

ADVANCED CONCEPT TRAINING REINFORCED CONCRETE (EN1992) – 2D MEMBERS

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

© Copyright 2018 SCIA nv. All rights reserved.

le of Contents	
Introduction	5
Plate design	6
Model	
1_Input of geometry	
2_Loads	
3_Finite element mesh	
4_Results for the linear calculation	
Concrete setups	
1_General setups	
2_ Member data	
ULS design	
1_Reinforcement design	
2_Practical reinforcement design	
Crack control	
1_Input data for crack control	
2_Results for required theoretical reinforcement	
Annex 1: Conventions for the results on 2D members	
1_ Basic magnitudes = Characteristic values	
2 Principal magnitudes	
3_Design magnitudes	
Annex 2: Results in Mesh elements and Mesh nodes \rightarrow 4 Location	
1_ In nodes, no average	
2_ In centres	
3_ In nodes, average	
4_ In nodes, average on macro	
Accuracy of the results	
Annex 3: Local Coordinate system for 2D members	
1_ Perpendicular to vector	
2_ Tilt of vector normal to line	
3_ Congruent with line	
4_ Smallest angle with vector	
5_ Tilt of vector defined by point	

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

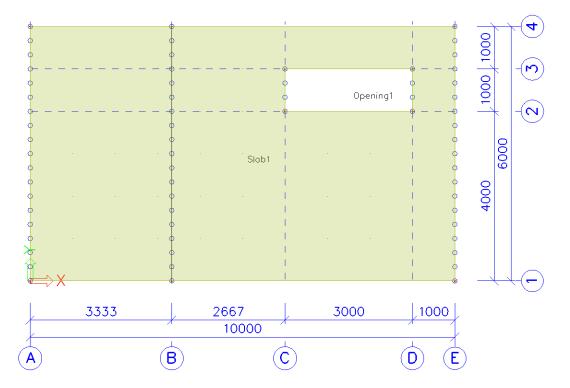
Plate design

Model

1_Input of geometry

Project data	: 2D enviro	nment = Plate XY	
Project data			>
Basic data Fur	nctionality Actio	ons Unit Set Protection	
	Data		Material
	Name:	Example project	Concrete
FREE			Material C20/25
AAA	Part:	ACT Reinforced Concrete	Reinforcement B 500A 👻
7 CT			Steel
TAT	Description:	Plate design	Masonry 🗆
11/2			Aluminium 🗆
PAL SE	Author:	Scia Engineer	Timber 🗆
			Steel fibre conc
THE OWNER	Date:	27. 07. 2011	Other 🗌
	Structure:	Post processing	Code National Code:
	ondotaro.	environment	
	🐑 Plate XY	- 🔹 v17 -	EC - EN *
	Model:		National annex:
The second Co	🗑 One	*	(`) EC-EN
			OK Cancel

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



Name	
Element type	Standard
Element behaviour	Standard FEM
Туре	plate (90)
)α Shape	Flat
T Material	C20/25 -
FEM model	Isotropic
FEM nonlinear model	none
Thickness type	constant
Thickness [mm]	250
LCS type	Standard
LCS angle [deg]	0,00
Layer	Laag1 -
A Nodes	
Nodes Actions	
Actions Table edit geometry	OK Can
Actions	
Actions Table edit geometry	OK Can
Table edit geometry mber edge	OK Can Sie2 Hinged
Actions Table edit geometry mber edge	OK Can
Actions Table edit geometry mber edge	OK Car Sie2 Hinged
Actions Table edit geometry mber edge	Sle2 Hinged Rigid Free Free
Actions Table edit geometry nber edge Constraint Z Rx	Sie2 Hinged Rigid Free
Actions Table edit geometry ember edge Constraint Z R R R Ry	OK Can Sle2 Hinged Rigid Free Free Slab1
Actions Table edit geometry mber edge Constraint Z Rx Ry 2D member	OK Can Sle2 Hinged Rigid Free Free Slab1 GCS
Actions Table edit geometry ember edge Constraint Z R: Ry 20 member 2 Geometry	Sie2 Hinged Rigid Free Free Slab1 GCS
Actions Table edit geometry mber edge Name Constraint Z Rx Ry 20 member System	OK Can Sie2 Hinged Rigid Free Free Slab1 GCS
Actions Table edit geometry mber edge Name Constraint Z Rx Ry 20 member Geometry System Edge	OK Can Sle2 Hinged Rigid Free Free Slab1 GCS 2 Rela 0,000
Actions Table edit geometry her edge Constraint Z Rx Ry 2D member Geometry System Edge Coord. definition	Sle2 Hinged Rigid Free Slab1 GCS 2 Rela

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 cases		×
🔎 🤮 🗶 📴	💁 🗠 🎒 💕 🖬 🛛 Al	• 7
LC1 - Self-weight	Name	LC1
LC2 - Walls	Description	Self-weight
LC3 - Sevice load	Action type	Permanent 👻
	Load group	LG1
	Load type	Self weight 👻
	Direction	-Z *
New Insert Edit	Delete	Close

Load groups		×
🏓 🤮 🗶 😫	2 🗠 🎒 🈂 🖬 🛛 Al	• 7
LG1	Name	LG2
LG2	Relation	Standard 👻
	Load	Variable -
	Structure	Building
	Load type	Cat B : Offices 👻
New Insert Edi	t Delete	Close

Load combinations

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

Combinations		:	×
🔎 🕃 🗶 📸 💽 🗅	🖞 🖨 Input combinations 🔹		
EN-ULS (STR/GEO) Set B	Name	EN-SLS Quasi-permanent	
EN-SLS Quasi-permanent	Description		
	Туре	EN-SLS Quasi-permanent	
	Structure	Building	
	Active coefficients		
	4 Contents of combination		
	LC1 - Self-weight [-]	1,00	
	LC2 - Walls [-]	1,00	
	LC3 - Sevice load [-]	1,00	
	Actions		
	Explode to envelopes	>>>	>
	Explode to linear	>>>	>
	Show Decomposed EN combinations	>>>	>
New Insert Edit	Delete	Close	e

Result classes

All ULS+SLS

Result classes		×
🎜 🤮 🗶 📸 💺	🖸 🗠 🎒 🗛	• 7
All ULS+SLS	Name	All ULS+SLS
	Description	
	4 List	
		EN-ULS (STR/GEO) Set B - EN-ULS (STR/GEO) Set B
		EN-SLS Quasi-permanent - EN-SLS Quasi-permanent
New Insert	Edit Delete	Close

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:

Name	SolverSetup1	1
pecify load cases for linear calculation		
Advanced solver settings		
General		
Neglect shear force deformation (Ay, Az >> A)		
Bending theory of plate/shell analysis	Mindlin	-
Type of solver	Direct	*
Number of sections on average member	10	
Warning when maximal translation is greater than [mm]	1000,0	
Warning when maximal rotation is greater than [mrad]	100,0	
Coefficient for reinforcement	1	
Effective width of plate ribs		
Number of thicknesses of rib plate	20	
Parallelism tolerance for automatic calculation [deg]	10,00	
Span length ratio L/beff, max (1 side) for automatic calculation [-]	8,00	
Span length correction		

Mesh generation

Via the Main menu \rightarrow Calculation, mesh \rightarrow 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 \rightarrow Mesh \rightarrow Draw mesh

- Labels tab \rightarrow Mesh \rightarrow Display label

OR

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

Structure	×	Panel
Labels	•	✓ Structure nodes
Model	×	Mesh N
Loads/masses	+	Local axes
Concrete		
Composite	•	φ
Modelling/Drawing	•	
Attributes	×	
Detailed on/off		
Setup dialog		
🖷 🖭 🗮		

Mesh refinement

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

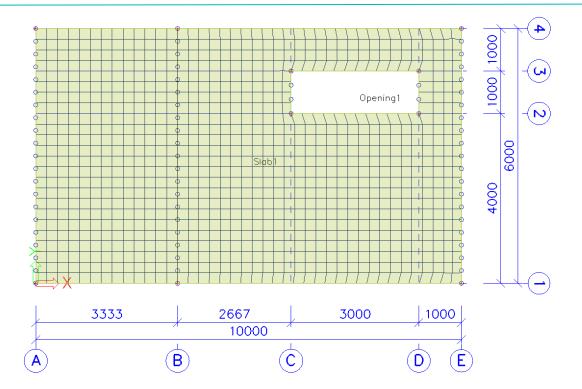
E Mesh setup	×	<
Name	MeshSetup1	
Average number of tiles of 1d element	1	
Average size of 2d element/curved element [m]	1,000	
▷ Advanced mesh settings		
	OK Cancel	

OR

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

E analysis		×
Calculations	4 Mesh setup	
✓ Linear analysis	Average number of tiles of 1d element 1	
Load cases: 3	Average size of 2d element/curved ele 1,000	
Other processes	Advanced mesh settings	
☐ Test input of data	4 Solver setup	
	Specify load cases for linear calculation	
	Advanced solver settings	
Calculate		

'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 \rightarrow Results \rightarrow 2D Members \rightarrow 2D Internal Forces.

Specify the desired result in the Properties menu:

Descention.		
Properties)
2D internal forces (1)	- 10	7/ /
	5	8
Name	2D internal forces	
Selection		
Type of selection	All	-
Filter	No	-
A Result case		
Type of load	Combinations	-
Combination	EN-ULS (STR/GEO) Set E	+
Envelope (for 2D drawing)	Absolute extreme	-
Averaging of peak		
Location	In nodes avg. on macro	-
System	LCS mesh element	-
Extreme	Global	-
Type of values	Basic magnitudes	-
Values	m_x	-
4 Output settings		
Print combination key	\checkmark	
Standard result	\checkmark	
Results on sections		
Results on edges		
Table setup		
Errors, warnings and notes		
Drawing setup 2D		
Actions		
Refresh		>>>
New combination from Combinat	ion key	>>>
Table results		>>>
Preview		>>>

System:

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

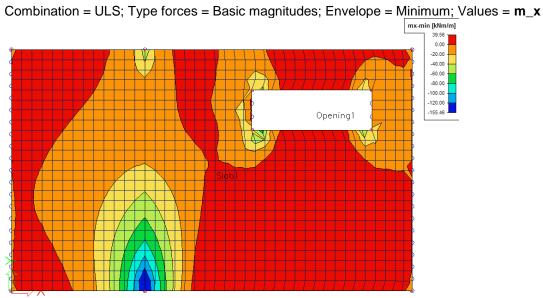
Location: 4 different ways to ask for the results, see Annex 2 **Type forces**: Basic, Principal or Design magnitudes, see Annex 1

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

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

Types of results

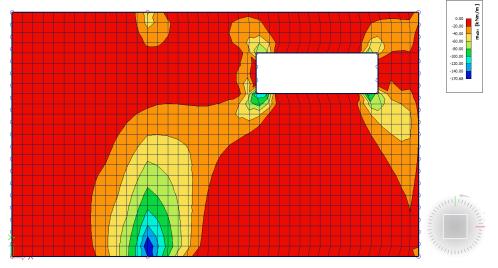
Basic magnitudes



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

Elementary design magnitudes

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



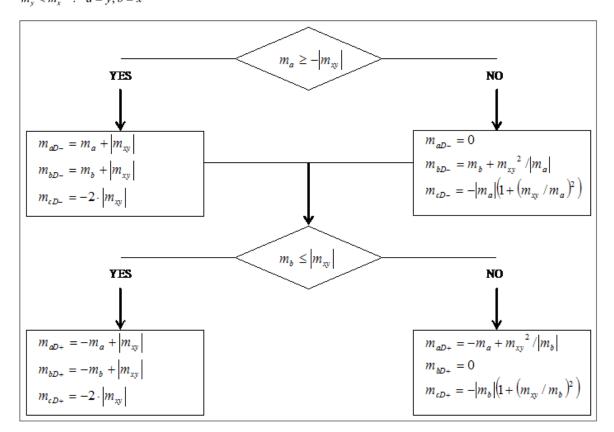
The convention for the sign of the design moments has been changed since the v17 post-processor. Now a moment is positive when it causes a tensile force on the bottom of the plate and negative when it causes tensile force at the top of the plate.

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

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

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

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



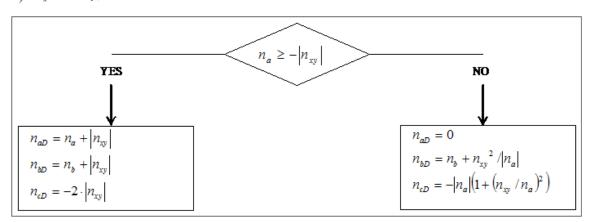
What happens, is that for the 3 characteristic (bending and torsion) moments an equivalent set of 3 design moments is calculated:

mx		mxD
my	≈	myD
mxy		mcD

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

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

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



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

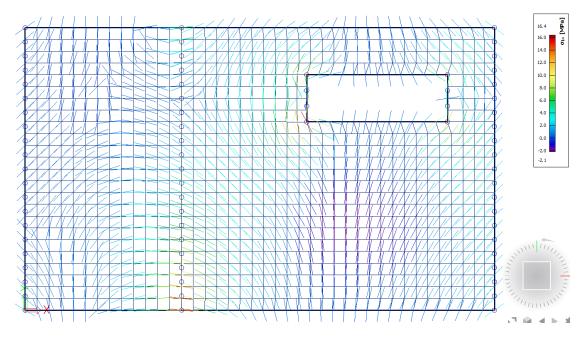
nx		nxD
ny	~	nyD
nxy		ncD

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 \leq fcd.

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

Principal magnitudes

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



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

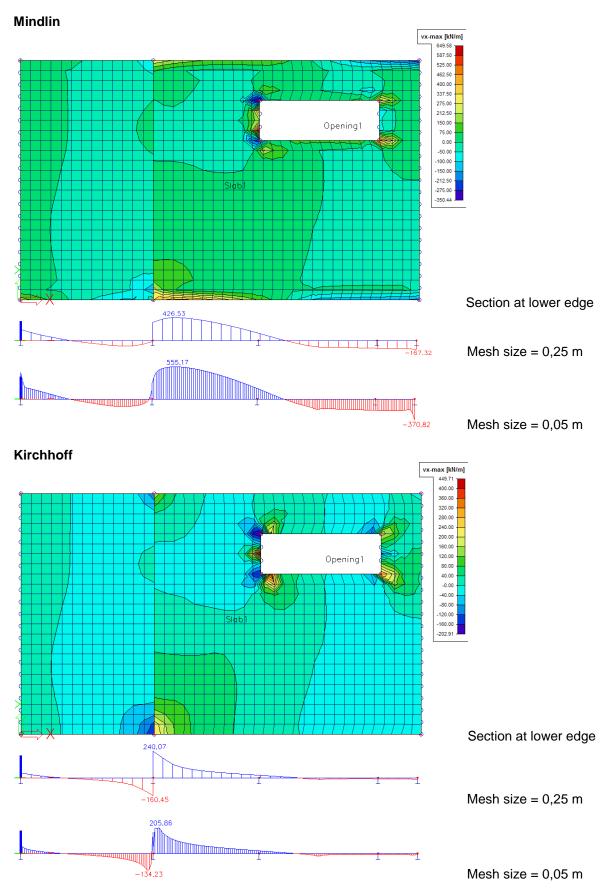
The first direction is the direction of maximum tension (or minimum compression). The second direction is the direction of maximum compression (or minimum tension).

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

Comparison Mindlin ↔ Kirchhoff

Shear force vx

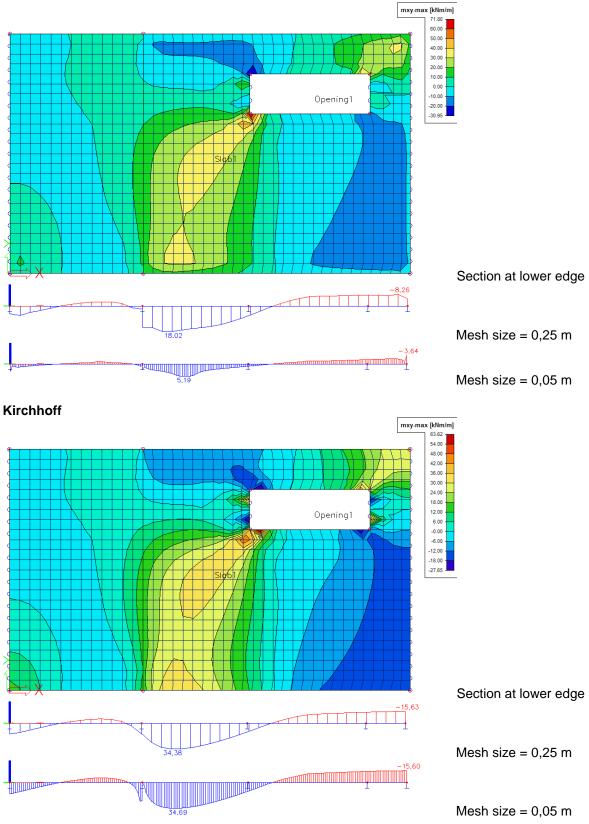
Combination = ULS; Type forces = Basic magnitudes; Envelope = Maximum; Values = v_x



Torsion moment mxy

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





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

Concrete setups

1_General setups

Setup 1: National Determined Parameters

```
Main menu \rightarrow Project data \rightarrow National annex [...] \rightarrow EN 1992-1-1 [...]
```

OR

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

Hollow core beams Prestressing U U U U U U U U U U U U U U U U U U U	al ncrete n-prestressed reinforcement rsbillity and concrete cover neral nching neral ng provisions	Name Concrete General General Concrete National annex KIN1992_1_1 YsH-Partial factor for shrinkage a Value [-] Yolue [-] Values [-]	1,00	
- 20	mmon detailing provisions structures and slabs nching	 f_{ck.max} - maximum value of the Value [MPa] a_{cc} - coeff. taking account of low Value [-] a_{ct} - coeff. taking account of low Value [-] k_{1red} - coeff. for calculation of r Value [-] k_{2,red} - coeff. for calculation of r Formula k_{3,red} - coeff. for calculation of r Value [-] k_{4,red} - coeff. for calculation of r Formula k_{5,red} - coeff. for calculation of r Value [-] 	90,00 igenational setup	
Select all Unselect all	Refresh	 4 k_{6,red} - coeff. for calculation of r Value [-] Coeff. taking account of log 	0,80	OK Canc

Setup 2: Concrete settings

Concrete menu \rightarrow Concrete settings

ncrete settings															- C	
itional annex:											ind	View	-	Advance	d Defau	łt
Description		Symbol		Value		Default		Unit		Chapter	Code		Str	ucture	Check Type	^
all>	P	<al></al>	P	<all></all>	P	<all></all>	P	<all></all>	ρ	<al></al>) <all></all>	5	D <a< td=""><td>D P</td><td><all></all></td><td>)</td></a<>	D P	<all></all>)
Solver setting																
✓ General																
Limit value of unity check		Lim.check		1,0		1,0					Indep	endent	All	(Beam,Bea	Solver setting	
Value of unity check for not calculated unity check		Ncal.check		3,0		3,0					Indep	endent	All	(Beam,Bea	Solver setting	
The coefficient for calculation effective depth of cross-section		Coeffd		0,9		0,9					Indep	endent			Solver setting	
The coefficient for calculation inner lever arm		Coeffz		0,9		0,9					Indep	endent	All	(Beam,Bea	Solver setting	
The coefficient for calculation force, where member as under compression		Coeffcom		0,1		0,1					Indep	endent	All	(Beam,Bea	Solver setting	
✓ Creep																
Type input of creep coefficient		Туре ф		Auto		Auto				Annex B.1	EN 1	992-1-1	All	(Beam,Bea	Solver setting	
Relative humidity		RH		50		50		%		Annex B.1	EN 1	992-1-1	All	(Beam,Bea	Solver setting	
Age of concrete at loading		to		28,00		28,00		day		Annex B.1	EN 1	992-1-1	All	(Beam,Bea	Solver setting	
Age of concrete at the moment considered		t		18250,00		18250,00		day		Annex B.1	EN 1	992-1-1	All	(Beam,Bea	Solver setting	
▷ SLS																
A Default sway type																
Sway around y axis		Sway yy		\checkmark		\checkmark					Indep	endent			Solver setting	
Sway around z axis		Sway zz		\checkmark		\checkmark					Indep	endent	All	(Beam,Bea	Solver setting	
4 Minimal concrete cover																
Design working life				50,00		50,00		year		4.4.1.2(5), table 4	EN 1	992-1-1	All	(Beam,Bea	Solver setting	
 A Risk of corrosion attack 																
Corrosion induced by carbonation				XC3		XC3				4.4.1.2(5)	EN 1	992-1-1			Solver setting	
Corrosion induced by chlorides				None		None				4.4.1.2(5)		992-1-1			Solver setting	
Corrosion induced by chlorides from sea water				None		None				4.4.1.2(5)	EN 1	992-1-1	All	(Beam,Bea	Solver setting	
Freeze / thaw attack				None		None				4.4.1.2(12)	EN 1	992-1-1	All	(Beam,Bea	Solver setting	
Chemical attack				None		None				4.4.1.2(12)		992-1-1	All	(Beam,Bea	Solver setting	
Risk of abrasion attack				None		None				4.4.1.2(13)	EN 1	992-1-1	All	(Beam,Bea	Solver setting	
Possibility of special control																
Risk of casting on atypical surface				Standard		Standard				4.4.1.3(4)	EN 1	992-1-1	All	(Beam,Bea	Solver setting	
 Concrete characteristics 																
Type of concrete				In-situ		In-situ				4.4.1.3(1P, 3)	EN 1	992-1-1	All	(Beam,Bea	Solver setting	
 Internal forces 																
Absolute limit ratio for modification of internal forces		Ratiniat also		5.00		5.00		кN			Inder	endent	1D	(Ream Re	Solver setting	~

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

2_ Member data

It is possible to **overwrite** the data from the general setups per 2D member, namely by means of Member data; see Concrete menu Setting per member – 2D member data. On a plate with Member data appears a label, e.g. CMD1 (= Concrete member data). This label can be selected at any time to view or to adapt the data via the Properties menu. Since Member data are

additional data, it is possible to copy them to other plates, via 'Geometry manipulations' toolbar ¹ or via a right mouse click.

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

ULS design

1_Reinforcement design

Internal forces

Concrete menu \rightarrow Reinforcement design \rightarrow 2D members \rightarrow Internal forces

Basic (centroid)

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

Design (centroid)

The values shown here are different from those in the Results menu.

- The design magnitudes in the **Results** menu are calculated by the **FEM** solver according to some simple formulas specified in EC-ENV.

- The design magnitudes in the **Concrete** menu are calculated by the **NEDIM** solver, where a much finer transformation procedure is implemented, based on the theory of Baumann.

These are the values that will be used for the SCIA Engineer reinforcement design.

Theory of Baumann.

1) Calculation of the lever arm.

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

Principal stresses and directions at both surfaces

 $\sigma_{I-} = 0.48 \text{ MPa} \sigma_{II-} = 0.11 \text{ MPa} \rightarrow \alpha_{7-} = -5.86 = -5.86 \circ$

 $\sigma_{I+} = -0.11 \text{ MPa} \sigma_{II+} = -0.48 \text{ MPa} \rightarrow \alpha_{z+} = -5.86 ^{\circ}$

-> direction for calculation inner lever arm

$$\alpha_{z} = -5.86$$

Recalculated forces to direction of inner lever arm

 $n_z = 0.0 m_z = 4970.4$

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

$$d = 210 \text{ mm}$$

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

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

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

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

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

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

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

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

$$z_z = 85 \text{ mm}$$

If value z can not be calculated it will be calculated according to formula: z = 0.9. d

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

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

Lower surface

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

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

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

Upper surface

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

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

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

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

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

Lower surface

Principal forces at lower surface:

$$n_{I-} = \frac{n_{x-} + n_{y-}}{2} + \frac{1}{2} \cdot \sqrt{(n_{x-} - n_{y-})^2 + 4 \cdot n_{xy-}^2}$$
$$= \frac{23.6 + 5.8}{2} + \frac{1}{2} \cdot \sqrt{(23.6 - 5.8)^2 + 4 \cdot -1.8^2} = 23.8 \text{ kN/m}$$
$$n_{II-} = \frac{n_{x-} + n_{y-}}{2} - \frac{1}{2} \cdot \sqrt{(n_{x-} - n_{y-})^2 + 4 \cdot n_{xy-}^2}$$
$$= \frac{23.6 + 5.8}{2} - \frac{1}{2} \cdot \sqrt{(23.6 - 5.8)^2 + 4 \cdot -1.8^2} = 5.7 \text{ kN/m}$$

Direction of principal forces:

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

Upper surface

Principal forces at upper surface:

$$n_{I+} = \frac{n_{x+} + n_{y+}}{2} + \frac{1}{2} \cdot \sqrt{\left(n_{x+} - n_{y+}\right)^2 + 4 \cdot n_{xy+}^2}$$
$$= \frac{-23.6 + -5.8}{2} + \frac{1}{2} \cdot \sqrt{\left(-23.6 - -5.8\right)^2 + 4 \cdot 1.8^2} = -5.7 \text{ kN/m}$$
$$n_{II+} = \frac{n_{x+} + n_{y+}}{2} - \frac{1}{2} \cdot \sqrt{\left(n_{x+} - n_{y+}\right)^2 + 4 \cdot n_{xy+}^2}$$
$$= \frac{-23.6 + -5.8}{2} - \frac{1}{2} \cdot \sqrt{\left(-23.6 - -5.8\right)^2 + 4 \cdot 1.8^2} = -23.8 \text{ kN/m}$$

Direction of principal forces:

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

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

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

Lower surface

Angles for Baumann's transformation formula

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

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

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

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

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

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

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

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

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

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

Upper surface

Angles for Baumann's transformation formula

 $\begin{aligned} \alpha_{1+} &= \alpha_{inp,1+} - \alpha_{I+} = 0 - .96 = 96 \ ^{\circ} \\ \alpha_{2+} &= \alpha_{inp,2+} - \alpha_{I+} = 90 - .96 = 186 \ ^{\circ} \\ \alpha_{3+} &= \alpha_{con+} - \alpha_{I+} = 135 - .96 = 231 \ ^{\circ} \end{aligned}$

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

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

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

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

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

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

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

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

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

Virtual forces at both surfaces

 $\frac{\text{Lower surface}}{\text{Angles for Baumann's transformation formula}}$ $\alpha_{1+} = \alpha_{\text{inp},1+} - \alpha_{I-} = 0 - -6 = 6^{\circ}$ $\alpha_{2+} = \alpha_{\text{inp},2+} - \alpha_{I-} = 90 - -6 = 96^{\circ}$ $\alpha_{3+} = \alpha_{\text{con-}} - \alpha_{I-} = 45 - -6 = 51^{\circ}$ Recalculated virtual forces at lower surface (acc. to Baumann) $n_{\text{Edsvirt1-}} = \frac{n_{I-} \cdot \sin(\alpha_{2+}) \cdot \sin(\alpha_{3+}) + n_{II-} \cdot \cos(\alpha_{2+}) \cdot \cos(\alpha_{3+})}{\sin(\alpha_{2+} - \alpha_{1+}) \cdot \sin(\alpha_{3+} - \alpha_{1+})}$

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

$$n_{Edsvirt2-} = \frac{n_{I-} \cdot \sin(\alpha_{3+}) \cdot \sin(\alpha_{1+}) + n_{II-} \cdot \cos(\alpha_{3+}) \cdot \cos(\alpha_{1+})}{\sin(\alpha_{3+} - \alpha_{2+}) \cdot \sin(\alpha_{1+} - \alpha_{2+})}$$

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

$$n_{Edsvirt3-} = \frac{n_{I-} \cdot \sin(\alpha_{1+}) \cdot \sin(\alpha_{2+}) + n_{II-} \cdot \cos(\alpha_{1+}) \cdot \cos(\alpha_{2+})}{\sin(\alpha_{1+} - \alpha_{3+}) \cdot \sin(\alpha_{2+} - \alpha_{3+})}$$

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

Upper surface

Angles for Baumann's transformation formula $\alpha_{1-} = \alpha_{inp,1-} - \alpha_{1+} = 0 - .96 = 96^{\circ}$ $\alpha_{2-} = \alpha_{inp,2-} - \alpha_{1+} = 90 - .96 = 186^{\circ}$ $\alpha_{3-} = \alpha_{con+} - \alpha_{1+} = 135 - .96 = 231^{\circ}$

$$\begin{aligned} \text{Recalculated virtual forces at upper surface (acc. to Baumann)} \\ n_{\text{Edsvirt1+}} &= \frac{n_{1+} \cdot \sin(\alpha_{2-}) \cdot \sin(\alpha_{3-}) + n_{11+} \cdot \cos(\alpha_{2-}) \cdot \cos(\alpha_{3-})}{\sin(\alpha_{2-} - \alpha_{1-}) \cdot \sin(\alpha_{3-} - \alpha_{1-})} \\ &= \frac{-5.7 \cdot \sin(186) \cdot \sin(231) + -23.8 \cdot \cos(186) \cdot \cos(231)}{\sin(186 - 96) \cdot \sin(231 - 96)} = -21.7 \text{ kN/m} \\ n_{\text{Edsvirt2+}} &= \frac{n_{1+} \cdot \sin(\alpha_{3-}) \cdot \sin(\alpha_{1-}) + n_{11+} \cdot \cos(\alpha_{3-}) \cdot \cos(\alpha_{1-})}{\sin(\alpha_{3-} - \alpha_{2-}) \cdot \sin(\alpha_{1-} - \alpha_{2-})} \\ &= \frac{-5.7 \cdot \sin(231) \cdot \sin(96) + -23.8 \cdot \cos(231) \cdot \cos(96)}{\sin(231 - 186) \cdot \sin(96 - 186)} = -4.0 \text{ kN/m} \\ n_{\text{Edsvirt3+}} &= \frac{n_{1+} \cdot \sin(\alpha_{1-}) \cdot \sin(\alpha_{2-}) + n_{11+} \cdot \cos(\alpha_{1-}) \cdot \cos(\alpha_{2-})}{\sin(\alpha_{1-} - \alpha_{3-}) \cdot \sin(\alpha_{2-} - \alpha_{3-})} \\ &= \frac{-5.7 \cdot \sin(96) \cdot \sin(186) + -23.8 \cdot \cos(96) \cdot \cos(186)}{\sin(96 - 231) \cdot \sin(186 - 231)} = -3.7 \text{ kN/m} \end{aligned}$$

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

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

Lower surface

Dimensional forces of lower surface transformed to centroid $n_{Ed1-} = n_{Eds1-} + n_{Edsvirt1+} = 25.4 + -21.7 = 3.7 \text{ kN/m}$ $m_{Ed1-} = n_{Eds1-} \cdot z_{-} - n_{Edsvirt1+} \cdot z_{+} = 25.4 \cdot 85 - -21.7 \cdot 124 = 4.9 \text{ kNm/m}$ $n_{Ed2-} = n_{Eds2-} + n_{Edsvirt2+} = 7.7 + -4.0 = 3.7 \text{ kN/m}$ $m_{Ed2-} = n_{Eds2-} \cdot z_{-} - n_{Edsvirt2+} \cdot z_{+} = 7.7 \cdot 85 - -4.0 \cdot 124 = 1.2 \text{ kNm/m}$ $n_{Ed3-} = n_{Eds3-} + n_{Edsvirt3+} = -3.7 + -3.7 = -7.4 \text{ kN/m}$ $m_{Ed3-} = n_{Eds3-} \cdot z_{-} - n_{Edsvirt3+} \cdot z_{+} = -3.7 \cdot 85 - -3.7 \cdot 124 = 0.1 \text{ kNm/m}$

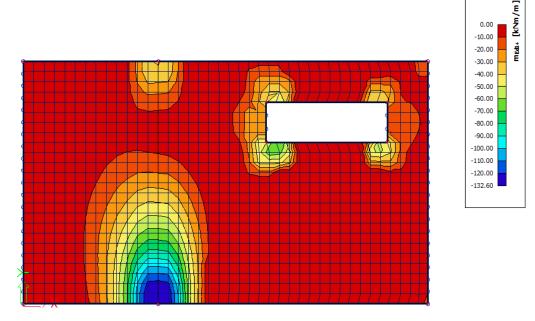
Upper surface

Dimensional forces of upper surface transformed to centroid

$$\begin{split} n_{Ed1+} &= n_{Eds1+} + n_{Edsvirt1-} = -21.7 + 25.4 = 3.7 \text{ kN/m} \\ m_{Ed1+} &= -n_{Eds1+} \cdot z_{+} + n_{Edsvirt1-} \cdot z_{-} = -21.7 \cdot 124 + 25.4 \cdot 85 = 4.9 \text{ kNm/m} \\ n_{Ed2+} &= n_{Eds2+} + n_{Edsvirt2-} = -4.0 + 7.7 = 3.7 \text{ kN/m} \\ m_{Ed2+} &= -n_{Eds2+} \cdot z_{+} + n_{Edsvirt2-} \cdot z_{-} = -4.0 \cdot 124 + 7.7 \cdot 85 = 1.2 \text{ kNm/m} \\ n_{Ed3+} &= n_{Eds3+} + n_{Edsvirt3-} = -3.7 + -3.7 = -7.4 \text{ kN/m} \\ m_{Ed3+} &= -n_{Eds3+} \cdot z_{+} + n_{Edsvirt3-} \cdot z_{-} = -3.7 \cdot 124 + -3.7 \cdot 85 = 0.1 \text{ kNm/m} \end{split}$$

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

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

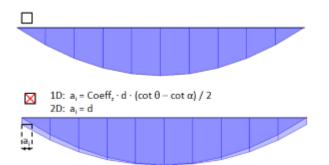


Compare the result for this value mEd,1+ (Concrete menu) with the result for the equivalent value mxD+ (Result menu) shown on p.10.

Despite the different transformation procedures, the general image of the results will be similar for *orthogonal* reinforcement directions (acc. to the local x and y axes). The largest difference is caused by the shift rule that is only taken into account in the design magnitudes calculated by the NEDIM solver (values mEd,1 and mEd,2).

Shift rule

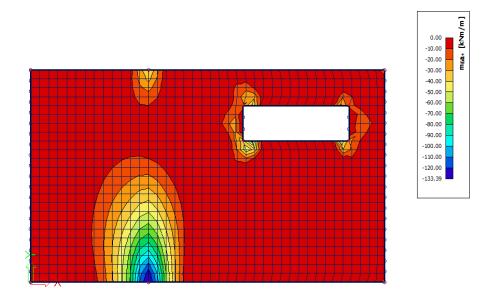
The shift rule takes into account the additional tensile force caused by the shear force by shifting the moment line by a distance a_i. a_i is determined as in the image below.



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

tional annex:									Find		View	Ŧ	Advanced	d Defa	ult	
Description	Symbol		Value		Default		Unit	C	apter	Co	de		Structure	Check Type	^	
I> 🔎	<all></all>	P	<al></al>	P	<all></all>	P	<al></al>	P <	ll> /	P <a< td=""><td> ></td><td>P</td><td>kalb 💫 🔎</td><td><al></al></td><td>ρ</td><td></td></a<>	>	P	kalb 💫 🔎	<al></al>	ρ	
A Risk of corrosion attack																
Corrosion induced by carbonation			XC3		XC3			4.	4.1.2(5)	EN	1992-1-1		All (Beam,Be	Solver setting		
Corrosion induced by chlorides			None		None			4.	4.1.2(5)	EN	1992-1-1		All (Beam, Be	Solver setting		£.
Corrosion induced by chlorides from sea water			None		None			4.	4.1.2(5)	EN	1992-1-1		All (Beam, Be	Solver setting		
Freeze / thaw attack			None		None			4.	4.1.2(12)	EN	1992-1-1		All (Beam, Be	Solver setting		
Chemical attack			None		None			4.	4.1.2(12)	EN	1992-1-1		All (Beam, Be	Solver setting		
Risk of abrasion attack			None		None			4.	4.1.2(13)	EN	1992-1-1		All (Beam,Be	Solver setting		
Possibility of special control																
Risk of casting on atypical surface			Standard		Standard			4.	4.1.3(4)	EN	1992-1-1		All (Beam, Be	Solver setting		
Concrete characteristics																
Type of concrete			In-situ		In-situ			4.	4.1.3(1P, 3)	EN	1992-1-1		All (Beam, Be	Solver setting		
 Internal forces 																
Absolute limit ratio for modification of internal forces	Ratioint.abs		5,00		5,00		kN			Ind	lependent		1D (Beam,B	Solver setting		
Relative limit ratio for modification of internal forces	Ratioint, rel		0,10		0,10		-			Ind	lependent		1D (Beam,B	Solver setting		
Modification of internal forces			No		No					Ind	lependent		1D (Beam,B	Solver setting	J	
Use equivalent first order value			\checkmark		\checkmark			5.	3.8.2(2)	EN	1992-1-1		Column	Solver setting		
Determination of unfavourable direction			Auto		Auto			5.	3.9	EN	1992-1-1		Column	Solver setting		
/ Internal former LILS																
Take into account additional tensile force caused by shear (shift rule)	1				\checkmark			9.	2.1.3(2)	EN	1992-1-1		Beam, Rib, Pl	Solver setting	J	
Use minimum value of eccentricity					\checkmark			6.	1(4)	EN	1992-1-1		Column	Solver setting	J	
Use geometric imperfection			\checkmark		\checkmark			5.	2(5)	EN	1992-1-1		Column	Solver setting		
Use second order effect			\checkmark		\checkmark			5.	3.8	EN	1992-1-1		Column	Solver setting		
Estimation ratio of longitudinal reinforcement for recalculation internal forces in des	μs		2,00		2,00		%	5.	3.3.1	EN	1992-1-1		Column	Solver setting		
Shear force reduction above supports								6.	2.1(8)	EN	1992-1-1		Beam,Beam	Solver setting		
Moment reduction above supports								5.	3.2.2 (4)	EN	1992-1-1		Beam,Beam	Solver setting	J	
Internal forces SLS																
Use geometric imperfection								5.	2(5)	EN	1992-1-1		Column	Solver setting	J	
✓ Design As																
Coefficient for reduction of strength of the concrete in compressive concrete	Redfod		0,85		0,85		-			EN	1992-1-1		Plate,Wall,S	Solver setting		
A Beam, Column, Rib, Beam Slab																
Limit ratio of bending moment for uniavial method	OM lim		0.10		0.10					Ind	lenendent		1D (Beam B	Solver setting	×	

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



Provided reinforcement

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

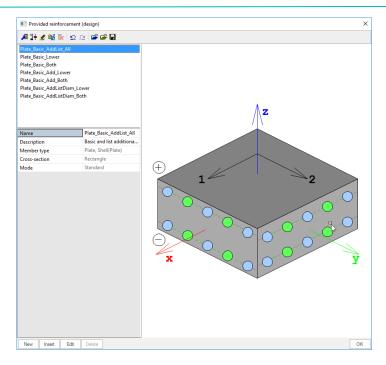
- Compare the template with the calculated theoretical reinforcement. By doing this it is easy to see where this basic template is not sufficient.
- Perform the punching design, Crack width check and the code dependent deflections.

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

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

ign defaults																	
tional annex:										Find		View	-	Advanced		Default	
Description		Symbol		Value		Default		Unit	C	Chapter	Code			Structure	C	heckType	
ll>	ρ	<al></al>	P	<all></all>	P	<al></al>	P	cal> 🏅	0 0	al> 🔎	<al></al>		P	(al)	ρ <	all> 🔎	
Design defaults																	
Beam (Rib)																	
▷ Beam slab																	
▷ Column																	
4 Plate, Shell(Plate)																	
✓ Longitudinal																	
Use a template of provided reinforcement				\checkmark							Indep	pendent		Plate,Shell(F	PI D	esign defaul	
Template of provided reinforcement (Plate)				Plate_Basic_A	d	late_Basic_	A				Indep	pendent		Plate,Shell(F	PI D)esign defaul	
 Upper (z+) 																	
Type of cover of upper surface		Typec1,u		Auto		Auto			4.	.4.1	EN 1	992-1-1		Plate,Shell(F	PI D	esign defaul	
Angle of reinforcement of first upper layer		α1.u		0.00		0,00		deg			EN 1	992-1-1		Plate,Shell(F	PI D	esign defaul	
Angle of reinforcement of second upper layer		α.2.u		90,00		90,00		deg			EN 1	992-1-1		Plate,Shell(PI D)esign defaul	
Lower (z-)																	
Shear																	

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



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

ided reinforcement (design) edit - Plate_Basic_AddList_All bertype Plate, Shell(Plate) section section Standard	✓ ■0 ■0	nt						- 0	
	Definition of Basic reinfor	cement: By	Diameter ×						
			Basic reinforceme	nt		Additiona	al reinforcement		
	Surface [direction]	Diameter	Spacing	Area	Туре	Diameter	Spacing	Area	
		[mm]	[mm]	[mm^2/m]		[mm]	[mm]	[mm^2/m]	
	Upper [1+]	10,0	200	393	List by spacing	10,0	0;100;150;200	0;785;524;39	
	Upper [2+]	10,0	200	393	List by spacing	10,0	0;100;150;200	0;785;524;39	
	Lower [1-]	10,0	150	524	List by spacing	10,0	0;100;150;200	0;785;524;39	
	Lower [2-]	10,0	150	524	List by spacing	10,0	0;100;150;200	0;785;524;39	93
							0	(Ca	

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

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

Note:

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

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

Description			Symbol		Value	De	efault	Unit		Chapter		Code		Structure	Ch	neckType
>		P	<all></all>	P	<al></al>) <a< th=""><th>all> 🔎</th><th><all></all></th><th>ρ</th><th><all></all></th><th>P</th><th><all></all></th><th>ρ</th><th><all></all></th><th>) <a< th=""><th>all></th></a<></th></a<>	all> 🔎	<all></all>	ρ	<all></all>	P	<all></all>	ρ	<all></all>) <a< th=""><th>all></th></a<>	all>
Design defa	aults															
Beam (R)	λib)															
Beam sla	ab															
D Column																
✓ Plate, St	hell(Plate)															
 Longi 	itudinal															
Us	se a template of provided reinforcement				×	\checkmark						Independent		Plate,Shell(Pl	De	esign defa
1	Template of provided reinforcement (Plate)				Plate_Basic_Ad	. Pla	ate_Basic_A					Independent		Plate,Shell(Pl	De	esign defa
.⊿ Up	pper (z+)															
	Type of cover of upper surface		Typec1,u		Auto	Au	ıto			4.4.1		EN 1992-1-1		Plate,Shell(Pl	De	esign defa
	Angle of reinforcement of first upper layer		α1.u		0,00	0,0	00	deg				EN 1992-1-1		Plate,Shell(Pl	De	esign defau
	Angle of reinforcement of second upper layer		α2.u		90,00	90),00	deg				EN 1992-1-1		Plate,Shell(Pl	De	esign defau
⊿ Lo	ower (z-)															
	Type of cover of lower surface		Typec1,I		Auto	Au	to			4.4.1		EN 1992-1-1		Plate,Shell(Pl	De	esign defau
	Angle of reinforcement of first lower layer		α1.1		0,00	0,0	00	deg				EN 1992-1-1		Plate,Shell(Pl	De	esign defau
	Angle of reinforcement of second lower layer		α2.1		90,00	90),00	deg				EN 1992-1-1		Plate,Shell(Pl	De	esign defa
Shear	r															

- Theoretically required reinforcement

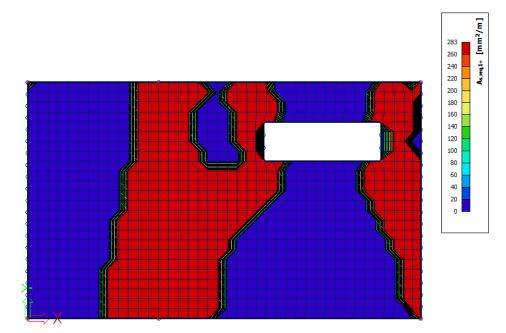
a.

Concrete \rightarrow Reinforcement design \rightarrow 2D members \rightarrow Reinforcement design (ULS)

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

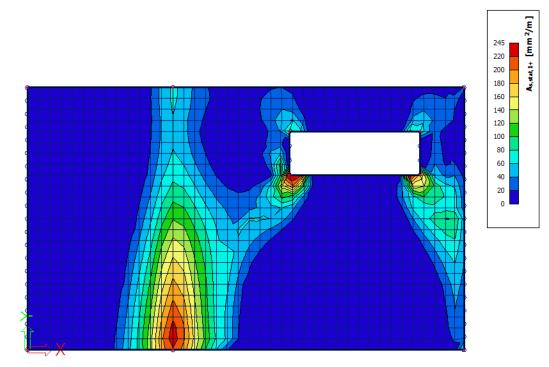
- **Required:** These values represent the theoretical reinforcement calculated by SCIA Engineer. This takes into account the detailing provisions.

Plat	e, Shell(Plate)		
4 L	ongitudinal		
	Check min. ratio of principal reinforcement		\checkmark
	Type of the minimum tension principal reinforcement for the upper surface		Auto
	Type of the minimum tension principal reinforcement for the lower surface		Auto
	Check max. ratio of principal reinforcement		\checkmark
	Check min. transverse ratio of secondary reinforcement		
	Check min. bar distance		\checkmark
	Minimal bar distance	slp.min	20
	Check max.spacing of principal longitudinal reinforcement		\checkmark
	Check max.spacing of secondary longitudinal reinforcement		\checkmark
4 5	ihear		
	Check min. ratio of shear reinforcement		\checkmark
	Check min. thickness of member with shear reinforcement		\checkmark
	Min. thickness of member with shear reinforcement	hmin	200
	Check max. spacing of shear links		\checkmark
	Max. spacing of shear links	Coeff _{smax.p.s}	0,8



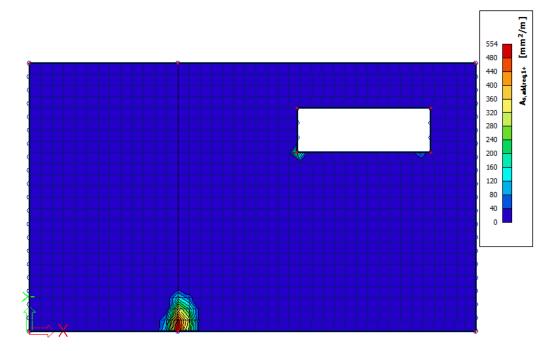
As,req1+: Theoretical required reinforcement on the top side of the plate (positive z direction) in the first reinforcement direction. Taking into account the detailing provisions.

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



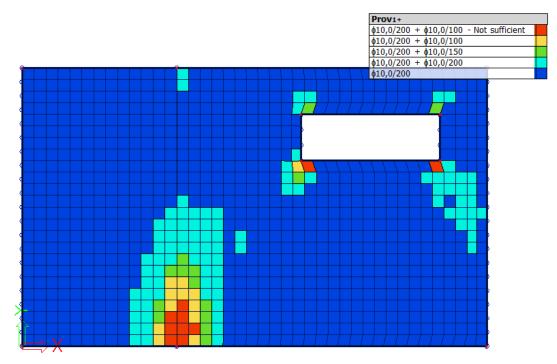
As,stat1+: Theoretical required reinforcement on the top side of the plate (positive z direction) in the first reinforcement direction. **Without** taking into account the detailing provisions.

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



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

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



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

Calculation of longitudinal reinforcement

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



Calculation of shear reinforcement

Before calculating the shear reinforcement two checks are done:

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

$$v_{\text{Rd,max}} = \frac{\alpha_{\text{cw}} \cdot b_{\text{w}} \cdot z \cdot v_{1} \cdot f_{\text{cd}}}{\left(\cot g\left(\theta\right) + tg\left(\theta\right)\right)}$$

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

$$\begin{aligned} v_{Rdc} &= \max\left(v_{Rdc}; v_{Rdcmin}\right) \\ v_{Rdc} &= \max\left(10^{6} \cdot \left(C_{Rdc} \cdot k \cdot (100 \cdot \rho_{1} \cdot f_{dk})^{\frac{1}{3}} + k_{1} \cdot \sigma_{qp}\right) \cdot d; 0\right) \\ v_{Rdcmin} &= \max\left(10^{6} \cdot \left(v_{min} + k_{1} \cdot \sigma_{qp}\right) \cdot d; 0\right) \end{aligned}$$

Check shear capacity (without shear reinforcement)

 $\frac{Check v_{Rd,max}}{v_{Ed} = 145 \text{ kN/m} \le v_{Rd,max} = 653 \text{ kN/m} \text{ (OK)}$ $\frac{Check v_{Rdc}}{v_{Rdc}}$

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

Statically required shear reinforcement

 $f_{ywd,req} = \frac{f_{ywk,req}}{\gamma_{s}} = \frac{500}{1.15} = 435 \text{ MPa}$ $A_{sw,req} = \frac{v_{Ed}}{\not{z} \cdot f_{ywd,req} \cdot \cot g(\theta)} = \frac{145.5}{0.18 \cdot 435 \cdot \cot g(40)} = 1559 \text{ mm}^{2}/\text{m}$ Required shear reinforcement $32\varphi 8/m^{2} = > A_{sw,req} = 1559 \text{ mm}^{2}/\text{m}^{2}$

Shear resistance (vRD,max) is not sufficient

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

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

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

Averaging strips

Concrete menu \rightarrow Result tools 2D \rightarrow Averaging strip

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

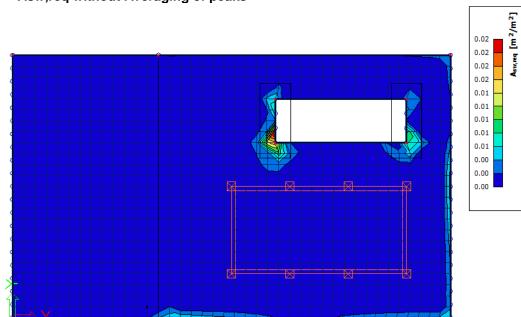
RS RS	×	
Name	RS1	
Туре	Strip	·
Width [m]	0,700	
Direction	both	*

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

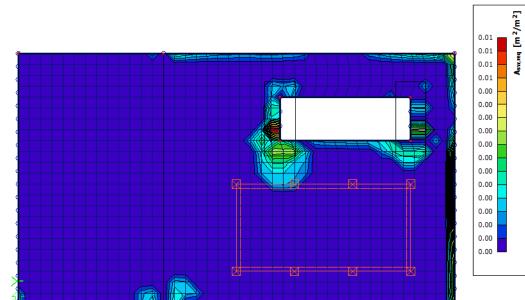
When asking results now, select the option 'Averaging of peak' in the Properties menu!

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

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



Asw,req without Averaging of peaks



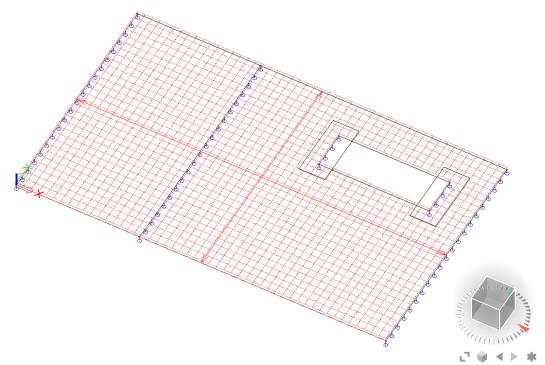
Asw,req with averaging of peaks

2_Practical reinforcement design

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

	Name	RR1	_
	2D member	Slab1	
	4 Reinforcement		
<u> </u>	Туре	Bars	÷
	Material	B 500A +	
	Surface	Upper	Ŧ
1	Number of directions	2	Ŧ
	Direction closest to surface	1	Ŧ
	Angle of first direction [deg]	0,00	
	1		
	Diameter (dl) [mm]	10,0	
	Concrete cover (cl, cu) [mm]	30	
	Bar distance (sl) [mm]	200	
	Offset [mm]	0	
	Reinf. area [mm^2/m]	393	
	4 2		
	Diameter (dl) [mm]	10,0	
	Concrete cover (cl,cu) [mm]	40	
	Bar distance (sl) [mm]	200	
	Offset [mm]	0	
	Reinf. area [mm^2/m]	393	
	Total weight [kg]	58,26	
	4 Geometry		
	Geometry defined by	Polygon	Ŧ
	Actions		
	Load from setup	>>>	
			_
		OK Cance	4

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



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

Crack control

1_Input data for crack control

Maximum crack width

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

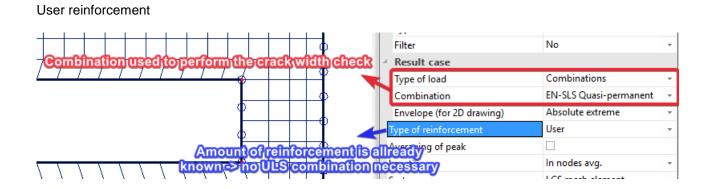
Concrete setup	X
Type of values NA building Type of functionality Hollow core beams Prestressing Image: State of the state of	Name EC-EN Concrete General ULS SLS General National annex k_3creack - coefficient for calculation maxin Value [-] 3,40 k_dotack - coefficient for calculation maxin Value [-] 0,42 wmax - for non-prestressed structure 7.3.1 Values (nm) 0,42.03/0,3 Detailing provisions 0,4/0,3/0,3 P Detailing provisions 0,4/0,3/0,3 P Detailing provisions 0,4/0,3/0,3
Select all Unselect all Refresh	Load default NA parameters OK Cancel

Type of used reinforcement

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

стаск міцін (эсэ) vame Selection All Type of selection No Filter Type of load Combinations EN-SLS Quasi-permanent Combination Absolute extreme Envelope (for 2D drawing) Combination used to calculate the theoretical reinforcement used in the crack width check Required Result case for required rei.. Type of load Combinations EN-ULS (STR/GEO) Set B Combination Averaging of peak

Required/provided reinforcement



Theoretical background

Crack appearance

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

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

With:

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

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

Calculation of crack width

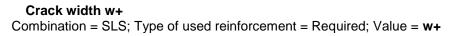
$$w = \varepsilon_{sm_cm} \cdot s_{r,max}$$

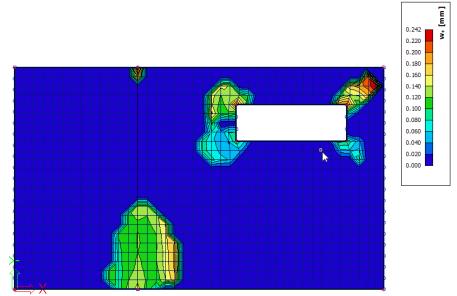
With:

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

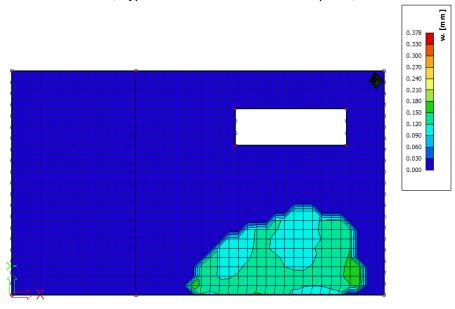
2_Results for required theoretical reinforcement

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



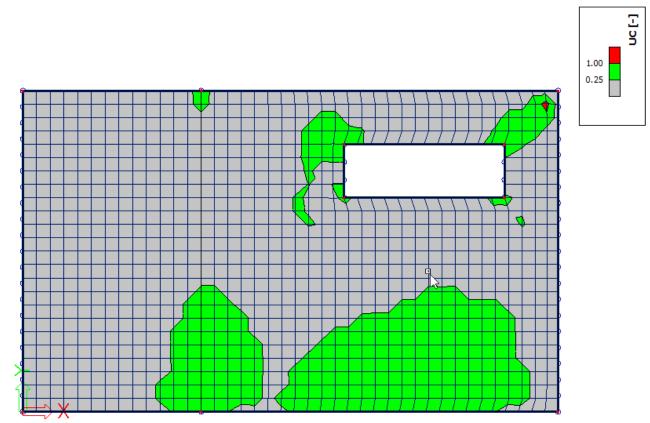


Crack width w-Combination = SLS; Type of used reinforcement = Required; Value = **w-**



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





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

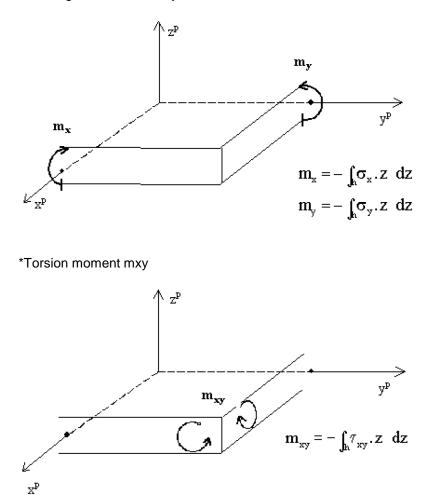
Annexes

Annex 1: Conventions for the results on 2D members

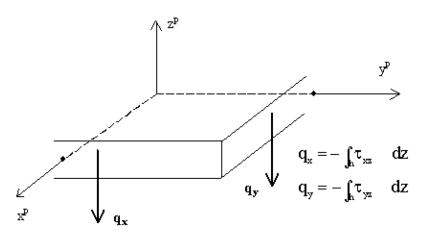
1_ Basic magnitudes = Characteristic values

Bending (plates, shells)

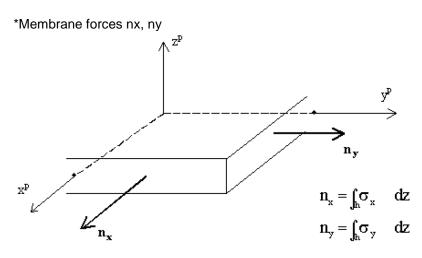
*Bending moments mx, my



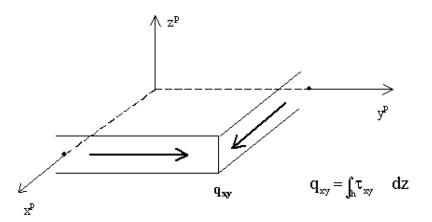
*Shear forces qx, qy (=vx, vy)







*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 \rightarrow 4 Locations

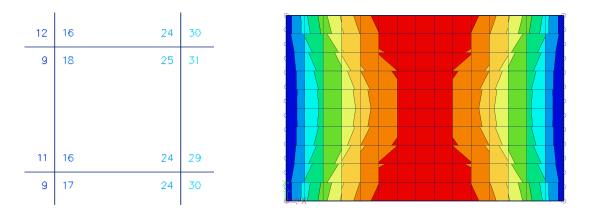
During a calculation in SCIA Engineer, the node deformations and the reactions are calculated exactly (by means of the displacement method). The stresses and internal forces are derived from these magnitudes by means of the assumed basic functions, and are therefore in the Finite Elements Method always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

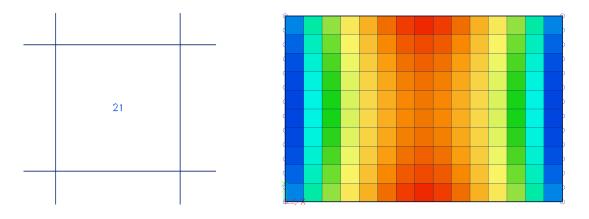
All of the values of the results are taken into account, there is no averaging. In each node are therefore the 4 values of the adjacent mesh elements shown. If these 4 results differ a lot from each other, it is an indication that the chosen mesh size is too large.

This display of results therefore gives a good idea of the discretisation error in the calculation model.



2_ In centres

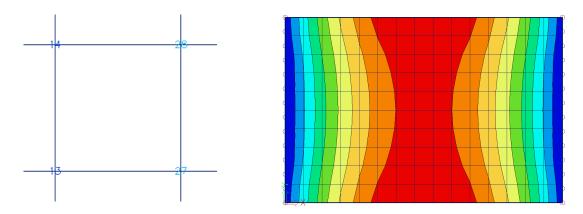
Per finite element, the mean value of the results in the nodes of that element is calculated. Since there is only 1 result per element, the display of isobands becomes a mosaic. The course over a section is a curve with a constant step per mesh element.



3_ In nodes, average

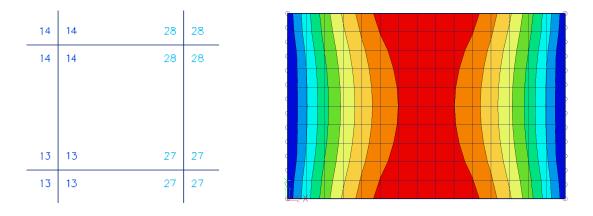
The values of the results of adjacent finite elements are averaged in the common node. Because of this, the graphical display is a smooth course of isobands.

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 and which have the same directions of their local axes. This resolves the problems mentioned at the option 'In nodes, average'.



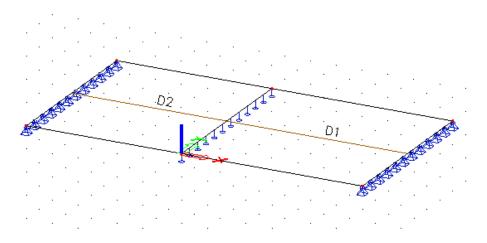
Accuracy of the results

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:

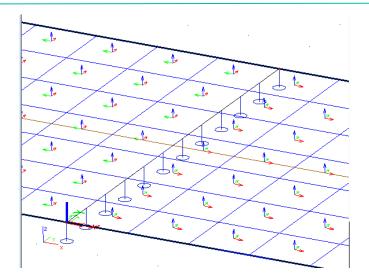


In order to display the mesh local axes, you need first to generate the mesh. You can use the button 'Mesh generation' or **Main menu > Calculation, Mesh > Mesh generation**

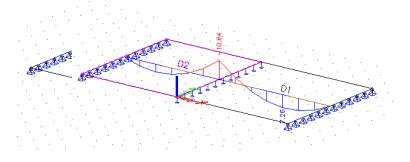
Afterwards, the mesh and local axes can be displayed from 'Set View parameters for all': right click of the mouse on the screen

Check / Uncheck group			Lock position		
		Attributes	🌌 Misc. 🚽	🔍 View	
	🕿 Structure 🛛 🔠 Labels	📥 Mode	I 🕹 I	_oads/masses	
	Check / Uncheck all				
Ŧ	Service				
_	Structure				
-	Panel				
Ξ	Structure nodes				
	Display	~			
	Mark style	Dot		•	
F	Mesh				
	Draw mesh	~			
	Free edges	V			
	Display mode	wire	d	_	
Ξ	Local axes				
	Nodes				
_	Members 2D				
Ļ	Mesh elements	V			
Ы	Sections Members 1D				
	Members 1D Members 2D	<u>۲</u>			
	Members 20	•			

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

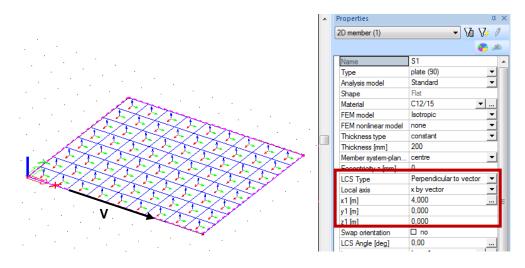
Moment diagram 2

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:

Properties # ×				
2D member (1) - 🗸 🕅				
	🐔 🙈			
Name	D2			
Туре	plate (90)			
Analysis model	Standard 💌			
Shape	Flat			
Material	C30/37 💌			
FEM model	Isotropic 💌			
FEM nonlinear model	none 💌			
Thickness type	constant 💌			
Thickness [mm]	200			
Member system-plan.	centre			
Eccentricity z [mm]	0			
LCS Type	Standard 🗸 🗸			
Swap orientation	Standard			
LCS Angle [deg]	Perpendicular to vector			
Layer	Congruent with line Smallest angle with vector			
□ Nodes	Tilt of vector defined by point			
N1	Tilt of vector normal to line			
N4	abso			

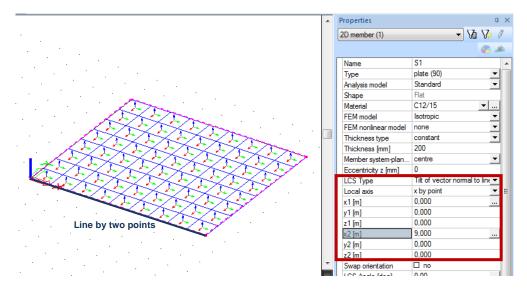
1_ Perpendicular to vector

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



2_ Tilt of vector normal to line

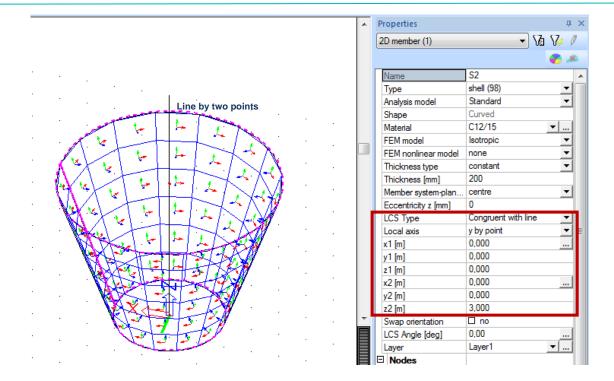
This method is similar to the one above. Instead of introducing a vector, a line is defined between two points. (x1 y1 z1) are the coordinates of the first point and (x2 y2 z2) are the coordinates of the second point. The x(y) axis is perpendicular to the introduce line and points towards it.



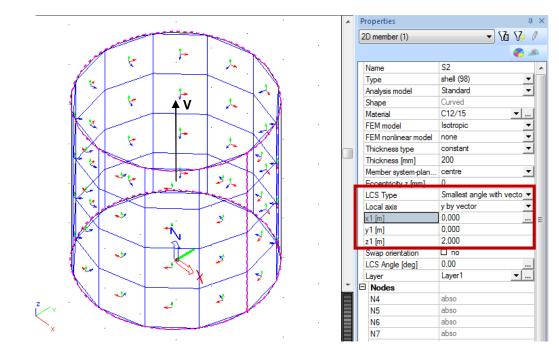
The coordinates of the points that define the line can be introduced in the properties window. You can also click on the $\boxed{}$ button next to x1 and x2 and define these points graphically.

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 :



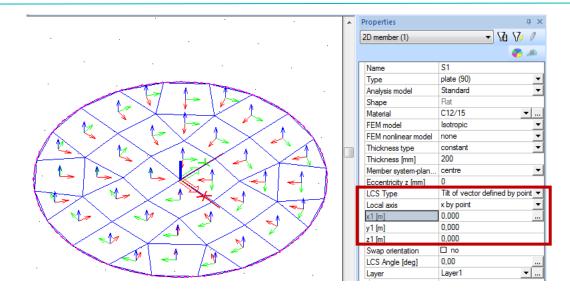
4_ Smallest angle with vector



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.



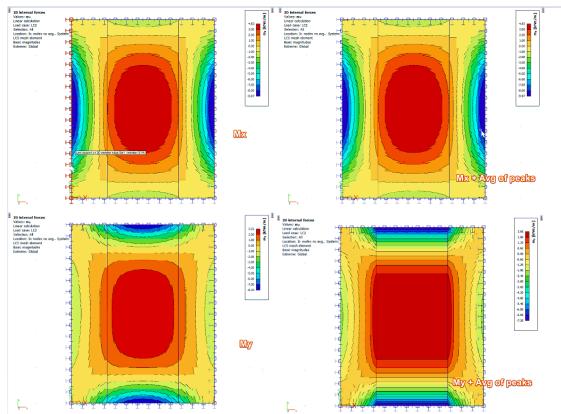
Remark: All the methods above are about how to adapt the x and y axes. The local z axis is defined automatically by the program but its orientation can be changed by ticking the box 'Swap orientation' in the Properties menu

y i [m]	0,000
z1 [m]	0,000
Swap orientation	🗆 no
LCS Angle [deg]	0,00

Annex 4: Averaging strips

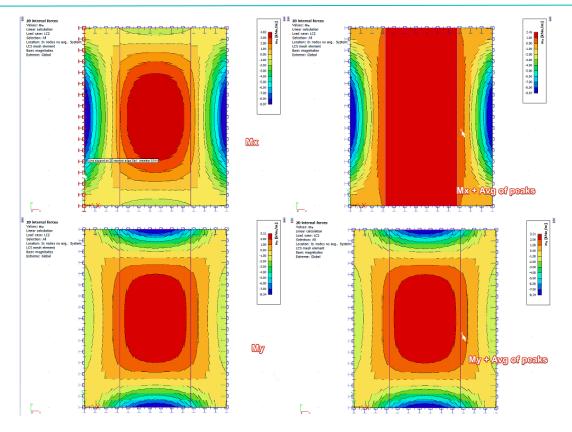
Averaging strips average the results perpendicular to the direction of the internal force. The option 'Direction' in the settings of the averaging strip is dependent on how the averaging strip is drawn.

1) Direction = Longitudinal

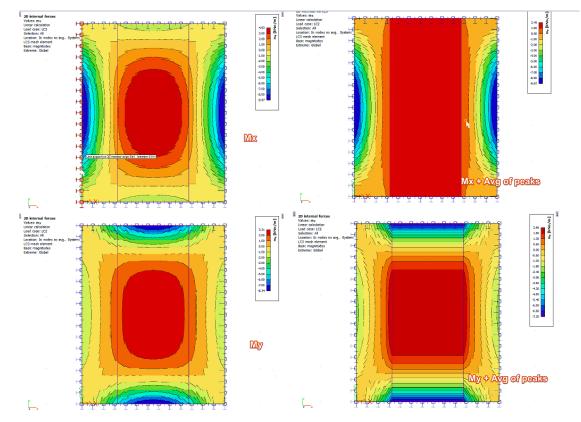


Longitudinal means that the averaging is done in the longitudinal direction of the strip. In the example above this is the y-direction. This means that the averaging is done for my. The values my are averaged in the x-direction.

2) Direction = perpendicular



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



3) Direction = Both

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