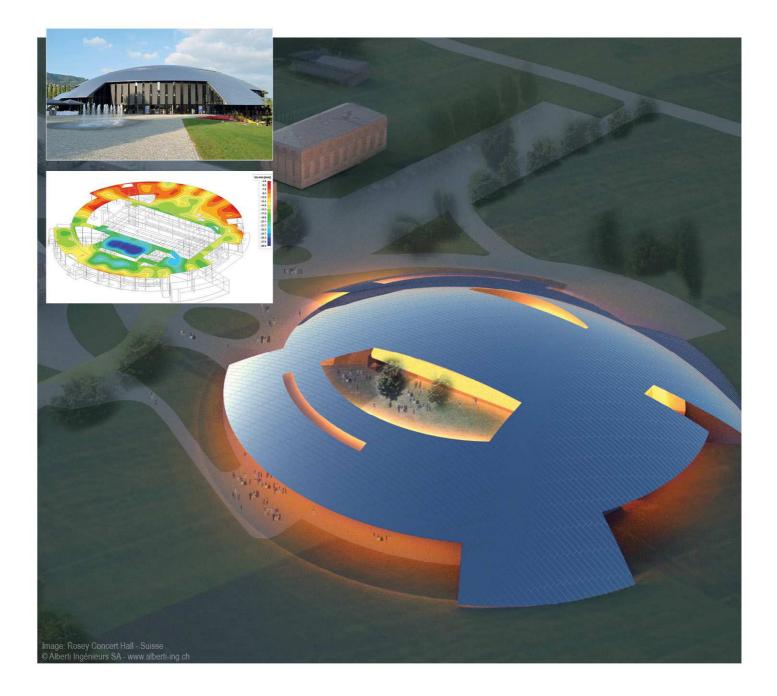
# **SCIAENGINEER**



# Advanced Concept Training Reinforced concrete (EN 1992) – 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 2015 SCIA nv. All rights reserved.

# Table of contents

Introduction	.5
Plate design	.6
Model	6
1_Input of geometry	6
2_Loads	7
3_Finite element mesh	8
4_Results for the linear calculation	
Concrete setups	
1_General setups	
2_ Member data	
ULS design	
1_Theoretical reinforcement design	
2_Practical reinforcement design ULS + SLS design	
1_Input data for crack control	
2_Theoretical reinforcement design	
3_Practical reinforcement design	
Crack control	
1_Input data for crack control	33
2_Results for required theoretical reinforcement	33
3_Results for basic reinforcement	
Annexes	37
Annex 1: Conventions for the results on 2D members	
1_Basic magnitudes = Characteristic values	37
2_Principal magnitudes	38
3_Design magnitudes	
Annex 2: Results in Mesh elements and Mesh nodes $\rightarrow$ 4 Locations	
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	
2D Reinforcement Concrete Design of Walls acc. to EN 1992-1-1:2004	46

# 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

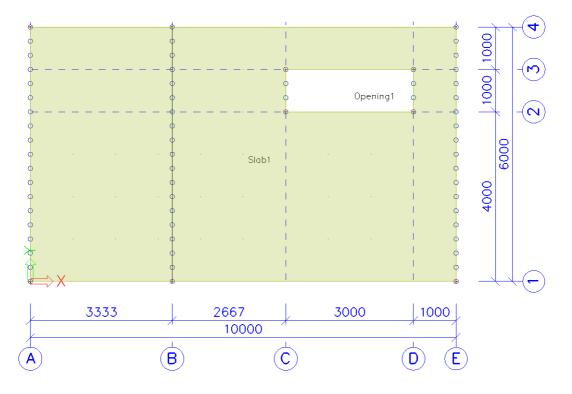
# Model

# 1\_Input of geometry

Project data: 2D environment = Plate XY

	Data		Material	
NAME OF TAXABLE PARTY.	Name:	Example project	Concrete	
and the			Material	C20/25
60	Part	ACT Reinforced Concrete	Reinforcement mat.	AND INCOMENTATION OF THE OWNER OF
The state	Part	ACT Reinforced Concrete	Steel Timber	
100			Other	
	Description:	Plate design	Aluminium	
			Automotion	
No.	Author:	Scia Engineer		
-	, iduitori			
	Date:	27.07.2011		
and a	Date.	27.07.2011		
alle				
100	<b>O</b> 1 <b>1</b>		Code	
	Structure:		National Code:	
110	Plate XY	Y	EC-EN	*
West .	Project Level:	Model:	National annex	
3.57	Advanced	V One V	EC-EN	~
	ravancea	One		100

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:

Name	Slab1	
Туре	plate (90)	•
Analysis model	Standard	τ.
Shape	Flat	
<sup>-</sup> δα Material	C25/30	×
FEM model	Isotropic	•
FEM nonlinear model	none	*
Thickness type	constant	•
Thickness [mm]	250	
LCS Type	Standard	•
LCS Angle [deg]	0,00	
Layer	Laag1	▼
I Nodes		
Actions		
Table edit geometry		>>

Line support on 2D member e	dg	e		X
RX x1 x1 x2 z		Name Z Rx Ry Geometry System Position x1 Position x2 Coord. definition Origin	Sle Rigid Free GCS 0,000 1,000 Rela From start	• • •
X Y			ОК	Cancel

# 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 🛛 🔀			
🔎 🕃 🖋 🖬 🖉 🖂	2   😂   😂 🖬   📶	▼ 7	
LC1 - Self weight	Name	LC1	
LC2 - Walls	Description	Selfweight	
LC3 - Sevice load	Action type	Permanent	•
	LoadGroup	LG1	▼
	Load type	Selfweight	•
	Direction	-Z	•
New Insert Edit De	lete		Close
🎜 🤮 🗶 📸 🔛 🗠 🗠	😂 🖙 🖬 🔤 🔤	▼ 7	
LG1	Name	LG2	
LG2	Relation	Standard	-
	Load	Variable	•
	EC1 - load type	Cat B : Offices	•
New Insert Edit De	elete		Close

#### Load combinations

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

Combinations			
🏓 🤮 🗶 😫 🕰	Input combinations	•	
ULS	Name	ULS	
SLS	Description		
	Туре	EN-ULS (STR/GEO) Set B	-
	Contents of combination		
	LC1 - Self weight [-]	1,00	
	LC2 - Walls [-]	1,00	
	LC3 - Sevice load [-]	1,00	
	Actions		
	Explode to envelopes		>>>
	Explode to linear		>>>
New Insert Edit [	Delete		Close

#### **Result classes**

All	UL	S+	-SI	S
/ WI			0	-0

Result classes	
🔎 🤮 🛃 💽 🗠 🎒 🗛	• 7
All ULS+SLS Name	All ULS+SLS
Description	
🗆 List	
	ULS - EN-ULS (STR/GEO) Set B
	SLS - EN-SLS Quasi.
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:

Solver setup	
Name	
Solver	
Advanced solver options	
Neglect shear force deformation (Ay, Az >> A)	
Bending theory of plate/shell analysis	Mindlin
Type of solver	Direct
Number of thicknesses of rib plate	20
Number of sections on average member	10
Maximal acceptable translation [mm]	1000,0
Maximal acceptable rotation [mrad]	100.0
Coefficient for reinforcement	1
i 🖻 🖬	OK Cancel

**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

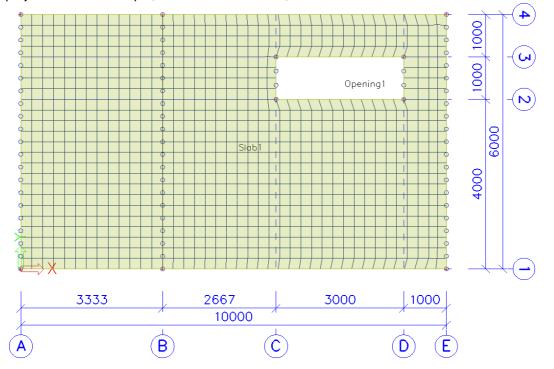
- Structure tab > Mesh > Draw mesh
- Labels tab > Mesh > Display label

**Mesh refinement** 

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

	Nesh setup	
	Mesh	
	Minimal distance between two points [m]	0,001
	Average number of tiles of 1D element	1
	Average size of 2D element/curved element [m]	1,000
Ð	1D elements	
	Minimal length of beam element [m]	0,100
	Maximal length of beam element [m]	100,000
	Average size of cables, tendons, elements on subsoil, nonlinear soil	1,000
	Generation of nodes in connections of beam elements	
	Generation of nodes under concentrated loads on beam elements	
	Generation of eccentric elements on members with variable height	
	No. of FE per haunch	5
	Apply the nodal refinement	No members
Ð	2D elements	
	To generate predefined mesh	

'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:

Properties	>
2D element - Interne krad	chten (1) 💽 🔽 🌾 🦉
Name	Results
Selection	All
Type of loads	Combinations -
Combinations	ULS 🗸
Filter	No
System	Local 🔹
Rotation [deg]	0,00
Averaging of peak	
Location	In nodes, avg. on macro 🔹
Type forces	Basic magnitudes 🔹
Envelope	Minimum
Drawing	Standard -
Values	mx 💌
Extreme	Global
Drawing setup	
Actions	
Refresh	>>>
Detailed results in mesh	node >>>
Preview	>>>

-System

Local: according to the local axes of the *individual* mesh elements LCS-Member 2D: according to the LCS of the 2D member <u>Attention</u> 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

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

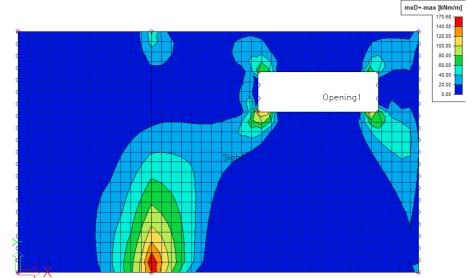
#### **Types of results**

#### **Basic magnitudes**

 Image: market in the second second

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:

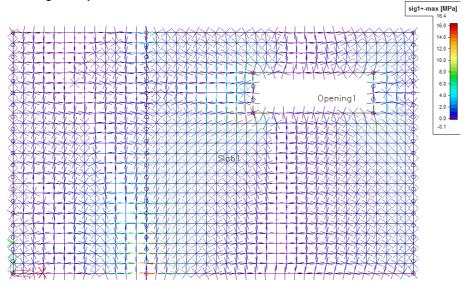
nx		nxD
ny	~	nyD
nxy		ncD

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  $\leq$  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 > 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.

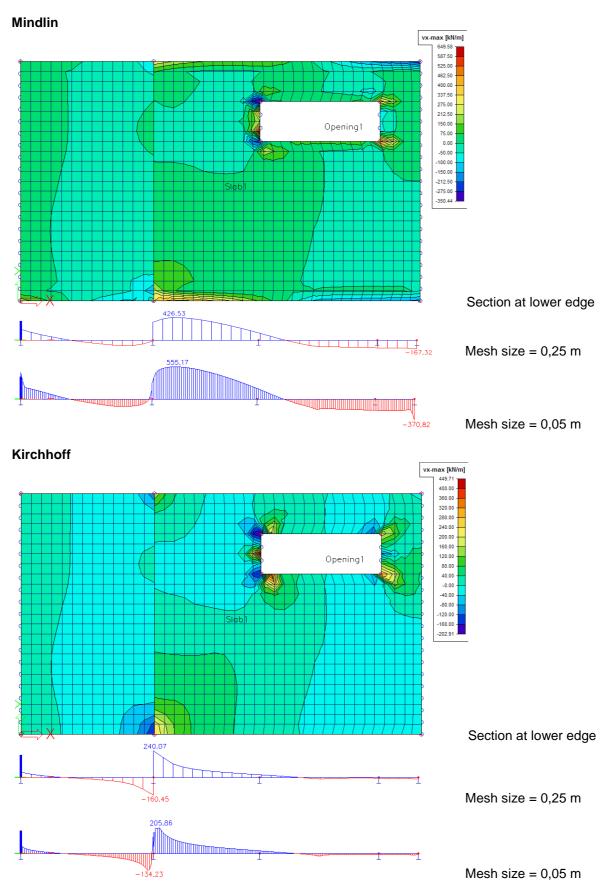
The first direction is the direction of maximum tension (or minimum compression). The second direction is the direction of maximum compression (or minimum tension).

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

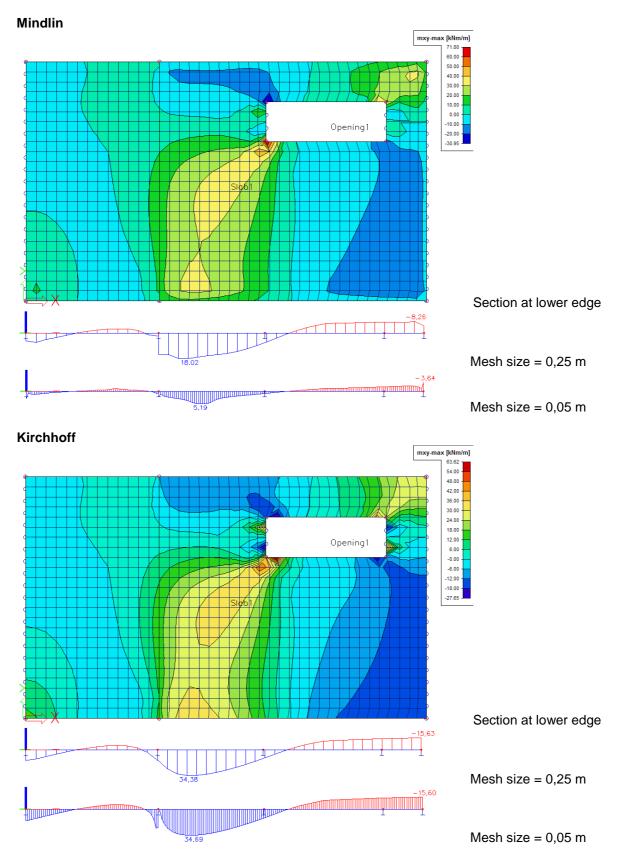
Combination = ULS; Type forces = Basic magnitudes; Envelope = Maximum; Values = vx



13

## Torsion moment mxy

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



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

# **Concrete setups**

# 1\_General setups

**Setup 1: National Determined Parameters** 

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

Type of values		Standard EN	Concrete	
NA building	12	Concrete	General	
Type of functionality		🖶 General	Concrete	
Hollow core beams	⊠	Concrete	National annex	
Prestressing	Ø		E EN 1992 1 1	1
			Values [-]	1,50 / 1,20
			E fck_max - maximum value of the characteristic	
			Value (MPa)	90,00
		General	□ alpha cc - coeff. taking account of long term ef	
		Prestressing	Value [-]	1.00
		Allowable stress	alpha ct - coeff. taking account of long term eff	
		<ul> <li>Stress limitation during tensioning</li> </ul>	Value [-]	1,00
			E k1_red - coeff. for calculation of ratio of distrib	
			Value [-]	0,44
			E k2_red - coeff. for calculation of ratio of distrib	
			Formula	Formula
		- Doanno	k3_red - coeff. for calculation of ratio of distrib	
			Value [-]	0,54
			k4_red - coeff. for calculation of ratio of distrib	
			Formula	Formula
			k5_red - coeff. for calculation of ratio of distrib	
			Value [-]	0,70
			k6_red - coeff. for calculation of ratio of distrib	
			Value [-]	0,80
			□ alpha_cc.pl-coeff. taking account of long term	
			Value [-]	7,00
			alpha_ct.pl-coeff. taking account of long term e	
			Value [-]	7,00
			Non-prestressed reinforcement	
			National annex	
			□ EN_1992_1_1	
			🖯 gamma_s	
			Values [-]	1,15/1,00
			eps_ud/eps_uk - ratio of design and characteri	
			Value [-]	0,90
	iding       i       i       General       i         of functionality       i       General       i         core beam       i       General       i         essing       is       i       General       i         Pestressed reinforcement       Durbility and concrete cover       i       Mational annex       i         ULS       General       isonral       genma_c       isonral       isonral         Pestressed reinforcement       Durbility and concrete cover       genma_c       isonral       isonral         Pestresseing       SLS       General       isonral       genma_c       isonral         Pestresseing       SLS stess limitation       Stess limitation       isonral       isonral       isonral         SLS stess limitation       Detailing provisions       Columns       Columns       Formula       kL_red - coeff. for calculation of ratio of distrib       Value []       0.54         Value []       Ocide       Coeff. for calculation of ratio of distrib       Formula       KS.reg - coeff. for calculation of ratio of distrib       Formula         K2_red - coeff. for calculation of ratio of distrib       Value []       0.54       0.70       Kd_red - coeff. for calculation of ratio of distrib       Value [] <td< td=""><td></td></td<>			
			11 000 1	C00.00
Select all Ur	iselect all	Refresh	Load default NA parameters	OK Cano

# Setup 2: Basic settings

Concrete menu > Design defaults

Type of values			Name	EC-EN	
Design default	Ø	Concrete	Concrete		
Drawing settings		😑 Design defaults	Design defaults		
and a second distance of the second dis		Concrete cover	Concrete cover		
		- 2D structures	Use min concrete cover	8	
		Punching	Design working life [years]		
			Exposure class	XC3	
			Abrasion class	None	
			Type of concrete		
		Punching         Use min concrete cover         Ide           Design working life (years)         50         ¥           Exposure class         XC3         ¥           Abrasion class         None         ¥			
				Normal surface	× × ×
				🗆 no	
					Y
	Upper reinforcement         10,0           Diameter (mm)         0,00           Angle (deg)         0,00           Unwer reinforcement         0           Diameter (mm)         10,0           Angle (deg)         0,00           Punching         0,00				
			0.00		
				0.00	
				Rectangle	
				Rectangle	
				Upper reinforcement     10.0       Diameter (mm)     10.0       Angle (deg)     0.00       Lower reinforcement     0.00       Diameter (mm)     0.00       Shear reinforcement     0.00       Diameter (mm)     10.0       Geometrical shape     Rectangle       Support data     10.0       Support shape     Rectangle       Column position     10.0	
			Default distance x*h; x = [-]	6,00	

#### Setup 3: Advanced settings

#### Setup menu > Concrete solver

ype of values		EC-EN	Name	EC-EN
ode independent values		Concrete	Concrete	1
ode dependent values		😑 General	□ General	-
		Calculation	Calculation	
		2D structures	General	
		B-ULS	Number of iteration steps	100
		- Shear	Precision of iteration [%]	1
		2D structures	Limit value for checks [-]	1.00
		Construction joint	Limit bending pressure zone ratio xu/d	
		Punching	Automatic calculation (steel yield limit)	🗆 no
		⊜ SLS	for C12/15 till C50/60 5.6.3(2) [-]	0.45
		Creep	for concrete classes higher than C50/60 5.6.3(2) [-]	0.35
		- Crack proof Code Dependent Deflections	2D structures	
		Code Dependent Detections     Detailing provisions	Req. shear reinforcement -> c-s height >= 20cm	🖾 yes
	- 2D structures and slabs	Design of pressure reinforcement in plates	🖾 yes	
		Punching	2D user reinforcement	
		Warnings and errors	Check of concrete cover for subtracting 2D user rei.	_ 🗆 no
			Special design control	
			Virtual strut reduction factor [%] [-]	80.00
			B Shear	
			2D structures	
			Shear strut inclination control 6.2.3	variable strut inclination method
			Shear effect control 6.2.3(7)	shear effect considered in SR 2
			Construction joint	
			Take into account cos_alpha to formula 6.25	⊠ yes
			Punching	
			Loaded area	0.00
			Control perimeters for slab of ceiling at a distance x	
			Control perimeters for foundational slab at a distanc	
			Distance between the perimeter of the loaded area	
			Dimensions of column for use the complete perimet.	. 5,00
				200.0
			Min thickness of plate (5.4.3.3 (1)) [mm]	200,0
			Calculation of shear resistance	
			Include normal force to punching calculation	🗆 no
			Column heads	

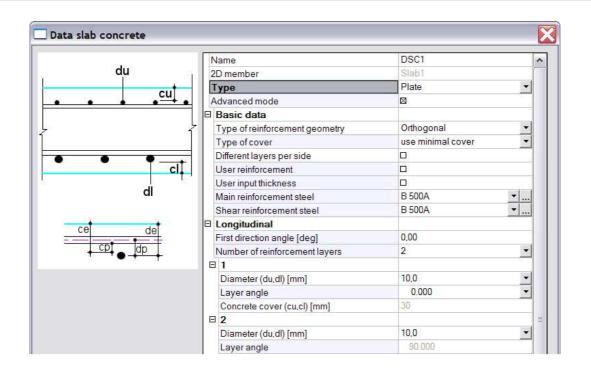
All of the adjustments made in one of the three general setups are valid for the **whole project**, except for the members to which 'Member data' are added.

#### 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 <sup>III</sup> or via a right mouse click.



# 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.

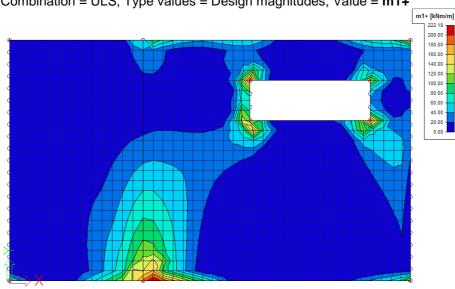
For more detailed information, reference is made to the Benchmark examples added at the end of this manual.

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 zaxis of the 2D member.

"1" and "2" stand for the reinforcement directions, which are by default respectively the local x- and ydirection 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,  $\Delta F_{td}$ , in the longitudinal reinforcement due to shear  $V_{Ed}$  is calculated from  $\Delta F_{td} = 0.5 V_{Ed} (\cot \theta - \cot \alpha)$  (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)

 $\rightarrow$  For members <u>without</u> shear reinforcement,  $\Delta F_{td}$  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:

				50.51	
Type of values	 ⊕ EC-EN		Name	EC-EN	
Code independent values	Concrete	E	Concrete		
Code dependent values	🖨 General		General		
	Calculation		ULS		
	General		□ Shear		
	□ 2D structures □ ULS		2D structures		
	- OLS		Shear strut inclination control 6.2.3	variable strut inclination method	
			Shear effect control 6.2.3(7)	no shear effect considered	
	Construction joint		Construction joint	no shear effect considered	
	Punching		Punching	shear effect considered in SR 2	
			I SLS	shear effect considered unconditionally	
	Creep		Detailing provisions		
	Crack proof		Warnings and errors		
	Code Dependent Deflections				
	<ul> <li>Detailing provisions</li> </ul>				
	<ul> <li>2D structures and slabs</li> </ul>				
	Punching				

#### a) Shear effect considered in SR2

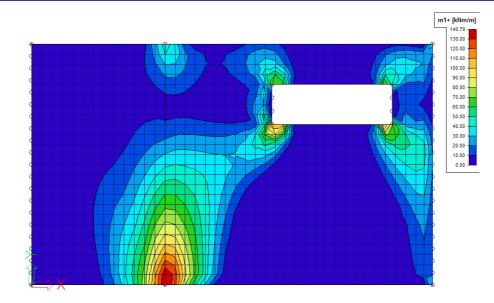
The value of  $\Delta F_{td}$  is calculated directly (EN §6.2.3(7)), which is in fact the most accurate approach. 'SR2' stands for shear region 2, which is defined by the DIN 1045 terminology as the region where shear reinforcement is required to resist  $v_{Ed}$ .

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  $\Delta F_{td}$  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  $\Delta F_{td}$  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.

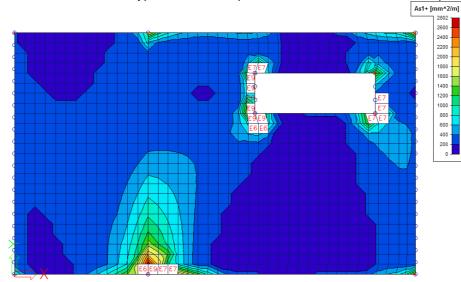
#### Theoretically required reinforcement

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-

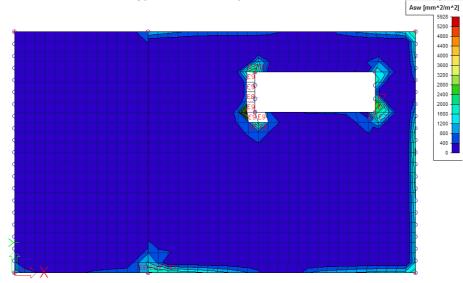
Combination = ULS; Type values = Required areas; Reinforcement = Required reinf.; Value = As1+



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 ydirection of the 2D member.

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

	Name	DSC1	
С <u>т</u>	2D member	Slab1	
	Туре	Plate	-
cu ,	Advanced mode		
	Basic data		
	Type of reinforcement geometry	User	*
	Type of cover	use minimal cover	*
	Different layers per side		
cl	User reinforcement		
1.5	User input thickness		
	Upper reinforcement steel	B 500A	
	Lower reinforcement steel	B 500A	
	Shear reinforcement steel	B 500A	•
. 8	Upper		V     I       V
T P	Number of directions	3	
া	First direction angle [deg]	45.00	
	Second direction angle [deg]	135.00	
	Third direction angle [deg]	90,00	
	Number of reinforcement layers	10	•
E	1		
	Diameter (du) [mm]	10,0	
	Layer angle	45.000	•
	Concrete cover (cu) [mm]	30	_
	Basic distance [mm]	200	•
E	2		-
	Diameter (du) [mm]	10,0	•
	Layer angle	135.000	-
	Type of cover	layer on previous layer	
	Concrete cover [mm]	40	
	Basic distance [mm]	200	•
E	3		
	Diameter (du) [mm]	10.0	
	Layer angle	90.000	
	Type of cover	layer on previous layer	
	Concrete cover [mm]	50	
	Basic distance [mm]	200	•
E	3 4		
	<b>N</b> (1)(1)	10.0	<ul> <li></li> <li></li></ul>
100.000	stions		
	oad default values oncrete Setup		

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:

🗌 Calcula	ation Info			X
🗈 🛄 📑	<b>6</b>     H	100	% 🔹 🖬 🚺 default 🔹 🕮 🚍 default 🔹 💷 📾	
warnings	and error	S	for slabs	^
Slab	No.	Туре	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.	Ш
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.	
🏥 Ready [I	nl]		<ul> <li>(٤)</li> </ul>	~
Show List for			for slabs	-

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.

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.

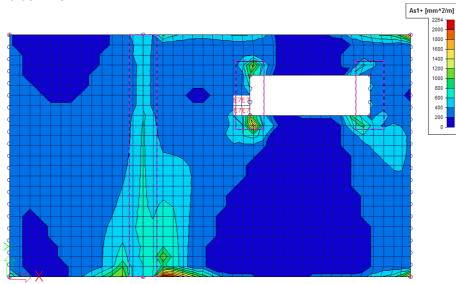
When asking results now, select the option 'Averaging of peak' in the Properties menu!

System	Local	
Show errors and warnings		
Print explanation of errors		
Use user scale isolines		
Averaging of peak	⊠	
Location	In nodes, avg. 🔹	
Type values	Required areas	
	1	×

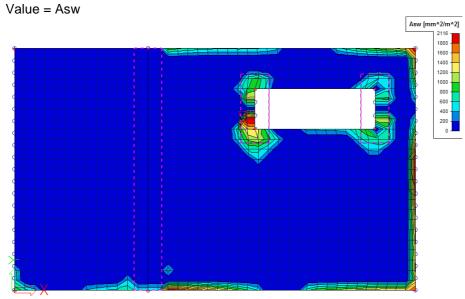
When asking the values for the theoretical reinforcement again, the errors E8 don't appear anymore:

#### Longitudinal reinforcement

Value = As1+



Shear force reinforcement



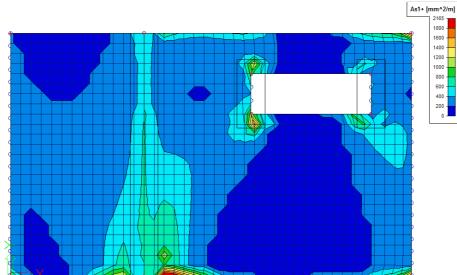
#### Measure 2: Increasing the concrete grade

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

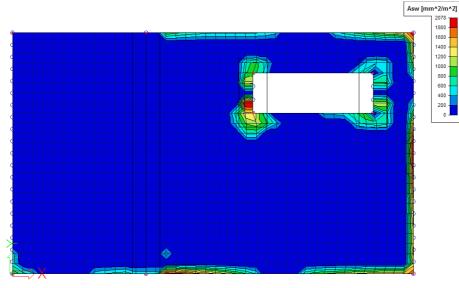
After a new linear calculation, and when asking the values for the theoretical reinforcement again, the error E6 has also disappeared:

#### Longitudinal reinforcement

Value = As1+



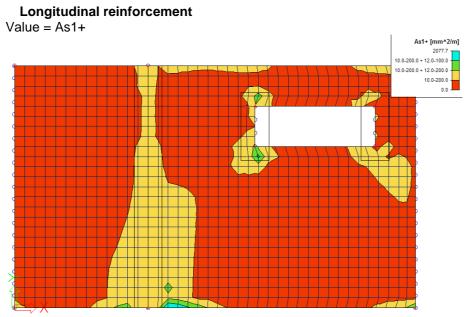
**Shear force reinforcement** Value = Asw



#### 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:

1.11001				
System	Local			
Show errors and warnings				
Print explanation of errors.	🗆			
Use user scale isolines	⊠			
User scale isolines	Isolines	1	▼	
Averaging of peak	⊠			
Location	In nodes	s, avg.		
T	Dequire	d araba	×	
Scale of isolines				
Name Isolines 1				
,				
New level			Mean	_
Diameter 0,0	0,0	0,0	0,0	mm
Distance 0	0	0	0	mm
Amount 0	0	0	0	 mm^2
Legend User default reinforcement 10.0-200.0 + 12.0-100.0 10.0-200.0 + 12.0-200.0 10.0-200.0	Copy to I	As [mm^2 .2 1523.67 .2 958.19	Clear le	
,	Delete a	ct.level		all Cancel



#### 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.

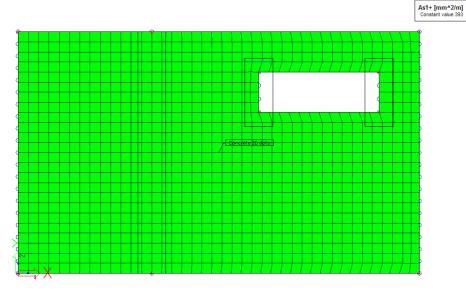
Select the plate, choose the option 'User reinforcement', and input a Diameter and Basic distance for directions 1 and 2.

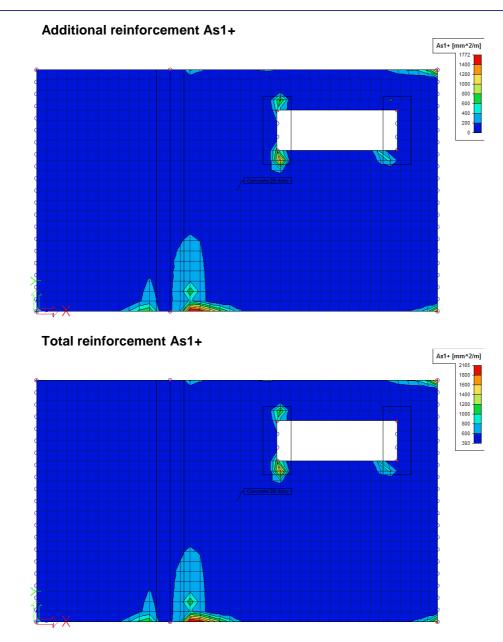
If you select also the option 'Different layers per side', it is possible to define the upper and lower reinforcement independently from each other.

1.000 <b>-</b> - 1.000	Name	DSC1	
du	2D member	Slab1	
	Туре	Plate	•
	Advanced mode		
	Basic data		
	Type of reinforcement geometry	Orthogonal	•
	Type of cover	use minimal cover	•
	Different layers per side		
	CI User reinforcement		
	User input thickness		
a	Upper reinforcement steel	B 500A 🔹	
	Lower reinforcement steel	B 500A 🔹	
dat	Shear reinforcement steel	B 500A 🔹	
	Upper		
	First direction angle [deg]	0,00	
<i>k</i> a	Number of reinforcement layers	2	•
	B 1		
	Diameter (du) [mm]	10.0	*
	Layer angle	0.000	*
	Concrete cover (cu) [mm]	30	
	Basic distance [mm]	200	•
	□ <b>2</b>		
	Diameter (du) [mm]	10,0	•
	Layer angle	90.000	

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+





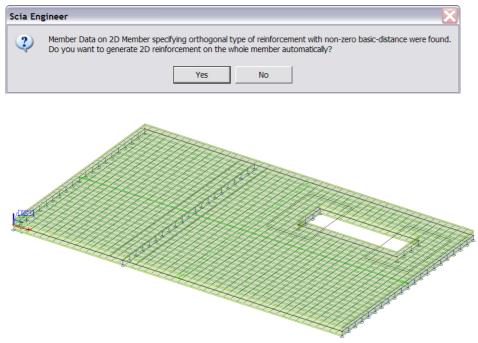
Take a look at the legend to see the difference in result values between additional and total reinforcement.

#### 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 specific 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 [...].

Concrete setup			X
Concrete setup	ECEN     General     Concrete     Concrete     Concrete     Concrete     Pestessed reinforcement     Ourbality and concrete cover     ULS     General     Concrete     Detailing provisions     Costructures and slabs     Punching	Name         Concrete         B General         ULS         Sts         General         National annex         National annex         K_crack - coefficient for calculation maxi         Value [-]         Watue [-]         Watue [-]         Watue [-]         Detailing provisions	EC-EN 3.40 0.43 0.4/0.3/0.3
Select all Unselect all	Refresh	Load default NA parameters	OK Cancel

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.

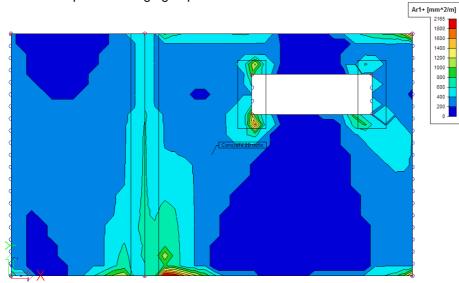
#### 2\_Theoretical reinforcement design

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

#### Longitudinal reinforcement

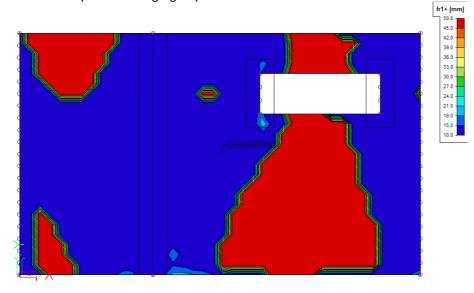
Analogously to the Member design ULS, the available values here are: Ar1+, Ar1-, Ar2+, Ar2-The subscription 's' is replaced here by 'r' to make clear that the requirements for cracking are accounted for in these values.

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

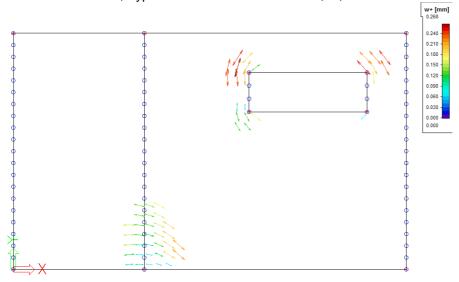
Concrete menu > 2D member > Member check - Crack control

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).

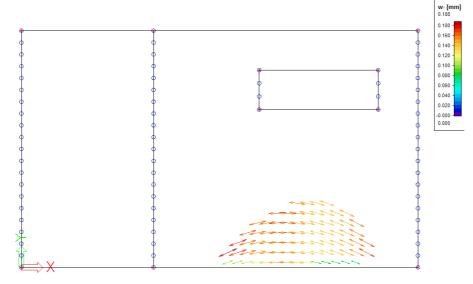
<u>Attention</u>: This is nevertheless only the case if the results for Member design ULS+SLS are stored *without* averaging of peaks!

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

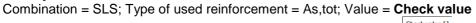


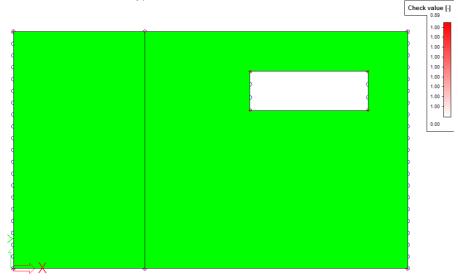
#### Crack width w-

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



#### Unity check





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

#### 3\_Results for basic reinforcement

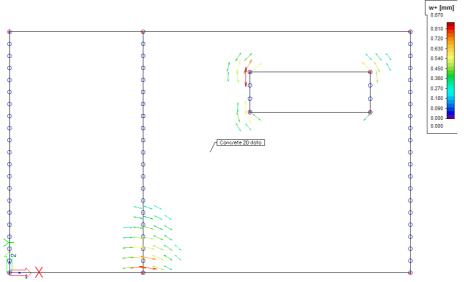
Concrete menu > 2D member > Member check - Crack control

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+

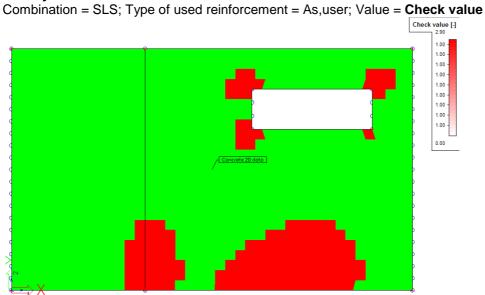
Combination = SLS; Type of used reinforcement = As, user; Value = **w+** 



#### Crack width w-

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

Unity check



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

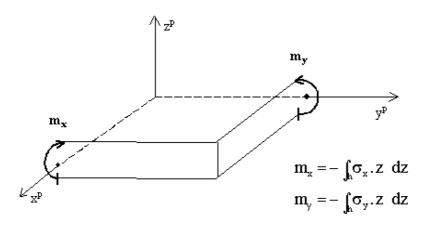
# Annexes

# Annex 1: Conventions for the results on 2D members

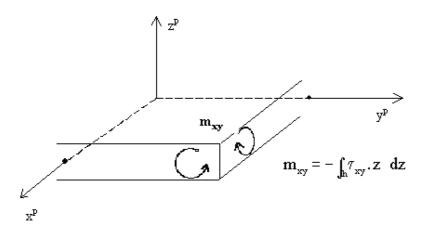
## 1\_ Basic magnitudes = Characteristic values

Bending (plates, shells)

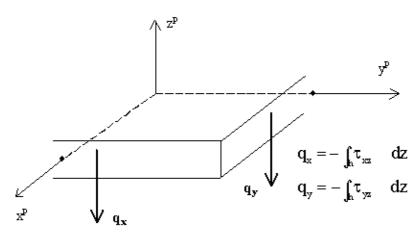
\*Bending moments mx, my



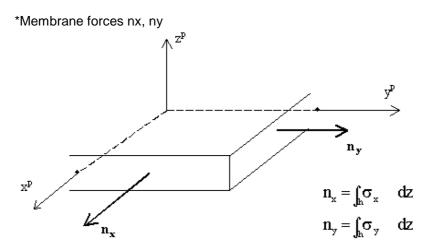
\*Torsion moment mxy



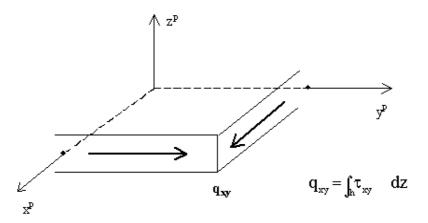
\*Shear forces qx, qy (=vx, vy)



Membrane effects (walls, shells)



\*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

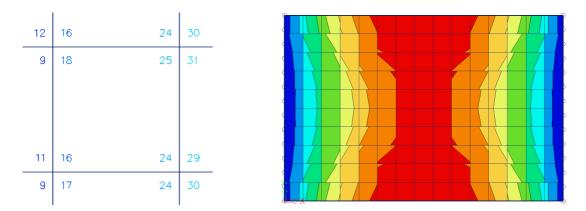
During a calculation in SCIA Engineer, the node deformations and the reactions are calculated exactly (by means of the displacement method). The stresses and internal forces are derived from these magnitudes by means of the assumed basic functions, and are therefore in the Finite Elements Method always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

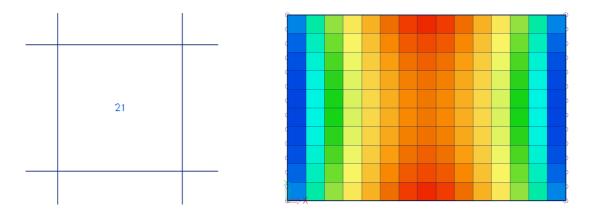
All of the values of the results are taken into account, there is no averaging. In each node are therefore the 4 values of the adjacent mesh elements shown. If these 4 results differ a lot from each other, it is an indication that the chosen mesh size is too large.

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.



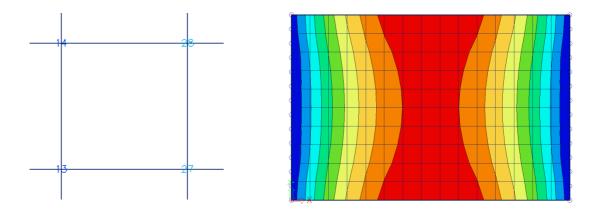
### 3\_ In nodes, average

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:

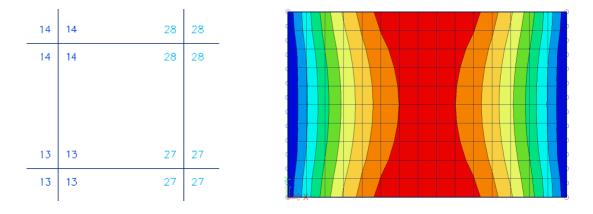
- At the transition between 2D members (plates, walls, shells) with different local axes.

- 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'.



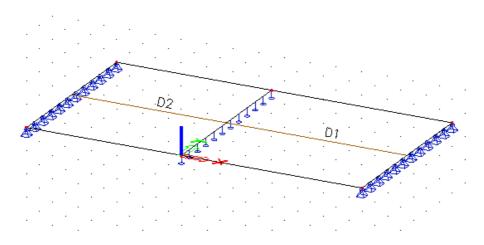
## Accuracy of the results

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:

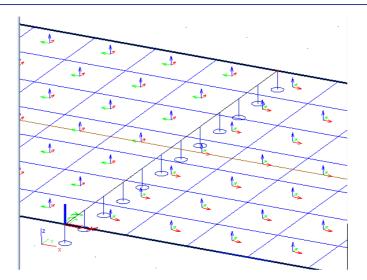


In order to display the mesh local axes, you need first to generate the mesh. You can use the button 'Mesh generation' or Main menu > Calculation, Mesh > Mesh generation

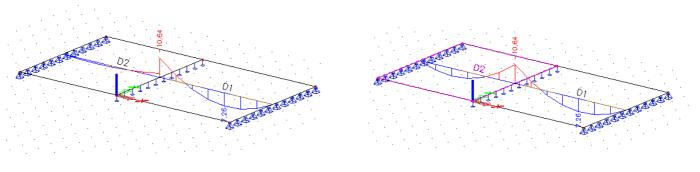
Afterwards, the mesh and local axes can be displayed from 'Set View parameters for all': right click of the mouse on the screen

C	heck / Uncheck group			Lock posit	tion		
1	Modelling/Drawing	🚱 Attributes		Misc.	View		
	🕿 Structure 🛛 🔤 La	bels 🔼	Model	J L	oads/masses		
	Check / Uncheck all						
-	Service						
-	Structure						
_	Panel						
_	Structure nodes		-				
-	Display		~				
	Mark style		Dot				
F	Mesh		000				
	Draw mesh		V				
-	Free edges		~	1			
	Display mode		wired				
Ξ	Local axes						
	Nodes		Г				
	Members 2D						
	Mesh elements		~	2			
Ξ	Sections						
	Members 1D		7				
	Members 2D		~				

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.



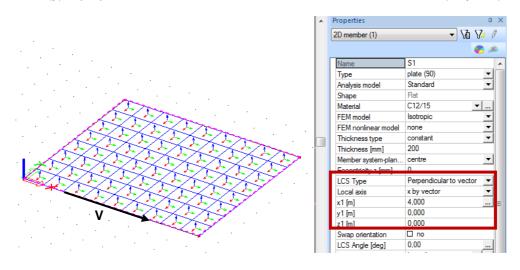
#### Moment diagram 1

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	ą ×							
2D member (1)	- 🕼 🎶 🖉							
	<b>8</b>							
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 Tilt of vector defined by point							
□ Nodes								
N1	Tilt of vector normal to line							
N4	abso							

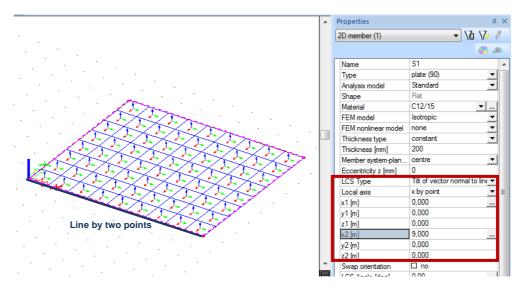
### 1\_ Perpendicular to vector



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

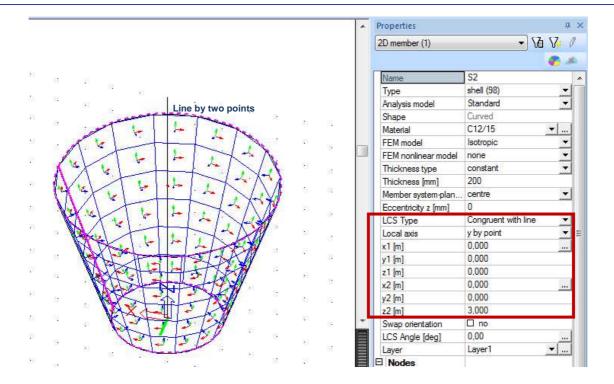
This method is similar to the one above. Instead of introducing a vector, a line is defined between two points. (x1 y1 z1) are the coordinates of the first point and (x2 y2 z2) are the coordinates of the second point. The x(y) axis is perpendicular to the introduce line and points towards it.



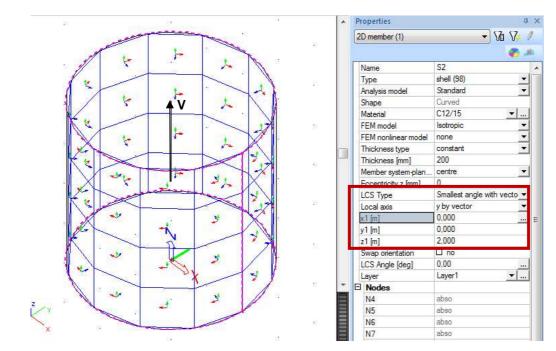
The coordinates of the points that define the line can be introduced in the properties window. You can also click on the  $\boxed{}$  button next to x1 and x2 and define these points graphically.

### 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 :



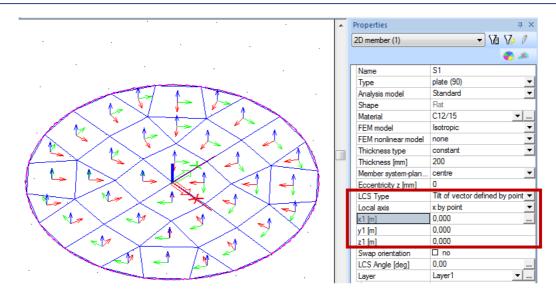
## 4\_ Smallest angle with vector



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

## 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 crosssection. 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 ( $n_l > 0$ ,  $n_{ll} < 0$ ) 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 2<sup>nd</sup> principal force  $n_{ll} < 0$ , 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 crosssection 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:

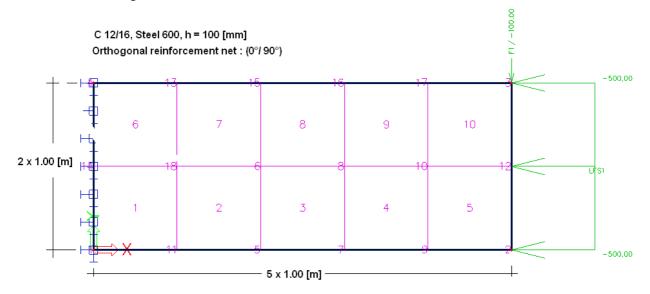


Fig.1 SEN model of Wall – Geometry and FE mesh

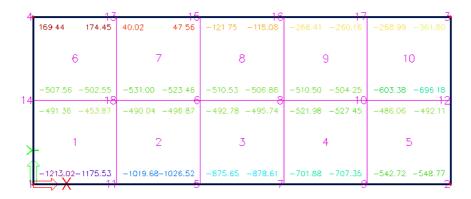


Fig.2 Results of FEM analysis: Inner forces  $n_x$  – Direct values in element nodes (LC1)

4	13			15		16		17			
	46.83	71.90	34,86	72 57	33 44	51,78	6 58	37 83	182.47	-281.56	
	6			7		8		9		10	
	-88 57		-79.35	-41 64	- 44-31	-25.98	-42.24	-10 99	115 59	- 348.44	
14	-104 77	<del>18</del> 82.67	74.54	40 35	46 52	31.72	38.07	10 72	-3.22	-33.48	
>	1		:	2		3		4		5	
	-249.11	-61.66	-31.39	-65.58	- 30.06	-44.86	2.09	-25.26	-14.55	-44.81	

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

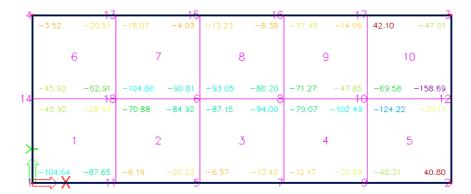


Fig.4 Results of FEM analysis: Inner forces  $n_{xy}$  – Direct values in element nodes (LC1)

## 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.

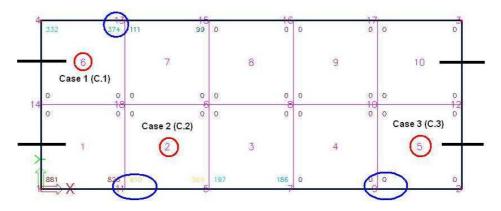


Fig.5 Design results: stat. required horizontal reinforcement  $a_{s1}$  [mm<sup>2</sup>/m] – direct values in nodes

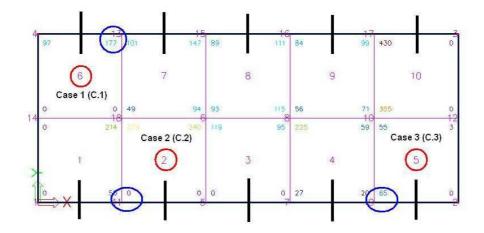


Fig.6 Design results: stat. required vertical reinforcement 
$$a_{s2}$$
 [mm<sup>2</sup>/m] – direct values in nodes

#### 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 <sub>I</sub>	$n_{\mathrm{II}}$	$lpha_{ m I,II}$	$n_{1d}$	<i>n</i> <sub>2d</sub>	n <sub>cd</sub>
174 .5	71. 9	- 20. 5	178 .4	68. 0	- 10. 9°	195 .0	92. 4	- 41. 0

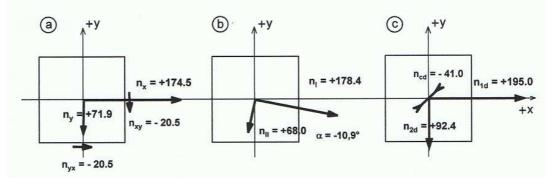


Fig.7 Graphic representation of Table 1 (Case 1 of inner forces transformation)

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

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

This is a standard transformation case. The design forces  $\{n_{1d}, n_{2d}, n_{cd}\}$  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:

$$\mathbf{a}_{si,req} = n_{id} / f_{yd}$$
(1)  
$$\mathbf{a}_{s1,req} = n_{1d} / f_{yd} = 0.1950 / (600 / 1.15) = 3.74_{10} - 4 \text{ [m^2/m]} = 374 \text{ [mm^2/m]}$$
(agreement with Fig.5)  
$$\mathbf{a}_{s2,req} = n_{2d} / f_{yd} = 0.0924 / (600 / 1.15) = 1.77_{10} - 4 \text{ [m^2/m]} = 177 \text{ [mm^2/m]}$$
(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 1<sup>st</sup> default value of the reduction factor  $r_{fac} = 0.80$  (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:

(2)

(3)

$$-\boldsymbol{n}_{cd} \leq \boldsymbol{r}_{fac} \times f_{cd} \times \boldsymbol{H}$$

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

 $41.0_{10}$ -3 = **0.041** << 0.80 × 12/1.5 × 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 <sub>y</sub>	$n_{xy}$	n <sub>I</sub>	$n_{\mathrm{II}}$	$lpha_{ m I,II}$	$n_{1d}$	n <sub>2d</sub>	n <sub>cd</sub>
- 101 9.7	- 31. 4	-6.2	- 31. 3	- 101 9.7	90. 4°	- 101 3.5	- 25. 2	- 12. 4

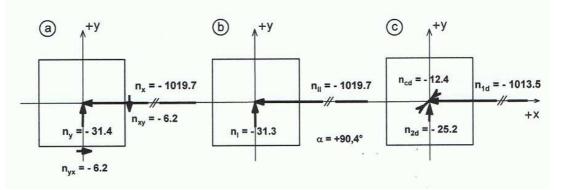


Fig.8 Graphic representation of Table 2 (Case 2 of inner forces transformation)

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

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

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

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

In (3)  $f_{ycd}$  is steel strength in compression (1<sup>st</sup> input default:  $f_{ycd} = f_{yd}$ );  $E_s$  – Young's modulus;  $\varepsilon_{c1}$  – concrete yield point (bilinear stress-strain concrete diagram specified,  $\varepsilon_{c1}$  = 0.175 %).

<u>Hint</u>: 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.  $f_{ycd} \le E_s \varepsilon_{c1}$  in most cases. From (3) it follows:

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

$$\mathbf{a}_{s1,req} = (1.0135 - 0.10 \times 12.0/1.5)/350.0 = 6.10_{10} - 4 \text{ [m^2/m]} = \mathbf{610} \text{ [mm^2/m]} \qquad (agreement with Fig.5)$$
  

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

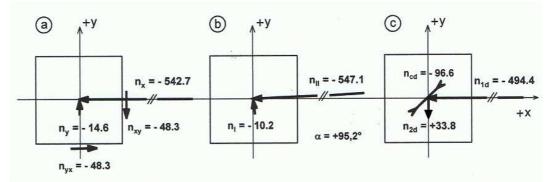
Obviously, the virtual concrete strut possesses sufficient resistance.

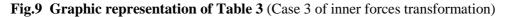
### C.3 Design Case 3: Statically Required Reinforcement – Hyperbolic Stress

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.

$n_x$	n <sub>y</sub>	$n_{xy}$	n <sub>I</sub>	$n_{\mathrm{II}}$	$lpha_{ m I,II}$	$n_{1d}$	n <sub>2d</sub>	n <sub>cd</sub>
- 542 .7	- 14. 6	- 48. 3	- 10. 2	- 547 .1	95. 2°	- 494 .4	+33 .8	- 96. 6

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





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

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

This is a very special NEDIM transformation case. *Baumann* [1] and all other transformation theories would yield the  $2^{nd}$  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:

$\mathbf{a}_{s1,req} = (0.4944 - 0.10 \times 12.0/1.5)/350.0 < 0 \rightarrow \mathbf{a}_{s1,req} = 0$	(agreement with Fig.5)
$\mathbf{a}_{s2,reg} = 0.0338/(600/1.15) = 6.48_{10} - 4 \text{ [m^2/m]} = 65 \text{ [mm^2/m]}$	(agreement with Fig.6)

The *tension* force  $n_{2d} = +33.8_v$  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.

<u>Hint</u>: 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 1<sup>st</sup> 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.

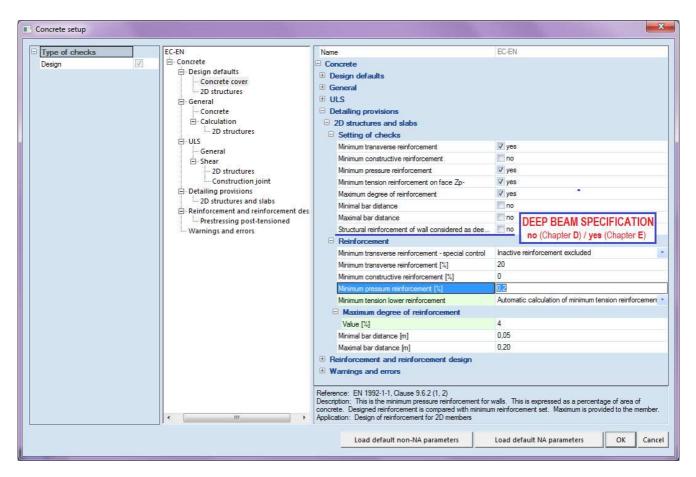


Fig.10 Concrete Setup > Detailing provisions > 2D structures - Minimum reinforcement

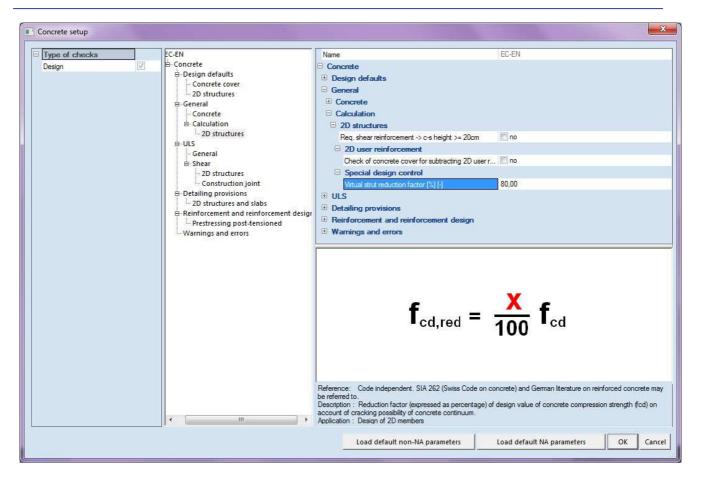


Fig.11 Concrete Setup > Calculation > 2D structures > Special design control – Virtual strut reduction

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

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

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 2<sup>nd</sup> term in (4) is decisive, thus  $a_{st,min} = 0.0013 d = 0.00013 x 0.065 = 0.0000845$  [m<sup>2</sup>/m]. However, in Walls the total reinforcement is presented; thus, the resulting reinforcement is double as much:

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

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

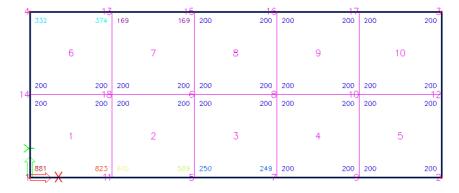


Fig.12 Horizontal reinforcement  $a_{s1,req+min}$  [mm<sup>2</sup>/m] – Direct values in nodes

(4)

169		177	169		15 169	169		169	169		169	430		200
	6			7			8			9			10	
200		200	169		169	169		169	169		169	355		200
200		214	279		240	169		169	225		169	169		169
	1			2			3			4			5	
200	,	169	200		200	200		200	169		169	169		200

Fig.13 Vertical reinforcement  $a_{s2,req+min}$  [mm<sup>2</sup>/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  $a_{sc,min} = 0,002 H \text{ [m}^2/\text{m]}$ . Hence, in Fig.12 and Fig.13 the value of

 $a_{sc,min} = 200 \ [\text{mm}^2/\text{m}],$ 

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:

 $a_{s,DBmin} = max(0.001H, 0.00015)$  [m<sup>2</sup>/m]

(5)

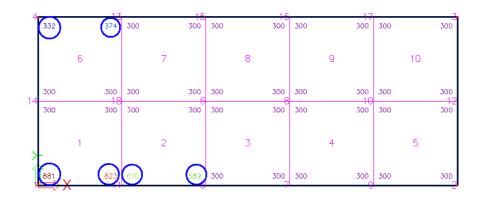
Since in Walls the total reinforcement is showed, the resulting constructive reinforcement is double as much.

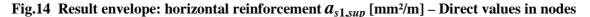
Numerically:

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

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 "\*".





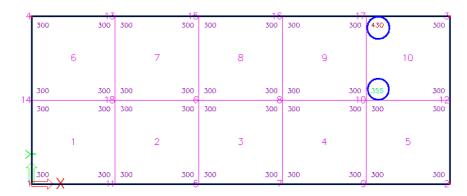


Fig.15 Result envelope: vertical reinforcement $a_{s2,sup}$ [mm <sup>2</sup> /m] – Direct va	values in nodes
--	-----------------

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

Elem/Node	Hori	zontal reinforce [mm <sup>2</sup> /m]	ement (1)	Vertical reinforcement (2) [mm <sup>2</sup> /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 (6<sup>th</sup> 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 crosssection 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 ( $m_I \ge m_{II} > 0$ ) 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/m<sup>2</sup> 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).

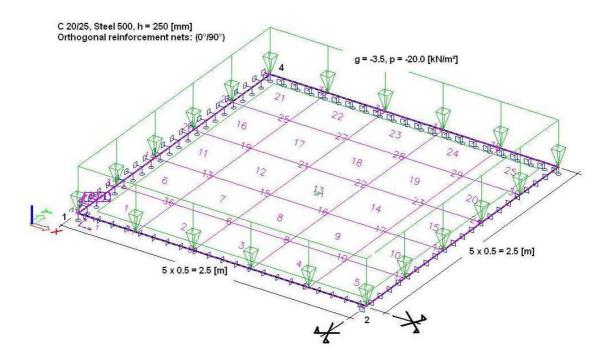


Fig.1 SEN model of Plate – Geometry, FE mesh, Loads

4	-	71		2.7		2.4		75		
	-0.3	2.8	-0.1	2.3	-0.0	2.1	0.6	2.0	1.2	1.7
	21		23	2	2	3	2	4	2	5
26	1.2	6.7 	8.0	11.7	12.3	14.7 	15.0	16.4 	16,5	16.9 30
20	-0.3		4.2	13.7	9,7	1002203	13.8	17.7	16.4	
	16		15	7	1	8	1	9	2	0
00	-0.0		10.3	19, <mark>8</mark> 21	18.5				27.5	28.8
20	-0.3	19 12.7	7.3	20.1	15.7		22.0	27.7	(全株川)	
	11		12		13		14		15	
14	-0.1	14,1	11.3	24.0	21.5	and the second	29.0	34.6	33.6	35.5
14	-0.2	14,4	9.3	15 23.8	19,3	16 30.2	27.0	33.7	32.1	34.3
	6		7		8		9		1	0
32	-0.2	15.2 	11.5	26.0		33.5	31.0	1920	36.4	38.7
22	-0.2	15.3	10.6	25 7	21 5	33.0	29.9	<del>10</del> 37.1	35.4	37.9
>	. 1		2			5	4	6	1	5
4	-0.2	15.5	11.3	26.4	22.5	34.1	31.2	38.4	36.9	39.3
11		11								

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

4	331	31 30.0	30.3	23 I	24.6	15.3	17.5		9.6	-0.1
	Z	1	2	2		23		24		25
26	30.0	26.8	28.3	21.1	23 3	14.0 28	16.8	6.3 29	9,4	-0.2 
20	30.3	28.3	27.7	22.5	22.4	15.2	15.7	(-27 × (1))	8.3	-0.0
	1	6	1	7		18		19		20
20	23.1	211	225	17.2	18.9	11.7 22	13.7	5.4 23	7.6	-0.1
20	24.6	23/3	22.4	18,9	18.2	13.1	12.7	6.7	6.5	0.1
	1	1	1	2		13		14		15
14	15.3	14.0	15.2	11.7	13,1	8.1	9.7	.3.7	5.5	-0.1
	17.5	16.8	15.7	13.7	12.7	9.7	8.7	5.2	4.3	0.4
		12	7	2		8		9		10
32	7.0	6.3 	7.5	5,4	6.7	3.7	5.2	1.6	3.1	-0.1
.04	9.6	9.4	8.3	7.6	6.5	5.5	4.3	3,1	1.9	0.6
>		t	2	2		3		4		5
5	-0.1	-0.2	-0.0	-0.1	0.1	-0.1	0.4	-0.1	0,6	-0.1

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

1	the state of the state of the	71		2.7	the second second	7.4	and the local data in the	75	Color States		7
	-0.1	-0.1	0.5	0.5	0.7	0.7	0.5	0.5	0.2		0.2
	2	1	2	2	2	3	2	4		25	
26	25.4	25,4 25	13,5	13.5	7,7	7.7	4.0	4.0	1,3		1,3
20	20.9	20.9	13.2	13.2	8.5	8.5	4.8	4.8	1.6		1.6
	1	6	1	7	1	8	1	9		20	
20	39.0	<u>39.0</u> 19	24.9	24.9 	15.2	15.2 22	8.2	8.2 	2.6		2.6
20	34.8	34.8	23.1	23.1	14.9	14.9	8,4	8.4	2.7		24
	1	18	1	2	1	3	1	4		15	
14	46.6	46.6 13	32.0	32.0	20.5	20,5	11.3	11.3	3,6		3.6
1.70	43.6	43,6	30,1	30.1	19.7	19.7	11.2	11.2	3.6		3.6
	(	5	5	7	(	8	Ś	9		10	
32	50.3	50.3 36	35 6	35.6	23.3	23.3	13,1	13.1	4.2		4.2
34	48.8	48.8	34 5	34.5	22.8	22.8	12,9	12.9	4.2		4.2
>		K.	2	2		E.	2	4		кЛ	
1	51.0	51.0	36.3	36.5	24.0	24,0	13.6	13.6	4,4		4,4
11		Contract of the Property of th	Statement and the state	0	Contraction of the second s	and the state of the state of the	distant state of som	0		in Research the	2

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 1<sup>st</sup> 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.

esign defaults	Detailing provisions
Vernings and errors	2D members and beam-slabs         Reinforcement         Image: Minimum transverse reinforcement         Image: Minimum constructive reinforcement         Image: Minimum constructive reinforcement         Image: Minimum percentage in pressure         Image: Minimum tension reinforcement on face Zp+         Image: Minimum tension reinforcement         Image: Minimum tension reinforcement         Image: Minimum tension reinforcement
	Min tension reinforcentage     S     Minimum tension reinforcement on face Zp-     9.2.1.1(1)
	Automatic calculation of minimum tension reinforcement
	🌔 Min tension reinf percentage 👘 🖉 🎖
	✓ Maximum degree of reinforcement 4 9.2.1.1(1.2)
	Minimum shear reinforcement 0 %

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.

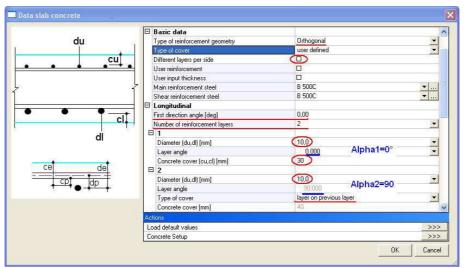


Fig.6 Concrete data input window - reinforcement and cross-section geometry

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].

4			- 71	1100 C	1000	7.7	51.0 27 - 13		- 7.4		20110	7.5	SULLES ST		7
	361		390	343		299	285		202	209		103	123		29
		21		8	22		a K	23			24			25	
	369		395	429					and the second s	368		V. Starter			211
26	326		25 447	376		27 424			28 365	341		-29 -288	283		36 207
		16			17	2		18			19			20	
-	265			383			435			441			408		
20	257		19 423	348		21 455			22 444			23 398	376		24 324
		11			12			13			14			15	
	162		323			and the second	401		CALLS COLOR			447	456		423
14	174		1 <u>3</u> 360	289		15 435	370		16 465			17 453	422		403
		6			7			8			9			10	
	61		246	217		362	338		432	419			459		
32	84			216		385	322		8 447			10 469	433		12 448
>		1			2			3			4	016		5	
1	10	1	195	134		317	260		405	366		455	435		465
H	->1	~					-					-			

Fig.7 Design results: stat. required reinforcement  $a_{s2}$ . [mm<sup>2</sup>/m] – Direct values in element nodes (ULS)

.

4	-		21		-	7.7			2.4	and the second		75		-10 or 10000	
	428	>	321	376		243	292		152	196		58	96		0
		21		6	22			23	1/5		24			25	
20	341		234	237		108	127		0	20		0	0		0
26	394		220	275		100	146		0	22		0	0		0
		16			17			18			19			20	
20	277		108	139		0	0		0	0		0	0		0
20	321		122	174		0	28		0	0		0	0		0
		11			12			13			14			15	
14	191		0	33		0	0		0	0		0	0		0
14	231		26	72		0	0		0	0		0	0		0
		6			7			8			9			10	
32	99		0	0		0	0		0	0		0 10	0		0
JZ	135		0	0		0	0		0	0		0	0		0
>	_	1	24		2	0.400	1	3	11	R.	4			5	
4	35	V	0	0		0	0		0	0		0	0		٥
12	~ ,	~	1.1						1			0			2

-

Fig.8 Design results: stat. required reinforcement  $a_{s2+}$  [mm<sup>2</sup>/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.

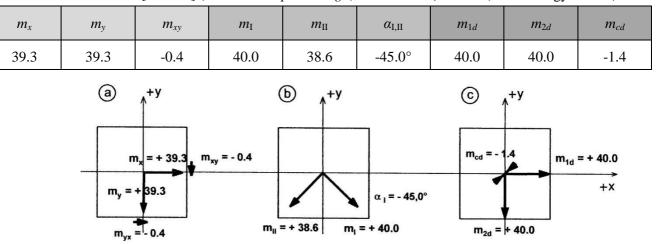


Table 1. Inner moments [kNm/m] (Basic/ Principal/ Design) in element 5, node 2 (Test Strategy line 1#)

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

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

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

The design moments  $\{m_{1d}, m_{2d}, m_{cd}\}$  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):

$$z_i = \zeta_{mean} \times h_i \tag{2}$$

$$h_1 = h - c - \phi/2 = 0.250 - 0.030 - 0.010/2 = 0.215$$
 [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:

$$a_{si,req} = m_{id} / (f_{yd} \times z_{id})$$
(3)  
$$a_{s1-,req} = m_{1d} / (f_{yd} \times z_{1d}) = 0.040 / (500 / 1.15 \times 0.20751) = 4.43_{10} - 4 \text{ [m^2/m]} = 443 \text{ [mm^2/m]} \text{ (not displayed here)}$$
  
$$a_{s2-,req} = m_{2d} / (f_{yd} \times z_{2d}) = 0.040 / (500 / 1.15 \times 0.19785) = 4.65_{10} - 4 \text{ [m^2/m]} = 465 \text{ [mm^2/m]} \text{ (agreement with Fig.7)}$$

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:

$$\xi_{strut} < \xi_{lim}$$
 [-]

(4)

where  $\xi_{lim}$  is the limit value of the relative bending zone height;  $\xi_{strut}$  is the actual relative height of the bending zone under the impact of  $m_{cd}$ . With the 1<sup>st</sup> default values of the reduction factor  $r_{fac} = 0.80$  (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:

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

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

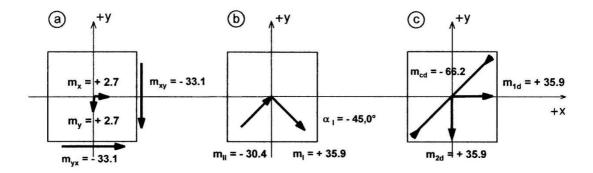
Fig.10 Concrete Setup - limit value of the relative bending zone height and strength reduction factor

#### 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 <sub>y</sub>	m <sub>xy</sub>	m <sub>I</sub>	$m_{ m II}$	$lpha_{ m I,II}$	$m_{1d}$	<i>m</i> <sub>2d</sub>	m <sub>cd</sub>
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:

$$m_{1d} + m_{2d} + m_{cd} = 35.9 + 35.9 - 66.2 = 5.5 \equiv m_{I} + m_{II} = 35.9 - 30.4 = 5.5 \text{ [kN/m]}$$
 (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:

$$a_{s1+,req} = m_{1d}/(f_{yd} \times z_{1d}) = 0.0359/(500/1.15 \times 0.20196) = 4.08_{10}-4 \text{ [m^2/m]} = 408 \text{ [mm^2/m]}(\text{not displayed here})$$

$$a_{s2+,req} = m_{2d}/(f_{yd} \times z_{2d}) = 0.0359/(500/1.15 \times 0.19257) = 4.28_{10}-4 \text{ [m^2/m]} = 428 \text{ [mm^2/m]}(agreement with Fig.8)$$

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

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

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

<u>Hint</u>: 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:

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

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

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

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 ( $-135^{\circ}/45^{\circ}$ ) 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 1<sup>st</sup> default input, as shown in Fig.12, is dealt with. As a fact, the user may modify the settings, but accepting the 1<sup>st</sup> default, legal solution according to the Norm is obtained.

Design defaults	Detailing provisions
General Calculation ULS Shear SLS Creep Crack proof CDD Detailing provisions Warnings and errors	2D members and beam-slabs Reinforcement ✓ Minimum transverse reinforcement Minimum constructive reinforcement Maximum percentage in pressure bending zone ✓ Minimum tension reinforcement on face Zp+ 9.2.1.1(1 Automatic calculation of minimum tension reinforcement Minimum tension reinforcement on face Zp+ 9.2.1.1(1 Automatic calculation of minimum tension reinforcement Automatic calculation of minimum tension reinforcement
	C Min.tension reinf.percentage     □
	☐ Minimum shear reinforcement

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<sup>st</sup> 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:

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

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

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

We find the value of 280 mm<sup>2</sup>/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.

		21			7,7		_	7.4			7,5			7
361		390	343		299	285	(	280	280		280	28()		280
	21			22			23			24			25	
369			429			417		- mession						250
3215		SPHORE S	376			377		Contraction of the local data				283		<u>280</u>
	16			17			18			19			20	
280		and the second second	383			435		510-50C	441			408		346
280			348			393			402			37.6		324
	11			12			13			14			15	
280		323				401		1010100	449			456		423
280		360	289		435 435	370		1. 1. 1. 1. 1. 1. 1. 1. 1. 1. 1. 1. 1. 1	414			422		18 403
	6			7			8			9			10	
280		280	280		362	358		2123623	10000		N13655			460
280		283	2,823		385	327		-						448
	1			2			CA			4			ц	
10		280	280		317	280		405	366		455	435		465
	369 326 280 280 280 280 280	21 369 16 280 11 280 280 280 280	369       395         326       447         16       382         280       382         11       423         11       360         280       360         6       360         280       260         280       280         11       360         12       360         13       360         14       283         15       283         16       283         17       283         18       360         19       283         10       283	21       369     395     429       25     447     376       16     1       280     382     383       19     423     348       11     1       280     325     305       280     323     305       280     325     305       280     360     289       6     280     36       280     280     280       1     1     1	$\begin{array}{c c c c c c c c c } 21 & 22 \\ \hline 369 & 395 & 429 \\ \hline 326 & 447 & 376 \\ \hline 16 & 17 \\ \hline 16 & 17 \\ \hline 280 & 382 & 383 \\ \hline 19 & 348 \\ 11 & 12 \\ \hline 280 & 423 & 348 \\ 11 & 12 \\ \hline 280 & 360 & 280 \\ \hline 280 & 360 & 280 \\ \hline 280 & 280 & 280 \\ \hline 0 & 280 & 280 \\ \hline 0 & 280 & 280 \\ \hline \end{array}$	$\begin{array}{c ccccccccccccccccccccccccccccccccccc$	$\begin{array}{c c c c c c c c c c c c c c c c c c c $	$\begin{array}{c c c c c c c c c c c c c c c c c c c $	$\begin{array}{c c c c c c c c c c c c c c c c c c c $	$\begin{array}{c c c c c c c c c c c c c c c c c c c $	$\begin{array}{c c c c c c c c c c c c c c c c c c c $	21 $22$ $23$ $24$ 369       395       429       384       417       351       368       280         326       447       376       424       377       365       341       286         16       17       18       19       19       23       22       23         280       382       383       430       435       429       441       388         19       21       22       23       24       23       23         280       382       383       430       435       429       441       388         19       21       21       22       23       23       23       23         280       423       348       455       393       444       402       398         11       12       13       14       401       452       449       447         280       360       29       435       370       465       414       453         6       7       8       9       366       432       419       457         280       283       385       522       447	21       22       23       24       100         369       395       429       384       417       331       368       280       297         326       447       376       424       377       365       341       266       283         16       17       18       19       19       19       21       22       23       23       368       408         280       382       383       430       435       429       441       388       408         280       423       348       455       393       444       402       398       376         11       12       13       14       14       14       17       18       19       17         280       525       305       414       401       452       449       447       456         11       12       13       14       17       16       17       17       18       9       10       17       18       9       10       17       18       17       15       16       117       14       12       16       17       18       9       10       10 <t< td=""><td><math display="block">\begin{array}{c c c c c c c c c c c c c c c c c c c </math></td></t<>	$\begin{array}{c c c c c c c c c c c c c c c c c c c $

Fig.13 Reinforcement *a*<sub>s2-,req+min</sub> [mm<sup>2</sup>/m] – Direct values in nodes

4			31	-		33		9400	34			35	1	42 M.	3
021	428		321	376			292	(	2 <b>8</b> ()	280		280	280		0
		21			22			23			24			25	
26	341		280	280		280) 27	250		0 	280		0	0		0
20	394		280	280		280	280		0	280		0	0		- <del>3</del> 6 0
		16			17			18			19			20	
00			250	280		0	0		0	0		0	0		0 -24 0
20	321		280	280			280		0	0		0	0		0
		11			12			13			14			15	
1 4	280		0	280		0	Ō		0	ο		0	0		0
14	230		280	280		0	0		0	0		0	0		
		6			7			8			9			10	
70	280		0	0		0	0		0	0		0	0		0
32	280		<del>36</del> 0	0		0	0		0	0		0	0		0
>		1			2			3			4			5	
1	280		0	0		0	0		0	0		0	0		0

Fig.14 Reinforcement  $a_{s2+,req+min}$  [mm<sup>2</sup>/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  $1^{st}$  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

$$w_{cal} \leq w_k$$

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:

#### $a_{s2,ULS+SLS \geq} a_{s2,req+min \geq} a_{s2,req}$

(compare with Fig.16,17 and Tab.3,4). In the Crack Proof algorithm not only the bar diameters  $\phi$  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)  $\rightarrow a_{s,min,constr} = 393$  mm<sup>2</sup>/m will be encountered at several design points, both upper and lower reinforcement – as marked in Fig.16,17.

(6)

(7)

Design defaults	Crack proof
General Calculation ULS Shear SLS Creep	General kt - load duration factor according \$7.3.4(2) 0.6 ( 0.4 for long term loads; 0,6 for short term loads ) - 2D structures
CDD CDD Detailing provisions Warnings and errors	Limit bar distances bar distance on face Zp+200 on face Zp-200 mm
	Concrete tension strength fct,eff in early stage of hardening

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

4	439		461	427		397	(393		393	393		393	393		393
		21			22			23			24			25	
4	446		467	489		461	the second second		426	452		393 29	400		
	415		501	453		488	455		451	432		394	393		393
		16			17			18			19			20	
	393		461	461			499		494	502		466 23	480		436
	393		487	434		21 511	470		-	476		474	459		420
		11			12			13			14			15	
	393		420	406			476		511	508		506	512		49
	393		13 447	393		499 499	454		<del>16</del> 518	485		510	490		477
		6			7			8			9			10	
	393		393	393		449	431		497	488		514	515		515
	393		<u>36</u> 393	393		465	419		507	472		1 <del>0</del> 520	498		507
Ľ		1			2			CA			4			5	
Pa	C	ſ	393	393		415	393	٦	479	451		512	499	ł	(519

Fig.16 Reinforcement  $a_{s2-, ULS+SLS}$  [mm<sup>2</sup>/m] – Direct values in nodes

	486	<del>31</del> 412	451	3	93	393		393	393		393	393	)	0
	21		22		23		24		25					
26	426	393 25	393	3	93 27	393		0	393	]	0	0		0
20	463	393	393	3	93	393		0	393	J	0	0		<u>3</u> 0
	16	5		17			18			19			20	
20	393	393	393	Ĭ	0	0		0	0		0	0		0
20	413	393	393		0	393		0	0		0	0		0
	1	1		12			13			14			15	
14	393	0	393		0	0		0 	0		0	0		0
14	393	393	393		0	0		0	0		0	0		0
	6	(		7			8			9			10	
70	393	0	0		0	0		0	0		0	0		0
32	393		0		0	0		0	0		0	0		-12 0
>	1			2			CA	_		4			5	
1	393	0	0		0	0		0	0		0	0		0

Fig.17 Reinforcement  $a_{s2+,ULS+SLS}$  [mm<sup>2</sup>/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).

Elem/Node	Reinford [mm²/m]	ement at face -	$Z_p \sim a_{s1-}$	Reinforcement at face $-Z_p \sim a_{s2-}$ [mm <sup>2</sup> /m]			
	a <sub>s1-,req</sub>	$a_{s1-,req+min}$	$a_{s1-,ULS+SLS}$	$a_{s2\text{-},req}$	$a_{s2-,req+min}$	a <sub>s2-,ULS+SLS</sub>	
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 3. Comparison of the reinforcement stages in design points 5/2, 21/4 and 1/1 ~ Lower face

Table 4. Comparison of the rei	nforcement stages in design	1 points 5/2, 21/4 and 1	/1 ~ Upper face
L	8 8	1 /	11

Elem/Node	Reinforc [mm²/m]	ement at face +	$-Z_p \sim a_{s1+}$	Reinforcement at face $+Z_p \sim a_{s2+}$ [mm <sup>2</sup> /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)
- 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 (3<sup>rd</sup> release)