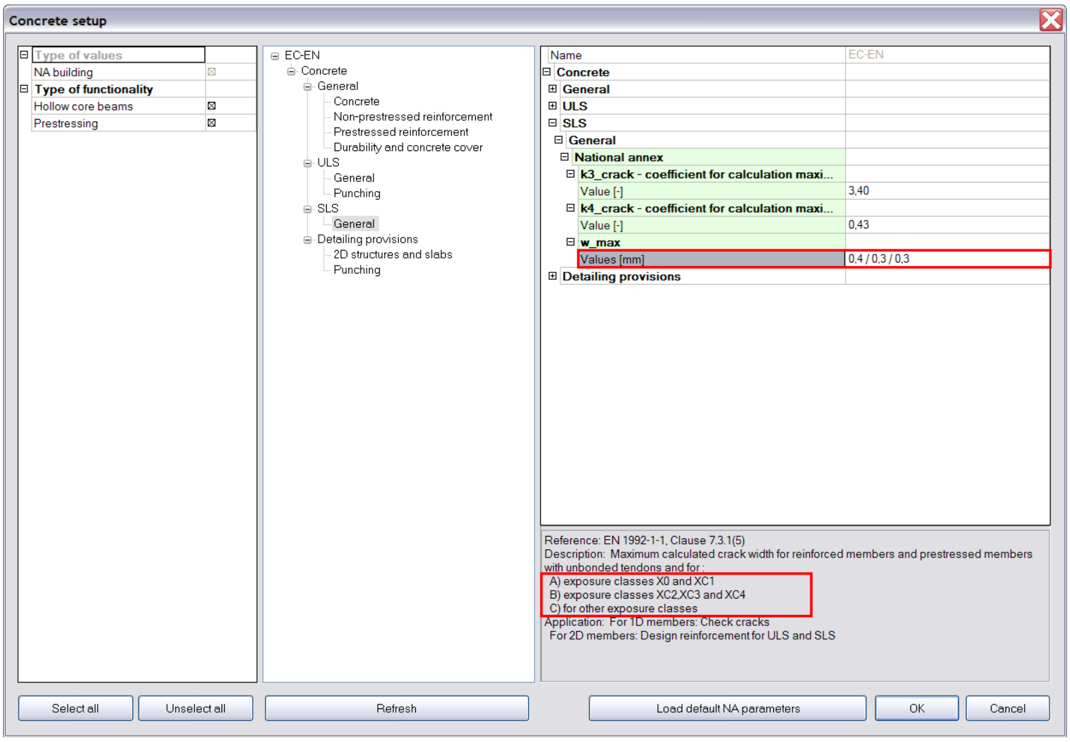
ULS + SLS design

In this design process, next to the ULS requirements, also the requirements for cracking (SLS) are met.

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 […].



The **NEDIM** solver will perform the crack control for 2D members.
The approach differs from the crack control for 1D members, where a value of the crack width is calculated and then checked against the value of w_{max}. For 2D members, the crack width is automatically limited during an iteration process (to 0,3 or 0,4 mm - based on the chosen exposure class).

Class ‘All ULS+SLS’

According to the Eurocode, the requirements for cracking have to be met under the quasi-permanent load combinations.
The intention of the ULS + SLS design is to give the user a required reinforcement amount that meets the ULS *and* cracking requirements. Therefore a class ‘All ULS+SLS’ has to be created, where at least one ULS combination and at least one quasi-permanent SLS combination are included. The NEDIM solver will first calculate the theoretical required reinforcement based on the ULS combination(s), and store the results in its memory. For this reinforcement amount (As,ULS) is then a crack control performed, based on the SLS combination(s). In the finite elements where $w_{calc} \le w_{max}$, the value of As,ULS is sufficient (As,ULS > As,SLS). In the finite elements where $w_{calc} > w_{max}$, extra reinforcement is added during an iteration process, until $w_{calc} \le w_{max}$.

The required reinforcement that results from the ULS + SLS design is thus the maximum of (As,ULS; As,SLS), and will be used in most cases as minimum value for the practical design.

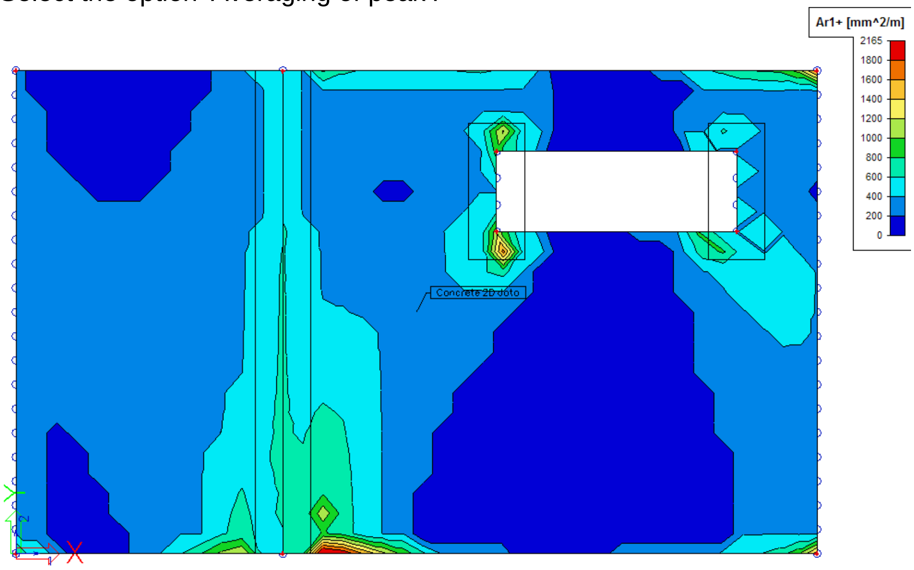
2_Theoretical reinforcement design

Concrete menu > 2D member > Member design > Member design ULS+SLS

Longitudinal reinforcement

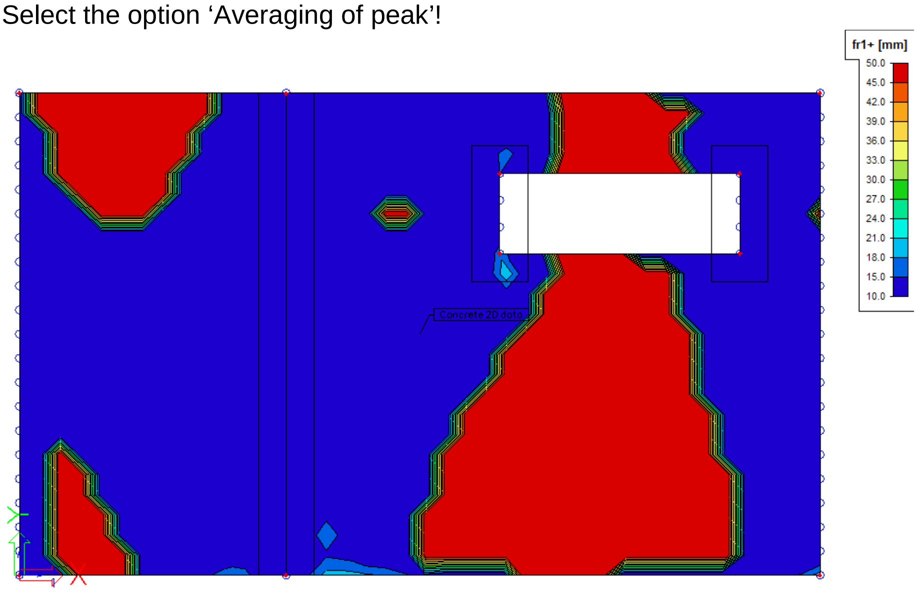
Analogously to the Member design ULS, the available values here are: Ar1+, Ar1-, Ar2+, Ar2-
The subscription ‘s’ is replaced here by ‘r’ to make clear that the requirements for cracking are accounted for in these values.

Class = All ULS+SLS; Type values = Required areas; Reinforcement = Required reinf.; Value = **Ar1+**
Select the option ‘Averaging of peak’!



Maximal bar diameters

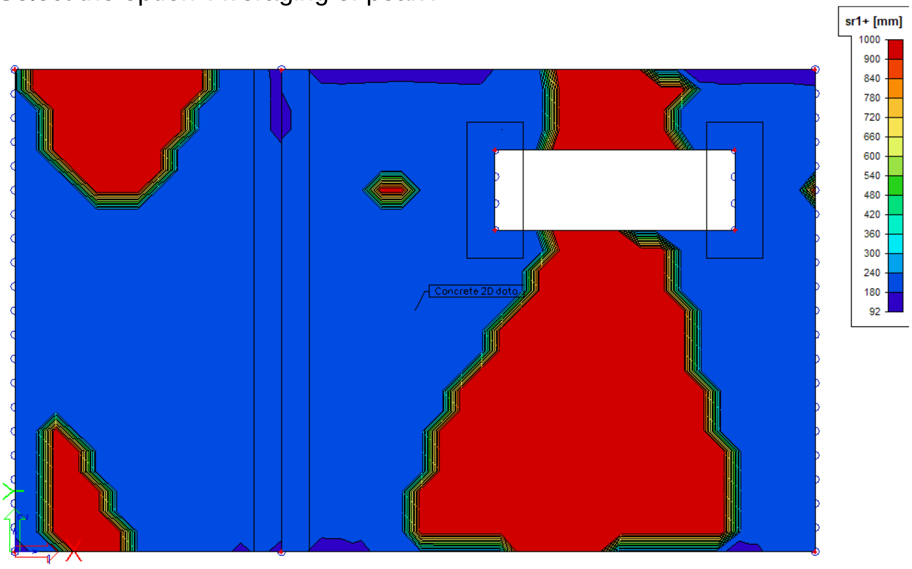
Class = All ULS+SLS; Type values = Maximal diameters; Value = **fr1+**
Select the option ‘Averaging of peak’!



Maximal bar distances

Class = All ULS+SLS; Type values = Maximal distances; Value = **sr1+**

Select the option ‘Averaging of peak’!



3_Practical reinforcement design

The procedure is exactly the same as for only ULS design, therefore reference is made to the previous chapter ‘ULS design’.

Crack control

1_Input data for crack control

Maximum crack width

For information about the values of the maximum crack width (w_{max}) taken into account by SCIA Engineer, reference is made to the previous chapter ‘ULS + SLS design’.

Combination SLS

According to the Eurocode, the requirements for cracking have to be met under the quasi-permanent load combinations.
The intention of the Crack control is to check if these requirements are met for 2D members with a *certain reinforcement amount*. The theoretical or practical reinforcement design has already been done in advance, therefore no class ‘All ULS+SLS’ is needed. The check will be executed only for the SLS combination(s).

Type of used reinforcement

The reinforcement amount in a 2D member for which the Crack control will be executed, is referred to as As,tot or As,user.

User reinforcement As,user

= Basic reinforcement	(as defined via Member data)
= Practical reinforcement	(as defined via Reinforcement 2D – if practical reinf. is designed, then its amount overwrites any basic reinf.)

Total reinforcement As,tot

= Required theoretical reinforcement	(if As,user = 0)
= As,user + As,additional	(if As,user ≠ 0)

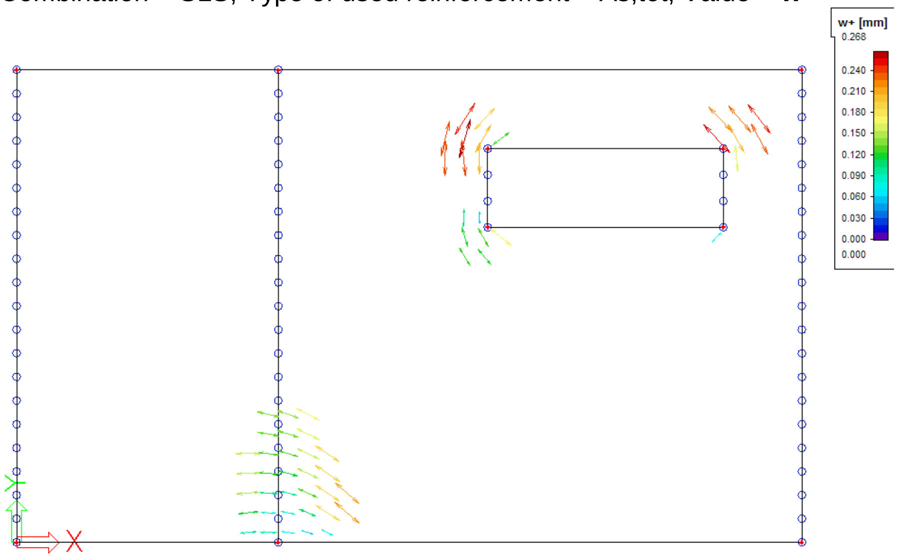
2_Results for required theoretical reinforcement

Concrete menu > 2D member > Member check - Crack control

If you are interested in the results of the Crack control for the required theoretical reinforcement amount, then no basic nor practical reinforcement may be present! In the Properties menu, the ‘Type of used reinforcement’ = ‘As,tot’.
To be able to get any results for the Crack control, the required theoretical reinforcement has to be stored at first. So go to Member design ULS or Member design ULS+SLS, and refresh the results.
For this example, the results for Member design ULS+SLS are stored, where the calculated required reinforcement meets the requirements for cracking. Therefore we expect to find crack widths with a maximum value of 0,3 mm (according to the chosen exposure class XC3).
Attention: This is nevertheless only the case if the results for Member design ULS+SLS are stored *without* averaging of peaks!

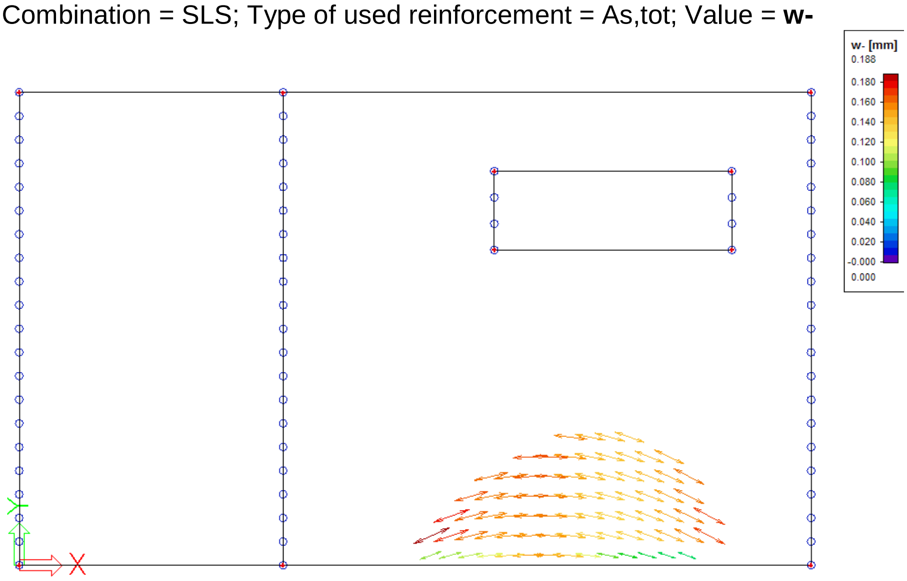
Crack width w+

Combination = SLS; Type of used reinforcement = As,tot; Value = **w+**



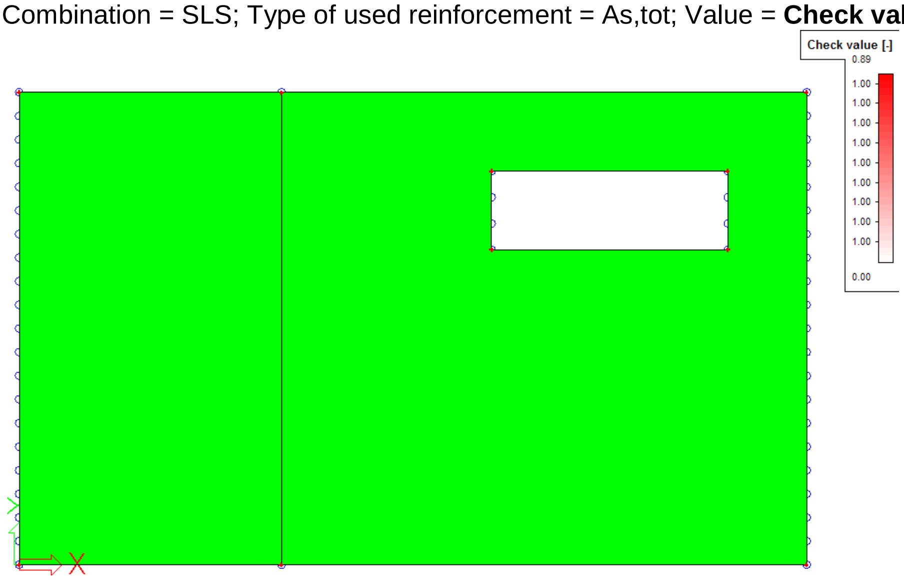
Crack width w-

Combination = SLS; Type of used reinforcement = As,tot; Value = **w-**



Unity check

Combination = SLS; Type of used reinforcement = As,tot; Value = **Check value**



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

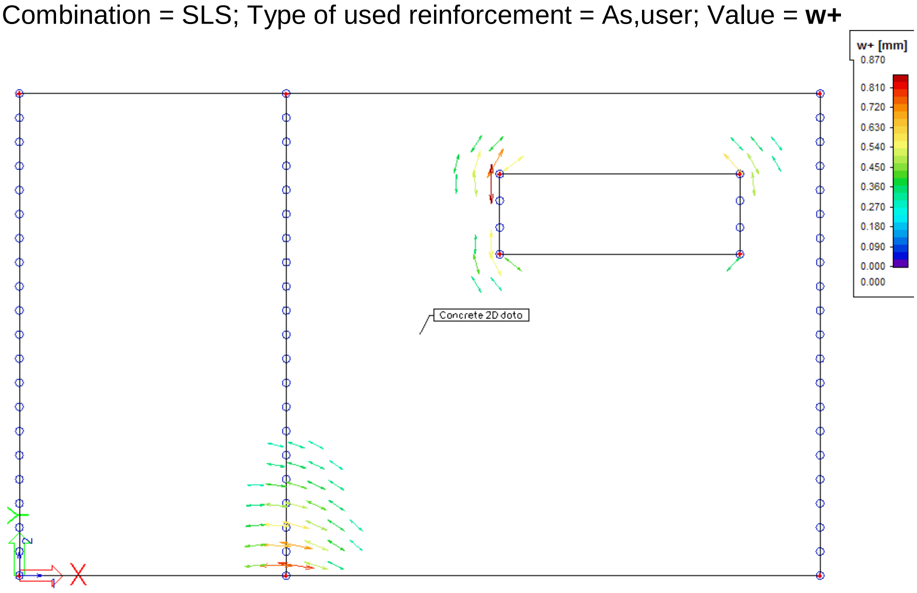
3_Results for basic reinforcement

Concrete menu > 2D member > Member check - Crack control

If you are interested in the results of the Crack control for the basic reinforcement amount defined via Member data, then no practical reinforcement may be present! In the Properties menu, the 'Type of used reinforcement' = 'As,user'.
The necessary additional reinforcement is not taken into account here, so it is probable that we will find crack widths larger than 0,3 mm.

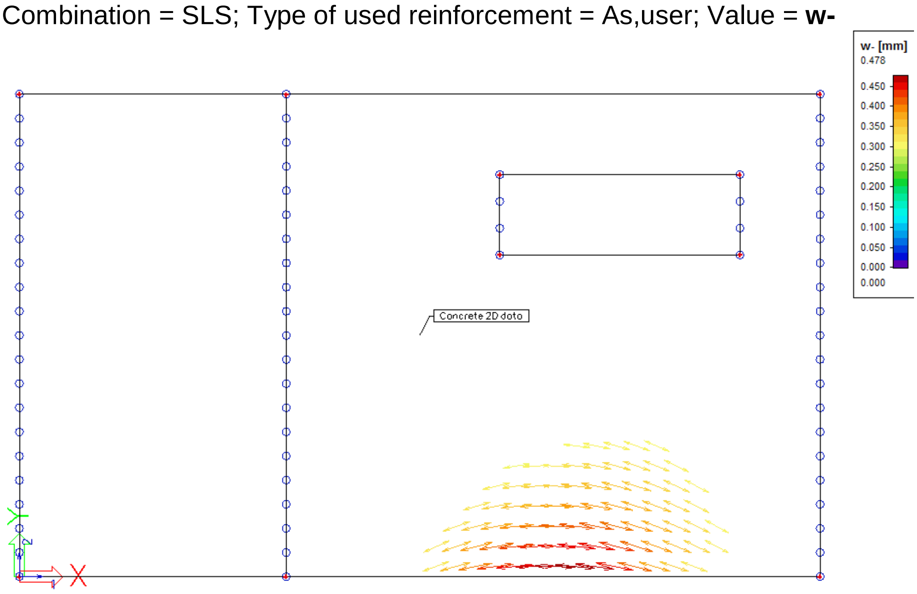
Crack width w+

Combination = SLS; Type of used reinforcement = As,user; Value = **w+**



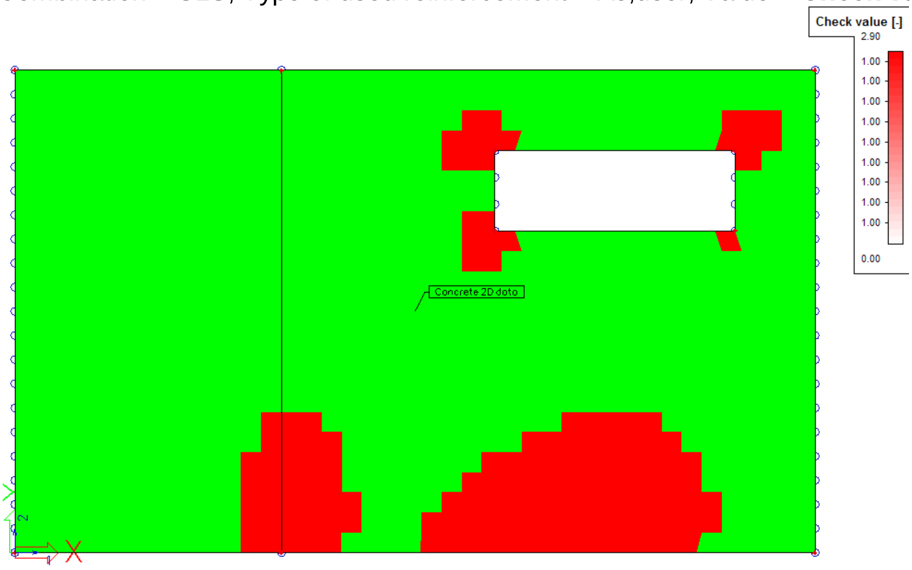
Crack width w-

Combination = SLS; Type of used reinforcement = As,user; Value = **w-**



Unity check

Combination = SLS; Type of used reinforcement = As,user; Value = **Check value**



A green value stands for a Unity check ≤ 1 ($w_{calc} \leq w_{max}$), while a red value means that w_{max} is exceeded.

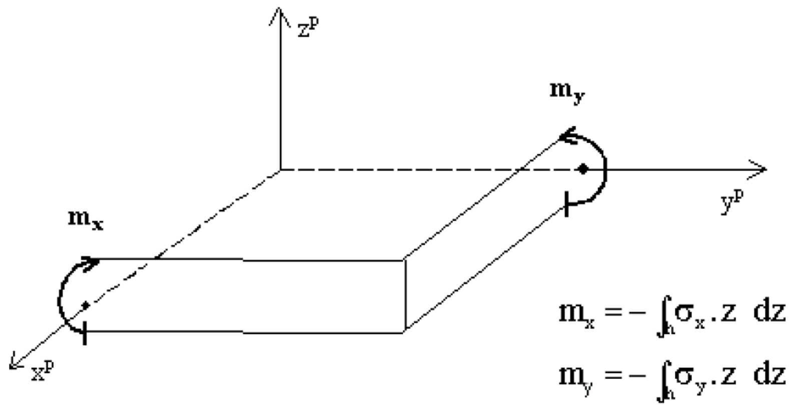
Annexes

Annex 1: Conventions for the results on 2D members

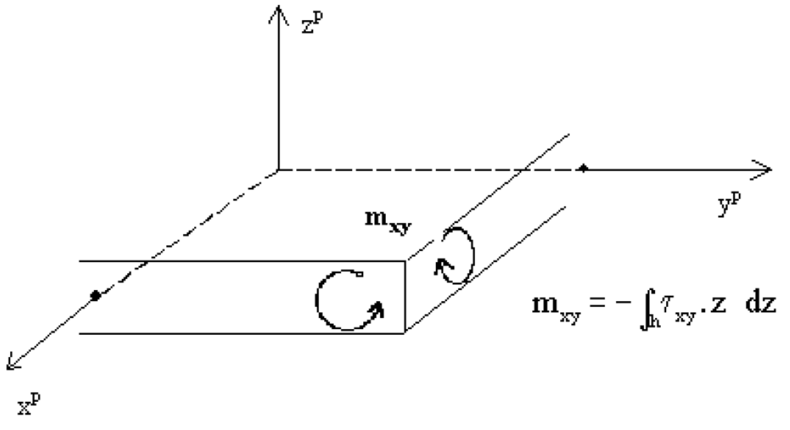
1_ Basic magnitudes = Characteristic values

Bending (plates, shells)

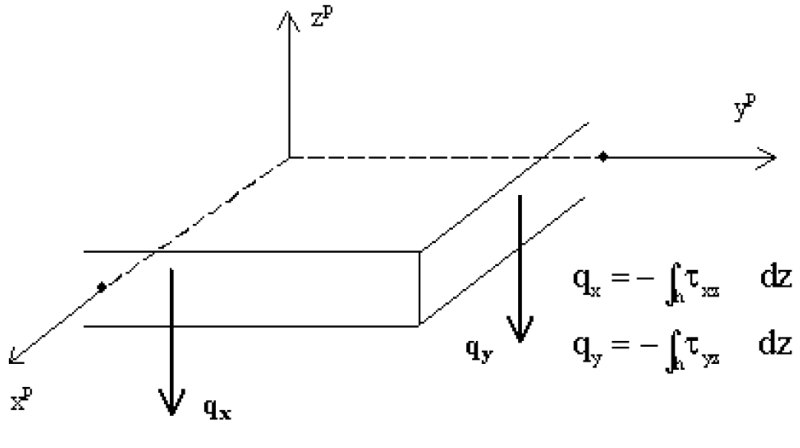
*Bending moments mx, my



*Torsion moment mxy

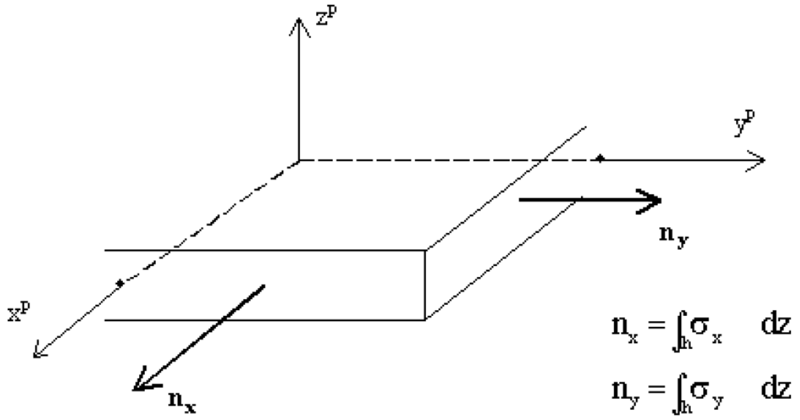


*Shear forces qx, qy (=vx, vy)

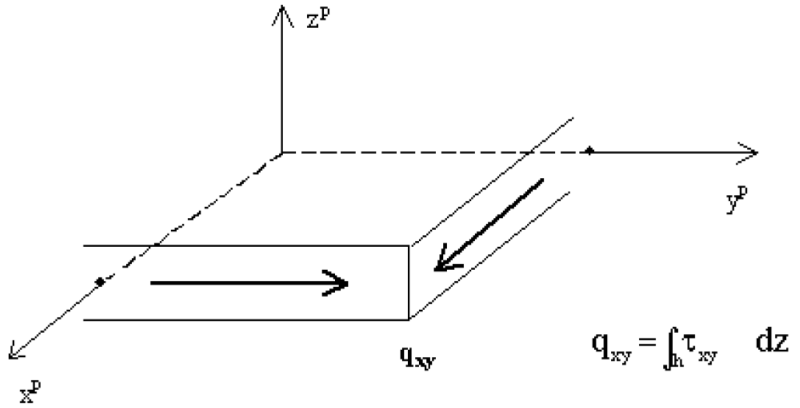


Membrane effects (walls, shells)

*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.

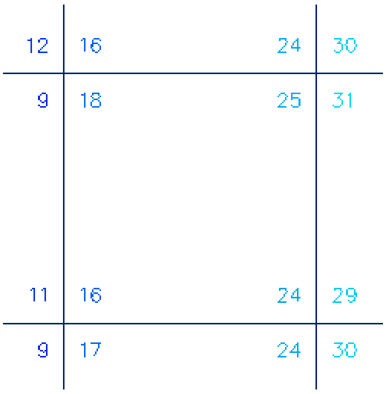
Annex 2: Results in Mesh elements and Mesh nodes → 4 Locations

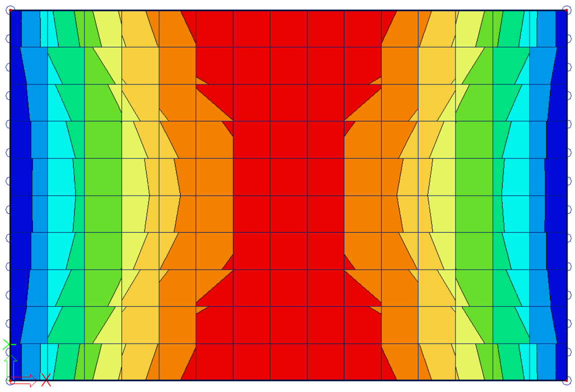
During a calculation in SCIA Engineer, the node deformations and the reactions are calculated exactly (by means of the displacement method). The stresses and internal forces are derived from these magnitudes by means of the assumed basic functions, and are therefore in the Finite Elements Method always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

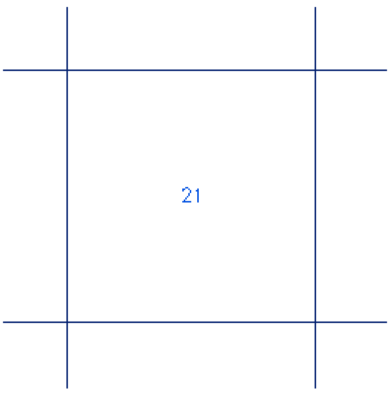
All of the values of the results are taken into account, there is no averaging. In each node are therefore the 4 values of the adjacent mesh elements shown. If these 4 results differ a lot from each other, it is an indication that the chosen mesh size is too large.
This display of results therefore gives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

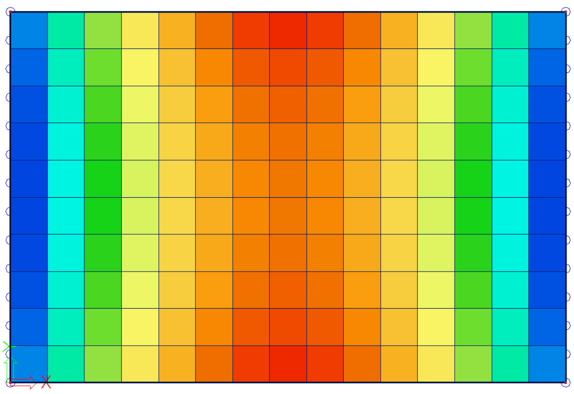




2_ In centres

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.



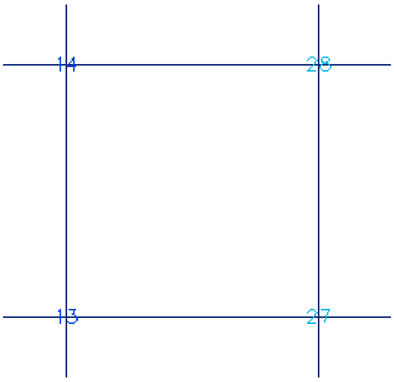


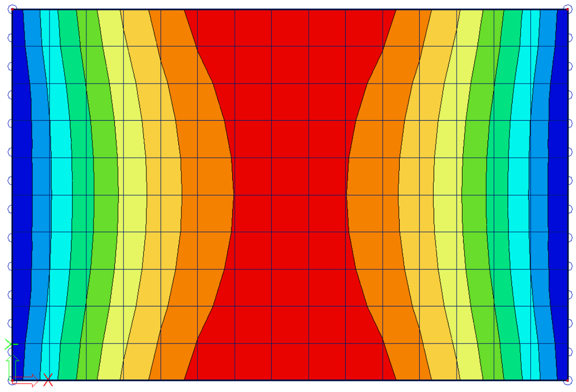
3_ In nodes, average

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:

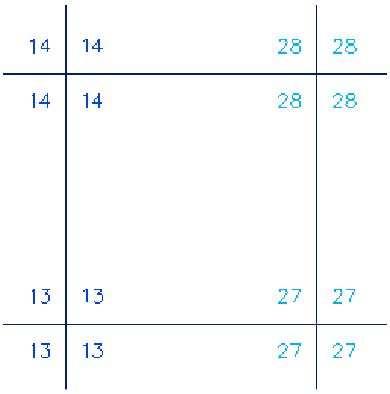
- At the transition between 2D members (plates, walls, shells) with different local axes.
- If a result is really discontinuous, like the shear force at the place of a line support in a plate. The peaks will disappear completely by the averaging of positive and negative shear forces.

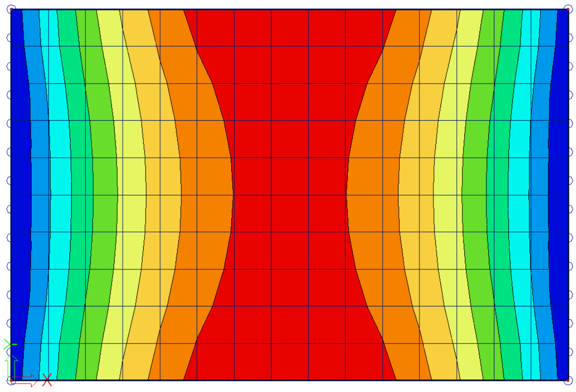




4_ In nodes, average on macro

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’.





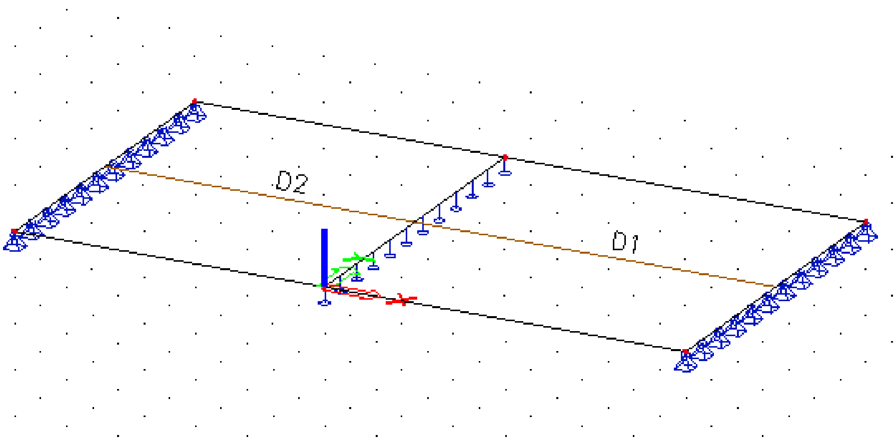
Accuracy of the results

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

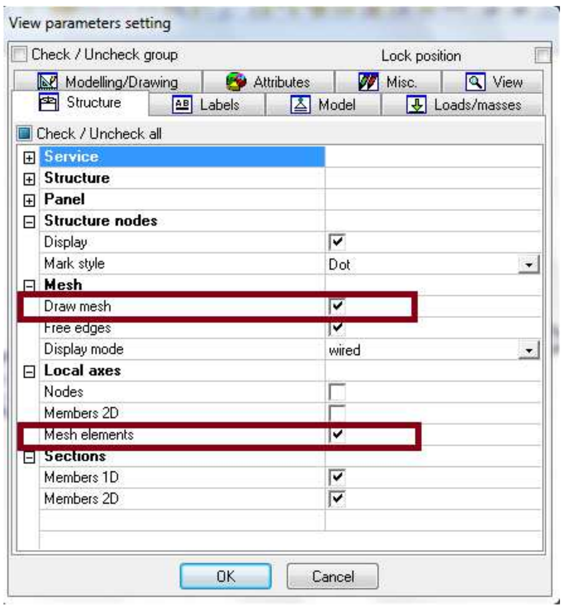
The results for a Finite Element analysis are computed according to the Local Coordinate System (LCS) of each mesh element. As a consequence, these results depend on the way the local axes for mesh elements are defined. A wrong definition of local axes can lead to very misleading results.

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

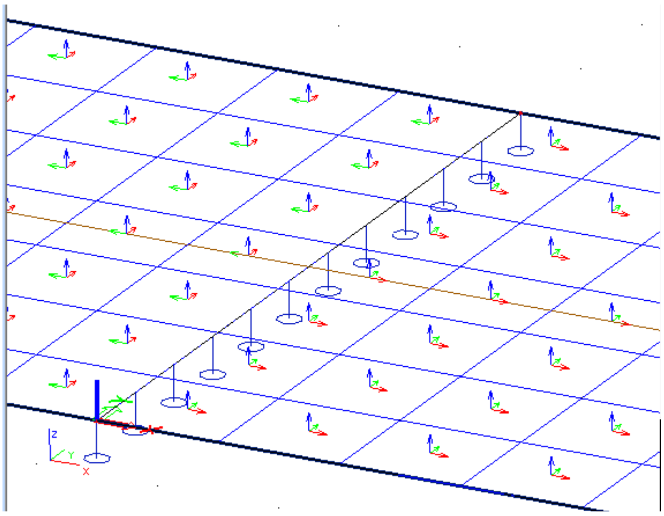


In order to display the mesh local axes, you need first to generate the mesh. You can use the button ‘Mesh generation’  or **Main menu > Calculation, Mesh > Mesh generation**

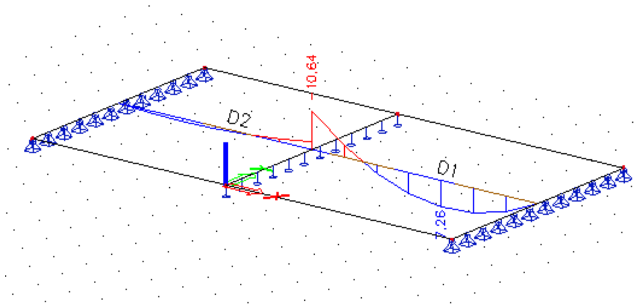
Afterwards, the mesh and local axes can be displayed from ‘**Set View parameters for all**’: right click of the mouse on the screen



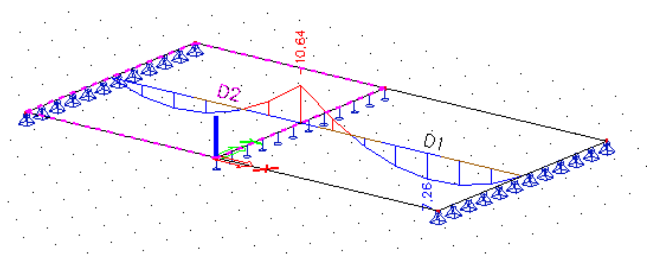
It is clear from the orientation of the axes that the continuity of the moments mx and my cannot be satisfied. The moment mx on D1 corresponds in this case to the moment my in D2.



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

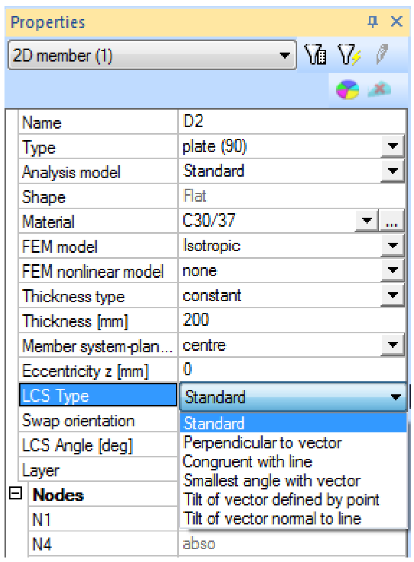


Moment diagram 1



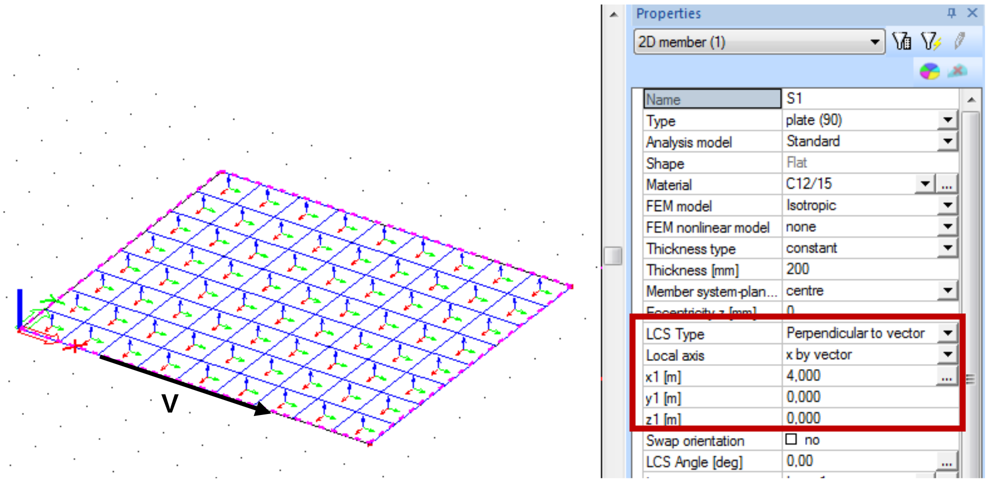
Moment diagram 2

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

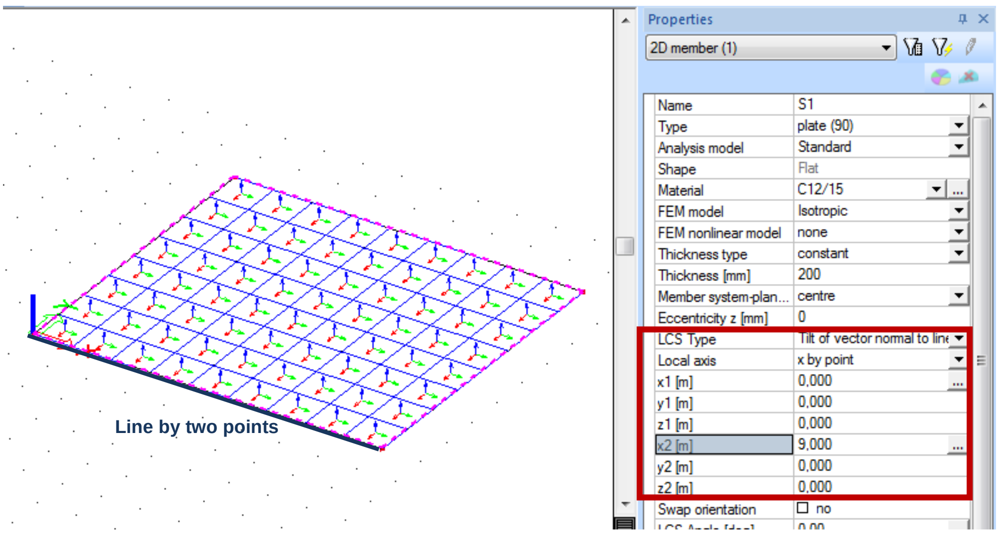
The local axis x(y) is perpendicular to a vector that is defined with its coordinates V(x1 y1 z1).



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

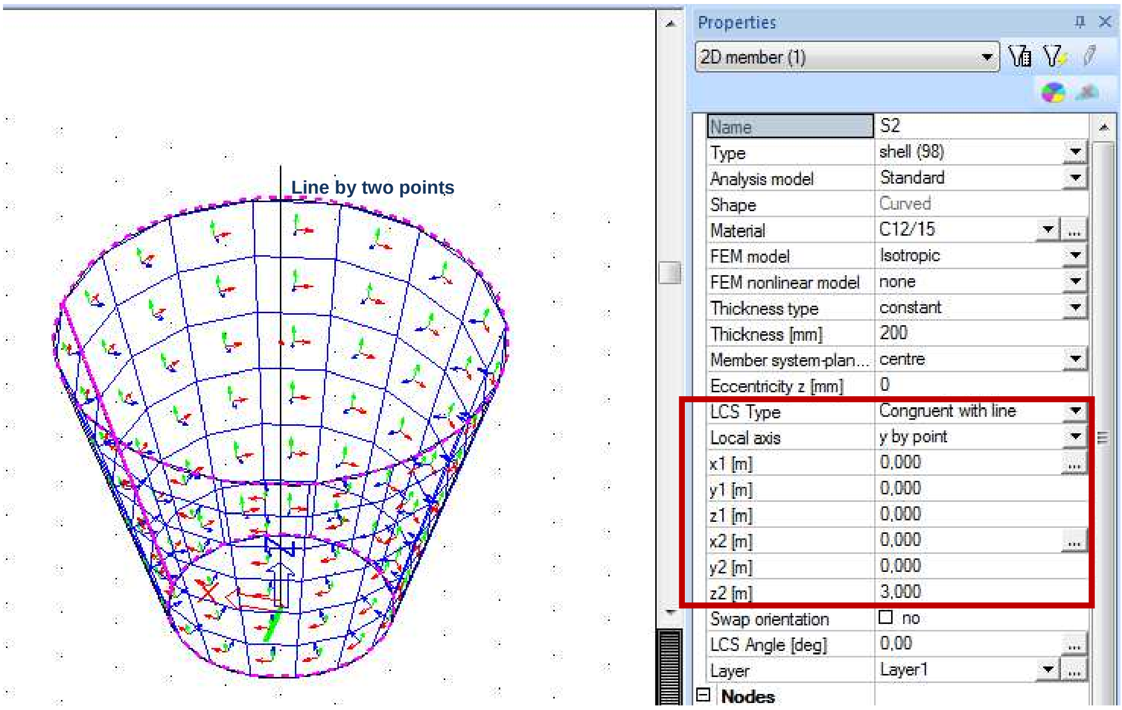
This method is similar to the one above. Instead of introducing a vector, a line is defined between two points. (x1 y1 z1) are the coordinates of the first point and (x2 y2 z2) are the coordinates of the second point. The x(y) axis is perpendicular to the introduce line and points towards it.



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.

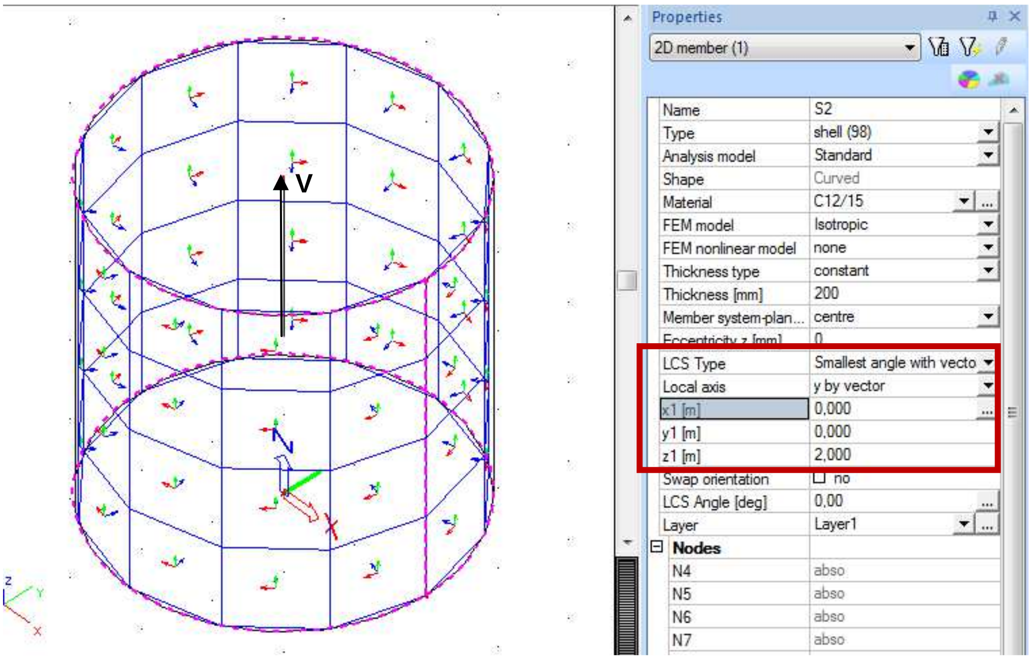
3_ Congruent with line

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



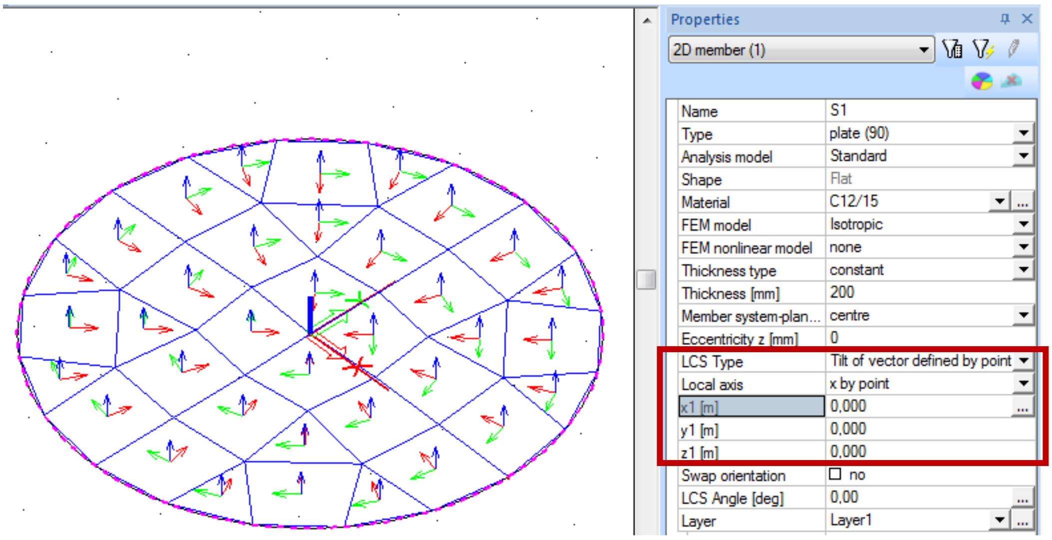
4_ Smallest angle with vector

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

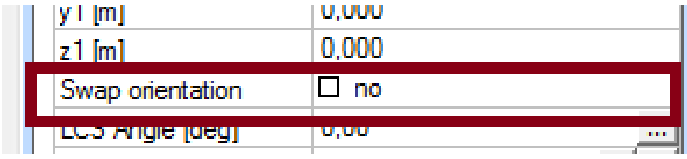


5_ Tilt of vector defined by point

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



2D Reinforcement Concrete Design of Walls acc. to EN 1992-1-1:2004

Benchmark Example on SCIA Engineer NEDIM Performance (v6)

Dipl.-Ing. Eduard Hobst Ph.D., Development Partner (Concrete) SCIA

A. Fundamental Considerations

Introductory note

This benchmark example presents and explains the basic features of the 2D reinforcement concrete design module NEDIM of the program system SCIA Engineer 2013.1. The related Code/Norm on reinforced concrete is the EN 1992-1-1:2004. The analysis model is *Wall*, i.e. plane structure subject to inner membrane forces only.

WALL design model

Wall is the most simple of 4 design models dealt with by NEDIM (*Wall, Plate, Shell, One-Layer*). Since it is restricted to plain shape and subject to membrane forces, represented by the inner forces vector $\{n_x, n_y, n_{xy}\}$, only (no bending), the concrete cover has impact on the Crack Proof only. Because of the transversal symmetry, there is no difference between the reinforcement on either face $+Z_p/–Z_p$. Thus, in Walls NEDIM calculates the total reinforcement (in each reinforcement direction). However, the very distinguishing feat of NEDIM, the processing of general 2/3 direction reinforcement nets is active also in the *Wall* design model. Thus, not only the standard orthogonal reinforcement net but arbitrary skew-angle and three-directional nets are available options.

Special note on compression reinforcement

Since concrete, according to all Norms, does not resist tension stresses (in strength calculations) the main task of reinforcement is to take on tension stresses appearing in a reinforcement concrete cross-section. However, if heavy compression forces act upon a 2D structure, compression reinforcement may be required to support plain concrete. Typically for 2D structures, no statically required compression reinforcement but minimum compression one is needed according to most Norm stipulations. As a fact, none of the known (published) 2D design theories, including the *Baumann*'s theory [1], which was used as the theoretical base of the NEDIM design algorithm, do give an instruction how to design the compression reinforcement. Thus, its design is left to the program user's subsequent hand calculation. The 2D situation is unlike the 1D design, where there is no problem to consider both tension and compression normal forces for reinforcement design.

The NEDIM algorithm has been equipped with very special procedure, enabling the calculation of compression reinforcement in all stress situations as a generalization of the *Baumann*'s algorithm [1]. In elliptic pressure states ($n_{II} < n_I < 0$), there is a consistent solution available, which yields compression design forces to both/ all 3 reinforcement directions, leading directly to either statically required or minimum compression reinforcement.

The hyperbolic pressure state ($n_I > 0$, $n_{II} < 0$) appears as a real challenge, also to the 2D design algorithm. NEDIM offers here a special solution, too, described in some detail in [1] ("Design of Walls", Fig.4). The elementary idea is based upon the *principle of redistribution*: (1) the resistance to compression develops in the concrete continuum in the direction of the 2nd principal force $n_{II} < 0$, no matter if there is laid reinforcement (exactly) in this direction or not; this fact "explains" the failing of all classical 2D design theories, which have to cope with arbitrary reinforcement geometry; (2) the NEDIM algorithm focuses to the *Ultimate Load State* (ULS) as the stress situation where the cross-section may fail due to exhaustion of the resistance of plain concrete to compression. In such a case, another stress distribution takes place in the design point, engaging the existing reinforcement to participate in resisting the pressure force; (3) for this, NEDIM carries out a special inner forces transformation, which may exceed the elementary energetic level described by the *generalized invariant relation* (5) in [1]; however, it represents a virtually possible state of stress. If there is found acceptable security for this state of stress, the structure is considered to be designable with the compression and tension reinforcement corresponding to this state of exploitation.

In this Paper, paragraph C.4 presents numerical processing of a hyperbolic stress case with a consistent solution, i.e. meeting exactly the invariant relation (5) in [1].

References
The numerical calculations presented in this Paper have been based upon the SCIA Engineer projects Wall_Benchmark_EN(0), Wall_Benchmark_EN(1) and Wall_Benchmark_EN(2), which are integral parts of this benchmark document [0]. The fundamental theoretical information on 2D design is presented by the SCIA *Theoretical Background* manual [1] (with advanced references there in).

B. Wall Example – Model Definition and FEM Results

A 2D cantilever member subject to considerable horizontal line load –500 [kN/m] and an additional vertical tip force –100 [kN/m] was chosen to demonstrate the feats of the 2D design algorithm NEDIM under SCIA Engineer:

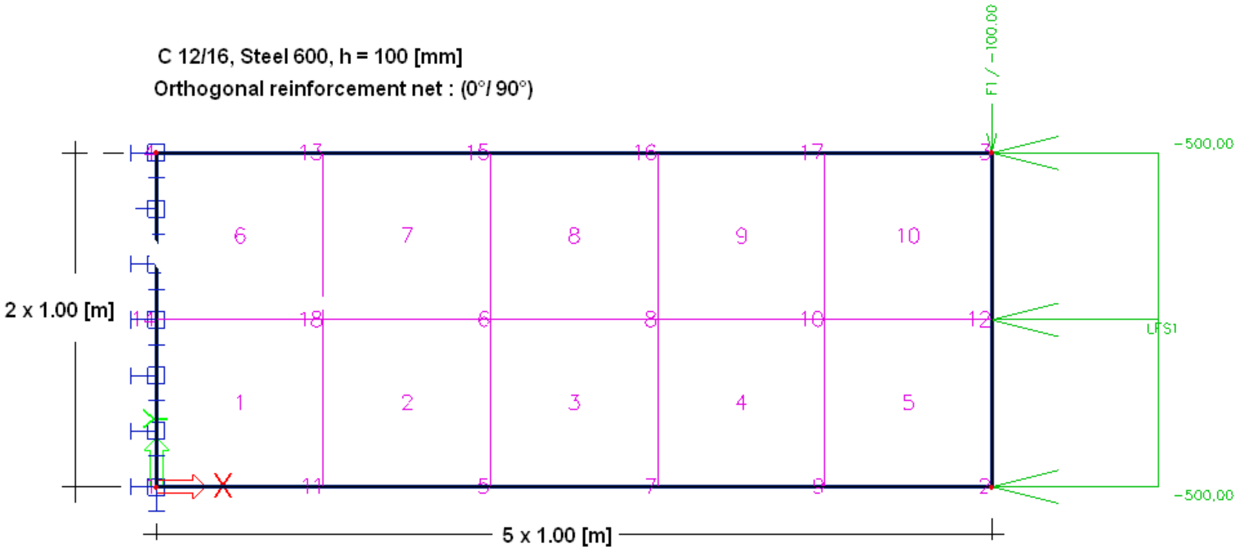


Fig.1 SEN model of Wall – Geometry and FE mesh

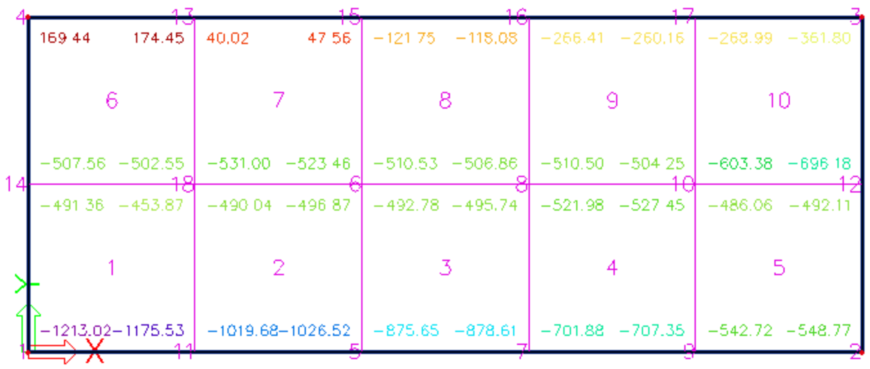


Fig.2 Results of FEM analysis: Inner forces n_x – Direct values in element nodes (LC1)

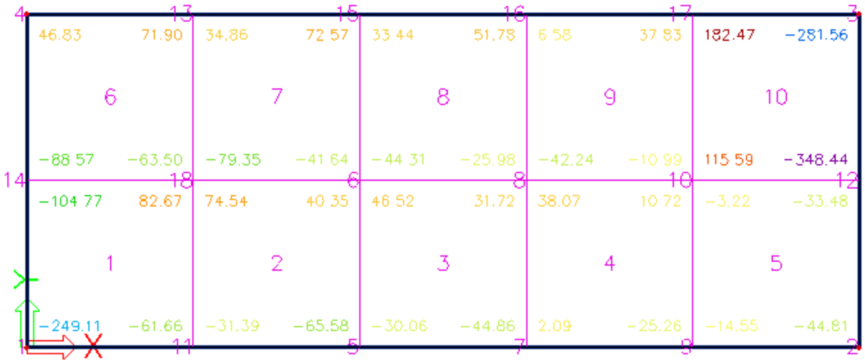


Fig.3 Results of FEM analysis: Inner forces n_y – Direct values in element nodes (LC1)

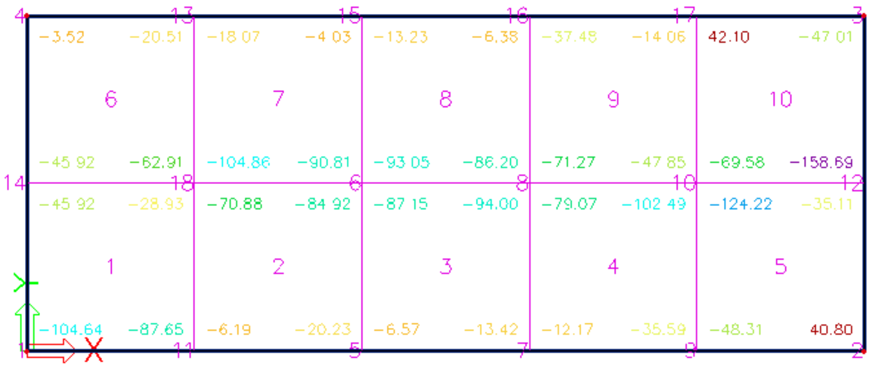


Fig.4 Results of FEM analysis: Inner forces n_{xy} – Direct values in element nodes (LC1)

C. ULS Design – Statically required reinforcement

In this Chapter, pure statically required reinforcement is dealt with by the project **Wall_Benchmark_EN(0)**. To disable the determination of the minimum required reinforcement, which may superpose the statically required values, its specification has to be suppressed on input (see Chapter D, Fig.10, showing the SCIA Engineer *First default* Concrete Setup dialogue window for non-zero minimum reinforcement). One consequence of suppressing the calculation of minimum compression reinforcement is vanishing of compression reinforcement in all design points where the bearing capacity of plain concrete is sufficient to resist compression stress.

The NEDIM design results will be scrutinized in detail in three element nodes, as marked in Fig. 5 and 6, using the special numerical protocol of the NEDIM Test Strategy: **C.1** Elliptic tension design case; **C.2** Elliptic pressure design case; **C.3** Hyperbolic stress state design case.

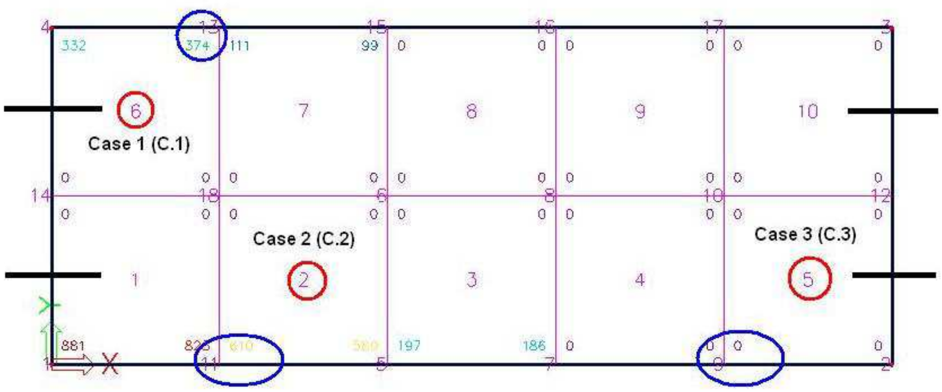


Fig.5 Design results: stat. required horizontal reinforcement a_{s1} [mm²/m] – direct values in nodes

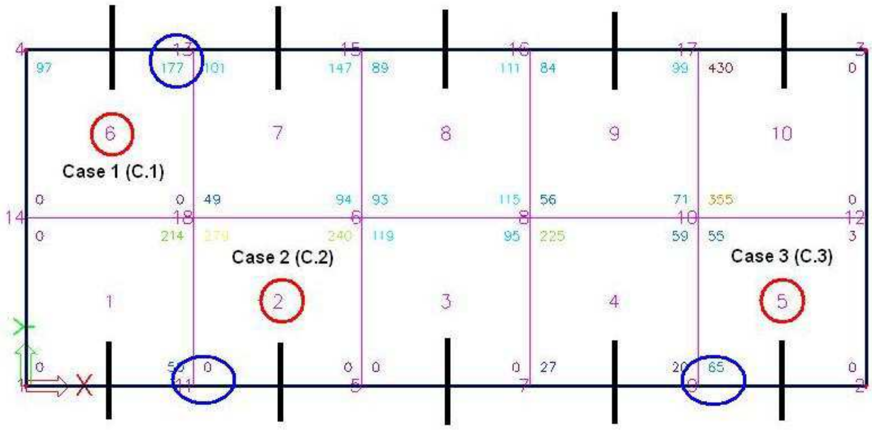


Fig.6 Design results: stat. required vertical reinforcement a_{s2} [mm²/m] – direct values in nodes

C.1 Design Case 1: Statically Required Reinforcement – Elliptic Tension

As seen from Fig.2–4, in element 6, inner node 3 (mesh node 13), both basic membrane forces are positive: $n_x > n_y > 0$, i.e. tension. Thus, an elliptic tension state is expected.

Table 1. **Inner forces** [kN/m] (*Basic/ Principal/ Design*) in **element 6, node 13** (Test Strategy line 1#)

n_x	n_y	n_{xy}	n_I	n_{II}	$\alpha_{I,II}$	n_{1d}	n_{2d}	n_{cd}
174.5	71.9	-20.5	178.4	68.0	-10.9°	195.0	92.4	-41.0

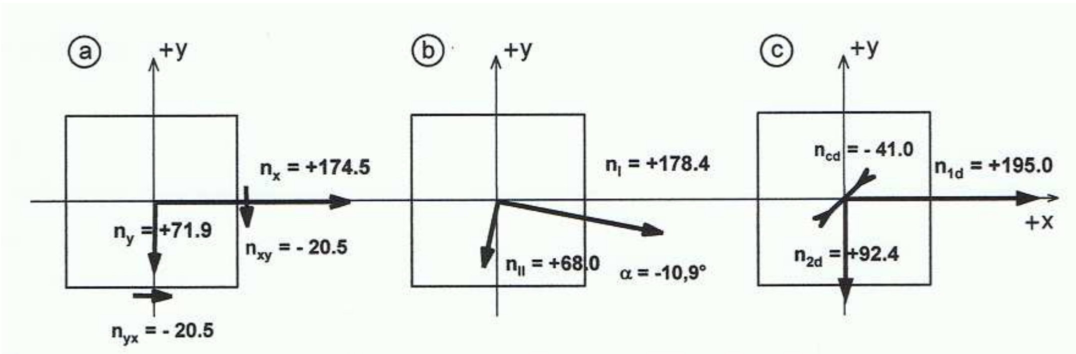


Fig.7 Graphic representation of Table 1 (Case 1 of inner forces transformation)

The generalized invariant relation (5) in [1] is satisfied:

$\boldsymbol{n_{1d}} + \boldsymbol{n_{2d}} + \boldsymbol{n_{cd}} = 195.0 + 92.4 - 41.0 = \mathbf{256.4} \quad \sim \quad \boldsymbol{n_I} + \boldsymbol{n_{II}} = 178.4 + 68.0 = \mathbf{256.4}$ [kN/m]

This is a standard transformation case. The design forces {$\boldsymbol{n_{1d}}$, $\boldsymbol{n_{2d}}$, $\boldsymbol{n_{cd}}$} may be, in analogy to mathematic terminology, considered as *separated result variables* of the 2D design problem, which thus disintegrates into three individual pseudo 1D design cases. The required reinforcement amount is calculated as in two mutually independent 1D members:

$\mathbf{a}_{si,req} = \boldsymbol{n_{id}} / \boldsymbol{f_{yd}}$ **(1)**

$\mathbf{a}_{s1,req} = n_{1d} / f_{yd} = 0.1950/(600/1.15) = 3.74_{10}{-4}$ [m²/m] = **374** [mm²/m] (agreement with Fig.5)

$\mathbf{a}_{s2,req} = n_{2d} / f_{yd} = 0.0924/(600/1.15) = 1.77_{10}{-4}$ [m²/m] = **177** [mm²/m] (agreement with Fig.6)

The virtual concrete strut, which function may be formulated as "stiffening of the reinforcement net against distortion in its plane" is to be checked as plain concrete section with reduced concrete strength [1] (see Fig.11). With the 1st default value of the reduction factor $\boldsymbol{r_{fac}} = 0.80$ (i.e. only 80% of full design strength of concrete, weakened by parallel cracks, may be applied), the resistance check of the virtual strut follows the formula:

$$-n_{cd} \leq r_{fac} \times f_{cd} \times H \qquad (2)$$

with H – full cross-section height (representing the “unit” area of $H \times 1\,[\text{m}]$). From (2) it follows:

$41.0_{10}{-}3$ = **0.041** << $0.80 \times 12/1.5 \times 0.10$ = **0.640** [MN/m]

The check reveals satisfactory bearing capacity reserve of the stiffening concrete.

C.2 Design Case 2: Statically Required Reinforcement – Elliptic Pressure

As seen from Fig.2–4, in element 2, inner node 1 (mesh node 11), both basic membrane normal forces are negative: $n_x < 0$ and $n_y < 0$, i.e. compression. An elliptic compression state is expected. Table 2 shows that it is true also in the transformed state: $n_{1d} < 0$; $n_{2d} < 0$.

Table 2. **Inner forces** [kN/m] (*Basic/ Principal/ Design*) in **element 2, node 11** (Test Strategy line 2#)

n_x	n_y	n_{xy}	n_I	n_{II}	$\alpha_{I,II}$	n_{1d}	n_{2d}	n_{cd}
-1019.7	-31.4	-6.2	-31.3	-1019.7	90.4°	-1013.5	-25.2	-12.4

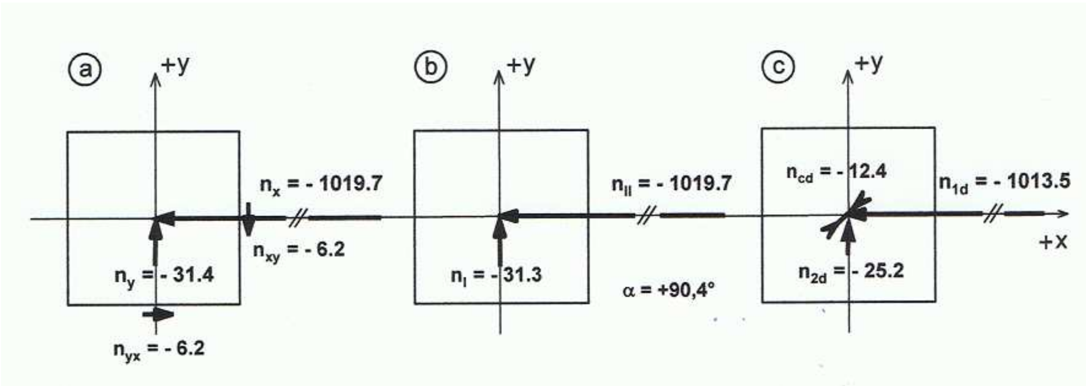


Fig.8 Graphic representation of Table 2 (Case 2 of inner forces transformation)

The generalized invariant relation (5) [1] is satisfied ("numerical" precision):

$\boldsymbol{n_{1d}} + \boldsymbol{n_{2d}} + \boldsymbol{n_{cd}} = -1013.5 - 25.2 - 12.4 = \mathbf{1051.1} \quad \sim \quad \boldsymbol{n_I} + \boldsymbol{n_{II}} = -31.3 - 1019.7 = \mathbf{1051.0}$ [kN/m]

This is a special NEDIM transformation case. The required reinforcement amount is calculated as in two mutually independent 1D members:

$$\mathbf{a}_{si,req} = (-n_{id} - H\,f_{cd}) / f'_{ycd} \quad \text{with} \quad f'_{ycd} = \min(f_{ycd}, E_s\,\varepsilon_{c1}) \qquad (3)$$

In (3) f_{ycd} is steel strength in compression (1st input default: $f_{ycd} = f_{yd}$); E_s – Young’s modulus; ε_{c1} – concrete yield point (bilinear stress-strain concrete diagram specified, $\varepsilon_{c1} = 0.175$ %).
Hint: the adapted compression steel strength f'_{ycd} respects the fact that concrete under centric pressure should not be allowed for higher than yield strain; thus, higher class reinforcement steel does usually not attain its yield point, i.e.) $f_{ycd} \leq E_s\,\varepsilon_{c1}$ in most cases. From (3) it follows:

$f'_{ycd} = \min(600.0/1.15, 200000.0 \times 0.00175) = 350.0$ [MPa]

$\mathbf{a}_{s1,req} = (1.0135 - 0.10 \times 12.0/1.5)/350.0 = 6.10_{10}{-}4$ [m²/m] = **610** [mm²/m] (agreement with Fig.5)

$\mathbf{a}_{s2,req} = (0.0025 - 0.10 \times 12.0/1.5)/350.0 < 0 \quad \rightarrow \quad \mathbf{a}_{s2,req} = \mathbf{0}$ (agreement with Fig.6)

Obviously, the virtual concrete strut possesses sufficient resistance.

C.3 Design Case 3: Statically Required Reinforcement – Hyperbolic Stress

As seen from Fig.2–4, in element 5, inner node 1 (mesh node 9), both basic membrane normal forces are negative: $n_x < n_y < 0$, i.e. compression. An elliptic compression state is thus expected like in Case 2. However, the “mixed” membrane component n_{xy}, which generally imposes a tension effect upon both reinforcement directions *x*, *y*, reverts the elliptic pressure state of the homogeneous continuum to a virtual hyperbolic stress state, as Table 3 shows.

Table 3. **Inner forces** [kN/m] (*Basic/ Principal/ Design*) in **element 5, node 9** (Test Strategy line 3#)

n_x	n_y	n_{xy}	n_I	n_{II}	$\alpha_{I,II}$	n_{1d}	n_{2d}	n_{cd}
-542.7	-14.6	-48.3	-10.2	-547.1	95.2°	-494.4	+33.8	-96.6

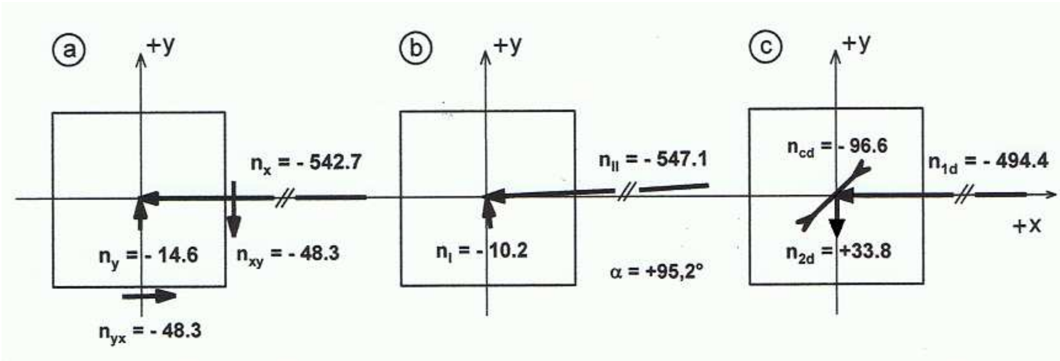


Fig.9 Graphic representation of Table 3 (Case 3 of inner forces transformation)

The generalized invariant relation (5) [1] is satisfied ("numerical" precision):

$\boldsymbol{n_{1d}} + \boldsymbol{n_{2d}} + \boldsymbol{n_{cd}} = \mathbf{-}494.4 + \mathbf{33.8_v} - 96.6 = \mathbf{557.2}$ **~** $\boldsymbol{n_I} + \boldsymbol{n_{II}} = \mathbf{-}10.2 - 547.1 = \mathbf{557.3}$ [kN/m]

This is a very special NEDIM transformation case. *Baumann* [1] and all other transformation theories would yield the 2nd principal force $n_{II} = -547.1$ as the design result, assigning it to the plain concrete as virtual strut force; no reinforcement would be designed. Effectively, plane concrete design would have taken place.

The required reinforcement amount is calculated once for compression, once for tension, i.e. according to the formulae (3) and (1), respectively:

$\mathbf{a}_{s1,req} = (0.4944 - 0.10 \times 12.0/1.5)/350.0 < 0 \quad \rightarrow \quad \mathbf{a}_{s1,req} = \mathbf{0}$ (agreement with Fig.5)

$\mathbf{a}_{s2,req} = 0.0338/(600/1.15) = 6.48_{10}-4$ [m²/m] $= 65$ [mm²/m] (agreement with Fig.6)

The *tension* force $\boldsymbol{n_{2d}} = +\mathbf{33.8_v}$ has virtual character (trailing symbol “v”). Under SLS, the reinforced concrete continuum would act as plain concrete without participation of the reinforcement, whereas in ULS, a redistribution of inner forces could take place, evoking the tension in steel as presented above. As a fact, this statement holds, in cases with required compression reinforcement. Obviously, in cases of minor exploitation, like this one, no redistribution would take place. However, this special design solution makes it generally possible, to assign minimum compression reinforcement according to the Code stipulations – see Table 4.

Hint: the virtual reinforcement amount (here $\mathbf{a}_{s2,req}$) is designed, consequently, as **tension reinforcement** !!!

D. Required Minimum Reinforcement

Numerical results and screen copies of this Chapter were obtained by the project **Wall_Benchmark_EN(1)**.

In this Chapter, the effect of activated minimum reinforcement specification according to NEDIM 1st default input, as shown in Fig.10, is dealt with. As a fact, the user may modify the settings, but accepting them, legal solution according to the Norm is obtained. The position of the virtual strut input in the Setup dialogue is shown in Fig.11.

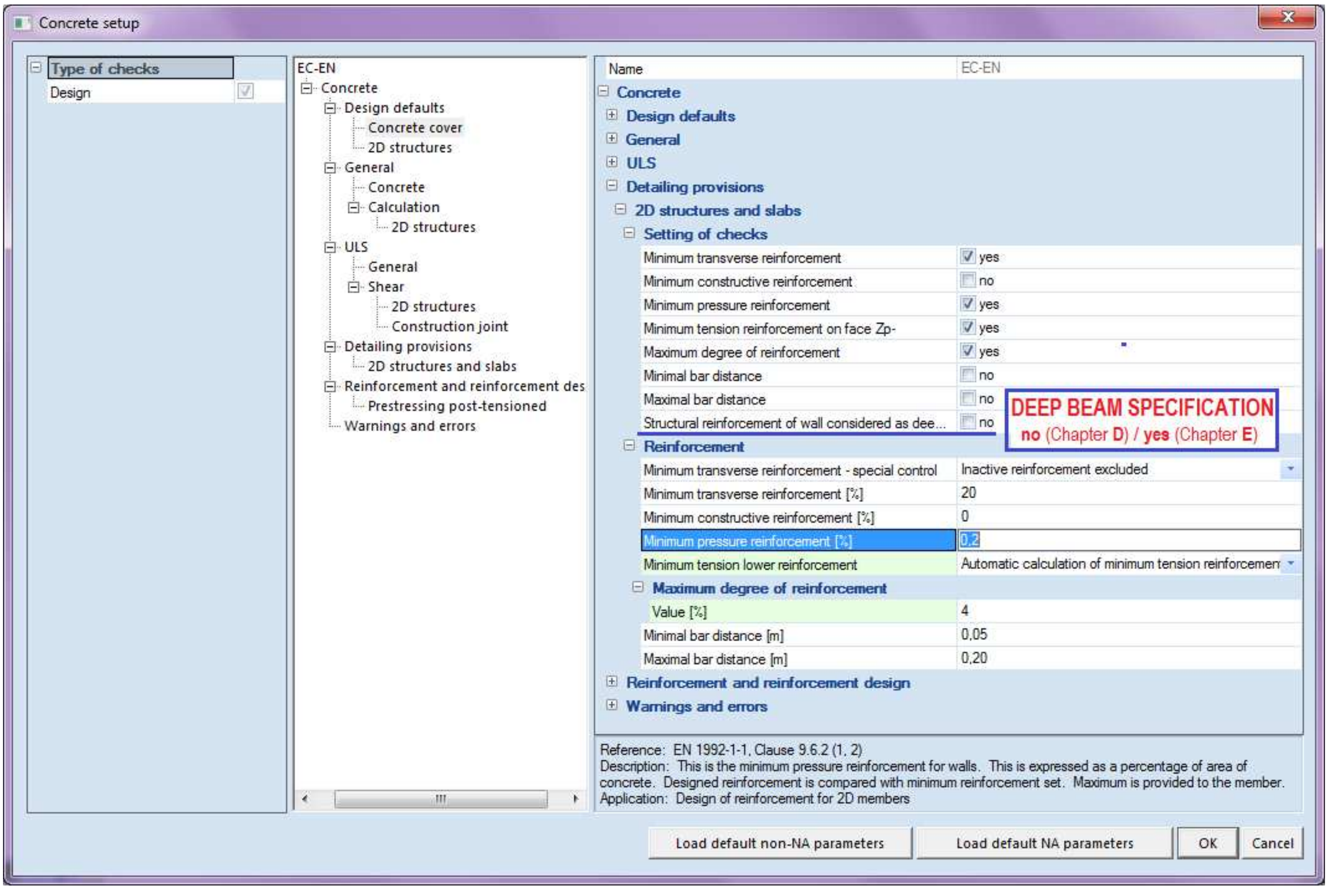


Fig.10 Concrete Setup > Detailing provisions > 2D structures – *Minimum reinforcement*

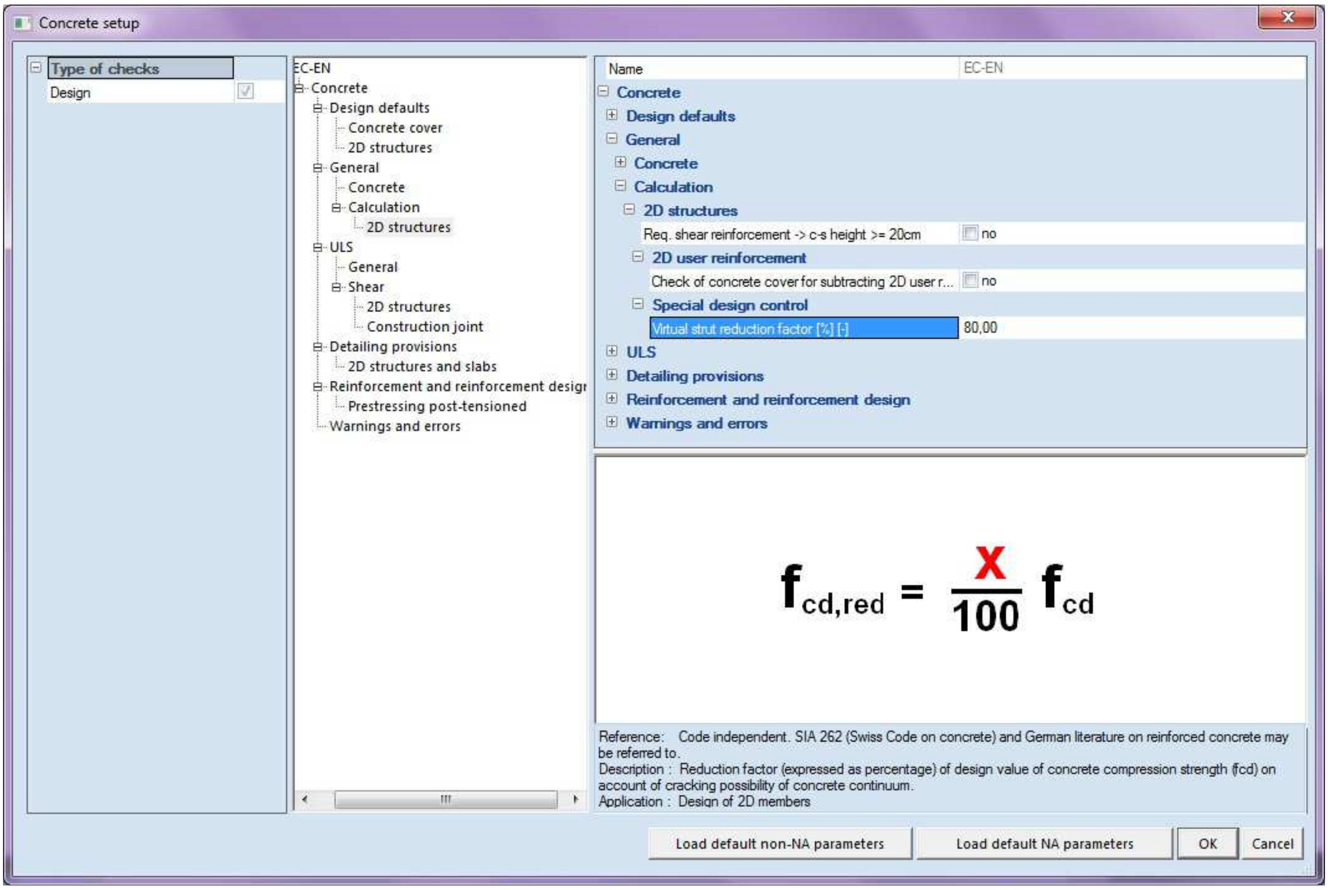


Fig.11 Concrete Setup > Calculation > 2D structures > Special design control – ***Virtual strut reduction***

The minimum tension reinforcement is stipulated by §9.2.1.1, formula (9.1N). In NEDIM terms:

$$\mathbf{a}_{st,min} = \boldsymbol{max}\,(\mathbf{0.26}\,\boldsymbol{f}_{ctm}/\boldsymbol{f}_{yk}\,\mathbf{,\ 0.0013}) \times \boldsymbol{d} \quad [\mathrm{m^2/m}] \qquad \mathbf{(4)}$$

With $f_{ctm} = 1.6$, $f_{yk} = 600$ [MPa] and $d = H - a_1 = 0.100 - (0.030{+}0.10/2) = 0.065$ [m] (a_1 is the effective static height: $a_1 = c_1 + \phi/2$) of the outer reinforcement layer, for consistency with a corresponding bending case), the 2nd term in (4) is decisive, thus $a_{st,min} = 0.0013\,d = 0.00013\text{x}0.065 = 0.0000845$ [m²/m]. However, in Walls the total reinforcement is presented; thus, the resulting reinforcement is double as much:

$\mathbf{a}_{st,min} = 2 \times 0.0013\,d = 0.000169$ [m²/m] = **169** [mm²/m],

as displayed in Fig.13 in all nodes where $a_{st,req} < a_{st,min}$.

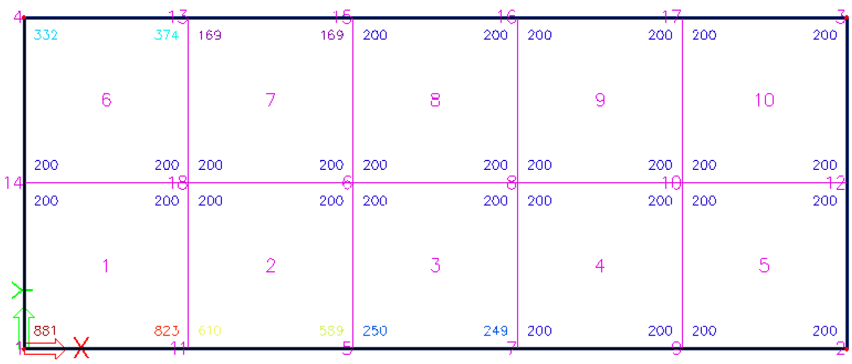


Fig.12 Horizontal reinforcement $a_{s1,req+min}$ [mm²/m] – Direct values in nodes

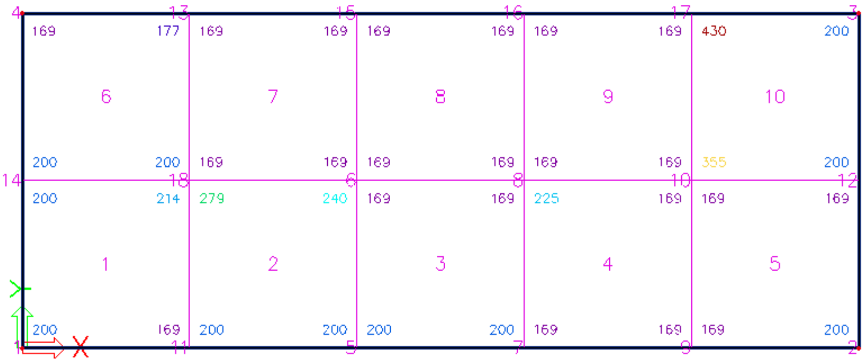


Fig.13 Vertical reinforcement $a_{s2,req+min}$ [mm²/m] – Direct values in nodes

The minimum compression reinforcement is governed by §9.6.2. In 2D structures it is simply specified as 0.2% of gross cross-section. The corresponding formula in NEDIM is $a_{sc,min}$ = 0,002 H [m²/m]. Hence, in Fig.12 and Fig.13 the value of

$\boldsymbol{a_{sc,min}} = \mathbf{200}$ [mm²/m],

prevails since there is low intensity normal force in most element nodes active. Only in element nodes at the lower edge, the statically required values of compression reinforcement exceed the minimum values (compare with Fig.5).

E. Minimum Structural Reinforcement of Deep Beams

Numerical results and screen copies of this Chapter were obtained by the project **Wall_Benchmark_EN(2)**.

EN 1992-1-1:2004, §9.7 deals with (minimum) structural reinforcement of so called Deep Beams. NEDIM controls this assignment by the pertinent input option in Concrete Setup > Detailing provisions > 2D structures, as shown in Fig.10; the checkbox has then to be activated (yes). The Deep Beam control option is set inactive in the project variants (0) and (1). On the other side, the specification of minimum reinforcement (tension and compression) from the variant (1) is maintained in variant (2). Thus, the reinforcement results obtained here constitute the absolute "constructive reinforcement envelope". The following result displays of $a_{s1,sup}$ and $a_{s2,sup}$ in Fig.14 and Fig.15 the original values of $a_{s,req}$ which have not been exceeded by both minimum/constructive reinforcement controls are marked by encircling.

The minimum constructive reinforcement of *Deep Beams* according to §9.7(1) takes on in NEDIM terms the following shape:

$$\mathbf{a}_{s,DBmin} = \mathbf{max}(\mathbf{0.001}\boldsymbol{H}, \mathbf{0.00015}) \quad [\text{m}^2/\text{m}] \qquad \mathbf{(5)}$$

Since in Walls the total reinforcement is showed, the resulting constructive reinforcement is double as much.

Numerically:

$\mathbf{a}_{s,DBmin} = 2 \times \max(0.001 \times 0.10, 0.00015) = 0.0003$ [m²/m] $= \mathbf{300}$ [mm²/m],

This is the value displayed in Fig.14 and Fig.15 in most nodes where $a_{s,req+min} < a_{s,DBmin}$.

Following Table 4 compares the reinforcement designed at the three discussed design stages. *Green* shadowing marks the stage *Required* when it is superior to *Minimum* and *DB* reinforcement. *Blue* and *Red* mark the stages *Minimum* and *DB* exceeding the preceding stage(s). Pressure reinforcement values are marked with trailing "*".

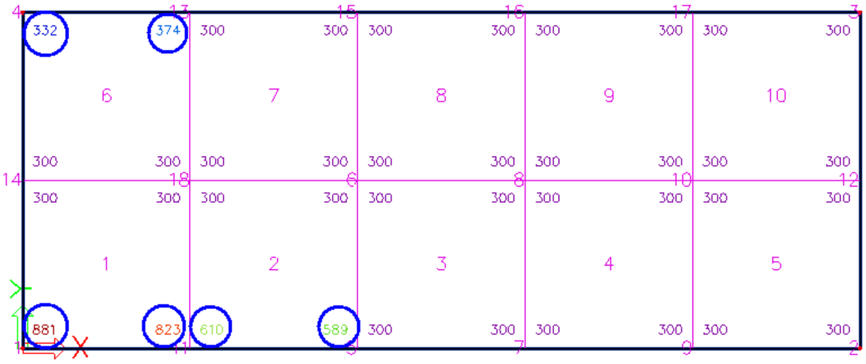


Fig.14 Result envelope: horizontal reinforcement $a_{s1,sup}$ [mm²/m] – Direct values in nodes

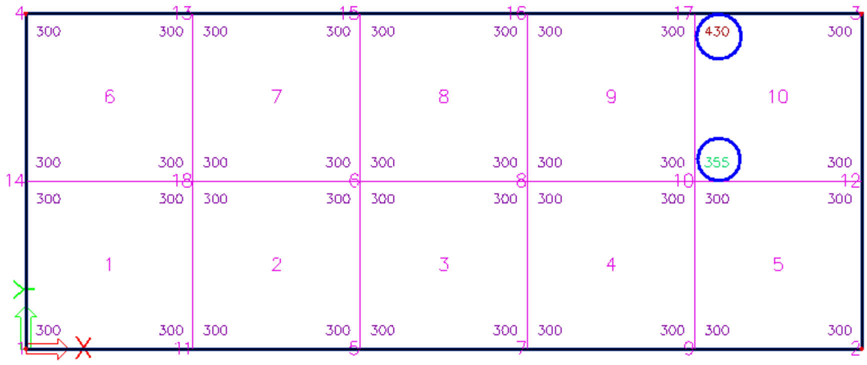


Fig.15 Result envelope: vertical reinforcement $a_{s2,sup}$ [mm²/m] – Direct values in nodes

Table 4. Comparison of the reinforcement stages in investigated design points 6/13, 2/11 and 5/9

Elem/Node	Horizontal reinforcement (1) [mm²/m]			Vertical reinforcement (2) [mm²/m]		
	$a_{s1,req}$	$a_{s1,req+min}$	$a_{s1,sup}$	$a_{s2,req}$	$a_{s2,req+min}$	$a_{s2,sup}$
6/13	374	374	374	177	177	300
2/11	610*	610*	610*	0	200*	300
5/9	0	200*	300	65	169	300

REFERENCES

[0] Hobst, Ed.: Project files >Wall_Benchmark_EN(0).esa<, >Wall_Benchmark_EN(1).esa< and >Wall_Benchmark_EN(2).esa<, created under SCIA Engineer V13.1.47 and NEDIM V13.0.0.0. Distributed with this Paper as its integral part.

[1] Hobst, Ed.: *SCIA Engineer Reinforced Concrete Design of 2D Structures*. *Theoretical Background*. Ingenieurbüro Dr. Eduard Hobst, Nürnberg 2013, Pages I-V+1-84 (89)

Nürnberg, 1 December 2013

Dr. Eduard Hobst

Development partner (concrete) SCIA

(6th release)

2D Reinforcement Concrete Design of Plates acc. to EN 1992-1-1:2004

Benchmark Example on SCIA Engineer NEDIM Performance (v6)

Dipl.-Ing. Eduard Hobst Ph.D., Development Partner (Concrete) SCIA

A. Fundamental Considerations

Introductory note

This benchmark example presents and explains the basic features of the 2D reinforcement concrete design module NEDIM of the program system SCIA Engineer (SEN). The related Code/Norm on reinforced concrete is the Novella of EC2 – EN 1992-1-1:2004. The structural/design model is PLATE, i.e. plane structure subject to pure bending.

PLATE design model

Plate is one of 4 design models supported by SEN/NEDIM (*Wall, Plate, Shell, One-Layer*). It is restricted to pure bending, represented by the generalised force vector $\{m_x, m_y, m_{xy}, v_x, v_y\}$ (membrane forces being absent by definition). The very distinguishing feat of NEDIM – the processing of general 2/3 direction reinforcement nets – is active also in the *Plate* model. Thus, not only the standard orthogonal reinforcement net but arbitrary skew-angle and three-directional nets are possible options of the *Plate* design model of NEDIM. The position of each of 2/3 specifiable reinforcement courses at either face $+Z_p/-Z_p$ of the cross-section is of fundamental meaning for the design, i.e. the concrete cover has not only impact on the Crack Proof.

Special note on compression reinforcement

Since structural concrete, according to all known Norms, does not resist tension stresses (in strength calculations) the main task of reinforcement is to resist tension stresses appearing in the tension zone of a reinforcement concrete cross-section under bending. However, if pronounced bending moments act upon a plate, compression reinforcement may be statically required to strengthen the concrete compression zone. On the other part, no minimum compression reinforcement is required with pure bending by any of the known Norms! NEDIM imposes a restriction to the design of compression reinforcement: the reinforcement directions (and their number, i.e. 2 or 3) at the faces $+Z_p/-Z_p$ must be pair-wise congruent; however, not necessarily in the same specification order. The position of each reinforcement course within the cross-section is checked in order to establish its state of strain: courses lying deeply in the cross-section might not have reached the yield strain; the required compression reinforcement is then augmented appropriately.

The hyperbolic bending ($m_I > 0$, $m_{II} < 0$) in plates does not constitute such a challenge to the 2D design algorithm as in other models, which involve the membrane forces. This is, simply speaking, due to the fact that no *minimum* compression reinforcement is required in plates. Thus, it is in every way a standard solution, when in plates under the elliptic stress state ($m_I \geq m_{II} > 0$) one face is designed without defined reinforcement. However, there are also in plates distinguishing theoretical and algorithmic features of the hyperbolic state of stress, which have been dealt with great thoroughness [1].

References

The numerical calculations presented in this Paper have been based upon the SCIA Engineer projects Plate_Benchmark_EN(0), Plate_Benchmark_EN(1) and Plate_Benchmark_EN(2), which are integral parts of this benchmark document [0]. The fundamental theoretical information on 2D design is presented by the SCIA *Theoretical Background* manual [1]; in [1] advanced references may be found and examined.

B. Plate Example – Model Definition and FEM Results

A quadratic plate supported along all 4 edges, subject to a single Load Case consisting of plane load $g = -3.5$ and $p = -10.0$ kN/m² and reinforced by congruent orthogonal nets 0°/90° at both faces (in this basic variant) has been defined to demonstrate the feats and design results of the 2D design algorithm NEDIM under SCIA Engineer. Since the system is double-symmetric, only the “upper/left” quarter of plate’s model will be processed here by introducing the appropriate symmetry boundary conditions on the central symmetry axes (Fig.1).

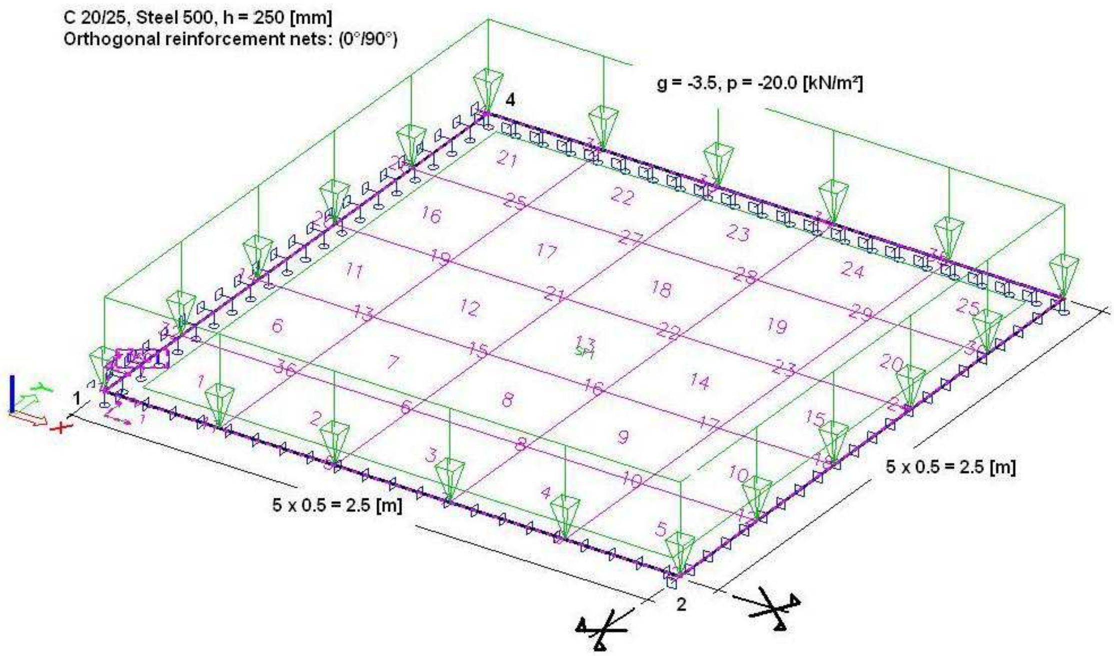


Fig.1 SEN model of Plate – Geometry, FE mesh, Loads

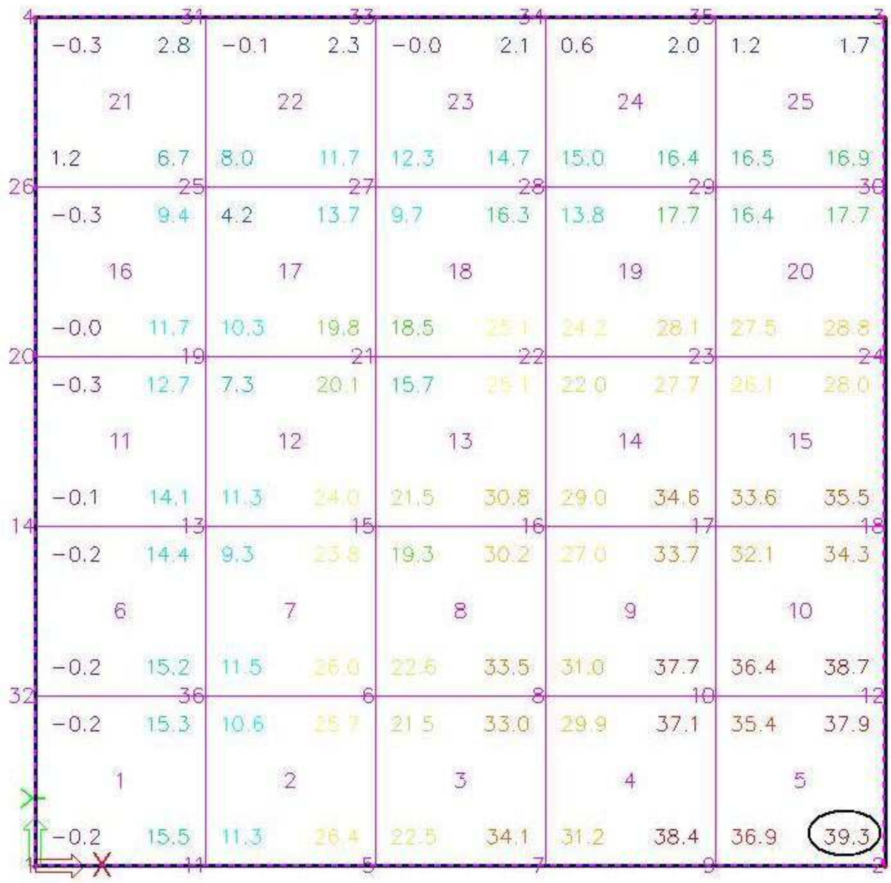


Fig.2 Results of FEM analysis: Inner moments $m_{y,max}$ – Direct values in element nodes (ULS)

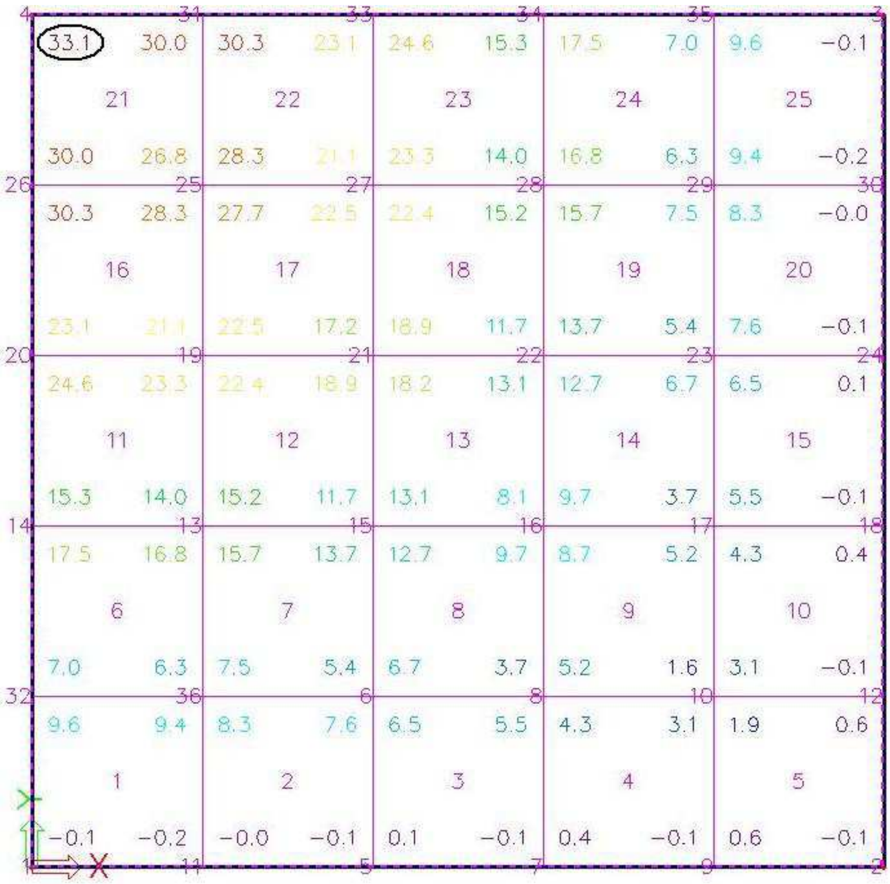


Fig.3 Results of FEM analysis: Inner moments $m_{xy,max}$ – Direct values in element nodes (ULS)

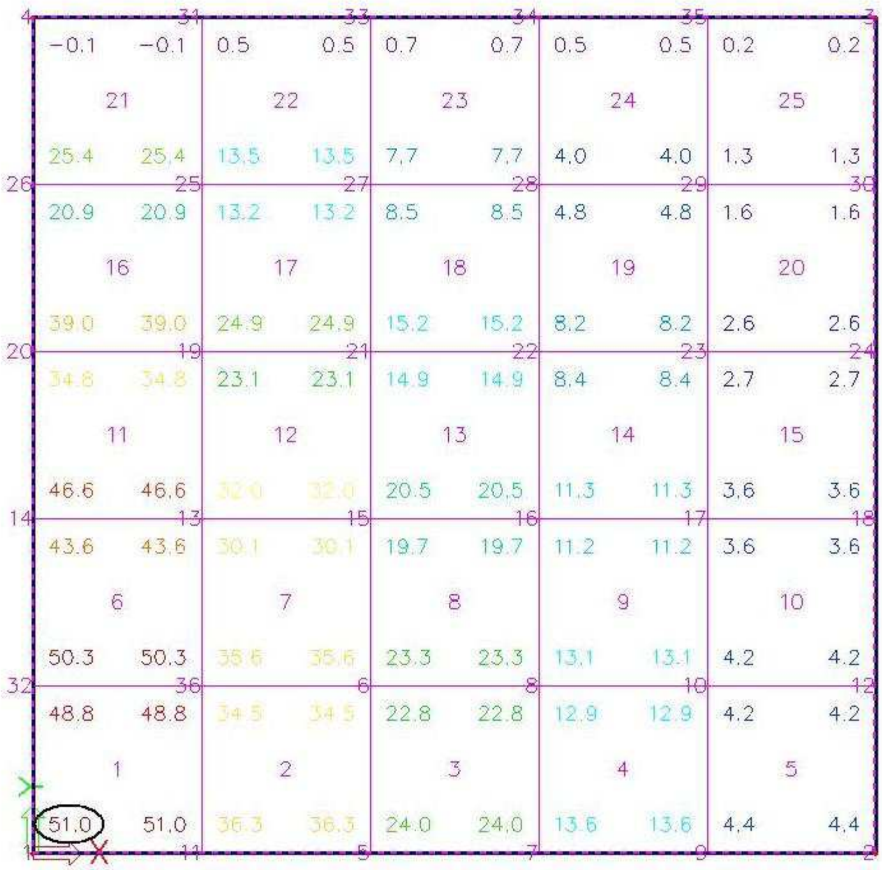


Fig.4 Results of FEM analysis: Inner shear forces v_x – Direct values in element nodes (ULS)

Concluding notes on inner forces

(1) Due to system symmetry, the inner bending moments m_x and m_y are mutually exchangeable when rotated by 90° in the plate plane. However, due to "numerical pollution" of the FEM approximation, m_x and m_y are not exactly equal at central node 2, since they belong to the inner element node. However, in the output mode "mean values in mesh nodes" the symmetry is maintained, since here all adjacent inner nodes contribute.

(2) The statement of (1) is true also for the inner shear forces v_x and v_y. However, since shear force is an "anti-symmetrical" quantity, the sign can change.

(3) A comparison of Fig.2 and 3 confirms the well known fact that the intensity of bending in the centre and in the corner of a (simply supported) quadratic plate is of about the same quantity. There may be found other ratios when looking for tabled values in literature, but the differences are due to different plate solutions applied.

(4) The shear force v_x in Fig.4 is not the *Kirchoff*'s combined "shear reaction" ($v_n + \partial m_{xy}/\partial t$); thus, $v_x \rightarrow 0$ in the corner(s). By fixing the radial rotation of the cross-section perpendiculars, as it is obvious from the boundary conditions representation in Fig.1, the torsion moment singular effects upon the function of the tangential shear force v_t along the edges (v_x – along the edges y = const; v_y – along x = const), have been eliminated from the FEM solution. Thus, neither the well-known *Kirchhoff*'s singular corner force R is not explicitly present in the model. Refer to [1] for detailed description of this phenomenon, having already caused so many irritations on the hotline.

C. ULS Design – Statically required reinforcement

In this Chapter, pure stat. required reinforcement is dealt with by the variant project **Plate_Benchmark_EN(0)**. To disable the determination of the minimum required reinforcement, which might superpose the statically required values, its specification is suppressed on input for purpose of this Chapter. As shown in Fig.5, the control options "Minimum tension reinforcement" are unchecked for both faces. The options "Minimum transverse reinforcement" and "Maximum degree of reinforcement" are important control values defaulted to the Norm stipulations. The non-standard option "Maximum percentage in pressure bending zone", which limits the amount of compression reinforcement (see Chapter A), relates the resisting force in the compression reinforcement to the concrete force activated in the compression bending zone of concrete cross-section. The 1st default value of 50% is borrowed from SIA 162, §3.24.16; EN 1992-1-1:2004 does not stipulate this kind of control! See also the alternative control as described in Chapter D, Fig.12.

Fig.5 Input dialogue window – max/min tension reinforcement set inactive

The NEDIM design results will be scrutinized in detail in three element nodes, as marked in Fig.6 and 7, using the special numerical protocol of the NEDIM Test Strategy: **C.1** Elliptic stress design Case 1; **C.2** Hyperbolic stress state design Case 2; **C.3** Shear Proof Case 3.

The specification data of the reinforcement and cross-section geometry are defined in the input window "Data slab concrete" as shown in Fig.6. For this example, identical orthogonal reinforcement at both faces is specified. The outer reinforcement courses go parallel to X_p, whereas the inner courses are parallel to Y_p. All reinforcement bar diameters are $\phi = 10$ mm; the concrete cover is $c = 30$ mm, the bars crossing in contact. These data are necessary from begin with to estimate the static heights of individual reinforcement courses.

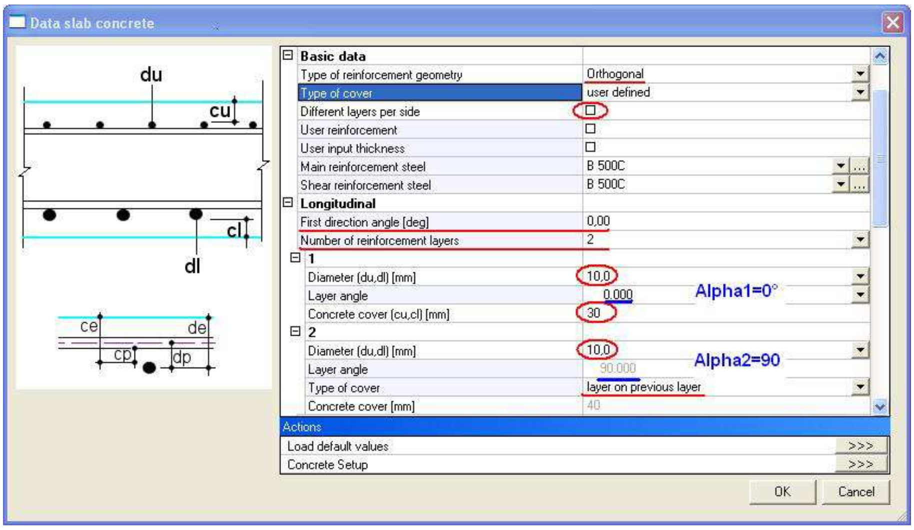


Fig.6 Concrete data input window – reinforcement and cross-section geometry

The reinforcement courses pertaining to $+Z_p$/$-Z_p$ are distinguished by the indices +/–. The natural order of courses at a face is marked by indices 1, 2 – from the outermost to innermost course within the cross-section. In the following numerical analysis, the first reinforcement courses will be discussed with no loss of universality. In Cases 1, 2, which deal with design points on the diagonal symmetry axis, the reinforcement amounts a_{s1} and a_{s2} are approximately in the ratio of the corresponding static heights [1].

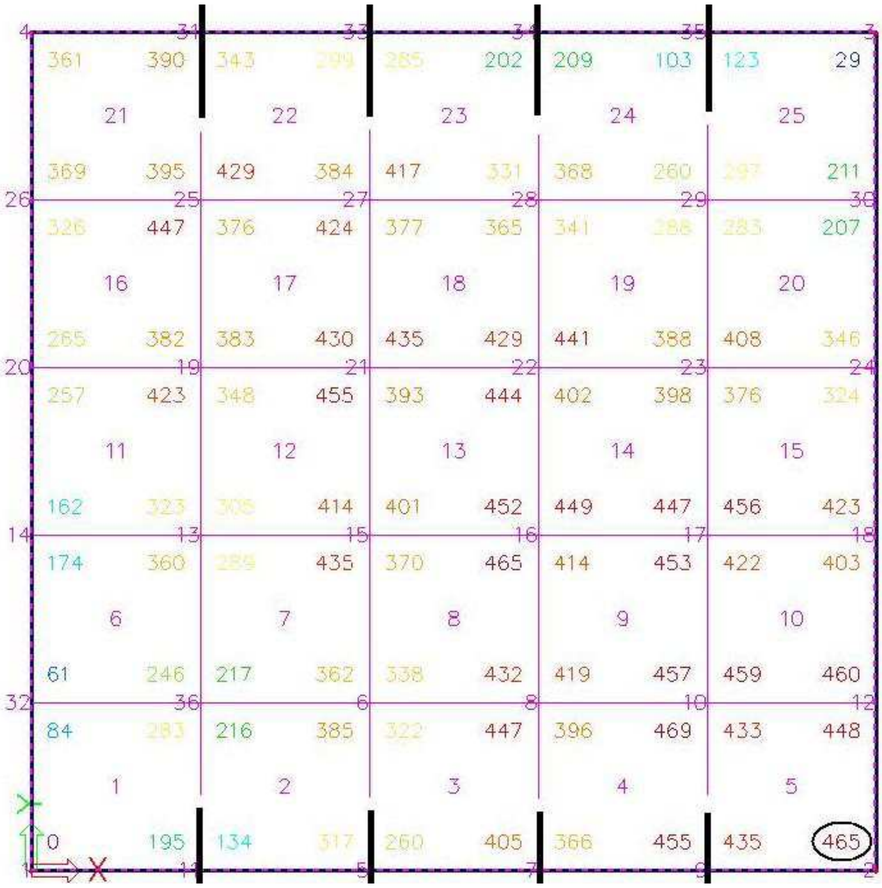


Fig.7 Design results: stat. required reinforcement a_{s2-} [mm²/m] – Direct values in element nodes (ULS)

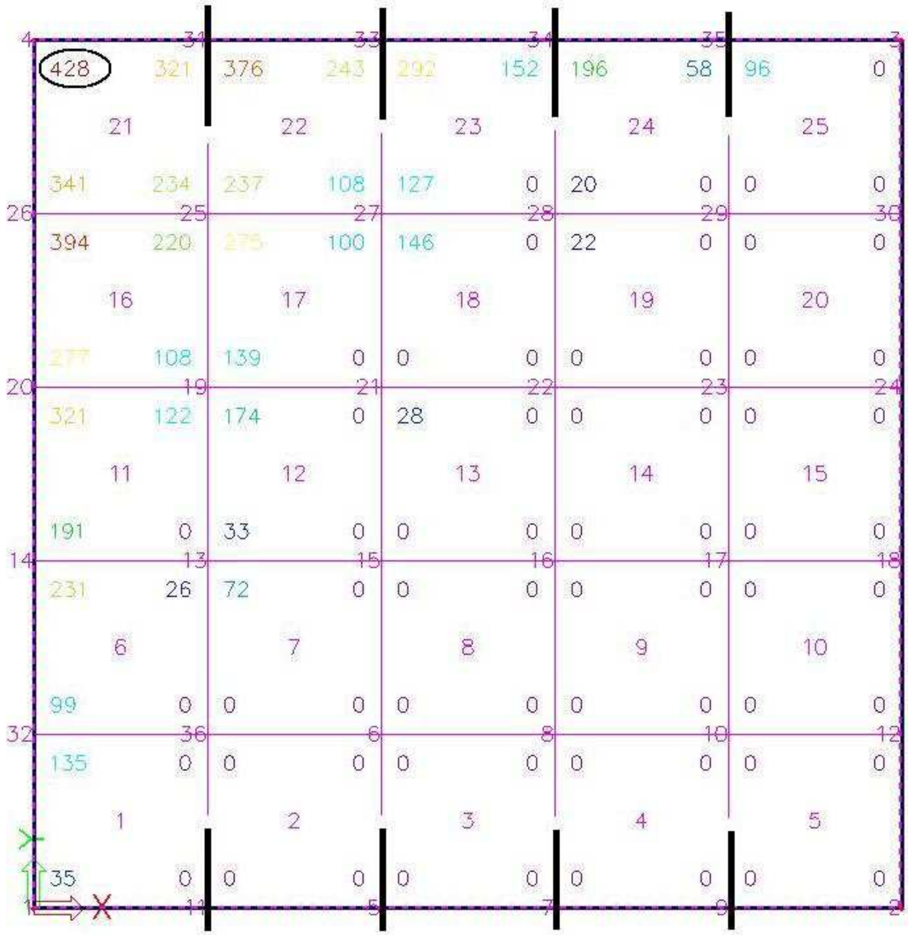


Fig.8 Design results: stat. required reinforcement a_{s2+} [mm²/m] – Direct values in element nodes (ULS)

C.1 Design Case 1: Statically Required Reinforcement – Elliptic Tension

As seen from Fig.2,3, in element 5, inner node 2 (coincident with the FE mesh node 2) both bending moments are positive: $m_x > m_y > 0$, i.e. tension at the lower face $–Z_p$. The torsion moment m_{xy} tends to zero; however, the theoretically exact zero value is "polluted" by the FEM approximation. Thus, an elliptic state of stress with tension at face $–Z_p$ is to examine in this design point. From Fig.2,3 and the Test Strategy protocol, the following Table 1 has been set up. Its graphic representation is given by Fig.9.

Table 1. **Inner moments** [kNm/m] (*Basic/ Principal/ Design*) in **element 5, node 2** (Test Strategy line 1#)

m_x	m_y	m_{xy}	m_I	m_{II}	$\alpha_{I,II}$	m_{1d}	m_{2d}	m_{cd}
39.3	39.3	-0.4	40.0	38.6	-45.0°	40.0	40.0	-1.4

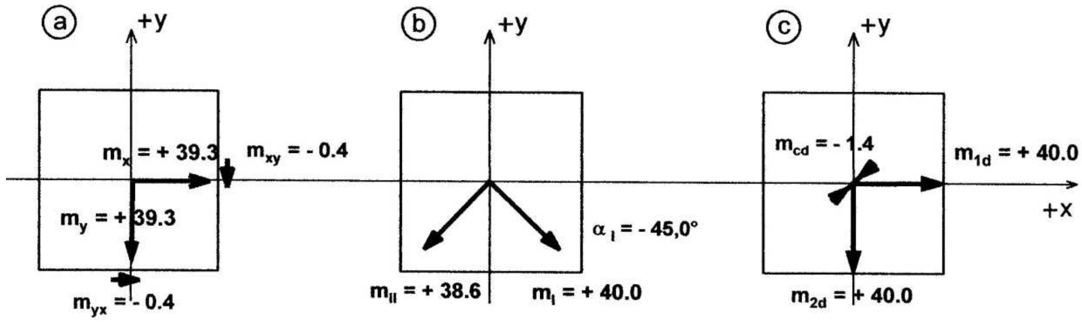


Fig.9 Graphic representation of Table 1 (elliptic inner moment transformation, face $–Z_p$)

The generalized invariant relation (5) in [1] is satisfied:

$$\boldsymbol{m_{1d}} + \boldsymbol{m_{2d}} + \boldsymbol{m_{cd}} = 40.0 + 40.0 - 1.4 = \mathbf{78.60} \quad \equiv \quad \boldsymbol{m_{I}} + \boldsymbol{m_{II}} = 40.0 + 38.6 = \mathbf{78.60} \quad [\text{kN/m}] \qquad \textbf{(1)}$$

The design moments $\{\boldsymbol{m_{1d}}, \boldsymbol{m_{2d}}, \boldsymbol{m_{cd}}\}$ may be, in analogy to mathematic terminology, considered as *separated result values* of the 2D design problem, which thus disintegrates into three individual pseudo 1D design cases. With the inner forces levels (the *Plate* design model enables exact distinguishing of reinforcement courses):

$$\boldsymbol{z_i} = \boldsymbol{\zeta_{mean}} \times \boldsymbol{h_i} \qquad \textbf{(2)}$$

$h_1 = h - c - \phi/2 = 0.250 - 0.030 - 0.010/2 = 0.215$ [m], $h_2 = h_1 - \phi = 0.215 - 0.010 = 0.205$ [m], $\zeta_{mean} = 0.96514$ [-]

$z_1 = 0.96514 \times 0.215 = 0.20751$, $z_2 = 0.96514 \times 0.205 = 0.19785$ [m]

The required reinforcement amount is then calculated formally as in two mutually independent 1D members:

$$\boldsymbol{a_{si,req}} = \boldsymbol{m_{id}} / (\boldsymbol{f_{yd}} \times \boldsymbol{z_{id}}) \qquad \textbf{(3)}$$

$\boldsymbol{a_{s1-,req}} = m_{1d} / (f_{yd} \times z_{1d}) = 0.040 / (500/1.15 \times 0.20751) = 4.43_{10}{-4}$ [m²/m] = **443** [mm²/m] (not displayed here)

$\boldsymbol{a_{s2-,req}} = m_{2d} / (f_{yd} \times z_{2d}) = 0.040 / (500/1.15 \times 0.19785) = 4.65_{10}{-4}$ [m²/m] = **465** [mm²/m](agreement with Fig.7)

The virtual concrete strut, which function may be characterised as "stiffening of the reinforcement net against distortion in its plane" is checked as "compression concrete zone without compression reinforcement" [1]. The strut check is governed by the formula:

$$\boldsymbol{\xi_{strut}} < \boldsymbol{\xi_{lim}} \quad [-] \qquad \textbf{(4)}$$

where ξ_{lim} is the limit value of the relative bending zone height; ξ_{strut} is the actual relative height of the bending zone under the impact of m_{cd}. With the 1st default values of the reduction factor $\boldsymbol{r_{fac}} = 0.80$ (i.e. 80% of full design strength of concrete, weakened by parallel cracks, is applied) and the limit value of the relative bending zone height $\xi_{lim} = 0.450$ (Fig.10), the following relation is met:

$\xi_{strut} = \mathbf{0.012} < 0.450$

There is a substantial resistance reserve in this design point; (to be expected, since $m_{cd} \rightarrow 0$ (double symmetry)).

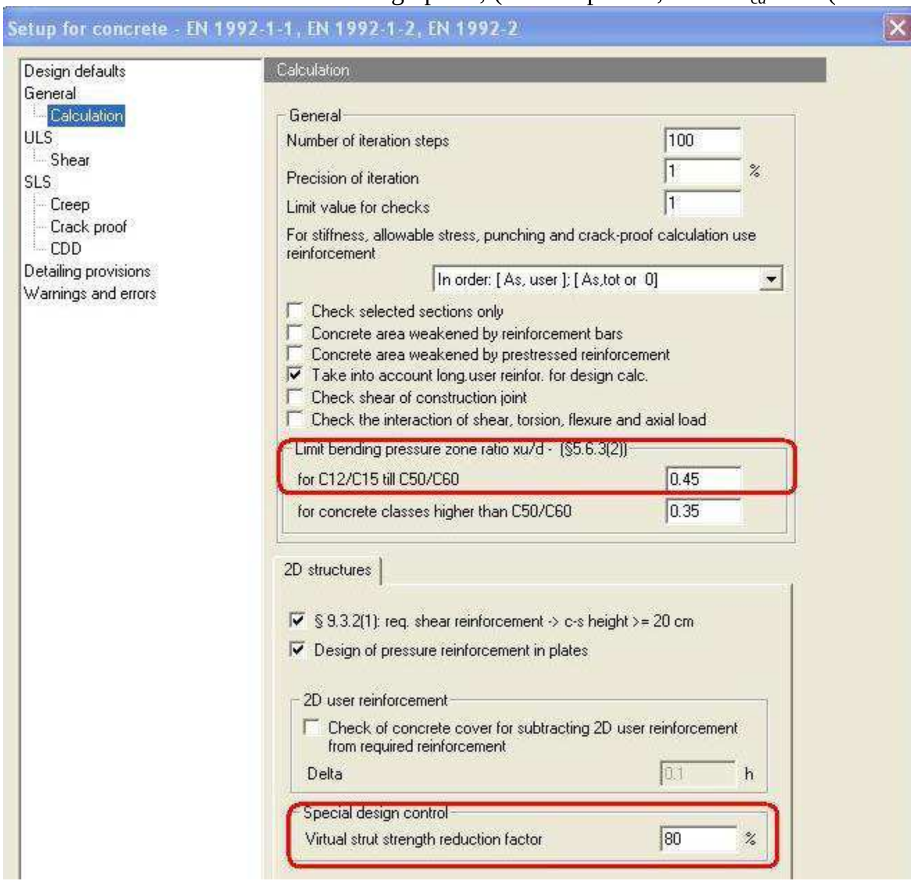


Fig.10 Concrete Setup - limit value of the relative bending zone height and strength reduction factor

C.2 Design Case 2: Statically Required Reinforcement – Hyperbolic Stress

As seen from Fig.2,3, in element 21, inner node 4 (coincident with the FE mesh node 21), the bending moments m_x and m_y tend towards zero while the torsion moment attains an intensity level comparable with the bending moments in element 5. This is a typical hyperbolic state of stress, as the altering signs of the principal moments $m_I > 0$ and $m_{II} < 0$ demonstrate – see Table 2. The graphic representation is given by Fig.11.

Table 2. **Inner moments** [kNm/m] (*Basic/ Principal/ Design*) in **element 21, node 4** (Test Strategy line 2#)

m_x	m_y	m_{xy}	m_I	m_{II}	$\alpha_{I,II}$	m_{1d}	m_{2d}	m_{cd}
2.7	2.7	-33.1	35.9	-30.4	-45.0°	35.9	35.9	-66.2

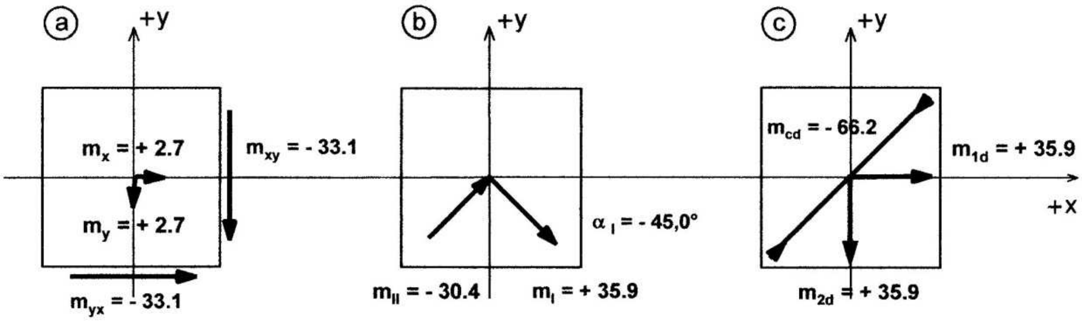


Fig.11 Graphic representation of Table 2 (hyperbolic inner moments transformation, face $+Z_p$)

The generalized invariant relation (5) [1] is satisfied:

$$\boldsymbol{m_{1d}} + \boldsymbol{m_{2d}} + \boldsymbol{m_{cd}} = 35.9 + 35.9 - 66.2 = \mathbf{5.5} \ \equiv \ \boldsymbol{m_I} + \boldsymbol{m_{II}} = 35.9 - 30.4 = \mathbf{5.5} \ \text{[kN/m]} \qquad \mathbf{(5)}$$

The most distinguishing difference of the invariant relations (1) and (5) is that of the intensity level of the design moments m_{cd}: in the elliptic stress case (1), it is a value tending to zero; in the hyperbolic stress case (5), it is a significant quantity, here absolutely about twice as high as the sum of absolute values of the principal moments. This is characteristic a phenomenon for the hyperbolic cases: the stiffening function of the concrete, the virtual strut, becomes the crucial factor of the designability.

Analogously to the elliptic Case 1, the required reinforcement in both directions at face $+Z_p$ is calculated as follows:

$h_1 = 0.215$ [m], $h_2 = 0.205$ [m], $\zeta_{mean} = 0.96514$ [-]

$z_1 = 0.93936 \times 0.215 = 0.20196$, $z_2 = 0.\ 93936 \times 0.205 = 0.19257$ [m]

The required reinforcement amount is then calculated formally as in two mutually independent 1D members:

$\boldsymbol{a_{s1+,req}} = m_{1d} / (f_{yd} \times z_{1d}) = 0.0359 / (500/1.15 \times 0.\ 20196) = 4.08_{10}{-4}$ [m²/m] = **408** [mm²/m](not displayed here)

$\boldsymbol{a_{s2+,req}} = m_{2d} / (f_{yd} \times z_{2d}) = 0.0359 / (500/1.15 \times 0.\ 19257) = 4.28_{10}{-4}$ [m²/m] = **428** [mm²/m](agreement with Fig.8)

The virtual concrete strut is checked by the relation (4):

$\xi_{strut} = \mathbf{0.204} < 0.450$

Thus, there is still a resistance reserve of the virtual strut in this hyperbolic state of stress.

Hint: the procedure of checking the virtual strut bearing capacity is a very distinguishing feat of SEN/NEDIM, which hardly any competing software comprises. The topic of checking the stiffening function of concrete in bent continua has been dealt with in some detail in [2].

C.3 Design Case 3: Shear Proof

Some features of the SEN/NEDIM Shear Proof procedure will be demonstrated on element 1, inner node 1 (coincident with the FE mesh node 1). Fig.4 displays here the maximum value of v_x. As seen from Fig.2,3, the bending stress tends here towards zero, which is a plausible result. From the Test Strategy line 3# document, the following relations can be established:

$$\boldsymbol{v_x} = 51.0,\ \boldsymbol{v_y} = -0.2\ \text{[kN/m]},\ \boldsymbol{\beta_0} = -0.2\ [°] \quad \rightarrow \quad \boldsymbol{v_{Ed}} = 51.0\ \text{[kN/m]}$$

The "theoretical" values are: $v_y = 0$, $\beta_0 = 0$. Since there are no accidental load cases specified, the material partial security coefficients are $\gamma_s = 1.15$ and $\gamma_c = 1.50$, thus $f_{yd} = 434.8$ and $f_{cd} = 13.33$ [MPa]. The shear resistance without shear reinforcement according to §6.2.2 (6.2a) is estimated as

$$\boldsymbol{v_{Rd,c}} = \mathbf{91.3} > \boldsymbol{v_{Ed}}\ \text{[kN/m]}$$

Thus, no shear reinforcement is required in this design point, and nowhere in the model, since here the most critical design situation appears.

C.4 Alternative Design Cases 1+2: Reinforcement Geometry (0°/60°/120°) and (–135°/45°)

Numerical results and screen copies of this Paragraph were obtained by the project **Plate_Benchmark_EN(1)**.

To demonstrate the distinguishing features of the 2D design module SEN NEDIM, the use of a 3-course reinforcement net is presented and briefly discussed. In engineering practice, ~95% of design cases represent the elementary arrangement of orthogonal reinforcement with congruent nets at faces $-Z_p$ /$+Z_p$, the outer course parallel to X_p, the inner course parallel to Y_p – as if it were a "nature law". NEDIM offers, however, general reinforcement arrangements – combining different specifications of orthogonal, skew angular or 3-course reinforcement, respectively, at both faces. Here we will discuss some features of a "nonstandard design", using the reinforcement nets (0°/60°/120°) at face $-Z_p$ and (–135°/45°) at face $+Z_p$.

(This Paragraph C.4 will be completed in Release 4 of this Benchmark document)

D. Required Minimum Reinforcement

Numerical results and screen copies of this Chapter were obtained by the project **Plate_Benchmark_EN(2)**.

In this Chapter, the effect of activated minimum tension reinforcement specification according to NEDIM 1st default input, as shown in Fig.12, is dealt with. As a fact, the user may modify the settings, but accepting the 1st default, legal solution according to the Norm is obtained.

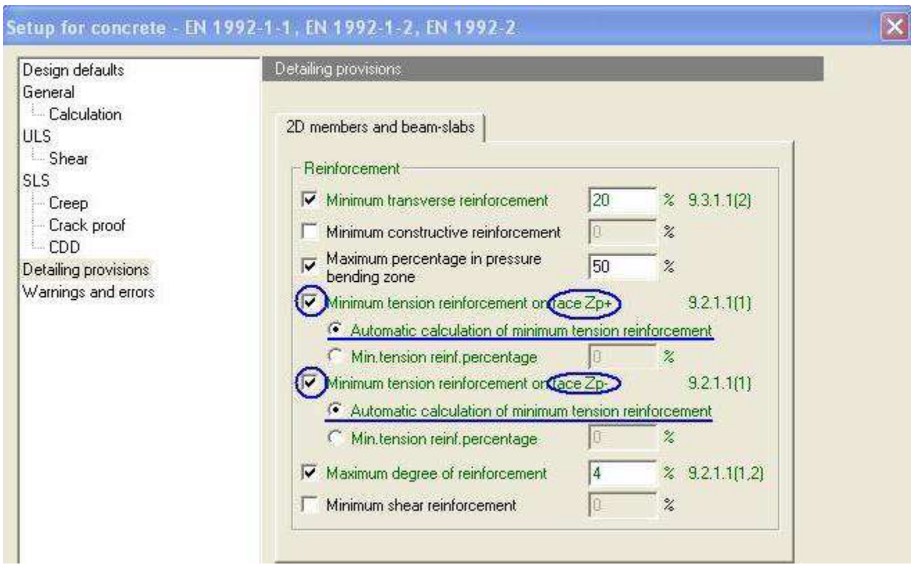


Fig.12 Input dialogue window – max/min tension reinforcement set active

The choice of the "Automatic calculation of minimum tension reinforcement" ensures the consideration of the EN provisions to prevent so called *brittle fracture*, according to §9.2.1.1(1) (9.1N). The alternative to this option is the input of percentage which will be superposed in each point to required tension reinforcement; this is non-standard control under EN 1992-1-1:2004. The control option "Maximum percentage in pressure bending zone" serves to limit the amount of required compression reinforcement to the given percentage related to the pressure bending zone force; the 1[st] default of 50% limits thus the steel force to 50% of the concrete force. In this example, however, this option is without practical impact since there is no compression reinforcement required.

From the relation §9.2.1.1(1) (9.1N) the following formula for 2D calculation of the minimum reinforcement against brittle fracture follows:

$$\mathbf{a}_{st,min} = \mathbf{max}(\mathbf{0.26}\, \boldsymbol{f_{ctm}/f_{yk}}, \mathbf{0.0013}) \times \boldsymbol{d} \quad [\text{m}^2/\text{m}] \qquad \mathbf{(6)}$$

With $d_1 = 0.215$ m, $f_{ctm} = 2.20$ and $f_{yk} = 500$ NEDIM estimates

$$\boldsymbol{a}_{s,min} = \max(0.00114, 0.0013) \times 0.215 \times 10^6 = 279.5 \ [\text{mm}^2/\text{m}]$$

We find the value of 280 mm²/m in the display of the functions $a_{s2-,req+min}$ and $a_{s2+,req+min}$ in Fig.13 and 14. All values 280 in the reinforcement display (marked) signalize that the pure statically required tension reinforcement is less than this minimum value. This can visually be checked in comparison with Fig.7,8.

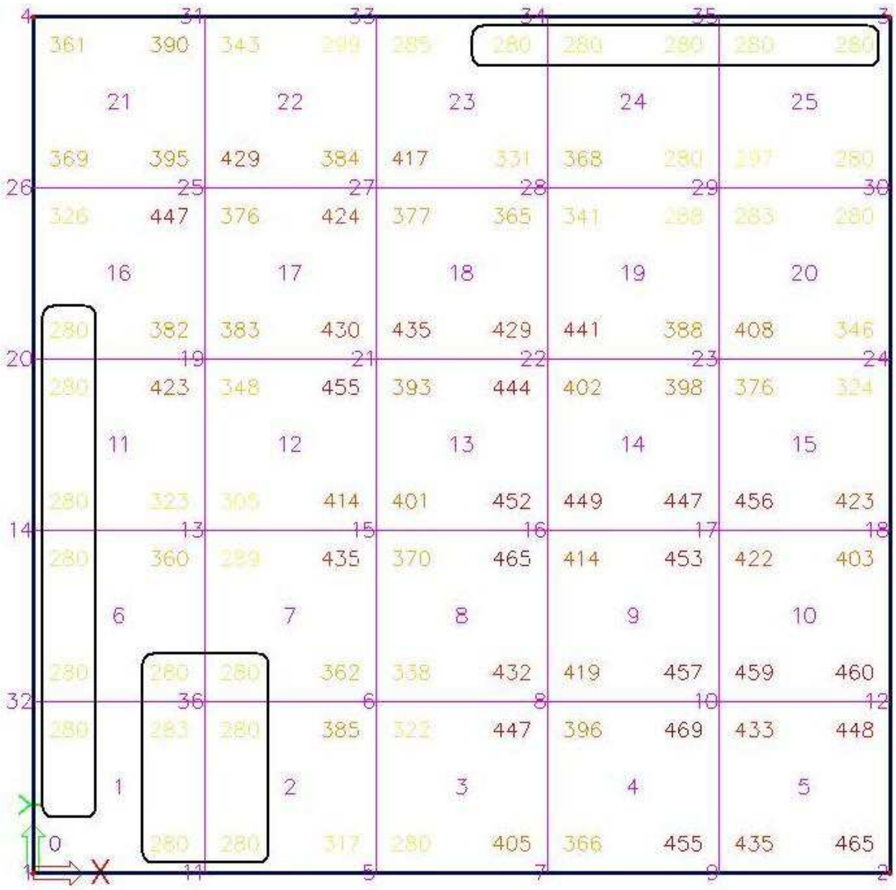


Fig.13 Reinforcement $\boldsymbol{a}_{s2-,req+min}$ [mm²/m] – Direct values in nodes

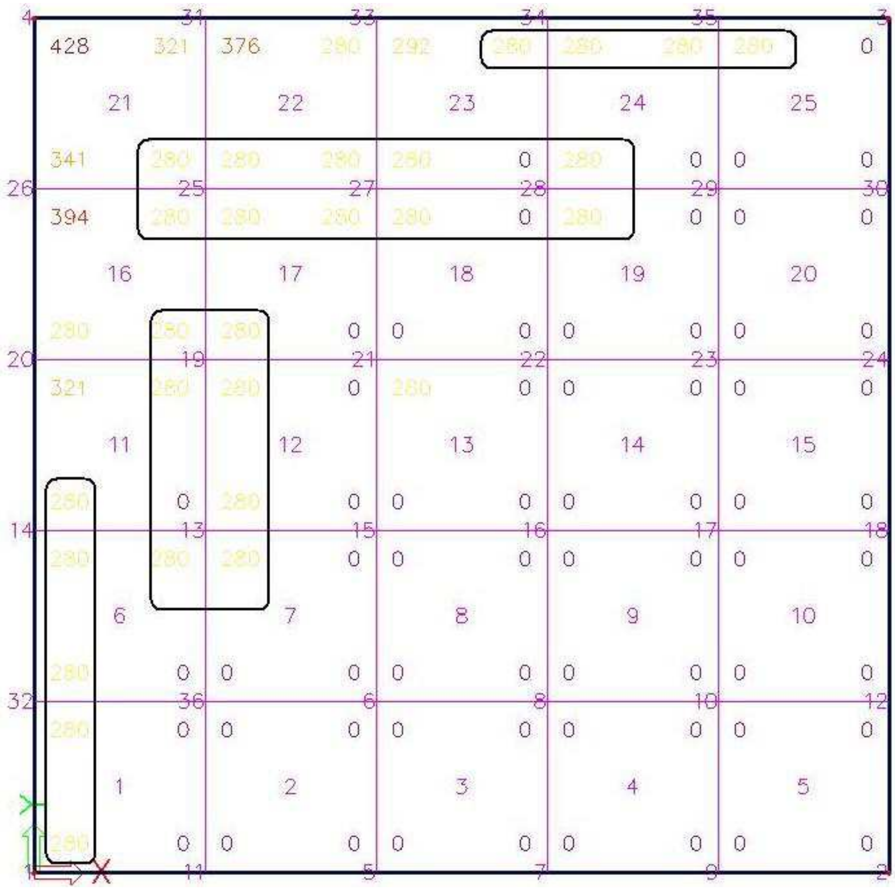


Fig.14 Reinforcement $a_{s2+,req+min}$ [mm²/m] – direct values in element nodes

E. SLS Design – Reinforcement Augmented to Limit Crack Widths

Numerical results and screen copies of this Chapter were obtained by the project **Plate_Benchmark_EN(2)** (continued from Chapter D).

In this Chapter, the effect of activated minimum tension reinforcement specification according to NEDIM's 1st default input, as shown in Fig.12, is dealt with. As a fact, the user may modify the settings, but accepting the default value, a legal solution according to the Norm stipulations will be obtained.

The declared task of the Crack Proof (particularly of the 2D design) is checking the crack widths in all design points according the stipulations of EN 1992-1-1:2004, §7.3.4 "Calculation of crack widths" and augmenting the reinforcement delivered via the database from the ULS phase so as to meet the condition

$$w_{cal} \leq w_k \qquad (6)$$

with w_{cal} – the actual crack width calculated from the formula §7.3.4, (7.8). As a fact, SEN/NEDIM seeks the value fulfilling the equality in (6) by a sophisticated (sub-linearly controlled) iteration process. The result is the reinforcement amount $a_{s2,ULS+SLS}$ which meets the requirements of both ULS and SLS design phases:

$$a_{s2,ULS+SLS} \geq a_{s2,req+min} \geq a_{s2,req} \qquad (7)$$

(compare with Fig.16,17 and Tab.3,4). In the Crack Proof algorithm not only the bar diameters ϕ but also the specified bar distances ***s*** (Fig.15) play a crucial role: they represent the user's decision to use reinforcement bars of this diameter, spaced at maximum by the distance specified. Thus, the final reinforcement amount $a_{s2,ULS+SLS}$ respects this restriction as a kind of "overall minimum reinforcement". In this example, the reinforcement amount corresponding to $\phi = 10/s = 200$ (Fig.6,15) → $a_{s,min,constr} = 393$ mm²/m will be encountered at several design points, both upper and lower reinforcement – as marked in Fig.16,17.

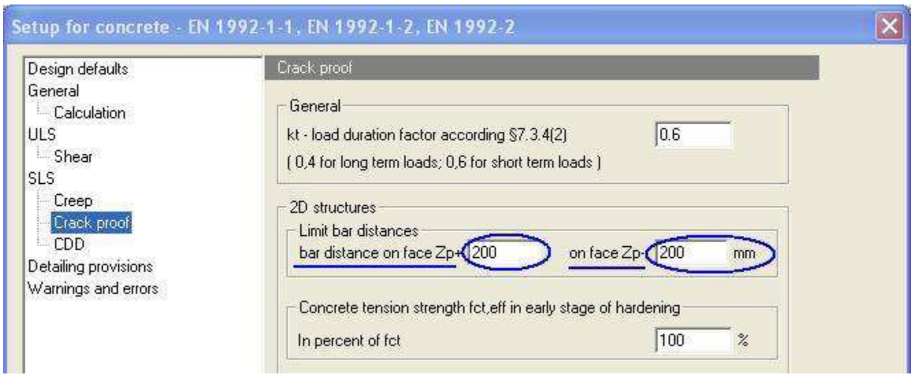


Fig.15 Concrete setup window – maximum allowable reinforcement bars distances

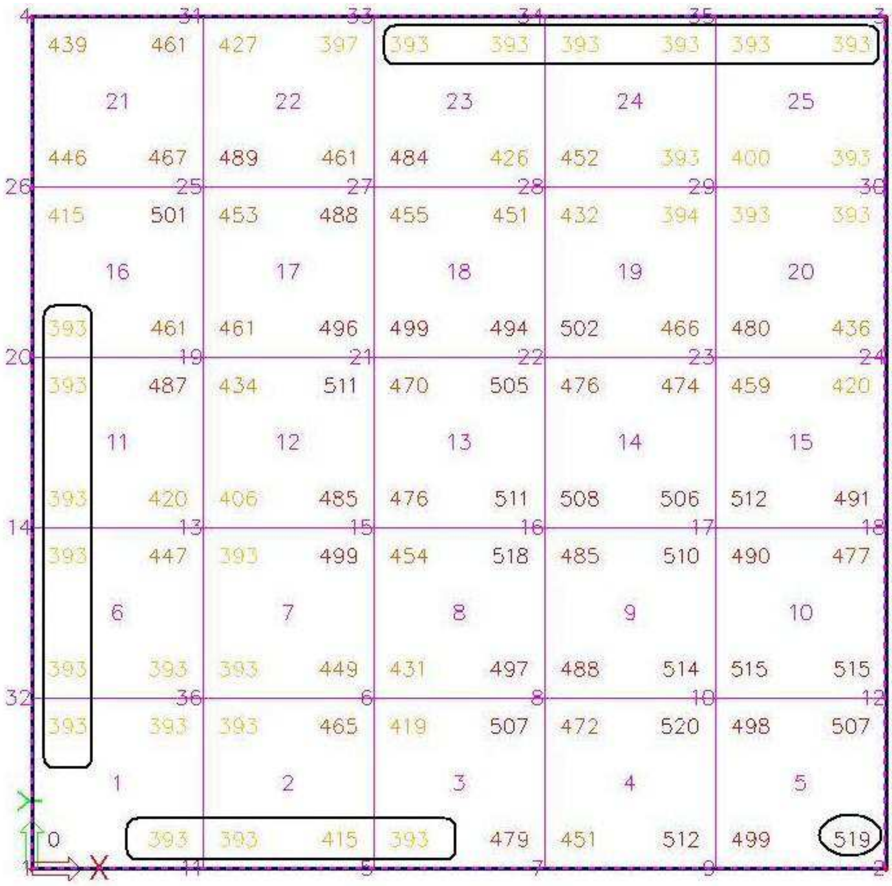


Fig.16 Reinforcement $a_{s2-,\ ULS+SLS}$ [mm²/m] – Direct values in nodes

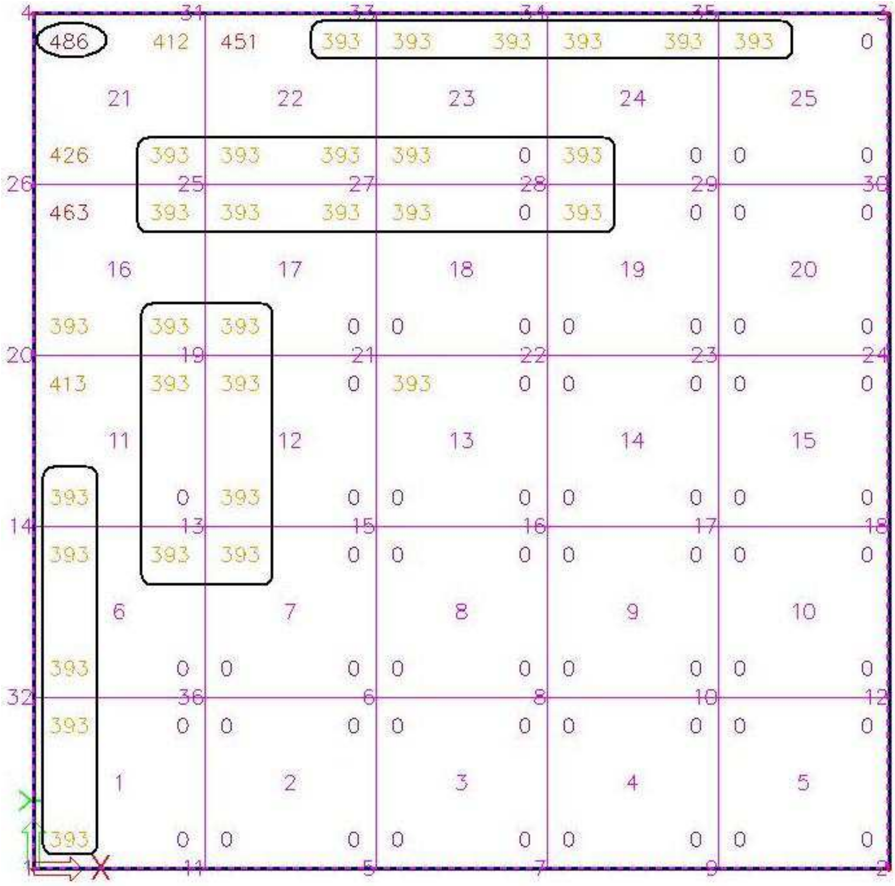


Fig.17 Reinforcement $a_{s2+,ULS+SLS}$ [mm²/m] – direct values in element nodes

Following Tables 3,4 compare the reinforcement designed at the three design stages discussed above. *Blue* marks the stage *Minimum* and *Red* marks the stage *Crack Proof* – when the reinforcement amount obtained here is higher than in the previous design stage(s). Only reinforcement values $a_{s2\pm}$ (direction 90°) presented in the following tables are shown above. All of them can be checked by viewing the display data of the projects (0) to (2).

Table 3. Comparison of the reinforcement stages in design points 5/2, 21/4 and 1/1 ~ Lower face

Elem/Node	Reinforcement at face $-Z_p$ ~ a_{s1-} [mm²/m]			Reinforcement at face $-Z_p$ ~ a_{s2-} [mm²/m]		
	$a_{s1-,req}$	$a_{s1-,req+min}$	$a_{s1-,ULS+SLS}$	$a_{s2-,req}$	$a_{s2-,req+min}$	$a_{s2-,ULS+SLS}$
5/2	443	443	443	465	465	519
21/4	344	344	393	361	361	439
1/1	28	280	393	0	0	0

Table 4. Comparison of the reinforcement stages in design points 5/2, 21/4 and 1/1 ~ Upper face

Elem/Node	Reinforcement at face $+Z_p$ ~ a_{s1+} [mm²/m]			Reinforcement at face $+Z_p$ ~ a_{s2+} [mm²/m]		
	$a_{s1+,req}$	$a_{s1+,req+min}$	$a_{s1+,ULS+SLS}$	$a_{s2+,req}$	$a_{s2+,req+min}$	$a_{s2+,ULS+SLS}$
5/2	0	0	0	0	0	0
21/4	408	408	408	428	428	486
1/1	0	0	0	35	280	393

The examination of Tables 3,4 enables the following conclusions, which possess general validity:

(1) Pronounced values of statically required reinforcement are typically “resistant” to augmentation by minimum reinforcement or Crack Proof requirements;

(2) Either the minimum reinforcement or Crack Proof requirements may yield higher reinforcement; the superposition result represents always the highest amount;

(3) If the statically required reinforcement amount is zero, the application of the minimum reinforcement or Crack Proof requirements has no impact upon its augmenting; the SEN/NEDIM result value $a_{s,ULS+SLS}$ remains zero! This may sometimes be found surprising if not irritating: it is, however, not the task of SCIA software to present non-existing results. However, the user is not obliged to apply zero reinforcement where he will use a nonzero reinforcement constructively.

(4) FEM is an approximate (numerical) method of solving differential mechanical problems. The results (inner forces) are *mean values on element* and represent basically non-continuous functions. Thus, “unexplainable” reinforcement values, as compared with “theoretical” values of other (analytical) solutions or models, are hardly erroneous; they are true counterparts of the inner forces, which are used by NEDIM without any biasing “adaptations” [3]. The design point in element 1, node 1 is a typical example of these relations: “theoretically”, all 4 reinforcement values are zero, but this is never true in a FEM model. Thus, we have got $a_{s1+} = a_{s2-} = 0$ (as expected), but a_{s2+}, $a_{s1-} > 0$ – due to the “numerical polution” of the FEM solution.

REFERENCES

[0] Hobst, Ed.: Project files >Plate_Benchmark_EN(0).esa<, >Plate_Benchmark_EN(1).esa< and >Plate_Benchmark_EN(2).esa<, created under SCIA Engineer V9.0.426 and NEDIM V04.8.3.0; distributed with this Paper as its integral part.

[1] Hobst, Ed.: *ESA-Prima Win & SCIA Engineer Reinforced Concrete Design of 2D Structures*. *Theoretical Background*. SCIA, Herk-de-Stad (B) 2009, Pages I-V+84 (89)

[2] Hobst, Ed.: *Critical notes on the design of plates according to DIN 1045, E DIN 1045-1 and ÖNORM B 4700* (in German: *Kritische Hinweise zur Bemessung von Platten nach DIN 1045, E DIN 1045-1 und ÖNORM B 4700*). In: "Bautechnik" 77 (2000), Issue 10, Pages 707-717.

[3] Hobst, Ed.: *Boundary conditions and singularities - how exact is the Finite Element Method?* (in German: *Randbedingungen und Singularitäten - wie genau ist die Finite-Elemente-Methode?*). In: "Beton- und Stahlbetonbau" 95 (2000), Issue 10, S. 572-583.

Nürnberg, 5 February 2010

Dr. Eduard Hobst

Local Product Ingenieur (Concrete) SCIA

(3rd release)