lateral buckling restraint - attaches - steel check - creep - charges climatiques - dynamic analysis - lateral buckling brandweerstandsanalyse - timber - 1st order - verstijvers - buisverbinding - diseño de planos de armaduras - pandeo lateral verbindingen - shear connection - verificación - armatures longitudinales - pórtico - unión base columna - voorontwerp - unión tubular - haunch - connexion moment - cimbras - vérification acier - unity check - Eurocode 2 - mesh - retaining wall - raidisseur - Eurocode 3 - longitudes de pandeo - connections - ACI 138 - acero - 2nd ordre - portal frame - Eurocode 8 - andamios - kip dwarskrachtverbinding - BS 8110 - dalle de fondation - seismische analyse - armaduras longitudinales - BIM - gelaste verbinding - 2de orde - buckling - funderingszool - poutre sur plusieurs appuis - maillage - malla - uniones - 2D raamwerken - fire resistance analysis voiles - cracked deformation - gescheurde doorbuiging - longueurs de flambement - pandeo - reinforcement - unity check - cantonera - dynamische analyse - hout - ossatures 3D - koudgevormde profielen - placa de extreme - 1er orden - continuous beam connexion soudée - momentverbinding - praktische wapening - renforts au déversement - fluencia - estribos - déformation fissurée - EHE - beugels - Eurocódigo 3 - platine de bout - análisis dinámico - column base plate - kruip - rigid link - welded connection - charpente métallique - moment connections - estructuras 2D - kniestuk - assemblage métallique - 3D raamwerken - second ordre - beam grid – cargas climáticas – Eurocode 2 – Eurocode 5 – wall – deformación fisurada – lien rigide – enlace rígido – 2D frames – estructuras 3D – éléments finis - vloerplaat - steel connection - scheurvorming - integrated connection design - armatures pratiques - analyse sismique - nieve y viento - practical reinforcement - charges mobiles - dalle - wapening - perfiles conformados en frío - Eurocode 3 connexion tubulaire - unión a momento - 3D frames - treillis de poutres - roof truss - practical reinforcement design - portique kipsteunen – análisis sísmico – Eurocode 8 – seismic analysis – B.A.E.L 91 – uniones atornilladas – bolts – ossatures 2D – eindige elementen – losa de cimentación - restricciones para el pandeo lateral - optimisation - wand - kniklengtes - end plate - dakspanten kolomvoetverbinding - stirrups - acier - staalcontrole - cálculo de uniones integrado - paroi - dessin du plan de ferraillage - stiffeners mobiele lasten – Eurocódigo 8 – Eurocódigo 5 – longitudinal reinorcement – doorlopende liggers – rigidizador – beton armé – fluage – CTE - connexion pied de poteau - langswapening - connexions - hormigón - neige et vent - elementos finitos - armaduras - cold

Getting started

# Diamonds



formed steel - jarret - uittekenen wapening - puente grúa - analyse dynamique - flambement - keerwanden - optimisation - steel - cercha - 2º orden - slab on grade foundation - entramado de vigas - Eurocode 5 - prédimensionnement - multi span beam - bouten - armatures - floor slab - poutre continue - pared - staal - 1er ordre - NEN 6770-6771 - connexion cisaillement - losa - déversement - viga continua - predimensionering - 1ste orde - unión metálica - CM 66 - madera - análisis resistencia al fuego - verbindingen - 2nd order - bois - Eurocode 2 - profilés formés à froid - verificación acero - predesign - unión soldada - fisuración - beton - muro de contención - Optimalisatie - foundation pads - fissuration - concrete - AISC-LRFD - HCSS - assemblage métallique - Eurocode 3 - viga con varios apoyos - armaduras prácticas - balkenroosters - unión a cortante - buckling length - boulons - cracking - Eurocode 8 - knik - Eurocode 2 - radier - eindplaat - Eurocódigo 2 - FEM - tornillos - NEN 6720 - moving loads - balk op meerdere steunpunten - cargas móviles - funderingsplaat - étriers - analyse resistance au feu- cercha - globale knikfactor - dynamische analyse- wapening - mailage - malla - uniones - radier

© BuildSoft, version 18.2

Any representation or reproduction, of all or part of the materials is strictly prohibited without a prior written permission from BuildSoft.

By purchasing the **Diamonds** software, the purchaser acquires a license of use. Under no circumstances can the user assign the license in part or in total to a third party without a prior written consent of the publisher.

The publisher holds no responsibility for possible faults that the program and/or the here mentioned manual may include and declines any responsibility for damages arising from misuse of the **Diamonds** software and/or the here mentioned manual.

# Table of contents

1	IN	TR	ODUCTION	1
2	BA	<b>SI</b>	C SKILLS	2
2	2.1	Pro	OJECT MANAGEMENT	2
2	2.2	Mo	DDEL MANAGEMENT	3
2	2.3	Dr	AWING	4
2	2.4	Sei	LECTING	5
2	2.5	WI	NDOW CONFIGURATION	6
2	2.6	TY	PES	9
2	2.7	Но	т кеуs1	0
2	2.8	Co	NFIGURATION GRAPHICAL CARD	1
	2.8	8.1	Problem 1	1
	2.8	3.2	Problem 2 1	2
	2.8	3.3	Problem 3 1	3
3	EX	KAN	<b>IPLES IN REINFORCED CONCRETE1</b>	15
3	8.1	Ex	AMPLE 1: A CONTINUOUS BEAM 1	17
	3.1	.1	Purpose of the exercise	17
	3.1	.2	Defining the structure	17
	3.1	.3	Defining the loads	22
	3.	.1.3.1	Creating the load groups	22
	3.	.1.3.3	Making combinations	.26
	3.1	.4	Generating the mesh	28
	3.1	.5	The global elastic analysis	28
	3.1	.6	Calculating the reinforcement	36
	3.1	.7	Calculating the cracked deformation	41
3	3.2	Ex	AMPLE 2: A PRESLAB FLOOR	16
	3.2	2.1	Purpose of the exercise	16
	3.2	2.2	Defining the structure	16
	3.2	2.3	Defining the loads	56
	3. 3.	.2.3.1	Creating the load groups Filling up the load groups	.66
	3.	.2.3.3	Making combinations	.68
	3.2	2.4	Generating the mesh	58
	3.2	2.5	The global elastic analysis	/0
	3.2	2.6	Calculating the reinforcement	17
	3.2	2.7	Calculating the cracked deformation	32
	3.2	2.8	Making a report	36
	3. 3.	.2.8.1	Sub report 1: Geometry	.87
	3.	2.8.3	Sub report 2: Loads	.96 07
	3. 3.	.2.8.5	Sub report 4: Detailed results	.00
~	3.	.2.8.6	Preview	03
1	).3 ~~~	EX	AMPLE 5: A FOUNDATION SLAB	J4 کر
	3.3	0.1	Defining the structure	
	3.3	.2	Defining the loads	13

	3.3.2.1	Creating the load groups	115
	3.3.2.2 3.3.2.3	Making combinations	115
	3.3.3	Generating the mesh	119
	3.3.4	The global elastic analysis	120
	3.3.5	Calculating the reinforcement	124
	3.3.6	Calculating the cracked deformation	127
3	.4 Ex	AMPLE 4: MAKING A $3D$ model with bearing walls in masonry.	131
	3.4.1	Defining the structure	132
	3.4.2	Defining the loads	148
	3.4.2.1 3.4.2.2	Creating the load groups	148
	3.4.2.3	Making combinations	151
	3.4.3	Generating the mesh	151
	3.4.4	The global elastic analysis	151
	3.4.5	Calculating the reinforcement	152
-	3.4.6	Calculating the cracked deformation	154
3	.5 Ex	AMPLE 5: MODELLING THE BASEMENT	156
	3.5.1	Defining the structure	157
	3.5.2	Defining the loads	167
	3.5.2.1 3.5.2.2	Creating the load groups Filling up the load groups	167
	3.5.2.3	Making combinations	170
	3.5.3	Generating the mesh	170
	3.5.4	The global elastic analysis	170
	3.5.5	Calculating the reinforcement	172
4	EXAN	IPLES IN STEEL	174
<b>4</b> 4	<b>EXAN</b> .1 Ex	<b>IPLES IN STEEL</b> AMPLE 1: 2D FRAME	<b>174</b> 175
<b>4</b> 4	<b>EXAN</b> .1 Ex 4.1.1	<b>IPLES IN STEEL</b> AMPLE 1: 2D FRAME         Purpose of the exercise	<b>174</b> 175 175
<b>4</b> 4	<b>EXAN</b> .1 EX 4.1.1 4.1.2	<b>IPLES IN STEEL</b> AMPLE 1: 2D FRAME         Purpose of the exercise         Defining the structure	174 175 175 175
<b>4</b> 4	<b>EXAN</b> .1 EX 4.1.1 4.1.2 4.1.3	<b>IPLES IN STEEL</b> AMPLE 1: 2D FRAME         Purpose of the exercise         Defining the structure         Defining the loads	174 175 175 175 182
<b>4</b> 4	EXAN .1 EX 4.1.1 4.1.2 4.1.3 4.1.3.1 4.1.3.2	<b>IPLES IN STEEL</b> AMPLE 1: 2D FRAME         Purpose of the exercise         Defining the structure         Defining the loads         Creating the load groups         Filling up the load groups	172 174 175 175 175 182 183 187
<b>4</b> 4	EXAN .1 EX 4.1.1 4.1.2 4.1.3 4.1.3.1 4.1.3.2 4.1.3.3 4.1.4	<b>IPLES IN STEEL</b> AMPLE 1: 2D FRAME         Purpose of the exercise         Defining the structure         Defining the loads         Creating the load groups         Filling up the load groups         Making combinations	172 174 175 175 175 182 183 187 195
<b>4</b> 4	EXAN 1 EX 4.1.1 4.1.2 4.1.3 4.1.3 4.1.3.1 4.1.3.2 4.1.3.3 4.1.4 4.1.5	<b>IPLES IN STEEL</b> AMPLE 1: 2D FRAME         Purpose of the exercise         Defining the structure         Defining the loads         Creating the load groups         Filling up the load groups         Making combinations         Generating the mesh         The alphal elastic analysis	<b>174</b> 175 175 175 182 183 187 195 197
<b>4</b> 4	EXAN 1 EX 4.1.1 4.1.2 4.1.3 4.1.3 4.1.3 4.1.4 4.1.5 4.1.6	<b>IPLES IN STEEL</b> AMPLE 1: 2D FRAME         Purpose of the exercise         Defining the structure         Defining the loads         Creating the load groups         Filling up the load groups         Making combinations         Generating the mesh         The global elastic analysis	<b>174</b> 175 175 175 182 183 187 197 197
<b>4</b> 4	EXAN 1 EX 4.1.1 4.1.2 4.1.3 4.1.3.1 4.1.3.2 4.1.3.3 4.1.4 4.1.5 4.1.6 4.1.6	<b>IPLES IN STEEL</b> AMPLE 1: 2D FRAME         Purpose of the exercise         Defining the structure.         Defining the loads         Creating the load groups         Filling up the load groups         Making combinations         Generating the mesh.         The global elastic analysis         Parameters for steel verification	172 174 175 175 175 175 182 183 187 195 197 197 206 206
<b>4</b> 4	EXAN 1 EX 4.1.1 4.1.2 4.1.3 4.1.3 4.1.3 4.1.4 4.1.5 4.1.6 4.1.61 4.1.62	<b>IPLES IN STEEL</b> AMPLE 1: 2D FRAME         Purpose of the exercise         Defining the structure.         Defining the loads         Creating the load groups         Filling up the load groups.         Making combinations         Generating the mesh.         The global elastic analysis         Parameters for steel verification         Buckling         Lateral torsional buckling	172 174 175 175 175 175 182 183 187 197 197 206 206 211
<b>4</b> 4	$\begin{array}{c} \textbf{EXAN} \\ \textbf{EXAN} \\ \textbf{.1}  \textbf{EX} \\ \textbf{4.1.1} \\ \textbf{4.1.2} \\ \textbf{4.1.3.1} \\ \textbf{4.1.3.2} \\ \textbf{4.1.3.3} \\ \textbf{4.1.4} \\ \textbf{4.1.5} \\ \textbf{4.1.6} \\ \textbf{4.1.6.1} \\ \textbf{4.1.6.2} \\ \textbf{4.1.7} \end{array}$	<b>IPLES IN STEEL</b>	<b>174</b> 175 175 175 175 182 183 187 197 197 197 206 206 211 212
<b>4</b> 4	$\begin{array}{c} \textbf{EXAN}\\ \textbf{EXAN}\\ .1  \textbf{EX}\\ \textbf{4.1.1}\\ \textbf{4.1.2}\\ \textbf{4.1.3}\\ \textbf{4.1.3}\\ \textbf{4.1.3}\\ \textbf{4.1.3}\\ \textbf{4.1.4}\\ \textbf{4.1.5}\\ \textbf{4.1.6}\\ \textbf{4.1.6}\\ \textbf{4.1.6}\\ \textbf{4.1.7}\\ \textbf{4.1.8}\\ \end{array}$	<b>IPLES IN STEEL</b> AMPLE 1: 2D FRAME         Purpose of the exercise         Defining the structure.         Defining the loads         Creating the load groups.         Filling up the load groups.         Making combinations         Generating the mesh.         The global elastic analysis         Parameters for steel verification         Buckling         Lateral torsional buckling.         Steel verification .         Cross-section optimization.	172 174 175 175 175 182 182 183 197 197 197 206 206 212 212 220
<b>4</b> 4 4	$\begin{array}{c} \textbf{EXAN}\\ \textbf{1}  \textbf{EX}\\ \textbf{4.1.1}\\ \textbf{4.1.2}\\ \textbf{4.1.3}\\ \textbf{4.1.3.1}\\ \textbf{4.1.3.2}\\ \textbf{4.1.3.3}\\ \textbf{4.1.4}\\ \textbf{4.1.5}\\ \textbf{4.1.6}\\ \textbf{4.1.6}\\ \textbf{4.1.6.2}\\ \textbf{4.1.7}\\ \textbf{4.1.8}\\ \textbf{2}  \textbf{Ex} \end{array}$	<b>IPLES IN STEEL</b> AMPLE 1: 2D FRAME         Purpose of the exercise         Defining the structure         Defining the loads         Creating the load groups         Filling up the load groups         Making combinations         Generating the mesh         The global elastic analysis         Parameters for steel verification         Buckling         Lateral torsional buckling         Steel verification         Cross-section optimization         AMPLE 2: 3D HALL	$\begin{array}{c} 172 \\ 174 \\ 175 \\ 175 \\ 175 \\ 175 \\ 182 \\ 182 \\ 197 \\ 197 \\ 197 \\ 206 \\ 211 \\ 212 \\ 220 \\ 223 \end{array}$
<b>4</b> 4 4	$\begin{array}{c} \textbf{EXAN}\\ \textbf{1}  \textbf{EX}\\ \textbf{4.1.1}\\ \textbf{4.1.2}\\ \textbf{4.1.3.1}\\ \textbf{4.1.3.2}\\ \textbf{4.1.3.3}\\ \textbf{4.1.4}\\ \textbf{4.1.5}\\ \textbf{4.1.6.1}\\ \textbf{4.1.6.2}\\ \textbf{4.1.7}\\ \textbf{4.1.8}\\ \textbf{2}  \textbf{Ex}\\ \textbf{4.2.1} \end{array}$	<b>IPLES IN STEEL.</b> AMPLE 1: 2D FRAME.         Purpose of the exercise         Defining the structure.         Defining the loads         Creating the load groups.         Filling up the load groups.         Filling up the load groups.         Making combinations.         Generating the mesh.         The global elastic analysis .         Parameters for steel verification         Buckling.         Lateral torsional buckling.         Steel verification         Cross-section optimization.         AMPLE 2: 3D HALL         Purpose of the exercise	172 174 175 175 175 182 183 197 197 197 206 206 211 212 220 223 223 223
<b>4</b> 4	$\begin{array}{c} \textbf{EXAN} \\ \textbf{1}  \textbf{EX} \\ \textbf{4.1.1} \\ \textbf{4.1.2} \\ \textbf{4.1.3} \\ \textbf{4.1.3.1} \\ \textbf{4.1.3.2} \\ \textbf{4.1.3.3} \\ \textbf{4.1.4} \\ \textbf{4.1.5} \\ \textbf{4.1.6} \\ \textbf{4.1.6.1} \\ \textbf{4.1.6.2} \\ \textbf{4.1.7} \\ \textbf{4.1.8} \\ \textbf{2}  \textbf{EX} \\ \textbf{4.2.1} \\ \textbf{4.2.2} \\ \textbf{4.2.2} \end{array}$	<b>MPLES IN STEEL.</b> AMPLE 1: 2D FRAME.         Purpose of the exercise         Defining the structure.         Defining the loads         Creating the load groups.         Filling up the load groups.         Making combinations         Generating the mesh.         The global elastic analysis .         Parameters for steel verification         Buckling.         Lateral torsional buckling.         Steel verification .         Cross-section optimization.         AMPLE 2: 3D HALL .         Purpose of the exercise .         Defining the structure.	$\begin{array}{c} 172 \\ 174 \\ 175 \\ 175 \\ 175 \\ 175 \\ 182 \\ \dots 183 \\ \dots 195 \\ 197 \\ 197 \\ 206 \\ \dots 211 \\ 212 \\ 220 \\ 223 \\ 223 \\ 223 \\ 223 \\ 223 \\ 223 \end{array}$
<b>4</b> 4	$\begin{array}{c} \textbf{EXAN} \\ \textbf{.1}  \textbf{EX} \\ \textbf{AN} \\ \textbf{.1}  \textbf{EX} \\ \textbf{4.1.1} \\ \textbf{4.1.2} \\ \textbf{4.1.3.1} \\ \textbf{4.1.3.2} \\ \textbf{4.1.3.3} \\ \textbf{4.1.4} \\ \textbf{4.1.5} \\ \textbf{4.1.6} \\ \textbf{4.1.6.1} \\ \textbf{4.1.6.2} \\ \textbf{4.1.7} \\ \textbf{4.1.8} \\ \textbf{.2}  \textbf{EX} \\ \textbf{4.2.1} \\ \textbf{4.2.2} \\ \textbf{4.2.3} \\ \textbf{4.2.3} \end{array}$	<b>IPLES IN STEEL</b> .         AMPLE 1: 2D FRAME.         Purpose of the exercise         Defining the structure.         Defining the loads         Creating the load groups.         Filling up the load groups.         Making combinations         Generating the mesh.         The global elastic analysis         Parameters for steel verification         Buckling         Lateral torsional buckling.         Steel verification         Cross-section optimization.         AMPLE 2: 3D HALL         Purpose of the exercise         Defining the loads	172 174 175 175 175 182 183 197 197 197 206 206 211 212 220 223 223 223 223 223 236
<b>4</b> 4 4	$\begin{array}{c} \textbf{EXAN}\\ \textbf{.1}  \textbf{EX}\\ \textbf{4.1.1}\\ \textbf{4.1.2}\\ \textbf{4.1.3}\\ \textbf{4.1.3.1}\\ \textbf{4.1.3.2}\\ \textbf{4.1.3.3}\\ \textbf{4.1.4}\\ \textbf{4.1.5}\\ \textbf{4.1.6}\\ \textbf{4.1.6.1}\\ \textbf{4.1.6.2}\\ \textbf{4.1.7}\\ \textbf{4.1.8}\\ \textbf{.2}  \textbf{EX}\\ \textbf{4.2.1}\\ \textbf{4.2.2}\\ \textbf{4.2.3}\\ \textbf{4.2.3.1}\\ \textbf{4.2.3.2} \end{array}$	<b>IPLES IN STEEL</b> AMPLE 1: 2D FRAME         Purpose of the exercise         Defining the structure.         Defining the loads         Creating the load groups         Filling up the load groups         Making combinations         Generating the mesh.         The global elastic analysis         Parameters for steel verification         Buckling         Lateral torsional buckling.         Steel verification         Cross-section optimization.         AMPLE 2: 3D HALL         Purpose of the exercise         Defining the loads         Creating the load groups	$\begin{array}{c} 172 \\ 174 \\ 175 \\ 175 \\ 175 \\ 175 \\ 182 \\ \dots 183 \\ \dots 195 \\ 197 \\ 197 \\ 206 \\ \dots 211 \\ 212 \\ 220 \\ 223 \\ 223 \\ 223 \\ 223 \\ 223 \\ 223 \\ 223 \\ 236 \\ \dots 236 \\ \dots 239 \end{array}$
<b>4</b> 4	$\begin{array}{c} \textbf{EXAN} \\ \textbf{.1}  \textbf{EX} \\ \textbf{AN} \\ \textbf{.1}  \textbf{EX} \\ \textbf{4.1.1} \\ \textbf{4.1.2} \\ \textbf{4.1.3.1} \\ \textbf{4.1.3.2} \\ \textbf{4.1.3.3} \\ \textbf{4.1.4} \\ \textbf{4.1.5} \\ \textbf{4.1.6} \\ \textbf{4.1.6} \\ \textbf{4.1.6} \\ \textbf{4.1.6} \\ \textbf{4.1.6} \\ \textbf{4.1.6} \\ \textbf{4.1.7} \\ \textbf{4.1.8} \\ \textbf{.2}  \textbf{EX} \\ \textbf{4.2.1} \\ \textbf{4.2.2} \\ \textbf{4.2.3.1} \\ \textbf{4.2.3.3} \\ \textbf{4.2.3.3} \\ \textbf{4.2.3.3} \\ \textbf{4.2.3.3} \end{array}$	<b>IPLES IN STEEL</b>	172 174 175 175 175 175 182 197 197 197 206 220 223 236 238 236 238 236 238 236 238 236 238 236 238 236 238 236 238 236 238 236 238 236 238 236 238 236 238 238 238 236 238 248 388
<b>4</b> 4	EXAN 1 EX 4.1.1 4.1.2 4.1.3 4.1.3 4.1.3 4.1.4 4.1.5 4.1.6 4.1.6 4.1.6 4.1.7 4.1.8 2 EX 4.2.1 4.2.2 4.2.3 4.2.3 4.2.3 4.2.4 4.2.4	<b>IPLES IN STEEL</b>	$\begin{array}{c} 172 \\ 174 \\ 175 \\ 175 \\ 175 \\ 175 \\ 182 \\ \dots 183 \\ \dots 195 \\ 197 \\ 197 \\ 206 \\ \dots 211 \\ 212 \\ 220 \\ 223 \\ 223 \\ 223 \\ 223 \\ 223 \\ 223 \\ 223 \\ 223 \\ 223 \\ 223 \\ 223 \\ 223 \\ 223 \\ 223 \\ 223 \\ 224 \\ 248 \\ 248 \end{array}$

4.2.6	Parameters for steel verification	. 250
4.2.6.1	Buckling	250
4.2.6.2 1 2 7	Staal varification	252 252
4.2.7	Cross section ontimization	. 252
4.2.0	Calculating connections	. 233
<b>4.2.9</b> 4291	Detailed calculation of a connection	. 2 <b>3</b> 4
4.2.9.2	Selecting combinations	256
4.2.9.3 4 2 9 4	Calculating the connection	256
4.2.9.5	Assigning the connection.	261
4.2.9.6	Verifying the nodes in Diamonds	263
5 EAAN	WIPLES IN TIMBER	. 204
5.1  EX	AMPLE 1: 2D FRAME	. 264
5.1.1	Purpose of the exercise	. 264
5.1.2	Defining the structure	. 264
5.1.3	Defining the loads	. 271
5.1.3.1 5.1.3.2	Filling up the load groups	272
5.1.3.3	Making combinations	283
5.1.4	Generating the mesh	. 284
5.1.5	The global elastic analysis	. 285
5.1.6	Parameters for timber verification	. 293
5.1.6.1	Buckling Lateral torsional buckling	293
5.1.7	Timber verification	. 297
518	Cross-section optimization	301
5 2 Ex	AMPLE 2: 3D HALL	305
521	Purpose of the exercise	305
522	Defining the structure	305
523	Defining the loads	318
5.2.3.1	Creating the load groups	
5.2.3.2	Filling up the load groups	319
5 2 <i>A</i>	Generating the mesh	370
525	The global elastic analysis	329
526	Parameters for timber verification	330
5.2.6.1	Buckling	
5.2.6.2	Lateral torsional buckling	332
5.2.7	Timber verification	. 332
5.2.8	Cross-section optimization	. 333

# **1** Introduction

The best way to become familiar with the operation of Diamonds is making a few examples.

The examples introduce you to the many functions of the software without discussing them one by one. Depending on the acquired licenses, you will be able to go through one or more examples. At the start of each example the required licenses are communicated.

With Diamonds you can model and calculate a wide range of structures. We distinguish the following exercises in this manual:

- Examples in concrete
  - Calculating a continuous beam
  - Calculating a floor slab
  - Calculating a foundation slab
  - Making a 3D model with bearing walls in masonry
  - Calculating the 3D structure with the basement
- Examples in steel
  - Calculating a 2D frame
  - Calculating a 3D hall
- Examples in timber
  - Calculating a 2D roof
  - Calculating a 3D hall

# In order not to overload the examples, we have summarized a small description of all the functions in the gray frames. For detailed information, consult the Reference manual of Diamonds.

More information on the **implementation of the Eurocodes** in Diamonds can be found on our website <u>www.buildsoft.eu</u> under the section 'Support – About Eurocodes' (for the moment only available in Dutch).

Paragraph §2 handles some **basic skills** in Diamonds. It is strongly recommended you read this paragraph before starting with the examples. The examples assume that you have mastered these skills.

## Paragraph §3 describes **examples in reinforced concrete**, paragraph §4 **examples in steel** and paragraph §5 **examples in timber.**

Don't start with 'example 3' when you haven't tried 'example 1'. The explanation/ comments/ tips ... discussed in 'example 1', will not be repeated in 'example 3'.

# 2 Basic skills

Start Diamonds. You'll see this screen:

																1	Dia	amo	ond	s -	[Wir	ndo	wc	1 (	m)]																	×
🥮 <u>F</u> ile	Edit	View	Sel	ect	Disp	play	A	naly	rsis	0	otio	ns J	Win	dow	s j	lelp																										- 6 >
	2	1 Fð	<i>_</i>	m	ĸ	7 6	54	F		H	m			Т.,	11	XX		¥	4.0	4	10	lė -		<b>(</b> <del>1</del> <b>)</b>	Feg	IÝ.	11	1E	康	40												
* 6		Wir	ndow	1				•		1	lu	lser co	onfig	urati	on						•		\$	F14	<b>X</b>	2		和	白	14	ŋ €	20	Q D	3	Ē		D	R	ST.	P		
		1	18 35	3		18 8			8 8	12		25.3	5 8	- 5	- 25	35.3	8	58-1	2.8	1.15	- 25	35	12		8 B		- 22	葱	8. 83	- 5	- 35	82	10.2	8.35	- 82	- 28		8 S	- 5		Active lev	rel
T p		$\sim$	3235	13	52	32.3	8. 12	- 6	2.12	- 92	12	32.3	N (S	- 8	- 32	8.	1	81.3	8.9	6.13	32	22	8	62.1	92 - 28	12	- 61	28.3	8.13	- 8	22	33	8.3	1.8	$^{22}$	61	25.3	18 19	- 8		Level 1	-
∩⊞		1	23 32	88	-8	83 3	8 8	- 3	8.8	- 58	20	83	2 8	- 2	- 83	8.3	×.	39	8 8	8.89	83	2	25	-8	83 52	80	33	8	98 - 36	8	32	98	-8-9	3.32	8	39	83	8 8	35		A	
<u> </u>			\$P \$P	12	- 55	ф з	8 8	1 2	2.9	- 12	82	62.5	2 3	- 53	- 42	83	22	38. I	8 9	2.83	10	22	38	<u>85</u>	ф 92	- 84	22	98 I	9.33	- 22	- 82	28	15 1	2.42	83	82	83	<b>a</b> 8	- 8		E AI	2.70 m
A	1	1	<u>e</u> 30	$\mathbb{R}^{2}$	53	<u>63</u> 8	5 33	2	8.8	13	83	22.3	5 8	- 8	10	(S)	32	85	8 3	8.33	10	35	22	8	8 B	13	25	8	8 B	23	- (5)	32	8.4	8.00	83	28	8.3	(h. 13)	- 8		A.	2 70 -
U	¢۷ [		85.38	аñ,	12	85.3	8 33	- 3	8 8	- 83	54	8.2	5 8	- 10	- 83	8.7	64	16	8 8	6 - 58	83	18	8	10	8 8	- 24	10	10	8 8	10	36	ΰł.	82.8	8.38	81	10	8.3	8 8	Ŕ		E **	2.70 m
$\square$			30 38	- 12	-8	90 B	66 58 		8 88 	- 68	34 	- 22 - 3	8 8 	- 2	- 20	83	54 	89 - I 10 - I	8.9	2.54	- 20	36 	a .	-8	80 - 98 	- 24	- 22	8.1	09. 34 		- 36	99 	8.8	0 36 	- 84	-	83	28 - 88 00 - 14			<i>—</i>	
-	¥		49	<u>.</u>	50,	49 9	p 164		- 12	- 14	64	49.5	a 8		- 40	101.1	04	40 0	90 - 14	1 104	45	545	а Т	201	g) 540	124	20	40.0	a iig	- 22	- 145	्र	10 A	n 146	104	<u>20</u>	41.1	φ 10	- 20		HH Leve	ei managei
1		~	AL 85	12	-23	AR 8	5 83	8	88	12	87	A 8	5 X	8	2	3	3	88	8.8	1.13	10	35	8	8	82 (B)	107	85	88	8 8 	8	- 35	32	88	8.85	89	88	83	8 N	8	Ť		and lower
1.	999 B	11	85 . 35	100	10	8 8	6 33	1.1	8.8	100	28	8.2	5 8	1	1.85	28.7	19	10	8 8 	5.58	- 85	28	a	10	8 8	- 28	- 50	- 25	8 De		- 25	88. 	83	5.96	- 28	58 	83	8.8	10		Sine Grou	una ievei
1	14141	ter an	48.04	25	255	<b>9</b> 8 0	W 88		2. 9	- 14	224	21.0	2 - 10		- 27	190 - 1 1	24	492	as 16	7 24	22	24	254	-55	ar ar	- 24	200	35	9 - 59 	- 20		28	10 1	8 - 34 -	- 24	201	-	92 (S			Draw	ing plane
17		1.5		88	- 33	18 B	8 8	1	8 8	8	85	- 28	\$ 8	- 3	- 28	2	8	<u>88</u> -	2 8	1.55	- 10	35	88	-32	18 38	35	- 53	82	8 8	- 22	35	88	32 8	8 35	85	<u>\$8</u>	Ø. 1	2 S	- 22		X = 0	).00 m
T	H~ 1		22 20 20 20	153	- 22	22 3 25 4	N 82 1. 1.1		8 22		100		8 (8) 8 (8)			81.1		80 I 160 I	85 98 U 10	1.12	392 475	35	23 	10	82 80 01 01	100	- 60	75 3		8	1.00	38 	10.1	2.55	12	82 100	15 3	20 12 10 10			Y = 2	2.70 m
N	.JA 9		10 10		- 55				8 8	100	8		5 8 8 8			83	a Si	88 I 38 I		5 08. 5 03.	- 15	8	8	52	83 (8) (6) (8)	100	- 22			- 2		98 32	10 10	1.00	23	- 52	8		2	1	Z = 0	).00 m
-	×6 * 7																					£.,																			Represent	tation
UT.	DE	5	25 50	107	- 22	25 S 20 B	8 88 8 82			: 85 : 62	200 200	- 25 - 5	8 88 8 88		- 15	20 - 3 20 - 3	8 2	200 - 1 120 - 1	2 8 2 8	5 85. 2 64.	÷		102 104	- 22	00 00 40 00	- 22	- 23	- 00 - 1 - 34 - 1	85 85 67 85	- 22	200 200	202 202		5 55 2 32	- 22 - 24	20	21	5 8 6 8	100 - 400		63.6	AA
250	Apr -	¥4	25 . 56		18	25 9	4 IS			- 65	14	25.5		1	- 25	<u>.</u>		8		6.84	- 25	2	2	23	25 56	14	- 62		n 14	- 6	- 94	a.	11	6 94	33	- 12	12.1	a 8			~~~	v < v
闸	este I	4	**	87	37	28.3	8 12	1	8 12	18	22	23.3	2 8	3	- 23	32-3	8	88 - I	2.8	8 8	28	32	87	37	2 2	12	23	12	8 8	2	32	87	8.5	8 32	83	18	12	8.8	1		Sale 1	1
-	~ ~ ~	A.,	as 94		12	a	6 33				-	a				a					25	38		<b>1</b> 211	85 98.		25		a		- 20	-			-	85					~	× .
<i>**</i> ~	1	$\mathcal{T}$	89.38	<i>8</i> 8	-8				8.8	- 88	5÷	- 88 - 8	9 X			8.1		35		8 39	89	÷	æ	-2		- 58	- 55			- 33	- 33	<i>3</i> 8			se.	33	81	8 8	- 8	Ť,	Size	
-	1	т.	10 S	33	55	63 S	8 8	1 3	8.9	- 87	84	12.3	2 8	ß	- 32	83	2	88	8 8	2.81	100	2	3	85	e s	23	22	98 I	8 8	33	- 82	38	is 1	8.8	8	32	$\otimes$	a s	- 23		Font	10
Ble	M 4	10°	20 20	12	23		a 53			12	82	21.2		- 92		8.1	22	62.3		8.85	10	88	10	88.3		32	62	25	a 15			10	s. 1		32	52	8	a s				
E.			32 8	13	12	32.3	N 12	- 8	6.92	12	22	32.3	N 35	. 6	32	8	3	8	8.9	6.18	32	88	13	62	5. R	12	10	28	1.12	8		18	8.3	6.8	22	10	83	18 19	- 8		Symbols	10 韋
			83 38	28	-83	83 3		3	8.9	- 58	8	83	8 8	- 10	- 8	8.3	8	39	8.5	8.8	93	8	25	-	88 58	8	32	8	8 8 S	8	32	88	8.3	8.8	8.	32	83	8 8	35		Loads	10 🚖
			\$3 S	33	55	83 S	9 8	1 3	8.9	12	82	42 S	2 8	13	- 47	8	a.	3X	8 3	5 81	10	2	35	\$S	17 SP	22	22	98 I	9 8	33	- 82	22	68 B	2.32	81	22	83	9 X	23			_
			88 - 38	82	13	8.8	8 8		8 8	8	85	83	8 8	- 8	.8	8.3	8	28 -	2.8	2.82	38	33	12	- 82	8 8	35	23	20	8 B	22	33	82	8.8	8.35	82	28	8	8 S	2		Results	10 💠
			85.38	$\partial t$	$\hat{n}$	85.3	6 3	- 3	8.8	- 83	81	8.3	6 8	- 10	8	8.3	8	38	8.8	6.58	- 83	8	8	10	8 8	St	10	80	8. 6	10	36	88	82.8	8.38	$^{St}$	$\overline{S}$	8.3	8 8	10	13	Show grou	ups
			25 32	122	-	35 3	6 33		1	65	88	25.3	8	8	20	96-3	8	81	8.3	1.55	25	96	8	88	20 SF	35	- 62	32	14 35 -	- 53	36	8	83	5.36	35	83	32.3	19 - 18 -	- 13	1	None	
			28 . 22	84	-55	28 8	2 22		8 8	89	82	83	8 8	3	1.63	39 I	8	<u>48</u> -	8 S	1 18	28	22	84	-55	28 22	124	23	92 I	8 8	20	- 322	84	38 3	8 32	84	38	19 I	°	- 2	- 1	hine	
			23 - 35	$\gtrsim$	22	<u>es</u> 8	5 8	2	8.8	- 33	83	<u>s</u> :	5 8	- 8	- 22	83	3	85	8 3	1.57	12	35	22	2	8 8	33	35	8	a 8	3	- 35	12	83	8.30	$\otimes 2$	85	8.3	3	->			
			83 38	8	10	85.2	6 . 59	- 3	8 8	- 19	58	8.3	6 8	- 10	- 8	36.0	9	<u>85</u>	8.8	6.58	- 83	38	28	82.1	83 - 38	- 28	- 55	36.1	9 . S	- 5	36	88	8.3	5.36	-58	53	8.3	3	z			

In the paragraphs that follows we review briefly:

- How to manage a project.
- How to manage a model.
- How you can draw things in Diamonds.
- How to select elements.
- What a 'configuration window' is and how you can adjust it.
- What a 'design type' is and how you can adjust it.

## 2.1 Project management

With the icon D you create **a new project**. This function is also available through the menu 'File – New'.

With the icon  $\checkmark$  you open **an existing project**. This function is also available through the menu 'File – Open'.

With the icon  $\blacksquare$  you save the current project. This function is also available through the menu 'File – Save'.

With the icon  $\square$  you obtain **a preview of the active window**. This function is also available through the menu 'File – Print preview'.

With the icon solution you **print the content the active window**. This function is also available through the menu 'File – Print window'.

With the icon vou **open the report manager**. This function is also available through the menu 'File – Report Manager'. The report manager is discussed in §3.2.8.

## 2.2 Model management

The following functions allow you to manipulate the view of the model:

- <image>
  - Moving the model
    - Mouse: keep the scroll (wheel) of the mouse pressed and move the mouse.
    - o Icon: use the button 🖤
  - Zoom in and out
    - o Mouse: move the scroll (wheel) of the mouse
    - $\circ$  Icon: use the buttons and and  $\Huge{eq}$
  - 3D rotate
    - Mouse: keep the scroll (wheel) of the mouse and the SHIFTkey pressed. Move the mouse.
    - Sliders: move the slider on the right/ under the model to rotate the view of the 3D model.
  - Centre the model on the screen: X or F12.
  - Choose representation

wireframe tra

transparent

transparent

non-



colour solid

colour solid wit borders



- The size of the font can be changed in the pallet on the size of Diamonds. Note that you can also adjust the size of the symbols, the loads and the results.

Size	
Font	15 🚖
Symbols	15 🜲
Loads	15 🚖
Results	27 🚖

- You can change the units and the number of decimals thought the menu 'Options – Units and decimals'.

# 2.3 Drawing

Lines can be drawn in different ways:

- Directly on the screen (only in a 2D view):
  - Click once to set the start point. It is not required to draw points first.
  - Move the mouse to the position of the end point. The line will follow the cursor while you move.
  - Edit □☞ ■ 집書 □ ∽ ~ | 田田田 Ⅲ | ℡ | 図画 2 没 & ※ (A) Fe ≠ || |= || ↓A # # MQQ # / 868 User cor D R H 101 200 Then once here Û Click once AAA 倒感的 80 10 🚖 Ly
  - Click once more to set the end point.

- Though coordinates in 2D/3D view:
  - Enter 2 (or 3) coordinates of the axes that you see on the right hand side below, separated by a semi-colon.

Ex: Front view: XY plane => distance in x-direction; distance in y-direction

Ex: 3D view => distance in x-direction; distance in y-direction; distance in z-direction

 Enter an @sign in front of the coordinates to use relative coordinates.

Ex: Top view: XZ plane => @distance in x- direction; distance in z- direction

- ENTER: End drawing of current line. You will be free to continue drawing elsewhere in the model.
- ESC or 🔄 End drawing function completely.

# 2.4 Selecting

Selected points and lines (without cross section) are represented 'bolder'. Selected lines (having a section) or plates are represented in another colour – by default yellow.

- 1 element: click once on the element
- Multiple elements
  - Adding to the selection: hold the SHIFT-key down while clicking once the other elements.
  - Removing from the selection: hold the SHIFT-key down while clicking once the selected elements.
- Selection window: keep the left mouse button pressed down and move the mouse. The selection window will be drawn with a dotted line.
  - From left to right: only the elements which **are completely** within the selection window will be selected.



• From right to left: all elements that are completely or partially in the selection window will be selected.



While moving the mouse, the selection window will be drawn in a dotted line.

- Elements from the same design type: hold the CTRL key down while clicking on one element of this design type (either one).

## 2.5 Window configuration

Diamonds has 4 standard window configurations with accompanying icon pallet. You can easily switch between configurations by using the pull-down list or the icons in the toolbar.





The **1** button allows you to change the model visualization. Depending on the chosen configuration, you can check different options which will be shown in the configuration.



# 2.6 Types

When you build up a model in Diamonds, you'll notice that each element (such as a beam, a column, a plate, a footing...) will get a certain colour .

This is because Diamonds recognizes a number of 'Design Types' (later just called 'Types').

You'll find all the standard defined 'Types' in Diamonds when you click on the button in the icon bar.



Working with 'Types' has two advantages:

- You maintain **overview** thanks to the colour s.
- You can **select all elements from a type at once**: hold the CTRL key down while clicking on one element of this design type (either one).

To assign a type to an element, select the relevant element(s) and indicate the desired type from the list of types (or define a new type).

## 2.7 Hot keys

- SHIFT
- CTRL
- ALT
- DELETE
- ENTER
- ESC
- CTRL + A
- CTRL + N
- CTRL + O
- CTRL + P
- CTRL + Q
- CTRL + S
- CTRL + Z
- CTRL + C
- SHIFT + CTRL + Z
- SHIFT + CTRL + DELETE
- F1
- F2
- F3
- F9
- F10
- F11
- F12
- SCROLL
- SHIFT + SCROLL down
- Right click in the drawing field
- Double click in the drawing field
- Letter 'a' or 'A'
- Letter 't' or 'T'
- Letter 'r' or 'R'
- Letter 'e' or 'E'
- Letter 'd' or 'D'
- Letter 'm' or 'M'

Select additional elements

• Aid when drawing orthogonal lines Select all elements of the same design type

• Select elements of the same group

 Select plate border together with plate Delete geometry elements or loads (depending on configuration) Draw a new line (in drawing function) End the drawing function (in drawing unction) Select all elements New file Open file Print screen Close Diamonds Save file

- Undo
  - Copy data from tables
  - Redo
  - Delete plate but not the borders
  - Delete a cutline but not the intersection point
  - Open Diamonds Help

Calculate concrete reinforcement quantities Steel and timber verification

- Elastic analysis
- Maximize
- Minimize
  - Show everything
  - Hold scroll down while moving mouse: pan
  - Scroll: zoom in and out
  - Rotate (3D orbit)

Open 'Window configuration' Open 'Grid settings'

Draw a new line Opens the translation window Opens the rotation window Opens the extrude window Opens the divide window Opens the mirror window

# 2.8 Configuration graphical card

When using Diamonds for the first time the configuration of the graphical card is not always optimal. Therefore, we review some common problems and their solutions.

### 2.8.1 Problem 1

Symptoms:

- I don't see the cursor of the mouse.
- When I draw a line, the line doesn't appear immediately but after a few seconds.
- When I select elements, it takes several seconds before I see the selection.

Operating system:

- Windows Vista
- Windows 7 (not on Windows 8 or 10)

Solution:

- Close Diamonds.
- Right click the Diamonds icon on your desktop.
- Click on 'Properties' and go to the tab 'Compatibility'.
- Check the option 'Disable desktop composition'.
- Click 'OK'.
- Restart Diamonds.



### 2.8.2 **Problem 2**

Symptoms:

- When drawing, I see the axes multiple times. It's like Diamonds is stuck.
- Diamonds seems to be reacting slowly.
- I tried the solution of §2.8.1, but the problem is not (completely) solved.

Solution:

- In Diamonds go to 'Options' -> 'Preferences'.
- Click on the tab 'Drawing'.
- Check or uncheck the option 'Graphical input using OpenGL'.
- Click 'OK'.

<del>\</del>	Diamonds - [Window 1 (m)]	- 🗆 🗙
💝 File Edit View Select Display Analysis Op	tions Windows Help	- 8 ×
🗅 📽 🖪   🖾 🚳 🖗   いっつ   田   🛄	Language ・ 変形 & 参加 ④ Fジ ゴ     = ■ +9	
🗑 🗟 🖷 🖂 Window 1 💽 📘	Units And Decimals 🗾 🖗 🖻 🖾 💀 🖉 🖉 🖉 🖉 🖉 🖉	3   🗰 🕅   a
k, X	Preferences	Active level
8 0 0	Preferences	
		A 2.70 m
	File Model Drawing	Level manager
	Show line distances next to the cursor	Cround level
2 m 12	Graphical input using OpenGL	Drawing plane
r <sup>y</sup> n,5 P+1	Fast rendering	X = 0.00 m Y = 2.70 m
NT AT A		Z = 0.00 m
LI O O		
		S .
		Size
Te h Il		Font 10 🚖
E,		Symbols 10 🚖
	Adapt with default parameters	Loads 10 🚖
		Results 10
	<u>H</u> elp <u>Cancel</u> <u>QK</u>	None 🔻
	Ľx	
		1.

### 2.8.3 **Problem 3**

Symptoms:

- I don't see the names of the axes, although the font size is large enough.
- When looking at the loads, I don't see the values, although the font size is large enough.
- When looking at the results, I don't see the values, although the font size is large enough.



#### Solution:

- Go to 'Options' -> 'Preferences'
- Click on the tab 'Drawing'
- Check or uncheck the option 'Fast rendering'.

- Click 'OK'.

🌳 Dia	amonds - [Window 1 (m)]	- 🗆 🗙
Sile Edit View Select Display Analysis Options Windows Help		- 8 ×
D 📽 📓 🗟 ∯ 🗍 ∽ ⇔ 🗍 🕂 🚺 Language 🔶 )	業報 &   ◎ Ⅲ ◎ 1♥ ☞     ⊨ ■ ゆ	
😨 🗟 🖷 🖂 Window 1 💽 📍 Units And Decimals	- M - K - K + K + K + K + K - K - K - K - K	
Preferences		Active level
8 0 0	Preferences	
		FTFF1.# 2.70 m
	File Model Drawing	Evel manager
	Show line distances next to the cursor	Cround level
夕 斑 空	Graphical input using OpenGL	Drawing plane
t <sup>y,7</sup> B, J P ↔	Fast rendering	X = 0.00 m Y = 2.70 m
南北兵		Z = 0.00 m
DT 69 (4)		Representation
		A 40
		Size
		Font 10 🚖
E.		Symbols 10 🚖
	Adapt with default parameters	Loads 10 🚖
		Results 10 🚖
	Help Cancel OK	Show groups
		INone 💌
	L <sub>X</sub>	1
		1.

# 3 Examples in concrete

This classic building consists of two identical apartments on the 1st and 2nd floor and a retail space on the ground level. Unlike the main building, which has a rather introverted character, the ground level has a rather extrovert character with large glass windows. The main building has a basement.



The outer walls of the main building are constructed of load-bearing masonry. To ensure a certain flexibility regarding the layout of the building, no bearing interior walls are provided.

The floor height of the ground level is 3,2m. The apartments and the basement have a height of 2,8m.

There has been opted for a system of preslabs whose bearing direction is defined in the figure below. The preslabs bear from front to back and are hinged imposed on all walls and beams. During the second phase concrete there's no joint foreseen in the middle of the span. The floor of the ground level is poured on site. Between the main building and the development, no moments can be transferred.



The loads on all floors, except the roofs:

- Dead loads: 3kN/m<sup>2</sup>
- Life load: 2kN/m<sup>2</sup>

The loads on the roofs:

- Dead loads: 1kN/m<sup>2</sup>

For all plates, basement walls, beams and column a concrete C25/30 is used. We assume the water level is 1m below the ground level.

The model will be made in different steps:

- We start by calculating the continuous beam on the first floor (§3.1).
- Next we calculate the plate on the first level (§3.2).
- In the next model we dimension the base plate assuming there's no basement (§3.3). We take the results of a soil investigation into account.
- In the next step we make 3D model starting from the floor plate we just calculated (§3.4).
- Finally we model the basement and calculate the complete 3D building (§3.5).

In the first two examples sufficient attention will be spend to the organic calculation (reinforcement, cracked deformation). The last three examples focus rather on the extensive modeling functions of Diamonds.

# 3.1 Example 1: Continuous beam

Required licenses:

✓ 2D Bars✓ Concrete Design

### 3.1.1 Purpose of the exercise

In the first example we model and calculate the continuous beam in the floor of level 1. We calculate the elastic forces in the beam and determine the required reinforcement. We will also calculate the cracked deformation and we will check if cracking width is admissible.



### 3.1.2 Defining the structure

#### Step 1: Go to the 'Geometry' configuration

Defining the structure is always done in the 'Geometry' configuration. Click on B in the icon bar, or select the 'Geometry' configuration in the adjacent pull down menu.

	Diamonds - [Window 1 (m)]																																							
🤗 Ei	ile <u>E</u> di	t <u>V</u> iev	v S	elect	<u>[</u>	<u>)</u> ispla	ay	An	nalysi	is (	2ptic	ons	Wi	ndo	NS	He	p																						-	e ×
D	÷ 1	<b>1</b>   [2	4	5		к?)	Č.		F	1H	m	:	:	Т	. 11	1920		Ŵ	茶	1	13	2ó		( <sup>1</sup> / <sub>2</sub> )	Feg	IÝ	11	1E	肁	10										
	20000 10000	1		- 4º	11						1.	1	•							· •		1 -						11						đ			000000 10			
*	5 7	X	/indo	w 1					-	1		User	cont	figura	ation	Ú					-		<b>m</b>	<b>L</b> H				和	Þ	14	? ∈	્ર	. 00	-		R	ET?		1 152	
			- 28	35 (3		3 18	35	85	23	85 B	1.55	\$3	35	82	St. 1	8 33	82	53	85 I	8.8	- 23	35	82	53	18 B	8 85	<u>\$3</u>	22	8 S	2	35	82	3 8	35 - 82	28 8	\$ 5	5 53	Active	level	
1°p	•		32	8.3		61 30 	- 88	12	- 62	8.8	1.11	32	8	33 	68 I 	12.8	10	- 63	18 I	10	30	8	35	12	32 8	19	- 63	28	10.10	8	- 88	33	11 IX	88 - 88	63.92	100 1	8 6	Level	1	•
	•	1	83	8 8 9 8		2000 2000		40 23	- 52	9 8 8 8	6 58 6 53	- 25	2	ः ११	93 - 1 66 - 1	5 8 5 8	- 22	- 52	100 I 100 I	is is Bi bi		2	32	-22	80 A 80 A	2 - 24 2 - 23	- 22	ж 68	85 86 86 93			28 32	2 H K K	90 10 90 93	- 22 G	1 25 4 1 25 1	• •8 1 98	fort	1 2 70	i m
	-	_																				2	÷.														10	PH-P		
П	Ø	P	- 92	0 8 N 5		64 96 63 96	- 30	22 54	- 10	00 S 26 S	8 100 8 164	- 52	30 36	28 29	194 - 1 82 - 1	0. 20 61 16	122	10	00 1 36 1	n n R G	902 913	30 36		- 22	103 (2) 103 (2)	0 100 0 104	10		08 09 08 06	- 2	- 30	28 29	61 FR 61 FR	30 10 30 10	- 10 C	i se s List s	9 88 9 88	ATTR	2.70	) m
Ā	415		20	86 18		6.3	- 56	84	-	8.9	1.14	26	96	8	-	8.9	85	-		a 14	20	96	12	-35	26 S	6.84	12	32	a 15	-	8	8	8.35	96 SS	S 16	- 22 - 3	s (s)	rt-p		
			- 23	82 B		8 8	32	84	28	92 S	8 82	23	12	84	8	8 3	$\sim$	$\overline{2}$	92 I	8 8	- 23	$\overline{\Omega}$	82	37	28 3	1 12	23	12	8 8	22	5							f	evel ma	inager
		-		8 8			- 35	33	25	8.8	1.12	\$3	3	8	8	8.8	83	35	8	a 23		s	-2	-	8 8	9 89	25	8	0.15	3		14	۲.,		т					
-			-83	10-10		0.8	- 35	$\Im$	$\mathcal{B}$	8.8	6	8	35	88	82.1	8.3	$^{38}$	$\mathcal{B}$	80	8.9	- 83	$^{18}$	88	10	83 38	- 28	30	$(\mathbf{x})$	8.9	- 51	2	15	Y		rop v	lew			Ground	level
14	[†[†]	12	10	82 ×8		S - S	32	22	22	63 X	1 22	4	2	32	5S	5 8	22	38	93 - F	G 33	0	22	12	<u>88</u>	10 S	8 82	22	68	0 33	33	- 8		Zray	(				声D	rawing	plane
ज	<u>er</u>	15 -	- 13	8 S		2 2	35	8	23	10 B	1.5	\$	33	88	32 - 1	8.8	8	23	10 I	8 S	13	3	85	-33	18 B	100	23	20	8 S	2	្រ								0.00	m
Τ.*	R₽Ĵ	1.1	32	8. 33		2.22	28	82	8	15 Q	1.12	32	83	33	- 18	2.8	82	8	98 B	8 (P	32	88	33	-22	32 3	1.55	5	28	18 B	8	- 8	R	V		Front	view	·	Y =	2.70	m
Ň	.ju	G2	- 88	10 10 10 10		0 8 0 8	1.00	- 28	- 82	88.8	6 108 1 103	- 83	8	98 39	85 1 76 1	83 (X) (X) (X)	- 22	- 82 - 92	81	8 8 8 8	- 33	8	100	- 65	93 94 107 5	0 108 - 100	- 42	- 36 - 22	66 128 No 108	- 8		¢T.	z.i.,	È I				Z =	0.00	m
- rup r	24. 1	1-		0. 20				0.04						20			-				- °	и <b>т</b>									1					_		Repres	entatio	m
DI	D	10	- 28	88		28	1.00	8	- 22	100 S 100 S		8	8	58 	汉 : :::::::::::::::::::::::::::::::::::	8.8	8	- 52	8 1 2 1	8 8 9 9	Ę.	53. 	100	- 22	18 3 10 10		- 22	- 20	8 8 9 9	- 2		15	1		Back	iew		æ.		A
20	-Vat.	*34	- 25	N 10		a a	- 94	14	- 20	100 10	1.14	- 25	34	04 10		25 SF	34	- 10	12.1		- 25	34	12	23	25 54	1 14	- 10	100	10 10 10 10	- 22	÷.	-11	Ľ	-	Dack	VIEW		24	<v< td=""><td>SV.</td></v<>	SV.
匣	gene .		28	92 B		8 8	32	84	28	12 8	18	23	12	84	8	8 8	85	78	92 - I	8 8	28	32	82	237	28 3	8 18	23	12	8 8	2			∠ <b>*~</b> •γ	{				A	80	
-	1	1.		85 33			- 95	82	25	8.8				82			- 32				8	35	-	13	8.3			33		2	<b>(</b>	٩.						~	~	
<i>***</i>	, d	V.	- 83	8 3	-	2.8	28	3÷	35	8.8	8 59	8	33	æ	2	8.8	52	33	8	8.9	88	3	88	-2	89. 28	1	35	1	8.8	3	2	13	V		Left v	iew		Size		
7		Te	100	8° 88		8	S.	82	22	83		12	$^{\odot}$	88	65 I	8.8	$\gtrsim 1$	18	93 - B	9.3	10	$^{\odot}$	35	65	s s	2	22	68	9 8	23	- 5		Z.44	í l				Font	1	0 🜲
-43(e		Aller -	- 23	8 8		2 2	12	82	53	8 8	- 33	23	35	23	81.3	8.8	82	28	82.1	s 8	33	35	12	53	18 3	10	23	22	8 B	2	1							Comba	ь <u>Г</u>	
I.			30	8. 33		0.33	8	12	61	18 B	1.12	32	8	13	68 1	12. 23	93	8	18	2.12	3.2	8	3	62	32 3	9.93	- 63	28	2.12	8	- 2	12	2		Right	view		Symbo	15   10	1
1 2 2			- 88	泉 湖			8	*	- 35	(現 8) (二 9)	8 88 	8	8	98 00	88 I 	88	20	- 32	展 :	8.8	93 	8	22	-8	28 E	1.00	- 89	展 : 二	8 8 	- 8	1		J.					Loads	1	0 🜲
			- 42	34 33		61.42	- 34	84	22	25.5	2 22	22	32	24	155 1	22.32	83	10	35		42	24	22	155	45.54	1.52	- 22	35	92 - 334 -	25			<u> </u>	•				Pecult	e 1	
			- 23	8 8		2 8	35	8	- 12	28	8.85	8	35	88	级 : 	8.8	- 55	- 53	8 I			35	100	- 23	18 B 10	100	- 22	*		10		1	-31	D					•   •	· 🔳
			- 25	20 12 12 12		63 - 89 23 - 39	1.25	28	10	10 10	6 28 1 33	- 25	8	98. 101	88 - 1 23 - 1	8 8 8 9	28	10	100	8 34 6 3		20 02	10	10	45 R 26 G	6 28 7 83	10	12	88 28 19 23	10		F	Y		3D vie	w		Show	groups	
			- 28	8 8		8 8	- 22	8	- 28	12 8	1 12	- 23	2	84		8.8	8	- 23	÷.	8 8	- 28	2	84	33	28 8	1 12	- 23	12	8 8	- 2			Zerty	(				None		•
				95 82		2.0	St	se				25	25	82	2.1		æ	38	8	a 94	23	35	32	-	a ==	s	55	22	a	2				95 SP						
			- 83	10 10		0.8	- 38	Se.	- 55	8.8	6 .54	8	18	8	80.1	8.8	38	- 35	81	8.98	- 83	8	iit.	10	83 18		- 81	æ		- 8	38	18 18	8.8	36 .58	- 10 - 10		TX.		34000	
1																																				him				1
	_						_		_		_	_																												

Then check if you are in a front view. If this is not the case, then click on

the button  $\stackrel{\text{result}}{=}$  in the icon bar or on the button  $z \xrightarrow{-x}$  in the lower right corner and select the viewpoint 'Front view'. This way you activate a vertical drawing area.

#### Step 2: Structure generator

You can insert the structure in Diamonds using different ways:

- Draw immediately on the screen with the mouse
- Draw on the screen by means of coordinates with the keyboard.
- Use the structure generator
- Import a DXF-file.

In this example we'll use the structure generator. Drawing a structures will be handled in example 2 §3.2.

Click on the icon in the pallet. A dialog window will appear in which you can select the form of the structure you would like to generate. Opt for a continuous beam.

<del>\$</del>	Diamonds - [Window 1 (m)]		_ 🗆 🗙
💝 Eile Edit View Select Display Analysis 🕻	2ptions <u>W</u> indows <u>H</u> elp		- 8 ×
💥 🙆 🖷 🖂   Window 1 🔄 📘 🧵	User configuration	@   ₽   ₹" ♥ ♥ ♥   /   ●	
🔽 🗸 baa bbaa baa			Active level
			Level 1
			2.70 m
			rin
U C C BRANNARE		2	2.70 m
	Wizard menu		
			Level manager
			Ground level
- Ay (編 2位) エエエエエエエエ			da Desuise elses
PH BREAKS		L L ARAR	X = 0.00 m
. 14 (b)			Z = 0.00 m
<u>na 24 /&gt;</u>			Representation
ot 🔗 🖉			
	$\square \land \land$		
角 🖉 🖻			🚮 🍂
"⊕ / V			Size
			Font 10 🚖
			Symbols 10
			Loads 10 🚖
a at a as at a at a at a			Results 10 🜩
			Show groups
			None
			poore T
			-x 2
			ż
			1.

Next Diamonds will ask you the geometric data of the continuous beam. Enter the details as shown below:

 Set the beam to be drawn on the level Y=2,7m. For the remainder of this exercise, the level doesn't matter. But in the next exercise (§3.2) we will extend this model to a plate and then it's useful the beam is located on the correct level.



- Change the lengths of the fields:

<b>~</b>		Diamonds - [Window	1 (m)]	- • <b>×</b>
🥮 <u>F</u> ile <u>E</u> di	it <u>V</u> iew <u>Select</u> <u>Display</u> <u>Analysis</u>	2ptions <u>W</u> indows <u>H</u> elp		- 8 ×
	Coordinates of first point: X = 0,00 m Y = 2,70 m Z = 0,00 m	Fields         Number of fields:       3 €         Number of fields:       0 €         Field lengths:       1 5.00         m       m	beams     -     ×       Left projection     Projection length:     3,00 m       Number of intermediary nodes:     0     •       Right projection     Projection     9	ievel 1 2.70 m 3 2.70 m wel manager
		2 5,00 m 3 5,00 m	Number of intermediary nodes:     0 \$       Sections and materials     IPE 300 - \$235       Projections     IPE 300 - \$235	round level awing plane 0.00 m 2.70 m 0.00 m entation
	· <u>A</u>	Δ	<u>∆</u>	x 10 € 10 € 10 € 10 € 10 € 10 € 10 €
			Cancel OK	

- Set the lengths of the cantilevers to 0,00m.

	Diamonds - [Window 1 (m)]	
🤗 <u>F</u> ile	Edit <u>V</u> iew <u>S</u> elect <u>D</u> isplay <u>A</u> nalysis <u>O</u> ptions <u>W</u> indows <u>H</u> elp	- 6 ×
D 🛩	Image: Wizerd: Continuous beams	
	Coordinates of first point:       Fields         X = 0,00       m         Y = 2,70       m         Z = 0,00       m         Field lengths:       Image: Space state stat	Lo      L     Co     L     Co     L     Co     L     Co     L     Co     L     Co     C
	Cancel OK	▼           

- Change the cross section of the beam to a rectangle 250x400mm.
  - Click on the button Fields . You will then see the following window:

Section	×	- D ×
Cross-section Name R250/400  Shape Constant Constant		· 문 · · · · · · · · · · · · · · · · · ·
Dimensions       B     250     mm       H     400     mm	ensions   Axes	Left projection Projection length: 0,00 m Number of intermediary nodes: 0 ♀ Right projection Projection length: 0,00 m Number of intermediary nodes: 0 ♀ Sections and materials
Automatic calculation General Elastic Plastic		Fields         IPE 300 - S235           Projections         IPE 300 - S235
Sy         20000000         mm³         Sz         125000000         mm³           Iy'         133333333         mm4         Iz'         52083333         mm4           iy'         115,5         mm         iz'         72,2         mm           Wel,y',t         6666667         mm³         Wel,z',t         4166667         mm³           Wel,y',b         6666667         mm³         Wel,z',t         4166667         mm³	alpha 0,0 ° Iyz 0 mm4 It 1273451646 mm4 Iw 0 mm6 Twm 6060606 mm3	Δ Δ
		Cancel OK
Material     Reinforcement       Local axes     0rientation angle		
Help	Cancel OK	

- Choose a rectangular shape.
- Enter the dimensions of the section (B250xH400mm).
- Select the correct material (Concrete C25/30).

Then click twice on 'OK' the draw the structure. You'll see the image below. In the image below we opted for a wireframe representation  $^{\textcircled{O}}$ . But you could also opt for a solid representation  $^{\textcircled{O}}$ .

<del></del>							Di	amono	is - B	etonV	B1.bsf -	[Venster	1 (n	n)]						- 🗆 🗙
🂝 <u>F</u> ile	<u>E</u> dit ⊻ie	w <u>S</u> elec	t <u>D</u> ispla	ay A	nalysis <u>O</u>	ptions <u>V</u>	Vindows	Help			1									- 8 ×
0 🗳		1 🕹 🚺		Cit.	H H	⊞∣₩			a v i	be b	8: <u>1</u>	5 🕒 F% 🗆	6	■利	ゆ					
80		enster 1			- 1	User co	nfiguratio	n			• B	F# 🕅 🗖	3	M D	<">	. 🔍 🖾		DR		
<b>k</b> <sub>p</sub>				24											•				<u>.</u>	Active level Verdieping 1
 		-0	89 <b>-</b>	83	8	Ξ.	89	35	-22	i).	8	25	8	83	-0	3÷	53 -	8	8	2.70 m
		-12	12	14	8	25	32	8	52	8	12	25	25	32	10	12	35	55	81	2.70 m
	¥ 	53	8	8	3	*	28	10	53	8	8	25	8	18	33	15	8	8		Level manager
14 H	1 21	27	15	2	22	12	-	25	10	8	2	22	12	-	57	12	8	8	82	Ground level
т <sup>7</sup> в.	5PE	10	84	17	2	68	101	18	10	8	13	<i>w</i>	8	10	61	51	81	a.	ŵ	X = 0.00 m Y = 2.70 m
NP1 2	1 /2		1										1					Δ		Representation
lt (	2	-83	15	15	94	12	35	51	-15	10	14	96	8	20	-25	14	8	22	86	**
m s	Fe (	10	Y	8	2	31	28	×.	85	15	8	38	8	83	10	3	93	8	18	D
		-	L	Х	a	15	30	8	52	8	it.	a	8	22	53	87	85	10	a	Size Font 25 🚖
E)		14	87	a,	8	8	101	8	ţs.	8	12	35	8	43	10	87	82	2	35	Symbols 10 🜩
		8	8	2	35	痰	*	20	53	82	2	35	2	18	3	15	82	8	8	Results 20 🔶
		27	8.	8	2	12	28	23	11	8	8	12	12	18	27	12	84	8	2	Show groups
		48	12	17		68	10	10	10	8	13	<i>w</i>	8	¢	ki.	84	38		YL <sub>X</sub>	
																		1		11.

There, the structure is completely defined. Now we define the loads.

### 3.1.3 Defining the loads

Step 3: Go to the 'Loads' configuration

We now leave the 'Geometry' configuration and activate the 'Loads' configuration to enter the loads. Click on the button  $\square$  in the icon bar or select in the adjacent pull down menu the 'Loads' configuration.



#### About the 'Loads' configuration

With the 'Loads' configuration windows comes a separate pallet containing all the functions for defining loads and generating combinations. Note that the point of view remains unchanged when switching between the configurations.

### 3.1.3.1 Creating the load groups

#### Step 4: Creating load groups

Before defining any loads, you have to make the different load groups. Click on the button  $\gamma_g^{\gamma_u}$ . You'll see the following screen:

Image: Second Secon	💝 Eile Edit View Select Display											
$\begin{array}{c c c c c c c c c c c c c c c c c c c $												
Image: Service 1       Image: Service 1 <t< td=""><td></td></t<>												
Active level       Active level         Selfneight       Image: selfneight <th< td=""><td>🛞 🗟 🖷 🔀 Venster 1</td></th<>	🛞 🗟 🖷 🔀 Venster 1											
$\begin{array}{c c c c c c c c c c c c c c c c c c c $	yg <sup>y</sup> u d1± ⊙ tm											
Load factors for       EN 1990       Image: Consequence dass Design lifetime       2       Service dass       1       Image: Consequence dass Design lifetime       2       Image: Consequence dass Design lifetime       1       Image: Consequence dass Design lifetime       2       Image: Consequence dass Design lifetime       1       Image: Consequence dass Design lifetime       2       Image: Consequence dass Design lifetime       1       Image: Consequence dass Design lifetime       2       Image: Consequence dass Design lifetime       1       Image: Consequence dass Design lifetime       2       Image: Consequence dass Design lifetime       1	Load groups – – ×											
Add load group       Insert load group       Delete load group       Delete load group       Several load cases per group         Name load group $\gamma_{uls-}$ $\gamma_{uls+}$ $\gamma_{sls-}$ $\gamma_{sls+}$ $\psi_0$ $\psi_1$ $\psi_2$ $\varphi$ $\xi$ $t_o$ Combination for cracking $k_{mod}$ Load       Action $\checkmark$ Selfweight       1.35       1.00       1.00       1.00       1.00       1.00       0.85       0       permanent       - $4 \pm 1 \pm $	Load factors for EN 1990  Consequence dass 2  Service dass 1											
$ \begin{array}{ c c c c c c c c c c c c c c c c c c c$	Add load group Insert load group											
✓         Selfweight         1.35         1.00         1.00         1.00         1.00         1.00         1.00         0.85         0         permanent         —         ±±           ✓         dead loads         1.35         1.00         1.00         1.00         1.00         1.00         1.00         0.85         0         permanent         —         ±±           ✓         live loads A : housing         1.50         0.00         1.00         0.50         0.30         1.00         1.00         0         medium term         —         ±±	Name load group $\gamma_{uls}$ - $\gamma_{uls}$											
✓       dead loads       1.35       1.00       1.00       1.00       1.00       1.00       1.00       0.85       0       permanent       —       ±±         ✓       live loads A : housing       1.50       0.00       1.00       0.70       0.50       0.30       1.00       0       medium term       —       ±±												
✓         live loads A : housing         1.50         0.00         1.00         0.70         0.50         0.30         1.00         0         medium term         —         ⊥⊥	✓ Selfweight											
	✓     Selfweight       ✓     dead loads											
	✓     Selfweight       ✓     dead loads       ✓     live loads A : housing											
Incompatible load groups Linked load groups Combine loadcases Ungroup loadcases	✓     Selfweight       ✓     dead loads       ✓     live loads A : housing											
Help Cancel QK												
	✓     Selfweight       ✓     dead loads       ✓     live loads A : housing       Incompatible load groups											

#### About the window 'Load groups'

- In the menu on top you select to which standard the safety and combinations factors should answer. Currently this is set to 'EN 1990 [--]' which means Eurocode 0 without a national annex.
- In some national annexes the safety coefficients also depend on the consequence class and the design lifetime of the structure. Both are linked to the economic and/ or social interest of the structure. A higher/ longer consequence class/ design lifetime will lead to higher safety factors.
- On the right you can enter **the climate class**. This climate class is representative for a certain moisture content of the air/ the timber. Diamonds uses the climate class you determine the modification factor  $k_{mod}$ . The modification factor  $k_{mod}$  takes the influence of the load duration and the moisture content on the strength properties into account. The modification factor  $k_{mod}$  depends not only on the climate class but also on the type of timber and the load duration class. The **load duration class** must be specified for each load case in the last column.
- In the table below the load cases 'Self-weight', 'Dead load' and 'Life load' are defined by default. You can freely rename or delete them, except for 'Self-weight'. The fill-in boxes to the right of the name of each load case include the safety *γ* and combination factors *Ψ* required for the automatic generation of the load combinations.
- We don't discuss the other parameters in this window.

To keep the example simple, leave the parameters untouched as in the figure above. Then click the 'OK' button.

### 3.1.3.2 Filling up the load groups

Now the loads groups are defined, we can assign loads to the structure.

For simplicity, we assume that the beam carries 50% of the loads on the floors. The other 50% is distributed between the front and back façade. So the beam bears 5m width of the plates.

Step 5: Filling in the load groups 'Self-weight', 'Dead loads' and 'Life load'

- The **self-weight** of the beams is calculated automatically by Diamonds and cannot be adjusted.

9			Dia	amonds	- Betor	VB1.bsf -	[Venst	er 1 -	Selfw	eight	- (kN, kN	m, mn	n, kN/m,	kNm/i	n, <mark>kN/</mark> m	², °C)]			- 🗆 🗙
🥮 <u>F</u> ile <u>E</u> dit	<u>V</u> iew <u>S</u> e	lect	<u>D</u> isplay	Analysis	Options	Windows	Help												_ <i>6</i> ×
0 🗳 📱	0.4		00		田田二	#   <b>T</b>		B X I	be b	08	221 (B) F%	IÝ	厚厚	49					
808	🖄 Venster	1		•	Loa	ls				•	🚯 💷 🔯		,用口	ংশ্য ভ	2 🗨 🖬		DR		
y <sup>y</sup> u mit	<u>د</u>			1												-94	18		Active level
Selfweight	-	18	16		æ	8	31	-2	<i>3</i> 2	8		8	83	-0	<u>8</u> 4	<i>3</i> 1	8	з	Verdieping 1
12 4		12	12	8	98	30	8	52	3	38	22	25	32	-	12	33	88	8	2.70 m
i i,		8					12	33				8	8	33	8				Evel manager
+ ~ .				-	2.5			-		2.5				- 7	5	- iii			Ground level
「イン」			14	1		- 22		35	88		<u>84</u>		10	33 <sup>-</sup>		84		22	Drawing plane
S.	27		10		6	1		14	8				10	61				æ	X = 0.00 m Y = 2.70 m
* Temperature	•	1		¥.	<b>W</b>	¥	Ť	Ŷ			¥	Ť	¥		¥	¥	Ă		Z = 0.00 m
Wind     Seismic	-	85	14	S.	22	20	87	-33	8	14	96	8	20	-33	85	8	15	8	
<ul> <li>Dynamic</li> <li>Moving</li> </ul>		Y	<i>.</i> .	×	31	83	æ	ĸ	95	8	12	×	8	80	8	93	8	×	D .
			→ X	( *	18	20	8	12	8	10	8	8	50	-	12	8		е 8	Size Font 25
	10						15	12	82		25		85	10	121	a.		25	Symbols 10 🗢
																			Loads 10
	10	85	8	33	82	18	22	8	12	8	<u>88</u>	22	88	53	8	82	8	*	Results 20 🚖
																			Show groups
																			None
	10	83	13	2	98	41	33	15	33	8	ŵ	8	10	85	8	28	a	Ľx	

- The **dead loads** on the beams are determined by the self-weight of the floor  $(4,5kN/m^2 \cdot 5m = 22,5kN/m)$  and the dead loads on it  $(3,0kN/m^2 \cdot 5m = 15kN/m)$ . Total dead loads: 37,5kN/m
  - Use the pull down-menu to activate the load group 'Dead loads'.
  - Now select the beams and click on the button . Note that only those icons will be active that can be applied on the selected elements.
  - Complete the window as follows:

<del>ô</del>	Diamonds - BetonVB1.bsf - [Ve	enster 1 - dead loads - (kN, kNm, mm, kN/m, kNm/m, kN/m², °C)] – 🗆 🗙
<u> </u>	lisplay <u>A</u> nalysis <u>O</u> ptions <u>W</u> indows <u>H</u> e	elp – e ×
	<u> ゃ ~   冊冊</u>	थ
😨 🗟 🖷 🔀 Venster 1	Loads	- 📾 Fi 🖾 🚽 🖉 Q. Q. 🖾 🥒 🔟 🖾 🖾 🗉
y <sup>y</sup> mit O		Active level
dead loads		
·		Force on line
NO ST Y		
Vrig T		
+ 10 50		start value 0.0 kN/m
€ 2 œ		x end value 0.0 kN/m
		Z <sub>4</sub> start value 27.5 kV/m
* Temperature		kN/m and value 37,5 kN/m
* Wind		z start value 0.0 kN/m
▼ Seismic		
Moving		x end value 0.0 kiv/m
		G f the Barlength = 5.00 m (first bar selected)
24 AL CA.		Physical element length = 5.00 m (first bar selected)
2 8 2		distance from begin 0.00 m
- K G		distance from end
1 N N		
A 16 3		values and distances related to group 'Loads'
9 8 86	N N 12 N N N N N	Help <u>Cancel OK</u>

In the 'Loads' configuration window the entered loads are graphically represented.



Using the image above, verify if you've defined the load correctly. If you made a mistake, you could:

- double click the relevant element and change the value of the loads in the window that appears.
- OR select the elements and delete the loads with <sup>1</sup>/<sub>K</sub>. Define the correct load afterwards.

- The life load on the beams is determined as  $2,0kN/m^2 \cdot 5m =$ 10.0kN/m.
  - Now select the load group 'Life load' from the pull down-menu.
  - Define a life load of 10kN/m.

### 3.1.3.3 Making combinations

Before starting the calculations we need to generate the combinations first.

#### **Step 6: Making combinations**

<u>11</u>

Click on the button in the pallet in the 'Loads' configuration window A dialog box appears with an empty list of combinations.

Click on the button <sup># Generate all combinations automatically</sup>. indicate in the pull-down menu that you wish to use the classic but conservative Eq. 6.10 and select all limit states.



After pressing 'OK', all the combinations required by the standard will show, grouped by limit state. If desired, you can change these combinations Modify a combination or define combinations yourself Create a combination. One combination can be deleted with Delete a combination. To delete all combinations click Delete all combinations

<b>?</b>	Loads combinations – 🗖 🗙												
	Image: Senerate all combinations automatically     Image: Create a combination												
ULS	ULS FC (ultimate limit state - fundamental combination)												
Loa	Load group												
	JLS FC (ultimate limit state - fundamental combination)												
ULS	JLS SC (ultimate limit state - seismic combination) JLS FI (ultimate limit state - fire)												
SLS	SLS RC (serviceability limit state - rare combination)												
I SLS	SLS FC (serviceability limit state - frequent combination)												
4	4 ULS FC 4 1.00 x 1.00 1.00 x 1.35 0.00												
5	5 ULS FC 5 1.00 x 1.35 1.00 x 1.00 1.00 x 1.50												
6	6 ULS FC 6 1.00 x 1.35 1.00 x 1.00 0.00												
7	7 ULS FC 7 1.00 x 1.00 1.00 x 1.00 1.00 x 1.50												
8	8 ULS FC 8 1.00 x 1.00 1.00 x 1.00 0.00												
	Modify a combination Delete a combination Delete all combinations												
	Find the second												
ļ	Help <u>C</u> ancel <u>O</u> K												

Click 'OK' to close the window with the load combinations.

The names of the different load combinations are now listed in the pull down menu of the pallet 'Loads'. When you select one of these combinations, then the whole of the loads that will act during this combination will be shown.



## 3.1.4 Generating the mesh

#### Step 7: Generating the mesh

Click on the button in the icon bar or select the menu instruction 'Analysis – Mesh'. Leave everything on default and hit 'OK'.

	Mesh creation	×
Bars Number of divisions Minimum element size		3 <b>≑</b> 0.00 m
Help	<u>C</u> ancel	<u>o</u> ĸ

#### About the mesh generator

Here you enter the number of elements in which a bar should be divided: 8 division is a good value.

Meer information on our support website:

http://buildsoftsupport.com/knowledge-base/how-to-pick-the-mesh-size/

### 3.1.5 The global elastic analysis

#### Step 8: Elastic analysis

To start the analysis, select the command 'Analysis – Elastic Analysis'. You can also start the analysis directly using the function key **F9** or use the icon  $\blacksquare$  on the icon bar. Following dialog box appears:

) 🗟 🖷 🔀   Venste	
y" tit 😞	Analysis settings
5 QP 1 ▼	Structural Soil Dynamic , PAI 2
2 - X	▲ Analysis
المتح مل _	
	C 2nd order analysis Precision for normal forces in bars: max. number of second order iterations: 10 ↓ Y = 2 Y = 2
Snow Wind Seismic Dynamic Moving	max. number of iterations for nonlinearities: 50 文 Represent
	▼ Joints
	▼ Concrete Cracking
	▼ Timber Loads

We choose a first order calculation and confirm with 'OK'. A dialog box displays the progress of the calculation.

Performing analysis
Static analysis
Solving combinations
Combination 17

The button ellows you to stop the calculation. If you stop the calculation, you'll have to completely restart it later.

#### Step 9: Go to the 'Results' configuration

To see the results of the calculations, you click on  $\square$  in the icon bar or select in the adjacent pull down menu the 'Results' configuration.

1	Results	R	<b>L</b> II	Ж	-
	Geometry Loads				
	Results				
	Mesh	]			

#### About the 'Results' configuration

In the corresponding pallet on the left side of the model window, you'll see several buttons, each representing a specific group of results.

• Only those buttons for which a calculation is carried out are available.

- Once one of these buttons is pressed, the corresponding partial results can be retrieved through buttons located below.
- Then you indicate for which load combination you wish to see the results. In a first pull-down menu, select the type of load combination (individual load group, ULS FC, ULS SC, SLS RC, SLS FC or SLS QP), then specify which load group or load combination must be shown. In the case of a load combination you can choose between either an individual load combination (indicated by a number) or the envelope. In those situations where the result suggests an envelope, it may be possible that for some results you can opt for the minimum (min) or maximum (max) results to be displayed.

Below, we list some results.

#### **Step 10: Deformation**

By default Diamonds will first show you the vertical deformation in de Ydirection for the first combination (or the first load group when you also made the combinations for the load groups – which is the case in this

example). You'll notice that the button for viewing the displacements  $I_{\text{sr}}$  is active. Below it, the button for vertical displacements according to the global Y-axis  $\P$  is active.



Now select the combination group 'SLS QP' and choose for the envelope of the results. We notice that:

- The maximum deflection is 4,3mm. Pay attention! This deflection is an elastic deflection! The cracked deformation after creep can be 3 to 5 times lager!

- In all combinations SLS QP and on each position of the beam, Diamonds will look for the minimum value of the deflection. Those values are represented by the **thin line**.

In all combinations SLS QP and on each position of the beam, Diamonds will look for the maximum value of the deflection.

Those values are represented by the **thick line**.

Hence this image is called an 'envelope'.



Note that you can arrange the size of the results representation using the pallet 'Size' located on the right-hand side of the model window.

#### About the scale

A symmetrical colour scale is by default used for all results. However, you may choose a different scale in the 'Results' configuration **1**.

You should understand the default scale as follows: the limits of the colour pallet correspond to the largest positive OR negative value. The colour scale runs from -4,3 to +4,3mm. However, the largest or smallest value is displayed above and below the scale. Consequently, for this example, only the lower half of the colour pallet is used.

#### Step 11: Deformation in the detailed window

Now select all beams. Click on the icon  $\blacksquare$  on the right of the icon bar to ask a detailed result. A new window will open. On the left you will find all
the buttons of the 'Results' configuration applicable for line elements (beams and columns).



Note that in this window the deformations are given according to the local axes of the bars. Also the angular rotations  $\varphi_{x'}$  (round the local x'-axis) is displayed.

#### Step 12: Stress in the detailed window (bar level)

Now show the elastic stresses (based on the cross section in concrete!) for the combination ULS FC envelope.



You can retrieve the results at any position using the slider. Moreover, you can also enter a distance. Consult for example the results on 2,45m. Enter '2,45' under the white arrow.

With a combination envelope the determining combination appears. You can disable this by clicking once on the button  $\square$ , this will change in  $\square$ .

Click 'OK' to close this window.

#### Step 13: Stress in the detailed window (section level)

Now show the elastic stresses in the cross section for the combination ULS FC1.

- Select the first beam and click on the icon <a>[</a> on the right of the toolbar.
- Or double click the first beam.

This window opens:



You can retrieve the results at any position using the slider. Moreover, you can also enter a distance. Consult for example the results on 2,50m. Enter '2,50' under the white arrow.

Move the mouse over the cross section to see the stresses at any position.

#### About detailed results on cross section level

Choose on the left top for which **load group** or load combination you would like to see the stresses.

With the **slider** on top of the window you can set the section for which you would like to see a detail of the stresses. By clicking on the distance below the slider, you can enter a position of your choice.

#### Results field with scale:

- In the results field the selected profile is graphically represented together with its principal axes of inertia. When a cross section is double symmetric, these axes will coincide with the local axes.
- On the principal axes you'll see red points. These are the points for which the stress results  $(N + M_y \text{ and } N + M_z)$  are presented in the global results window of Diamonds. The position of these points is determined as the intersection of the principal axes with the cross section's bounding box.
- When you come near these red points with the cursor, Diamonds will snap to them.
- Move the mouse over the section to see the stresses at the desired place. Enter 'x' and 'y' coordinates to show the stresses at a point of your choice. The stresses you find in this window are based on  $N + M_y + M_z$
- Compression is negative, tension is positive.

Click 'OK' to close this window.

#### Step 14: Bending moments $M_{y}$

Now we visualize the bending moments  $M_y$ . Click on  $\mathbb{N}_y^{\mathbb{R}}$  in the pallet and select the bending moment  $M_y$ . Choose the combination ULS FC envelope.



The moment diagram is always displayed on the tension side of the element. The sign of the bending moment corresponds to direction of the local axes. In this case the local z'-axis is directed upwards and causes a positive moment thus tension on the upper side.

#### Step 15: Reactions

Once back in the model window, we click on the button fine B in the pallet to show the reactions. All reactions are displayed separately by Diamonds. In this example we are interested in the vertical node reactions in the combination 'SLS RC': we select the support reactions  $fine B_{Y}$ .

<b>%</b>				D	liamor	nds - E	Beton	VB1.bsf	- [Win	dow 1	- Rea	action	Ry on po	int -	SLS R	C Envelo	pe N	Max (kN	J)]			- D X
🤗 <u>F</u> ile	<u>E</u> dit <u>V</u>	iew S	elect [	isplay	Analysi	s <u>Optic</u>	ons <u>V</u>	Vindows	<u>H</u> elp						e							- 8 ×
	¥ 📕	D. #		5 0	FF	ΠĦ	:::	Tii		I X K	1 6	8	🖂 🕒 Fe/	IÝ	ᆌ	歐椅						Dorien 🔻 📕 🖵
× 5		Window	v 1		•	1	Results					•	<b>%</b> 🕶 🗵		88	\$	Q	۹ 🖬	Ø	D	R 🚰 🛛	
1.7			r.	2	-81	8	68	13	21	e.		-	21	<sup>1</sup>	8	22			10	max 275.0	= 275.0	Active level
1.00	140 PA		10	*	83	52	<u>8</u> 2	8	8	8	35	1	13	2	8	15	18			206.2 137.5		2.70 m
Į			32	8	23	8	28	12	12	32	83	12	81	25	×	27	30	8		68.7		Level manager
SLS RC	pe		83		93	32	(2)	8	8	8	52	82	-55	8	8	22	8		98	0.0 · 0.0 ·		Ground level
Min	Max	1	f	1	-	25	12	ł	15	8	12	10	ł	10	2	12	8	4		-68.7 -137.5	+ - +	X = 0.00  m Y = 2.70  m
Rayz	T Ry T Rz		100	.0		2	8		3	2 1	8					3	0	100	.0	-206.2		Z = 0.00 m Representation
Ra	Ry Rz		25	8			12		89	26	s			10	ä	32	20			-275.0 ·	= 100.0	
Rayz	<b>R 1</b>		18		82	12		275.	0	2		82	275.0	<b>)</b> .	8	12	28		12	8		Size
Site e	ny Chz	ł .	32	8	8	8	25	12	1	22	8	13	11	25	22	88	30		3	8		Font   25     Symbols   10
Mayz Mayz Ma	Hy Hz	<b>e</b>	23	я	ах	ĸ	30	8	8	8	ж	а,	85	ж	8	38	8		8	10	×	Loads 10 🜩 Results 20 🜩
k	<u> </u>	- 2	17	2	88	8	6	13	21	τ.	8		51	8	8	22	2	2	2	<i>li</i>	8	Show groups
	Ŕ		0	3	12	25	8	2	39	¢:	*	8	84	8	0	89	2		8	8	Y L <sub>X</sub>	
																						11.

So far the overview of the functionalities in the 'Results' configuration.

# 3.1.6 Calculating the reinforcement

#### About dimensioning reinforced concrete

With Diamonds you calculate the required longitudinal reinforcement in beams, columns, plates and walls. For beams and columns the required transverse reinforcement is additionally determined. There is no shear verification for plate elements! Punching shear reinforcement is automatically calculated for footings, but for plates this calculation has to be started manually!

Before starting the calculation, we check the properties of the used concrete and reinforcement steel.

Choose the menu instruction 'Edit – Material library' and select the material 'Concrete C25/30' from the left column. Concrete C25/30 is a default material.

<b>?</b>	Material library	
Er      L	Name     C25/30       Material type     concrete       Mechanical properties     Thermal properties       Advanced     Image: Concrete       Mechanical properties     Thermal properties       Advanced     Image: Concrete       Voung's modulus     31476       Poisson ratio     D.2       Transverse Young's modulus     13115       Therm. dilatation coeff.     0.000010       Density     2548.4	

Default materials are marked by #.

The properties of default materials *determined* using the standards and can't be edited. Should you wish to make adjustments, you'd have to make a new material. User defined materials are marked with the icon **1**.

On the right of this window you'll find:

- The **mechanical properties**: the Young's modulus, the (transverse) Poisson ratio, the thermal dilatation coefficient and the density.
- The thermal properties used in a fire analysis.
- The strength properties in the tab page 'Advanced'. In particular we review the properties that apply for 'Eurocode 2: EN 1992-1-1 [--]'.

In the first tab we find **the properties of the concrete**: the characteristic compressive cylinder strength  $f_{ck}$ , the tensile strength  $f_{ctk}$ , the partial safety factor  $\gamma_c$  and the creep factors  $\varphi$  are displayed. Also the limits to which the stresses are limited are given.

🗈 🕼 🎝 t² 🍸 🛨 🖃 🗈	
Search	Name C25/30
Concrete C12/15     Concrete C16/20     Concrete C20/25	Material type concrete
Concrete C30/37     Concrete C35/45     Concrete C40/50     Concrete C40/50     Concrete C45/55     Concrete M15 (D5)     Concrete M30 (D5)     Concr	Image: Teurocode 2 : EN 1992-1-1       Image: Teurocode 2 : EN 1992-1-1         Concrete       Reinforcement         Compr. strength fck       25.0         Tension strength fck       2.6         Y <sub>c</sub> 1.5         Creep factor O(t, to) for stress limit       1.4         Creep factor O(t, to) for deformation       2.0         If Maximum allowable compressive stress for rare combinations       15.0         If Maximum allowable compressive stress for quasi-permanent combinations         II.3       N/mm <sup>2</sup> IV after creep       Im 0.6 fck

The **properties of the reinforcement steel** can be found in the second tab: the steel quality of the longitudinal  $f_{yk}$  and transverse reinforcement  $f_{ywk}$ , the partial safety factor  $\gamma_s$ , the stress limit and the minimum/maximum reinforcement ratio.

Na	C3E(20	
Concrete C16/20     Concrete C20/25     Concrete C20/25     Concrete C20/25     Concrete C30/37     Concrete C30/37     Concrete C40/50     Concrete C40/50     Concrete C45/55     Concrete C45/55     Concrete K15(15)     Concrete M25(15)     Concrete M25(15)     Concrete M40(15)     Concrete M45(15)     Concrete M45(	inial type       concrete         hanical properties       Thermal properties         é       Eurocode 2 : EN 1992-1-1         i       Image: Second se	

Now click on 'OK' to close the material library.

#### Step 16: Choosing the concrete standard

Now select the menu instruction 'Analysis – Concrete standard' and indicate you wish to calculate the reinforcement using the European standard EN 1992-1-1. We don't use a national annex [--].



#### Step 17: Calculating the reinforcement

Next choose the menu instruction 'Analysis – Calculate reinforcement', press **F2** or click on the button B in the icon bar. A window shows you the progress of the calculations.

Once the calculation has ended, the button for showing the reinforcement results will become active. The reinforcement results for line and surface elements (e.g. beams, columns, walls and plates) are

represented by the same button . As with all the other results, you select the desired reinforcement below in the 'Results' pallet. Note that this time you can't select a combination. The reinforcement is determined using the envelopes of the different limit states.

Visualize the longitudinal reinforcement in the beams: select the longitudinal reinforcement  $A_y$ . The longitudinal reinforcement  $A_y$  is the reinforcement needed to withstand the bending moments  $M_y$  (bending around the strong axis).



The calculated reinforcement corresponds with the optimal (most economic) reinforcement.

The amount of reinforcement is always drawn on the side where it is needed.

The **thin line** represents the amount of reinforcement needed to meet the ultimate limit state ULS only.

The **thick line** represents the amount of reinforcement needed to meet the ultimate limit state ULS AND the service limit states SLS (stress, minimum required amounts and buckling).

In places where the thicker line is far away from the thinner line, the service limit states SLS are important. This points to a slender beam with a height smaller than the optimum height. The three other results are the longitudinal reinforcement for bending around the weak axis  $A_z$  and the transverse reinforcement for bending around the strong  $A_{wz}$ , resp. weak axis  $A_{wy}$ .



#### Step 18: Reinforcement results in a table

Select the centre span and then click the icon  $\mathbb{R}$ . The reinforcement's results for the centre span are now represented in a table. You will find the 4 required longitudinal reinforcements (top, bottom, left and right) and the transverse reinforcement according to the two main directions ( $A_{wy}$  and  $A_{wz}$ ), for each mesh point of the bar.

🥹 Dia	nonds - BetonVB1.bsf - [Venster 1 - Awz in ba	ar (mm²/m)]	- 🗆 🗙											
Eile Edit View Select Display Analysis Options	<u> M</u> indows <u>H</u> elp	191	- 8 ×											
□ 🛎 📕 🖪 🧶 🖗 🗍 ∾ ∾ 🛛 🖽 🖽 🖽 🗰	🔚 🖾 🖩 X 🌬 ठ 😤 🎞 🕒 🕬 🕬	事で												
😨 🗟 🗮 🛛 Venster 1 💽 🚺 Result	• 🔹 🖌 🕅 🗹	那 🛱 🖑 Q Q 📓 🥒 🖪 🗷												
<b>1.7</b> 0 0 0 0	370	max = 870	Active level											
580 580	580.	Results -												
Ay Az Awz Awy Az Awy (mm <sup>2</sup> ) (mm <sup>2</sup> /m) (mm <sup>2</sup> /m)														
	position	(mm <sup>2</sup> ) (mm <sup>2</sup> ) (mm <sup>2</sup> /m) (mm	2/m)											
	* * * * * * * *	(max) (max) (max) (max)	ax)											
A9 XA2		sup. =1819 right =0												
	2 - 1 (0.00 m)	inf. =728 left =0 725 (	)											
No ceres ceres		sup. =536 right =0												
	2 - 2 (0.63 m)	inf. =150 left =0 543 0	)											
		sup. =150 right =0												
NE CAREARES	2 - 3 (1.25 m)	inf. =150 left =0 362 0	)											
<b>Арм</b>		sup. =150 right =0												
▶ 🖉 💥	2 - 4 (1.88 m)	inf. =195 left =0	)											
		sup. =150 right =0												
🧑 👻 ся в кал в кал в	жажыса 2 - 5 (2.50 m)	inf. =287 left =0	)											
			- ·											
			~											
	akaaska <u>H</u> eip													

# 3.1.7 Calculating the cracked deformation

The deflection that you have calculated so far is based on elastic material properties as defined in the material library. Nevertheless, in practice the deflection of structures in reinforced concrete depends highly on the extent to which the concrete is cracked.

The cracking occurs when the bending moment in the rare combinations exceeds the cracking moment  $M_r$ . The cracking moment  $M_r$  depends on the tensile strength of the concrete and the amount of reinforcement placed.

If you enter no further specifications, Diamonds will assume that in practice just as much reinforcement is placed as calculated by the program (or the minimum amount imposed by the standard). But you can also force Diamonds to the take the provided reinforcement into account by defining practical reinforcement.

#### **Step 19: Assigning practical reinforcement**

Visualize one of the four reinforcements results on beams. Select all beams using the CTRL-key and press the right button of the mouse once.

<del>\$</del>	Diamonds - BetonVB1.bsf - [Window 1 - Ay in b	par (mm²)]	- 🗆 ×
🂝 Eile Edit View Select Display Analysis Optic	ons <u>W</u> indows <u>H</u> elp		_ 8 ×
□ 🖆 🖪 🖪 🖉 🗍 느୦ ୦୦ 🛛 🖽 🖽 🖽	ⅲ   № 🖩 ¥ № ৪ 😤 竝 9 № 🕫	平會を	Dorien 🔻 📕 🖵
🗑 🐻 🗮 🛛 Window 1 💽 🚺	Results 🗾 🖗 🖽 🖾 🗟	# b ?QQ / D .	
<b>7.7</b> DU 20		max = 18	Active level
File Wo Dea		1819	2.70 m
150 I = 0		1304 ↔ - 150 909 ↔	2.70 m
			PH-1'
.150	728	150	
12	120	1207 0 +	Cround level
	Practical Reinfor	cement Quantities	Ng plane
			70 m
Awy	462 mm <sup>2</sup>	0 mm <sup>2</sup>	ation
Ang Azz	Max calculated : 1819 mm <sup>2</sup>	Max calculated : 0 mm <sup>2</sup>	38
× × ×	Number of bars : 3 🚖	Number of bars : 2 🚖	<b>1</b>
N a a a a a	,		
AN	1256 mm <sup>2</sup>	0 mm <sup>2</sup>	25 🚖
★ A X	Max calculated : 1207 mm <sup>2</sup>	Max calculated : 0 mm <sup>2</sup>	10 🚖
	Number of bars : 4	Number of bars : 2	10 🜲
🔊 👻 a a a a			20 🜲
	Help	Cancel	
38 108 40 100 0	8 45 93 38 58 48 46 58 3		-* <u> </u> == , ==

We provide in all beams:

- upper reinforcement: 3xØ14 (462mm<sup>2</sup>)
- lower reinforcement: 4xØ20 (1256mm<sup>2</sup>)

In this window you will also find the maximum reinforcement calculated for each of the four longitudinal reinforcements. In places where the theoretical calculated reinforcement is greater than the practical reinforcement, Diamonds will disregard the practical reinforcement when calculating the cracked deformation.

We also indicate that this reinforcement will be distributed on 3 and 4 bars. This number is important for the calculation of the crack width. Confirm with "OK".

#### **Step 20: Graphical representation of the practical reinforcement**

You can include the practical reinforcement in the representation of the results. Click on the button in the icon bar to edit the settings of the active ('Results') configuration. Select the tab 'Results' and indicate that you wish to see the practical reinforcement. Click 'OK'.

\$			Diamon	ds - Beto	Window Conf	iguration ×
🤗 Eile Edit View Selec	t <u>D</u> isplay	<u>A</u> nalysis <u>O</u> p ∏∏∏∏∏	tions <u>W</u> indov	s Help	Current configuration :	Save current configuration
😿 🗟 🖷 🔀 (Window 1	SK. [1]	•	User configura	ion	User configuration	Delete selected configuration
₽	а в	19	101010	ю ж	Configuration name Results	
f the two	a s	10		а в	General Geometry Mesh Results	
46	62				Representation Default scale Manual scale	
	~.	1.			Results on lines         Results on surfaces           Magnitude         10         Magnitude         10	Reactions Force :
Y <sup>12</sup>	256	16 B	.1256	0 Q	vith values	Magnitude 20 Moment :
	·Х	10 E	10 - 10	a s	Maximum only     Max, begin and end     So-contours	Magnitude 20
Awy	a a	8 B	10 R	н ж	i with values	Show resulting vector
A2. A2.	a a	8 16	14 - 38	a a		
(Ang.) (Ang.)	3 S	煎 製	52 Sž	8 8		
Asw	SI 13	9 R	15 N	а «	practical reinforcement     practical reinforcement     practical reinforcement	
	10 IX	12 E	8 B	a w	reinforcement grid	
	а в	K R	10 A	н ж	X' 1.00 m	
	25 12	25 - 85	e:	ж ж	Z' 1.00 m	
13	a a	8 N	53 Sž	8 B		
48	a a	9 B	10 10	9 9		
					Help	<u>C</u> ancel <u>O</u> K

The figure above shows at a glance the places where additional reinforcement should be provided.

#### Step 21: Calculating the cracked deformation

Now we can start the calculation of the cracked deformation. First we would like to highlight the fact that with Diamonds either calculate

- the total deflection  $\overline{223}$
- or the deflection at a specific time  $^{\bigodot}$  can be calculated.

The second approach takes into account the time at which a given load is active and allows you to estimate the additional deflection which could possibly cause damage to partition walls.

We restrict ourselves to the calculation of the total cracked deformation. Select the menu command 'Analysis – Cracked deflection' or click on  $\frac{1}{223}$  in the icon bar. The following dialog box will appear:



Leave the parameter  $\beta$  unchanged and select the option 'Compute deformation after creep'. Confirm with 'OK'. A dialog box will show you the progress of the calculations.

#### Step 22: Looking at the results

Once the calculation is finished, you will no longer find the elastic deformation but the cracked deformation under this  $r_{sr}$  button. Moreover, you can now visualize the cracking widths using the button  $v_{sr}$ .

Below you can see the cracked deformation  $\delta_y$  after creep for the SLS QP envelope.



Note that this deformation is nearly 3 times larger than the elastic deformation. This increase is partly due to creep effects and partly due to the cracking of the concrete.

Also the cracking widths need to be evaluated. Thus we find in this example a maximal cracking width of 0,23mm under SLS QP.



With this first example you've met the main functions of Diamonds. Many of these features will be discussed again in the next examples, but we also tried to discuss other features of Diamonds.

We don't make a report for this (rather simple) example. The following example is better suited for this, as this model also contains plates.

Note: you don't need to save this model, in the next exercise (§3.2) we start from scratch.

# 3.2 Example 2: A preslab floor

Required licenses:

- ✓ 2D Bars
   ✓ 2D Slabs
- ✓ Concrete Design

# **3.2.1** Purpose of the exercise

In this second example we model and calculate the complete floor of the first level. We calculate the elastic forces in the beams and determine the required reinforcement. We will also check if the cracked deformation is admissible. Finally we make a report.



# **3.2.2 Defining the structure**

#### Step 1: Start a new project

Start a new project through the menu command 'File–New' or click on D.

We could expand the geometry of §3.1, but to show as much functions as possible, we don't.

#### Step 2: Go to the 'Geometry' configuration

Defining the structure is always done in the 'Geometry' configuration. Click on B in the icon bar, or select the 'Geometry' configuration in the adjacent pull down menu.

<del>\$</del>	Diamonds - [Window 1 (m)]			- 🗆 🗙
Eile Edit View Select Display Analysis Options Windows	Help			- 8 ×
				Dorien 🔻 📕 L
😨 🗟 🖷 🛛 Window 1 💽 Geometry	<u>-</u> 瞬 巴 図 マ   ポ 」 中 (**) Q	€ 🛛 🖉	DR	
<b>★</b> <sub>p</sub> ×				Active level
<u>☆</u> • /				61-12,70 m
8 0 0				
				F 1.* 2.70 m
		4		Evel manager
		्रिंग न	op view	Ground level
$\gg$ He $\odot$ Y		Zerty		Drawing plane
+7 BJP+ ↓ ×		F	ront view	X = 0.00 m Y = 2.70 m
南北区	· · · · · · · · · · · · · · · · · · ·	Z=1-X	-	Z = 0.00 m
		<i>.</i>	ackuieu	
HO WA WAR		z ty	ack view	
		-		S. S. S.
	1		eft view	Size
the m 112		∠•~+X		Font 25 📚
[平)		R	ight view	Symbols 10
		Z-1-X		Loads 10
		3D .	Durinuu	Results 20
		zł s	D VIEW	None
			ľ.	
				//

Then check if you're in a top view. If this is not the case, then click on the

button  $\blacksquare$  in the icon bar or on the button  $z \xrightarrow{i} x$  in the lower right corner and select the viewpoint 'Top view'. This way you activate a horizontal drawing area.

#### **Step 3: Defining the grid**

Before we start with defining a grid. In the icon bar, select the button and opt for a standard grid of 1m in all directions. Set the grid active and visible.

9	Diamonds - [Window 1 (m)]		- 🗆 🗙
💝 Eile Edit View Select Display Analysis Options	Mindows <u>H</u> elp		_ & ×
	■ 1 回 X & & ※ 五 ⊕ ™ ▼	均	
😨 🗟 🧮 🔀 Window 1 💽 🚺 User of	onfiguration 🔄 🏚 🖳 🕅 🔁	(?) Q Q Z / D D D D	
$\mathbf{k}$ $\times$ has been been been been been been been bee			Active level
△. /	Crid settings = U X		
			A 1 2.70 m
000	General grid		2.70 m
	Grid		Level manager
	(* On		
- 人/ (新 2日)	C Off X 1.00 m		the Drawing place
7 年15:11 11:11 11:11	Display Y 1.00 m		X = 0.00  m
T <sup>2</sup> R <sub>2</sub> J [ <sup>27</sup> ] SASESAESAESAES	⊙ Visible Z 1.00 m		Y = 2.70 m
Nil W 12			Representation
	Variable grid		AAA
fff all all and a second s			
*** / V			Size
27 T P			Font 10 🚖
E. BARKBARKAR	New Modify Delete		Symbols 10 🚖
	East East		Loads 10 🗲
	Help Cancel OK		Results 10 🜩
			Show groups
			None 💌
2.5.2 ¥ 2.5.5.2 ¥ 2.4.2 × 1. N N D K N N D K K K D N	5 X X 4 5 X X 6 X X 4 5 X X 4 5 X X 4 X 4 X 4 X 4 X 4 X		
			//

You're now ready to draw.

#### Step 4: Drawing the first line

We start by drawing the perimeter of the plates. Active the drawing function with the button of the 'Geometry' pallet and draw the back edge of the plate. You have the choice to:

- either draw the line using the mouse (click once on both the start an end position of the line with the left mouse button). You can rely on the coordinates displayed below in the information bar and adapt them depending on the movement of mouse. Once the starting point is assigned – choose for example the origin as starting point – you will also find the projected distances to the starting point and the length of the segment to be defined. Move for example the mouse to the point where dx = 15m and dz = 0m and indicate this point with the left button of the mouse.



- either enter the coordinates of both endpoints in the foreseen bar at the bottom of the model window. The coordinates of a point are

separated by a ";". If you are in a top view, you only need to enter 2D coordinates.



Each time you enter the coordinates of a point, press the ENTER-key of the keyboard. In addition to the input of absolute coordinates, it is also possible to enter the endpoint of the line using relative coordinates using "@".



So, you've drawn you first line. Turn off the drawing function with real or use the ESC-key from the keyboard.

Step 5: Defining the other lines of the perimeter

Select the line by clicking on it with the mouse. Immediately, a number of buttons on the pallet – those that can be applied on the selected element

- become active. Click on the button to copy the selected line. Complete the dialog box as shown below:

<del></del>	Diamonds - [Venster 1 (m)]     P														- • ×										
🤗 <u>F</u> ile	<u>E</u> dit	<u>V</u> iew	Select	<u>D</u> isplay	Ana	lysis .	Option	s <u>W</u> ind	dows	<u>H</u> elp	8														- 8 ×
🗅 🖻		₫. 4	9 🗘	n	2	FFI IF	<b> </b>	##	Tei	⊠	II X	Ĭ.e	9	\$ <u>111</u>	⊕ Fe%	Iý	IF-	<b>I</b> : +	Ø						
85		Venst	er 1		÷	]	]   Ge	ometry					•	ß	<del>1</del> 1		南	户	<i>هم ا</i>		3	/	R		
N.,	>	<b>(</b> )	8	55	82	18	13	5	-8	- 23	83	92	53	10	5	38	- 68	84	82	51	8	13	18		Active level
	• /	, .	95	-		X														-	œ	-	15	-	2.70 m
n 4	0 1	2	12	27	*		~	8	84	122	85	22	37	-	8	8	-	8	32	27	-	23	87	2	77711 ± 2.70 m
	5 U				2																				
	~	-	155	20	257	20			- 25			25		15		95	-55	101	2		225	30	98		Evel manager
1.1	11 2		25	23	22							Tr	ansla	ate						<	82	53	82	8	Ground level
	ш., сп.		33	-8	58) 194	Г	-	_	_			-	-	-	-	-	-	-	-		68	30	25	8	C Drawing plane
T <sup>×</sup> R	μť	*1	8	12	85		V =	2		∩"́∔	ዀ	ŝ		crea	te con	nect	ing lin	es			0	25	18	0	Y = 2.70 m
Nî∩ ∌	H &	2	12	23		L							V	crea	te con	nect	ing su	irface	s		75		192		Representation
IT S	24					lr.					_		_												***
鋼。	Fe D	<b>5</b>	25	10	88					dx	:  0.0	00	п	n							12	63	15	22	â 📠
I I	1	5	92 1	57	<u> </u>					dy	: 0.0	00	n	n							84	23	87	8	
<b>≡</b>	70	2		-	33					dz :	5,0	00	n	n							88	30	<i>5</i> 2	8	Size Font 25 🕏
E,		-	22	53	25																10	22	82	-	Symbols 10 文
1			33	35	92																8	32	93	8	Loads 10 🗢
							H	elp						Ca	ncel			<u>0</u> K							Results 20 🚖
		*	6	64	80								-				-				1	28	22	*	Show groups
		3	32	-23	86	10	14	20	12	12	8	88	-33	15	25	3	12	8	(ii)	-25	12	87	8	- 22	INone 💌
		8	98	8	22	8	×	30	3	98	22	8		12	N	.9	25	2	81	0	12	2		zX	

Click 'OK' and you will obtain this drawing:

÷											Diam	nonds	- [Ver	nster	1 (m)]	l.									- 🗆 🗙
🤗 Eile	e <u>E</u> dit <u>V</u>	iew S	elect	<u>D</u> isplay	Anal	/sis (	option	s <u>W</u> in	ndows	Help															- 8 ×
	ê 📕	D. #	Ø	50	•   +	ΠĦ	Ħ	##	Tii	× 1	i Y	že -	6 8	<u>; 111</u>	(1) F%	r⁄	4	庫	4						
*		Venste	r 1		•	1	Ge	ometry					•	B	F11 🕅		南	户	<">	Q		Ø		R	
k,		1		15	82	18	-	ų.	-38	98	81	8	10	10	10	38	- 22	>1	9	10	8				Active level
^⊞	• /		18	52		X														-	10				2.70 m
Û	00	1	17	27	ž	1	8	8	84	17	8	10	8	8	2	87	12	81	53		8				2.70 m
$\widehat{\mathbb{Q}}$		a	×	10		55	8	8	2	ж	29	26	10	8	8	ŝŔ.	ж	29	18		8				Evel manager
1	-	].	25	33		53	2	23	12	2	8		-	8	23	82	20	8	10		8				Ground level
4	HH ()		35	85		ĸ	8	8	15	×	8	18	45	8	8	15	35	3	38		8				Drawing plane
<i>√</i> ∕	BJ ₽+ 124 /*		8	8	-															_	0				Y = 0.00 m Y = 2.70 m Z = 0.00 m
- Digit	211 75	- 2	10	15		10	15	25	12	12	34	86	15	15	20	12	10	8	86		15			2 25	Representation
IO		2	28			0	S.	52	3	28	12	85	12	12	32	.0	25	12	82		10				
E	83 P	1	17	27		18	8	22	8	12	8	а.	23	8	2	8	12	8	12		8				S 4
*	= 1		30	10		51	8	8	a.	ж		18	10	8	8	12	ж	- 22	25		8				Size Font 25 🚖
144(0	11 .46	-	10	33	_																8				Symbols 10 🚖
			35	15	92 92	ĸ	8	8	15	93	8	22	13	8	8	94	35	24	12	13	8				Loads 10 🚖
				•10		*:					-	×.	•0					20		•00					Results 20 🚖
																									None
					96				28			×		5.0		0		8.		•0	50			x	25
		1.0	28	2	20	0	N.	32	3	- 25	12	20	5	12	32	-0	25	19	88	10	100		2 2	ż	1

#### About the 'Translation' function

- In the field 'N' you enter the amount of copies you want. When just want a translation (= move something), 'N' should remain equal to 0.
- In the three fields below you enter the translation (or copy) vector.
- When you check the boxes 'create connecting lines' or 'create connecting plates', Diamonds will automatically draw lines or plates between the copied items.

Select the centre line and click on  $\checkmark$  to divide the line in 3 equal parts. Confirm with 'OK'.

<b></b>													[	Diar	non	ds -	[Ve	nste	1	(m)]														- 🗆 🗙
🂝 <u>F</u> ile	Edit \	/iew S	elect	Disp	lay ,	Analy	sis	Opt	ions	Win	ndow	s <u>F</u>	lelp			-						m			1147.1	1								- 8 ×
		Là. @	<b>9</b>	5	CH.	H R	H H	H H	1   8		Ţ.				( 1)	e e	<b>b</b>   8	5: 22	1 Œ	) Fe//	V		IF	۱.	10					ы.				- i i
85		Venste	er 1		10	•		1	Geor	netry		57	- 54	~			•	ß	<b>F11</b>			Ĥ	神	白	- Su		•				D			3   00 100   5
<b>♦</b> <sub>p</sub>	×		• ×														_	e 14	13	-			8	41	10	84	42	æ	84	10	10	84		Active level
	- /		2 .	53	8	8	53	8	8	53	8	8	2	8	8	8		8	8	2		8 8	8	8	8	25	8	3	8	8	10	12		2.70 m
00	l b	)	(3) (3)	- 53 - 53	8	8	-8	- 83	8	- 22	8	88 20	24 15	8	8	34		8	34 53	3		8 5 8 8	94 13	8	8	24 10	8	35 32	59 53	8	33 32	24 13	8	2.70 m
	Þ			5	20		53	2		53	18	8	8	18	8	8		8					8	8	8	8	8		8	8	10	81		E Level manager
			-				_					_		_		_	_	e 8	8	-		8 3		8	83	8	8	33	Se	8	35	Se	1	Ground level
i i i i i i i i i i i i i i i i i i i	HJ	-	14	15	35	15	15	30	15	13	20	15	35	20	15	12		N.	33	3		6 8	8	31	11	34	30	8	39	20	36	89		Drawing plane
T <sup>T</sup> B.	۶P۰	ł	8	- 53	8	88 88	53 -33	8	8 8	53 - 73	8	8	81 54	- 8	8	83 53		8	18 18			8 8 8 8	а ж	8	8	81 52	8	25 25	81 53	8	25 33	81 54		X = 0.00 m Y = 2.70 m
NÎA 🖌	4 p	2	N.	23	25	14	25	25	15	13	25	15	33	25	15	1		14	33	3		a 1	a	25	8	33	25	56	84	25	56	84		Representation
HT (	24				_												_	ξ. μ	53			<u>e s</u>	1	81	320	32		22	12	23	35	82		***
<b>M</b> 8	Fe 🗗	]	6 68 6 68	83 23		Ŷ									Di	vi	de	lin	e									×		8	33 52	54 54	8	<b>A</b>
<i>™</i> ⇔ ∕	ľ ()	> .		53															ſ		_	-								23	35	62		Size
The -	7 🕸	e ,	8 8	12		N	lum	iber	of	seg	jmei	nts								اد -	_									8	36	39		Font 25
E,			6 6	23		E	qui	al se	egm	ent	ts									ŀ	/									25	56	85		Symbols 10
1			8 8	3																										-8	33	8		Loads 10 🗢
			8	80				He	b		1								Ca	nce			Γ	-	c	ж	-	٦		8	36	9	2	Results 20
		3	11 IX	-33		_			*	_											_			_	_		_			25	56	88		None
		2	a 14	53 62	32	90 14	- 22	20	in a		20	14	12	20 30	in an	10	;	u u	12	,	5 8 2 3	с ;	2 9	20	a.	22 13	20	8	12	20 20	25 25	Ţ	-×	

Thus the position of the columns in the main building is established.

#### **Step 6: Visualizing the node numbers**

Before drawing the outline of the extension, we would like to make the node number visible. To do so, we need to adjust the window configuration 'Geometry' . Click on in the icon bar and select the second tab 'Geometry'. Here you'll find all the model information that can be shown on the drawing.

<b>9</b>	Diamonds - [V	Window Configuration	×
	Options Windows Help		
🗅 🚅 📕 🖪 🖉 🛑 🕫 🖓 🗐 🛱	王田 🔢 🚺 📓 🏌 🎤 👌 Current configuration :		Save current configuration
😨 🗟 🥅 🛛 Venster 1 💉 🚺	I Geometry User configuration	•	Delete selected configuration
	Configuration name Geomet	у	
	General Geometry Mesh		
🖾 · 🖌 Zannan			1
	i e e e e e e e e e e <mark>Points</mark>	Lines Surfac	205
	Show point	Show axis Transpo	arency : 50 🚖 %
	Point number	Line number	face number
	Connection name	Name complete Nam	ne
		Length	kness
A H M A A A A A A A A A A A A A A A A A		Section orientation	a
AT # 15 1		Local coordinate system	al coordinate system
T <sup>*</sup> B,J I <sup>*+</sup> I		V End releases	releases
南斑惑		Material Mat	ierial
		Border lines	der lines
		Buckling length f     Pra	ctice reinforcement
ATTAL TE ATTAL A REAL AND A REAL AND A		Distance between LT suspende	entation
w s. l-		750	
		Distance between LT supports	
		7<0	
the m 12 and a second			
T		LT supports z>0	
		Fire curve	
		Required fire resistance	
			Creat Original
	<u>H</u> elp		

Indicate in the first column that you wish to see the point numbers and click on the button Save current configuration. Confirm that you wish to replace the current configuration with these new settings. Each time you now ask this configuration, the point numbers will be shown. Click 'OK' to close this window.

#### Step 7: Drawing the outline of the extension

We now draw the outline of the extension and we start at node 5. Again

click on the button  $\checkmark$ . The node numbers show you the sequence of drawing. Note that you can draw contiguous bars in one movement by indicating the consecutive points. End the drawing function using the ESC-key.

Diamonds - [Venster 1 (m)]														- 🗆 🗙																
🥮 <u>F</u> ile <u>E</u> dit ⊻	ew <u>S</u> ele	ect J	Displa	y i	<u>A</u> naly:	is (	<u>Optio</u>	ns	Wind	lows	He	lp																		- 8 ×
D 🗳 日	161	D	5	Ċł.	F	H	E			T.:	$\boxtimes$		X	6e	6	18:	111	5) Fg	× I		F	<b>I</b> :	Ð							
	😨 🗟 🕾 Venster 1 🔽 User configuration 🚽 瞬 🗉 🗑 対 🔯 🌾 由 🥐 🗨 🖉 🖉 🖉																													
	1 2	36	-84	12	14			30	<u></u>	10	36	4	19	î.			30			1	301.1		30				1	- 62		Active level
₹ <sub>₽</sub> <u>×</u>		33	2	53	35	32	52	33	32	52	8	82	23		2	22	10		22		2	8		12	83 B		12	25		Level 1
	÷ 8	35	<u>.</u>	1	٠X						15	.00m	1	-					2		<i>i</i> te	35	38	a.	- -	6 (R	33	3	a.	2.70 m
6 0 0	- 31	96	35	ž	96 96	8	62	96	8	6	96	35	82	ł	12	92 192	86	12		88	12	8	88	9	45: S	8 19	82	32	-	
		33	2.2		1	82	22		82	53	8	82	23	ł		22	88		-		25	8	8	2	8 8	8 8	22	25		
~ ~		35		5.ψι	201	-	10	33	-	10	33	28	85	ł	a	80	35		5.001	0 8	a.	10	38	a	en 12	6 B	10	35	1	Evel manager
-	l	96	35		8	85	<b>S</b> 2	8	32	49	98	32	12	ł	12	192	8	12		88	12	5	88	3	19: - 9	8 19	12	12	a <sup>11</sup>	Ground level
4 HH 23		22		3	í.	5.0	00m		7		5.	00m	2	8		5.0	00m		5		5.0	0m		9	•2 - 8		12	25		1 Drawing plane
A. 5 D.			12		22		8	20		8	82	32	83	ł	3	85	8	13		8	13	8	22		8 8	8 18	8			X = 0.00 m
T B <sub>4</sub> J 1		86	8			85	52	8	5	10	36	82	32	1	12	-	56	а.		8	12	8	8. <b>.</b> .		20 S	8 18	5	12	-	Y = 2.70 m 7 = 0.00 m
N/1 2/1 /3	- 			5.00	Jm		50	22		50	22		50	ł		50	22		5.00 <b>m</b>	<u>,</u>		50	. 5	.00m			50			Representation
		20			20			20			- 20			Į.		- 20	20					8	20			0 33		- 00		AAA
(4441 : Fe at				4							15	.00m		1					6					10						
w s. r																														SP 🛷
1 to 1 C		33		- 53	20		- 20	20		- 20	33		- 23		10	- 22	100		53 			53 					- 20			Size
																							5	.00m						Font 25 🚖
	-	340	2.5	10	ас		112	36	2.4	10	30	2.5			24	10	36	24	15	20		10	30		*E 3	6 34	15			Symbols 10 🜲
		33	10	- 53	35	15	23	33	35	23	33	15	28	13	8	50	20m	88	12	88	5.0	Om	35	11	58 8	8 88	- 23	22	10	Loads 10 🜲
		88	100	- 65	88 	88	80 	28	38	8	88	10	8	علم					-Le	~~			~		55 3 		83. 	28	35 	Results 20 🜩
		36	10	40	36	10	10	30	53	10	30	0.9	80	1	38	40	36	- 17	40	10	38	80	30		40 Y	0 37	10	16		Show groups
	S 8	35	85	53	25	85	52	33	8	52	35	25	53	i i	12	23	35	12	50	3	12	53	25	12	53 S	8 8	53	22	12	None
	Give ne	xt poi	nt:	83	23	32	85	28	8	85	25	85	83	Ť	3	85	28	35	8	80	35	8	38	3	<u>12 3</u>	8 35	8			a sila il geografi
	X = 10	.00m	Y = 2	2.70r	n Z=	10.0	0m	dx	= 0.0	0m (	dy =	0.00m	dz :	= -5.0	00m	L =	5.00	m											z	
																														1.

To draw the last line you make clever use of the intelligent. This is a cursor that detects when he comes near e.g. an existing point or bar. In particular the intelligent cursor snaps to the orthogonal projection on the existing bar when drawing the second point of the last line. The line between nodes 4 and 6 will be divided automatically.

To display the structure as large as possible on the screen, click on the button in the icon bar or use the function key F12.

#### **Step 8: Generating plates**

Once the perimeter of the plates has been drawn, we can generate the plates. Select the menu command 'Select – All', draw a selection window over the complete model or use 'CTRL + A'.

The entire drawing will be represented in bolt to indicate all the elements are selected.

Now we click on  $\checkmark$  in the pallet to find the plates.

#### About the function 'Generate plates' 🍣

In particular Diamonds searches between which selected bars a plate can be formed. Two conditions must be met:

- All the selected bars should lay in the same plane.
- The bars need to form one or more closed contours.

Diamonds defines three independent plates which are represented in grey. If you don't see the plates, you're probably in a wireframe representation

2. In such a view only the edges of the plates will be recognizable. You can change the representation with the toolbar on right side of the model

window. If you opt for a transparent view , then the plates are displayed in a colour in addition to the contours. However, the model remains transparent so you can see the other elements that might be under the plate.



#### **Step 9: Editing the plate properties**

By default the plates are isotropic and have a thickness of 20cm. You can change the properties of one plate by double clicking it. Double click for example the plate on the right and change the thickness of the plate to 15cm. Note that a selected plate element is standard displayed in yellow when it's selected. This is because the plate belongs to a certain 'Type'. For more information see §2.6.

Properties of plate number: 3	- • ×
	- 8 ×
	F IF 40
	⊉ (?) Q Q 📓 🖉 🖻 🖻 🖻 🖾 🗇 👌
Shape	A Ctive level
Dimensions Stiffness matrix	
thickness (e) 150.0 mm	3 3 3 5 5 5 5 5 5 5 5 5 5 5 5 5 5 5 5 5
	· · · · · · · · · · · · · · · · · · ·
	Ground level
	III Drawing plane
	Y = 2.70 m
	2 Z = 0.00 m
	Representation
Properties	
Material Concrete C25/3C 🔽 🍸 Gross cover	
Local axes	Font 25 🜩
Local xz-axes orientation $0.0 \circ 1^2$	S 10
	Loads 10 \$
Local y-axis orientation	Results 20 ±
2 × 3 3 3 1 1	Show groups
Level = Afdek gelijkvloers	None 🔻
Help Caprel OK	

The two other plates are preslabs. Such plates have an orthotropic character, which means that they have a different structure in the two perpendicular directions and thus have a different rigidity in both directions. For orthotropic plates the main directions coincides by definition with the local x'-axis. By default, the local coordinate system of a plate will always coincide with the global coordinate system (x'-axis // X-as and z'-axis // Z-axis). Because the preslabs now bear from top to bottom, i.e. // to the global Z-axis, the local coordinate system of a plate visible in an analogous manner as displaying the node numbers. Click on and indicate that the local coordinate system of the plates should be displayed. Optionally, you can also decide to show other plate information. This time we click immediately 'OK' so we make a temporary user configuration. The desired model information will be visible until you select one of the standard configurations (Geometry, Loads, mesh or Results).



Now select both plates and click on the button  $\overset{\text{W}}{\overset{\text{W}}}$  to adjust the plate properties. The modifications being carried out now, will be applied to both selected elements.

First give the plate a name and select the second plate type from the pull down menu. Enter the total thickness (so preslabs + second phase concrete) and the thickness of the preslabs on the right. This means that in the preparation of the stiffness matrix in the main direction (// x'-axis) the full thickness of the plate is taken into account and the secondary direction (// z'-axis) only the thickness of the second phase concrete is counted.

<b>⇔</b> □	Properties of plate number: 1	×
♥ File       £dit       Yiew       Select       Display       Analysis       Options       Windows       Help         □	General Name Slab 0.20	1
Image: Second secon	Shape	
	Dimensions Stiffness matrix thickness (e) 1150.0 mm (1) 50.0 mm	
Seb 0.20 」 夕 附 改		
<i>₽</i> <sub>в</sub> 5 P+1 № <u><i>1</i></u> <u>1</u> <u>2</u>		
	Properties	
	Material Concrete C25/3C  Gross cover  Local axes	
	Local xz-axes orientation 90.0 °	
	Level = Afdek gelijkvloers	
	Help Cancel	

Leave the assigned material (Concrete C25/30) unchanged. By means of the button Gross cover you open a new dialog box in which you can define the gross coverage of the longitudinal reinforcement. The gross coverage on the upper and lower longitudinal reinforcement is the distance from the centre of gravity from the reinforcement tot the upper/lower edge of the section. Immediately the useful height of the cross section – needed for the calculation of the longitudinal reinforcement – can be deduced.

The thickness of the preslabs is 5cm. We assume that the longitudinal reinforcement  $(A_{xi})$  is placed in the middle of the preslabs. We don't calculate the distribution reinforcement in the preslabs. However we are interested in the number of distribution bars  $(A_{zi})$  we need above the joints. From this we know that the coverage of this reinforcement must be greater than the thickness of the preslabs. The upper reinforcement  $(A_{xs} \text{ and } A_{zs})$  is placed with a coverage of 35mm.



Complete the dialog box with the information below and confirm with 'OK'.

C	iross Concrete Cover	×
Concrete Cover		
X' - Reinforcement ( $Y' > 0$ ) (1)	35.0 mm	ŹŹ¥→X ∠
X' - Reinforcement (Y' < 0) (2)	25.0 mm	
Z' - Reinforcement (Y' > 0) (3)	35.0 mm	ŹŻŹ¥X <sup>(3</sup> ∕
Z' - Reinforcement (Y' < 0) (4)	60.0 mm	(4)
Help		<u>C</u> ancel <u>OK</u>

You'll see the plate properties again. We now rotate the local coordinate system of the plate so the local x'-axis of the plates lies parallel with the global Z-axis, for reasons mentioned above.

🤪 Dia	Properties of plate number: 1
Pile Edit View Select Display Analysis Options Windows Help         Pile Edit View Select Display Analysis Options Windows Help         Pile Edit View Select Display Analysis Options Windows Help         Pile Edit View Select Display Analysis Options Windows Help         Pile Edit View Select Display Analysis Options Windows Help         Pile Edit View Select Display Analysis Options Windows Help         Pile Edit View Select Display Analysis Options Windows Help         Pile Edit View Select Display Analysis Options Windows Help         Pile Edit View Select Display Analysis Options Windows Help         Pile Edit View Select Display Analysis Options Windows Help         Pile Edit View Select Display Analysis Options Windows Help         Pile Edit View Select Display Analysis Options Windows Help         Pile Edit View Select Display Analysis Options Windows Help         Pile Edit View Select Display Analysis Options Windows Help         Pile Edit View Select Display Analysis Options Windows Help         Pile Edit View Select Display Analysis Options Windows Help         Pile Edit View Select Display Analysis Options Windows Help         Pile Edit View Select Display Analysis Options Windows Help         Pile Edit View Select Display Analysis Options Windows Help         Pile Edit View Select Display Analysis Options Windows Help         Pile Edit View Select Display Analysis Options Windows Help         Pile Edit View Select Display Analysis Op	General Name Slab 0.20 Shape Dimensions Stiffness matrix Dimensions Stiffness matrix thickness (e) 180.0 mm (1) 50.0 mm
	Properties Material Concrete C25/3C
	Local xz-axes orientation 90.0 • 2 X
化化化化化 化化化化化化 化化化化化	Level = Afdek gelijkvloers
	<u>Lep</u> <u>Cancel</u> <u>C</u>

Click 'OK' so that the new cross section, the orientation and the gross covers are assigned to the plates.

Have you taken the previous steps successfully, then an arrow will indicate the direction of the preslabs, at least if in the configuration settings 1 the option for showing the plate orientations is checked.



The clarity of this symbol depends on the type of graphical card in your computer. In any case, you must recognize this symbol when you take a

wireframe representation. Simply select the button  $\overset{\textcircled{}}{\textcircled{}}$  in the toolbar on the right side of the model window.

#### **Step 10: Defining line supports**

The floor is supported by a few walls and columns which we will model as simple point and line supports. We start with the line supports. Select the

following lines using the SHIFT-key and press the button in the 'Geometry' configuration.



Because the endpoints of the lines are also selected when you select the lines, you will see two tab pages in this window. We choose a simple

support in the second tab (Lines) by clicking immediately on  $\stackrel{\triangle}{\rightarrow}$  in the left column. Or you can check the three checkboxes (X, Y and Z displacement) on the right. Confirm with 'OK'.

#### **Step 11: Defining the points supports**

Now select all the points that did not belong to the first selection, in particular the point numbers 6, 7, 8, 9, 10, 11, 12 and 13. Again click on

 $\stackrel{\mathbb{Z}}{\longrightarrow}$  and assume a simple support.

<del>.</del>	Diamonds - [Venster 1 (m)]	- D ×
Eile Edit View Select Display Analysis Options Windows E	<u>i</u> elp	- 8 ×
	▩■▮◪◪◓ ᄬ▥©▫~~  ⊢▫▫▫	Supports
😨 🗟 🖷 🗵 Venster 1 💽 🚺 User configuration	@ ⊡ [	
	Points	
	Xdisplacement	X rotation
4/ INI 🖓    <mark>Baaaaaaaaaaaaaaaaaaaaaaaaaaaaaaaaaa</mark>	ked	Free 💌
_∕7 "¶ ₽.4	cannot resist to cension	
* 14 0	Y displacement	Y rotation
	Cannot resist to tension	1 Free
HT 🖓 🌒	Cannot resist to compression	7
鋼 alt alt		Free V
	cannot resist to tension	
	Angle relative to h	orizontal plane α : 0,0 °
	Angle relative	to vertical axis β :  0,0
机能振动器 化化化化合金 化化化合金		
	n a ar na ar an an a <mark>Help</mark>	<u>Cancel</u> <u>QK</u>
		None 🔽
		a a a a a a
		z i == set

#### Step 12: Assigning the beam section

Select all lines who don't have a line support and click on for defining a section based on a standard shape.

	Section	×
Select Display Analysis Options Windows Help		
□□☞ ■ 🖪 ● 🔍 🕫 🕫 🕫 🛛 🗎 🛛 📾	Cross-section	
😨 🗟 🗮 🛛 Venster 1 🔹 🚺 User configuration	Name R250/400 👻 🎵	
	Shape Constant	
🔥 🗙 сколования сконсколова		
	Dimensions	noc Aves
	B 250.0	
	B 250.0 mm	
	H  400.0 mm	+ +
		±
· 夕 ₩ 29 · · · · · · · · · · · · · · · · · ·		
J S DJ		
τ' BJ Γ'I Cressesses		
NA # Q		
IO 🔍 💱 setty a a a a set te set ty y a a a		
m e t	Section properties	
Σ. × A hard and a state stat		Material dependent
T T It contractions and the second seco	✓ Automatic calculation	
	General Elastic Plastic Cold-formed members	
	strong axis v-v weak axis z-z -	
	Sy 2000000 mm <sup>3</sup> Sz 12500000 mm <sup>3</sup>	alpha 0.00
	Iy' 133333333 mm" Iz' 520833333 mm"	Iyz 0 mm*
	iy' 115.5 mm iz' 72.2 mm	It 1273451646 mm*
	Wel,y',t 6666667 mm <sup>3</sup> Wel,z',l 4166667 mm <sup>3</sup>	IW 0 mm <sup>4</sup>
	Wel.y'.b 66666667 mm <sup>3</sup> Wel.z'.r 4166667 mm <sup>3</sup>	Twm 6060606 mm <sup>3</sup>
	the second	La constant a
	Material	
	Material Concrete C25/30 🔽 🍸 Reinforcement	
	Local axes	
	Orientation angle 0.00 °	Mirroring
	Help	Cancel QK

- Note the name of the new cross section.
- Using the pull down menu under the name, select the desired shape. Choose for a rectangular section.
- Enter the dimensions of the cross-section. They are represented in the schematic drawing on the right.
- All the properties of the cross section are calculated automatically.
- Underneath select the material. We opt for 'Concrete C25/30'. Leave the coverage on the longitudinal reinforcement as well as the orientation of the bars unchanged. By default a gross cover of 35mm is used.

Confirm with 'OK' and verify if the beams have the defined section. Most likely you'll have to change the configuration settings again. Click on and indicate you wish to the see the 'Border lines' of the cross sections. Click 'OK' and opt for a 3D view. You can change the point of view by move the sliders on the right and at the bottom of the model window.



Now opt for a solid representation ( $\checkmark$  or  $\checkmark$ ), all the elements will be presented volumetrically. The representation in colour also shows you which elements belonging to the same design type.

#### Step 13: Defining types

From the drawing it follows that a number of standard types have already been defined. Beam and plates will be recognized as a separate type. You can find all types defined in Diamonds when you click on <sup>1</sup> in the icon bar.



For example we wish to place the preslabs in a separate design type.

- Select both plates using the SHIFT-key and click on <sup>1</sup>
- Using the button Add new type you define a new design type for which you can change the name immediately.
- We give this type a different colour so that we easily recognize the plates. We can even change the colour of the element in case it's selected.
- Confirm with 'OK' and deselect the plates.

If the colour of the plates has changed, you know the design type was assigned successfully. In the figure below we opted for a transparent view



and we turned off the border lines.

🥹 Diamonds - [Venster 1	(m)]			-	
					- 8 ×
	) Fe%				-
🐮 🔄 🖷 🖄 (Venster 1 💽 🚺 User configuration 🗾 🎆	) )	Desig	n types		< <u>-</u>
	·	2 00.9			
		Add new type	Delet	te selected type	1 n
000	_				- n
	⋟	Туре	Normal	Selected 🔛 ^	ager
		Beam			vel
今回望		Column			ane
PajPi 🦓 💦	۶	Footing			
No the second se	۵	Opening			
	۶	Plate			R
Mela 🔶 🔶	۵	Rigid link			~
Z/0	۵	Wall			_
	٨	Preslabs			¢
				v	
	H	lelp	<u>C</u> ar	ncel <u>O</u> K	
			_	Show a	OUDS
				▼ None	•
7				X angen	

#### Step 14: Adding boundary conditions to the plate's edges

The penultimate step in building our model consists off adding a few hinged lines. It has been said that no moment can be transmitted between the main building and the extension. In other words a joint is foreseen between both.

Select the preslab adjacent to the extension together with the edges adjacent to the plate. Click the button 🔗

- In the dialog box that appears, you recognize below an image of the selected plate. The selected borders are detected immediately.
- Deselect all the moments around the edges in the lower half of the window
- Confirm with 'OK'.

It's also possible to only select the plate, click  $\checkmark$ , and then indicate the edges in the dialog box using the SHIFT-key.



Given both plates along this edge are supported by the beam, you'll have to add another hinged line to the other side. Therefore, repeat the same procedure on the other side.



For the last time change the configuration setting (via 1) so the edge releases are displayed.



#### Step 15: Hinge at beam end

We define a hinge at the end of beam 9 with the aim of disconnecting the continuous beam from the extension. Select the menu command 'Select – Bar number ...' to select bar number 9. Click in the pallet on the button

. Turn off the transition of moments at the rod end with the largest Xcoordinate and confirm with 'OK'.



#### Step 16: No buckling check for beams in plate surface

We assume that the beams in the plate surface can't buckle. Therefore, we'll turn off the buckling check for the beams.

Select the beams and then click the button  $\mathbb{I}^{\ell}$ . Uncheck all boxes for buckling.



There, the structure is completely defined. Now we define the loads.

# 3.2.3 Defining the loads

Step 17: Go to the 'Loads' configuration

We now leave the 'Geometry' configuration and activate the 'Loads' configuration to enter the loads. Click on the button  $\square$  in the icon bar or select in the adjacent pull down menu the 'Loads' configuration.

## 3.2.3.1 Creating the load groups

#### Step 18: Creating load groups

Before defining any loads, you have to make the different load groups. Click on the button  $\gamma_{g}^{\gamma_{u}}$ . You'll see the following screen:

				Diamo	nds - [V	enster 1	I - Selfv	veight -	(kN, kl	Nm, mn	n, kN/m	, kNm/n	n, kN/m², °C)]			_ □	×
Ś	Eile	Edit View Select Displa	ay <u>A</u> nal	ysis <u>O</u> pt	ions <u>W</u>	indows	<u>H</u> elp									-	8 ×
1	ם ב	· 🖬 🖪 🎒 🗠	~	HHA	7   III	Tei		L Ke	<b>9</b> 8.	<u>111</u> (B)	°% I%	= II	¢ +0				
Ĉ	* 5	🖷 🔀 Venster 1	•		Loads				•	<b>R</b> 💷		ri de	* 🕅 Q Q 🖾		R 🖉		8
y y all ⊗ im ⊗ Istwardt ▼															•		
Selfweight 💌														A 1 2.1	70 m		
5																	
4	Coad groups – D																
Г																	
	Load	factors for		EN 1990		-		•	Conseq	uence d	ass 🛛	2 韋		Service class			
									Design	lifetime	Γ	50 韋	years				
			_			1							1				
		Add load group		Insert loa	ad group					Delete	oad grou		Several load cases per group				
		Name load group	$\gamma_{\rm uls}$ -	$\gamma_{uls+}$	$\gamma_{\rm sls}$ -	$\gamma_{\rm sls+}$	ψ <sub>0</sub>	$\psi_1$	ψ <sub>2</sub>	φ	ξ	to	Combination for cracking	k <sub>mod</sub>	Load	Action	
	$\checkmark$	Selfweight	1,35	1,00	1,00	1,00	1,00	1,00	1,00	1,00	0,85	0		perm	—	<u>+++</u>	
	<ul> <li>Image: A start of the start of</li></ul>	dead loads	1,35	1,00	1,00	1,00	1,00	1,00	1,00	1,00	0,85	0		perm	—	<u>+++</u>	
	<b>√</b>	life loads A : housing	1,50	0,00	1,00	0,00	0,70	0,50	0,30	1,00	1,00	0		mediu	—	<u>+++</u>	
		Incompatible load groups			Linked lo	ad group	)S			Comb	ine load	ases	Ung	roup loadc	ases		
	Help Cancel QK																

To keep the example simple, leave the parameters untouched as in the figure above. Then click the 'OK' button.

## 3.2.3.2 Filling up the load groups

Now the loads groups are defined, we can assign loads to the structure.

Step 19: Filling in the load groups 'Self-weight', 'Dead loads' and 'Life load'

- The **self-weight** of the beams is calculated automatically by Diamonds and cannot be adjusted.
- The **dead load** on the plates is 3kN/m<sup>2</sup>.
  - Use the pull down-menu to activate the load group 'Dead loads'.
  - o Now select the preslabs (use the CTRL-key) and click on the

button B. Note that only those icons will be active that can be applied on the selected elements.

• Complete the windows as follows:
Diamonds - IVenster 1 - dead loads - (kN, kNm, mm)	. kN/m, kNm/m, kN/m <sup>2</sup> , °C)] – 🗆 🗙	
Eile Edit View Select Display Analysis Options Windows Help	_ <i>6</i> ×	
□☞ ■ 집 ● ♥   ∽ ~   田田田     ■   1 ■    2 ■ 米 2 と 参 ⊡ ⑤ !!	× x     ⊨ ∎ +0	
😨 🗟 🎮 🛛 Venster 1 🔹 🚺 Loads 🔹 🗣 🕅		
vy	Active level	
	Force on surface	×
dead loads		
YI3D		
2% P		-
¥ > 2 (1.0)		7
	© Uniform load 3,0 kN/m <sup>2</sup>	·
A A TOTAL	O Trapezium load	
	Y(m)	
- Snow	Level 1 0,00 Value 1 0,0	kN/m²
v Wind	Level 2 1,00 Value 2 0,0	kN/m²
v Dynamic	C Non-uniform load	
* Moving	X(m) X(m) Z(m)	
0	Point 1 1.00 0.00 0.00 Value 1 0.0	kN/m²
	Point 2 0.00 1.00 0.00 Value 2 0.0	kN/m <sup>2</sup>
4	Point 3 0.00 0.00 1.00 Value 3 0.0	kN/m <sup>2</sup>
	Help	<u>o</u> ĸ
	× z × == set	

In the 'Loads' configuration window the entered loads are graphically represented.

Select the plate from the extension and place a load from 1kN/m<sup>2</sup> on it using the same manner.

Now select the load group 'Life load A: housing' and define a distributed load on the preslabs of 2kN/m<sup>2</sup>.
 We don't provide any life load on the extension.

# 3.2.3.3 Making combinations

Before starting the calculations we need to generate the combinations first.

#### Step 20: Making combinations

Generate the combinations  $\frac{41}{4m}$  as described in §3.1.3.3.

# 3.2.4 Generating the mesh

Step 21: Generating the mesh

Click on the button  $\square$  in the icon bar or select the menu instruction 'Analysis – Mesh'. Enter a maximum element size of 0,8m and a minimum of 0,4m.

Mesh creation		×
Plates		
Maximum element size	0.80	m
Minimum element size	0.40	m
<ul> <li>Advanced parameters</li> </ul>		
<u>H</u> elp <u>C</u> ancel	<u>O</u> K	

Now click 'OK' to start the generation of the finite element net. A window shows you the progress of the generation.

#### About the mesh generator

Diamonds is an 'on displacements based' finite element program. This
means that the model will be divided in a limited number (= finite) elements.
The composition of such a finite element mesh is called the 'mesh
generation'.

The properties in **the upper half** of this dialog box are related to the division of plate elements.

- The maximum element size allows you to define a general fineness of the mesh. Diamonds will try to subdivide the plate into equilateral triangular elements whose edges are equal to the maximum dimension.
- With the minimum element size, you can manipulate subdivision of elements around the smallest lines or very close points. Because this model doesn't contain any points very close to each other, this parameter will not have an influence. We can leave this value equal to 0.
   Note: there are no general rules that guarantee the accuracy of the finite

elements calculation. Some guidelines concerning the choice of the element size can be found in the reference manual.

The meaning of the other parameters can be consulted on our support website: <u>http://buildsoftsupport.com/knowledge-base/how-to-pick-the-mesh-size/</u>.

#### Step 22: Checking the mesh

Once the mesh is generated, we make the mesh visible with  $\square$ . In a top view the model now looks like this:



We see that with the entered dimensions, we obtain a dense and regular mesh. A mesh consisting of regular ( $\approx$  equilateral) triangles is necessary, given the quality of the results strongly depend on the shape of the mesh.

# 3.2.5 The global elastic analysis

Step 23: Elastic analysis

Follow the same method as described in §3.1.5.

## Step 24: Go to the 'Results' configuration

To see the results of the calculations, you click on  $\square$  in the icon bar or select in the adjacent pull down menu the 'Results' configuration.

Below, we list some results.

#### Step 25: Deformation

- Click in the 'Results' pallet on the button for viewing the displacements.
- Select the vertical displacement  $\delta_y$  according to the global Y-axis.
- Then select the combination group 'SLS QP' and choose for the envelope of the results.

Because we are looking for the largest downward deflection we select the option "min", which indicates the smallest real value. Since the downward deflection is oppositely directed to the positive Y-axis, we can understand this choice of combination. The maximum deflection is 3,8mm. The figure below is shown in wireframe and we opted for a top view.



## Step 26: Deformation in the beams only

If for example, you only wish to see the deflection in the beams, select all the beams (using CTRL-key) and click on  $\mathbf{H}$  in the icon bar. Choose for a 3D view:



Click on  $\square$  to make all elements visible again and choose for a top view.

Now we visualize the bending moments  $M_{xx}$  in the plates. Click on the pallet and select the plate result  $M_{xx}$ . In particular show the maximum values of the ultimate limit state FC (ULS FC) envelope. The notation 'max' refers to the biggest real value. A bending moment is always drawn on the side under tension (which shows clearly in a 3D view). The sign of the bending moment corresponds to the direction of the local coordinate system. In normal cases, the local y'-axis is directed upward, so that tension arises in the lower fibers. Here you'll find the largest moments at the supports.



If you're looking for the biggest field moments (biggest negative bending moments) then you should choose the combination ULS FC – envelope – min. Based on these moments the (lower) longitudinal reinforcement in the preslabs is determined.



Select both preslabs (with the CTRL-key), press the SHIFT-key and select the 2 supports in the middle of these preslabs. Click on the icon 🗐 on the right top of the model window. A new window will open with a plan view of the selected plates. On the left you will find all the buttons of the 'Results'

configuration applicable for plate elements. The figure below shows the maximum shear force  $V_x$  for the envelope 'ULS FC'.



We recognize a symmetrical shear diagram relative to the beam support. Move the mouse over the drawing, below the corresponding values of the selected results will appear. When you come near the selected supports, Diamonds will snap to them. This way you can quickly obtain the desired result.

With a combination envelope the determining combination appears. You can disable this by clicking once on the button  $\square$ , this will change in  $\square$ . To show the combinations again, click again on  $\square$ .

Click 'OK'.

Once back in the model window, we click on the button  $\square$  in the pallet to show the reactions. All reactions are displayed separately by Diamonds. In this example we're interested in the vertical node and line reactions in the combination 'SLS RC envelope'. We select the line reactions  $\square \square \square \square \square$ . We only make the results visible for the back edge of the main building and double click this line. A dialog box shows the total and average reaction force on that line.

🤪 Diamonds -	BetonVB2.bsf - [Window 1 - Reaction Ry on bar - SLS RC Envelope (kN/m	1)] 🗕 🗆 🗙
Eile Edit View Select Display Analysis Option	s <u>W</u> indows <u>H</u> elp	_ <i>8</i> ×
	ⅲ 🛯 🖬 桨 挫 8 幣 ☲ ⊕ 5% ☞ 🗍 🕨 🦗	Dorien 🔻 📕 🖵
🖹 🗟 🖷 🔀 Window 1 💽 📘 R	sults 🔄 🕅 🖬 🕅 🔂 🔤 👫 🖨 🥙 🗨 🕲	
		max = 23.4 Active level
File Do Ro		27.5 + 2.70 m
A I I A	A A A	13.7 ← 2.70 m
-77.5	A A A A	6.9 🛱 Level manager
SLS RC		0.0 ← 0.0 ←
		-6.9 ←
	Average: 1 - Reaction : force Y on	- X = 0.00 m -13.7 ↔ Y = 2.70 m
No Ny Ny		-20.6 ← Representation
Rayz	total value $\Sigma = 192.8$ kN/m x m (min)	-27.5
R <sub>8</sub> R <sub>y</sub> R <sub>2</sub>	total value Σ = 254.3 kN/m x m (max)	min = -27.5
Rayz	line length L = 15.00 m	(\$\$\\$\$\$ \$\$\$
	average value $\Sigma / L = 12.9$ kN/m (min)	Size
SH SH CHI	average value $\Sigma/L = 17.0$ kN/m (max)	Font 19 🚖
5 th		Symbols 15
Have The FER	<u>QK</u>	Loads 15 🗢
M <sub>g</sub> M <sub>y</sub> M <sub>2</sub>		Results 25
		None
		Y and to accept
I		▶ z <sup>►</sup> x

Down the line we find a uniform distributed upward reaction of  $\pm 17$ kN/m. Only in the corners the plate will light up and cause downward reactions. Close this dialog.

Now select all line supports and click on  $\mathbb{R}$ . You obtain a table with the minimum and maximum values of the reaction forces on the selected line supports.

Ŷ												Result	s										-	
SLS RC	ISRC Envelope																							
bar numbe	R Fx (kN/m)	R Fx (kN/m) (max)	R Fy (kN/m) (min)	R Fy (kN/m) (max)	R Fz (kN/m) (min)	R Fz (kN/m) (max)	R Mx (kNm/m) (min)	R Mx (kNm/m) (max)	R My (kNm/m) (min)	R My (kNm/m) (max)	R Mz (kNm/m) (min)	R Mz (kNm/m) (max)	ΣR Fx (kN/m) (min)	ΣR Fx (kN/m) (max)	ΣR Fy (kN/m) (min)	ΣR Fy (kN/m) (max)	ΣR Fz (kN/m) (min)	ΣR Fz (kN/m) (max)	ΣR Mx (kNm/m) (min)	ΣR Mx (kNm/m) (max)	ΣR My (kNm/m) (min)	ΣR My (kNm/m) (max)	ΣR Mz (kNm/m) (min)	ΣR Mz (kNm/m) (max)
1	0.0	0.0	-25.6	23.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	193.0	253.9	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0
2	0.0	0.0	-28.3	67.3	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	58.3	78.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0
4	0.0	0.0	-37.6	417.5	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	137.1	173.7	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0
7	0.0	0.0	-26.4	285.6	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	206.6	257.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0
total	-	-	-	-	-	-	-	-	-	-	-	-	0.0	0.0	595.1	762.5	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0
Hel		3 🖪	4																				[	<u>0</u> K

Close this dialog and make the complete structure visible again by clicking on  $\square$  in the icon bar.

You now have seen all the available results except for the buttons below the pallet. They make is possible to view the results along a cut line. We are for example interested in the bending moments  $M_{xx}$  along a cut line parallel to the global Z-axis and through the left column of the main building. First show the maximum bending moments  $M_{xx}$  for the combination ULS FC envelope and then click on <sup>4</sup> to draw the cut line. Make use of the intelligent cursor.



The cut line will cross the beam exactly at the height of the column and will therefore be divided in two. Click on to end the drawing function and select the button  $\stackrel{\frown}{\longrightarrow}$ . The result:



Also for cutline a detailed result can be asked. Select the complete cutline (use the SHIFT-key) and click on 📓 in the icon bar.

<del></del>	- 1	Лхх	- 🗆 🗙
ULS FC	Cuttine 1 of plate 2	0 5.00 m I Cutline 1 of plate 1 - UI S FC - UI S FC 1	
8 			
D v D v	0.00		10.00 m
2.0	Mxx 0.5 -25.3	68.3	
	Mzz 1.0 -0.8	24.6 24.6	24.6 kNm/m 1.00.8 kNm/m
	Mxz 1.21.7	0.0 -0.1 -1.7	11.7 kNm/m 
	M1 2.0 -0.7	68.3 68.3	
	M2 0.5 -25.4	24.6 24.6	24.6 kNm/m -0.625.4 kNm/m
	1	1	QK

Close the window with 'OK' and click on  $\bigcirc$  to see the global results again. A cutline can be deleted using  $\bowtie$ . Obviously, this functions is only available for the selected cutline.

Now we can calculate the reinforcement and the cracked deformation.

# 3.2.6 Calculating the reinforcement

Step 27: Choosing the concrete standard

Now select the menu instruction 'Analysis – Concrete standard' and indicate you wish to calculate the reinforcement using the European standard EN 1992-1-1. We don't use a national annex [--].

#### **Step 28: Calculating the reinforcement**

Next choose the menu instruction 'Analysis – Calculate reinforcement', press on **F2** or click on the button  $\stackrel{\textcircled{\baselineskip}{\label{eq:F2}}$  in the icon bar. A windows shows you the progress of the calculations.

#### **Step 29: Viewing the results**

Getting started with Diamonds

Once the calculation has ended, the button for showing the reinforcement results will become active.

About the message 'The cross section of some elements is insufficient' If the thickness (or the effective height) of the plates or beams is insufficient to provide a valid reinforcement proposal, the following message appears:



You don't need to panic immediately with such a message. For plates, the area where there is a problem is often very local. In most cases, the reason lies in the peak moments that occur at intermediate support or in the vicinity of discontinuities, see in this case no averaging of the moments is done. Other elements, not taken into account in this calculation will void the peaks. For example the width of a support will cap the peak. Therefore always check whether a thicker plate is necessary!

For beams on the other hand a larger cross section will almost always be necessary.

The reinforcement results for line and surface elements (eg. beams,

columns, walls and plates) is represented by the same button  $\mathbb{W}^{\mathbb{N}}$ . As with all the other results, you select the desired reinforcement.

Visualize for example the lower reinforcement parallel to the local x'-as  $A_{xi}$  in the plates.



Reinforcement results are always defined locally. Consequently, we can derive the required longitudinal reinforcement in the preslabs from this drawing. The number of reinforcements bars for to the global X-direction (which is identical to the local x' direction) an isotropic plate parallel follows from the reinforcement  $A_{xi}$ .

You will notice that the triangles for which the reinforcement could not be calculated, don't have a colour. We recognize such triangles in the connection between the plate and the lower preslab. In addition, each plate which an incomplete reinforcement result, is provided with a death's head. This death's head is always drawn in the geometric centre of gravity of the plate on in the point where the problem arises!

You can find the cause of the death's head by double clicking the plate. The dialog for defining practical reinforcement will appear. At the bottom you will find the death's head again indicating the limit state for which no valid reinforcement proposal could be made. In our example we see that a solution has been found in ULS, while the check in SLS results in too much reinforcement to comply the stress control in the rare combinations.

Close this dialog with 'OK' and now select the reinforcement  $A_{xs}$  to visualize the upper reinforcement parallel to the x'-axis.



The largest upper reinforcement should be provided near the columns.

Hereafter we show the longitudinal reinforcement in the beams. Select the longitudinal reinforcement  $A_{\gamma}$  and opt for a 3D representation.



The amount of reinforcement is always drawn on the side where it is needed.

Note that next to the upper and lower reinforcement we also need web reinforcement  $A_z$ :



#### **Step 30: Eliminating the web reinforcement**

You can impose in Diamonds you only wish upper and lower reinforcement  $A_y$  (so no web reinforcement  $A_z$ ). To do so, return to the 'Geometry' configuration B and select the beams. Click on the button for the cross section properties D and then on Reinforcement. Choose for 'only upper and lower reinforcement'.



You simply need to recalculate the reinforcement for these elements. In the image below you'll find the longitudinal reinforcement  $A_y$ . There'll be no longer web reinforcement  $A_z$ .



For the further development of the exercise: make sure you have reinforcement along all 4 sides of the beams!

# 3.2.7 Calculating the cracked deformation

Step 31: Assigning practical reinforcement to the beams

Visualize one of the four reinforcements results on beams in the 'Results' configuration window . Select all beams using the CTRL-key and press the right button of the mouse once.

We provide in all beams:

- upper reinforcement: 2xØ16 (402mm<sup>2</sup>)
- lower reinforcement: 2xØ16 (402mm<sup>2</sup>)

<b>\$</b>	Diamonds - BetonVB2.bsf - [Window 1 - Ay in	n bar (mm²)]	- 🗆 🗙
💝 Eile Edit View Select Display Analysis Optic	ns <u>W</u> indows <u>H</u> elp		- 8 ×
	🏭   🚺 🖩 X & 8 🕏 🖬 🕀 🕬 🗈		Dorien 🔻 📕 🖵
😨 🗃 🗮 🛛 Window 1 💽 🚺	Results 🗾 🖗 🖽 💹 束	# Þ 《? Q Q X / / D R 2	
	A <sup>1242</sup> Practical Reinford	max = 1242 1242 931 + 621 + 621 + Cement Quantities	Active level
545			
150 <sup>2</sup> 150 <sup>2</sup> 150 <sup>2</sup>	$M_{Ay}$ $402$ mm²Max calculated :1255mm²Number of bars :22 $M_{Ay}$ $402$ mm²Max calculated :592mm²Number of bars :2 $2$	$\begin{array}{c c} \hline & & & \\ \hline & & \\ \hline & & \\ Max calculated : & 49 & mm^2 \\ Number of bars : & 2 & \textcircled{\bullet} \\ \hline & & \\ \hline & & \\ \hline & & \\ \hline & & \\ A_z & 0 & mm^2 \\ Max calculated : & 49 & mm^2 \\ Number of bars : & 2 & \textcircled{\bullet} \\ \hline \end{array}$	ound level awing plane 0.00 m 2.70 m 0.00 m entation entation €
	<u>н</u> ер 178	Cancel QK 15050 15650	s 15 € 15 € 25 € None ▼

Confirm with "OK".

## Step 32: Assigning practical reinforcement to the plates

We will now assign practical reinforcement to the plates. Visualize one of the four reinforcements results on plates in the 'Results' configuration window <sup>[]</sup>. Select both preslabs and press the right button of the mouse once.



Provide all plates of a

- upper longitudinal reinforcement: rebar mat Ø8 each 150mm
- upper longitudinal reinforcement: rebar mat Ø8 each 150mm

You either write down the corresponding amount in  $mm^2/m$  in the fill in boxer or you either click on the button  $\blacksquare$  and mark the desired rebar mat in the left column.



In this window, you'll find all the rebar mats included in the rebar mat library of Diamonds. For more information about the rebar mat library we refer to the Reference Manual. When you click 'OK' all the required information for both the x' as the z' direction will be entered. Note that also here, the amount of reinforcement bars is requested.

## Step 33: Calculation of the cracked deformation

Choose the menu command 'Analysis – Cracked deformation' of click on the button  $\overline{223}$  in the icon bar. Leave the parameter  $\beta$  unchanged and select that you wish to take creep into account.

Note: in the area for which no reinforcement proposal could be calculated, Diamonds will assume a reinforcement section that corresponds to the maximum reinforcement percentage specified with the properties of the reinforcement steel (material properties). This way you are not obligated to solve the death's heads and still obtain a realistic cracked deformation.

#### Step 34: Looking at the results

Once the calculation is finished, you will no longer find the elastic deformation but the cracked deformation under this rest formula to the formation of the 'Results' pallet. Moreover, you can now visualize the cracking widths using the button <math>rest formation to the formation the formation the second secon

Below you can see the cracked deformation  $\delta_y$  after creep for the SLS QP envelope.



Note that this deformation is about 3 to 4 times larger than the elastic deformation. This increase is partly due to creep effects and partly due to the cracking of the concrete (based on the rare combination).

Also the cracking widths need to be evaluated:



Note: In §3.4 this model will be further expanded. Hence, you should have this file.

# 3.2.8 Making a report

In this task you will be asked to make a report of this model which will illustrate the strength of the Diamonds Report Manager.

You can generate multiple report simultaneously and print them at once afterwards. This, it is possible to display the geometry and the results in a different view in the same report. It's also possible to show in each report a different part of the structure. You will be asked to define these sub reports:

- 1. A sub report with the following data
  - $\circ~$  a top view of the plate with the dimensions
  - o the cross sections properties of the beams
  - o an overview of the used materials
- 2. A sub report with a graphical 3D representation of all the load cases. Also add the load case data and data about the load combinations.
- 3. A sub report with an image of the moments  $M_{xx}$ ,  $M_{xz}$  and  $M_{zz}$  for the combination ULS FC envelope 'max' and 'min'.
- 4. A sub report with a representation of the reinforcement results  $(A_y, A_{wz})$  for all beams, one after the other, together with a 3D view, showing geometry with beam numbers.

Generate a global table of contents for these 4 sub reports.

# 3.2.8.1 Dimensions

#### Step 35: Dimensions

Before making the reports, we're defining the dimensions first. We use a grid for this. Make sure you're in the geometry configuration B. Choose

wireframe representation P in a top view  $z^{\perp x}$ 

Open the dialog window for setting the grid . Click on \_\_\_\_\_. The following windows displays:



This window allows you to define a (variable) grid that you can use for drawing a structure but it can also be used to dimension a structure. Proceed as follows:

- Give the grid a name and choose a colour (eg. grey)
- Define in which plane you'd like dimensions. This can be done in 2 ways:
  - Either fill out the coordinates for 'Origin', 'Point on x' axis' and 'Point on y' axis'.
  - Or, mark these coordinates directly in the model geometry using the mouse, by clicking the button
     Choose points in the drawing environment
     Start by clicking on the desired point of origin. Then an x'-axis will appear, following your cursor. Click on a second point to determine the direction of

the x-axis. Once the x-axis is fixed, you can analogously define the y-axis by clicking on another point.



Note: zoom out sufficiently before defining dimension lines, which makes the identification of the direction of the axes easier.

- Then you must specify in the distances you would like to see in the columns 'x' and 'y' direction. This can be done in 2 ways:
  - If the icon I is active, you will need to add point by point using the button New
     The coordinates of these points can be entered manually enter or you can select them in the model geometry.
  - When you click once on I, it changes in k, now you can select multiple points in the model geometry without needing to click New each time. Press the SHIFT-key while selecting. When you select a bar, both points (end & beginning) are included in the list. You can specify either direction (x ', y') at the same time by changing each I in k, or you can select each direction separately.



You can remove point from the list by selecting the point and clicking

- Finally, specify how you wish to number the directions and whether the numbers or annotation lines should be visible in the corresponding direction.
- In the image below only the annotation lines are visible.

-	Diamonds - BetonVB2.bsf - [Venster 1 (m)] – 🗖 🗙																												
۴	🖗 <u>F</u> ile	<u>E</u> dit ⊻i	ew Select	<u>D</u> ispla	<u>An</u>	alysis	Opt	tions	Wind	dows	Hel	р																	- 8 ×
	D 🗳		l 🗧 🚺	n l	Cir	F##	HA	R ∷	:	T.:	$\boxtimes$		ĽĽ	6 6	8	: 111	9 ا	% I			<b>正   </b>	Ð							
	* 5		Venster 1		1	-	1	User	config	uration	1				•	R	<b>FII</b> [	0		\$A   \$	∍∣	<u> </u>	<b>Q</b>		Ø		DR		
			68	28 - 38	8	18	21	10	10	65	101	92	38	35	2	38	33	10	101	48	12	98	22	38	32	18	81	12	Active level
	. р ОШ	,	56	8 8	X	,	5.0	)0 r	n	1		5.0	0 n	n			5.0	)0 n	n			5.0	10 n	n	$\downarrow$	22	24	8	Level 1
	<u></u>	• /							5	2			1			a	52	N	a	M									2.70 m
	Î d	00	25		N N		~			<b>N</b>	200			1231			123		<b>3</b> 63										2.70 m
	5	UN UN		E	N IN																		10				04		PER 1
	4 ª	₹⁄	1	. 9								1	9746							CE CE	15	32	87	17	86	12	35	14	Level manager
ļ		*		5.0	XX							ų	-							THE R	88	98	8	33	22	63	12	10	Ground level
	44	해 한문		22																	8	85	22	12	33	22	8		Drawing plane
	1.	" p.:	) a	1	- 20																_	_	_	_		28	83	3	X = 0.00 m
	*	174 (* )		-	X															88						-25	14	æ	Y = 2.70 m Z = 0.00 m
	Dijî 👷	M 75	-	Ξ	B							0	>							**									Representation
	HT &	2 4		00	No.							8								00 00						25	83 		AAA
		Fe 🚵		, ro	X							Ų	۶."													10	83	9	
	wy e	50 IN 2	ų a	8	E A															848						- 22	8	8	S. S
	*~~~ ^	° V	×				×	⊠				Ø			82	***	***		***	#麗						83	12	13. <sup>17</sup>	Size
		7 []e		81	32	18	84	87	23	57	84	22	23	82												28	83	8	Font 25 🚖
*	TR.		-	5	86	-	82	iii.	25	-3	12	2	53	12												-	32	14	Symbols 15 🜲
1			_	d	a.			24			2.4	~	12	-															Loads 15 🚖
				CJ																									Results 25 🜲
			10	10	33	\$8	8	8	8	53	8	22	23	82												58	23	8 -	Show groups
			8	\	5	22	82	8	20	55	12	ŝ	28	-		-										22	82	3	None
				8 8	88	53	29	88	83	-0	88	8	80	88. 1	8	80	38	8	83	88	8	35	30	<i>8</i> 2	38	55	84 T		angen angen
_				N 32	35					e				-					- 01						35		.,	z	

The dimensions are only asked in the first report, not in the others. So you have two possibilities:

- Either you set the grid **visible** and you turn if off in the report manager when you don't want to see it.
- Either you set the grid **invisible** and you turn it on in the report manager when you want to see it.

Since the last option demands the least work, we'll choose this on. Set the grid invisible.

Diamonds - BetonVB2.bsf - [Venster 1 (m)] – 🗖 🗙													
🧇 File Edit View Select Display Analysis Options Windows Help	- 8 ×												
1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1													
😨 🗟 🗟 Venster 1 🔄 🚺 User configuration 💽 📾 🖂 🔯 🏚 🖓 🔍 🍭 📓 🥒 💽 🗷 🦉													
	Active level												
<b>∧</b> ₩ ,													
$\overline{\mathcal{A}}$	2.70 m												
	2.70 m												
😒 🧶 General grid													
Grid Spacing													
	Ground level												
^> Mi (2) a s a 🛱 s a a s a a s a a	Drawing plane												
<i>I J β β β β β β β β β β</i>	X = 0.00 m Y = 2.70 m												
The second secon	Z = 0.00 m												
	Representation												
M & A	<b>A A</b>												
To the second se													
	Size												
	Pont 25 😴												
	Symbols 15 🚖												
n e a a a a a a a a a a a a a a a a a a	Loads 15 🜩												
	Results 25 🚖												
Help <u>Cancel</u> <u>QK</u>	Show groups												
	None 💌												
	-X -												

## 3.2.8.2 Sub report 1: Geometry

## Step 36: Sub report 1

Choose the menu command 'File – Report manager' or click on the icon to open the report manager.

Let us first define the page setting. Click on Page setup and complete the dialog window as follows.

<del>\$</del>	Diamonds - BetonVB2.bsf - [Venster 1 (n	Page Setup ×
Image: Select Display Analysis Optic         Image: Select Display Analysis         Image: Select Display Analysis         Image: Select Display Analysis         Image: Select Display Analysis         Image: Select Display Analysis      <	ins Windows Help   III   Tui   図画 光光 た 啓 亚 ① F% ゴ Jser configuration	Margins [10,0 mm [20,0 mm]
	Papate manager	10,0 mm
		Orientation       Orientation
	Modify Copy Delete	Font Arial
 夕田健 夕時PH 商歴へ		Font Size         P         Title 1         18           Header         7         Title 2         14           Footer         7         Title 3         12           Title report         22         12         12
IT Q Q Import configuration	Sin Export configuration All	Logo
	Preview 🛓 RTF-file	Width 100%
Help	<u></u> Cancel	left : None  middle : None  icht : None  Set-up
		Footer left : None
	8 8 6 8 8 8 8 8 8 8 8 9 8 8 9 8 9 9 8 8 8 8 8	middle : None Use right : Page number : start at 1 Set-up
		Help Cancel QK

Make use of the option 'logo' the import your own logo into the report.

Provide a header and footer on each page. Check the advanced settings on the right. Click on the button <u>Set-up</u> and note the title of the document in right field. Here you can add your logo if desired.

Advar	nced header and footer configu	uration	×
			7
Lett	Middle Add date	<u>Right</u>	
	Add file name	Add file name	-11
Add file name with path	Add file name with path	Add file name with path	-111
Add page number	Add page number	Add page number	-111
Add logo	Add logo	Add logo	-
^	^	Calculation of a floor	
< >	< >	<	
Help		<u>C</u> ancel <u>O</u> K	

Click on 'OK'. For the footer we choose to show the page numbers on the right, starting from '1'.

Confirm the page set up with 'OK' and click in the button **I** to start composing the first sub report.

9	Diamonds - BetonVB2.t	Print report ×
Eile Edit View Select Display Analysis Options	Windows Help	
	። 🚺 🖩 🦹 🗽 ሪ 笔	Name : Geometry
🗑 👼 🛛   Venster 1 💽 🚺   User	configuration 🔄	General Geometry Loads Global results Detailed results
$   \mathbf{x}, \times  $		
∩⊞ / ↔	Reports manager	Selection and view
	( <b>–</b> (	<ul> <li>all visible elements</li> <li>selected elements</li> </ul>
	1odify ECopy	🕅 👯 🌄 5:3 Modify selection and view
+7 B,Ĵ ₽+:		
▲ <b>护</b> ⊗		
Timport configuration	Export configuration	General information
E Str. E	Page setup	Title report
		I✓ Table of contents
💮 📊 🚺	A Preview	add to previous table of contents, with continuous pagenumbers
		V Paragraph numbering Start at 1
Help		Contractor
		Structural engineer     Principal
	* * * * * * *	
		Insert solution options
8 6 8 8 8 8		jinser claicula uon opuons
		Help Cancel OK

Click on the button it to open the grid settings. Set the grid with the dimensions visible.

<del>\$</del>	Diamonds - BetonVB2.bsf - [Venster 1 (m)]	- •	x
	<u>W</u> indows <u>H</u> elp		7 ×
	ⅲ┃┓┓⊻ѡѳ҄๛ҧ⊕҄҂҂│⊫ҏҏ		
😨 🗟 🗮 🛛 Venster 1 💽 🚺 Us	er configuration 💽 🖬 🖂 🕅 庄 🖑	Q Q X / ■ E Z ■ ■	E
K Y Y Y Y Y Y Y Y Y Y Y Y Y Y Y Y Y Y Y	🕨 🚽 🚽 Grid settings 🚽 🗖 🗡	Active level	_
∩ <b>⊞</b> . /			-
	General grid		n
	Crid Spacing	2.70	n
🖓 🧶 🐘 🖁	© On	Evel mar	ager
	C Off X 1,0 m		
4/ HH 24	-Directory Y 1.0 m	the Description	vel
	Visible     Z     IO     T	X = 0.00 m	n
τ <sup>2</sup> R <sub>+</sub> J <sup>-1</sup>	C Invisible	Y = 2.70 m 7 = 0.00 m	1
MI W /		Representation	e K
	Vaviable evid	B.A.	R
A C C C C C C C C C C C C C C C C C C C	variable grid	Set Set	s.
	Variable grid		
		Size	
the m 112		Font 25	2
		Symbols 15	
	New Modify Delete	Loads 15	
		Results 25	١
	July Consul Consul	Snow groups	-
	Heip <u>C</u> ancel <u>O</u> K		-

Turn back to the report manager with 'OK'. Click on the button , opt for a top view.



Check it the elements are depicted as desired. If that's not the case, follow the advanced procedure in the grey box below.

#### About the option 'Modify selection and view'

Regardless what you can to display (geometry, loads or results) in the calculation note, you have to fix the viewing point (top, front or side view) in the first tab. In addition you indicate for which elements this sub report applies. How can you do this? Well, you have two possibilities:

- Using the button Modify selection and view you return to the model window of the active configuration where you change the orientation and visibility of the elements as you please.
- Via you can choose from the default view in Diamonds. Diamonds will centre and display the selection as large as possible in the window.

At the bottom of the 'General' window, check 'Report Title' and 'Table of contents'. The title which will be used is the name of the report, namely 'Geometry'. The option 'Table of contents' makes a new table of contents for this report. The paragraph numbering should be checked.

<del>9</del>	Print report	×		- 🗆 🗙
🥮 Eile Edit View Select Display				- 6 ×
D 📽 📕 🖾 🎒 🗠 🗠	Name : Geometry			
🛞 🗟 🖷 🔀 Venster 1	General Geometry Loads Global results Detailed results	L)	D R 2	
<b>K</b> X 2 2 2 3	Colortian and view	1	8 3 3	Active level
<u>∩</u> ⊞. / @	Selection and view		8 3 8	
	all visible elements     C selected elements		a 18 14	12.70 m
	Modify selection and view		N 16 18	2.70 m
		6	и и и	Evel manager
<u> </u>		2	a n n "	Ground level
夕照贺			8 8 8	📇 Drawing plane
₽ B 戸北		3	8 3 3	X = 0.00 m
N. 17 (2)			a a a .	Z = 0.00 m
			8 37 38	Representation
	General information		8 8 8	
	Intereport		8 8 8	D 🔊
I I I I I I I I I I I I I I I I I I I	add to previous table of contents, with continuous pagenumbers	a		Cira
	Raragraph numbering			Font 25 📚
			8 13 16	Symbols 15 🚖
Help	Structural engineer			Loads 15 🜲
	Architect			Results 25 🚖
	Insert software reference			Show groups
	Insert calculation options	3,		None
K 6.2			-×	
	Help Cancel OK	1 -	ź	
				11.

Now select the second tab 'Geometry' and indicate you wish to see the geometry graphically.

9	Print report ×	- 🗆 🗙
<u> </u>		- 8 ×
D 📽 🖬 🖾 🎒 🗠 🕾	Name : Geometry	
😨 🗟 🖷 🔀 Venster 1	General Geometry Loads Global results Detailed results	
K, X · · · ·		Active level
<u>∩</u> ⊞. /	Show geometry	
	Configuration Other configuration I	2.20 m
	Insert geometry data in table	
🛛 🖓 🥮 🔹 v is is is		Level manager
<u> </u>	Table parameters	Ground level
夕風登している		J Drawing plane
-7 85 P÷	Insert cross-section data	X = 0.00 m
▲	Insert connection data	Z = 0.00 m
	Insert material data	Representation
	Insert soil laver profile	
∰s <sup>e</sup> in		🖗 🏘
		Size
		Font 25 🚖
		Symbols 15 🚖
		Loads 15 보
75 84 38		Results 25 🜲
		Show groups
		None 💌
H H (H		<b>7</b> ×
	Help Cancel OK	1. 1. 1. 1. 1. 1. 1. 1. 1. 1. 1. 1. 1. 1

The viewpoint is fixed, but the representation method hasn't. Well in the adjacent pull down menu indicate the configuration according to which the model should be depicted. We wish to display the geometry of the model, so select the 'Geometry' configuration. Remember that a lot of model information is related to a configuration. Optionally, you can change the settings of this configuration via the button **1**.



Finally implement the cross-section and material data in the report and click 'OK'.

9	Print report	×			- 🗆 🗙
<u>File Edit View Select Display</u>		_			- 8 ×
D 📽 🖪 🖪 🚳 🖗 🗠 🗠	Name : Geometry				
😿 🗟 📇 🔀 Venster 1	General Geometry Loads Global results Detailed results				
$\mathbf{k}$ $\times$ $\mathbf{k}$			2 10	a a	Active level
<u>∩</u> ⊞. /	Show geometry		8 8	8 8	
	Configuration Other configuration 🗾 🚺				2.70 m
			- 25	8 8	2.70 m
2 🗶 🖉 👘 🗤 👘 🖉	) Insert geometry data in table			55 BC	Evel manager
<u> </u>	Table parameters		81	19 DE -	Cround level
夕田 建二十十十			- 22		Drawing plane
, J _ = = = = = = = = = = = = = = = = = =	✓ Insert cross-section data		1	8 8	X = 0.00 m
it in⊅ It it	Insert connection data		-	a 8	Y = 2.70 m Z = 0.00 m
	✓ Insert material data				Representation
	E Tagert seil laver grafte		-	n a	888
∰s <sup>e</sup> È			-	8 8	D 🔊
			1		Cine
					Font 25 🜩
					Symbols 15 🚖
					Loads 15 🗢
			60	10 M	Results 25 🕏
					Show groups
			3	a a	None
K K A		<u>'</u>	5 B)	-×	
<u> </u>	Help QK		5 N	, Z	1

The first sub report is defined. Next we will make a sub report from the loads.

# 3.2.8.3 Sub report 2: Loads

## Step 37: Sub report 2

Click on ا 🕅 New

- Name this sub report 'Loads'.
- With the button you can change the scale of the images. Opt for
- Change the orientation of the plate to a perspective view with the button .
- Join the table of contents from this report with the table of contents from the other reports. For this purpose, check the option 'Add to previous table of contents, with continuous page numbers'.

Image: Select Display Analysis Ontions Window	Viamonds - BetonVB	Print report	×
	⊠ II X & &	Name : Loads	٦
😨 🗟 🗮 🛛 Venster 1 💽 🚺 User configurati	ion	General Geometry Loads Global results Detailed results	
		Selection and view	
	Reports manager		
New Modify	Copy	Modify selection and view	
✓ ● Geometry ✓ H 20 ✓ B 5 P 4 前 北 20			
Ho & M I Import configuration	Export configuration	General information	
The setup	Page setup	✓ Table of contents	
Print	A Preview	✓ add to previous table of contents, with continuous pagenumbers         ✓ Paragraph numbering         Start at	
Help		Project data     Contractor     Structural engineer     Principal	
	а а а а ( <b>)</b> а а а а ( <b>)</b>	Architect     Insert software reference     Insert calculation options	
		Help Cancel QK	

Go to the tab page 'Loads'.

- Indicate that you want to view the loads graphically.
- Choose the 'Loads configuration'. Optionally, you can change the settings of this configuration via the button 1.
- Then select the type 'Load group' and select the 3 load groups.

<b>9</b>	Print report	- • ×
<u> </u>		- 8 ×
D 📽 🖬 🖪 🗿 🎙 က က	Name : Loads	
😨 📴 🔀 Venster 1	General Geometry Loads Global results Detailed results	
A R R R		Active level
<u>р</u>	I✓ Show loads	Level 1
	Configuration Loads I	2.70 m
100	Load group	2.70 m
	Selfweight	
	✓ dead loads ✓ life loads A : bousing	Level manager
		Ground level
夕田 役		J Drawing plane
⊅.≒p.:		X = 0.00 m
* 114 -		Y = 2,70 m Z = 0,00 m
Ni 21. 75	Tipsert loads data in table Table parameters	Representation
		AAA
APPT - Fe are	Selfweight	
III) St. P	dead loads	🖓 🏘
		Size
R R II		Font 25 🗢
		Symbols 15 🕏
		Loads 15
95 (2 3)	✓ Insert load case information	Partite 25 A
10 N N N	✓ Insert combinations information	Kesuits 25 -
16 BK 88	Insert generated loads	Show groups
	j index generated 10005	, , , , , , , , , , , , , , , , , , ,
		Ţ <sup>X</sup> → 50°c
	Help QK	

There is also asked to implement the data on the different load groups and the composition of the load combinations in the report. Therefor select the penultimate and final box. Then click 'OK'. The second sub report is defined.

## 3.2.8.4 Sub report 3: Global results

Step 38: Sub report 3

Now, create a new calculation note with **I** New .

- Name this report 'Global results'.
- Choose a top view with the button 1.
- Join the table of contents from this report with the table of contents from the other reports.

Diamonds - BetonVB2.t     Elie Edit View Select Direlay Analysis Ontions Windows Help	Print report ×
Image: Select Display Analysis Options Windows Help         Image: Select Display Analysis Options Windows Help         Image: Select Display Analysis Options Windows         Image: Select Display Analysis Options         Image: Select Display Analysis Options <th>Name :       Global Results         General       Geometry       Loads         Selection and view</th>	Name :       Global Results         General       Geometry       Loads         Selection and view
Import configuration	General information ✓ Title report ✓ Table of contents ✓ add to previous table of contents, with continuous pagenumbers ✓ Paragraph numbering Project data
	Structural engineer       Principal         Architect:       Insert software reference         Insert calculation options         Help       Cancel

Go to the tab page 'Global results'.

- Indicate that you want to view the results graphically.
- Choose the 'Results configuration'. Optionally, you can change the settings of this configuration via the button 1.

9	Print report	_ 0 ×
🥮 <u>F</u> ile <u>E</u> dit <u>View</u> <u>Select</u> <u>D</u> isplay <u>A</u> nal		- <del>6</del> ×
	Name : Global results	
😵 🗟 🖷 🛛 Venster 1 🗨	General Geometry Loads Global results Detailed results	
K <sub>p</sub> × <sup>n</sup> <sup>n</sup> <sup>n</sup> <sup>n</sup>	Show dobal results	Active level
	Configuration Results <u>i</u>	2.70 m
		* 2.70 m
<u>₽♥</u>		🛱 Level manager
	Load group	Ground level
夕田健	dead loads	Drawing plane
<i>₽</i> 85 P4	<sup>γ</sup> γ φ <sup>γ</sup> φ <sup>γ</sup> γ φ <sup>γ</sup> φ <sup>γ</sup> γ φ <sup>γ</sup> φ	X = 0.00 m Y = 2.70 m
NA AR A	¥ ¥	Z = 0.00 m
	т. Ф	Representation
	All	
	☐ Insert min-max results in table Load group ▼	🕸 🏘
	Selfweight	Size
the mile	V V V V V V U U U U U U U U U U U U U U	- Font 25 🕏
Į		Symbols 15 👤
× a a	N 12	Loads 15 🜩
51 50 54 💻		Results 25 🜩
		None V
K K K K	J Insert equilibrium check	
	Help Cancel OK	

Next you select which results should be implemented in the report:

- Click on 2.
- Indicate the bending moment  $M_{xx}$  from the pull down menu below.
- Select both envelopes (min and max) from the list with ULS FC combinations.



Repeat this procedure for the bending moments  $M_{zz}$  and  $M_{xz}$  (so immediately select a new force from the pull down menu and indicate which combinations should be displayed) and click 'OK'. In total, this sub report will contain 6 images.

Select the option 'Insert equilibrium check'. This will give an overview of all the (vertical and horizontal) loads and the (vertical and horizontal) reactions for each combination.

<b>\$</b>	Print report ×	<b>-</b> ×
💝 Eile Edit View Select Display Anal		- 8 ×
D 🖉 🖬 🗟 🎒 🔊 🖓 🛛	Name : Glboal results	
🗑 🗃 🗮 🛛 Venster 1 💽	General Geometry Loads Global results Detailed results	
$\mathbf{x}$ $\times$ $\mathbf{x}$ $\mathbf{x}$		Active level
<b>∩</b> ⊞. ∕	V Show global results	
	Configuration Results 1	× 2.70 m
		2.70 m
	*8* AAA IAA IAA AAA AAA AAAA AAAA	Level manager
	(Haz) (Haz) ELU CF	
14 HA 201	(H) (H) (Y) Enveloppes min	the Desuise share
2 · · · · · · · · · · · · · · · · · · ·	ULS FC 1	X = 0.00  m
τ <sup>×</sup> 8,J Γ**:	Number of the second se	Y = 2.70 m
NT 2/14 /2	(N1) (N2) (X2) ULS FC 4 ULS FC 5	Representation
	Insert min-may results in table Groupe de charges	🕸 🏘
**** / V = + + + + + + + + + + + + + + + + + +		Size
	I alead loads	- Font 25 文
<u> </u>		Symbols 15 🚖
	5 Y 1/ V V V V V	Loads 15 文
		Results 25 🚖
		Show groups
	▼ Insert equilibrium check	None 💌
11 B. A. H		
	Help QK	

Finally click 'OK'. The third sub report is defined.

# 3.2.8.5 Sub report 4: Detailed results

#### Step 39: Sub report 4

In a final report should contain a detail result of the reinforcement in the beams together with a perspective view showing the geometry with the line numbers.

Now, create a new calculation note with

- Name this calculation note 'Detailed results'.
- Click the button Modify selection and view to record the orientation of the model. Then choose a perspective view and just make the beams visible through the function  $\mathbb{H}$ .
- Turn back to the report manager with Click here to return to print report configuration .



- Implement this sub report in the table of contents.

*	Print report	- • ×
<u>File Edit View Select Display</u>		_ 8 ×
D 🛎 📕 🖪 🚳 🕅 🗠 🗠	Name : new report	
🐼 🔂 🖶 🔀 Venster 1	General Geometry Loads Global results Detailed results	
$\mathbf{k}$ $\times$ $\mathbf{k}$		Active level
∩⊞. ∕	Selection and view	
	all visible elements     O selected elements	2.70 m
	da ::: 下。 5:3 Modify selection and view	2.70 m
2 🖤 🖉 🖉 🖉		Evel manager
		Ground level
夕田 建二十十		🖽 Drawing plane
₽ 85 P4		X = 0.00 m
1. 1. 1. 1. 1. 1. 1. 1. 1. 1. 1. 1. 1. 1		Y = 2.70 m Z = 0.00 m
		Representation
	General information	\$ \$ \$
m et in	Title report	Ents An
Z. / />	✓ Table of contents	
	add to previous table of contents, with continuous pagenumbers	Size
the mille	Paragraph numbering Start at 1	Font 25 📚
E,	🗖 Project data 🦳 Contractor	Symbols 15
	🗖 Structural engineer 📄 Principal	Loads 15 文
	Architect	Results 25 🚖
	Insert software reference	Show groups
	Insert calculation options	None
45 56 62		<b>, ,×</b> , , , , , , , , , , , , , , , , ,
	Help QK	

Now select the tab 'Geometry'.

- Check the first selection box.
- Then click on the button 💷 and select the line numbers in the tab page 'Geometry'.

<del>\$</del>	Print report	
Image: Select Display Analysis Options Windo       D     D       D     D       D     D       D     D	Name : Detailed results	- 8 ×
😵 🗟 🗮 🛛 Venster 1 🗾 📘 User configure	Genera Geometry Loads   Global results   Detailed results   0	🔤   🔳
	Image: Show geometry]     Image: Show geometry]	
	Configuration Other configuration	2.70 m ∷≱ 2.70 m
General Geometry Mesh	ameters	evel manager
Points     Lines       ✓ finit     ✓ finit       ✓ finit     ✓ finit	Surfaces Transports Surfaces Transports Surfaces Transports Surfaces Transports Surfaces Transports Surfaces Surfac	round level awing plane 0.00 m 2.70 m 0.00 m entation entation 25 ¢ 15 ¢
Help	Cancel OK	roups
	<u>H</u> elp <u>Cancel</u> <u>QK</u>	

Confirm with 'OK'.

Now activate the last tab 'Detailed results'.

- Indicate that you want to view the detailed results graphically.

- Click the button and select the longitudinal reinforcement  $A_y$  and shear reinforcement  $A_{wz}$  from the pull-down menu located below. Check each time the cross check box on the right side.



Click 'OK' to close this window. All four sub reports have been defined. The report is ready to be printed.

Diamonds - BetonVB2.bsf - [Venster 1 (m)]					
🥮 Eile Edit View Select Display Analysis Options Windows Help		- 8 ×			
□☞ ■ 집● ◎ ∽ ∽ 田田田 ※ ■ 図圖光波 6 窓五 9 19 ☆   F■ ◎					
😨 巻 🕾 Venster 1 👤 🚺 User configuration 🔹 🍘 已 🗑 市 💹 👌 🖑 Q Q 🗃 .					
🗙 🗙 sensensensensensensens	2 A A	Active level			
AH	8 B B	Level 1			
	40 79 00	2.70 m			
		2.70 m			
🔊 🧶 🖉 🖉 New 📝 Modify 📳 Copy 👔 Delete 🎒 Delete all	10 IA IA	Evel manager			
Geometry					
Contraction of the second sec		tt Drawing plane			
✓ Detailed results		X = 0.00  m			
T <sup>*</sup> B <sub>2</sub> J [ <sup>-7</sup> ]		Y = 2.70 m			
m M B	N 19 19 19	Representation			
	16 27 28	AAA			
Import configuration	8 34 58	S. S			
A d d 🖉 🍯 Print setup	lan na 🕅	Size			
	5 2 2	Font 25 🚖			
Print Q. Preview 🔄 RTF-file 🔻	10 H H	Symbols 15 🚖			
	41 17 14	Loads 15 🚖			
<u>H</u> elp <u>QK</u>		Results 25 🚖			
	1	Show groups			
		None 💌			
	, <b>r</b> ×				
	5 3 L 4				

# 3.2.8.6 Preview

## Step 40: Preview

At any time you can make preview of either the entire calculation note either the selected sub report using the button **Q** Preview.

Once a calculation note has been properly prepared, you can print it by clicking the button Print. Also you can write the calculation note to an RTF file RTF-file so that it later can be opened / edited with a text editor.
## 3.3 Example 3: A foundation slab

Required licenses:

- ✓ 2D Bars✓ 2D Slabs
- ✓ Concrete Design

We now proceed to the calculation of a general foundation slab on soil. We assume no excavation, i.e. the foundation level equals the ground level. Furthermore, a cone penetration test has been performed which gave us these soil properties:

Layer	С	Α	OCR	СС	$\gamma_d$	$\gamma_n$
thickness [m]	[-]	[-]	[-]	[%]	[kg/m <sup>3</sup> ]	[kg/m <sup>3</sup> ]
1	120	180	1,0	0,0	1600	2000
3	350	525	1,0	0,0	1600	2000
4	260	390	1,0	0,0	1600	2000
10	340	510	1,0	0,0	1600	2000

- The compression constant *C* and the recompressibility constant *A* are derived from the cone resistance.
- The overconsolidation ratio *OCR* takes into account the effect of preloading (or overconsolidation) of the soil.
- Drainage ratio *CC* is a measurement of how much of the pore water is already drained.
- $\gamma_d$  and  $\gamma_w$  are the dry and wet density of the soil.

The groundwater level is 1m below the ground level.



In the floor pan above the lines correspond to the lower edge of the walls. The column bases are represented by points. The foundation slabs has and overhang of 0,5m relative to the walls and the columns. The thickness

of the plate is 30cm and again we use a concrete quality C25/30. The gross cover is 35mm, for both the upper and lower reinforcement.

We assume the following data concerning the loads:

- Each wall transfers a uniformly distributed vertical line load of 85kN/m on the foundation slab as dead load (including the self-weight of the upper structure), and 10kN/m for life load.
- All central columns bear a point load of 600kN on the foundation slab as dead load (including the self-weight of the upper structure) and 100kN for life load.
- All central columns near the edges bear a point load of 60kN on the foundation slab as dead load (including the self-weight of the upper structure).

## **3.3.1 Defining the structure**

#### Step 1: Start a new project

Start a new project using the menu command 'File – New' or click on  $\Box$ .

#### Step 2: Go to the 'Geometry' configuration

Defining the structure is always done in the 'Geometry' configuration. Click on in the icon bar, or select the 'Geometry' configuration in the adjacent pull down menu.

Then check if you are in a top view. If this is not the case, then click on the

button  $\blacksquare$  in the icon bar or on the button  $z^{\perp x}$  in the lower right corner and select the viewpoint "Top view". This way you activate a horizontal drawing area.

-	2													Dia	mon	ds -	[Ven:	ster 1	(m)	]									- 0	×
	🖗 <u>F</u> ile	<u>E</u> dit \	/iew	Sele	ct 👖	<u>)</u> isplay	/ <u>A</u>	nalysis	s Or	otions	Wir	ndows	Help	þ															-	- E ×
	🗅 🖨		<u>a</u> e	\$ (	D	к	CH	F	H	₽ :	III	Tei			X Ke	6	de	201	5) Fe	ľ	ļ		49							
	* 5		Vens	ter 1				-	1	Geo	metry						•	6	<b>1</b>		- 1 1 1 1	臣	<b>«</b> ۳	Q	Q 🖾		D [	R IP (		- E
Î	٨.							10	•2														10	-0	10.				Active level	
	ΛĦ	. /	•		8		<b>b</b>			8	8		8			2				8		8	18			8			Toundation	
	<u> </u>					v	- /	×																					FILI 0.	00 m
	Ū	l l				z						04 		Ċ													og og	0 <b>7</b>	<b>□□1 *</b> 0.	00 m
		<b>\$</b>			0	20	28	10	23	81	84	35	355	10	12	8	82	8	98	0	0	22	25		10	3	a a	23	F Level r	manager
		2	-		5	28	22	15	28	83	12	38	35	10	13	2	82	8	98	10	101	22	38		2	1	Top vie	N	7777 Croun	d lough
	4	(H 3)	-8		20	10	87	-88	-35	85	85	10	122	14	14	8	86	8	10	35	30	97	99	1	z.L.x	í.	· ·		Drawin	o plane
	1	, # P.			21	87	87	83	23	35	35	12	39	15	16	96	86	12	12	21	20	10	87						X = 0.0	00 m
	1 I	12 I 14 C			8	89	32	-85	-83	8	25	95	25	8	8	2	8	R	50	8	28	32	32		Z.L.	, I	Front vi	ew	Y = 0.0 Z = 0.0	00 m 00 m
	Injei nj	81.72			8	10	ĸ	88	13	24	28	8	-	8	8	12	12	8	33	8	33	8	10						Representa	tion
	IT	Z 4			8	10	16	10	-02	59	19	őł.	10	8	8	ж	5	æ	32	8	83	35	15	J	A P		Back vie	w		1
	節	Se P	Ĵ.		22	10		-53	-	54	124	<i>8</i> 7		æ	22		3 <b>7</b>		30		33	-	-		Zelax	i l			<b>A</b>	
	$\overline{\lambda}_{a}$	10	>																					4			eft vie	v		-
	=	₹ŭ.	0																						z.L.X				Size Font	25 🜲
	-13(e -77	M 7.	-		31	53.	8	-51	-11	82	82	155	155	10	36	8	83	15	25	22	32	63	63						Symbols	15 \$
1	<del>(</del> )				61	201	25	14	14	89	89 1	12	18	10	10	85	85	8	00	<u>\$1</u>	10	25	25			۱ ا	Right vi	ew	Loads	15 🜲
			8		22	25	25	53	54	85	85	88	82	12	85	8	8	8	8	50	63	25	25		2***X				Results	15 🜲
			2		13	28	52	53	53	8	82	82	82	8	8	35	32	25	20	8	18	22	28		3	וי	3D view		Show group	IS
			5		8	28	52	53	53	22	8	82	82	8	2	35	35	22	22	8	15	53	20		Z	8			None	•
			3		8	18	75	10	33	8	8	84	82	8	8	22	10	12	12	2	8	25	18	33	8	8	8 8	<b>F</b> ×		41 H
			-																									Z		
-																														11.

**Step 3: Setting the level manager** 

It doesn't matter at which level (= global Y coordinate) you draw the structure, but because we want to recover the foundation plate in §3.4, we'll define levels correctly.

Press the button *Hevel manager*. Select the 'Level 1' and click on 'Delete'. Click 'OK' to close the window.

<b>\$</b>	Diamonds - [Venster 1 (m)]	- 🗆 🗙
🥮 Bestand Wijzig Scherm Selecteer Toon	Analyseer Opties Venster Help	_ & ×
		Dorien 🕶 📕 🖵
🗑 🖥 🔀 Venster 1 🛛 🔹 📔	1   Geometrie 📩 📾 🖽 💹 📩   🏦	⊉   १७ Q Q X   ∥   ■ ■ ⊠ 2   ■ ⊠   =
<b>N N A B B B B B B B B B B</b>		Actief niveau
<b>`р</b>		fundering 💌
		naan xaan xaa naa 🕂 🛄 🛄
	Niveau beheer	×
$\odot \square $		
		🛱 Niveau beheer
	Nieuw Splitsen Kopiëren Verwijderen	
Z. 11 ML A REAL REAL		Maaved
		Ē⇒ Tekenvlak
· 27 승두 만나 이 이 이 이 이 이 이 이 이 이 이 이 이 이 이 이 이 이		X = 0.00 m
	R X Vertering 1 2 70 m	7 = 0.00 m
NIC 2/T /~	Verdieping 1 2.70 m 2.70 m	Voorstelling
17 6 Ab	🖉 🏷 foundation 0.00 m 0.00 m	
m et a sesse		an sean sea sea 🔊 🚠 🌨
	1	
😤 🖉 🖉 - а кназа кна		Grootte
	Help Appuleer OK	Lettertype 10 🚖
<u>Ale M 447</u>		
		Symbolen 10 হ
		Lasten 5 🕏
		Resultaten 10 🚖
10 40 96 06 10 96 08		
		Toon groepen
		Geen
	法法保证 法法保证 建成体的 化化合合 化化合合 化化合合	

#### Step 4: Changing the step of the grid

Click on <sup>III</sup> and change the default step to 5m. Confirm with 'OK'.

9					Diamonds - [\	/enster 1 (m)]							- 🗆 🗙
🥮 Eile Edit Vie	w <u>S</u> elect	<u>D</u> isplay	Analysis	Options	Windows <u>H</u> elp		en -						- 8 ×
	160	$\kappa \gamma ~~ (\gamma )$	E E	Im	: N 回 X 秒 6	\$\$ <u>211</u> (D) F% IV		40					
🛛 🗑 🖲 🖾 🛛	/enster 1		•	Geo	etry	• 🖗 😐 🖾 🖬	南臣	ধ্যু 🖻	. 🗨 🖾		D		
			10						0 10	0. 0			Active level
			v .		🤪 Grid se	ettings 🗕 🗆	×			8.8		84	
~ ·			~										
		z			General grid								0.00 m
S 🖗		10 IS	15	10 01	Grid	Spacing		20 1	10 10	8 <b>2</b> 8			Evel manager
	0 0	20 - 30;	61	15 10	© On	x 500 m		31	81 - 81	84 8	1 88	88	
小丽型	20 20	8 8	-25	25 SA		x 1.00 m		8	8 8	8 <b>.</b> 8	10	10	
27 . 5 P.	21 21	w = w	88	41 - 14	<ul> <li>Visible</li> </ul>	7 5.00 m		8	6 - K	34 - 32	- 12	10	X = 0.00 m
τ κ, μ. τ . 	8 8	82 82	35	8 B	C Invisible	- <u>1975-1</u>		ю -	8 8	85 8	: 05	95	Y = 0.00 m Z = 0.00 m
<u></u>		w w	83	8 8				ю.	8 8		a si	- 10	Representation
		15 15	12	en 16	Variable grid		_		a a	54 S		ä	888
育 🖉													6 A
In I On		36 36	201					30 1		38 - 33	e 08	o:	
	34 30	0.0	61	61 19				60 I	61 61	98 - 9	13	13	Size
<u>Me h 112</u>	32 32	0 0	-9	-s (e		1		8	si -si	8 <b>*</b> - 8	· 35	2	Symbole 15
- <del>(</del> *)	0 0	8 8	12	93 - 59	<u>N</u> ew Mod	lify <u>D</u> elete		8	14 IV	87 - 8	12	-12	Inads 15 +
	0 0	8 8	8	8 <i>8</i>	Ush	Creat 1		28	a 14	8 <b>7</b> 8	: *	*	Regults 15 A
	8 8	51 - 52	12	s. 8			N	10	a 15	85 - 83		8	Show groups
	8 N	0.0	52	52 AT			N 10	12 1	a a	s: s			None 💌
		R 8	10				2 2	18		8 8		X	17일 27일
												z	
													11.

Step 5: Drawing the lines and points on which loads will act

Using the button draw all construction line to which a lines load will be assigned. All segments have a length which is a multiple of 5m. Let the left top corner of the drawing coincide with the origin of the global coordinate system.

<b>9</b>	[	Diamonds - [Venster 1 (m)]		- 🗆 🗙
🥮 Eile Edit View Select	Display Analysis Options Windows Help			- 8 ×
D 🛩 📕 🖪 🚳 🕅	∽ ~   ⊞ ⊞ ⊞ <b>™   ™  </b> ⊠ ■	I X № 6   % ± 0 % ₽		
😨 🐻 🖷 🔀 Venster 1	User configuration	🔹 🚯 💷 🔯	🛉 🕁 🕅 Q Q 🛛 🖉 🖻 🖻	
	■ Vuer configuration	<u>15.00m</u>	▲ 小 <b>Q Q 図 / ●</b> ■ 5.00m	Image: Constraint of the second level         Active level         Foundation         Image: Constraint of the second level         Image: Constraint of the second level
				z^  ======
11				11.

Then click on the button and draw all constructions points. For drawing a point, navigate the mouse to a grid point and click once with the left mouse button.

<del>9</del>	Diamonds - [Venster 1 (m)]	- 0 ×
🥮 <u>F</u> ile <u>E</u> dit ⊻iew S	Select <u>D</u> isplay <u>A</u> nalysis <u>Options</u> <u>Windows</u> <u>H</u> elp	- 8 ×
D 🛩 日 🖪	▶ ◎ ● ☆ ◇   田田田   Ⅲ   Ⅲ   図 圖 業 数 6   欧 ⊡ 彤 ☞     F 申 ゆ	
😨 🗟 🖷 🔀 Venste	er 1 🖃 🚺 Geometry 🖃 🏟 🕂 💹 🕅 🖉 🕴 🖉 🖉 🖉 🚺 🖉	
N. X		Active level
	+	
000		4+11
⊗ ♦		Level manager
Z. 121 24		Ground level
12 HH G.		X = 0.00 m
τ <sup>2</sup> R,J I <sup>2</sup> *:		Y = 0.00 m
<u>_ Mar 200 785</u>		Representation
HT 🔍 🎝		***
⑪ sf: 庐		- D - D - D - D - D - D - D - D - D - D
		Size
Te n Il		Font 25 🚖
Ę,		Symbols 15 🚖
		Loads 15
		Results 15 🜩
		None 👻
		- V 197 197
		2 1 == 2 == C

End the drawing function with  $\stackrel{\bullet}{\blacktriangleright}$ . To display the structure as large as possible on the screen, click  $\stackrel{\blacksquare}{\blacksquare}$  or F12.

#### Step 6: Changing the grid step again

Click on <sup>III</sup> and change the grid step to 0,5m. Close this window with 'OK'.

#### Step 7: Drawing the perimeter of the foundation slab

Again activate the drawing function and draw the contour of the foundation slab. Make use of the grid to realize an overhang of 0,5m.

<del>\$</del>	Diamonds - [Venster 1 (m)]	- 🗆 💌
Select Display Analysis Options Windows	Help	_ 8 ×
	M R V W A CO TO PER TY   I LE ME	(A)
		**
😨 🗟 🧮 🔀 Venster 1 💽 Geometry		? Q Q 🖾   🖉 🔟 🖻 🖻 🐨 🖾   🗇 🖾   5</th
X = 2 + 2 + 2 + 2 + 2 + 2 + 2 + 2 + 2 + 2		Active level
n ∧ contraction contraction in the second s		foundation 💌
		주 10.00 m
A / /		
·····		Level manager
	1 403 M 21 403 M 21 104 M 26 10 10 M 27 10 M 28	
7 111 TT		Ground level
夕 囲 迂		🛱 Drawing plane
AT the second se	i na n' n' na 11 n' n' ni n' n' ni n' ha n' n' na	$X = 0.00 \mathrm{m}$
†″ вµЈ Г*1 вкажен   вкаженаженаж		Y = 0.00 m
志 (X-3)		Z = 0.00 m
191 <b>%</b> 1 /		Representation
IT AL PRANE PRANERANCE		
🗄 🗟 🖓 💱 (1997) 👘 (1997) (1997)		
All the second s		
		анка 🖬 как клан ка 🖓 🛷 👘
The A 11111		
		Size
R R II I I I I I I I I I I I I I I I I		Font 25 🚖
	- 2014 12 20 20 20 20 20 20 20 20 20 20 20 20 20	Symbols 15 -
		Loads 15 🕏
		Show groups
		None
		prone
	1 NOT N 11 NOT N 11 NOT N 10 N 1 NOT N 10 NO NOT NOT N 10 NOT	
		z z
h.(.		16

The foundation can now be defined:

- Either you only select the external edges of the foundation plate using the SHIFT-key.
- Either you select the entire structure by drawing a selection window or pressing CTRL+A.

We conveniently choose the last option: press CTRL+A. Then click on the button so diamonds can start looking for plates. There's only one closed contour present so that one plate will be formed.

<b>e</b>		Diamonds - [Venster 1 (m)]	- 🗆 🗙
🥮 File Edit View Select	Display Analysis Options Windows	Help	- 8 ×
		M R V Ro A CO TO PO FOR THE LE LE LE	5 J
😿 🔕 🖷 🔀 Venster 1	Geometry	● ● ● ● ● ● ● ● ● ● ● ● ● ● ● ● ● ● ●	*) q q 🖬 🖉 📴 🖪 🐨 🖾 🛛 着 🛛
			Active level
5 🔨			foundation 💌
		a state a	eraeraeraera
	· · · · · · · · · · · · · · · · · · ·		₩10.00 m
A A			
	<b>Z</b> e os s e os s e os s		
<b>₩₩</b>			Hi Level manager
<u>v</u> 21 402 56	21 43 21 402 96 21 402 96 21 402 96	n ang sa na ang sa na na na na na na na 🚺 na na na n	
Z			Ground level
	0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0		🛱 Drawing plane
AT 5 D.1	21 10 21 102 10 21 102 10 20 103 10	n en a n en à n na c a na c a <sup>n</sup> la c a na c	X = 0.00 m
<b>τ</b> <sup>*</sup> <b>Β,</b> Ĵ Γ**Ι   • • • • •	8 C		X 2010 X 2010 X 2010 X 2010 Y = 0.00 m
NT 14 0			Z = 0.00 m
niu 11 12	25 K 25 K 26 K 26 K 26 K 26 K 26 K 26 K		Representation
0T 6 44	ы к		
IO 😪 💱 🔡			
HA SE BI			
W 8 4	8 8 <b>-</b>		ска 🗉 казиказика 🖑 🛷
🗐 🛼 🥒 🔿 01000			
100 × 100 ×	20 KOZ W 20 KOZ W 20 KOZ W 20 KOZ W		Size
T 12			e e e e e e e e e e e e e e e e e e e
		5 191 5 5 <mark>- 5 5 1 5 5 1 5 5 1 5 5 1 5 5 1 5 5 1 5 5 1 5 5 1 5 5 1 5 5 1 5 5 1 5 5 1 5 5 1 5 5 1 5 5 1 5 5 1 5 5</mark>	Sumbole 15
En State	21 AUX 10 AUX 10 AUX 10 AUX 10 AUX 10		Symbols 13
			Loads 15 🗢
0.108.2			Deally Internet
2 2 2 2			Results 15
81 6526 26	N KA K K KA K K KA K KA K		Show groups
0.108.0			None 🔻
21 2012 12	20 2008 12 20 2008 12 20 2008 12 20 2008 12 30 2008 12 30 2008 12 30 2008 12 30 2008 12	27 2010 12 27 2010 12 27 2010 12 28 2010 12 2010 10 10 2010 10 10 2010 10 N 2010 12 N 2010 12 N 1010 10 N 1010 N 10 10 101 101 10 1010 N 10 10 101 N 101	
6 103 2	A 100 0 A 100 0 A 100 0 A 100 0	<u>E 188 U E 188 U E 189 U E 189 U E 189 U E 189 U</u>	n nora n nora n nora Z

From the toolbar on the right side of the model window choose a transparent representation  $\stackrel{\text{loc}}{\longrightarrow}$  so that the plate and all the construction lines and points are displayed.

S.	>											Di	amo	nds	- [Ver	nster	1 (r	n)]											- C	×
1	🖗 <u>F</u> ile	<u>E</u> dit <u>V</u> ier	w Select	<u>D</u> ispla	y £	nalysi	s <u>O</u>	ptions	Wi	ndows	Hel	р																		- 8 ×
1	D 🚅		A D	6	à.	II pu	171	ΠÌ		<b>T.</b> .	₩.		¥ ¥		4 0	2 371	(T4)	e/ I	1	IE.	自	A)								
+													<i>m</i> •	<u> </u>	1 -		~				·+				÷	11				- 4 - 4
	* 5		enster 1			•	1	Use	er conf	iguratio	on				•	酚	L11	0 🖻		<b>御</b>	由	<b>E</b>	0.0		-	D	R	51° 0	0	<b>N</b> =
Ť			20 100 0	30 6135	2.1	1.100	35.35	1000 0	5 30 	1985 - 24 	31 100	1.1	30 100	1.1	51 (USP 1. 1	18.75	1.1	10.05	2010		10.1	3 25	2019 0	75 8		1. 1	10.7		Active le	vel
	۳	$\sim$	0.131.0	0.89	8.		88	191.1	8.8	191 S	8.8	1.0	0.3	1.8	9.98	8.8	985.	8.8	985. I	9.8		9.9	181 I	181	S. 6	8.98	8.8		foundatio	on 💌
	ΛĦ	. /	31 1035 32	8.69		2. 14													. [	8 8	100	8 - 8	3938-8		8.8	H 100	8.8	- 800 F	A	
	<u> </u>	•	10 1000 M	20 M	1.	2.1		toot s	а. н.:	oor a	15 - 50		10.00		to the			a n	11	a s a a	ACC 1	10 10 10 10 10 10 10 10 10 10 10 10 10 1	5007 G		107 - 50 107 - 50	25 5535 36 5535	10 10 10 10	5 - 5002 C - 5002	E HI	0.00 m
	n.	00	8 188 8	8.8	¥ :					202-0	4.43		41.40		1.10		2.04			9.9	201	4.4	981.8	1 1 1	8. Q	a 28.	8 G	- 283	And .	0.00 m
	0	v v	20 1028 28	8.5	4		81 AS		e e:				80 80 10	а ас.		194 (H) 194 (H)	* *	-		98 (S) 	333	8 . S.	3938 8		8.8	H 1993	8.9	6 8939 -	AT P	0100 111
	26		N 1005 0	- 20 - 53 - 50 - 50									10 10 10 10		10 1 1 1 1 10 1 1 1 1					10 10 11 11	1000		5007 G		त्व ३६ त्व क	10 1000	- 10 - 10 - 10 - 10	5 8005 5 8005	<b>G</b>	Inner
	-		2 22 2	2.3	1					84 Q										4.4		4.4	1992, 19		S. S.	9 PS	8.8	1.55	HH LEV	ermanager
		Ŧ	20,623,9	25 - 25	1				÷ +:				1.1		1.10		* 24			14 12 	1939	14 HZ	1335 0			NE 1934	10.10		THE Gro	undlevel
	1.1	11 M.	0 100 0	88		e teet. S teet	a a.	101.0	5 5. 5 50	eor a	10,000	5 (5) (5 (5)	10 100 10 100		to toot to toot		- 1001 - 1001			05 (5)	1691	85 (85	3605-0			45 1601 (8 1631	0 0 0 0	5 1500 5 1505	10000 010	
	12.1	111 A.F.	2.23.3	2.8	1					• • •									1.				1.4.9			10 10 M	8.8	1254	Drav	ving plane
	47 .	5 P.:	20 K02 W	20.00		o esta. S esta	ar an ar an		n ni National	esta la Local la	10.00	ана. Ана	40 400 40 400		10 1004 10 1004		- 4654 - 4654		- 600 - 1 - 600 - 1	а с. А. А.		а а. А. А.	4004 00 4004 00		194 - 194 - 194 - 194 -	160 1000 100 1000	10.10	( 1004 ( 2004	X =	0.00 m
	1.1	1427 - 8 - 8 - 147 - 8 - 8	6 100 0 6 100 0	0.0			a				1.10		10.00						1011						87 (B) 87 (B)	(S 160)		1000	Y =	0.00 m
	NA 2	KIR	2 .53 5	2.13	Ť	1994	2.2	54.2					8.6		· · ·					• •		• •			8 B.	9 P.9	8.8	· · · · · ·	2 =	J.UU M
		~	10 KOR W	20 KB 50 KB		e estat. Stestat	анас. А. 40		e el e el	era a Fra a	- 40 - 400 - 40 - 400	ана. Анас	46 - 65 16 - 65		10 1004 10 1004		- 4004 - 4004		- 4004 - 1 - 4004 - 1	a a	- 4004 - 1 - 4004 - 1		404-0 404-0		04 58. 54 58	NE 1004	100 100 100 100	- 1004 - 1004	Represen	tation
	HT S	2.0	10 108 2	0.0																					87 B.	8 887	a a	1997	A	A A
	000	En the	26 2832 52	20.20	1	5 - 5794 5 - 5574		1994 - 2 1994 - 2			- 81 - 83 - 100 - 100		2.50		2.100		100		2014	* *		4 4) 			22 22 	12 F.12	10.1	2 2002 	~	
	EE g	91 F)	30,003,00	81.00		e •200	× ×		× *:-	•><+ -×	99.900	* *	(*) •):)	_	на насе на насе				• • • •	a a			+04-0 400-0		04 54 04 58	N 1004	24 A	- 1004 - 1004	20	0
	Ξ	+ /2	0.038.2											1												0.897	12 13	107	*	·
	~~~~~ ·	e v	20 2009 02 No. 2008 02	20 2002	2.1	5 25354 6 23356	96 - 96 07 - 96	2002 0	1 - 10 - 10		10.000	a	10 10	14											22 22 22 22	10 1010			Size	
	<u></u> =	P Tre	30 -000 B	- 94 - 604 - 98 - 608	3.3	a 19034 8 19039	an an 18 88	1000 0	6 38 8 38	-004 -00 -0059 -00	38 88	а ж. Э.Э.	38 - 55	-	10 10 10 10 10 10 10 10 10 10 10 10 10 1								+		58 58. 58 58	00 1000 30 1000	32.3	5 808 5 809	Font	25 🜲
	-13(e	M 7.	0.138.8	0.138	8.5	133	8.8	138 8	0.0	136 B	0.33	8.8	0.3									• •				8 887	8.8	107		
	E.		10 1000 10 10 2000 00	20.000	3.3	5 2009 6 2009	ar ar Grain	2002 0	41 - 421 26 - 256 -	2003 SK 2003 SK	10.000	9 19 8 00	10.10												104 NG 014 NG	10 1014	26 14 26 15	2 - 40704 2 - 1903	Symbols	15
1	2.2		8 6.8 8	8 103	8.8	8 80.58	8.8	1031	8.8	10.1	8 10	8.8	8 8	-	6.609				101						SF 82	81 853	8.8	6 859	Loads	15 🜲
			0.138.2	8 38	2.5	1.53	8.8	136 8	8 8	38.8	83	8.8	8 S			33	100	a a	101	: :	-	: :	507 S		8.8	8 88	8.9	1927	0.1	
			201003	25 1004	2.1	5 2007 5 2007	10 - 20 57 - 36	2004 0	6 6 8 8	2004 NG 2004 NG	10 100	4 14 2 12	40 10 26 882	1	no nora	~~~	- 2014		<u>.</u>	4 ¥	2.14		*** 9		104 FO	10.1014	10 10 10 10		Results	15 💽
			8 6.8 8	81.003	8.8	10.0	8.8	10.0	6.81	6.R. H	83.82	8.36	10 100	8.16	8 8.08	8.8	35.52	8.8	808	a a	8.9	a a	808 B		a a	10.00	8.8	1.00	Show gro	ups
			0.138.13	0.195	8.5	1531	8.8	1331 1	2.0	31.2	8 8	8.8	8 13	8.8	5 KS	8.5	182	8.8	621	8.8	127	8.8	182.6	1.8.1	8.5	S 182	8.8		None	-
			20 102 9	35 883	8.3	6 839	96 - 26	282 3	8.95	882 86	21 43	8.8	10 RE	18	20 832	10.10	- 6938	16 18 16 18	1933	н 18 18	635	a 12	635 B		99 39	10 1014	2Ê		Lott Lo	
			8 63 8	8 8.8	8.8	80.9	8.8	1000	8.8	608 B	83.82	εж	8 83	8.8	8 8.0	8.8	633	8.8	83	ж ж	831	в в	8.08 8		a a	8.63	8	L_X		ě.
-			0.030.9	4: 102	3.3	1004	35 - 45	038.3	6 K.	<u>esa 35</u>	<u>6</u> (3	a .a	<u>e e</u>	1.35	<u>a neg</u>	0.0	1859	0.0	332	3. 3	837	3. 3	887.8	0.0	<u> </u>	8.882	<u>a</u>	Z		
2																														

#### Step 8: Adjusting the plate properties

By default the plate has a thickness of 20cm. Double click the plate and change its thickness to 300 mm.

<b>\$</b>	Diamonds - BetonVB3.bsf - [Fene	Propertie	s of plate number: 1
Eile Edit View Select Display Analysis Options V	<u>V</u> indows <u>H</u> elp		
	🚺 🖬 🕱 🥻 🕹 🕾 🗔 🕒 🛛	Cross-section	
😨 🗟 🖷 🛛 Fenêtre 1 💽 🚺 Geomet	гу 💽 🕅 🖬 [	Name Foundation	<b>_</b>
$\mathbf{k}$ $\mathbf{X}$		· · · · · · · · · · · · · · · · · · ·	
		Shape	thickness (e) 300,0 mm
🗠 · 🖉 👬 👘 👘 👘 👘			
000	11111 233322233333333333333333333333333		
🔊 🐠 👘 🔤 🔤 🚛			
Exundation			
₽ <sup>7</sup> в5 P·I			
M JH A			
	<mark></mark>		
H 🎗 🎲	Gross Concr	ete Cover ×	
₩ é <sup>p</sup> k			<ul> <li>Gross cover</li> </ul>
Σ / Α	X' - Reinforcement (Y' > 0) (1)	mm / z', 🏹 X' 🚄	
	X' = Reinforcement(X' < 0)(2) 35.0		↑² ×′
te n P	x - Reinforcement (r < 0) (2)		
E. EEEEEEEEEEE	Z' - Reinforcement (Y' > 0) (3) 35,0	mm $z_{1}^{2} \xrightarrow{\uparrow \gamma} \chi^{(3)}$	
	Z' = Reinforcement(Y' < 0)(4) 35.0	\$	
		£4)	×8 94×8
	Help	Cancel OK	
			-
		Help	<u>C</u> ancel <u>O</u> K

Also check the gross cover on the longitudinal reinforcement. Click on Gross cover and accept the default cover of 35mm. Conform twice with 'OK'.

#### Step 9: Defining the ground level

Because we will be using soil layers, you have to set the ground level. Set the ground level to 0,0m.

<del>\$</del>	Diamonds - BetonVB3.bsf - [Fenêtre 1 (m)]	- D ×
	<u>/</u> indows <u>H</u> elp	_ & ×
□ 🛎 🖬 🖪 🖉 🚺 ∽ ↔ 🛛 🖽 🖽 🖽 🗰	1 🖬 💹 📾 🗶 🎉 👌 🕸 亚 🕀 🕬 🍽 📗	Dorien 🔻 📕 🖵
😨 🗟 🗮 🔀 Fenêtre 1 💽 🚺 Geomet	ry 💽 👰 🖳 🕅 🔁 🥙	
× ×		Active level
		foundation 💌
		0.00 m
A 0 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1	Ground level ×	
V 🖤 – – – – – – – – – – – – – – – – – –	Position	🕮 Level manager
Foundation		
Louis 25 Contraction	Y A	Cound level
12 PR LE		• Drawing plane
- <del>,</del> <i>γ</i> , <i>β</i>		X = 0.00 m Y = 0.00 m
.₹4		Z = 0.00 m
	y= 0,00 m	Representation
E 🎗 🖒 anna 🛛 anna anna		A A A A A A A A A A A A A A A A A A A
fffi 2Fe are the second second second		
		ST 47 47
🚡 / 🖉 na ka 🛌 🔤		
	Help Cancel OK	Font 25 🜲
		Symbole 15
• • • • • • • • • • • • • • • • • •		Symbolis 15
		Loads 15 호
		Results 15 🚖
		Show groups
		• None 💌
		and a second second second second
		z z z z z z z z z z z z z
		1.

#### Step 10: Defining the soil properties

Select the plate and click on



- You could prevent the X- and Z-displacement but is not necessary since there'll be no horizontal forces action on the model.
- For the Y-direction, indicate you wish to use soil layers.



Now click on . A new soil profile is defined with the button Add soil layer profile. Indicate that the soil parameters have been obtained using a static penetrometer test (CPT) and note that the ground water level is 1 m below the ground level.

🤪 Diamonds - BetonVB3.bsf - [Fenêtre 1 (m)] 🗕 🗖 🗙	
Pick Edit View Select Display Analysis Options Windows Help           ● 定         ● 合 無 下         ● 一 一 一 平           ● ご         ● 日 一 一 一 一 一 一         ● ○ ○ ○ □ □ □ □ □ □ □ □ □ □ □ □ □ □ □ □	
Image: Second	×
Image: Control in the image: Cont	24

Next we start defining the different soil layers and we start with the soil layer right below the ground level. Click on Ref Add layer and note the layer thickness, the compressibility constant *C*, the recompressibility constant *A*, the dry  $\gamma_d$  and wet  $\gamma_n$  mass of the layer in the dialog box.

<b>\</b>	Diamonds - Beton	VB3.bsf - [Fenêtre 1 (m)]	- • ×
Image: Select Display     Image: Select Display       Image: Select Display     Image: Select Display	Analysis Options Windows Help 日日日日日:111 日日:211 日日 図 園 文 谷々。		_ ♂ × Dorien ▼ ■ 🖵
Fenêtre 1		Soil layer profile –	×
	Profile 1         Soil parameters from	Add sol layer profile  Add sol layer profile  Add sol layer profile  Soil layer  Layer thickness  Layer thickness  Compressibility constant C  Recompressibility constant C  Recompressibility constant A  Overconsolidation ratio OCR  Drainage ratio CC  Specific mass (dry)  Specific mass (dry)  Help  Cancel  QK  Cancel  QK	A Served QK

Confirm with 'OK'. Repeat the steps for the other layers in the profile. All defined soil layers are neatly displayed under each other:

\$		Diamonds - Bet	onVB3.bs	f - [Fenê	tre 1 (m)]			- <b>D</b> ×	
Eile Edit View Select Display Anal	ysis Options <u>W</u> inde	ows <u>H</u> elp	• 1 Hos	-	a a filme	m s		_ 8 ×	
	╈		Soil	laver p	% ₽      ► rofile	∎	_ 🗆 🗙	Donen C	×
	▼		301	layer p	ome				
	Profile 1				<b>-</b>	🚟 k Add soil lay	/er profile		
						🧱 Remove soil I	ayer profile	khi/m 3	-
000	Soil parameters from	m						Kingin-	
	static pene	tration test (CPT)							
Foundation	C Menard pre	essiometer							
る 副 2月 111	Ground water level	below original ground l	level	1.00	m				
27 .5 P.1	Soil layers below or	iginal ground level							
τ <sup>*</sup> κ, <b>, , , , , , , , , , , , , , , , , , </b>	🔽 Bottom lay	er continues into infinit	ty						
	Add	d layer		子 Import	layers			kN/m³	
HJ 🏹 🕼	👥 🙀 Remo	ove layer	[	Export	layers			- ,	
⑪ \$ 的	thickness (m)	C A	OCR	CC (%)	γ-dry (kg/m³)	γ -humid (kg/m³)			
🚡 / 🖉 eeennnn	1.00	120.0 180	.0 1.0	0.0	1600.0	2000.0			
	3.00	350.0 525	.0 1.0	0.0	1600.0	2000.0			
	4.00	260.0 390	.0 1.0	0.0	1600.0	2000.0			-
<u>. ~</u>	10.00	340.0 510	.0 1.0	0.0	1600.0	2000.0			
	1								
55533333555 5553333555	Help					<u>C</u> ancel	<u>o</u> k	Cancel OK	
* * * * * * * * * * * * * * * * * * *							·		
			3333	2.2.2				z arist	

Since we will use these soil layers in a later project (see §3.5), we click on Export layers. A text file containing all data of the soil layers will be saved at a location designated by you. Save the file for example on the desktop. Now exit all dialog windows with 'OK'. That the soil profile is indeed assigned to the model is identified by the symbol in the centre of gravity of the plate.



There, the structure is completely defined.

## 3.3.2 Defining the loads

Step 11: Go to the 'Loads' configuration

We now leave the 'Geometry' configuration and activate the 'Loads' configuration to enter the loads. Click on the button  $\square$  in the icon bar or select in the adjacent pull down menu the 'Loads' configuration.

## 3.3.2.1 Creating the load groups

```
Step 12: Creating load groups
```

Before defining any loads, you have to make the different load groups. Click on the button  $\gamma_g^{\gamma_u}$ . You'll see the following screen:

49			Diamo	nds - [V	enster '	I - Selfv	veight -	(kN, k	Nm, mr	n, <mark>kN/</mark> m	n, kNm/r	n, kN/m², °C)]			- 🗆 🗙
🤗 <u>F</u> i	le <u>E</u> dit <u>V</u> iew <u>S</u> elect <u>D</u> ispl	ay <u>A</u> nal	/sis <u>O</u> pt	ions <u>W</u>	indows	<u>H</u> elp									_ <i>8</i> ×
	🗳 📕 🖪 🖨 🚺 🗠	~	H H H	7   <b>:::</b>   F	Tii		X Ke	ନ   ଞ୍ଚ	<u>, (</u> )	9% IÝ		t +\$			
*	🗟 🖷 🔀 Venster 1	•	1	Loads				•	<b>6</b>		₩ ₫	* * • • •		R	
γ <sub>g</sub> γ <sub>g</sub>  Selfv →Ω	mi⊥ © fm ⊷ recipht ▼				*~~~	2								-	Active level foundation
÷							Loa	d grou	ıps						- • ×
Lo	ad factors for		EN 1990		•		•	Consec Design	juence d lifetime	ass [	2 🔶 50 🔶	years	Service d	ass 1	\$
	Add load group		Insert loa	ad group					Delete	load gro	up		Several	oad cases	per group 🥅
	Name load group	$\gamma_{\rm uls}$ -	$\gamma_{uls+}$	$\gamma_{\rm sls}$ -	$\gamma_{sls+}$	ψ <sub>0</sub>	$\psi_1$	ψ <sub>2</sub>	φ	ξ	to	Combination for cracking	k <sub>mod</sub>	Load	Action
<b>•</b>	<ul> <li>Selfweight</li> </ul>	1,35	1,00	1,00	1,00	1,00	1,00	1,00	1,00	0,85	0		perm	—	<u>+++</u>
	dead loads	1,35	1,00	1,00	1,00	1,00	1,00	1,00	1,00	0,85	0		perm	-	<u>+++</u>
	life loads A : housing	1,50	0,00	1,00	0,00	0,70	0,50	0,30	1,00	1,00	0		mediu	-	<u>+++</u>
	Incompatible load groups         Linked load groups         Combine loadcases         Ungroup loadcases														
	Help												<u>C</u> anc	el	<u>o</u> ĸ

Close this dialog box using 'OK'.

### 3.3.2.2 Filling up the load groups

Now the loads groups are defined, we can assign loads to the structure.

Step 13: Filling in the load groups 'Self-weight', 'Dead loads' and 'Life load'

- The **self-weight** of the beams is calculated automatically by Diamonds and cannot be adjusted.
- We now define the **dead loads**:
  - Use the pull down-menu to activate the load group 'Dead loads'.
  - Select all lines except the edge of the foundation slab and click on the button in the pallet. Enter a uniform distributed load of 85kN/m.

🧇 Diamonds - [Venster 1 -	dead loads - (kN, kNm, mm, kN/m, kNm/m, kN/m², °C)] – 🗆 🗙
<u>File Edit View Select Display Analysis Options Windows Hel</u>	elp - #×
	1圓 % 秒 6 ◎ 盃 ⊕ ☞ ☞ ↓   ⊨ ■ ゆ
😨 🗃 🖷 🗵 Venster 1 💽 🚺 Loads	- • • • • • • • • • • • • • • • • • • •
y <sup>y</sup> u <u>mit</u> O	Active level
	foundation 🔽
	10.00 m
No. at	2 0.00 m
7ñu +*	met Level manager
+ + F	· ·
9 9 9 9 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1	Verdeelde last op lijn
A. A	
· · · · · · · · · · · · · · · · · · ·	Z Beginwaarde 0.0 kN/m
Temperature     Snow	Eindwaarde 0.0 kN/m
* Wind	
Seismic     Dynamic	Beginwaarde [85.0] kN/m V
* Moving	xk Eindwaarde  85.0 kN/m
	Z Beginwaarde 0.0 kN/m
	Eindwaarde 0.0 kN/m
	± €
	€ Staaflengte = 15.00 m
	Lengte fysisch element = 15.00 m
	C afstand van begin 0.00 m
	afstand van einde 0.00 m
	Waarden en afstanden gekoppeld aan groep 'Lasten'
	<u>H</u> elp <u>Annuleer</u> <u>Q</u> K

- Confirm with the button 'OK'.
- o Next select the two points in the left top of the model and click

on +. Define a point load of 600kN on them:

🥹 Diamonds - [Venster 1 - dead loads -	(kN, kNm, mm, kN/m, kNm/m, kN/m², °C)] – 🗆 🗙
🧐 Eile Edit View Select Display Analysis Options Windows Help	- # ×
	♦ ※ 10 9 1 1 1 4 1 4
😨 🗟 🗮 🛛 Venster 1 💽 🚺 Loads	- M - M - M - M - A - M - M - M - M - M
y <sup>y</sup> <sub>a</sub> mit o	Active level
dead loads 🔽	
	年期 <sup>#10.00 m</sup>
	• Provide the second se
	Ground level
▼ / / ♥ └┘	Force at point
17. 17 A	X = 0.00 m Y = 0.00 m
* Temperature	Z = 0.00 m
v Snow	presentation
v Seismic	🔶 0,0 kN 🖓 🤁 🕸
Dynamic     Moving	
	nt. 25 🚖
	mbols 36 🜩
	0,0 KN ads 15 €
	sults 15 🚖
	Help Cancel OK
	Z <sup>×</sup>

- o Click on 'OK'.
- Repeat these steps to assign a dead load of 60kN to the other points.
- Finally define a surface load of 3kN/m<sup>2</sup> on the foundation slab.

Select the foundation slab and click on  $\overset{\textcircled{}}{\longrightarrow}$ . Enter 3kN/m<sup>2</sup> and confirm with 'OK'.

Here below we represent the life load in a perspective view. You'll notice that each type of loads is presented in a different colour. If you wish the same configuration, then choose the menu command 'Show – Window configuration' or click on **I**. A dialog box appears containing all the settings of the 'Loads' configuration.

Select the tab 'Loads' and define a different colour for each load type by clicking in the corresponding colour field with your mouse.

Also note that you can impose a different scale factor for each load type. For example in the image below the point loads will be represented bigger than the uniformly distributed loads.

Diamonds - BetonVB3.bsf - [Fenêtre		Window Configuration
♥ File       Edit       Yiew       Select       Display       Analysis       Options       Windows       Help         □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □	Current configuration :	Save current configuration
😨 🗟 🗮 🛛 Fenêtre 1 💽 User configuration	Loads	Delete selected configuration
y <sub>2</sub> <sup>y</sup> mi ⊗ im ∞	Configuration name Loads	
dead loads 600.0	General Geometry Loads Mesh	
× +	Size Color for	ractive load Color for inactive load
+ -> -> + -> -> + -> ->	20	
	20	
Temperature     Snow     Wind	20	
V Seismic     V     Vynamic     Woving	20	
600		
	20	
	20	Show mass
4	20	
	20	
	Help	<u>C</u> ancel <u>Q</u> K

Click on the button Save current configuration to save the current settings. Confirm that you wish to replace the active configuration by the new one. Close this dialog box with 'OK'. If the loads in general are displayed to small, then change the size in the pallet 'Size' on the right side of the model window (see §2.2).

- Now select the load group 'Life loads A' from the pull down list.
  - Define a line load of 10kN/m on each wall.
  - Define a point load of 100kN coming from the two columns in the main building on the two points in the left top of the model.
  - Define a surface load of 2kN/m<sup>2</sup> on the foundation slab.



### 3.3.2.3 Making combinations

#### Step 14: Making combinations

Generate the combinations  $\frac{414}{4m}$  as described in §3.1.3.3

## 3.3.3 Generating the mesh

Step 15: Generating the mesh

Generate the mesh  $\square$  as described §3.2.4.

#### Step 16: Verifying the mesh

Once the mesh is generated, we make the mesh visible with  $\square$ . In a top view the model now looks like this:



We obtain a dense and regular mesh, which will give us good quality results.

## 3.3.4 The global elastic analysis

#### Step 17: Elastic analysis

To start the analysis, select the command 'Analysis' – 'Elastic Analysis'. You can also start the analysis directly using the function key **F9** or use the icon  $\blacksquare$  on the icon bar. Following dialog box appears:

<b>9</b>	Diamonds - BetonVB3.bsf - [Fenêtre 1]		- 🗆 🗙
<u> </u>	t <u>D</u> isplay <u>A</u> nalysis <u>Options</u> <u>W</u> indows <u>Help</u>		- 8 ×
	▏ <u>▏</u> ऽ○○│───────────────────────────────────		
🛞 🙆 🖷 🔀  Fenêtre 1	🗾 🚺 Mesh 🗾 🕅 🖽 💹 🔁 🗍 🖄	ѝ   ⊈  <") Q Q  2   ∥  ■  ■  ⊡	
N, X	Analysis settings	×	Active level
	Structural Soil Dynamic		₩10.00 m ### 0.00 m
	Analysis     Ist order analysis		Evel manager
今開登 같時円	2nd order analysis     Precision for normal forces in bars:     1     max. number of occessing and effections	%	↓ Drawing plane           X = 0.00 m           Y = 0.00 m           Z = 0.00 m
	max. number of iterations for nonlinearities: 50 5		Representation
	Global Imperfections     Inints		Size
	Concrete Cracking		Symbols 15
	▼ Timber		Loads 15 文 Results 15 文
20 10 40	Help         Cancel         < Previou	IS Next > OK	Show groups
-			

We opt for a first order analysis and click 'Next'.

The exact value of the spring constant is not yet known. Therefore the first step in calculating the model is determining the correct function of the spring constant based on the soil data and a suitable load combinations. A number of parameters must be set:

9	Diamonds - BetonVB3.bsf - [Fenêtre 1]	- 🗆 🗙
File Edit View Select Disp	play <u>A</u> nalysis <u>O</u> ptions <u>W</u> indows <u>H</u> elp	- 8 ×
D 📽 📕 🖪 🖉 🖻	○ ○   田田  ⅲ   №   図圖業数 6  ※五色 ※ゴ    ⊨ ♥	19 19 19 19 19 19 19 19 19 19 19 19 19 1
😨 🐻 🖷 🔀 🛛 Fenêtre 1		
N. X	Analysis settings	Active level
	Structural Soil Dynamic	0.00 m
Highe	al ground level 0.00 m n	
夕田 受 2 85 円 2 85 円 Verti	erence load group / combination SLS QP 1   itical resolution for stress evaluation 0.60 m	Ground level
	imum number of iterations	Z = 0.00 m Representation
m ster	uss tolerance ju %	
	Help         Cancel         < Previous	Size Font 25
E)		Symbols 15 文 Loads 15 文 Results 15 文 Show groups None 💌

The parameters in **the second tab page** of this dialog window are linked to the soil and terrain properties.

- First Diamonds will show you the level of the highest foundation to which soil layers where assigned. In this example only one foundation slab is defined in the plane Y=0m.
- In this example we assume no excavation, so that the initial ground level is located at Y=0m.
- Then select a load combination based on which Diamonds should calculate the spring constants. Because the largest settlement will be obtained with the first quasi-permanent load combination, we select 'SLS QP1' from the pull down list.
- The vertical resolution determines the thickness of the successive layers Δh which will be considered in the calculation of the settlements. Diamonds assumes the vertical resolution to be equal to the chosen maximum size of the triangle elements. For a resolution of 60 cm, the setting will be evaluated every 60 cm. Do you have a soil profile with C-values each 20 cm, then each three layers will be combined to an average C-value.
- Finally we impose a maximum of 2 iterations.

Once all parameters are entered, we can start the calculations using the 'OK' button.

A dialog box displays the progress of the calculation.

Performing analysis	
Generating a complete mesh	
Adding inclined supports and hinges	

Once the progress of the spring constant is determined, the actual elastic analysis will be performed.

#### Step 18: Go to the 'Results' configuration

To see the results of the calculations, you click on 🔄 in the icon bar or select in the adjacent pull down menu the 'Results' configuration.

1	Resultaten	🖗 म 🕅 🔁
	Geometrie	
	Lasten	
	Resultaten	
	Mesh	

In the first place we are interested in the settlement of the foundation slab.

- Click on the button *in the results pallet for viewing the deformations.*
- Select the vertical displacement  $\delta_{\gamma}$  according to the global Y-axis.

- The combination 'SLS QP – envelope - min' shows you the largest downward (elastic) deflection in each point under all combinations SLS QP.

In the figure below we opted for a plan view and wireframe representation



The main building will undergo the largest settlement.

We are also interested in the ground reactions:

- Click in the pallet on the button for viewing the reactions;
- Select the plate reactions  $R_Y$  according to the global Y-axis.
- In particular we opt for the largest (real) values of the envelope 'SLS RC max'.



Upward reactions are always considered positive. Given the entire plate undergoes a downward movement, each feather will be compressed. In other words, we find the greatest reactions when we choose the upper (max) envelope of the rare combinations. The largest ground stresses are concentrated in the corners of the plate.

## 3.3.5 Calculating the reinforcement

Step 19: Choosing the concrete standard

Now select the menu instruction 'Analysis – Concrete standard' and indicate you wish to calculate the reinforcement using the European standard EN 1992-1-1. We don't use a national annex [--].

#### **Step 20: Calculating the reinforcement**

Next choose the menu instruction 'Analysis – Calculate reinforcement', press on **F2** or click on the button  $\stackrel{\text{(s)}}{\stackrel{\text{(s)}}{\stackrel{\text{(s)}}{\stackrel{\text{(s)}}{\stackrel{\text{(s)}}{\stackrel{\text{(s)}}{\stackrel{\text{(s)}}{\stackrel{\text{(s)}}{\stackrel{\text{(s)}}{\stackrel{\text{(s)}}{\stackrel{\text{(s)}}{\stackrel{\text{(s)}}{\stackrel{\text{(s)}}{\stackrel{\text{(s)}}{\stackrel{\text{(s)}}{\stackrel{\text{(s)}}{\stackrel{\text{(s)}}{\stackrel{\text{(s)}}{\stackrel{\text{(s)}}{\stackrel{\text{(s)}}{\stackrel{\text{(s)}}{\stackrel{\text{(s)}}{\stackrel{\text{(s)}}{\stackrel{\text{(s)}}{\stackrel{\text{(s)}}{\stackrel{\text{(s)}}{\stackrel{\text{(s)}}{\stackrel{\text{(s)}}{\stackrel{\text{(s)}}{\stackrel{\text{(s)}}{\stackrel{\text{(s)}}{\stackrel{\text{(s)}}{\stackrel{\text{(s)}}{\stackrel{\text{(s)}}{\stackrel{\text{(s)}}{\stackrel{\text{(s)}}{\stackrel{\text{(s)}}{\stackrel{\text{(s)}}{\stackrel{\text{(s)}}{\stackrel{\text{(s)}}{\stackrel{\text{(s)}}{\stackrel{\text{(s)}}{\stackrel{\text{(s)}}{\stackrel{\text{(s)}}{\stackrel{\text{(s)}}{\stackrel{\text{(s)}}{\stackrel{\text{(s)}}{\stackrel{\text{(s)}}{\stackrel{\text{(s)}}{\stackrel{\text{(s)}}{\stackrel{\text{(s)}}{\stackrel{\text{(s)}}{\stackrel{\text{(s)}}{\stackrel{\text{(s)}}{\stackrel{\text{(s)}}{\stackrel{\text{(s)}}{\stackrel{\text{(s)}}{\stackrel{\text{(s)}}{\stackrel{\text{(s)}}{\stackrel{\text{(s)}}{\stackrel{\text{(s)}}{\stackrel{\text{(s)}}{\stackrel{(s)}}{\stackrel{(s)}{\stackrel{(s)}}{\stackrel{(s)}{\stackrel{(s)}}{\stackrel{(s)}{\stackrel{(s)}}{\stackrel{(s)}{\stackrel{(s)}}{\stackrel{(s)}{\stackrel{(s)}}{\stackrel{(s)}{\stackrel{(s)}}{\stackrel{(s)}{\stackrel{(s)}}{\stackrel{(s)}{\stackrel{(s)}}{\stackrel{(s)}{\stackrel{(s)}}{\stackrel{(s)}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}{\stackrel{(s)}}{\stackrel{(s)}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}{\stackrel{(s)}}$ 

#### Step 21: Viewing the results

Once the calculation has ended, the button <sup>100</sup>/<sub>100</sub> for showing the reinforcement results will become active.

Visualize for example the lower reinforcement parallel to the local z'-as  $A_{zi}$  in the plates. As expected, the lower reinforcement is situated under the columns of the main building.



On the other hand we find the larges upper reinforcement in the fields between the walls and the columns. Here we visualize the upper reinforcement parallel to the local z'-axis.



The peak reinforcement under the columns probably follows from the stress verification in SLS. In that place extra compression reinforcement will be provided to limit the compression stresses in SLS QP and RC.

As an example we turn off the stress verifications in SLS for a moment:

- Choose the instruction ' Edit Material library'
- In the left column select the material 'Concrete C25/30'.
- Then click on the button 🗈. So we make a user-defined material for which we can change its properties.
- Click on the tab page 'Advanced'
- Select the standard 'Eurocode 2 : EN 1992-1-1 [--]'.
- In the tab 'concrete' in deselect the options for the allowable stresses.

conc	Name (25/30(1)
Concrete C12/15     Concrete C16/20     Concrete C20/25     Concrete C25/30     Concrete C35/37     Concrete C35/45	Name     CC5/30(1)       Material type     concrete       Mechanical properties     Thermal properties       If     Eurocode 2 : EN 1992-1-1
Concrete C40/50     Concrete C45/55     Concrete C50/60     Concrete M15 (IS)     Concrete M20 (IS)     Concrete M20 (IS)     Concrete M30 (IS)     Concrete M30 (IS)     Concrete M35 (IS)     Concrete M40 (IS)     Concrete M45 (IS)     Concrete M45 (IS)	Concrete       Reinforcement         Compr. strength fck       25.0         Tension strength fct       2.6         Y <sub>c</sub> 1.5         Creep factor O(t, to) for stress limit       1.4         Creep factor O(t, to) for deformation       2.0         Maximum allowable compressive stress for rare combinations
= 'cunaete c23/30(1)	15.0     N/mm²     I after creep     0.6 fck       Aximum allowable compressive stress for quasi-permanent combinations       11.3     N/mm²     I after creep       0.45 fck

Click on to save the changes. Close both dialog window with 'OK'. Assign this new material to the plate using the button in the Geometry configuration.

Restart the elastic analysis and the calculation of the reinforcement  $\overset{\textcircled{}}{\textcircled{}}$ . Then you'll obtain the following result for  $A_{zs}$ :



Now we see no upper reinforcement is needed under the columns in ULS. Which proves that the upper reinforcement came from the stress verification in the service limit states.

For the further progress of the exercise, set the material back to the default material 'Concrete C25/30' and recalculate the model.

# 3.3.6 Calculating the cracked deformation

#### **Step 22: Assigning practical reinforcement to the beams**

We provide the foundation slab with a practical rebar mat:

- Go to the 'Results' configuration 2.
- Visualize one of the four reinforcements results on plates.
- Double click the foundation slab and plate a rebar mat B8-150 in the upper and lower reinforcement.

9	Diamonds - BetonVB3.bsf - [Fenêtre 1 - Az-sup. in plate (mm²/m)]	- 🗆 🗙
🥮 Eile Edit View Select	Display Analysis Options Windows Help	- 8 ×
	● ♀ □ 田田田 Ⅲ ■ 2 20 6 18 五 9 18 1     ■ 10	Dorien 🔻 📕 🖵
💽 🐻 🖷 🔀 Fenêtre 1	・ 「1」 Results ・ 一 務 中 図 マ 本 や Q Q 図 / ● 直 配 翌	
<b>F.7</b> 00 00	max = 1105	Active level foundation
	Practical Reinforcement Quantities	× nd level
	Image: Signal state in the second state is second state in the second state in the second state in the second state is second state in the second state in the second state is second state in the second state in the second state is second state in the second state in the second state is second state in the second state in the second state is second state in the second state in the second state is second state in the second state in the second state is second state in the second s	.00 m .00 m .00 m .00 m .00 m
	Number of bars :         6 ◆         Number of bars :         6 ◆           1000 000000000000000000000000000000000	
	$A_{x1}^{(r)}$ $J_{333}^{(r)}$ mm²/m       ### $\chi_{x1}^{(r)}$ $J_{333}^{(r)}$ mm²/m         Max calculated :       4202       mm²/m       Max calculated :       4268       mm²/m         Number of bars :       6 $\clubsuit$ Number of bars :       6 $\clubsuit$	
	Help Qancel OK	
		1.

Confirm with 'OK'.

#### Step 23: Calculating the cracked deformation

Choose the menu command 'Analysis – Cracked deformation' of click on the button  $\overline{223}$  in the icon bar. Leave the parameter  $\beta$  unchanged and select that you wish to take creep into account.

#### Step 24: Viewing the results

Below you can see the cracked deformation  $\delta_y$  after creep for the SLS QP envelope 'min'.



The settlement has not increase a lot compared to the elastic deformation. However, the largest settlement occurs not only at the edges of the plate but also in the area where the plate supports the two central columns.

When we evaluate the cracking of the plate, we can understand the increase in settlement in that zone.

Click on the button  $\bigvee$  in the pallet and select the cracking width  $w_{k,j}$  to show the cracking width in the bottom of the plate. We opt for the combination 'SLS QP max'.



Indeed, the foundation slab under the central columns in significantly cracked. The stiffness of the plate will also decrease in that zone which explains the increase in settlement.

Note: In §3.4 this model will be further expanded. Hence, you should save this model.

# 3.4 Example 4: Making a 3D model with bearing walls in masonry

Required licenses:	✓ 2D Bars	✓ 3D Bars	
	✓ 2D Slabs	✓ 2D Plates	✓ 3D Plates
	✓ Concrete Design		

Instead of calculation the preslab and the foundation slab separately, you can opt to enter the building as a whole. Where you have made some simple assumptions concerning the dissent of loads in the first examples, this time the actual stiffness of the supporting walls and column will be taken into account and the descent of loads will be done 'automatically'. A difference in stiffness as well as the differential settling of the foundation my cause the deflection pattern and the resultant reinforcement differ from the simple 2D model.



The fact that this is a classic building consisting of a few levels, allows us to use the level manager and its functionalities in Diamonds. We repeat that the level height of the ground level is 3,2m and 2,8m of all other floors.

To gain time, we won't remodel he floor slab but use the 2D model we made in §3.2. The model will be built up in different steps:

- 1. First we will expand the 2Dmodel to a full floor. In other words, we will remove the rigid support and replace them with walls and columns with a limited rigidity.
- 2. Then we copy this floor either using the translation function, either using the level manager depending on only a part of the floor or the entire floor should be copied.
- 3. Finally we define the foundation slab and the soil layers.

All walls are constructed of masonry for which you can find the elastic properties in the table below:

Ε	6170 N/mm <sup>2</sup>	
ν 0,15		
α	0,000008 /C	
ρ	1700 kg/m <sup>3</sup>	

The columns are made of reinforced concrete C25/30 and have a circular cross-section with a diameter of 300mm.

Once the geometry of the model is completed and the loads are places on the foundation slab, we calculate the structure. We review briefly some results.

## **3.4.1 Defining the structure**

Step 1: Open the project from §3.1

Open the file from §3.2 using the menu command 'File – Open' or click on  $\overrightarrow{P}$ . Next click on the icon  $\overrightarrow{P}$  in the icon bar go to the Geometry configuration. The preslab is displayed with the following model information:

Diamonds - BetonVE		Window Configuration	×
File Edit View Select Display Analysis Options Windows Help			
D 🚔 📕 🖪 🚳 🚺 🖉 🕾   田田田   田 🖬 🚺 📓 🧝 🧞 🎸	Current configuration :		Save current configuration
R B R Venster 1 Viser configuration	User configuration	•	Delete selected configuration
N, X <mark>g ≥ g ⊗ g ⊗ g ⊗ g ⊗ g</mark> e ≥ a e	Configuration name Geometry		
	General Geometry Mesh		
	Points Line	ies	Surfaces
	I Show point I Sh	Show axis	Transparency : 50 🔷 %
	Point number	Line number	Surface number
r 📕 📲 a sa sa a sa sa sa sa 🚦 a sa sa	Connection name	Name	Name
l i a ser ser a fra ser ser se 🖡 ser ser se		Length	Thickness
夕田 徑		Section orientation	T Area
AT 45 1		Local coordinate system	Local coordinate system
Ly B <sup>+</sup> 1 L	V 1	End releases	✓ End releases
南, 27 段		Material	Material
		Border lines	🗇 Border lines
How we have a set of the set of t		Buckling length Y	Practice reinforcement
🗰 🖉 👘 👘 👘		Buckling length Z	✓ Orientation
W 5. The second se		Distance between LT supports	
💑 🖌 🚫 r ka		z>0	
$\Rightarrow \equiv \pi$		Distance between LT supports	
		2<0	
		L1 supports z>0	
		L1 supports z<0	
	Help		

This plate is normally drawn in the horizontal drawing plane Y = 2,7m. This drawing plane corresponds to "Level 1" of the manager level. You can

check that the plate actually belongs to this floor by double-clicking it. At the bottom you discover to which level to the selected element belongs.

🥹 Diamonds - Be	Properties of plate number: 1
🥮 Eile Edit View Select Display Analysis Options Windows Help	
	General
	Name Prédallen 🗨
📓 🗐 🖷 🔀   Venster 1 🔄 📘 User configuration	
$\mathbf{k}$ $\mathbf{X}$	Shape
	Dimensions Stiffness matrix
е д д	thickness (a) [180.0] mm
	(1) [50.0 mm
夕靈 狸   ■   ■   ■   ■	
τ́́ R↓ I **! · <del>Έσαυασασασα κ</del> ερααταρα	
$\Lambda^{*}$ $\mathcal{H}^{*}$ $\mathcal{R}$	
Ho 🎇 殿 . A a a a a a a a a a a a a a a a a a a	- Duran Kar
🛲 👷 📴 🛛 a a a a a a a a a a a	
	Material Concrete C25/3C
💑 🖉 🕗 na	Local axes
	Local xz-axes orientation 90.0 • 12
-G(e h 4**	
	Local y-axis orientation
	Level = Level 1
· · · · · · · · · · · · · · · · · · ·	Help Cancel OK
b)	

Note: if you draw the plate from §3.1 in another horizontal plane, please first select the entire plate and translate it to the correct level (Y= 2,7m) using the function  $\overrightarrow{Y}$ ).

#### Step 2: Adjusting the level manager

Click on *Level manager* in the pallet on the right-hand side. The following dialog box appears:

<b>\$</b>	Diamonds - BetonVB2.bsf - [V	Vindow 1 (m)]		- 🗆 🗙
🍄 Eile Edit View Select Display Analysis O	ptions <u>W</u> indows <u>H</u> elp			- 8 ×
	ਸਿ≣∐ਙਿ⊠⊠⊻੬ਟਿਫ਼ਾ⊡	9 F% I   F 🖩 🖗		
🗑 🖥 🖄 Window 1 💽 🚺	User configuration	<u>■</u> 🔄 🏦 📥 🥙 🗨		
K, X N M M				Active level
				2.70 m
	😌 Level mana	iger	×	ΥΗΗ <sup>1</sup> Στηβ:≱ 2.70 m
		-		<u>⊬</u> ∰n'
	New Divide	Conv Delete		Level manager
		copy belete		Ground level
17 m 3	A b Level			X = 0.00 m
		ਸ਼ਮਾ <u></u>  ਸ਼ਿਮਾ		Y = 2.70 m
	🖉 ⊳ Level 1	2,70 m 2,70 m		Representation
	A b foundation	0,00 m 0,00 m		* ***
<i>翻读</i> 韵 <b>*</b>			64 SK	<b>A</b>
			- 16 - 16	· · · · · · · · · · · · · · · · · · ·
			8 X	Size Font 25 🜩
	1			Symbols 15 📚
<u>– – – – – – – – – – – – – – – – – – – </u>			8.8	Loads 15 🚖
	Help	Cancel OK		Results 15 🚖
				Show groups
				None
				Z <sup>X</sup>
D				

Here you will find a list of all levels in the project. At present, only two levels defined: a foundation level to 0,00m and a first level with floor height 2,70 m. Now select the 'Level 1' and edit the floor height to 3,20m. The level of the floor is automatically adjusted.

<b>9</b>	Diamonds - BetonVB2.bsf - [Window 1 (m)]	- 🗆 🗙
🥮 <u>F</u> ile <u>E</u> dit <u>V</u> iew <u>S</u> elect <u>D</u> isplay <u>A</u> nalysis	2ptions <u>W</u> indows <u>H</u> elp	- 8 ×
🗋 🖬 🖬 🖪 🚳 🚺 🖉 🖓 🗐	田   Ⅲ	
🗑 🖷 🛛 Window 1 💌 🚺	User configuration 💽 🏟 🖽 💹 🤿 🔍 🔍 🔝	
	<u> </u>	Active level
<sup>4</sup> <sup>p</sup> ∧ , <del>β ⊳ β ⊘</del>		Level 1 💌
		2.70 m
	💝 Level manager 🗙	
		Evel manager
	New Divide Copy Delete	Ground level
夕田登  -		🛱 Drawing plane
₽ <sub>8</sub> 5 P+1		X = 0.00 m Y = 2.70 m
◎ 水 灸   ● ■ ● ● ● ●		Z = 0.00 m
	🖉 🏷 Level 1 2,70 m <sup>3,20</sup>	Representation
	// 🏷 foundation 0,00 m 0,00 m	
	4     <sup>  </sup>	🕸 🏘
		Size
te h Il		Font 25 🚖
E		Symbols 15 🚖
		Loads 15 호
ed ed 50e 50e 50e	Help <u>C</u> ancel <u>O</u> K	Results 15
et et to 15		Show groups
et et in in in		
	AL 18 18 18 18 18 18 18 18 18 18 18 18 18	· · · [ ]* = = =

Note: With 'level' we mean the slab with underlying walls and columns.

9	Diamonds - BetonVB2.bsf - [Window 1 (m)]	- 🗆 🗙
Eile Edit View Select Display Analysis	<u>O</u> ptions <u>W</u> indows <u>H</u> elp	- 8 ×
	▋╡ <u> </u>   ⊻ % ⊕ ⊡ %   <b>♪ ☆ ێ 圖 図   ₽</b>   <b>!</b>   11	1 10
🗑 🖗 🔀 Window 1 💽 🚺	] User configuration 💽 🕅 🖽 🕅 🛃 🏥	→ 《? Q Q 🖾 🥒 🖻 🗷 🖾 🖾 🖾
		Active level
		× 2.70 m
	/ Lever manager	FH-1
🔊 🕸 🕺 🗛 🛔	New Divide Copy De	lete
		Cround level
27 .5 Pt		x = 0.00 m
T H,J I	Cover ground level 3,20 m 3,20 m	Y = 2,70 m Z = 0.00 m
	/ isolation 0,00 m 0,00 m	Representation
Ĩ⇔ / ⊘ · <mark>¤ ¤ ¤</mark>		Size
	1	Font 25 🗲
		Symbols 15 文
<u> </u>	Help Cancel	QK Loads 15 🛬
40 40 59 59		Results 15 🛫
11 11 12 12 13	<mark>.</mark>	Show groups
40 40 00 00 0 <u>0</u>		None 💌
ta ta az az az		Z <sup>X</sup> Z
D		

Close this dialog window and verify if the coordinates of the nodes indeed have been adjusted. Double click for example a single node:



Close this window with 'Cancel'.

#### Step 3: Replace point and line supports with walls and columns

We now wish to replace all point and line supports with columns and walls.

We start by removing the supports.

- Select the entire model (CTRL+A).
- Then click on the button in the 'Geometry' configuration and go the tab page 'Lines.

The locks a on the right side make you aware that the X-, Y- and Z-displacement was not set fixed for all bars. You should therefore unlock them before you can set all the displacements free.

Image: Support       Image: Support         Image: Support       Image: Support         Image: Support       Image: Support	- # ×
D So In Control International Supports     X       Supports     X       Image: Support	Active level
	Active level
Points Lines Surfaces	Active level
	3.20 m
	3.20 m
	Evel manager
X displacement X rotation	Ground level
今 Wil 沙 vs. at better 「Free ■	Drawing plane
F and P+1 control con	X = 0.00 m
Y displacement Y rotation	Z = 0.00 m
cannot resist to tension	Representation
I T Control result result for a control result of the control resu	\$ \$ \$
	D 40
	Font 25 📚
$z_{\alpha=0}^{2}$ $\alpha=0$ $\alpha z_{y}^{2}$ Restraint defined relative to line orientation	Symbols 15 🚖
💙 α το	Loads 15 🚖
	Results 15 🜩
Lieb Cancel QK	Show groups
	None
	X STATE

Repeat this procedure for the tab page 'Points'. Click 'OK'.

#### Step 4: Drawing walls

In the pallet click on the button to start drawing walls. Drawing walls is analogous to drawing lines with that difference that a surface (wall) with a height equal to the story height is immediately drawn under the line. Once all the lines to which a line support was granted are redrawn, click

to stop the drawing function.

Choose for a perspective view the right dialog box and discover the walls have just drawn. The figure below is displayed from a lower point of view.

Image: Second	- 🗆 🗙
🤗 Eile Edit View Select Display Analysis Options Windows Help	- 8 ×
□☞ ■ 집● ◎ ∽ ~   時田田 Ⅲ Ⅲ Ⅲ 図 圖 ※ 秒 8 盃 ⊕ ◎ ☞	
	Active level
	Cover ground le
	3.20 m
	3.20 m
	Evel manager
	Ground level
夕田登	道- Drawing plane
PT af PH	X = 0.00 m Y = 3.20 m
The strength and the st	Z = 0.00 m
	Representation
	- 🔊 🌒
	Font 25
	Symbols 15
	Loads 15
the state	Results 15 🜩
	Show groups
	▼ None ▼
	Z ¥X

#### Step 5: Editing the properties of the walls

Select all walls:

- Either you use the menu instruction 'Select Vertical plates'.
- Either you select them by click on one plate using the left mouse button while keeping the CTRL-key pressed in. This selection mode

is possible since all walls defined using the  $\square$  function as stored in a separate design type.

Click on the button *in the pallet to change the name and thickness of the walls. We leave the material untouched for the moment.* 

Pie Edit Vew Select Display Analysis Options Windows   Def D   Aff · /   D d d   D d d   D d d   D d d   D d d   D d d   D d d   D d d   D d d   D d d   D d d   D d d   D d d   D d d   D d d   D d d   D d d   D d d   D d d   D d d   D d d   D d d <	🥹 Diar	Properties of plate number: 4, 5, 6, 7
Properties   Material Concrete C25/3C T   Gross cover   Local axes   Local axes   Local y-axis orientation   Image: Cover ground level     Help     Cancel	File       Edit       Yiew       Select       Display       Analysis       Options       Windows         Image: Select       Windows       Image: Select       User configuration         Image: Select       Image: Select       Image: Select       Image: Select       Image: Select         Image: Select       Image: Select       Image: Select       Image: Select       Image: Select       Image: Select       Image: Select       Image: Select       Image: Select       Image: Select       Image: Select       Image: Select       Image: Select       Image: Select       Image: Select       Image: Select       Image: Select       Image: Select       Image: Select       Image: Select       Image: Select       Image: Select       Image: Select       Image: Select       Image: Select       Image: Select       Image: Select       Image: Select       Image: Select       Image: Select       Image: Select       Image: Select       Image: Select       Image: Select       Image: Select       Image: Select       Image: Select       Image: Select <t< th=""><th>General Name Muur 200,0 Shape Dimensions Stiffness matrix thickness (e) 190.0 mm</th></t<>	General Name Muur 200,0 Shape Dimensions Stiffness matrix thickness (e) 190.0 mm
		Properties         Material Concrete C25/3C $\checkmark$ Local axes         Local x2-axes orientation         0.0       • $2 \times \frac{1}{2}$ Local y-axis orientation $\frac{3}{2} \times \frac{2}{2}$ Local y-axis orientation $\frac{3}{2} \times \frac{2}{2}$ Level = Cover ground level         Help       Cancel

Note: if the model would already contains walls with different properties (name, plate type and thickness), you can assign this to the selected walls (or plates) using the pull down menu in the geometry pallet. The material will always refer to the default material.

#### Step 6: Expanding the material library

The walls are currently made of concrete. Because the material doesn't contain a material with the properties of masonry, we have to define a new material:

- Open the material library using the menu command 'Edit Material library'.
- Click on the button •.



First you'll be asked to which type the new material belongs to. The material type is important for the type of verification that can be performed after the elastic analysis. The verification for masonry is not included in Diamonds, so no additional verification can be made for this material. We select the forth option 'Other' and click 'OK'.

The new material is included in the library. Remains for us to note the name and the elastic properties:



Confirm with 'OK'.

#### Step 7: Assigning the material 'Masonry' to the walls

We now assign the material 'Masonry' to the walls:

- Select all walls.
- Click on the button
- Select the just defined material from the list.
- Then click 'OK'.

Filter	-	
<ul> <li>Material type</li> </ul>	$\mathbf{x}$ $\mathbf{y}$ $\mathbf{J}_{z}^{A}$ $\mathbf{f}_{z}^{A}$	
Steel	Search	
Concrete		
Timber	#Timber GL32c	^
Other	# Timber GL32n	
	# Timber LVL ULTRALAM-R	
	#Timber LVL ULTRALAM-Rs	- 10
	Timber LVL ULTRALAM-X	
Clear all	I Masonry	*

#### Step 8: Impose the behaviour of masonry to the walls

The masonry walls cannot prevent the floor slab from being lifted up from the walls. In addition, for such an elements will always be assumed that along its edges no moments can be transferred.

These very specific conditions can be attributed to vertical walls in Diamonds in a simple manner:

- Select all walls.
- Click on the button  $\checkmark$  in the geometry pallet.
- On top, choose for the option 'No moment transfer along the edges'. Also check that no tension and no arch effect is allowed.

The masonry walls have mainly a load descending function, not a load spreading function.



Confirmed with 'OK'. All walls are now marked with the symbol



Note: you should be aware that this symbol indicates that there are only a number of specific conditions along the edges of the element imposed. It does not mean that also the material properties of masonry are assigned, nor that no tensile stress may arise in the element.

#### **Step 9: Defining the columns**

Now the properties of the walls are adjusted, we can start defining the columns.

- Choose a top view.
- Click on the button  $\square$  in the pallet.
- To draw a column, simply click a point in the model window. Under this point, a vertical line is immediately generated. The height of the line again corresponds to the story height of the current level.
- Define a column among all the nodes which previously were a point support.
- End the drawing function through button
- Again opt for a 3D view and you'll obtain this image:



#### Step 10: Changing the cross-section of the columns

All column have by default a rectangular cross section of 200x200mm in reinforced concrete C25/30. To change the cross section:

- Use the CTRL-key to select all columns at once (this is possible because columns defined with are in a separated design type).
- Click on in the pallet. Complete the window like this:



Note: through the pull down menu under the drawing functions in the 'Geometry' pallet you'll find the already defined cross-sections.



So you can immediately select the correct cross-section after you clicked on

#### Step 11: Making the first floor

With this operation the geometry of the ground level is completed. As mentioned in the introduction, only the main building will be provided of floors. Thus we only need to copy a part of the structure.

- Select the part you wish to copy. Choose a top view and draw a selection window around the main building. This way you will select all elements belonging to the active floor (thus the floor slab, the supporting walls and the columns).
- Then click on the button in the 'Geometry' pallet and copy the elements once (N =1) in the positive Y-direction. Make sure the distance is equal to the height of the selected level (3,2m). If not, the elements will overlap. Confirm with 'OK'.



Now choose a front view and select the upper floor slab with edges (use a selection window). Click on  $\overrightarrow{r}$  and perform a translation (N = 0) this time in the negative Y-direction.

<del></del>	Diamonds - BetonVB2.bsf - [Window 1 (m)]	
🥮 Eile Edit View	ct Display Analysis Options Windows Help	
0 🛩 🖬 🖪 🤅	▶ ◎ ○ 田田田 Ⅲ № 図目光投る  ※五① ◎ ☞	
🗑 🐻 🖷 🔀 (Wind	✓ 【 User configuration ・ ● ● ■ 図 ○ お ゆ ペ ○ ● ■ ○ □ ○ □ ○ □ ○ □ □ □ □ □ □ □ □ □ □ □ □	
$\mathbf{k}$ $\mathbf{X}$	Active level	
08		
∞ ·		
000		
9 🖑	rada da babababababababababababababababab	×
₩ ¥	nanna a bha	
<u> 19 10 21</u>		
-27 .5 P+1		
<u>niri 11 700</u>		
HT 🐑 🎝		
GEET AFE RES	dy: -0,40 m	
un s. tol	dz: 0.00 m	
Ĩ⇔ \$ <>		
🔜 🔜 Te	rescale physical elements	
	Cancel OK	
3		
9		
9	n n n n n n n n n n n n n n n n n n n	
9	None V	
1	i i i i i i i i i i i i i i i i i i i	

Again click 'OK'. The floor slab of the second level is now located at the level +6,00m, the height of the first level is 2,80m.

#### Step 12: Expand the level manager

Although the model consists of two levels (ground level + first floor), Diamonds will recognize only one. Open the level manager with the button Level manager and click on New to add a new level. Change the name to 'Cover first floor'. Set the height to 2,8m.

	New Level ×
Level name	Cover floor 1
On top of	Cover ground level
Level height	2,80 m
	<u>C</u> ancel <u>O</u> K

The level will be adjusted automatically.

<b>&gt;</b>	Level manager											
	Ne	w	Divide Copy Del									
Ø	۵	Level										
Ø	4	Cover	floor 1	6,00 m	6,00 m 2,80 m							
Ø	٨	Cover	ground level	3,20 m	3,20 m							
0	٨	found	ation	0,00 m	0,00 m 0,00 m							
0	• ♦	found	ation	0,00 m	3,20 m 3,20 m 0,00 m							
						-1						
į	<u>H</u> elp			<u>C</u> ancel	<u>O</u> K							

From the moment you click 'OK', all points, lines and surfaces belonging to the new floor will be recognized by Diamonds. For example double click on one of the nodes located at a height of +6,00m.

Properties of point number: 31
Coordinates
X = 0,00 m
Y = 6,00 m
Z = 10,00 m
Footing
Level = Cover floor 1
Help Cancel

#### Step 13: Making the second and third level

We will now continue with the definition of the second and third level.

Again, we can copy the desired parts of the structure. However, since this floor is identical to the first level, there's a much faster method.

In the level manager select the level 'Cover first floor' and click on the button Copy . Complete the window like here below. This feature has the additional advantage that the new floor is directly placed above the current floor.

coj	py a level ×
level to copy	Cover floor 1
number of copy	1
begin copy on top of	Cover floor 1
Help	<u>C</u> ancel <u>Q</u> K

Confirm with 'OK' and wait until the second level is assigned to the model. Note that the new level has the same height as the original level. Change the name to 'Cover second floor' and click 'OK'.



All levels have been defined.

#### Step 14: Modelling the foundation slab

The last thing we have to do is model the foundation slab. There are two options:

- You could redraw the foundation slab in the current project. Opt for a top view and select the level 'Foundation' from the pull down list on the right. Note that on the level (+0,00m) a number of points and lines have already been defined, namely the lower edges of the walls and the feet of the column of the level 'Cover ground level'. Follow the steps in §3.3.1 to complete the model. Use a spring constant of 1400kN/m<sup>2</sup> instead of the soil layers.

Or you could import the Diamonds file from the already modelled foundation slab from §3.3, including all sections, supports and loads. Via the menu command 'File – Import – Import Diamonds' choose the correct file. Enter the new coordinates for the point (0,0,0) or indicate them directly on the screen. In this case the point (0,0,0) comes to lie at the origin of the 3D model. Confirm with 'OK'.

<del>\$</del>	Import Diamonds file settings – 🗖 🗙
Geometry The origin ( 0,0,0 ) becom 0,0 0,0	nes 0,0 m
Loads 	Selfweight
$\begin{array}{ccc} & & & & \\ & & & & \\ & & & & \\ & & & & $	dead loads     dead loads     T
	Help Cancel QK

When importing you can decide which load groups should be transferred and whether new load groups should be created if necessary. **Only import the selfweight!** 

In case the foundation slab was imported on the wrong level, click on the button in and import the foundation slag again on the right level.

Both options give the same result.

Here below we show the model in a solid representation 4.



Note we didn't provide any hinges at the column ends. This means that the columns are rigidly connected to the beams and the plates.

### **3.4.2 Defining the loads**

Step 15: Go to the 'Loads' configuration

We now leave the 'Geometry' configuration and activate the 'Loads' configuration to enter the loads. Click on the button 🖽 in the icon bar or select in the adjacent pull down menu the 'Loads' configuration.

### 3.4.2.1 Creating the load groups

Step 16: Creating load groups

The load groups do not need to be regenerated. After all, we started from a file which already contained load groups.

If desired, you can retrieve them using the button  $\frac{\gamma_g}{2}$ .

### 3.4.2.2 Filling up the load groups

Step 17: Filling in the load groups 'Self-weight', 'Dead loads' and 'Life load'

We now continue with entering the various loads.

There will be some loads already defined. While copying and importing the model the loads were also included. We go over:

- The **self-weight** of the beams is calculated automatically by Diamonds and cannot be adjusted.
- Loads in the load group 'dead loads'
  - On the roofs
  - Select both roofs using the SHIFT-key.
  - Click on the icon <sup>■</sup>.

A table appears with a summary of all the dead loads defined on the selected plates.

- Click with the left mouse button on a value and change it to an uniform load of 1kN/m<sup>2</sup>.
- Repeat these steps until all values are set to 1kN/m<sup>2</sup>.
- Diamonds BetonVB2.bsf [Window 1 dead loads (kN, kNm, mm, kN/m, kNm/m, kN/m<sup>2</sup>, °C)] - 🗆 🗙 File Edit View Select Display Analysis Options Windows Help - 8 × 😵 🐻 🖷 🔀 🕅 Window 1 Loads 🛨 📾 🖽 🖂 📌 🖻 🖑 🍳 🍳 🖉 🖉 🔳  $\gamma_g^{\gamma_u} \stackrel{\text{and}}{\underline{\lim}} \stackrel{\mathfrak{G}}{\rightleftharpoons}$ Active level Cover ground le 💌 dead loads (3.0 3.20 m Y 3.0) 3.20 m \* 7 E Level man 4 2 2 Ground level 922 Drawing plane 1.00  $\begin{array}{rrrr} X = & 0.00 \mbox{ m} \\ Y = & 3.20 \mbox{ m} \\ Z = & 0.00 \mbox{ m} \end{array}$ Temperature 0 Snov Representation Wind 888 Dynamic \_ 🗆 🗙 Data Loads on surfaces dead loads ▼ Case 1 point 1 point 2 point 3 Value 1 Value 2 Value 3 load surface orientation (m) (m) (m) (kN/m², ℃ (kN/m², °C) (kN/m<sup>2</sup>, °C) (kN/m², ℃ Set all values (1;0;0) (0;1;0) (0;0;1) 3 1,00 1,00 1,00 1,00 global Y to 1kN/m<sup>2</sup> (1;6;0) (0;7;0) (0;6;1) 3.00 1.00 3.00 global Y 14 15 (1;6;0) (0;7;0) (0;6;1 3.00 3.00 3.00 3,00 global Y 4 Help Parameters . . . . 🚑 🛕 当 Loads • Cancel  $\underline{A}$ pply modifications
- Click on Apply modifications to confirm the adjustments.

 On the two préslab floors and foundation slab 3kN/m<sup>2</sup> is applied.





- Loads in the load group 'life load'
  - o On the roofs.
  - o Select both roofs using the SHIFT-key.
  - $\circ$  Remove the loads with  $\frac{1}{2}$ .
  - o On the two préslab floors 2kN/m<sup>2</sup> is applied.
  - On the foundation slab 2kN/m<sup>2</sup> is applied.

### 3.4.2.3 Making combinations

Step 18: Making combinations

Generate the combinations  $\frac{414}{4m}$  as described in §3.1.3.3

### 3.4.3 Generating the mesh

Step 19: Generating the mesh

Generate the mesh  $\square$  as described §3.2.4.

#### Step 20: Verifying the mesh

Once the mesh is generated, we make the mesh visible with  $\square$ . In a 3D view the model now looks like this:



We don't see any irregularities, so we can expect good quality results.

### 3.4.4 The global elastic analysis

Step 21: Elastic analysis

To start the analysis, select the command 'Analysis' – 'Elastic Analysis'. You can also start the analysis directly using the function key **F9** or use the icon icon bar. Use the same settings as in §3.3.4 (and make sure the ground level is set to 0,00m!).

This time an iterative calculation is performed. The reason for this is definition of the masonry walls: they can't bear tension at the top. You don't know in advance whether a load combination will cause tension in the top of the plates or not. The fact that for some combinations, multiple iterations are necessary indicates that some plates indeed will lift up (in the corners).

We don't pay attention to the results of the elastic analysis for this example.

### 3.4.5 Calculating the reinforcement

#### Step 22: Calculating the reinforcement

We calculate the reinforcement <sup>(b)</sup> with the same settings as in §3.2.6. Also here Diamonds will give the message that for some plates the reinforcement could not be calculated. We fail to comply with this warning and click 'OK'.

#### **Step 23: Viewing the results**

Once the calculation has ended, the button <sup>100</sup>/<sub>100</sub> for showing the reinforcement results will become active.

Visualize for example the lower reinforcement parallel to the local x'-as  $A_{xi}$  in the plates.



Although the supports of the base plate will follow the settlement of the foundation, the reinforcement in the fields (apart from those in the corners of the preslabs) is practically the same as obtained in §3.2.

Again choose for a 3D view and select the longitudinal reinforcement  $A_y$  in the beams and columns.

Optionally, you can also select elements and only make the results visible for this selection using the buttons  $\mathbb{H}$ ,  $\mathbb{H}$  and  $\mathbb{H}$ . The colour scale will adapt each time as a function of the visible elements in the model window.



Note: The actual buckling length of the columns was not taken into account when calculating the reinforcement. Since no calculation of the buckling lengths was performed, the buckling lengths will be set equal to the group length (in this case the length of the columns). Because all column in this model are build-in (en thus the buckling length will be smaller than the system length) we will obtain a safe result.

# 3.4.6 Calculating the cracked deformation

#### Step 24: Assigning practical reinforcement to the beams

Before starting the calculation of the cracked deformation, we verify if all beams and plate are provided with practical reinforcement.

When copying or importing the floor, the practical reinforcement is included. In principle, the practical reinforcement should be in order for all levels.

When you didn't import the foundation slab, but modelled it again from scratch, you should assign practical reinforcement to it. Therefor double click the foundation slab (make sure you're looking at a reinforcement result on plates) and complete the dialog window as below:

Practical Reinford	ement Quantities ×
Image: Signal state     Signal state     mm²/m       Max calculated :     1330     mm²/m       Number of bars :     6     €	335     mm²/m       Max calculated : 391     mm²/m       Number of bars :     6
335     mm²/m       Asi     335       Max calculated :     476       Mumber of bars :     6	335     mm²/m       Max calculated : 365     mm²/m       Number of bars :     6 ♀
Help	<u>C</u> ancel <u>OK</u>

Confirm with 'OK'.

#### Step 25: Calculating the cracked deformation

Choose the menu command 'Analysis – Cracked deformation' or click on the button  $\overline{\mathfrak{m}}$  in the icon bar. Leave the parameter  $\beta$  unchanged and select that you wish to take creep into account.

#### Step 26: Looking at the results

We show the maximum cracked deformation under the combination 'SLS QP min' for the floor slab on the first level.



Note: In §3.5 this model will be further expanded. Hence, you should save this model.

# 3.5 Example 5: Modelling the basement

Required licenses:	✓ 2D Bars	✓ 3D Bars	
-	✓ 2D Slabs	✓ 2D Plates	✓ 3D Plates
	✓ Concrete Design		

In the final step we model and calculate the basement. We assume that only the main building has a basement, which makes the model will contain two foundation slabs on a different level. We use the results of the cone penetration test on both foundation slabs.

The basement has a height of 2,8m. The thickness of the walls is 20cm. the columns from the above levels are assumed to be continuous. The dead load and life are  $3kN/m^2$  and  $2kN/m^2$ .

The water level is 1m below the ground level. The water level can be considered as permanent.



### 3.5.1 Defining the structure

#### Step 1: Open the project from §3.4

Open the model from §3.4 using the menu instruction 'File – Open' or click on  $\stackrel{\frown}{\Longrightarrow}$ . Then click on the icon  $\stackrel{\frown}{\boxtimes}$  in the icon bar to go to the 'Geometry' configuration. Opt for a top view. Make sure the plan of the foundation slab is visible.

#### Step 2: Deleting the old boundary conditions

Before expanding the model, we remove the soil layers we assigned to the foundation slab:

- Select the foundation slab.
- In the pallet click on the button
- Set all displacements free and confirm with 'OK'.

#### **Step 3: Drawing the basement walls**

Next draw with *lines* the two missing lines of the basement walls.

÷											Di	amo	nds	- Be	ton\	/B4.b	osf - [	Wind	dow	1 (r	n)]									- 🗆 🗙
Ψ.	Eile <u>E</u> dit	View	Sele	ct [	<u>)</u> ispla	y A	nalysi	s <u>O</u> p	tions	Win	dows	Hel	р																	- 8 ×
	🖌 🕻	1 3	6	D	кЛ	CH .	F	ΠF	Ħ   S	8	T	×		X Z	e e	<b>,</b>   @	ś <u>20</u>	(£) (	- % I	£	IF.	₿.	4							
	58	XW	indow 1				-	1	Use	r config	guratio	'n				*	酚	<b>F11</b>			南	声	877	Q @	20	1		R		
1	-	$\sim$							0.52									a. 112			3301			- 14		1. 2.				Active level
7	P	$\sim$	55	28	8	82	38	- 15	8	- 22	33	8	32	-8	35	8	32	- 53	35	8	18	8	1	38	98	8	88	36	28	Foundation 💌
		/	8	20	T															1	2	2	12	8	82	53	82	8	82	0.00 m
ſ	Ø	O	8	12			32		20		192	Ξ.	10	5		~		53			- 22	2	8	8	12	28	82	8	12	2-12 0.00 m
6	AUS .		15	12			- 6														12	- 13	18	15	12	12	85	14	32	KHH.
~	~		88	æ					13			э.					÷				35	-		8	22	35	<u>8</u> 9	8	25	E Level manager
									• 2				×.																	Ground level
14	× [4]4]														10						<u></u>					ť.				🔁 Drawing plane
4	T 8Ĵ	P.:													8					1										X = 0.00 m
1	.UA	0	10	100			- 1						÷.	•		a.		<ul> <li>.</li> </ul>	- 14						Ŷ.		34	18	<i>w</i>	Z = 0.00 m
	· 24 )	1	8	10			2		t:		1.2	а. Т					÷	<u>6</u> 2	N		*		( - 18 1		,	- 51	59	8	10	Representation
1 11	52.		13	8										5		L	÷	5	.)						Ċ.	-25	33	13	8	
節	J Ste		8	12			3		20									8	<b>K</b>							2	83	8	12	D 🔊
X	s de	0>	14	8											•					_	÷		- 2		•	-87	85	22	8	
	Š =	The	8	8	32	3	- 63	- 33	30	- 85	- 24	10	- 8	- 85	19	a.									÷.	$\cdot v$	28	8	33	Size Font 0 🜩
	<u> </u>			28	63			13	33	-22	.8	8	32	-8		2	*	-2		3					4	-8	88	38	28	Symbols 0 🗲
1			100	25	62	22		25	42	10.	2.5	22		82											<i></i>		131		25	Loads 10 🚖
				8	- 22	8		8	- 22	- 10	2	8	-	58		4		- 22		-	a a				4		8	12	8	Results 15 🗲
			-	- 22	10		-		20	27	01	55	-03	10					1.2					- 14	2		V.1			Show groups
																										1				None 💌
			52	98	63	82	92	15	63	- 52	33	81	30	- 12	35	80	32	-11	13	8	<u>98</u>	8	1	56	98	63	93	1		age age
			8	25	- 53	8	8	~	23	- 33	82	35	20	- 53	10	- 25	- 8	- 53	12	- 25	- 22	2	- 85	8	- 22	- 53	85	8	z×	active and a
																														1

Once the button is pressed again, Diamonds will verify if new plate contours can be formed. The foundation will be divided into two plates for which the enclosed plate will represent the ceiling of the basement.

#### **Step 4: Defining the basement**

Now draw a selection window around the basement contour and click on the button  $\checkmark$  in the pallet.



Copy the plate over a distance of -2,80m and let Diamonds connect the two plates by creating the walls. Confirm with 'OK' and choose a 3D view: an extra floor is added to the model.



The thickness of the foundation slab of the basement is 30cm. For the basement walls a default thickness of 20cm is assumed.

#### **Step 5: Expanding the level manager**

Although the model contains a basement, this floor is not yet recognized by the level manager. Open the level manager with the button  $\square$  Level manager and click on  $\square$  to add a new level under the foundation with a height of 2,8m. Change the name to 'Basement floor'.

	New Level ×
Level name On top of Level height	Basement Below foundation
	<u>C</u> ancel <u>Q</u> K

Optionally you can change the name of the 'foundation' to 'cover basement'.

	Level manager										
	Ne	w Divide	Divide Copy								
Ø	⊳	Level	<b>F</b>								
Ø	٨	Cover floor 2	8,8 m	2,8 m							
Ø	٨	Cover floor 1	6,0 m	2,8 m							
Ø	٨	Cover ground level	3,2 m	3,2 m							
Ø	4	Cover basement	0,0 m	2,8 m							
Ø	٨	Basement	-2,8 m	0,0 m							

Note: the hidden levels will be drawn in light grey. If you wish to hide them completely, then check the option 'Show hidden parts greyed' in the tab 'General' with the configuration settings.

#### Step 6: Cutting the model to easy define the water pressure

To define the water pressures later, it is necessary to mark the part of the basement below the water level. For that reason we will cut through the basement with a horizontal reference plane Y = -1m.

- Select all walls of the basement.
- Click on the button in the pallet.
- Complete the dialog window like this:



Click 'OK'. At the height of the cut a cutting line is defined which divides the walls. Now the basement walls under Y=-1,00m can be selected separately.



Note: as an alternative the basement can be split using the button <sup>Divide</sup> in the level manager. Then the basement walls will be organized in two separate levels. In the contest of this exercise, this solution may be less relevant.

#### Step 7: Assigning types

In order to facilitate selecting in the future, we wish for the foundation slabs and the cellar walls under the water level to stand out from the rest of the structure by the assignment of two new design types.

- Select both foundation slabs with the SHIFT-key (take a view where you look from the bottom of the model, i.e. move the vertical slider downward).
- Click on the button 🍱.
- Define a new type 'Foundation slab' and choose a colour.



Click 'OK' to assign the new design type to the selected elements.

Then select (again use the SHIFT key) all basement walls below the water level (optionally turn the model using the horizontal slider) and again click on **1**. Make a new design type 'Basement walls' and choose a different colour . Confirm with 'OK'.



#### Step 8: Assigning a cross section to the columns

To the two bars (columns) in the middle of the basement, no cross-section was assigned. We therefore assign a circular cross section to the two lines:

- Take a wireframe representation
- Select one of the columns with section R300.
- Click once with the right mouse button and click on 'Copy properties'.



- Now select the two columns in the basement.
- Click with the right mouse button and select 'Paste Section, material and type'.



#### Step 9: Defining the soil properties

We now define the soil properties.

- Select the two foundation slabs using the CTRL-key.
- Click on the button
- Prevent the X- and Z-movement.
- Use the soil layers in the Y-direction.

We'd like to use the same soil profile as in §3.3. Since we saved the different soil layers in a \*.txt-file, so now we can import is using the button Finder layers. Indicate the relevant \*.txt-file and click 'Open'. The following soil layers should appear:



Close all dialog window with 'OK'. The soil layers is now assigned to both foundation slabs although they lay on a different level.

Note: This soil layer profile is defined relative to a certain reference level (i.e. the original ground level). During the analysis you'll be asked to fix this reference plane. Once this as happened, Diamonds doesn't only know the soil layers immediately below the various foundation but also the depth of the excavation so that any preloading of the terrain can be taken into account. Therefore the same soil layers profile is used for both foundation slabs!

#### Step 10: Give the local axis of the plates the same orientation

In principle the geometry of the structure is finished now. However, before moving to the 'Loads' configuration, it is advisable to adjust the local orientation of the basement walls and the foundation slab below the water level in such a way that the local y'-axis is always directed outward. Doing so, the water pressures can be applied in one operation.

Open the dialog box with the configuration settings 1 and visualize the local coordinate system of the surfaces.



In the image above only the elements below the ground water level are visible. Now select the two basement walls in the back and the foundation

slab and click on the button 🔍 in the pallet. Change the orientation of the local coordinate system by selecting the second button.

Notice that the plate orientation is accompanied by a lock is which means that these properties are not identical for all selected elements. Only the 'free' properties will be applied on the selected elements when you close this window.

🥹 Diamonds - Be	Properties of plate number: 21, 25, 27, 29
Eile Edit View Select Display Analysis Options Windows Help	
	General
😨 🗟 🗮 🔀 Window 1 💽 🚺 User configuration	
	Shape v
	Dimensions Suttress matrix
	thidoness (e) 300.0 mm f
	Properties
	Material Concrete C25/3C - T Gross cover
	Local xz-axes orientation 0.0 • 2 x
	Local y-axis orientation
	<u>H</u> elp <u>Cancel QK</u>
1	

As you can see, the direction of the local y'-axis have changed. The x' and z' are changed so that the right-handed coordinate system is maintained.

### 3.5.2 Defining the loads

Step 11: Go to the 'Loads' configuration

We now leave the 'Geometry' configuration and activate the 'Loads' configuration to enter the loads. Click on the button  $\square$  in the icon bar or select in the adjacent pull down menu the 'Loads' configuration.

### 3.5.2.1 Creating the load groups

#### Step 12: Creating load groups

The dead and life load acting on the foundation slab of the basement are already defined since there were copied with the plate.

Because the water level is situated above the base of the cellar foundation, an extra load group is required to defined the water pressures:

- Click on the button  $\gamma_{g}^{\gamma_{u}}$
- Add a new load group with the button Add load group. When we assume that the level of the ground water does not vary, we can see it as a dead load. Thus the following coefficients apply:

÷	l.	Diamon	ds - Be	etonVB4	4.bsf - [\	Vindow	/ 1 - Sel	fweight	- (kN,	kNm, m	nm, kN/r	m, kNm/m, kN/m²,	°C)]		- 0	x
🌳 <u>F</u> ile	<u>Edit View Select Displa</u>	y <u>A</u> naly	ysis <u>O</u> pt	ions <u>W</u>	indows	<u>H</u> elp									-	- E ×
	6 🖬 🖪 🖉 🗠	~   F	H H H	7   III	T		X Ke	6 8	<u>111</u> (9)	9% IÝ		¢ +∌				
*	🕅 🖷 🔀 🕅 Window 1	-	1	Loads				•	<b>B</b> ••• [		髀」	+ «) Q Q 🖾		R		百
y y u 111 O Active level															i j	
Selfwe	ight 🚽														Foundation	
															FTR.I o.	00 m
10						1									2.1	80 m
/riq	τ^		<u> </u>							_						~
₩							Loa	d grou	ups							~
								_			- 11	r				
Loa	Load factors for EN 1990 V V Consequence dass 2 Service dass 1 Service dass 1															
Design lifetime 50 🚖 years																
Add load group Insert load group Delete load group Several load gases per group																
									Derette	iouu groi	-p					
	Name load group	γ <sub>uls-</sub>	$\gamma_{uls+}$	$\gamma_{\rm sls}$	$\gamma_{\rm sls+}$	Ψ <sub>0</sub>	$\psi_1$	ψ2	φ	ξ	t <sub>o</sub>	Combination for cracking	k <sub>mod</sub>	Load	Action	
$\checkmark$	Selfweight	1,35	1,00	1,00	1,00	1,00	1,00	1,00	1,00	0,85	0		perm	—	<u>+++</u>	
<ul> <li>✓</li> </ul>	dead loads	1,35	1,00	1,00	1,00	1,00	1,00	1,00	1,00	0,85	0		perm	-	<u>+++</u>	
<ul> <li>✓</li> </ul>	life loads A : housing	1,50	0,00	1,00	0,00	0,70	0,50	0,30	1,00	1,00	0		mediu	-	<u>+++</u>	
<ul> <li>✓</li> </ul>	Water pressure	1,35	1,00	1,00	1,00	1,00	1,00	1,00	1,00	0,85	0		perm	-	<u>+++</u>	
Tecompatible land groups							Combine loadcases					Ingroup loadcases				
	Incompatible load groups			inked lo	ad group	s			Comh	ine load	cases	Line	roun loade	ases		
	Incompatible load groups			Linked lo	ad group	s			Comb	ine load	cases	Ung	jroup loadc	ases		
	Incompatible load groups			Linked lo	ad group	is			Comb	ine load	cases	Ung	proup loadc	ases	<u></u> ĸ	

Confirm with 'OK'.

If the load combinations where already defined in the model, Diamonds will give you a warning they are no longer conform to the newly defined load groups. If desired, you can indicate that you don't wish to see this message in the future, but we recommend you don't to prevent possible errors.



### 3.5.2.2 Filling up the load groups

Step 13: Filling in the load groups 'Self-weight', 'Dead loads' and 'Life load'

These load are already on the structure.

#### Step 14: Filling in the load group 'Water pressure'

- Select the load group 'Water pressure' in the pull down list.
- Select all elements below the water level (use the CTRL key).
- Click on #. The following dialog box shows:



- First you indicate the surface load should be applied locally.
- Secondly, you will use a function specifically designed for defining ground and water pressures. Enter the size of the load on two know levels and Diamonds will generate a load which varies linearly in the depth.

Click on 'OK' and you'll obtain this result:



You'll discover a trapezoidal load on the basement walls and a uniform distributed upward load on the foundation slab.

### 3.5.2.3 Making combinations

#### Step 15: Making combinations

Generate the combinations  $\frac{4}{4\pi}$  as described in §3.1.3.3

### 3.5.3 Generating the mesh

#### Step 16: Generating the mesh

Generate the mesh  $\square$  as described §3.2.4.

### 3.5.4 The global elastic analysis

#### Step 17: Elastic analysis

To start the analysis, select the command 'Analysis' – 'Elastic Analysis'. You can also start the analysis directly using the function key **F9** or use the icon  $\blacksquare$  on the icon bar. Use the same settings as in §3.3.4 (and make sure the ground level is set to 0,00m!).

Continue the calculations with 'OK'.

#### Step 18: Go to the 'Results' configuration

To see the results of the calculations, you click on  $\square$  in the icon bar or select in the adjacent pull down menu the 'Results' configuration.

Below, we list some results.



Axial forces in the columns ULS FC envelope



Elastic vertical deformation  $\delta_{y}$  for SLS QP envelope min



Soil stresses SLS RC envelope max

### 3.5.5 Calculating the reinforcement

Step 19: Calculating the reinforcement

For calculating the reinforcement we assume the same settings as in §3.4.5.

However, it is also possible to perform the reinforcement calculation only for a few pre-selected elements. Therefore, select two foundation plates with the CTRL-key and click on the icon 😤 in the icon bar. The following dialog appears:



Confirm with 'OK' and visualize the reinforcement  $A_{xi}$ .



Note, given the local orientation of the lower plate has changed,  $A_{xi}$  stands for the upper reinforcement in the local x'-direction!

## 4 Examples in steel

In this chapter we model a shed consisting of 5 frames. We start from a 2D model (§4.1) which we expand later to a 3D-model (§4.2).



When calculating the 2D model we focus on the wind and snow generator. In the 3D model we use the surface load generator.

The calculation methods used for both examples are the same. The only difference is that in the 3D model we will calculate a connection in detail using PowerConnect (which is also possible in the 2D model).

We don't make a report for this example. The report manager is explained in the second example in reinforcement (§3.2.8). The principle is the same.

### 4.1 Example 1: 2D frame

Required licenses:

✓ 2D Bars
 ✓ Steel Design

### 4.1.1 Purpose of the exercise

This example handles the calculation of a steel frame. We calculate the internal forces and stresses in the bars and we execute a steel verification (strength and stability).

The frame looks like this:



The steel quality is S235.

### 4.1.2 Defining the structure

Step 1: Go to the 'Geometry' configuration

Defining the structure is always done in the 'Geometry' configuration. Click on B in the icon bar, or select the 'Geometry' configuration in the adjacent pull down menu.
49													Dia	mon	ds -	[Ven	ster 1	(m)	]									- 🗆 🗙
🤗 <u>F</u> ile	<u>E</u> dit	Viev	v <u>S</u> e	elect	<u>D</u> ispla	ay <u>A</u>	Analysi	s <u>O</u>	ptions	Wir	ndows	Help																- 8 ×
	6	1 🗋	. 8	$\bigcirc$	6	$^{\circ}$	F	H	$\square$	#	Tei	M		X Ke	6	18	211	5) F9;	Ľ	IF	₿.	$\downarrow \Rightarrow$						
* 2		× v	enstei	1			•	1	Geo	metry	2					-	R	14 🕅		南	甴	<del>ا</del> م	Q	<b>Q</b> 🖾	1	DR		
		X																										Active level
∩⊞		,				- 22																						foundation •
455		~				•	X																					0.00 m
0	Ø	Ø	27	1	7	18	28	38	84	84	84	82	26	14	22	- 22	2	12	20	20	28	73	38	28	14 U	. 8	84	0.00 m
$\bigtriangledown$			10	10		20	155	10	81	84	25	33	12	12	8	8	8	8	5	10	22	23	10	10 	a 8	1 31		E Level manager
		Ŧ	10	10	88	22	88	28	83	85	38	38	13	13	2	82	2	68	40	10	28	33		*	т	on view		
14		211	20	35	87	8	23	-23	8	8	13	10	33	14	8	96	32	10	35	30	10	82		z.L.X	•	op nen		Cround level
1		64) 15. 5	20	31	10	8	85	23	35	35	12	10	15	16	56	56	12	12	31	21	12	17		-	-			X = 0.00 m
T.*	R↓J 1 117		88	8	30	32	-83	-8		28	25	85	8	8	52	12	8		8	23	32	30		PP	, Fi	ront vie	N	Y = 0.00 m
N/d	Mr,	12	- 22	22	12	12	12	12	24	24	94	97	2	12	32	32	22	48	2	52	12	72		Z				Representation
UT IO	21																								В	ack viev	/	BBB
翻	e Fe	Å¥. I⇔																						Z.L.X				
Ξ.	ہ دروں ہے اور	$\sim$	85	8	20	30	-22	-22	88	88	<i>5</i> 8	62	8	8	25	25	8	8	8	33	55	20	-					
		≪ π.	30	31	8	8	61	62	99	32	13	33	32	32	8	89	25	98	32	32	65	8		J.	. L	eft view		Size
-Ble	In	118	32	32	81	8	-57	-11	85	82	33	15	10	55	8	8	25	28	32	32	8	8		25 . 4				Font 25 -
- F)			10	22	28	25	12	53	83	87	12	32	2	12	3	35	8	8	0	10	25	28		V	R	ight vie	N	Symbols 15 🛫
				22	88	25	~	-	83	87	-22		2	12	*	8	8	*	2	63	25	88		Z=1=X				
					62	52	28	52	82	82			10	12	88	85		25	22	11	12	62		3	)	D		Results 15 -
							50																	z	. 3	D view		None 👻
					8	- 22			8	2 22		14	8		22				- 10			8	- 23	- 1			744	anget anget
			<u> </u>																								z×	
																												//

Then check if you are in a front view. If this is not the case, then click on

the button in the icon bar or on the button  $z^{1}x$  in the lower right corner and select the viewpoint 'Front view'. This way you activate a vertical drawing area.

#### Step 2: Structure generator

You can insert the structure in Diamonds using different ways:

- Draw immediately on the screen with the mouse
- Draw on the screen by means of coordinates with the keyboard.
- Use the structure generator  $\square$ .
- Import a DXF-file.

In this example we'll use the structure generator. Drawing a structures will be handled in example 2 §3.2.

Click on the icon in the pallet. A dialog window will appear in which you can select the form of the structure you would like to generate. Opt for a continuous beam.

Image: Optimized State   Provide State     Image: Optimized State   Optimized State     Image: Optized State   Opti	- 🗆 🗙
🥮 Eile Edit View Select Display Analysis Options Windows Help	_ # X
😨 🗟 🕾 🖄 Window 1 🖃 🚺 Geometry 🔄 🍘 🖻 🖾 🐨 🖉 🖗 (中) (*) 🗨 🖉 🖉 🖉	
$\mathbf{k}$ $\times$	Active level
	2.70 m
	2.70 m
Wizard menu ?	Evel manager
	Ground level
	Drawing plane
PH A A A A A A A A A A A A A A A A A A A	X = 0.00 m
· A A A	Z = 0.00 m
	Representation
	\$ \$ \$ \$
	A 🖓
	Size
	Font 15 🚖
	Symbols 15 🚖
	Loads 10 🚖
	Results 15 🚖
	Show groups
X	None 🛨
L,	×
	1

Next Diamonds will ask you the geometric data of the roof.

- First enter the slope of the beams and the width of the frame.

9	-	Wizard: Portal frame 🗕 🗖 🗙	- • ×
		Beams       ID,0       ○         Roof slope:       ID,0       ○         Number of intermediary nodes (left):       0       •         Width portal frame:       I4,0       m         Beam section       IPE 300 - S235       •         Columns       Number of intermediary nodes (left):       0       •         Number of intermediary nodes (left):       0       •       •         Columns       Number of intermediary nodes (left):       0       •         Left column height:       3,00       m       •         Number of intermediary nodes (right):       0       •       •         Right column height:       3,00       m       •         Column section       HEA 300 - S235       •       •         Im       Column section       •	
		Cancel OK	

- Change the cross section of the beams to an IPE 270.
  - Click on the button Beam section. You will then see the following window:



- Click on and select an IPE 270 from the list.
- The used material will be the default material, which is by default 'Steel S235'.
- Click 'OK'.
- Repeat the steps for the columns.
  - The height of both columns is 4m.
  - o Each column has one intermediary node.
  - The cross sections is a **HEA 260**.
- Choose for simple supports.
- Set the frame to be drawn in the origin.



Then click 'OK' to draw the structure:



Diamonds doesn't always show all the element information. You can select which data should be displayed. Click on 1. In the tab 'Geometry' select 'Name' under the title 'Line'.

*	Diamonds	Window Configuration	×
🥮 Eile Edit ⊻iew	<u>S</u> elect Display Analysis Options Windows Help ■	Current configuration :	Save current configuration
R D R Vens	ter 1		Delete selected configuration
	  	Configuration name Geometry General Geometry Mesh  Points Lines	Surfaces
	IPE 270	Show point     Show axis       Point number     Line number       Connection name     Name	Transparency : 50 🚖 %
Υ™Ω ₽π5PH ©₩A	HEA 260	Length     Section orientation     Local coordinate system	Thickness Area Local coordinate system
		☐ End releases ☐ Material ☐ Border lines	End releases     Material     Border lines
	X	Buckling length Y     Buckling length Z     Distance between LT supports	Practice reinforcement     Orientation
E		z>0 Distance between LT supports z<0	
		LT supports z>0     LT supports z<0	
1	15 AF 18 15 AF 18 15		
		Heb	Cancel QK

Note that with the pull down menu on top of the window you can edit these parameters also for the 'Loads', 'Mesh' and 'Results' configuration.

#### Step 3: Adding supports in the z-direction

To indicate the frame actually comes from a 3D hall, we will fix all point (except for the simple supports who are already fixed in the z-direction) in

the z-direction. Select the points and click on  $\frac{\overline{x}}{\overline{x}}$ .

Image: StaalVB1.bsf - [V]	enster 1 (m)] _ 🗖 🗙
♥ Eile Edit View Select Display Analysis Options Windows Help       □ 26     □     □     ○     □     □     □     □     ○     □     □     □     □     ○     □     □     □     □     ○     □     □     □     □     □     ○     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □     □ <th>D f‰ x′    ⊨ ⊯ +0</th>	D f‰ x′    ⊨ ⊯ +0
😨 🗟 🗮 🛛 Venster 1 🗨 🚺 User configuration 💌 🕅 F	Supports
	Points         A       T         A       T         A       T         A       T         A       T         A       T         A       T         A       T         A       T         A       T         A       T         A       T         A       T         Cannot resist to tension       T
	lis-

As a result you'll see 'dashes' appear in global Z-direction when you take a 3D view. These indicate visually that the frame can't move in the global Z-direction.



#### Step 4: Defining a physical group

Finally, we define that the column parts form a physical group.

#### About physical groups

A physical group consists of separate collinear bars that behave as if they were a continuous bar. Physical groups are useful when assigning variable sections, eccentricities and defining loads. In this example we will use physical groups to define a trapezoidal load easily to the columns.

Choose a solid representation . On the right you'll find a pallet that allows you to define physical groups. Choose 'Physical' from the pull down list. Right below you will find a pallet allowing you to define groups. The button allows you to group the selected bars, with you ungroup them.



Select the two column parts on the right and click on *the same for the column parts on the left.* 

# 4.1.3 Defining the loads

#### Step 5: Go to the 'Loads' configuration

We now leave the 'Geometry' configuration and activate the 'Loads' configuration to enter the loads. Click on the button 🖽 in the icon bar or select in the adjacent pull down menu the 'Loads' configuration.



#### About the 'Loads' configuration

With the 'Loads' configuration windows comes a separate pallet containing all the functions for defining loads and generating combinations. Note that the point of view remains unchanged when switching between the configurations.

### 4.1.3.1 Creating the load groups

#### Step 6: Creating load groups

Before defining any loads, you have to make the different load groups. Click on the button  $\gamma_g^{\gamma_u}$ . You'll see the following screen:

<del>9</del> [	Diamon	ids - S	taalVB1	l.bsf - [\	/enster	1 - Self	weight	(kN, I	kNm, m	m, kN/n	n, kNm/m, kN/m²,	°C)]		- 0	×
<u> </u>	Analys	sis <u>O</u> pt	ions <u>W</u>	indows	<u>H</u> elp										- 8 ×
06 🖬 🖪 🗑 🗠	∼    F;	HHH	7   III	Tei		X Ke	8 8:	<u>111</u> (B)	Fe% I%		¢+ 10				
😨 🗟 🖷 🔀 Venster 1	•	1	Loads				•	<b>R -</b>		R d	- Q Q ⊠	1	R		
y <sup>y</sup> u mt O												14		Active leve	el
Selfweight														Verdieping	91 <b>-</b>
														<b>冊和1</b> 2	2.70 m
NO ST															2.70 m
*n T*														fillevel	manager
<b>\$</b>						Loa	ad aro	ups							×
Load factors for	E	N 1990		•		-	Consec	uence d	ass [	2 ᅌ		Service d	ass 1	•	
	,			_	,	_	Design	lifetime	ŕ	50 🚖	vears				
									1		,				
Add load group	Ir	nsert loa	d group					Delete	load groi	Jp		Several	load cases	s per group	
	- 1							1	1	1					
Name load group	γ <sub>uls-</sub>	$\gamma_{uls+}$	$\gamma_{sls}$ -	$\gamma_{sls+}$	$\psi_0$	$\psi_1$	Ψ2	φ	ξ	to	Combination for cracking	k <sub>mod</sub>	Load	Action	
✓ Selfweight	1,35	1,00	1,00	1,00	1,00	1,00	1,00	1,00	0,85	0		perm	_	<u>+++</u>	
✓ dead loads	1,35	1,00	1,00	1,00	1,00	1,00	1,00	1,00	0,85	0		perm	_	<u>+++</u>	
✓ life loads A : housing ▼	1,50	0,00	1,00	0,00	0,70	0,50	0,30	1,00	1,00	0		mediu	—	<u>+++</u>	
Incompatible load groups Linked load groups Combine loadcases Ungroup loadcases															
Help												<u>C</u> an	el	<u>о</u> к	

#### About the window 'Load groups'

- In the menu on top you select to which standard the safety and combinations factors should answer. Currently this is set to 'EN 1990 [--]' which means Eurocode 0 without a national annex.
- In some national annexes the safety coefficients also depend on the consequence class and the design lifetime of the structure. Both are linked to the economic and/ or social interest of the structure. A higher/ longer consequence class/ design lifetime will lead to higher safety factors.
- On the right you can enter **the climate class**. This climate class is representative for a certain moisture content of the air/ the timber. Diamonds uses the climate class you determine the modification factor  $k_{mod}$ . The

modification factor  $k_{mod}$  takes the influence of the load duration and the moisture content on the strength properties into account. The modification factor  $k_{mod}$  depends not only on the climate class but also on the type of timber and the load duration class. The **load duration class** must be specified for each load case in the last column.

- In the table below the load cases 'Self-weight', 'Dead load' and 'Life load' are defined by default. You can freely rename or delete them, except for 'Self-weight'. The fill-in boxes to the right of the name of each load case include the safety *γ* and combination factors *Ψ* required for the automatic generation of the load combinations.
- We don't discuss the other parameters in this window.

#### Step 7: Changing the type of the life load

Change the type of 'Life load A: Housing' to 'Life load H: roofs'.

÷		Diamo	nds - S	taalVB	l.bsf - [	Venster	1 - Self	weight	- (kN, l	kNm, m	m, kN/n	n, kNm/m, kN/m²,	°C)]		- 0	×
🂝 <u>F</u> ile	<u>E</u> dit <u>V</u> iew <u>S</u> elect <u>D</u> ispl	ay <u>A</u> nal	ysis <u>O</u> p	tions <u>W</u>	indows	Help									-	. 8 ×
	۰ 🚺 🛃 🖥 🖗	C4	HHF	RI	Tii		X Ke	6 8	$\overline{\underline{m}} \oplus$	Fe% IV		I +0				
<b>*</b> E	🔊 🖷 🔀 🛛 Venster 1	•		Loads				•	<b>f</b>	0	🗩 🏚	∋ «୬ <b>୧ ସ</b>		RE		
$\gamma_g^{\gamma_u}$	tiu 📎														Active leve Verdieping	1
<b></b>							Loa	ad gro	ups							×
Loar	d factors for	ſ	EN 1990		•		•	Consec Design	juence d	ass [	2 文	years	Service d	ass 1	•	
	Add load group		Insert loa	ad group					Delete	load gro	qu		Several	load cases	s per group	
	Name load group	γ <sub>uls-</sub>	$\gamma_{uls+}$	$\gamma_{\rm sls}$	$\gamma_{\rm sls+}$	$\psi_0$	$\psi_1$	ψ2	φ	ξ	to	Combination for cracking	k <sub>mod</sub>	Load	Action	
✓	Selfweight	1,35	1,00	1,00	1,00	1,00	1,00	1,00	1,00	0,85	0		perm	_	<u>+++</u>	
<ul> <li>✓</li> </ul>	dead loads	1,35	1,00	1,00	1,00	1,00	1,00	1,00	1,00	0,85	0		perm	_	<u>+++</u>	
~	life loads H : roofs 🗸 🗸	1,50	0,00	1,00	0,00	0,00	0,00	0,00	1,00	1,00	0		mediu	—	<u>+++</u>	
	Ine loads G : heavy ve A Iffe loads H : roofs snow (H > 1000 m) snow (H <= 1000 m) Wind Temperature Seismic Crane loads $\checkmark$															
	Incompatible load groups Linked load groups Combine loadcases Ungroup loadcases															
	Help												<u>C</u> an	cel	<u>O</u> K	

#### Step 8: Creating load group 'wind'

Now we add the load group 'Wind':

- Click on Add load group
- From de predefined list, select the load type 'Wind'. Note that all the safety factors and combination coefficients change when you do this.

	(		Diamo	nds - S	staalVB	l.bsf - []	/enster	1 - Self	weight ·	- (kN, k	«Nm, m	m, kN/n	n, kNm/m, kN/m²,	°C)]		- 0	×
-	Eile	Edit View Select Displa	ay <u>A</u> nal	ysis <u>O</u> pt	tions <u>W</u>	indows	Help		Turk			() 	en son il				_ 8 ×
	) 🖻	• 🖬 🖪 🖨 🖣 🗠 •	а	H H A	A 📰	<b>T</b>		L Fe	୫ ାଞ୍ଚ	<u>m</u> (9	Fe% IX		<b>1</b> +0				
1	¥ 5	Venster 1	•		Loads				<b>_</b>	<b>B</b> •		AR ¢	∋  १७ <b>Q Q ⊠</b>				
	γ <sub>σ</sub> <sup>γ</sup> u	यांग 📚														Active leve	5
9								Loa	ad gro	ups							×
	Load	factors for	[	EN 1990		•		•	Conseq Design	uence cla lifetime	ass	2 🔹	years	Service d	ass 1	•	
		Add load group		Insert loa	ad group					Delete l	load grou	чр		Several	load cases	per group	
		Name load group	γ <sub>uls-</sub>	$\gamma_{uls+}$	$\gamma_{\rm sls}$ -	$\gamma_{\rm sls+}$	ψ <sub>0</sub>	$\psi_1$	ψ2	φ	ξ	to	Combination for cracking	k <sub>mod</sub>	Load	Action	
	<ul> <li>Image: A start of the start of</li></ul>	Selfweight	1,35	1,00	1,00	1,00	1,00	1,00	1,00	1,00	0,85	0		perm	_	<u>+++</u>	
	1	dead loads	1,35	1,00	1,00	1,00	1,00	1,00	1,00	1,00	0,85	0		perm	_	<u>+++</u>	
	✓.	life loads H : roofs	1,50	0,00	1,00	0,00	0,00	0,00	0,00	1,00	1,00	0		mediu	_	<u>+++</u>	
	~	Wind 👻	1,50	0,00	1,00	0,00	0,60	0,20	0,00	1,00	1,00	0		short	(Pao	$\pm \pm \pm$	
	life loads G : heavy ve ∧ life loads H : roofs snow (H > 1000 m) snow (H <= 1000 m) Wind Temperature Seismic Crane loads ▼																
	Incompatible load groups Linked load groups Combine loadcases Ungroup loadcases																
	Help Cancel QK																

We consider 16 cases of wind:

- Wind left upward -> right upward ( $c_{pi} = -0.3$ )
- Wind left upward -> right downward ( $c_{pi} = -0.3$ )
- Wind left downward -> right upward ( $c_{pi} = -0.3$ )
- Wind left downward -> right downward ( $c_{pi} = -0.3$ )
- Wind left upward -> right upward ( $c_{pi} = 0,2$ )
- Wind left upward -> right downward ( $c_{pi} = 0,2$ )
- Wind left downward -> right upward ( $c_{pi} = 0,2$ )
- Wind left downward -> right downward ( $c_{pi} = 0,2$ )
- Wind right upward -> left upward ( $c_{pi} = -0.3$ )
- Wind right upward -> left downward ( $c_{ni} = -0.3$ )
- Wind right downward -> left upward ( $c_{pi} = -0.3$ )
- Wind right downward -> left downward ( $c_{pi} = -0.3$ )
- Wind right upward -> left upward ( $c_{pi} = 0.2$ )
- Wind right upward -> left downward ( $c_{pi} = 0,2$ )
- Wind right downward -> left upward ( $c_{pi} = 0,2$ )
- Wind right downward -> left downward ( $c_{pi} = 0,2$ )

We will implement these 16 cases of wind as 'sub load cases'. To define sub load cases check the option Several load cases per group .

Next to each load group you can now indicate how many sub load cases you want. You've also noticed the column with the little people in it ( $\frac{1}{2}$  or  $\frac{3}{2}$ ). Click on the icons ( $\frac{3}{2}$  or  $\frac{3}{2}$ ) until they don't hold hands  $\frac{3}{2}$ .

#### About 'Sub load cases'

'The little people' can hold hands or not. Click on the icon to switch between the two.

- If they do not hold hands **R**, this means that all sub load cases are incompatible (i.e. the sub load cases can never act together). This is for example the case with wind and snow.
- If they hold hands ﷺ, this means that all sub load cases act together. This is for example the case with dead loads.

Jau	factors for		JEN .	1990 _ [	3 8		lesign <mark>l</mark> if	ence das fetime	5	0 \$	years		JEI VIL	le uass	1				
	Add load group		Inse	ert load group				Delete lo	ad group	, ]							Several lo	oad cases p	er group <b>[</b>
	Name load group	#	Туре	Name load case	$\gamma_{\rm uls}$	$\gamma_{uls+}$	$\gamma_{\rm sls}$ .	$\gamma_{sls+}$	$\psi_0$	$\boldsymbol{\psi}_1$	ψ <sub>2</sub>	φ	ξ	to	Combinati	k <sub>mod</sub>	Load	Action	
1	Selfweight	1	-	2	1,35	1,00	1,00	1,00	1,00	1,00	1,00	1,00	0,85	0	1	perm	-	. <u>++</u>	
1	dead loads	1			1,35	1,00	1,00	1,00	1,00	1,00	1,00	1,00	0,85	0		perm	8 <del></del> 8	ŦŦŦ	
~	lif life loads H : roofs	1			1,50	0,00	1,00	0,00	0,00	0,00	0,00	1,00	1,00	0		mediu	1 <del></del> ))	+++	
1	Wind	16	ł£	wind l u -> r u (-0.3)	1,50	0,00	1,00	0,00	0,60	0,20	0,00	1,00	1,00	0		short	p	4+4	
				wind l u -> r d (-0.3)															
				wind l d -> r u (-0.3)															
				wind l d -> r d (-0.3)															
				wind l u -> r u (0.2)															
				wind l u -> r d (0.2)															
				wind I d -> u (0.2)															
				wind l d -> r d (0.2)															
				wind r u -> l u (-0.3)															
				wind r u -> l d (-0.3)															
				wind r d -> l u (-0.3)															
				wind r d -> l d (-0.3)															
				wind r u -> l u (0.2)															
				wind r u -> l d (0.2)															
				wind r d -> l u (0.2)															
				wind r d -> l d (0.2)															
	•	-	1			t			a lander		1	Lie			1				

#### Complete the table as below:

Note: because we calculate a 2D frame, we will not generate wind out of the plane, but this can also be done with the wind generator.

#### Step 9: Creating the load group 'snow'

Finally add the load group 'Snow':

- Click on Add load group.
- From de predefined list, select the load type 'Snow (H  $\leq$  1000m)'.
- Define three sub load case, namely:
  - o Case 1
  - o Case 2
  - o Case 3

oad factors for		EN	1990 💽 -	51 - K	•	Consequ Design lif	ence clas etime	s 2	¢ 0 ¢	years		Servi	ce class	1				
Add load group		Inse	ert load group				Delete lo	ad group								Several lo	oad cases p	er group 🎵
Name load group	#	Туре	Name load case	γ <sub>uls-</sub>	$\gamma_{uls+}$	$\gamma_{\rm sls}$ -	$\gamma_{sls+}$	$\psi_0$	$\psi_1$	ψ2	φ	ξ	t <sub>o</sub>	Combinat cracking	k <sub>mod</sub>	Load	Action	
			wind l u -> r u (-0.3)					-										
			wind l u -> r d (-0.3)															
			wind l d -> r u (-0.3)															
			wind l d -> r d (-0.3)															
			wind l u -> r u (0.2)															
			wind l u -> r d (0.2)															
			wind l d -> u (0.2)															
			wind l d -> r d (0.2)															
			wind r u -> l u (-0.3)															
			wind r u ->1 d (-0.3)															
			wind r d -> l u (-0.3)															
			wind r d ->1 d (-0.3)															
			wind r u -> l u (0.2)															
			wind r u -> l d (0.2)															
			wind r d -> l u (0.2)															
🖌 snow (H <= 1000 n	) 3	Ł	Case 1	1,50	0,00	1,00	0,00	0,50	0,20	0,00	1,00	1,00	0		short	*	4+4	
			Case 2															
			Case 3															
Incompatible load or	ouns	1	Linked load groups		1		Combin	e loadca	ses	1	Lb	ogroup la	adcases	.				

### 4.1.3.2 Filling up the load groups

Now the loads groups are defined, we can assign loads to the structure.

#### Step 10: Filling in the load groups 'Self-weight', 'Dead loads' and 'Life load'



- The self-weight is calculated automatically by Diamonds.

- On both rafters we add a permanent load.
  - Use the pull down-menu to activate the load group 'Dead loads'.
  - Now select the rafters and click on the button <sup>1</sup>. Note that only those icons will be active that can be applied on the selected elements.

🧇 Diamonds - StaalVI	81.bsf - [Venster 1 - dead lo	ads - (kN, kNm, mm, kN	l/m, kNm/m, kN/m	², °C)]	- 🗆 🗙
	<u>W</u> indows <u>H</u> elp				- 8 ×
	🚺 🖬 🕅 🖬 🗶 रेख ठ	양: <u></u> () F% 파      부	<b>□</b> +∂		
😨 🗟 🖷 🔀 Venster 1 💽 🚺 Loads		• 🖗 🗉 🖾 🗖 🎢	# * QQ		
y <sup>y</sup> H O					Active level
					verdieping 1
		-			2.70 m
¥ 17		Force on line	e	×	2.70 m
		start value	0.0	kN/m	Evel manager
		and value	0.0	khi/m	Ground level
<b>↓</b> → ≫	~~ <b>V</b> 3 <sub>V</sub>	enu value	10.0	NAU	Drawing plane
₩ 2 Δ × ×		start value	3.0	kN/m	X = 0.00 m Y = 2.70 m
* Temperature	xk	end value	3.0	kN/m	Z = 0.00 m
v Snow	Z	start value	0.0	kN/m	Representation
▼ Seismic		end value	0.0	kN/m	\$P \$P \$P
Vynamic     Moving	×< VSy		1		D 🔊
	c (*	Bar length = 7.11 m			
		Physical element length	= 7.11 m		Font 20 🜩
	0	distance from begin	0.00	m	Symbols 20 🚖
		distance from end	0.00	m	Loads 10 🚖
			related to group 1 on	de'	Results 15 🗢
			riciated to group Loa		Show groups
	Help		Cancel	<u>o</u> k	None
					1
					12

o Complete the window as follows:

After confirming with 'OK', you'll obtain this image:



Using the image above, verify if you have entered the loads correctly. If you made a mistake:

- o Double click the bar and edit values in the table.
- $\circ$  OR select the incorrect bars, delete the loads with  $\frac{3}{2}$  and regenerate the loads.
- Add a distributed load of 2kN/m to the rafters in the load group 'Life Load H: Roofs' using the same method.



On both columns we add a trapezoidal load (0,50kN/m to 0,75kN/m).

Select the column on the right and click on  $\checkmark$ . Complete the dialog window like here below. Make sure the option 'Values related to the physical element' is selected. Because the column is defined as a physical group, Diamonds will apply the trapezoidal load over the entire column and not on each column part separately.



Do the same for the column parts on the left, but this time enter negative values (-0,50 and -0,75 instead of 0,50 and 0,75).



#### Step 11: Filling in the load group 'wind'

#### To generate wind:



- Choose the load group 'wind' and the first sub load case 'wind I up -> r down (-0.3)' from the pull down menu.
- Click on The to define the wind standard and the terrain parameters.

	Terrain parameters ×	
	Norm: EN 1991-1-4 •	Active level
yg <sup>y</sup> u that v Wind v Geval 1 v	Reduction with factor 0.85 for lack of correlation	Verdieping 1
****	Direction factor Cdir: 1.0	Level manager
+ > \$ H &	Season factor Cseason 1.0	Ground level
Temperature     Snow     Wind	Air density: 1.3 kg/m³ I Probability factor Cprob: 1.0	Z = 0.00 m Representation
Seismic	C         V <sub>b,0</sub> by location:         Map           (●         v <sub>b,0</sub> = wind velocity:         26         m/s	Size
V Dyname	Terrain Type:     I     ✓     Choose	Font 20 🔶 Symbols 20 🜩 Loads 10 🜩
	Orography coefficient	Results 15 🚖 Show groups
	<u>H</u> elp <u>Cancel QK</u>	

- Select the standard EN 1991-1-4 [--].
- Opt for a basic wind velocity of 26m/s and a terrain type I.
- Click 'OK' to close this window.
- Select the entire structure ('CTRL+A' or use a selection window) and click on 😤 to start the wind generator on frames.





 Complete the window as above. Then click 'OK' to generate the wind.



#### About the wind generator

- The upper part presents a copy of the selected structure with the end points taken into consideration. The bars in bold represent the periphery of the frame on which the wind is expected to act.
- In the box 'Geometry' you note the total depth of the structure and the position of the frame relative to the façade (FRONT) and the distance between the previous and next frame.
- The image on the right shows you a floor plan of the structure indicating the position of the frame relative to the front and back of the building. Also the previous and next frame are shown in a dotted line. The coordinate system clearly explains the orientation of the building.
- In the tabs 'left -> right', 'right -> left', 'front -> back' and 'back -> front' you
  indicate which wind case should be generated, which internal pressure
  coefficient c<sub>pi</sub> should be taken into account and in which sub load cases the
  generated wind should be placed.

#### Step 12: Filling in the load group 'snow'

#### To generate **snow**:

- Select the load group 'Snow' and the first sub load case 'Case 1' from the pull down menu.
- Click on \* to select the snow standard and the terrain parameters.

<b></b>	Diamonds - StaalVB1.bsf - [Window 1 - snow (H <= 1000 m) - Geval 1 (kN, kNm, mm, kN/m, kNm/m, kN/m <sup>2</sup> , °C)] ↔ – □ 🗙
Sile Edit View	Select Display Analysis Options Windows Help
	●♥   ♡ ○   H:H H III   ■   図 ■ X 2/ 6   窓辺 9 1 I F ■ や
😿 🗟 🖷 🖂   Win	dow 1 🔄 🚺   Loads 🚽 🏟 🖻 🖾 🛃   244 (한 Q, Q, Q)   24   25   25   25   25   25   25   25
γ <sup>γ</sup> <sub>g</sub> <u>im</u> .⊕	Active level
snow (H <= 10(_▼	Settings snow generator
Geval 1 💌	
Ma P	Norm: EN 1991-1-3 – Zone: Central West –
1 10 12	
+ 	Area: 3 V Map
04 G	Altitude: 0.00 m
	Snow load at ground level (Sk): 0.41 kV/m <sup>2</sup>
Temperature     Enow	
*	V = 1.0: Shysk = 1.0
807	Ce: Exposure coefficient
Wind     Seismic	Ct: thermal coefficient
* Dynamic	
Y Moving	Take into account snow weight on overhang at edge
	Help Cancel OK
	None 💌

- Select the standard EN 1991-1-3 [--].
- Choose as country [--] 'Central West' 'Area 3'.
- Click 'OK' to close this window.
- Select the entire structure ('CTRL + A' or use a selection window) and click on To start the snow generator on frames.

♥ File     Edit     View     Select     Display       □     ∅     ∅     ∅     ∅     ∅     ∅       ∅     ∅     ∅     ∅     ∅     ∅     ∅       ∅     ∅     ∅     ∅     ∅     ∅     ∅       √     ∞     ∞     ∞     ∞     ∞       smow     ∅     ∞     ∅     ∅     ∅		
Geval 1 ** +7 + + -5 ** -5 ** ** ** ** ** ** ** ** ** *		
Vind Vind Vorseinic Vorseinic Vorseinic Vorseinic	Width supported by structural frame:       5,00       m         Snow load cases:       Image: snow (H <= 1000 m)       Image: Case 1       Image: snow (H <= 1000 m)         Image: snow (H <= 1000 m)       Image: Case 2       Image: snow (H <= 1000 m)       Image: Case 3       Image: snow (H <= 1000 m)         Image: snow (H <= 1000 m)       Image: Case 3       Image: snow (H <= 1000 m)       Image: Case 3       Image: snow (H <= 1000 m)         Image: snow (H <= 1000 m)       Image: Case 3       Image: snow (H <= 1000 m)       Image: Case 3       Image: snow (H <= 1000 m)       Image: snow (H <	<u>o</u> k

 Complete the window as here above. Then click 'OK' to generate the snow.



#### About the Snow generator

- The **upper part** presents a copy of the selected structure. The bars in bold represent the periphery of the frame on which the snow is expected to act.
- In **the middle** of this window you enter the width supported by the frame.
- In the lower part you indicate the snow cases that should be generated and in which sub load case they should be placed.

## 4.1.3.3 Making combinations

Before starting the calculations we need to generate the combinations first.

#### Step 13: Making combinations

111

Click on the button in the pallet in the 'Loads' configuration window.

Click on the button <sup># Generate all combinations automatically</sup>, indicate in the pull-down menu that you wish to use the classic but conservative Eq. 6.10 and select all limit states.



After pressing 'OK', all the combinations required by the standard will show, grouped by limit state. If desired, you can change these combinations Modify a combination or define combinations yourself <u>Create a combination</u>. One combination can be deleted with <u>Delete a combination</u>. To delete all combinations click <u>Delete all combinations</u>.

÷				Loads combin	ations	<b>+</b>	-	×
	🖸 Generate	all combinatior	ns automaticall	y	Create a co	mbination		
ULS	FC (ultimate	limit state - fu	ndamental com	bination)		•		
Load ULS ULS SLS SLS	d group FC (ultimate SC (ultimate FI (ultimate I RC (servicea FC (servicea	limit state - fu limit state - se imit state - fire bility limit state bility limit state	ndamental com ismic combinati 2) 2 - rare combin 2 - frequent com	bination) on) ation) mbination)		H <= 1000 m) 50 x 1.50 50 x 1.50	-	^
SLS 4	QP (servicea ULS FC 4	bility limit state 1.00 x 1.35	e - quasi-perma 1.00 x 1.35	anent combination) 0.00	0.00	0.00		
5	ULS FC 5	1.00 x 1.00	1.00 x 1.35	1.00 x 1.50	0.60 x 1.50	0.50 x 1.50		
6	ULS FC 6	1.00 x 1.00	1.00 x 1.35	0.00	1.00 x 1.50	0.50 x 1.50	]	
7	ULS FC 7	1.00 x 1.00	1.00 x 1.35	0.00	0.60 x 1.50	1.00 x 1.50		
8	ULS FC 8	1.00 x 1.00	1.00 x 1.35	0.00	0.00	0.00		
9	ULS FC 9	1.00 x 1.35	1.00 x 1.00	1.00 x 1.50	0.60 x 1.50	0.50 x 1.50		~
	Modify a co	mbination	Delet	e a combination	Delete	all combinations		
	<b></b>	Import combin	ations		Export com	binations		
E	<u>t</u> elp					<u>C</u> ancel		<u>O</u> K

Click 'OK' to close the window with the load combinations.

The names of the different load combinations are now listed in the pull down menu of the pallet 'Loads'. When you select one of these

combinations, then the whole of the loads that will act during this combination will be shown.



## 4.1.4 Generating the mesh

#### Step 14: Generating the mesh

Click on the button in the icon bar or select the menu instruction 'Analysis – Mesh'. Leave everything on default and hit 'OK'.



#### About the mesh generator

Here you enter the number of elements in which a bar should be divided: 8 division is a good value.

Meer information on our support website:

http://buildsoftsupport.com/knowledge-base/how-to-pick-the-mesh-size/

# 4.1.5 The global elastic analysis

The calculation of the structure is performed in three steps:

- First we calculate the internal forces with an elastic analysis
- Then we verify the strength and stability according a certain standard (see §4.1.6 and §4.1.7).
- Finally we will optimize the cross sections so we obtain the most economical cross sections (see §4.1.8).

#### Step 15: Elastic analysis

To start the analysis, select the command 'Analysis – Elastic Analysis'. You can also start the analysis directly using the function key **F9** or use the icon  $\blacksquare$  on the icon bar. Following dialog box appears:

9	Diamonds - StaalVB1.bsf - [Window 1 - SLS QP 1 - (kN, kNm, mm, kN/m, kNm/m, kN/m <sup>2</sup> , °C)]	+ _ 0 <mark>_ × </mark>
🥮 <u>F</u> ile <u>E</u> dit <u>V</u> iew <u>S</u> elect	t <u>D</u> isplay <u>A</u> nalysis <u>Options</u> <u>W</u> indows <u>Help</u>	_ 8 ×
0 🖌 🖬 🖪 🚳 🕅	りっ    田田田  🏭 🚺 📓 🕻 🇞 👌 警証の例 🖬    巨脚 🥼	
🐼 🗟 🖷 🔀 Window 1	✓ 1 Loads ✓ 第 日間 2 第 日 (7 Q Q 10 ) / ● 目目	
γ <sub>g</sub> <sup>γ</sup> u <u>dit</u> ⊕ <u>im</u> ⊷ SLS QP 1 ▼	Analysis settings ×	Active level
	Structural Soil Dynamic	2.70 m
+	Analysis     Ist order analysis	Level manager
キンジ	C 2nd order analysis Precision for normal forces in bars: 1 %	the Drawing plane           X = 0.00 m           Y = 2.70 m
Temperature     Snow	max. number of second order iterations: 10 文	Z = 0.00 m Representation
~ Ø ₽ ▼ Ø ₽	Take shear deformation into account for beams	
*         Seismic           *         Dynamic           *         Moving	Joints	Size Font 20 🚖
	✓ Concrete Cracking     ✓ Timber	Loads 10 -
	Help     Cancel      Next >       QK	Show groups

We choose a first order calculation and confirm with 'OK'. A dialog box displays the progress of the calculation.

Performing analysis	
Static analysis	
Solving combinations	
Combination 17	

The button ellows you to stop the calculation. If you stop the calculation, you'll have to completely restart it later.

#### Step 16: Go to the 'Results' configuration

To see the results of the calculations, you click on 🖻 in the icon bar or select in the adjacent pull down menu the 'Results' configuration.



#### About the 'Results' configuration

In the corresponding pallet on the left side of the model window, you'll see several buttons, each representing a specific group of results.

- Only those buttons for which a calculation is carried out are available.
- Once one of these buttons is pressed, the corresponding partial results can be retrieved through the buttons located below.
- Then you indicate for which load combination you wish to see the results. In a first pull-down menu, select the type of load combination (individual load group, ULS FC, ULS SC, SLS RC, SLS FC or SLS QP), then specify which load group or load combination must be shown. In the case of a load combination you can choose between either an individual load combination (indicated by a number) or the envelope. In those situations where the result suggests an envelope, it may be possible that for some results you can opt for the minimum (min) or maximum (max) results to be displayed.

Below, we list some results.

#### **Step 17: Deflection**

By default Diamonds will show you the vertical deformation in de Ydirection for the first combination (or the first load group when you also made the combinations for the load groups – which is the case in this

example). You'll notice that the button for viewing the displacements  $I_{\mathfrak{s}}I$  is active and that the button for vertical displacements according to the global Y-axis  $I_{\mathfrak{s}}I$  is active.



The figure above is shown in wireframe  $\textcircled{}^{\textcircled{}}$  and we opted for a front view.

Now select the combination group 'SLS RC' and choose the envelope of the results. We notice that:

- the maximum deflection is 61mm (downward).
- In all combinations SLS RC and on each position of the beam, Diamonds will look for the minimum value of the deflection. Those values are represented by the **thin line**.
- In all combinations SLS RC and on each position of the beam, Diamonds will look for the maximum value of the deflection. Those values are represented by the **thick line**.
- Hence this image is called an 'envelope'.



Note that you can arrange the size of the results representation using the pallet 'Size' located on the right-hand side of the model window.

#### About the scale

A symmetrical colour scale is used by default for all results in Diamonds. However, you may choose a different scale.

You should understand the default scale as follows: the limits of the colour pallet correspond to the largest positive OR negative value. The colour scale runs from -61,0 to +61,0mm. However, the largest or smallest value is displayed above and below the scale. Consequently, for this example, only the lower half of the colour pallet is used.

#### Step 18: Deformation in the detailed window

Now select the left rafter. Click on the icon  $\exists$  on the right of the icon bar to ask a detailed result. A new window will open. On the left you will find all the buttons of the 'Results' configuration applicable for line elements (beams and columns).



Note that in this window the deformations are defined according to the local axes of the bars. Also the angular rotation  $\varphi_{y'}$  (round the local y'-axis) is displayed.

#### Step 19: Stress in the detailed window

Now show the elastic stresses for the combination ULS FC envelope.



You can retrieve the results at any position using the slider. Moreover, you can also enter a distance. Consult for example the results on 2,45m. Enter '2,45' under the white arrow.

With a combination envelope the determining combination appears. You can disable this by clicking once on the button  $\underbrace{\text{H-}}$ , this will change in  $\underline{\text{H-}}$ .

Click 'OK' to close this window.

#### Step 20: Stress in the detailed window (section level)

Now show the elastic stresses in the cross section for the combination ULS FC1.

- Select the left beam and click on the icon <a>[</a> on the right of the toolbar.
- Or double click the left beam.

This window opens:



You can retrieve the results at any position using the slider. Moreover, you can also enter a distance. Consult for example the results on 2,50m. Enter '2,50' under the white arrow.

Move the mouse over the cross section to see the stresses at any position.

#### About detailed results on cross section level

Choose on the left top for which **load group** or load combination you would like to see the stresses.

With the **slider** on top of the window you can set the section for which you would like to see a detail of the stresses. By clicking on the distance below the slider, you can enter a position of your choice.

#### Results field with scale:

- In the results field the selected profile is graphically represented together with its principal axes of inertia. When a cross section is double symmetric, these axes will coincide with the local axes.
- On the principal axes you'll see red points. These are the points for which the stress results  $(N + M_y \text{ and } N + M_z)$  are presented in the global results window of Diamonds. The position of these points is determined as the intersection of the principal axes with the cross section's bounding box.
- When you come near these red points with the cursor, Diamonds will snap to them.
- Move the mouse over the section to see the stresses at the desired place. Enter 'x' and 'y' coordinates to show the stresses at a point of your choice. The stresses you find in this window are based on  $N + M_y + M_z$
- Compression is negative, tension is positive.

Click 'OK' to close this window.

#### Step 21: Bending moments $M_{\gamma}$

Now we visualize the bending moments  $M_y$ . Click on  $\mathbb{N}^{\mathfrak{g}}$  in the pallet and select the bending moment  $M_y$ . Choose the combination ULS FC envelope.



The moment diagram is always displayed on the tension side of the element. The sign of the bending moment corresponds to direction of the local axes. In this case the local z'-axis is directed upwards and causes a positive moment thus tension on the upper side.

#### Step 22: Reactions

Once back in the model window, we click on the button in the pallet to show the reactions. All reactions are displayed separately by Diamonds. In this example we are interested in the vertical node reactions in the combination 'SLS RC': we select the support reactions  $\mathbf{f} \in \mathbf{R}_{\mathbf{y}}$ .



So far the overview of the functionalities in the 'Results' configuration. Now we can calculate the reinforcement and the cracked deformation.

# 4.1.6 Parameters for steel verification

#### About steel design

The internal forces in the structure are known and we wonder if the structure can bear these forces? Checking this is done by a steel verification (see §4.1.7). This verification consists of two parts:

- Strength: is the structure strong enough to handle the internal forces?
- Stability: is the structure stable enough or will it buckle (laterally).

To perform the check for strength, you don't need to do anything extra. To perform the check for stability, you'll have to set some parameters for buckling and lateral torsional buckling.

## 4.1.6.1 Buckling

Buckling is the collapse / becoming unstable of a bar under a compressive force.

Because Diamonds works axis-to-axis, it may be that continuous elements (elements which are manufactured in one piece) are modelled as a series of consecutive individual elements.

In this example, the columns are executed as a whole even though we entered a support halfway in the Z-direction. The fact that the column

consists of 2 individual elements, may lead to an underestimation of the buckling lengths if we do not make the right definitions.

With the 'correct definitions' we mean setting groups for buckling. Hereby we impose which consecutive individual bar can buckle together in a certain direction.

#### Step 23: Setting the groups for buckling

Return to the 'Geometry' configuration  $^{\textcircled{M}}$ , where we choose a solid representation and a 3D view.

Because buckling always occurs around the (local) strong or weak axis, we make the local coordinate system of the bars visible with **1**.

Oiamonds		Window Configuration	×
Diamonds       File     Edit     Yiew     Select     Display     Analysis     Options     Windows     Help       Windows     Windows     Help     Image: Test of the select     Image: Test of the select     Image: Test of the select       Windows     Image: Test of the select       Image: Test of the select     Image: Test of the select     Image: Test of the select     Image: Test of the select       Image: Test of the select     Image: Test of the select     Image: Test of the select     Image: Test of the select       Image: Test of the select     Image: Test of the select     Image: Test of the select     Image: Test of the select       Image: Test of the select     Image: Test of the select     Image: Test of the select     Image: Test of the select       Image: Test of the select     Image: Test of the select     Image: Test of the select     Image: Test of the select       Image: Test of the select     Image: Test of the select     Image: Test of the select     Image: Test of the select       Image: Test of the select     Image: Test of the select     Image: Test of the select     Image: Test of the select       Image: Test of the select     Image: Test of the select     Image: Test of the select     Image: Test of the select       Image: Test of the select     Image: Te	Current configuration : User configuration Configuration name Geometry General Geometry Mesh   Points I✓ Show point I✓ Point number I✓ Connection name	Window Configuration         ▼         Lines         ✓ Show axis         □ Line number         Name         □ Length         ✓ Section orientation         ✓ Local coordinate system         ✓ End releases         Material         Border lines         Buckling length Y         Buckling length Z         Distance between LT supports z>0         □ LT supports z>0         □ LT supports z<0	Save current configuration         Delete selected configuration         Surfaces         Transparency:       50 € %         Surface number         Name         Thickness         Area         Local coordinate system         ✓ End releases         Material         Border lines         Practice reinforcement         ✓ Orientation
	Help		<u>C</u> ancel <u>Q</u> K

Right below you will find a pallet allowing you to define groups. The button allows you to group the selected bars, with you ungroup them.



First we look at the groups for buckling around the y'(u)-axis of the (selected) right-hand column.

- We select 'Buckling around y'(u)' from the pull down list (see image above).
- Buckling around the y'(u)-axis for this column means buckling to the left or the right in the plane of the frame. Since nothing prevents these columns parts from buckling in that directions, **they should be grouped.**
- Select the 2 columns and click on
- The same applies to the left-hand column (i.e., repeat the above operation for the left-hand column).

Now we look at the groups for buckling around the **z'(v)-axis** of the (selected) right-hand column.

- We select 'Buckling around z'(v)' from the pull down list (see image below)
- Buckling around the z'(v)-axis for this column means buckling to the left or the right in the plane of the frame. Since the support in the z-direction prevent these columns parts from buckling in that directions, they should be ungrouped.
- Select the 2 columns and click on .
- The same applies to the left-hand column (i.e., repeat the above operation for the left-hand column).

The rafter is not divided, so we don't need to define groups for it.



In the pallet on the right, select 'None' to stop showing the groups. Although the groups are not visible, they're still defined!



#### Step 24: Calculating the buckling lengths

Now the groups are defined, we can calculate the buckling lengths. Click on  $\frac{1}{2}$  in the icon bar.



In each direction (round y'(u)- or z'(v)-axis) Diamonds asks you for which type of structure and for which type of analysis (first or second order) you would like to calculate the buckling lengths.

It is important that you use the same type of analysis as what you indicate here. We choose 'semi displaceable nodes'. Click 'OK'.

To view the just calculated buckling lengths, click on  $\square$  and check the options with the buckling lengths.

Operation		Window Configuration	×
Diamonds         Pie Edit View Select Display Analysis Options Windows Help         Pie Edit View Select Display Analysis Options Windows Help         Pie Field Window 1         Pie Field Window 1         Pie Field View Select Display Analysis Options Windows Help         Pie Field View Select Display Analysis Options Windows Help         Pie Field View Select Display Analysis Options Windows Help         Pie Field View Select Display Analysis Options Windows I         Pie Field View Select Display Analysis Options Windows I         Pie Field View Select Display Analysis Options Windows I         Pie Field View Select Display Analysis Options Windows I         Pie Field View Select Display Analysis Options Windows I         Pie Field View Select Display Analysis Options Windows I         Pie Field View Select Display Analysis Options Windows I         Pie Field View Select Display Analysis Options View Select	Current configuration : User configuration Configuration name General Geometry Mesh Points Show point Point number Connection name	Window Configuration         Image: Strain	Surfaces Transparency : 50 💽 % Surface number Name Thickness Area Local coordinate system Grid releases Material Border lines Practice reinforcement Grientation
	Help		<u>C</u> ancel <u>O</u> K

## 4.1.6.2 Lateral torsional buckling

Lateral torsional buckling is the collapse/ becoming unstable of a bar under a bending moment.

#### Step 25: Setting the lateral torsional buckling length

We didn't model the purlins in the roof. The presence of these purlins does affect the lateral torsional buckling length. We will therefore represent the purling by lateral restraints:

- Go to the 'Geometry' configuration 👼.
- Select the rafters and click on  $I^{\ell}$ .

	- StaalVB1.bsf - [Window 1 (m)] -	
🥮 Eile Edit View Select Display Analysis Options Windows Help	_ <i>B</i> ×	
□☞ ■ 0.●♥  ゃ~  冊冊冊 Ⅲ  ℡  図圖	¥ 22 0 0 m = 1 = 1 = 1 = 1 = 1 = 1 = 1 = 1 = 1 =	
🐼 🔄 🕾 🛛 Window 1 💽 🚺 Geometry	· 酚 P 🛛 🔁 🕺 中 🖑 Q Q 🖬 🖉 🖬 🐨 🖉 💷 🗔 🔳	
× ×	Active level	
OH /	verdeping 1	
	HI2.7m	
	Buckling parameters	- 🗆 🗙
IPE 270 ▼ Δ→ ×		
夕風空		
-27 . ₩ P+I	About y(u)-axis 5,5 m	
	About z (v)-axis /, u m	
HU 🖉 🦚	ci ateral burding length	1
FFF FE BAR	Lateral bucking length	
E State	z' > 0 z' < 0	
1009 85; 47-1 7	-z' > 0  ✓ Lateral torsion budding check  ✓ Lateral torsion budding check	
	-z' > 0     -z' < 0	
	-z' > 0     -z' < 0	ports
	-z' > 0     -z' < 0	ports
	-z' > 0     -z' < 0	ports
	z' > 0     iv     Lateral torsion buckling check       iv     Equal segments     iv       b     intermediate LTB supports     intermediate LTB supports	ροrts ,β
	z' > 0     Z' < 0	ports ,β Φ
	z² > 0       Image: Constraint of the second s	ports ,β Φ • [m] Φ
	z² > 0       Image: Constraint of the second s	οοrts .8 Φ [m] Φ .8
	z² > 0	-,8 Φ (m) Φ ,8
	z² > 0	-,8 Φ ,8 (m) Φ ,8
	z² > 0	-,8 Φ ,8 (m) Φ ,8
	z² > 0	

In this window you can

- consult/ adjust the (calculated) buckling lengths
- add lateral restraints
- adjust the lateral torsional buckling parameters

We just calculated the **buckling lengths** so we don't need the make any changes here.

For the best, leave **the lateral torsional buckling parameters** on default at all times.
We assume that the purlins prevent lateral torsional buckling in the upper and lower flange of the beams. Since each rafter contains 3 purlins, increase the number of lateral restraints to '3'.

The small red dots on the rafters, indicate the presence of these lateral restraints.



# 4.1.7 Steel verification

#### About the steel verification

With the steel verification we check the strength and the resistance against buckling and lateral torsional buckling of the bars according to a certain standard.

Before starting the calculations we check the properties of the used steel quality.

Choose the menu command 'Edit – Material library' and select the material 'Steel S235' from the left column. 'Steel S235' is a default material. Standard materials are marked by #.

Er er +z Iz I E I I		
4 Steel E 450 /Ee 570) D		Name S235
A Steel E 450 (Fe 570) D	^	Material type steel
A Steel Fe 235W		
A Steel Fe 355W		Mechanical properties Thermal properties Advanced
A Steel Fe 360		
# Steel Fe 430		Eurocode 3 : EN 1993-1-1/3
A Steel Fe 510		
A Steel S235		Yound's modulus 210000 N/mm2
A Steel S250GD+Z		
A Steel S275		
A Steel S275 M		Transverse Young's modulus 80/69 N/mm2
# Steel S280GD+Z		Therm. dilatation coeff. 0.000012 /°C
A Steel S320GD+Z	~	Density 7850.0 kg/m3
		,

The properties of default materials are determined using the standards and can't be edited. Should you wish to make adjustments, you should make a new material. User defined materials are marked by the icon **1**.

On the right of this window you'll find:

- The **mechanical properties**: the Young's modulus, the (transverse) Poisson ratio, the thermal dilatation coefficient and the density.
- The thermal properties used in a fire analysis
- The strength properties. In particular we review the properties that apply for Eurocode 3: EN 1993-1-1 [--]. The mechanical properties like the yield  $f_y$  and ultimate  $f_u$  strength in relation to the thickness of the components (web and flange). The partial safety factor for steel depends on the application.

5earch	Name \$235
Steel ASTM-Grade60/75     Steel ASTM-Grade65/80     Steel E 165 (Fe 290)     Steel E 250 (Fe 410) W)     Steel E 350 (Fe 440)     Steel E 350 (Fe 570) D     Steel E 410 (Fe 570) D     Steel E 450 (Fe 570) D     Steel E 450 (Fe 590)     Steel Fe 350     Steel Fe 325W     Steel Fe 330     Steel Fe 330     Steel Fe 310     Steel S250GD+Z     Steel S250GD+Z     Steel S355GD+Z     Steel S355 M	Material type       steel         Mechanical properties       Thermal properties       Advanced         Mechanical properties       Thermal properties       Advanced         Strength

Now click on 'OK' to close the material library.

# Step 26: Choosing the steel standard

Now select the menu instruction 'Analysis – Steel standard' and indicate you wish to perform the steel verification using the European standard EN 1993-1-1 - [--] - Method 1.



#### Step 27: Steel verification

To start the verification, select the menu command 'Analysis – Steel and timber design' or click on F or press **F3**.

Otamonds - StaalVB1.bsf - [Window 1 - δy - Selfweight (mm)]	- 0 ×
💝 File Edit View Select Display Analysis Options Windows Help	- 8 ×
😨 図 🖷 🛛 Window 1 🔹 🚺 Results 🔹 📝 🔞 🖬 🔝 🔁 🎢 中 🥙 Q Q 🛛	
Code check for selected elements only Code check for selected elements only Selfweight Selfweight Code check for all elements Selection criteria Switch combination filter on Co Global value 95 % of maximum C Particular values Interaction method (EN 1993-1-1) remethod 1 remethod 2 Help Cancel QK	x = 0.0 0.0 0.0 0.0 0.0 0.0 0.0 0.0

Fill in the settings as above and click 'OK' to perform the verification.

#### About the window 'Steel / timber design'

• The box 'Action' you indicate whether you want to perform the verification on all bars or just the selected ones.

 In the box 'Selection criteria' you can specify whether Diamonds should run the verification for all combinations or only for the most determining combinations.

Once the calculation has ended, the button 2 will become active in the Results configuration 2. You'll see the following two icons:

- X for viewing the results of the check on strength
- f for viewing the results of the check on stability

Both results are expressed as a percentage of the maximum capacity. The maximum capacity equals 100%.



Results for strength (%)



Results for stability (%)

We see that the structure is insufficient for both strength and stability (= percentages larger than 100%).

To get more information about the problem, we double click for example the rafter on the left in the 'Results' configuration. Make sure you're looking at the results of the steel verification (22 or 2).



From the window about strength we learn that the check for bending + shear + axial forces gives us the highest percentage (146%). When we look at each 'check' separately, we see that bending is the determining 'check'. It gives us percentages higher than 100 AND higher than the percentages we find for axial and shear forces. The only solution is choosing a larger cross section (= larger moment of resistance) or a better material (= larger yielding stress  $f_y$ ).

From the window about stability we learn that the check for bending + compression gives us the highest percentage (199%). When we look at each 'check' separately, we see that lateral torsion (so bending) is the determining 'check'. It gives percentages higher than 100 AND higher than the percentages we find with buckling (so compression).

To solve the instability problem we could add more purlins. This will reduce the lateral torsional buckling length and therefore also the results of the stability verification. But since in this case the percentage is rather high, adding extra purlins, won't solve this problem. Opting for a larger crosssection is the only solution.

#### Step 28: Adding a haunch

We notice that the bending moment near the bar ends, (and thus the verifications), increases in relation to the centre of the beams. We could stiffen these sections locally by adding haunches. This allows us to usually get lighter sections for the beams and thus save weight.

Click on the icon Minimi in the icon bar, or select the 'Geometry' configuration

from the nearby pull down menu. Select the rafters and click on Complete the dialog like this:



Click 'OK' to add the haunches. The haunches will appear under the rafters.



Regenerate the mesh  $^{
mathbb{M}}$ . Perform the elastic analysis again  $^{
mathbb{M}}$ .

Perform the steel verification again <sup>Fg</sup>. You'll see the percentages have decreased.



# 4.1.8 Cross-section optimization

We now will use the optimization algorithm of Diamonds to find the most optimal cross-sections.

The optimization is based on the obtained percentages with the steel verification (see §4.1.7).

There are two optimization principles in Diamonds:

- The basic cross-section is provided in the **section library**. Diamonds will therefore search for the best suited cross-section in the cross-section library.
- The cross-section is built based on a **characteristic shape**. Diamonds will optimize by modifying either the height or width step by step defined by the user.

In this example the first method will be used, because the section we used come from the section library. The second method will be discussed in the example when we calculate a 3D structure in timber (§5.2.8).

#### Step 29: Cross-section optimization

To start the optimization, select the menu command 'Analysis – Optimization' or click on the button  $\mathbf{I}$  in the icon bar.

If you had elements selected, Diamonds will ask if you want to perform the optimization for all the elements or for the selected elements only.

Diamonds - StaalVB1.bsf - [Window 1 - Stability verification of bar (%) (%)]	- 🗆 🗙
🧇 Eile Edit View Select Display Analysis Options Windows Help	- 5 ×
🗑 🗟 🗟 Window 1 🔽 🚺 Results 🔹 🗑 🖻 🗑 🖬 🗐 🛃 🕫 🖗 🖉 🖗	
7.7 01 01       0         123.0       123.0         123.0       123.0         123.0       123.0         123.0       123.0         123.0       123.0         123.0       123.0         123.0       123.0         123.0       123.0         123.0       123.0         123.0       123.0         123.0       123.0         123.0       123.0         123.0       123.0         123.0       123.0         123.0       123.0         123.0       123.0         123.0       123.0         123.0       123.0         123.0       123.0         123.0       123.0         123.0       123.0         123.0       123.0         123.0       123.0         123.0       123.0         123.0       123.0         123.0       123.0         123.0       123.0         123.0       123.0         123.0       123.0         123.0       123.0         123.0       1.9         123.0       1.9         123.0<	Active level Verdeping 1 • 2.7 m 2.7 m Evel manager Cond level Drawing pane X = 0.0 m Representation Size Font 20 • Size Font 20 • Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size Size

# If this window pop up, select the option 'optimization for all elements'!

Then Diamonds will show you a window for entering the parameters for the optimization:

Oj	otimization
Optimization parameters Adjust cross-se	ections
Optimisation targets:	95 🚖 %
Optimize for:	I Resistance I Stability
Order sections by:	Resistance
Help < Previous	Next > Cancel

Optimization			
Optimization parameters Adjust cross-sections			
Adjust cross-sections			
C Automatic adaptation			
How to replace cross-sections:			
I Equal cross-sections remain equal after optimisation			
All bars of same type have the same section after optimisation			
Help      Previous     Next     Cancel			

Fill in the settings like in the images here above.

#### About the window 'Timber optimization'

- The **first tab** '**Optimization parameters**' contains a field with a percentage value. This is the value we should try to approach during the optimization.
- The second tab 'Adjust cross-sections' is present when the project contains cross-section coming from the section library. In this tab you can ask Diamonds to show you a summary of the optimization. If you do not ask the summary, Diamonds will adjust the cross-sections automatically.

Fill in the settings like in the images here above.

When the optimization is completed, a dialog box appears with the summary of the optimization.



Diamonds proposes you to change some cross-sections. You can accept or ignore the changes by (un)checking the corresponding line. Accept the changes.

Note: Save this model, you will need it in the next paragraph.

# 4.2 Example 2: 3D hall

Required licenses: ✓ 2D Bars

- ✓ 3D Bars
- ✓ Steel Design
- ✓ Steel Connection Design (for §4.2.9)

# 4.2.1 Purpose of the exercise

We now calculate the 3D hall. This is a sketch of the structure we will calculate:



We will make this model starting from the 2D frame we calculated in the previous exercise (§4.1).

# 4.2.2 Defining the structure

# Step 1: Opening 2D the model from §4.1

If the frame from §4.1 is not open, use the button 🖻 to open the project.



Defining the structure is always done in the 'Geometry' configuration. Click on in the icon bar, or select the 'Geometry' configuration in the adjacent pull down menu.

# Step 2: Erasing the supports in the Z-direction

Select all points except the simple supports and remove the restraints in the z-direction  $\frac{\overline{x}}{\overline{x}}$ .

# Step 3: Remove the lateral torsional buckling supports

Select the rafter and remove the lateral torsional buckling supports I. We will actually model the purlins.

🥹 Diamonds - StaalVB1 (met	kniestuk na optimalisatie).bsf - [Window 1 (m)]	- • ×
🥮 Eile Edit View Select Display Analysis Options Windows Help		_ 8 ×
□ ☞ ■ □. ● □ □ □ □ □ □ □ □ ■	¥₩ 6 \$3 ± 0 19 ≠     ⊨ # 4	12. 10 10.00
🐨 🗟 🗮 🕅 Window 1 👻 🚺 Geometry		
		ctive level
	5	/erdieping 1 💌
	É	2.7 m
	Puckling parameters	X
IPE 300 ▼ ×	Buckling length	
夕服費 ≜	About v/(u)-avis	
√ _ ≒ D.1	About z'(u)-axis	
NA M DA		
	Lateral torsional supports Advanced LT parameters	1
ffff ~E° R	Lateral buckling length	
W 8: 471	✓ Lateral torsion buckling check	n buckling check
<sup>#</sup> ↔ 8° <u>√</u>	✓ Foual segments     ✓ Foual segments	ats
	0 🔿 intermediate LTB supports 👂 🔷 inter	mediate LTB supports
I.	Representation	
	7,1	
	0	\$
	• 7,1 m	• [m]
	\$	<b>⊕</b>
	7,1	
L		
	Help	Cancel

# Step 4: Dividing the rafters

Select the rafter and divide them in 4 equal parts 🐓.

Diamonds - StaalVB1 (met kniestuk na optimalisatie).bsf - [Window 1 (m)]	- 🗆 🗙
💝 Eile Edit View Select Display Analysis Options Windows Help	_ 8 ×
□☞ ■ 国● ◎ ◎ □ 田田田 ※ ■ 2 2 2 2 3 2 5 2 2 1 1 1 1 1 1 2 2 2 2 2 3 2 3 2 2 2 2	10 M W 10 M
図 日 尼 🛛 Window 1 🔽 🚺 Geometry 🔽 🍘 印 💹 🗗 (外 Q Q 🖾 🖉 🗊 🗷	
X X	Active level
	Verdieping 1 •
	12.7 m
	2.7 m
	🛱 Level manager
	Ground level
夕  翔 (2)	🔁 Drawing plane
	X = 0.0 m
予 連 Divide line X	Z = 0.0 m
	Representation
	888
Final segments	Se the
	~ ~
	Size
<u>Tae II II Cancel</u>	
<b>I</b>	Symbols 20
	Loads 10 🚖
	Results 15 🚖
	Show groups
	None
Ĺ,	
	1.

# Step 5: Copy the structure to a 3D model

Now select the entire structure (use a selection window or press CTRL+A) and click on  $\checkmark$ .



Complete the window like here above and click 'OK'.

#### About the 'Translation' function

- In the field 'N' you enter the amount of copies you want. When you just want a translation (= move something), 'N' should remain equal to 0.
- In the three fields below you enter the translation (or copy) vector.
- When you finally check the boxes 'create connecting lines' or 'create connecting plates', Diamonds will automatically draw lines or plates between the copied items.

In order to see what Diamonds has made of it, you have to view the project in perspective. Click  $\square$  on the right bottom and choose a 3D view.

At the bottom and on the right scroll bars allow you to change the viewpoint. You can also use the button 🖾 or **F12** to see the entire structure on the window.



#### Step 6: Cross-section of the purlins

The cross-section of the purlins has not yet been defined:

- Select the purlins with:

Sele	ct	Display	Analysis	Opti	ons	Windows
	AI	I	Ctrl+A	Ē	1   #	: T.
	Point Number			User	configurati	
	Line Number		F			
	Ve	ertical Line	:	ι.		
	Horizontal Line					
	Inclined Line					
	Cross-Section			HEA	260	
	Surface Number			IPE 3	300	
	Vertical Surface			?		
	Horizontal Surface					
	In	clined Sur	face			
	PI	ate Sectio	n 🕨	ι.		
	Material •					
	Most loaded bars		L			

This way you select all bars without a cross-section.

- Click on III to open the section library and select an IPE 120 from the list.

If you always define the cross-sections at each step in the project, then the last drawn bars will always be without a cross section. Now it becomes very easy to select them.

We also could have chosen 'Select - Horizontal lines'.

# Step 7: Add hinges to the purlins

We assume that the purlins are imposed between the frames through L-sections, so transferring bending moments is not possible.

- Select all purlins (IPE 120)
- Click on the button .
- Complete the dialog box like here below:



When you confirm the settings, small round symbols will appear on the ends of the girders. These indicate the presence of a hinge. In this case, we keep the torsional moment  $M_x$  fixed. If you set it free, this means the bar (girder) can rotate freely around its own axis, which is not the case.



If don't see the red round symbols, do this:

- Click on 1 and activate the tab 'Geometry'.

- Select the option 'End releases' under the title 'Lines'.
- Make sure the size of symbols is large enough (see §2.2).

#### **Step 8: Drawing the facades**

We draw the façades using the extrusion function.

- For both facades (front and back) select all points on the rafters, except for those who connect to the columns (see image below).
- Next click on <sup>[+]+]</sup> in the pallet.
- Choose the option 'to plane' and indicate that you wish to extrude to the plane Y=0m.



With this function we create bars with start point the selected point and end point determined by the components of the input vector.

#### Step 9: Cross section and supports to the facades.

Assign the section HEA100 1 to the facades and place simple supports at the lowest endpoints.

#### **Step 10: Change orientation of the cross sections in the facade**

Diamonds assumes a default orientation. We wish to turn the HEA100 sections over 90° to use them more efficiently.

- Select the bars with cross section HEA100.
- Next click on the icon in the pallet.
- Enter a rotation angle of 90°.

Section	orientation ×
Orientation angle	90 •
Mirroring	
Help	<u>C</u> ancel <u>O</u> K

# Step 11: Adding a gate to the front facade

Now we make an opening in the façade for a gate:

- From the front facade: select the column on the left of the ridge.
- Divide it 2 equal parts 🔨.



- Double click the lower half of the column **near the support**. Doing so, the support will remain in place, while the other point moves when changing the length/slope of the bar. A dialog will appear allowing you to change the geometry of the bar.
- Set the length of the bar to 2,75m.

Note: we also could have divided the line in 2 unequal parts, wherein the first part is 2,75m and the second part 2,18m.



To draw the lintel click on the button  $\checkmark$  and draw in a 3D view a line from the left column to the right column. The intelligent cursor will help you finding the perpendicular position. Click on  $\checkmark$  to end the drawing function.



Now select the lintel and the centre column and click on  $\mathscr{H}$ . Select the option 'Intersection lines' to find the intersection of these two lines.

🤪 Diamonds - Sta	alVB1 (met kniestuk na optimalisatie).bsf - [Window 1 (m)] 🛛 🚽 🗖 💌
	/s Help
	図 圖 ¥ 社 & ◎ 亚 ⊕ F% ゴ   ⊨ ■ +9
😨 🗟 📉 Window 1 💽 Geometry	- R - C - C - C - C - C - C - C - C - C
▶,         ×           ∩用         ·	Active level
	Intersections
	r Intersection lines
Z ⊪5 Pri	C Intersection lines and plates with reference plane
	O Y=m
	C Keep only intersection points
	Help Cancel OK
	Show groups None
<u> </u>	► Z <sup>⊥</sup> X

Assign a section HEA 100 to the lintel. Remove the lower central column by selecting it and then clicking on  $\times$  or pressing the DEL button from the key board.

In order to get the gate on the other back façade, select the lintel and perform a translation.

- With 'N' enter '1'
- And enter the distance on the screen: just click once on a point in the front facade and once on a point in the back facade.
   Based on the selected point, Diamonds will calculate the translation distance.



Remove the central column in the back façade. Result:



# Step 12: Adding a bracing system

To complete the model, we add a bracing system to ensure the stability of the structure. Here for we use metal bars with a L30x30x5 section in steel S235.

- Click on the button to activate the drawing function.

- Then draw the bracing system using the intelligent cursor. Use the image at the start of this example as a reference for the position of the braces.



- Select the just drawn lines using the menu command 'Select – Section - ?' and click on <sup>IIII</sup> and search the section L30x30x5 in the list.

Finally, the rods must be defined as tie rods. This means that the rods can't take compression. Select the bars and click on  $\checkmark$ , choose the option 'tie rod'.

The geometry of the structure is now complete.



# 4.2.3 Defining the loads

Step 13: Go to the 'Loads' configuration

We now leave the 'Geometry' configuration and activate the 'Loads' configuration to enter the loads. Click on the button  $\square$  in the icon bar or select in the adjacent pull down menu the 'Loads' configuration.

# 4.2.3.1 Creating the load groups

# **Step 14: Deleting the loads on the frames**

The loads from §4.1 are still on the frames.

- Select the first load group 'dead loads' from the list.
- Then select the entire structure and click on  $\frac{1}{2}$ .
- Repeat these steps for all load groups. The loads in the load group 'self-weight' cannot be deleted.



# Step 15: Creating load groups

Click on the button  $\gamma_{g}^{\gamma_{u}}$ . You'll see the following screen:

🤌	Edit View Sele	ret Di	Diar	nonds - Staal <sup>1</sup> Analysis Ontions	VB2.bs Wind	f-[Ve	nster 1	1 - dea	d load	ds - (	kN, kN	m, mn	n, kN/r	n, kNr	n/m, kN/	m², °C)]			×
		n	ispidy 1			T. D	X 📾	¥ 7.0	4	126	(T4) Fe;	TY	IE1	<b>1</b> : 10				1	
		••			da				× 1	Je			30	h so			1773 1775 1775	ाजा हा	1 =
- <u>M</u>		-0			us					1 1999	*** @		1.514	=   Y	विष			Active leve	1 -
٧ <sup>٧</sup> ٣																		Verdieping	1 -
dead lo	oads 💌																		.7 m
~								-											
<b>9</b>								Lo	bad gr	oups									
Load f	actore for		EN 10	- -	<u> </u>		Conser	uence da	acc [	2			Sen	ice class	1				
Lodd I			Ind I:		1		Design	lifetime	T	50	vears		ber		1- 💌				
		142.0					buoign				,								
	Add load group		Inser	t load group				Delete l	oad grou	up	]					Ser	veral load cas	es per grou	N q
2	Name load group	#	Туре	Name load case	γ <sub>uls-</sub>	$\gamma_{uls+}$	$\gamma_{\rm sls}$ -	$\gamma_{\rm sls+}$	$\psi_0$	$\psi_1$	ψ2	φ	ξ	to	Combinati cracking	<sup>id</sup> k <sub>mod</sub>	Load	Action	^
~	Selfweight	1			1.35	1.00	1.00	1.00	1.00	1.00	1.00	1.00	0.85	0		permanent	-	414	ŕ
1	dead loads	1			1.35	1.00	1.00	1.00	1.00	1.00	1.00	1.00	0.85	0		permanent	-	+1+	
×	live loads H : roofs	1			1.50	0.00	1.00	0.00	0.00	0.00	0.00	1.00	1.00	0		medium term		***	
1	Wind	16	ł£	Case 1	1.50	0.00	1.00	0.00	0,60	0.20	0.00	1.00	1.00	0		short term	p.	414	
				Case 2															
				Case 3	-														
				Case 4															
				Case 5															
				Case 6															
				Case 7															~
1	ncompatible load group	os		Linked load group	IS			Comb	ine load	cases		â	Jngroup	loadcase	s				
н	Help Cancel OK																		
-		_	_		_	_	_	_	_	_	_			_	_			-	

We still find the load groups from the previous example §4.1 in this list. We will use the same load groups in this example but for a 3D structure with a

duo pitched roof we should consider 20 cases of wind. So set the number of cases wind to 20 and change the name of the added cases.

- Wind front -> back upward ( $c_{pi} = -0.3$ )
- Wind front -> back upward ( $c_{pi} = 0.2$ )
- Wind back -> front upward ( $c_{pi} = -0.3$ )
- Wind back -> front upward ( $c_{pi} = 0,2$ )

.oad fa	ictors for		EN 199	90 💌		-] Co	nsequen	ce class	2	¢		1	Service o	lass [	1 🚖				
						De	sign <mark>life</mark> ti	me	50	🔹 ye	ars								
	Add load group		Insert	load group			De	lete load	group							Ser	veral load case	es per grou	ম
	Name load group	#	Туре	Name load case	γ <sub>uls-</sub>	γ <sub>uls+</sub>	$\gamma_{\rm sls}$	$\gamma_{\rm sls+}$	ψ <sub>0</sub>	ψ1	ψ <sub>2</sub>	φ	ξ	t <sub>o</sub>	Comb cracki	k <sub>mod</sub>	Load	Action	ſ
				Case 13															
				Case 14															
				Case 15															
				Case 16															
				wind f -> b u (-0.3)															
				wind f -> b u (0.2)															
				wind b -> fu (-0.3)															
				wind b -> f u (0.2)															
✓	snow (H <= 100	3	ž£	Case 1	1,50	0,00	1,00	0,00	0,50	0,20	0,00	1,00	1,00	0		short term	举	$\downarrow \downarrow \downarrow \downarrow$	
				Case 2															~
In	compatible load groups	5		Linked load groups			(	Combine I	oadcase	5		Ungr	oup load	ases					
	1																3 1		

# Step 16: Defining linked load groups

The load groups 'self-weight' and 'dead loads' both contain vertical forces with a downward direction. Thus both load groups have the same effect on the structure. Consequently the most extreme (min and max) values for the internal forces are obtained when both load groups are multiplied by the same minimum/ maximum safety coefficients.

In Diamonds it's possible to define this kind of behaviour using 'Linked load groups':

- Click on the button Linked load groups at the bottom of this window.
- Indicate that the load group 'Self-weight' is linked to the load group 'Dead loads'.



Using 'Linked load groups' will reduce the amount of load combinations and is applicable on small and large structures.

Next click twice on 'OK' to close these windows.

# 4.2.3.2 Filling up the load groups

Step 17: Filling in the load groups 'Self-weight', 'Dead loads' and 'Life load'

- The self-weight is calculated automatically by Diamonds.
- A dead load of 0,6kN/m<sup>2</sup> is applied to the rafters.
  - Select the load case 'dead loads' from the pull down menu.
  - Select all rafters on one side of the roof like on the image below. The function <sup>3</sup>/<sub>2</sub> we will be using only works with bars that lay in the same 2D plane.
  - Click on the button <sup>♣</sup>.
  - Complete the dialog box like here below:



When you click 'OK', Diamonds will calculate the loads on each purlin.

 Repeat the same steps for the selection on the other side. Both sides can't be generated together since the selected bars have to be coplanar.

Result:



Using the image above, verify if you have entered the loads correctly.

- Repeat the steps for 'Life load H: roofs' of 0,4kN/m<sup>2</sup> on the same rafters.



# Step 18: Filling in the load group 'Wind'

# To generate **wind**:

- Click on the button *cond level* and set the ground level to 0m.



- Choose the load group 'wind' and the first sub load case 'wind I up r down (-0.3)' from the pull down menu.
- Click on 🎦 to define the wind standard and the terrain parameters.

🤗 Di	amonds - StaalVB2.bsf - [Venster 1 - Wind - Geval 1 (kN, kNm, mm, kN/m, kNm/m, kN/n	n², °C)] – 🗆 🗙
	Analysis Options Windows Help	- 8 ×
0 🛩 🖬 🖪 🚳 🕅 🗃	Terrain parameters	11 W. 1917
🐨 🗟 🛸 🛛 Venster 1		
		Active level
y <sup>y</sup> u <u>im</u> i⊗	Norm: EN 1991-1-4 💌 💌	Verdieping 1 V
Wind 💌		
Geval 1 💌	Reduction with factor 0.85 for lack of correlation	
10.7		2.7 m
Mr Nasan	Structural factor CsCd: 1	
		Hevel manager
	Direction factor Cdir: 1,0	Ground level
(f <sup>4</sup> ♪ 9 <sup>3</sup>	Season factor Cseason	t⊐- Drawing plane
AL SO		X = 0.0 m
		Y = 2.7 m
* Temperature	Air density: 1,3 kg/m³	Z = 0.0 m
Wind	Probability factor Corob: 1.0	Representation
1000	V <sub>k o</sub> = wind velocity: 26,0 m/s Belgium     ▼	
		👰 🦚
n 🖉 🕅 🔤 e e e e e	to q = wind pressure: 0,4 kW/m <sup>2</sup> Map	а вка вка
* Seismic		Size
Y Moving	Terrain Type:	Font 30 文
		Symbols 15 🚖
		Loads 10 🚖
	Orography coefficient	Results 15 🚖
		Show croups
	Help Cancel OK	None
		r · · · · · · · · · · · · · · · · · · ·
	<u>, a na n</u>	

- Select the standard EN 1991-1-4 [--].
- Opt for a basic wind velocity of 26m/s and a terrain type I.
- Click 'OK' to close this window.
- Take a top view and select the first frame of the 3D hall.

49						Dia	am	onc	is -	- S	taa	IVB	2.b	sf -	[V	ens	ter	1 -	Win	d -	Gev	al	1 ()	đN,	kNr	n, m	ım,	kN/	/m,	kNn	n/n	n, kľ	N/m	2,0	C)]					- 1		x
🤗 <u>F</u> ile	<u>E</u> dit <u>V</u> ie	w	Sele	ct	Dis	play	A	naly	/sis	0	ptio	ns	Wi	ndov	NS	He	р																								- 5	×
0 📾		1 é	6 (	D	1.	0	SI.	E	ĦI	Ħ	Ħ		:	Т		XX		1	Te.	è	1 9	ġ,	221	Ð	e//	r/	1		۵.	Ģ4	1											100
		21.53			1		-		11a		L F	a da			201	-277	0.800			- 50	111	1 0		-				-	den .	sm	0	æ	<b>K A</b>	sf	ß		100	1 122	8 MZ			-
		ens	er 1	8				-		1	lr	.oads									-		889	H .			- 63	FI   .	ш	57	Q	्य		1	¢/	D	IR	1 12	1 12	2   1001		÷.
$\gamma_g^{\gamma_u}$	tin 😞	8	15	55	82	12	8	3		82.3	8	22	8	1	33	82	35	8	$\approx$	33	55	33	2	$\otimes$	28	35	8	12	8	65	55	32	8	сs	57	-	8	82	-	Active le	vel	Ţ
Wind	•	2	83	-83	8	8	53	3		89 8		25	53	83	32	8	1	8		39	83	-	2		32	-83	93	88		8	83	23	88	8	8	-8	8	88	32		-9 -	_
Geval 1	-	35	83	-83	28	8	28	- 3		83 - S	3	a.	-	+ >	•	_	-	-	_		-	7	-	-	-	_	<b>.</b>	8	ж	8	10	88	88	35	85	13	8	8	35	AT AL	2.7 m	
		35	23	83	83	12	8	1		85.3	8	22	*	43	85	87	8	3	8		12	8	$^{\odot}$	8	-	12		13	8	52	83	12	13	8		53	83	13	35	ATT A	2.7 m	
Pring .	47 -	28	8	53	8	10	22	2		58 - 8	8	12	t	-	88	82	8		2		53	3	33	25	8	53		8	$\otimes$	8	53	82	8	82	$^{\odot}$	53	8	8		rt-p'		
1	1 1	8	53	55	22	10	8	5		31 3	ŝ	38		1	88	8	1		8		18	14	2	8	3	43		13	93	42	55	22	13	93	57	<b>3</b> 5	8	12	2	E Lev	el mana	iger
<b>+</b>		36	20	-3	8	12	9	3		81.3	8	12		20	87	8	8	9	8		38	1	96	33	-3	-25		36	2	20	25	12	88	2	20	-35	3 <b>2</b>	14	307	787 010	und los	
₽.	2 200	8	89	-0	8	8	23	3		38 S		2	•	_	•		-	-	-	•	_	•	-	•	•	-	•	8	8	88	-2	őł.	88	8	8	-8	8	8	35	the Deer		ei
		3	0	22	35	10	8	1		85 3	8	22		1	35	87	8	3	8		-	1	$\otimes$	8	- 3	2		13	8	$\leq 2$	22	38	13	8	2	-	8	3	35		wing pla	ine
	T Z	33	8	33	8	8	10	8		\$8 J	8	82		20	\$8	85	8		氮		32	ŀ	33	激	0	23		8	22	8	33	82	8	22	8	53	8	8	33	X = Y =	2.7 m	
- Ten	nperature	82	10	15	33	10	Si.	1		81 I	2	38		121	28	82	1		8		15	14	22	8	3	43		12	93	42	63	22	13	93	47	15	21	101	- 22	Z =	0.0 m	
¥ 1	Snow	36	20	-33	15	15	9	- 7		80 - 3	8	12		20	82	35	8	8	28		88	8	96	28	-3	-85		14	10	20	63	12	14	33	21	-88	85	15	36	Represer	ntation	
-	Wind	35	83	10	31	8	23	3		<u>8</u> 8 - 1	8	ŝŻ.	•		•		-	-		•	-	•		•	•	-	1	88	33	8	10	őł.	88	8	8	13	51	88	20	R	8.1	R
		8	22	-22	12	12	8	3		8.9	2	155		30	8	8	5	8	28		-88	8	8	18	3	-17		36	15	32	1	35	38	12	22	-	12	1	8	0		s.
849	a a	33	23	33	8	8	10	2		58 J	8	85		-	23	8	÷		2		33	ġ.	33	22	0	23		8	82	-83	33	82	8	82	23	33	8	8	33	499 A		
1 I + S	eismic .	8	23	10	84	14	- 9			<u>88 - 9</u>	2	84		20	18	84	î.		12		88	4	22	12	-	18		14	122	20	23	84	14	12	22	28	85	14	- 22 []	Cine		_
* D	ynamic	86	20	10	15	14	8	2		87 - J	8	12		20	87	38	8	8	18		88	12	96	18	8	85		26	18	20	10	12	25	98	20	23	88	15	- 56	Font	30	÷
Y N	Noving	26	83	90	- 28	8	38	- 8		<u>80 (</u>	1	88			•	1				1	_	1	-		1		7	88	35	8	10	88	88	35	8	10	59	8	35	Combala	10	-
-		8	30	13	12	10	8	- 3		81.1	2	15		32	8	10	8	.8	2		1.64	8	1	۲	्ह	1		52	98	31	63	35	32	98	32	52	12	32	28	Symbols	15	-
		33	33	53	8	8	12	2		<u>88</u> 8	8	12		-	2	8	8		12		22	2	33	2		(33		8	32	3	3	82	8	22	3	53	8	8	25	Loads	10	\$
		9	-	- 25	25	12	4			98 - P	2	82		1	1	84	÷.	Å	1		-	•	1	12	1	- 25		14	12	3	33	84	12	\$	- 23	-53	8.	1	4	Results	15	\$
		36	25	-	33	15	8	3		6 I	3	12		120	19	1	2/	8	2		198	19	1	9	- 63	-3		36	8	20	83	12	14	38	20	23	84	10	3/2	Show gro	ups	
		32	83	-55	28	8	33	- 8		89 - X	2	95	1		1	-		-			-	1	-	-	-	1	4	8	8	8	53	98	68	8	8	43	24	68	38	None		-
		85	32	53	33	12	85	- 3		81.1	3	15	83	31	83	22	15	8	25	81	52	35	82	25	53	22	35	12	98	31	52	33	38	98	31	51	°1			weets -		
		85	10	K	8	10	35	1		85 3	9	32	8	123	15	87	32	3	8	8	<u></u> [3]	32	35	8	25	12	12	12	8	22	12	32	13	8	12	53	8	z	- X		ěr.	
11-1																																										

Click on <sup>11</sup>/<sub>1</sub> to start the wind generator on frames.
 Complete the window like below:









 We assume that the roof and the lintel in the façade don't bear wind out of the plane. Then click 'OK' to generate the wind on the first frame.

🧇 Diamonds - S	Wind Generator for 2D frames
🥮 Eile Edit View Select Display Analysis Q	
	Bearing elements for loads in plane Bearing elements for loads out of plane
🐼 🖻 🗟 🛛 Venster 1 💽	
y <sup>y</sup> u <u>fii</u> 🔊	
Wind	
Geval 1 💌	
<b>紧</b> 望	
± 4 4	
	Geometry
	Total width of construction: 14,00 m Position of frame: 20,00 m
₩ <i>⊕</i>	Total depth of construction: 20,00 m Distance to frame in front: 5,00 m
	Distance to frame behind: 0 m
* Snow	
× Wind	left -> right   right -> left   mont -> back   back -> mont
• • • • • • • • • • •	Image: Provide the second s
899 A A	vind vind vind vind vind vind vind vind
* Seismic	🗌 downward
* Dynamic	
* Moving	
N 10 10 10 10 10 10 10	
	v upward wind v wind b -> fu (0 v
16 IN 46 16 16 16 IN 16 1	downward
N 30 60 10 10 10 10 10	FRONT
	<u>H</u> elp <u>QK</u>

- Now select the second frame and click on  $\overset{\text{m}}{\cong}$ . The only thing you have to change is the position of the frame. Diamonds will remember all the other parameters from the wind generation on the first frame.



- Repeat these steps for the all the other frames. When you come to the last frame, don't forget to set the bearing elements like in the first frame!
- Result:



# Step 19: Filling in the load group 'snow'

# To generate **snow**:

- Select the load group 'Snow' and the first sub load case 'Case 1' from the pull down menu.

- Click on \* to select the snow standard and the terrain parameters.

60 Die	
	amonds - staarvB2.dsi - [venster I - snow (H <= 1000 m) - Geval I (kiv, kivin, min, kiv/m, kivim, kiv/m, c)]
😿 🗟 🖷 🖂  Venster 1	
yg <sup>y</sup> u ∰ .⊖	Settings snow generator
Geval 1	Norm: EN 1991-1-3
	Country:  Zone: Central West
- D- D	Altitude: 0,00 m
Temperature     Snow	Snow load at ground level (Sk): 0,4 kN/m <sup>2</sup>
*	Return period adjustment 1,0 : Sn/Sk = 1,0
v Wind	Ce: Exposure coefficient
Dynamic     Moving	Ct: thermal coefficient 1
	Take into account snow weight on overhang at edge
1	<u>H</u> elp <u>Q</u> K
	, tz inter

- o Select the standard EN 1991-1-3 [--].
- Choose as country [--] 'Central West' 'Area 3'.
- Click 'OK' to close this window.
- Take a top view and select the first frame of the 3D hall.



- Click on To start the snow generator on frames. The first frame bears 2,5m of snow loads (because the distance between the frames is 5m).

🤗 Diamonds	Snow Generator	×
Eile Edit View Select Display		
D 🗃 🖬 🖪 🚳 🕅 🗠 ୍		
🛞 🗟 🖷 🔀 Venster 1		
γ <sup>ε</sup> <sup>∓m</sup> ⊕		
snow (H <= 10( -		ו ו
Geval 1 💌		
₩. 4 <sup>7</sup>		
+++++++++++++++++++++++++++++++++++++++		
€ 2×68		
Temperature     Snow		•
*		-1
· 天 <i>日 日</i>	Width supported by structural frame: 2,5 m	
Wind w w w w	Snow load cases:	
Seismic     Dynamic		- 11
v Moving		
3 6 7 7		

- Complete the window as here above. Then click 'OK' to generate the snow.
- Now select the second frame and click on <sup>1</sup>. This frame bears 5m of snow loads.

Diamonds - StaalVB2.bsf - [Vens	Snow Generator
♥ File       Edit       View       Select       Display       Analysis       Options       Wind         □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □       □<	
Viet         Viet <th< th=""><th>Width supported by structural frame:       5,00 m         Snow load cases:       Image: Case 1 minimum         Image: Case 1 minimum       Image: Case 1 minimum         Image: Case 2 minimum       Image: Case 2 minimum         Image: Case 2 minimum       Image: Case 3 minimum         Image: Cancel       Image: Cancel         Image: Lep       Image: Cancel</th></th<>	Width supported by structural frame:       5,00 m         Snow load cases:       Image: Case 1 minimum         Image: Case 1 minimum       Image: Case 1 minimum         Image: Case 2 minimum       Image: Case 2 minimum         Image: Case 2 minimum       Image: Case 3 minimum         Image: Cancel       Image: Cancel         Image: Lep       Image: Cancel

- Repeat these steps for the all the other frames. When you come to the last frame, don't forget to set the width supported by the frame to 2,5m.
- Result:


### 4.2.3.3 Making combinations

Step 20: Making combinations

Generate the combinations  $\frac{4}{10}$  as described in §4.1.3.3.

### 4.2.4 Generating the mesh

Step 21: Generating the mesh

Click on the button  $\square$  in the icon bar or select the menu instruction 'Analysis – Mesh'. Enter these distances:

Berekening van de mesh			
Staven Aantal verdelingen Minimum element grootte	8 🗲 0.35	m	
Help	<u>A</u> nnuleer <u>O</u> K		

Now click 'OK' to start the generation of the finite element net.

### 4.2.5 The global elastic analysis

#### Step 22: Elastic analysis

Follow the same method as described in §4.1.5.

Here below you find the bending moments for the combination ULS FC envelope and the vertical displacement for the combination SLS RC envelope.



#### Bending moments ULS FC



Deformation SLS RC

# 4.2.6 Parameters for steel verification 4.2.6.1 Buckling

Step 23: Setting up groups for buckling

See §4.1.6.1 for the method.

Define the groups for buckling around the y'(v)-axis as follows (in the image below the end releases, tie rods, supports and IPE120 are set invisible to increase the readability. The IPE's 120 should be ungrouped in both directions):



Define the groups for buckling around the z'(v)-axis as follows:



#### Step 24: Calculating the buckling lengths

Now calculate the buckling lengths 2.



In each direction (round y'(u)- or z'(v)-axis) Diamonds asks you for which type of structure and for which type of analysis (first or second order) you would like to calculate the buckling lengths.

It is important that you use the same type of analysis as what you indicate here. We choose 'semi displaceable nodes'.

With 'Combination for determination of non-linearities' you select 'SLS QP1'.

Click 'OK'.

### 4.2.6.2 Lateral torsional buckling

Step 25: Settings for lateral torsional buckling

The purlins are modelled in this structure, so that we don't have to do anything specific for lateral torsional buckling.

### 4.2.7 Steel verification

Step 26: Steel verification

Assume the same settings as for the 2D-frame in §4.1.7.

You'll obtain these results:



Results for strength (%)



Results for stability (%)

From the percentages we can deduct that the structure is sufficient for strength ( $\leq 100\%$ ), but not for stability (> 100%).

To get more information about the instability problem, you can double click the relevant bar. Then Diamonds will show you the detailed calculation.

### 4.2.8 Cross-section optimization

Step 27: Cross-section optimization

To start the optimization, click on the button  $\checkmark$  in the icon bar. Assume the same settings as in §4.1.8.

When the optimization is completed, a dialog box appears with the summary of the optimization.

٩	Optimisation report	×
	Bar number 13 changed from section IPE (EU) - IPE 120 to IPE (EU) - IPE 140     Bar number 14 changed from section IPE (EU) - IPE 120 to IPE (EU) - IPE 140     Bar number 15 changed from section IPE (EU) - IPE 120 to IPE (EU) - IPE 140     Bar number 16 changed from section IPE (EU) - IPE 120 to IPE (EU) - IPE 140     Bar number 17 changed from section IPE (EU) - IPE 120 to IPE (EU) - IPE 140     Bar number 18 changed from section IPE (EU) - IPE 120 to IPE (EU) - IPE 140     Bar number 19 changed from section IPE (EU) - IPE 120 to IPE (EU) - IPE 140     Bar number 20 changed from section IPE (EU) - IPE 120 to IPE (EU) - IPE 140     Bar number 21 changed from section IPE (EU) - IPE 120 to IPE (EU) - IPE 140     Bar number 21 changed from section IPE (EU) - IPE 120 to IPE (EU) - IPE 140	<
	Number of bars to adjust: 56 Number of bars not requiring any adjustment: 82	

Diamonds proposes you to change some cross-sections. You can accept or ignore the changes by (un)checking the corresponding line.

Once the optimization has ended, you should regenerate the mesh  $\square$ . Perform the elastic analysis again  $\square$ . Recalculate the buckling lengths  $\cancel{\mathbb{H}}$  and perform the verification again. The stiffness's and self-weights are indeed changed.

Note: under this amendment, the new verification sometimes gives results that are not yet close enough to 100%. A second optimization imposes itself.

### 4.2.9 Calculating connections

We illustrate how you can perform a detailed calculation of a connection.

The purpose of this calculation if to pass the geometry and the forces on a node to the calculation heart of PowerConnect. There the strength and stiffness of the connection is calculation as a function of its components (bolts, welds, stiffeners, ...). The calculated stiffness diagram is then taken into account by Diamonds for a more accurate global analysis.

### 4.2.9.1 Detailed calculation of a connection

Step 28: Detailed calculation of a connection

We restrict ourselves to two types of connections: the connection between a column and a beam and the connection of the ridge (beam – beam connection).

Make a selection of bears as shown in the image below. Also make the line numbers visible 1:



Now click on the icon IF in the icon bar. A dialog window will show all the possible connections in a list.

- Select the **beam-column** connection from the list. This concerns the connection of the column with the rafter.
- Choose the correct configuration.

<b>🖗</b>		Connection Design	- • ×
4	Point Nr. = 4 Left connection : Left beam Nr. = 11 Right connection : Right beam Nr. = 4	Similar connections list : point Nr:4, bar Nr: 11, 4 point Nr:22, bar Nr: 97, 69 point Nr:23, bar Nr: 98, 70 point Nr:24, bar Nr: 99, 71 point Nr:25, bar Nr: 100, 72	Select combinations Calculate connection Insert in library Insert on structure
5	Point Nr. = 5 Column Nr. = 5 Right beam Nr. = 12	Similar connections list : point Nr:5, bar Nr: 5, 12 point Nr:26, bar Nr: 73, 101 point Nr:27, bar Nr: 74, 102 point Nr:28, bar Nr: 75, 103 point Nr:29, bar Nr: 76, 104	Select combinations Calculate connection Insert in library Insert on structure
Heb Jan			<u>Cancel</u> <u>QK</u>

#### About the window 'Detailed calculation of a connection'

- All connections with the same geometric configuration are combined into one connection type.
- A connect may have different configurations (beam to column or beam on the column,...). The numbers on the drawing correspond to the line

numbers. This allows you to easily verify whether the proposed configuration is the desired one.

### 4.2.9.2 Selecting combinations

It goes without saying that not all combinations will be determining when verifying the connection.

By imposing limits (based on percentages) compared to the maximum occurring value, for each internal force (bending moment, axial and shear force), combinations can be filtered out.

However, don't set the limits too high: when the limits are for example 95%, the combinations for which both M, N, and V reach 90% at the same time, will not be verified. This combination may nevertheless be more dangerous than one with 95% for M and small percentages of N and V. Some caution is required.

Click on the button Select combinations to select the combinations.

<del>.</del>		Choose combination		- • ×
Number of selected co	ombinations :			9
Selection criterion				
Bending moment >	90 🚖	% of maximum bending moment		
Normal force >	90 🚖	% of maximum normal force		
Shear force >	90 🚖	% of maximum shear force		
Bending moment <	90 🜲	% of minimum bending moment		
Normal force <	90 🚖	% of minimum normal force		
Shear force <	90 🜲	% of minimum shear force		
Node Nr.4 bar Nr6, 12	2 : UGT FC 1			^
Node Nr.22 bar Nr77,	101 : UGT FC	1		
Node Nr.23 bar Nr78,	102 : UGT FC	1		
Node Nr.24 bar Nr79,	103 : UGT FC	1		
Node Nr.25 bar Nr80,	104 : UGT FC	1		¥
$\blacksquare \longleftrightarrow \Box$				
Help			<u>C</u> ancel	<u>OK</u>

If necessary, set all percentages to 90% and click 'OK'. A 'V' will appear in front of the button <sup>Select combinations</sup> to indicate this step is completed for the selected connection. The combinations will be saved, even when you witch to another connection.

### 4.2.9.3 Calculating the connection

At this point the program has all the information to calculate the connection. We start the PowerConnect module by clicking on Calculate connection.

The first dialog box asks you to choose the connection type. We opt for a connection with bolted end plate (first option).



PowerConnect generates a connection based on a number default parameters.



Now you can adjust the geometry of the connection. For example, select the column, then click once with the right mouse button on the column. Add a web plate.

🕼 Diamonds - PowerConn	ect 2015 - [Geometry] – 🗆 🗙
File Edit View Analysis Windows Options Help	
	Add or remove element ×
6.0	X Remove element
	- Add beam (LHS)
	4 Add backing plate (RHS)
	Add web plate
43	Add upper stiffener (RHS)
	Add lower stiffener (RHS)
	Add diagonal stiffener (RHS)
	Add transverse stiffener (RHS)
Ŭ	Help Qancel Qk
<	۷ ۲ ۱۵ ۱۵

Now double click on of the bolts to see their properties.



Click on it to calculate the connection. The 'Results' window (see figure below) will automatically come to the foreground. Herein, the results will be briefly summarized. In particular, you will find there the ultimate forces that the connection can resist, as well as the most critical combination.

In case the connection is not sufficient for the applied loads or when the connection contains certain dangers that require you attention, a warning appears in red.



In this report you will also find 2 coloured diagrams:

- The left-hand diagram is the work level based on the applied moment.
- The right-hand diagram is the work level based on the maximal moment (that the connection can resist).

From the left-hand diagram you can deduce which components should be adjusted to optimize the connection;

From the right-hand diagram you can deduce which components will fail if the connections is loaded to its maximum capacity.

Then close PowerConnect. In the first dialog window, a second 'V' appear, this time next to the button Calculate connection.

### 4.2.9.4 Add the connection to the library

Currently the connection and its results is saved temporarily. To save the connection in the intern library of the project, you will need to click the third button Insert in library. The internal library is related to the project. Later, you can add the connection to an external library. This way the connection can be used in other projects.

In the dialog bow, enter the name of the connection. After clicking 'OK' a third 'V' appears.

Connection name
Give a name to the connection
Beam-column
<u>C</u> ancel <u>O</u> K

The final step is assigning the connection to the model. With the button Insert on structure you firstly define the relation between the selected nodes and the connection from the library. Secondly the rigidity diagram is assigned to the bar ends of the node. When we recalculate the structure later, the correct value will be taken into account.

Ŷ		Connection Design	- <b>-</b> ×
Beam-Beam	Point Nr. = 21 Left connection : Left beam Nr. = 100 Right connection : Right beam Nr. = 68	Similar connections list : point Nr:3, bar Nr: 11, 3 point Nr:18, bar Nr: 97, 65 point Nr:19, bar Nr: 98, 66 point Nr:20, bar Nr: 99, 67 point Nr:21, bar Nr: 100, 68	Select combinations Calculate connection Insert in library Insert on structure
Beam-column	Point Nr. = 25 Column Nr. = 80 Right beam Nr. = 104	Similar connections list : point Nr:4, bar Nr: 6, 12 point Nr:22, bar Nr: 77, 101 point Nr:23, bar Nr: 78, 102 point Nr:24, bar Nr: 79, 103 point Nr:24, bar Nr: 80, 104	Select combinations Calculate connection Insert in library Insert on structure
			<u>C</u> ancel <u>QK</u>

The first connection is now dimensioned. The dialog window allows you to calculate other connection, despite a new elastic analysis is actually necessary. But this way you quickly and efficiently find a solution.

Repeat the procedure for example for the beam-beam connection from the list.

Once these two connections are calculated, we close the dialog box. Remember that you can display the name of the connection, if in the window configuration the option 'connection name' is active.



### 4.2.9.5 Assigning the connection

You may have noticed that only the connections on one side of the structure are calculated. Given the beam-column connections on the left have an identical configuration, we can assign the same connection to these nodes.

Select the beam-column connections like in the figure below and click on the icon <sup>III</sup>.



When clicking on the icon <sup>III</sup>, Diamonds will first check whether there are bars selected. Then there are three possibilities:

- Either a list of all defined connections appears in the library;
- Either the dialog box show the connection already assigned;
- Either Diamonds gives a list of all connection that can possibly be assigned to the selection.

In this case the last option is applicable, since Diamonds will find a connection in its (internal) library that can be assigned to the selected bars.

<del>\</del>	Connection library	- 🗆 🗙
Beam-column Column:HEA 260 Beam:IPE 300	Right connection : Sj,ini M+ = 101554.3 kNm/Rad Sj,ini M- = 72936.1 kNm/Rad Sj M+ = 36468.1 kNm/Rad Sj M- = 36468.1 kNm/Rad Category with M+ = rigid Category with M- = rigid General results	
Name Beam-column	Assign node to selection Comments	~
	Cancel	<u>O</u> K

To assign the connection to the selected bars, click on Assign node to selection. The name of the connection now appears in the model. The assignment of the same connection to the other nodes is done in the same manner.



### 4.2.9.6 Verifying the nodes in Diamonds

Since we have adjusted the stiffness of all connections, a new calculation is necessary.

Note: PowerConnect also calculates the connection class (rigid, semi rigid or flexible). When a connection is classified as rigid or flexible (so not semirigid!) and the option 'Calculate with the classification of the connections' in the window for the global analysis is active, then Diamonds will not use the stiffness diagram in the global analysis but assume the connection if completely rigid or perfectly flexible. When a connection is classified as semi-rigid, the stiffness diagram will always be taken into account, regards less the

After the global analysis  $\square$ , we should – in principle – calculate the connections again in PowerConnect, because the internal forces changed a little. However Diamonds is able to verify the applied forces without having to take the connections back to PowerConnect. As a result the number of manipulations and the computation time is realty reduced.

You perform this check with <sup>1</sup>/<sub>2</sub>. Then a dialog box appear containing four tables. Each tab corresponds to a specific type of connection.

ø					Connectio	n verification			×
þ	03	Ŧ							
ſ	Connection with mor	nent resistance							
	Connection	<mark>MSd-</mark> MRd- (kNm) < 1		NSd- NRd− (kN) < 1	NSd+ NRd+ (kN) < 1	<mark>VSd−</mark> VRd− (kN)	<mark>V5d+</mark> VRd+ < 1 (kN)	$\left[ \left( \frac{MSd}{MRd} + \frac{NSd}{NRd} \right)^2 + \left( \frac{VSd}{VRd} \right)^2 < 1 \right]$	î
	Beam-column	$\frac{-20,2}{-192,2} = 0,1$	$\frac{23,8}{269,5} = 0,1$	$\frac{0,0}{-768,3} = 0,0$	0,9 1341,2 = 0,0	$\frac{0,0}{-531,6} = 0,0$	$\frac{3,7}{531,6} = 0,0$	$\left(\frac{-20,2}{-192,2} + \frac{0,9}{1341,2}\right)^2 + \left(\frac{3,7}{531,6}\right)^2 = 0,0$	
	Beam-column	$\frac{-20,2}{-192,2} = 0,1$	$\frac{23,8}{269,5} = 0,1$	0,0 -768,3 = 0,0	0,9 1341,2 = 0,0	$\frac{-11,7}{-531,6} = 0,0$	$\frac{16,2}{531,6} = 0,0$	$\left(\frac{-20,2}{-192,2} + \frac{0,9}{1341,2}\right)^2 + \left(\frac{16,2}{531,6}\right)^2 = 0,0$	
	Beam-column	-51,4 -192,2 = 0,3	$\frac{165,5}{269,5} = 0,6$	0,0 -768,3 = 0,0	40,5 1341,2 = 0,0	0,0 -531,6 = 0,0	56,3 531,6 = 0,1	$\left(\frac{165,5}{269,5} + \frac{40,5}{1341,2}\right)^2 + \left(\frac{56,3}{531,6}\right)^2 = 0,4$	~
		ļa ij						QK	

For each node a verification of the various internal forces and the combination of them is performed. When a verification is not sufficient, it is marked in red. This way you get a quick overview of the connection that may cause problems.

## **5 Examples in timber** 5.1 Example 1: 2D frame

Required licenses:

✓ 2D Bars
 ✓ Timber Design

### 5.1.1 Purpose of the exercise

In this example we calculate a timber roof. We calculate the internal forces and stresses in the bars and after that we execute a timber verification (strength and stability).

The roof looks like this:



The timber quality is C18.

### 5.1.2 Defining the structure

Before building the geometry, we will expand the section library with crosssection frequently made off timber. This will make it easier to optimize the truss later.

#### Step 1: Expanding the section library

Go to 'Edit - Section library'.

Click on the [-] next to 'Groups' to add a new group. Name the group for example 'BEAMS' and hit the button 🛨 after that. Click on 'OK' to close this window.



Now click on ≡ to add a new cross section. Adjust the name to '38x150'. Click on '[-]' with 'Groups' and select the group 'BEAMS'.

<b>%</b>	Section library	×
End L2       T2       T2         ZETAVOR (Brausa) - Zv325x2,5       ZETAVOR (Brausa) - Zv325x3         ZETAVOR (Brausa) - Zv325x4         ZETAVOR (Brausa) - Zv350x3         ZETAVOR (Brausa) - Zv350x4         ZETAVOR (Brausa) - Zv350x4         ZETAVOR (Brausa) - Zv350x4         ZETAVOR (Brausa) - Zv350x4         ZETAVOR (Brausa) - Zv35x4         ZETAVOR (Brausa) - Zv400x3         ZETAVOR (Brausa) - Zv400x4         JSx150         LElp	Name       33x150         Groups       [-]         Geometry       Advanced         Image: Select section groups       -         Image: Select section group       Image: Select section group         Image: Select section group       Image: Select sectin group         Ima	•

Then click on for a the adjust the cross section properties. Choose for a rectangular form and enter the dimensions (B and H). The material is 'Timber C18'.

	Section
Cross-section Name 38x150 Shape Dimensions B 38 mm H 150 mm	Dimensions   Axes
Section properties           Image: Automatic calculation           General         Elastic         Plastic         Cold-formation	Material dependent  med members
Sy 427500 mm <sup>3</sup> Iy' 10687500 mm <sup>4</sup> iy' 43.3 mm Wel,y',t 142500 mm <sup>3</sup> Wel,y',b 142500 mm <sup>3</sup>	Sz     108300     mm³     alpha     0.0     o       Iz'     685900     mm²     Iyz     0     mm²       iz'     11.0     mm     It     2305872     mm²       /el,z',r     36100     mm³     Iw     0     mm²       /el,z',r     36100     mm³     Twm     62674     mm³
Material Material Timber C18	<u>C</u> ancel <u>QK</u>

Click 'OK' to place the section in the list.

>	Sectie bibliotheek	
E+         I 12         12         Y         I         I           Zoek         I         I         IIIIIIIIIIIIIIIIIIIIIIIIIIIIIIIIIIII	Naam default Groepen BALKEN Geometrie Geavanceerd B = 38.0 mm H = 150.0 mm	
Help		<u>A</u> nnuleer <u>O</u> K

Add a beam '38x175' the same way. Click on 1 to save the changes. Click 'Ok' to close the section library.

#### Step 2: Go to the 'Geometry' configuration

Defining the structure is always done in the 'Geometry' configuration. Click on B in the icon bar, or select the 'Geometry' configuration in the adjacent pull down menu.

Then check if you are in a front view. If this is not the case, then click on

the button in the icon bar or on the button  $z \xrightarrow{x}$  in the lower right corner and select the viewpoint 'Front view'. This way you activate a vertical drawing area.



You can either draw the structure with the mouse (see §2.3) or you could use the structure generator.

#### Step 3: Structure generator

You can insert the structure in Diamonds using different ways:

- Draw immediately on the screen with the mouse
- Draw on the screen by means of coordinates with the keyboard.
- Use the structure generator
- Import a DXF-file.

Click on the icon in the pallet. A dialog window will appear in which you can select the form of the structure you would like to generate. Opt for a roof like in the figure below.



Next Diamonds will ask you the geometric data of the roof. Enter the details as shown below.

Wizard: roof	- • <b>×</b>
	Left main rafter Roof slope: $35,0^{\circ}$ Number of intermediary nodes: $0 \ddagger$ Height Base point - Collar beam: $1,60^{\circ}$ m Horizontal projection length: $0,0^{\circ}$ m Section $38x175$ [C18] - C18 Right main rafter Roof slope: $35,0^{\circ}$ $\circ$ Number of intermediary nodes: $0 \oiint$ Height Base point - Collar beam: $1,6^{\circ}$ m Horizontal projection length: $0,0^{\circ}$ m Section $38x175$ [C18] - C18 Collar beam Number of intermediary nodes: $0 \oiint$ Height Collar beam - Crest: $1,6^{\circ}$ m Section $38x175$ [C18] - C18 Coordinates of left support $X = 0^{\circ}$ m $Y = 0^{\circ}$ m $Z = 0^{\circ}$ m
	Cancel OK

To change the cross section of the bars:

- Click on Section .
- Click on and select the desired section from the list. All beams have a rectangular section 38x175.

Section	/izard: roof -	. 🗆 🗙
Cross-section Name 38x175 Shape  Constant  Constant  Constant  Shape  Section Iibrary Constant  Section Iibrary Constant  Section properties F J <sup>A</sup> 1	Left main rafter Roof slope: 35.0 Number of intermediary nodes: 0 Height Base point - Collar beam: 1.60 Horizontal projection length: 0.00 Section 38x175 - Timber ( Right main rafter Roof slope: 35.0 Number of intermediary nodes: 0 Height Base point - Collar beam: 1.60 Horizontal projection length: 0.00 Section 38x175 - Timber ( Collar beam Number of intermediary nodes: 0 Height Collar beam - Crest: 1.60 Section 38x175 - Timber ( Coordinates of left support X = 0 m Y = 0 m Z = 0 m Cancel	•     •     m     m     m     m     m     m     m     m     m     m     m     m     m     m     m     m     m     m     m     m     m     m     m     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M     M
Help Cancel OK		

Then click on 'OK' to confirm you selection. Verify if the material is set to 'Timber C18'. Click 'OK' to assign the cross section to the beams. Then click twice 'OK' to draw the structure.

#### **Step 4: Completing the structure**

the centre of the bars.

Complete the structure with <. The intelligent cursor will help you find



Draw the other 2 lines as well.



#### Step 5: Copying the cross section and the material

The cross section and the material of the just defined bars are the same as the collar beam. We can copy these properties very easy by click with the right mouse button on one of the collar beams.



Select the option 'Copy properties'. Now select the 3 bars without cross section and material (use the SHIFT key), and click once with the right mouse button. Then paste the cross section and material on the bars.



### 5.1.3 Defining the loads

Step 6: Go to the 'Loads' configuration

We now leave the 'Geometry' configuration and activate the 'Loads' configuration to enter the loads. Click on the button  $\square$  in the icon bar or select in the adjacent pull down menu the 'Loads' configuration.

1	Loads	R	L11	Ж	-
1 .	Geometry Loads				
	Results Mesh				

#### About the 'Loads' configuration

With the 'Loads' configuration windows comes a separate pallet containing all the functions for defining loads and generating combinations. Note that the point of view remains unchanged when switching between the configurations.

### 5.1.3.1 Creating the load groups

Step 7: Creating load groups

Before defining any loads, you have to make the different load groups. Click on the button  $\gamma_{g}^{\gamma_{u}}$ . You'll see the following screen:

-				D	iamono	ds - [Ve	nster 1	- Selfw	eight (	kN, kNi	n, mm,	kN/m, l	kNm/m, kN/m², °(	[)]		_ 0	×
-	<u>File</u>	dit <u>V</u> iew <u>S</u> elect [	Display	<u>A</u> nalysis	Option	ıs <u>W</u> ind	ows <u>H</u>	elp								-	8 ×
	D 🖻	🖬 🖪 🚳 🕅 🛛	$\mathbf{r}$	FFE	ΗĦ	##    '	<b>1.:</b>    🗵	3 🖬 i	te d	5 8:	<u>a</u> ()	% I⁄	⊨ <b>⊫</b> +9				
[	* 5	🖹 🔀 Venster 1		•	1   La	ads				•	<b>F</b> 14		# 🖪 🖑 Q	0. 🛛 🖉 🔟	RA		8
	γ <sub>g</sub> <sup>γ</sup> u 41 1 Selfweight	i No												- AU		Active level Level 1	<b>•</b> 7 m
4	>								Load	l grou	os						×
	Load fa	ctors for		EN 19	90	T	] [	•	Con: Desi	sequence gn lifetim	e class	2	<ul> <li>♦ years</li> </ul>	Service class	1 👤		
		Add load group		Insert	load gro	up				Dele	te load g	roup		Several	load cases	per group	
		Name load group	$\gamma_{\rm uls}$	$\gamma_{uls+}$	$\gamma_{\rm sls}$	$\gamma_{\rm sls+}$	$\psi_0$	$\psi_1$	ψ2	φ	ξ	to	Combination for cracking	k <sub>mod</sub>	Load	Action	
	<ul> <li>Image: A start of the start of</li></ul>	Selfweight	1,35	1,00	1,00	1,00	1,00	1,00	1,00	1,00	0,85	28		permanent	-	<u>+++</u>	
	<b>√</b>	dead loads	1,35	1,00	1,00	1,00	1,00	1,00	1,00	0,00	0,85	0		permanent	_	<u>+++</u>	
	$\checkmark$	life loads A : ho 🔻	1,50	0,00	1,00	0,00	0,70	0,50	0,30	0,00	1,00	0		medium term	—	<u>+++</u>	
	In	compatible load group	)S		Linked	l load gro	ups			Cc	ombine lo	adcases					
	Hel	p												Cano	el	<u>О</u> К	
																	11.

#### About the window 'Load groups'

- In the menu on top you select to which standard the safety and combinations factors should answer. Currently this is set to 'EN 1990 [--]' which means Eurocode 0 without a national annex.
- In some national annexes the safety coefficients also depend on the consequence class and the design lifetime of the structure. Both are

linked to the economic and/ or social interest of the structure. A higher/ longer consequence class/ design lifetime will lead to higher safety factors.

- On the right you can enter **the climate class**. This climate class is representative for a certain moisture content of the air/ the timber. Diamonds uses the climate class you determine the modification factor  $k_{mod}$ . The modification factor  $k_{mod}$  takes the influence of the load duration and the moisture content on the strength properties into account. The modification factor  $k_{mod}$  depends not only on the climate class but also on the type of timber and the load duration class. The **load duration class** must be specified for each load case in the last column.
- In the table below the load cases 'Self-weight', 'Dead load' and 'Life load' are defined by default. You can freely rename or delete them, except for 'Self-weight'. The fill-in boxes to the right of the name of each load case include the safety *γ* and combination factors *Ψ* required for the automatic generation of the load combinations.
- We don't discuss the other parameters in this window.

#### Step 8: Changing the type of the life load

This window will contain 3 load groups by default, namely: 'Self-weight', 'Dead loads' and 'Life load A: housing'.

Set the type of the life load to 'Life load H: Roofs'. Note that the safety factors and combination coefficients change when you do this.

			D	iamon	ds - [Ve	nster 1	- Selfw	eight (	(kN, kN	m, mm	, kN/m,	kNm/m,	kN/m², °	'C)]			- 🗆 🗙
🤗 Eile	e <u>E</u> dit <u>V</u> iew <u>S</u> elect <u>D</u>	isplay	Analysis	<u>Option</u>	ıs <u>W</u> inc	lows <u>H</u>	elp					en:					- 8 ×
	🗧 🖬 🖪 🚳 🚺 📔	S OI	H H I	ΗĦ	##	<b>1.:</b>   🗵		l ke o	5 8:	<u>m</u> (9	Fe% 15/		- t-\$				
× 8	🗟 🖷 🔀 Venster 1		•	1	ads				•	B ==		<b>M</b> 🗖	< 77 Q	•			
$y_g^{\gamma_u}$	tim 😌																Active level
Selfw	eight 💌																2.7m
<b></b>								Load	d grou	ps							- • ×
	d fa abaar faa		-			_		Conce		daga		<u>л</u>		Foruisa	daga 🔽	1	
Loa	d factors for		JEN 199	0	-		-	Desig	n lifetime	uass	50 🛋	vears		Service		1	
								D'Congr				, , ca. s					
	Add load group		Insert	oad grou	ıp				Delet	e load gr	oup				Seve	ral load cases	per group 🦳
	Name load group	$\gamma_{\rm uls}$ -	$\gamma_{uls+}$	$\gamma_{\rm sls}$	$\gamma_{sls+}$	ψ <sub>0</sub>	$\psi_1$	ψ <sub>2</sub>	φ	ξ	to	Combinati cracking	k <sub>mod</sub>	Load	Action		
$\checkmark$	Selfweight	1,35	1,00	1,00	1,00	1,00	1,00	1,00	1,00	0,85	0		perm	—	<u>+++</u>		
<ul> <li>✓</li> </ul>	dead loads	1,35	1,00	1,00	1,00	1,00	1,00	1,00	1,00	0,85	0		perm	_	<u>+++</u>		
~	life loads H : roofs 🔻	1,50	0,00	1,00	0,00	0,00	0,00	0,00	1,00	1,00	0		mediu	—	<u>+++</u>		
	life loads G : heavy: ∧ life loads H : roofs snow (H > 1000 m) snow (H <= 1000 m Wind Temperature Seismic Crane loads ✓																
	Incompatible load groups	;		Linked	load grou	ıps			Cor	nbine loa	dcases		Unq	group load	cases		
	Help														G	ancel	<u>o</u> ĸ

Load groups can be added with Add load group and deleted with Delete load group.

#### Step 9: Making a new load group 'wind'

Now we will add a new load group 'Wind';

- Click on Add load group
- From de predefined list, select the load type 'Wind'. Note that all the safety factors and combination coefficients change when you do this.

			D	iamono	ds - [Ve	nster 1	- Selfw	eight	(kN, kN	m, mm	, kN/m,	kNm/m,	kN/m², °	C)]			. 🗆 🗙
🤗 <u>F</u> ile	<u>E</u> dit <u>V</u> iew <u>S</u> elect <u>D</u>	isplay	<u>A</u> nalysis	Option	ıs <u>W</u> ind	ows H	elp	<b>.</b>									- 5 ×
	• 🖬 🖪 🖨 🕅 🗍	n a		H m		<b>T.:</b>    Ø	( 🖩 )	l Ke o	5 8	ш®	Fe% IN		4				
* 6	🛛 🖷 🔀 🛛 Venster 1		•	1 Lo	ads				•	<b>B 1</b>		A D	87 Q	۹ 🖾	1	RED	
$y_g^{\gamma_u}$ Selfwe	nli ⊙ im ∾ ight ▼													<u> </u>			ive level rel 1 💌
<del>```</del>								Load	d grou	ps						-	
Load	factors for		EN 199	D	•		•	Conse Desig	equence n lifetime	dass	2	years		Service	class 1	•	
	Add load group		Insertik	oad grou	ıp				Delete	e load gro	oup				Sever	al load cases per	group 🥅
	Name load group	γ <sub>uls-</sub>	$\gamma_{uls+}$	γ <sub>sls-</sub>	$\gamma_{\rm sls+}$	$\psi_0$	$\psi_1$	ψ2	φ	ξ	to	Combinati cracking	k <sub>mod</sub>	Load	Action		
$\checkmark$	Selfweight	1,35	1,00	1,00	1,00	1,00	1,00	1,00	1,00	0,85	0		perm	_	<u>+++</u>		
<ul> <li>✓</li> </ul>	dead loads	1,35	1,00	1,00	1,00	1,00	1,00	1,00	1,00	0,85	0		perm	_	<u>+++</u>		
<ul> <li>✓</li> </ul>	life loads H : roofs	1,50	0,00	1,00	0,00	0,00	0,00	0,00	1,00	1,00	0		mediu	—	<u>+++</u>		
$\checkmark$	Wind	1,50	0,00	1,00	0,00	0,60	0,20	0,00	1,00	1,00	0		short	per la companya de la	<del>+++</del>		
	life loads G : heavy ' ∧ life loads H : roofs snow (H <= 1000 m) snow (H <= 1000 m Wind Temperature Seismic Crane loads ✓											1					
	Incompatible load groups			Linked	load grou	ips			Con	nbine loa	dcases		Ung	roup load	cases		
	Help														<u>C</u> a	ancel	<u>o</u> k

We consider 16 cases of wind:

- Wind left upward -> right upward ( $c_{pi} = -0.3$ )
- Wind left upward -> right downward ( $c_{pi} = -0.3$ )
- Wind left downward -> right upward ( $c_{pi} = -0.3$ )
- Wind left downward -> right downward ( $c_{pi} = -0.3$ )
- Wind left upward -> right upward ( $c_{ni} = 0,2$ )
- Wind left upward -> right downward ( $c_{pi} = 0,2$ )
- Wind left downward -> right upward ( $c_{pi} = 0,2$ )
- Wind left downward -> right downward ( $c_{pi} = 0,2$ )
- Wind right upward -> left upward ( $c_{pi} = -0,3$ )
- Wind right upward -> left downward ( $c_{pi} = -0.3$ )
- Wind right downward -> left upward ( $c_{pi} = -0.3$ )
- Wind right downward -> left downward ( $c_{pi} = -0.3$ )
- Wind right upward -> left upward ( $c_{pi} = 0,2$ )
- Wind right upward -> left downward ( $c_{pi} = 0,2$ )
- Wind right downward -> left upward ( $c_{pi} = 0,2$ )

- Wind right downward -> left downward ( $c_{pi} = 0.2$ )

We will implement these 16 cases of wind as 'sub load cases'. To define sub load cases check the option Several load cases per group .

Next to each load group you can now indicate how many sub load cases you want. You'll also noticed the column with the little people in it ( $\Re$  or  $\Re$ ). Click on the icons ( $\Re$  or  $\Re$ ) until they don't hold hands  $\Re$ .

#### About 'Sub load cases'

'The little people' can hold hands or not. Click on the icon to switch between the two.

- If they do not hold hands *x*, this means that all sub load cases are incompatible (i.e. the sub load cases can never act together). This is for example the case with wind and snow.
- If they hold hands #, this means that all sub load cases act together. This is for example the case with dead loads.

Complete the table as below:

oad	factors for		EN :	1990 💽 -		• •	Consequ Design <mark>l</mark> if	ence clas etime	s 2	•	years		Servi	ce dass	1				
Add load group Insert load group									Delete load group								Several lo	ad cases p	er group
	Name load group	#	Туре	Name load case	$\gamma_{\rm uls}$ -	$\gamma_{uls+}$	$\gamma_{\rm sls}$ -	$\gamma_{\rm sls+}$	ψ	$\psi_1$	ψ <sub>2</sub>	φ	ξ	to	Combinati cracking	k <sub>mod</sub>	Load	Action	
~	Selfweight	1			1,35	1,00	1,00	1,00	1,00	1,00	1,00	1,00	0,85	0		perm	-	<u>+++</u>	
~	dead loads	1			1,35	1,00	1,00	1,00	1,00	1,00	1,00	1,00	0,85	0		perm	1	***	
~	lif life loads H : roofs	1			1,50	0,00	1,00	0,00	0,00	0,00	0,00	1,00	1,00	0		mediu	() <del>, ()</del> ()	<del>***</del>	
~	Wind	16	ł£	wind l u -> r u (-0.3)	1,50	0,00	1,00	0,00	0,60	0,20	0,00	1,00	1,00	0		short	Pao	<u>+++</u>	
				wind l u -> r d (-0.3)															
				wind I d -> r u (-0.3)															
				wind I d -> r d (-0.3)															
				wind l u -> r u (0.2)															
				wind l u -> r d (0.2)															
				wind I d -> u (0.2)															
				wind I d -> r d (0.2)															
				wind r u -> l u (-0.3)															
				wind r u -> l d (-0.3)															
				wind r d -> l u (-0.3)															
				wind r d -> l d (-0.3)															
				wind r u -> l u (0.2)															
				wind r u -> l d (0.2)															
				wind r d -> l u (0.2)													_		
				wind r d -> l d (0.2)															
	Incompatible load gro	ups	1	Linked load groups		1		Combin	e loadca	ises	10	Ur	ngroup la	adcases	1				

Note: because we calculate a 2D truss, we will not generate wind out of the plane, but this can also be done with the wind generator.

#### Step 10: Making a new load group 'snow'

Finally add the load group 'Snow':

- Add another load group by using the button Add load group
- From de predefined list, select the load type 'Snow (H ≤ 1000m)'. Note that all the safety factors and combination coefficients change when you do this.
- Define three sub load case, named:
  - o Case 1
  - o Case 2
  - o Case 3

bad	factors for		EN	1990 💌 🖵	s	•	Consequ Design <mark>l</mark> if	ence da: etime	ss 2	•	years		Servi	te dass	1				
	Add load group		Inse	ert load group			1	Delete lo	ad group	,							Several k	oad cases pe	r group 🖡
	Name load group	#	Туре	Name load case	γ <sub>uls-</sub>	$\gamma_{uls+}$	γ <sub>sls-</sub>	$\gamma_{\rm sls+}$	$\psi_0$	$\psi_1$	ψ2	φ	ξ	to	Combinati cracking	k <sub>mod</sub>	Load	Action	
-				<sup>↓</sup> wind I u -> r u (-0.3)															
				wind l u -> r d (-0.3)															
				wind l d -> r u (-0.3)															
				wind I d -> r d (-0.3)															
				wind l u -> r u (0.2)															
				wind l u -> r d (0.2)															
				wind I d -> u (0.2)															
				wind l d -> r d (0.2)															
				wind r u -> l u (-0.3)															
				wind r u -> l d (-0.3)															
				wind r d -> l u (-0.3)															
				wind r d -> l d (-0.3)															
				wind r u -> l u (0.2)															
				wind r u -> l d (0.2)															
				wind r d -> l u (0.2)															
1	snow <mark>(</mark> H <= 1000 m)	3	Ł£	Case 1	1,50	0,00	1,00	0,00	0,50	0,20	0,00	1,00	1,00	0		short	*	+++	
				Case 2															
				Case 3															
	Incompatible load arou	ine	1	Linked load groups		1			e loadra	rec	1	1 kr	aroun la	adraces	1				

Next click on 'OK'.

### 5.1.3.2 Filling up the load groups

Now the loads groups are defined, we can assign loads to them.

#### Step 11: Filling in the load groups 'Self-weight', 'Dead loads' and 'Life load'

- The **self-weight** is calculated automatically by Diamonds.
- On both rafters we add a permanent load.
  - Use the pull down-menu to activate the load group 'Dead loads'.
  - Now select the rafters and click on the button . Note that only those icons will be active that can be applied on the selected elements.
  - o Complete the window as follows:







Using the image above, verify if you have entered the loads correctly. If you made a mistake:

- o Double click the bar and edit values in the table.
- OR select the incorrect bars, delete the loads with <sup>™</sup> and regenerate the loads.
- Work the same way to add 0,4kN/m on the rafters.



#### Step 12: Filling in the load group 'wind'

#### To generate **wind**:

- Click on the button **Ground level** and set the ground level to -6,0m. We suppose that this roofs stands on a building that's 6m tall.



- Click on <sup>1</sup> to define the wind standard and the terrain parameters.

	Terrain parameters ×	
D 🗃 🖬 🗟 🖨 🕅 🗑 🖥 ⊠ Venster 1 <sub>V</sub> V, nit હ	Norm: EN 1991-1-4	Active level
Vind  Geval 1	Reduction with factor 0.85 for lack of correlation       Structural factor CsCd:	Verdieping 1 2.70 m 2.70 m
7m 〒* + シージ 中 シージ	Direction factor Cdir: 1.0 Season factor Cseason 1.0	Evel manager
Temperature     Snow     Wind	Air density: 1.3 kg/m³ ✓ Probability factor Cprob: 1.0	X = 0.00 m Y = 2.70 m Z = 0.00 m Representation
r Selsmic ▼ Dynamic	C V <sub>h.0</sub> by location:         Map           If v <sub>b.0</sub> = wind velocity:         26         m/s           C q <sub>L</sub> = wind pressure:         0.42         kN/m²	Size Font 20 🗲
v Moving	D Terrain Type: I  Choose	Symbols 20 文 Loads 10 文
	Orography coefficient	Show groups
	<u>H</u> elp <u>Cancel OK</u>	

- Select the standard EN 1991-1-4 [--].
- Opt for a basic wind velocity of 26m/s and a terrain type I.
- Click 'OK' to close this window.
- Choose the load group 'wind' and the first sub load case 'wind I up -> r down (-0.3)' from the pull down menu.
- Select the entire structure ('CTRL + A' or use a selection window) and click on <sup>11</sup> to start the wind generator on frames.



🧇 Diamonds - H	🥹 Wind Generator for 2D frames 🗕 🗖 🔜
🥮 <u>F</u> ile <u>E</u> dit <u>V</u> iew <u>S</u> elect <u>D</u> isplay <u>A</u> nalysis <u>O</u>	
	Bearing elements for loads in plane Bearing elements for loads out of plane
🗑 🗟 🗟 🔟 Windows 1 🔄 🚺	$\wedge$
γ <sup>g</sup> <sub>n</sub> $\frac{1m}{\sqrt{11}}$ $\frac{1}{\sqrt{2}}$	
Wind	
Case 1	
** -	
+ + + + + + + + + + + + + + + + + + + +	
	Geometry
$\forall \geq \mathfrak{P}$	Total width of construction: 9,1 m Position of frame: 5,00 m
AT IT	Total depth of construction: 10,00 m Distance to frame in front: 0,45 m
✓ Temperature	
Wind	left -> right right -> left ront -> back back -> front
P 4 _	IF Cpi -0,30 ▼ 🗐 FACK
199 B B	Image: wind ru → lu (· m)     Image: wind ru → lu (· m)
× Seismic	Image: wind wind wind wind ru -> I d (
Vynamic     Moving	v right downward - left downward wind wind wind r d -> l u (·▼
1	
	v right upward -left downward wind wind v wind ru ->1 u ((↓
	I right downward - left upward wind wind wind r d -> I u (( ▼
	✓ right downward - left downward     wind     ▼     wind r d -> I d (( ▼
	<u>Help</u> <u>Cancel</u> <u>QK</u>

 Complete the window as above. Then click 'OK' to generate the wind.



#### About the wind generator

- The **upper part** presents a copy of the selected structure with the end points taken into consideration. The bars in bold represent the periphery of the frame on which the wind is expected to act.
- In the box 'Geometry' you note the total depth of the structure and the position of the frame relative to the façade (FRONT) and the distance between the previous and next frame. The image on the right shows you a floor plan of the structure that indicates the position of the frame with respect

to the front van back sides of the building. Also the previous and next frame are shown in dotted line. The coordinate system clearly explains the orientation of the building.

In the tabs 'left -> right, 'right -> left, 'front -> back' and 'back -> front' you
indicate which wind case should be generated, which internal pressure
coefficient c<sub>pi</sub> should be taken into account an in which sub load cases the
generated wind should be placed.

#### Step 13: Filling in the load group 'snow'

To generate **snow**:

- Select the load group 'Snow' and the first sub load case 'Case 1' from the pull down menu.
- Click on 🕸 to select the snow standard and the terrain parameters.

<b></b>	Diamonds - StaalVB1.bsf - [Window 1 - snow (H <= 1000 m) - Geval 1 (kN, kNm, mm, kN/m, kNm/m, kN/m <sup>2</sup> , °C)] 🐡 🗕 🗖 🗙
🥮 Eile Edit View	Select Display Analysis Options Windows Help
	●●● ● ● ● ● ● ● ● ● ● ● ● ● ● ● ● ● ●
🛛 😨 🖱 🔀 Win	dow 1 🔽 🚺 Loads エ 🏟 🖽 💹 🤤 🚀 🕁 🥐 🗨 🕲 🖉 🔟 🖾 👘
γ <sup>g</sup> <sup>μ</sup> mit ⊙	Active level
snow (H <= 10( 💌	Settings snow generator
Geval 1 💌	
X I	Norm: EN 1991-1-3
J. J. M.	
$+ \sim \sim$	Area: 3 V Map
+ > >	Altitude: 0.00 m
- 27 	Snow load at ground level (Sk)
* Temperature	
*	Return period adjustment V = 1.0 : Sn/Sk = 1.0
894	
- Wind	
Seismic     Dynamic	Ct: thermal coefficient
Y Moving	
	Help Cancel OK
	None 💌

- Select the standard EN 1991-1-3 [--].
- Choose as country [--] 'Central West' 'Area 3'. More information about these parameters can be found in the Reference Manual.
- Click 'OK' to close this window.
- Select the entire roof ('CTRL + A' or use a selection window) and click on <sup>1</sup> to start the snow generator on frames.

🤗 Diamonds	Snow Generator
Pile       Edit       View       Select       Display         D       Pile       Dit       Dit       Pile       Dit         View       View       View       View       Pile       Dit         View       View       View       View       View       Dit         View       View       View       View       View       View         View       View       View       View       View       View         View       View       View       View       View       View       View         View       View       View       View       View       View       View       View       View       View       View       View       View       View       View       View       View       View       View       View       View       View       View       View       View       View       View       View       View       View       View       View       View       View       View       View       View       View       View       View       View       View       View       View       View       View       View       View       View       View       View       View       View<	
Wed Selamic V Dynamic V Moving	Width supported by structural frame:       0,45 m         Snow load cases:       Image: Case 1 model         Image: Case 2 model       Image: Case 2 model         Image: Case 2 model       Image: Case 3 model         Image: Case 3 model       Image: Case 3 model         Image: Cancel       Image: QK

 Complete the window as here above. Then click 'OK' to generate the snow.



#### About the Snow generator

- The **upper part** presents a copy of the selected structure. The bars in bold represent the periphery of the frame on which the snow is expected to act.
- In the lower part you indicate the snow cases that should be generated and in which sub load case they should be placed.

### 5.1.3.3 Making combinations

Before starting the calculations we need to generate the combinations first.

#### Step 14: Making combinations

Click on the button in the pallet in the 'Loads' configuration window.



Click on the button <sup>#</sup> Generate all combinations automatically, indicate in the pull-down menu that you wish to use the classic but conservative Eq. 6.10 and select all limit states.

After pressing 'OK', all the combinations required by the standard will show, grouped by limit state. If desired, you can change these combinations Modify a combination or define combinations yourself <u>Create a combination</u>. One combination can be deleted with <u>Delete a combination</u>. To delete all combinations click <u>Delete all combinations</u>.
Loads combinations – 🗆 🗙											
1	Generate all combinations automatically     Create a combination										
ULS	ULS FC (ultimate limit state - fundamental combination)										
Load ULS	Load group ULS FC (ultimate limit state - fundamental combination) I <= 1000 m)										
ULS	SC (ultimate	limit state - se	ismic combinat	ion)		50 x 1.50					
SLS F	RC (servicea	bility limit state	=/ e - rare combin	ation)		50 x 1.50					
SLS F	FC (serviceal OP (servicea	bility limit state bility limit state	e - frequent co e - quasi-perma	mbination)		00 x 1.50					
4	ULS FC 4	1.00 x 1.35	1.00 x 1.35	0.00	0.00	0.00					
5	ULS FC 5	1.00 x 1.00	1.00 x 1.35	1.00 x 1.50	0.60 x 1.50	0.50 x 1.50					
6	ULS FC 6	1.00 x 1.00	1.00 x 1.35	0.00	1.00 x 1.50	0.50 x 1.50					
7	ULS FC 7	1.00 x 1.00	1.00 x 1.35	0.00	0.60 x 1.50	1.00 x 1.50					
8	ULS FC 8	1.00 x 1.00	1.00 x 1.35	0.00	0.00	0.00					
9	ULS FC 9	1.00 x 1.35	1.00 x 1.00	1.00 x 1.50	0.60 x 1.50	0.50 x 1.50		~			
	Modify a combination Delete a combination Delete all combinations										
	<b>1</b>	Import combin	ations		Export com	binations					
H	elp					<u>C</u> ancel		<u>o</u> ĸ			

Click 'OK' to close the window with the load combinations.

The names of the different load combinations are now listed in the pull down menu of the pallet 'Loads'. When you select one of these combinations, then the whole of the loads that will act during this combination will be shown.



### 5.1.4 Generating the mesh

Step 15: Generating the mesh

Click on the button  $\bowtie$  in the icon bar or select the menu instruction 'Analysis – Mesh'. Leave everything on default and hit 'OK'.



### About the mesh generator

Here you enter the number of elements in which a bar should be divided: 8 division is a good value.

Meer information on our support website:

http://buildsoftsupport.com/knowledge-base/how-to-pick-the-mesh-size/

## 5.1.5 The global elastic analysis

The calculation of the structure is performed in three steps:

- First we calculate the internal forces with an elastic analysis
- Then we verify the strength and stability according to a certain standard (see §5.1.6 and §5.1.7).
- Finally we will optimize the cross sections so we obtain the most economical cross sections (see §5.1.8).

### Step 16: Elastic analysis

To start the analysis, select the command 'Analysis – Elastic Analysis'. You can also start the analysis directly using the function key **F9** or use the icon  $\blacksquare$  on the icon bar. Following dialog box appears:

<del>o</del>	Diamonds - HoutVB1.bsf - [Windows 1 - SLS QP 1 - (kN, kNm, mm, kN/m, kNm/m, kN/m², °C)]	- 🗆 🗙
🥮 <u>F</u> ile <u>E</u> dit <u>V</u> iew <u>S</u>	elect <u>D</u> isplay <u>A</u> nalysis <u>O</u> ptions <u>W</u> indows <u>Help</u>	- 8 ×
	●■二★◎①日%~★★★■図■1	
🛞 🗟 着 🔀 Window	ws 1 🔄 🚺 Loads 🔄 🖗 🖳 💹 🖂 🚀 🗛 🕲 🖉 🔟 🗷	
γ <sub>g</sub> <sup>γ<sub>u</sub></sup> <u>im</u> ö	Analysis settings	Active level
SLS QP 1	Structural Soil Dynamic	2.7 m
₩. I	Analysis	2.7 m
$\downarrow \not \rightarrow \not \Rightarrow$		Level manager
$\varphi \gg \emptyset$	C 2nd order analysis	Ground level
R. 9	Precision for normal forces in bars: 1 %	X = 0.0 m
Tanana a	max, number of second order iterations;	Y = 2.7 m 7 = 0.0 m
<ul> <li>Snow</li> </ul>	max. number of iterations for nonlinearities: 50	Representation
*	Take shear deformation into account for beams	880
$\mathbb{R} \otimes \mathbb{P}$	▼ Global Imperfections	<b>A</b>
Wind     Seismic	▼ Joints	Cire
* Dynamic	▼ Concrete Cracking	Font 10 文
- Moving	▲ Timber	Symbols 15 🚖
		Loads 10 🚖
	Take creep into account : E = E / (1 + Kdef)	Results 15 🚖
	Help Cancel Previous Next > OK	Show groups
	Low Stowed Stowed A	None 💌
		X 2000
		1

We choose a first order calculation and confirm with 'OK'. A dialog box displays the progress of the calculation.

Performing analysis
Static analysis
Solving combinations
Combination 17

The button ellows you to stop the calculation. If you stop the calculation, you'll have to completely restart it later.

### Step 17: Go to the 'Results' configuration

To see the results of the calculations, you click on 🖻 in the icon bar or select in the adjacent pull down menu the 'Results' configuration.

Results	R	<b>L11</b>	Ж	
Geometry Loads				
Results Mesh				
	Results Geometry Loads Results Mach	Results  Geometry Loads Results Mech	Results  Geometry Loads Results Mach	Results Geometry Loads Results Mach

#### About the 'Results' configuration

In the corresponding pallet on the left side of the model window, you'll see several buttons, each representing a specific group of results.

- Only those buttons for which a calculation is carried out are available.
- Once one of these buttons is pressed, the corresponding partial results can be retrieved through the buttons located below.
- Then you indicate for which load combination you wish to see the results. In a first pull-down menu, select the type of load combination (individual load group, ULS FC, ULS SC, SLS RC, SLS FC or SLS QP), then specify which load group or load combination must be shown. In the case of a load combination you can choose between either an individual load combination (indicated by a number) or the envelope. In those situations where the result suggests an envelope, it may be possible that for some results you can opt for the minimum (min) or maximum (max) results to be displayed.

Below, we list some results.

### **Step 18: Deflection**

By default Diamonds will show you the deformation in de Y-direction for the first combination (or the first load group when you also made the combinations for the load groups – which is the case in this example).

You'll notice that the button for viewing the displacements I is active and that the button for vertical displacements according to the global Y-axis P is active.



The figure above is shown in wireframe  $\textcircled{1}{2}$  and we opted for a front view.

Now select the combination group 'SLS RC' and choose for the envelope of the results. We notice that the maximum deflection is 18,4mm.



Note that you can arrange the size of the results representation using the pallet 'Size' located on the right- hand side of the model window.

#### About the scale

A symmetrical colour scale is used by default for all results in Diamonds. However, you may choose a different scale in the 'Results configuration' .

You should understand the default scale as follows: the limits of the colour pallet correspond to the largest positive OR negative value. The colour scale runs from -1,0 to +1,0mm. However, the largest or smallest value is displayed above and below the scale. Consequently, for this example, only the lower half of the colour pallet is used.

#### **Step 19: Deformation in the detailed window**

Now select the left rafter. Click on the icon  $\exists$  on the right of the icon bar to ask a detailed result. A new window will open. On the left you will find all the buttons of the 'Results' configuration applicable for line elements (beams and columns).



Note that in this window the deformations are defined according to the local axes of the bars. Also the angular rotations  $\varphi_{\chi'}$  (round the local x'axis) and  $\varphi_{\gamma'}$  (round the local y'-axis) are dispalyed.

### Step 20: Stress in the detailed window

Now show the elastic stresses for the combination ULS FC envelope.



You can retrieve the results at any position using the slider. Moreover, you can also enter a distance. Consult for example the results on 2,45m. Enter '2,45' under the white arrow.

With a combination envelope the determining combination appears. You can disable this by clicking once on the button  $\mathbb{H}^{2}$ , this will change in  $\mathbb{H}^{2}$ .

Click 'OK' to close this window.

### Step 21: Stress in the detailed window (section level)

Now show the elastic stresses in the cross section for the combination ULS FC1.

- Select the left beam and click on the icon a on the right of the toolbar.
- Or double click the left beam.

This window opens:



You can retrieve the results at any position using the slider. Moreover, you can also enter a distance. Consult for example the results on 2,50m. Enter '2,50' under the white arrow.

Move the mouse over the cross section to see the stresses at any position.

#### About detailed results on cross section level

Getting started with **Diamonds** 

Choose on the left top for which **load group** or load combination you would like to see the stresses.

With the **slider** on top of the window you can set the section for which you would like to see a detail of the stresses. By clicking on the distance below the slider, you can enter a position of your choice.

Results field with scale:

- In the results field the selected profile is graphically represented together with its principal axes of inertia. When a cross section is double symmetric, these axes will coincide with the local axes.
- On the principal axes you'll see red points. These are the points for which the stress results  $(N + M_y \text{ and } N + M_z)$  are presented in the global results window of Diamonds. The position of these points is determined as the intersection of the principal axes with the cross section's bounding box.
- When you come near these red points with the cursor, Diamonds will snap to them.
- Move the mouse over the section to see the stresses at the desired place. Enter 'x' and 'y' coordinates to show the stresses at a point of your choice. The stresses you find in this window are based on  $N + M_{\gamma} + M_{z}$
- Compression is negative, tension is positive.

Click 'OK' to close this window.

### Step 22: Bending moments $M_{\gamma}$

Now we visualize the bending moments  $M_y$ . Click on  $\mathbb{N}^{\mathbb{N}}$  in the pallet and select the bending moment  $M_y$ . Choose the combination ULS FC envelope.

- In all combinations ULS FC and on each position of the beams, Diamonds will look for the minimum value of the bending moment. Those values are represented by the **thin line**.
- In all combinations ULS FC and on each position of the beams, Diamonds will look for the maximum value of the bending moment. Those values are represented by the **thick line**.
- Hence this image is called an 'envelope'.



The moment diagram is always displayed on the tension side of the element. The sign of the bending moment corresponds to direction of the local axes. In this case the local z'-axis is directed upwards and causes a positive moment thus tension on the upper side.

### Step 23: Reactions

Once back in the model window, we click on the button  $\[b]{\[mathebdotseline}\]$  in the pallet to show the reactions. All reactions are displayed separately by Diamonds. In this example we are interested in the vertical node reactions in the combination 'SLS RC': we select the support reactions for  $\[mathbb{R}\]_{R_{\gamma}}$ .



So far the overview of the functionalities in the 'Results' configuration. Now we can calculate the reinforcement and the cracked deformation.

## 5.1.6 Parameters for timber verification

### About dimensioning structures in timber

The internal forces in the structure are known and we wonder if the structure can take these forces? Checking this is done by a timber verification (see §5.1.7). This verification consists of two parts:

- Strength: is the structure strong enough to handle the internal forces?
- Stability: is the structure stable enough or will it buckle (laterally).

To perform the check for strength, you don't need to do anything extra. To perform the check for stability, you'll have to set some parameters for buckling and lateral torsional buckling.

### 5.1.6.1 Buckling

### Step 24: Taking the purlins into account

If you'd calculate the buckling lengths blindly, you'd notice that often the buckling lengths out of the plane are not realistic. This is because the position of the purlins could not be taken into account. A simple way to take them into account is prevent the movement out of the plane of a number of nodes. Therefore we will divide the rafters.

Active the 'Geometry' configuration  $\mathbb{B}$  and select the two longest rafters like indicated in the image below. Divide these two bar in 2 equal parts using this button  $\mathbb{A}$ .



No we will prevent the movement out of the plan for all nodes, except for the two the supports.

- Draw a selection window from left top to right bottom over the entire structure, except for the 2 supports.



Then click on  $\frac{1}{2}$  for defining supports and complete the window as below:

Image: Open state     Image: Open stat	HoutVB1.bsf - [Windows 1 (m)]
Eile Edit View Select Display Analysis Options Windows Help	_ <i>B</i> ×
	(施6)祭亩④隙ゴ   F■ゆ
😵 🗟 🚔 🛛 Windows 1 🔹 🚺 Geometry	
	Supports Active level
	Level 1
	2.7 m
A / 0	
	Evel manager
* * *	Ground level
夕 M 登	及 Drawing plane
∠T p t P+i	X = 0.0  m
ta ut	o tension Z = 0.0 m
Y displacement	Y rotation Representation
	Free 💌
「「 cannot resist 」 「 cannot resist 」 □ cannot resist 」	o compression
Z displacement	Z rotation
The	o tension
	o compression Font 20 🗲
	Symbols 15 👤
	jle relative to horizontal plane α: 0.00 ° Loads 10 €
🛛 🖓 🖓 🖓	Angle relative to vertical axis β :  0.00     °
	Show groups
Help	<u></u>
	-x

### Step 25: Setting groups for buckling

Next we will set the groups for buckling. See §4.1.6.1 for the method.





#### Step 26: Calculating the buckling lengths

Now calculate the buckling lengths *be*.



In each direction (round y'(u)- or z'(v)-axis) Diamonds asks you for which type of structure and for which type of analysis (first or second order) you would like to calculate the buckling lengths.

It is important that you use the same type of analysis as what you indicate here. We choose 'semi displaceable nodes'.

Click 'OK'.

### 5.1.6.2 Lateral torsional buckling

Step 27: Settings for lateral torsional buckling

The supports in the Z-direction are by default lateral torsional buckling supports, so that we don't have to do anything specific for lateral torsional buckling.

# 5.1.7 Timber verification

### About the timber verification

With the timber verification we check the strength and the resistance against buckling and lateral torsional buckling of the bars according to a certain standard.

The timber verification is always performed according to EN 1995-1-1, without a national annex.

Before starting the calculations we check the properties of the used timber quality.

Choose the menu command 'Edit – Material library' and select the material 'Timber C18' from the left column. Timber C18 is a default material. Standard materials are marked by #.

E+ II IIIIIIIIIIIIIIIIIIIIIIIIIIIIIIIIIIII

The properties of default materials are determined using the standards and can't be edited. Should you wish to make adjustments, you should make a new material. User defined materials are marked by the icon

On the right of this window you'll find:

- The mechanical properties like the strength for tension, compression, bending and shear parallel or perpendicular to the fiber direction.
- The thermal properties used in a fire analysis
- The strength properties



#### Step 28: Timber verification

To start the verification, select the menu command 'Analysis – Steel and timber design' or click on  $F_{2}$  or press **F3**.

4	ş		Diamonds - Ho	outVB1.bsf - [Windows 1 - δy -	Selfweight (mm)]		- 🗆 🗙
K	🥮 <u>F</u> ile <u>E</u> dit <u>V</u> iew	Select Display Analy	sis <u>O</u> ptions <u>W</u> indows <u>H</u>	elp			_ & ×
	D 🛩 📕 🖪	🖨 🚺 🗠 🗠 🗌 🗄	1 (FF (FF   🗰     🌆 🛛 🛛	A 🖩 X & & S 🖬 🖯 F%	11 平平平		
	🐺 🖾 🖷 🔀 🕅	ndows 1	Results	<b>•</b> 🖗 🖬 💹	🔤 🖉 🛤 🖕 🥙 🔍 🤇	2 🖾 🥒 🖻 🖻 🛛	
						max = 0	Active level
	FR Do Ro					0.1	
	AN DE YN					0.0 ← -	
				Code check	×	0.0 ←	A
			Action			0.0 ←	Level manager
	Selfweight		C Code check for s	elected elements only		0.0 <del>~</del> 0.0 <del>~</del>	Ground level
			Code check for a	ll elements		0.0 ↔	Drawing plane
	A. 57 s. A		Selection criteria			- 0.0 ←	Y = 2.7 m
			Switch combinati	on filter on		0.0 -	Representation
	15. 15 15 15		Global value	95 😂 % of maximum		-0.1	***
	2~x2∕0×2~×x ∲	Y	C Particular values			min = -(	0.1 🔊 🄊
	*						
		× ≼					Font 10 🜩
		-091					Symbols 10 🚖
	🧔 🍝		Help	<u>C</u> a	ncel <u>O</u> K	1	Loads 10 🚖
	1						Results 15 🚖
							Show groups
						N N	None
							-X 2000
F	1						1

Fill in the settings as above and click 'OK' to perform the verification.

#### About the window 'Steel / timber design'

- The box 'Action' you indicate whether you want to perform the verification on all bars or just the selected ones.
- In the box 'Selection criteria' you can specify whether Diamonds should run the verification for all combinations or only for the most determining combinations.

Once the calculation has ended, the button 2 will become active in the Results configuration 2. You'll see the following two icons:

- $\mathcal{K}$  for viewing the results of the check on strength
- $\mathcal{I}$  for viewing the results of the check on stability

Both results are expressed as a percentage of the maximum capacity. The maximum capacity equals 100%.



Results for strength (%)



Results for stability (%)

We see that the structure is sufficient for both strength and stability (= percentages larger than 100%).

To get more information about the calculations, we double click for example the rafter on the left in the Results configuration. Make sure you're looking at the results of the timber verification (22 or 2).



From this window we learn that both buckling and lateral torsional buckling determine the stability of the rafter. When we look at the results of the strength, the verification for bending is determining.

Adding more purlins will not solve a stability problem this big. The only solution is to choose a heavier cross section (= higher resisting moment) or a better timber quality (= higher allowable stresses).

## 5.1.8 Cross-section optimization

We now will use the optimization algorithm of Diamonds to find the most optimal cross-sections.

The optimization is based on the obtained percentages with the timber verification (see §5.1.7).

There are two optimization principles in Diamonds:

- The basic cross-section is provided in the **section library**. Diamonds will therefore search for the best suited cross-section in the library.
- The cross-section is built based on a **characteristic shape**. Diamonds will optimize by modifying either the height or width step by step defined by the user.

In this example the first method will be used, because the section we used come from the section library. The second method will be discussed in the next example when we calculate a 3D structure in timber (§5.2.8).

### Step 29: Cross-section optimization

To start the optimization, select the menu command 'Analysis – Optimization' or click on the button  $\checkmark$  in the icon bar.

If you had elements selected, Diamonds will ask if you want to perform the optimization for all the elements or for the selected elements only.



If this window pop up, select the option 'optimization for all elements'!

Then Diamonds will show you a window for entering the parameters for the optimization:

	0	ptimization			x					
Optimization parameters	Adjust cross-s	sections			1					
Optimisation target	s:	95 🔶 %								
Optimize for:		<ul><li>✓ Resistance</li><li>✓ Stability</li></ul>								
Order sections by:		Resistance	•		_					
Help	< Previous	Next >	<u>C</u> ancel	<u></u> K						
	0	ptimization			×					
Optimization parameters	Adjust cross-s	sections								
Adjust cross-sections					-					
Summarize and control	nfirm									
C Automatic adapta	tion									
How to replace cross-se	ections:				-					
🔽 Equal cross-sectio	ins remain equa	l after optimisation								
☐ All bars of same t	All bars of same type have the same section after optimisation									
Help	< Previous	Next >	<u>C</u> ancel	<u>O</u> K						

Fill in the settings like in the images here above.

When the optimization is completed, a dialog box appears with the summary of the optimization.

Optimisation re	eport	×
		_
		_
Number of bars to adjust: 0		
Number of bars not requiring any adjustment: 13		
	<u>C</u> ancel <u>O</u> K	

Diamonds proposes you to change no cross-sections. This is because the steel verification gave very good percentages (nearly 100%).

We don't make a report for this example. The report manager is explained in the second example in reinforcement (§3.2.8). The principle is the same.

# 5.2 Example 2: 3D hall

Required licenses:

- ✓ 2D Bars✓ 3D Bars
- ✓ Timber Design

### **5.2.1** Purpose of the exercise

We will now calculate a 3D structure in timber. This is a sketch of the structure we will calculate:



The timber quality is GL32h.

## 5.2.2 Defining the structure

### Step 1: Go to the 'Geometry' configuration

Defining the structure is always done in the 'Geometry' configuration. Click on B in the icon bar, or select the 'Geometry' configuration in the adjacent pull down menu.

49													Dia	mon	ds -	[Vens	ster 1	(m)	l									- 🗆 🗙
🤗 <u>F</u> ile	<u>E</u> dit	View	/ <u>S</u> e	elect	<u>D</u> ispla	ay 🧍	Analysi	s <u>O</u>	ptions	Win	dows	Help	p															- 8 ×
	6		. 🖶	$\square$	6	$\sim$	E	H	$\square$	#	Tii			l He	6	de:	111	5) F%	Iý	IF		$\langle \psi \downarrow \rangle$						
× 8	5 🖷 1	× Ve	enster	r 1			-	1	Geo	ometry						+	ß	4 🔯		南	声	877	Q	Q 🔯	1	DR		
1 k		$\times$				_																						Active level
<b>`</b> р		<u>,</u>		- 20	10	- 22	52 	- 23	8	25	81	8	8	8 			8		8	**	12 	83 	- 23	- 24	25 - 25 		88 	foundation 💌
ZZ		/	10	1		*	X	20	24	26	1	8	14	14	8	- 22	2	12	2	10	10	45	20	10	84 B	1 81	- 24	0.00 m
D	0	0	22	20	*	28	38	57	12	12	84	82	8	2	22	12	12	12	2	28	28	28	57	277	85 X	- 81	8	61-12 0.00 m
2			10	62	2	28	8	10	81	82	8	33	12	82	8	2	8	8	10	10	22	38	10	61	8 8	8	8	
	~	-	10	5	18	22	63	13	81	8	8	38	3	3	2	82	22	68	10	10	22	88		+				Level manager
1	a'a'a -5		20	35	-	-	-33	-33	84	82	12	12	24	14	s	8	2	12	35	26	-	-		J.	. 1	op view		Ground level
12	14141	£																					_	24 18	_		_	Drawing plane
7	вĴŦ	2.5																					1		F	ront viev	v	X = 0.00 m Y = 0.00 m
Ň.	,K )	R.	23	8	82	82	-55	-83	20	10	25	12	8	8	8	8	2		88	23	82	80	1	z.L.X			-	Z = 0.00 m
07	6-2 6	11	83	8	82	80	80	43	24	28	93	98	8	8	8	18	8	35	8	23	80	10			2			Representation
至0	She k	¥	83	8	75	35	63	12	8	39	őł.	10	8	8	35	35	8	32	8	88	10	10		P	. B	ack viev	'	
Ħ	85	e)	83	8	85	51	10	10	21	39	<i>8</i> 2	12	8	8	ж	8	×	8	8	83	35	10		2 X				S 🔊
₩¢	1 <	2	30	32	0	6	11	12	19	12	.0	13	12	36	8	82	28	98	32	32	0	82		- V	L	eft view	-	Size
		Te								~														Z.L.X				Font 25 🚖
																												Symbols 15 🚖
ettai)			12	61	25	25	63	12	87	87 1	32	18	12	10	85	35	8	0	20	12	25	25		1 E	s R	ight viev	v	Loads 15 🚖
			63	22	8	25	12	84	85	15	82	82	12	85	\$	8	8	8	52	62	25	8		2***X				Results 15 📚
			88	-8	28	53	53	53	8	85	82	82	8	8	35	33	2	22	8	10	52	28	- P	_ <b>3</b> [	3	D view	1	Show groups
			22	18	23	53	53	53	82	13	82	8	8	2	33	25	2	22	8	8	52	23		z.L.X				None 💌
			22	8	18	8	10	20	85	85	84	82	8	8	8	8	12	82	8	17	8	18	8	8	84 8	. 8	r× z	
)																												

Then check if you are in a front view. If this is not the case, then click on

the button  $\blacksquare$  in the icon bar or on the button  $z^{\perp}x$  in the lower right corner and select the viewpoint 'Front view'. This way you activate a vertical drawing area.

### Step 2: Structure generator

Click on the icon in the pallet. A dialog window will appear in which you can select the form of the structure you would like to generate. Opt for a frame in the figure below.

\$	Diamonds - [Venster 1 (m)]	- 🗆 🗙
🥮 Eile Edit View Select Display Analysis Opti	ons <u>W</u> indows <u>H</u> elp	- 8 ×
┃ □ ☞ ■   □ ● □   い ~   日田日	╡╫║┺╔┇╆╺╎╩╦╚╚ᢧ┙│╞┣ゅ	
😵 🗟 🖷 🛛 Venster 1 💽 📘	Geometry 💽 📾 🖂 🕅 🖽 🦑 🗨 🤅	R 10   🖉 🖪 R 🛛 🖉 🗐 🗔 🗧
		Active level
<u>∩</u> ⊞. ∕		
		年月10.00 m
000 z		
	Winned many ? X	Evel manager
	wizard menu	
<b>夜麗</b> 理		to Drawing plane
		$\frac{y_{\text{Exc}} \text{ brawing plane}}{X = 0.00 \text{ m}}$
T' 8, J 1 '.		Y = 0.00 m Z = 0.00 m
<u>Nil 2/1 7×</u>		Representation
		BBB
m e <sup>s</sup> A		â 🖍
The second se		
		Size
		Sumbole 15
		loade 15
		Results 15 A
		Show groups
		None 💌

After you clicked 'OK', Diamonds will ask you the geometric data of the frame. Enter the details as shown below.

⇔ Wi:	zard: Portal frame 🛛 🗖 🗙
	Beams       9,5       °         Number of intermediary nodes (left): $0 \Leftrightarrow$ $0 \Leftrightarrow$ Number of intermediary nodes (right): $0 \oint$ Width portal frame:       12,00 m         Beam section       150/300600 - GL32h         Columns       Number of intermediary nodes (left): $0 \oiint$ Left column height:       4,00 m         Number of intermediary nodes (right): $0 \oiint$ Right column height:       4,00 m         Column section       150/300600 - GL32h         Left boundary condition       (° Support         C Support       (° Support         C Restraint       (° Support         Cordinates of first point:       X = 0 m         X = 0 m       x = 0 m         X = 0 m       x = 0 m         X = 0 m       x = 0 m         X = 0 m       x = 0 m         X = 0 m       x = 0 m         X = 0 m       x = 0 m         X = 0 m       x = 0 m         X = 0 m       x = 0 m
	Cancel OK

The beams and columns all have the same tapped cross-section. To change the cross section of the bars:

- Click on Beam section Or Column section.
- Copy the parameters from the image below.

	Section
Cross-section       Name     150/300600       Shape     Variable	└── values related to physical member
Dimensions section 1	Section Cross-section Name 150/300600 Shape Values related to physical member Dimensions section 2 B 150 mm
Automatic calculation         General       Elastic       Plastic         strong axis v-v       -       v         Sy       6750000       mm³         Iy'       337500000       mm4         Iy'       86,6       mm         Wel,y',t       2250000       mm³       Wel,         Wel,y',b       2250000       mm³       Wel,	H   600        mm         F       Automatic calculation         General       Elastic         Plastic       Imma         Sy   27000000       mm <sup>3</sup> Sy   27000000       mm <sup>3</sup> Sy   27000000       mm <sup>4</sup> Iy       27000000         General       Iyz   0         mm4       Iyz   0         Iyz   16875000       mm4         Iyz   16875000       mm4
Material <u>Material</u> Timber GL32h Cocar axes Orientation angle 0,0 <u>Help</u>	Image: Second Section S

Then click 'OK' to confirm the selection. After you have completed the preceding parameters, you receive this image:



### Step 3: Change orientation of the beam

The cross-section of the left rafter has the wrong orientation. To correct this, select this rafter and click on  $\frac{1}{20}$ :



### Step 4: Placing the columns inclined

Now we only have to place the columns inclined. Double click the upper endpoint of the left column and increase the x-coordinate with 1m.

\$	Diamonds - [Venster 1 (m)]	- • ×
💝 Eile Edit View Select Display Analysis Option	ns <u>W</u> indows <u>H</u> elp	_ 8 ×
□☞∎◘霉◙■∽∝│⊞⊞⊞	🔠   🖬 🖾 📾 🕱 🦗 👌 🕸 📅 🕀 🕬 🖬 🖉	
🐺 🖻 🖷 🔀 Venster 1 💽 🔳 G	ieometry 🗾 🖍 🖽 🖾 🛃 🛱 🥵	Q Q 🗱 🖉 🖻 R 🕾 🛛 🗰 🗛 👘
		Active level Level 1 v 2.7 m 2.7 m 2.7 m 2.7 m Count level
	Coordinates     m $X = 100$ m $Y = 4.00$ m $Z = 0.00$ m       Footing     Image: Coordinate of the second se	Size Font 11 ± Symbols 15 ± Loads 10 ± Nione

Do the same for the right column but reduce the x-coordinate with 1m. Result:



### Step 5: Copy the structure to a 3D model

Select the rafter and divide them in 3 equal parts 🐓.

Diamonds - [Venster 1 (m)]	- 🗆 🗙
🥮 Eile Edit View Select Display Analysis Options Windows Help	- 8 ×
◎ 御司 (1111) [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1111] [1	
😨 巻 🕾 🛛 Venster 1 🔍 🚺 User configuration 🔽 🍘 🗔 😡 🗐 🗔 📈 (本) 代 包 色 図 🥒	
$\mathbf{k}$ $\mathbf{X}$	Active level
OH · /	
	₽ 2.7 m
	Evel manager
	Ground level
夕 · ··································	白 Drawing plane
₩ <sub>B</sub> 5 PH	X = 0.0 m Y = 2.7 m
NTA I Con Y	Z = 0.0 m
	Representation
io 🖉	10 10 10 10 10 10 10 10 10 10 10 10 10 1
	D 🔊
🕹 🖌 🔗 🛛 Divide line 🗡	Size
	Font 11
Number of segments 3	Symbols 15 🚖
Equal segments	Loads 10 🚖
	Results 15 🚖
Help Cancel OK	Show groups
	None 👤
	X AND AND A
	10

Diamonds will automatically adjust the height of the different parts of the beams.

Now select the entire structure (use a selection window or press CTRL +A) and click on  $\checkmark$ .

<b>\$</b>	Diamonds - [Venster 1 (m)]	- 🗆 🗙
💝 Eile Edit View Select Display Analy	iis <u>O</u> ptions <u>W</u> indows <u>H</u> elp	_ 5 ×
D 🛎 🖬 🖪 🚳 🕅 🗠 🗠 🛛 A	∄Ħ用 Ⅲ   ℡   図 ■ ≵ ≵ ♂ ♡ ♡ ♡ ♡ ↓   ⊨ ↓	
🐺 🔄 🖷 🔀 Venster 1 💌	1 User configuration ・ 瞬 中 🛛 🚽 🚈 🖓 🔍 🗶 🚺 🥒 💽 🗷 🖓	
	$\begin{tabular}{ c c c c c } \hline \hline \ $	Active level         Level 1         1         2.7 m         2.7 m         Evel manager         Cound level         Drawing plane         X = 0.0 m         X = 0.0 m         X = 0.0 m         Representation         Drawing plane         Size         Font       11 •         Symbols       15 •         Loads       10 •         Results       15 •         Show groups       None
	Lx	

Complete the window like here above and click 'OK'.

### About the 'Translation' function

- In the field 'N' you enter the amount of copies you want. When just want a translation (= move something), 'N' should remain equal to 0.
- In the three fields below you enter the translation (or copy) vector.
- When you finally check the boxes 'create connecting lines' or 'create connecting plates', Diamonds will automatically draw lines or plates between the copied items.

In order to see what Diamonds has made of it, you have to view the project in perspective. Click  $\square$  on the right bottom and choose a 3D view.

At the bottom and on the right scroll bars allow you to change the viewpoint. You can also use the button 🖾 or **F12** to see the entire structure on the window.



### Step 6: Cross-section of the purlins

The cross-section of the purlins has not yet been defined:

- Select the purlins with:

Sele	ct	Display	Analysis	Opti	ons	Wir	ndows	H
	AI	I	Ctrl+A	ĒĦ	1   I		Ţ	Į۵
	Po	oint Numb	oer		Geon	netry		-
	Li	ne Numbe	er					
	Ve	ertical Line	:	ι.				
	Н	orizontal L	ine	ι.				
	In	clined Line	e					_
	Cı	ross-Sectio	on 🔸		150/	3006	00	
	Su	urface Nur	nber		?			
	Ve	ertical Surf	ace	Г				_
	Н	orizontal S	ourface	ι.				
	In	clined Sur	face	ι.				
	PI	ate Sectio	n 🕨	L .				
	м	aterial	+					
	м	lost loaded	bars					

This way you select all bars without a cross-section.

- Assign a rectangular section 80x180mm to them. Name the section '80/180'. The used material is GL32h.

If you always define the cross-sections at each step in the project, then the last drawn bars are always without profiles. Now it becomes very easy to select them. We also could have chosen 'Horizontal lines'.

### **Step 7: Turning the purlins**

The purlins have (by default) a vertical orientation in the program. However, in practice the purlins follow the slope of the support beams. If we now double click a support beam, we see that the slope is 11,3°. We now will give the purlins the same slope. The purlin at the ridge will keep its orientation.

- Select all purlins on the right side like in the figure below.
- Then click on the button in the pallet and complete the dialog box as follows:



Do the same for the purlins on the left side but enter  $11,4^{\circ}$  (instead of  $-11,4^{\circ}$ ).

We could also use and eccentricity for the purlins but that's not considered in this example.

### Step 8: Setting the purlins hinged

We now define hinges at the ends of the purlins. We assume that the ends of the purlins can't take a bending moment of any importance.

- Select all the purlins using the menu command 'Select – Section – 80/180'.



- Next click on the button and complete the window as follows:

The value of 500kNm/rad allows to take a certain degree of rigidity of the connection into account.



### Step 9: Adding a bracing system

To complete the model, we add a bracing system to ensure the stability of the structure. We use metal bars with a diameter of 20mm in steel S235.

We first draw the bracing system. This is possible in a 3D view.

Click on the button it to activate the drawing function. Then draw the bracing system using the intelligent cursor.



Select the just drawn line using the menu command 'Select – Section - ?' and click on . Complete the window as follows:

Cross-section			S	ection			
ame Windver	band	<u> </u>	L				
ihape 💽 🗸	Constant	•					
Dimensions				1	Dimonsions	Aver	2
н 20.	m 0	m		1	Jinensions	nine -	
Section propertia	es						x
					Mat	erial depe	ndent 🥅
Automatic	c calculation						
☞ Automatio	Elastic P	lastic │ Co	old-formed membe <u>weak a</u>	rs   <u>dis z-z</u> - <b>}</b> -			
General	Elastic P Elastic P Strong axis y	lastic   Co - <u>y</u> - <u>T</u>	old-formed membe weak a 57 [3130	rs   ds z-z - <b>}</b>	aloba	0.00	ē
Cross-section Name Windve Shape Dimensions H 20 Section properti General Section properti General Sy Hy Wel,y',t Wel,y',t Wel,y',t Udates	Elastic P Flastic P Strong axis y 3130 7854	lastic   Co	old-formed member weak a Sz 3130 12' 785'	rs   ds z-z	alpha Iyz	0.00	° mm*
General    Seneral    Sy Iy'	Elastic P Strong axis y 3130 7854 5.0	lastic   Co	old-formed member <u>weak a</u> sz 3130 12' 785- 12' 5.0	rs   ds.z-z -	alpha Iyz It	0.00	° mm" mm"
General	Elastic P Strong axis y 3130 7854 5.0 788	lastic   Co 	bld-formed member weak a Sz 3134 12' 785- 12' 5.0 Wel <sub>2</sub> 2',1 788	rs   dis z - z - }	alpha Iyz It Iw	0.00 0 15708 0	e mm <sup>4</sup> mm <sup>4</sup>
General    General    Sy Iy' Iy' Wel,y',t Wel,y',t	Elastic P strong axis y 7854 5.0 788 788	lastic Co 2 - T - mm <sup>3</sup> mm mm mm <sup>3</sup> mm <sup>3</sup>	0ld-formed member weak a Sz 313( 12' 785- iz' 5.0 Wel,z',1 788 Wel,z',7 788	rs   ds z-z -	alpha Iyz It Iw Twm	0.00 0 15708 0 1571	e mm <sup>a</sup> mm <sup>a</sup> mm <sup>a</sup>
Automatic General	c calculation Elastic P P atrong axis y 3130 7854 5.0 788 788 788	lastic   Co y - I - mm <sup>3</sup> mm <sup>4</sup> mm mm <sup>3</sup> mm <sup>3</sup>	0d-formed member weak a sr 3133 fr/ 785- it/ 5.0 Wel,z',/ 788 Wel,z',r 788	rs   dis 2-2 mm <sup>3</sup> mm <sup>4</sup> mm mm mm <sup>3</sup> mm <sup>3</sup>	alpha Iyz It Iw Twm	0.00 0 15708 0 1571	e mm" mm" mm" mm"
Automatic General Sy IV IV Wel,V,r Wel,V,r Atterial Ste	c calculation Elastic P P strong axis y 3130 7854 5.0 7854 785 785 785 785 785 785 785 785 785	lastic Co y I mm <sup>3</sup> mm <sup>3</sup> mm <sup>3</sup> mm <sup>3</sup>	bld-formed member weak a Sz 3132 12' 785- 12' 5.0 Wel,z',J 788 Wel,z',r 788 Wel,z',r 788	rs   ds 2-2 mm <sup>3</sup> mm <sup>4</sup> mm <sup>3</sup> mm <sup>3</sup> mm <sup>3</sup>	alpha Iyz It Iw Twm	0.00 0 15708 0 1571	e mm <sup>a</sup> mm <sup>a</sup> mm <sup>1</sup> mm <sup>2</sup>
Automatic General	c calculation Elastic   P strong axis y [3130 [7854 [5.0 [788 [788 [788] [788] [788]	lastic   Co y I	bld-formed member weak a Sz 3130 12' 785- 12' 5.0 Wel,z',J 788 Wel,z',J 788 Wel,z',J 788	rs   ds z-z mm mm mm <sup>3</sup> mm <sup>3</sup> mm <sup>3</sup>	alpha Iyz It Iw Twm	0.00 0 15708 0 1571	e mm <sup>s</sup> mm <sup>1</sup> mm <sup>3</sup>
Automatic     General        Sy     IV     IV     Wel,v,t     Wel,v,t     Wel,v,t     tatenal     fatenal     See     caal axes     Xrientation angl	c calculation Elastic P Strong axis y 3130 7854 5.0 788 788 788 el \$235 e 0.00	lastic   Co x - I - mm <sup>3</sup> mm <sup>4</sup> mm <sup>3</sup> mm <sup>3</sup>	bld-formed member <u>weak a</u> Sz 3130 12' 785- 12' 5.0 ₩el,z',i 788 ₩el,z',i 788 ₩el,z',i 788	rs   ds z-z mm <sup>3</sup> mm <sup>4</sup> mm <sup>3</sup> mm <sup>3</sup> d	alpha Iyz It Iw Twm	0.00 0 15708 0 1571	r mm* mm* mm*

Finally, the rods must be defined as tie rods. This means that the rods can't take compression. Select the bars and click on , choose the option 'tie rod'.

The geometry of the structure is now complete.



# 5.2.3 Defining the loads

### Step 10: Go to the 'Loads' configuration

We now leave the 'Geometry' configuration and activate the 'Loads' configuration to enter the loads. Click on the button <sup>III</sup> in the icon bar or select in the adjacent pull down menu the 'Loads' configuration.

### 5.2.3.1 Creating the load groups

### Step 11: Creating load groups

We consider 5 load groups:

- Self-weight
- Dead load
- Life load H: Roofs
- Wind: 20 cases
  - Wind left upward -> right upward ( $c_{pi} = -0.3$ )
  - Wind left upward -> right downward ( $c_{pi} = -0.3$ )
  - Wind left downward -> right upward ( $c_{pi} = -0.3$ )
  - Wind left downward -> right downward ( $c_{pi} = -0.3$ )
  - Wind left upward -> right upward ( $c_{pi} = 0,2$ )
  - Wind left upward -> right downward ( $c_{ni} = 0,2$ )
  - Wind left downward -> right upward ( $c_{pi} = 0,2$ )
  - Wind left downward -> right downward ( $c_{ni} = 0,2$ )
  - Wind right upward -> left upward ( $c_{pi} = -0.3$ )
  - Wind right upward -> left downward ( $c_{pi} = -0.3$ )
  - Wind right downward -> left upward ( $c_{pi} = -0.3$ )
  - Wind right downward -> left downward ( $c_{pi} = -0.3$ )
  - Wind right upward -> left upward ( $c_{pi} = 0,2$ )
  - Wind right upward -> left downward ( $c_{pi} = 0,2$ )
  - Wind right downward -> left upward ( $c_{pi} = 0,2$ )
  - Wind right downward -> left downward ( $c_{pi} = 0,2$ )
  - Wind front -> back upward ( $c_{pi} = -0.3$ )
  - Wind front -> back upward ( $c_{pi} = 0,2$ )
  - Wind back -> front upward ( $c_{pi} = -0.3$ )
  - Wind back -> front upward ( $c_{pi} = 0,2$ )
- Snow: 3 cases
  - o Case 1
  - o Case 2
  - o Case 3

Follow the procedure in §5.1.3.1 to define them.

### Step 12: Defining linked load groups

The load groups 'self-weight' and 'dead loads' both contain vertical forces with a downward direction. Thus both load groups have the same effect on the structure. Consequently the most extreme (min and max) values for the internal forces are obtained when both load groups are multiplied by the same minimum/ maximum safety coefficients.

In Diamonds it's possible to define this kind of behaviour using 'Linked load groups':

- Click on the button Linked load groups at the bottom of this window.
- Indicate that the load group 'Self-weight' is linked to the load group 'Dead loads'.

						Design lifetime	50 文 yea	rs		1	<b>u</b>			
	Add load group		Insert lo	ad group	]	Delete load	group				Several	load cases	per group	F
	Name load group	#	Туре	Name load ca	se $\gamma_{uls}$ $\gamma_{uls+}$	$\gamma_{\rm sls-}$ $\gamma_{\rm sls+}$ $\psi_{\rm c}$	$\psi_1 \psi_2$	φξ	to	Combina cracking	kmod	Load	Action	
/	Selfweight	1			<b>~</b>	Linked load	groups –		5 0		permanent	-	+++	1
/	dead loads	1			Selfweight				5 0		permanent	1.00	ŦŦŦ	
/	life loads H : roofs	1			Linked with				0		medium term	-	<u>+++</u>	
~	wind	20	ł£	Geval 1					0 0		short term	pan.	+++	
				Geval 2	dead load	s								
				Geval 3		1: roots								
				Geval 4	snow (H <	<= 1000 m)								
				Geval 5										
				Geval 6	De	lete all links for the s	elected load grou	p						
				Geval 7		Delete all	links							
				Geval 8	Help	1	Cancel	ок						
				Geval 9		_								
				Geval 10										
													>	e,

Using 'Linked load groups' will reduce the amount of load combinations and thus also the calculation time (especially noticeable on large models).

Next click twice on 'OK' to close these windows.

### 5.2.3.2 Filling up the load groups

Step 13: Filling in the load groups 'Self-weight', 'Dead loads' and 'Life load'

- The **self-weight** is calculated automatically by Diamonds.
- A dead load of 0,3kN/m<sup>2</sup> is applied to the rafters.
  - o Select the load case 'dead loads' from the pull down menu.
  - Select all rafters on one side of the roof like on the image below.
     The function <sup>2</sup>/<sub>2</sub> we will be using, only works with bars that lay in the same 2D plane.
  - Click on the button <sup>₱</sup>/<sub>₱</sub>.
  - Complete the dialog box like here below:



When you click 'OK', Diamonds will calculate the loads on each purlin.

o Repeat the same steps for the selection on the other side.

Result:



- Repeat the steps for 'Life load H: Roofs' of 0,4kN/m<sup>2</sup> on the same rafters.



## Step 14: Filling in the load group 'Wind'

## To generate **wind**:

- Click on the button Ground level and set the ground level to 0m.

🥹 Diamonds - Hou	tVB2.bsf - [Venster 1 - Wind - Geval 1 (kN, kNm, mm, kN/m, kNm/m, kN/m <sup>2</sup> , °C)]	- 🗆 🗙
💝 Eile Edit View Select Display Analysis Optio	ns <u>W</u> indows <u>H</u> elp	_ & ×
□ ☞ ■ 🖪 ● 🖣 🗠 ∾ ⊂   日田田	🏭 🛛 🌆 🦹 独 👌 黎 西 ④ 豫 ゴ 🗍 💷 🖉 🛄	
🗑 🗟 🖄 Venster 1 💽 🔳	oads 🔹 🚱 🖻 🕅 🗟 👘 🖽 🖑 🗨 🕲 🖉 💽 🕅 🗺	
γ <sup>y</sup> <sub>u</sub> <u>mi</u> . →	-	Active level
Wind Geval 1 V		2.7 m
₩. + <sup>7</sup>	Ground level ×	2.7 m
4 20 22	Position	Evel manager
$\varphi \gg \varphi$	Y Y	Ground level
	· · ↑ · · · · · · · · · · · · · · · · ·	$\frac{y = 0 \text{ awing plane}}{X = 0.0 \text{ m}}$
* Temperature		Y = 2.7 m Z = 0.0 m
× Snow		Representation
× wind	Y = 0.00 m	
PH & A		<b>A</b>
Seismic     Dynamic		Size
* Moving	Help Cancel OK	Sumbole 15
		Joada 10
	Ψ	Results 15 1
		Show groups
		None
	Ϋ́×	
	<b>, , , , , , , , , ,</b>	1 35-4
		11.

- Choose the load group 'wind' and the first sub load case 'wind I up r down (-0.3)' from the pull down menu.
- Click on 🎦 to define the wind standard and the terrain parameters.

<del>9</del>	Terrain parameters ×	- • ×
🥮 <u>F</u> ile <u>E</u> dit <u>V</u> iew <u>S</u> elect <u>D</u> i		_ 8 ×
D 📽 📕 🖪 🚳 🚺		
😵 🗟 🖷 🔀 Venster 1	Norm: EN 1991-1-4 💌	
y y ant O		Active level
	Reduction with factor 0.85 for lack of correlation	Verdieping 1
Geval 1		2.70 m
	Structural factor CsCd: 1	2.70 m
Mr T		KHN.
the state of the	Direction factor Cdir: 1.0	Evel manager
the second se	Season factor Coason	Ground level
$\pm 22$		🔔 Drawing plane
N. B		X = 0.00  m Y = 2.70  m Z = 0.00  m
* remperature * Snow	Air density:	Representation
L Wind	Probability factor Cprob: 1.0	
	C V <sub>b.0</sub> by location: Map	
MB A		
Seismic     Dynamic	$C_{in} = wind pressure; 10.42 kN/m^2$	Size
* Moving	B minit product print within	
	Terrain Type:	Symbols 20 😴
		Loads 10
		Results 15
	Orography coefficient	Show groups
		None 💌
	Help Cancel OK	Lx see
		1

- Select the standard EN 1991-1-4 [--].
- Opt for a basic wind velocity of 26m/s and a terrain type I.
- Click 'OK' to close this window.
- Take a top view and select the first frame of the 3D hall.

-	🤪 Diamonds - HoutVB2.bsf - [Venster 1 - Wind - Geval 1 (kN, kNm, mm, kN/m, kNm/m, kN/m², °C)] – 🗖 🗙																																							
	🦻 <u>F</u> ile	<u>E</u> dit	<u>V</u> iew	<u>S</u>	elect	t <u>D</u>	<u>)</u> ispl	ay	Ana	lysi	5 <u>C</u>	ptio	ns	Win	dows	Н	elp																							- 8 ×
		-	D.	6	Ø		5	0		Ħ	Ħ	Ħ		:	Ter	X	8		ίř	e.	0	Ġ.	111	91	· %	¢,		= 0	<b>;</b> 1	¢										
Î	¥ 6		Ve	nste	r 1	111			+	11	1	1	oads	14	1840	1						-	B	1	0	1	Ma	¢		ery.	Q	Q (	3	1			I I	9 12		
Ť	v	ni (	3							1014		00.055										51,05		-112				<u>61</u>	- 11		- 5%	- 100	-	- 27	1			1	Active lev	el
	$\lambda^{2,n}$	tim ~	7	-	1	8	8	8	39	33	18	3		-2	-23	-23	-2	89	84	89	89	83. 1	68	iit.	8	8	88	88	3		3	10	65	10	8	8	88	83	Verdieping	g 1 💌
	Wind		-	8	15	18	32	32	22		8	8	1		-11	-12	- 11	88	82	83	88	155	155	8	100	100	88	55	8	8	8	28	98	18	15	32	22	22	frit :	2.7 m
	Geval :	1	-	8	98	18	22	32 22	32	30	8	8	1	- 5	-58	- 58 - 63	- 13	88	82	88	12	35	-55	3	10	10	14	55	80	8	83	28	98	98	15	32	22	20	AHP-	
	1.00	жī	1	8	98	95	22	<u>N</u>	32	30	8	8		1	T	1		1		1			1			Ĩ		55	88	25	20	98	98	98	95	32	32	30		2.7 m
	Trig	τ.		8	8	8	22	20	121	10	28	8	: 2		,		:4	8	83	33		38	100		10		13	10	30	8	85	88	8	8	6	10	121	10		manager
	+	و مر	3.	8	8	8	<u>82</u>	<u>\$</u> 2	63	10	18	2	5 23	1	12		14	8	25	83		32	22	10	12		101	10	3	35	85	65	8	8	8	£3	123	10	HHH LEVE	manager
	à	· · · ·	2	8	æ	8	22	22	62	<u>63</u>	23	8	: 2	: :2	22		**	8	S.	8			8	10	22		10	59	8	8	89	8	8	8	8	23	123	100	Grou	und level
	প	~ 2		8	8	æ	22	22	62	63	28	3	5 8	: 2	1.5	Î							1			Î	511	19	8	8	3	8	8	8	8	22	62	2	🕒 Draw	ing plane
		월 2	7	22	82	82 2	-3	10	18	10	20	22	3 8	3	- 33		33	8	S.	8	1	88	8	85	8		8	8	35	33	35	82	20	85	82	20	23		X = 0	.0 m
	✓ Ter	mperature		22	82	82	-3	23	18	28	<u>\$2</u>	2	3 8	3	53		33	8	85	85	1	88	8	8	8		8	8	33	33	35	22	82	82	82 2	20	28		Y = 2 Z = 0	./m .0m
	*	Snow		52 13	20 55	改 (5)	8	100	- 10		20 20	- 20	3 8		13		12	8	81	18	1	10	1	10	88			8	20	25 22	20 20	- 25 55	22 33	25 55	20 30	28 28	28 28	807 100	Represent	tation
	*	Wind													100			2	10		-1	12				1													AR A	A A
							a			- 52	10											28	12	- 22			88	- 22											4	$V \leq V$
	200	GA A	0		33																		12					~											1 - AB	Þ
	1 I + 9	Seismic																								-												10	Cita	
	+ D	ynamic													T	1			21	2	1				1		1					- 68							Font	11 🚖
	7 1	Moving			æ	a.		- 25	10	- 10						/		5					1				V.	25											Sumbole	15
				a.	a.	a		83		100								2				38	X	5	-	1	1	88				6				83				
							25	25	26	20					.]						2				1	1		12	a		G	12			10	25	26		Loads	
				~ 0			25	25	-	20						13	23	14	-	84	8			12	10			10	G		W		10		12		25		Results	15 🚖
							25	- 25	25	20					23	23		10	11	10	10		12	12	76	70	76	76				12				- 25	25	-	Show grou	ps
					a.	ě.	2	-			10				10				24	24	24											48			÷.	- 204			None	*
				8	8	×	8	8	88	83	87	8		- 6	-63	-85	-8	34	8	24	34	18	9	8	8	8	8	8	2	2	æ	35	34	8	R	8	Z	-x		ar.
																																								//

# - Click on 22 to start the wind generator on frames.









- Complete the window as above. Then click 'OK' to generate the wind on the first frame.
- Now select the second frame and click on <sup>11</sup>. The only thing you have to change is the position of the frame. Diamonds will remember all the other parameters from the wind generation on the first frame.

🤪 Diamonds - HoutVB2.bsf	🍄 Wind Generator for 2D frames – 🗖 🗾 🔀
File Edit View Select Display Analysis Options Wind	
│D ☞ ■│Q. @ Ŋ│ ∽ ~ │ ⊞ ⊞ ⊞│	Bearing elements for loads in plane Bearing elements for loads out of plane
🐼 🖻 🖷 🛛 Venster 1 📃 🚺 Loads	
γ <sub>g</sub> <sup>γ</sup> u <u>mi</u> ⊗	
Wind	
Geval 1	
K 47	1
+ +	
	Geometry Total width of construction: 12.00 m Position of frame:
A A A A A A A A A A A A A A A A A A A	Total depth of construction: 16,00 m Distance to frame in front: 4,00 m
▼ 9 9 5 27 (5 5 9 (5 7 7 8 9 7 8 7 9 7 8 9 7 8 9 7 8 7 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7 8 9 7	Distance to frame behind: 4,00 m
× Snow	left -> right         right -> left         front -> back         back -> front
	IV Cpi 0,30 ▼ 🗐 VACK
	✓     left upward - right upward         ✓     wind          ✓     wind lu -> ru (· ▼)
✓ Seismic	✓     left upward - right downward     wind     ▼       ✓     wind     ▼     wind lu -> r d (· ▼)
* Dynamic	✓     left downward - right upward       wind     ▼       wind l d -> r u ( ▼
* Moving	Image: Wind I d → r d (· m)     Image: Wind I d → r d (· m)
	I I I I I I I I I I I I I I I I I I I
	🔽 left upward - right upward wind 🔻 wind lu -> r u ( 💌
	✓ left upward - right downward wind
* * * * * * * * * * * * * * * * *	✓ left downward - right upward wind ✓ wind I d -> u (0, ✓
(1) (1) (1) (1) (1) (1) (1) (1) (1) (1)	✓     left downward - right downward       wind     ▼       wind I d -> r d ((▼   FRONT
	<u>H</u> elp <u>Q</u> K

- Repeat these steps for the all the other frames.
- Result:



## Step 15: Filling in the load group 'snow'

To generate **snow**:

- Select the load group 'Snow' and the first sub load case 'Case 1' from the pull down menu.
- Click on \* to select the snow standard and the terrain parameters.

<del>9</del>	Diamonds - HoutVB2.bsf - [Venster 1 - snow (H <= 1000 m) - Geval 1 (kN, kNm, mm, kN/m, kNm/m, kN/m <sup>2</sup> , °C)] – 🗆 💌
🥮 Eile Edit View S	jelect Display Analysis Options Windows Help
0 🛩 🖬 🖪 🤀	● ◎ ● 田田田 Ⅲ ■ 2 2 2 8 回 2 2 2 8 回 9 2 2 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1
🗑 🐻 👼 🔀 Venste	r 1 🔽 Loods 🔍 藤戸図 🗟 🕸 中 🕅 🔁 🕅 Q Q 🔯 🖉 🔟 🖾 🗄
γ <sup>γ</sup> <sub>ų</sub> <u>im</u> .⇔	Active level
snow (H <= 10( ▼	Settings snow generator
Geval 1 💌	
$\mathcal{M} \not \sim$	Norm: N 1991-1-3
	Area:
$\neq \gg \gg$	Altitude : 0.00 m
19- M	Snow load at ground level (Sk): 0.4 kN/m <sup>2</sup>
▼ Temperature Snow	☐ Return period adjustment V = 1.0 : Sn/Sk = 1.0
** ** 0 /	Ce: Exposure coefficient
Wind     Seismic     Dynamic	Ct: thermal coefficient
* Moving	Take into account snow weight on overhang at edge
	Help Cancel OK T
-	v None v
न	XZ 22 are

- Select the standard EN 1991-1-3 [--].
- Choose as country [--] 'Central West' 'Area 3'.

Take a top view and select the first frame of the 3D hall. Diamonds - HoutVB2.bsf - [Venster 1 - snow (H <= 1000 m) - Geval 1 (kN, kNm, mm, kN/m, kN/m, kN/m<sup>2</sup>, °C)] □ × 🍄 Eile Edit View Select Display Analysis Optio ns <u>W</u>indows Help \_ 8 × 🗑 🗟 😤 🔀 Venster 1 Loads I B H B d A L M Q Q B Ø tive level γ<sup>γ</sup><sub>g</sub> <u>im</u> ⊕ Verdieping 1 💌 snow (H <= 10( ▼ 2.7 m Geval 1 -2.7 m 紧护 Evel manage + + 5 Ground level € 2 ge Drawing plane 1 8 X = 0.0 m Y = 2.7 mZ = 0.0 m Repre \* 888 帘 R D 🔊 Font 11 🜲 Symbols 15 🚖 10 🚖 Loads Results 15 🚖 Show groups -None

o Click 'OK' to close this window.

- Click on <sup>1</sup> to start the snow generator on frames. The first frame bears 2m of snow loads (because the distance between the frames is 4m).

🥹 D	Snow Generator ×	
Eile Edit View Selec		- 8 ×
		-
😿 🖻 🖷 🔀  Venster 1		
y <sup>g</sup> <u>im</u> ⇔		ng 1 👻
snow (H <= 10(_▼		2.7 m
Geval 1 💌		-2.7m
***		
++0		rel manager
+		bund level
		wing plane
<b>*1</b> - 27		0.0 m 2.7 m
Temperature     Snow		0.0 m
*		
央 3 日	Width supported by structural frame: 2.00 m	
T T ≥	Snow load cases:	
Y Seismic     Dynamic	▼ Snow (H <= 1000 m) ▼ Geval 1 ▼	
* Moving		
8. 9	✓ Snow (H <= 1000 m)  Geval 2	
8.8		
	▼         snow (H <= 1000 m)	pups
65 (6		1
	<u>H</u> elp <u>C</u> ancel <u>Q</u> K	

- Complete the window as here above. Then click 'OK' to generate the snow.
- Now select the second frame and click on <sup>1</sup>. This frame bears 4m of snow loads.

Diamonds - HoutVB2	Snow Generator
Image: Series of the series	
	Width supported by structural frame:       4.00 m         Snow load cases:       Image: Comparison of the support of the suppo

- Repeat these steps for the all the other frames. When you come to the last frame, don't forget to set the width supported by the frame to 2m.
- Result:



# 5.2.3.3 Making combinations

### **Step 16: Making combinations**

Generate the combinations  $\frac{41}{100}$  as described in §5.1.3.3.

### Generating the mesh 5.2.4

# Step 17: Generating the mesh

Generate a mesh <sup>I</sup> as described in §5.1.4.

# 5.2.5 The global elastic analysis

Step 18: Elastic analysis

Follow the same method as described in §5.1.5.

Here below you find the bending moments for the combination ULS FC envelope and the vertical displacement for the combination SLS RC envelope.



Bending moments ULS FC envelope



Deformation SLS RC

It's possible to show the results for just a part of the structure:

- Select all relevant elements
- Then click on the icon II. The colour scale is adjusted to the results of the visible elements.

To set everything visible again, click on  $\square$ .

Look at some other results (normal and shear force, torsion, stress) by selecting the corresponding icon from the pallet.

# 5.2.6 Parameters for timber verification 5.2.6.1 Buckling

Step 19: Setting up groups for buckling

See §4.1.6.1 for the method.

Define the groups for buckling around the y'(v)-axis as follows (the end releases, the tie rods and the supports are set invisible to increase the readability of the image):



Define the groups for buckling around the z'(v)-axis as follows:



## Step 20: Calculating the buckling lengths

Now calculate the buckling lengths *ke*.



In each direction (round y'(u)- or z'(v)-axis) Diamonds asks you for which type of structure and for which type of analysis (first or second order) you would like to calculate the buckling lengths.

It is important that you use the same type of analysis as what you indicate here. We choose 'semi displaceable nodes'.

With 'Combination for determination of non-linearity's you select 'SLS QP1'.

Click 'OK'.

# 5.2.6.2 Lateral torsional buckling

## Step 21: Settings for lateral torsional buckling

The purlins are modeled in this structure, so that we don't have to do anything specific for lateral torsional buckling.

# **5.2.7** Timber verification

## Step 22: Timber verification

To start the verification, select the menu command 'Analysis – Steel and timber design' or click on F or press **F3**.



Results for strength (%)



Results for stability (%)

When we analyse the results we see that the structure is sufficient ( $\leq 100\%$  for both strength and stability).

# 5.2.8 Cross-section optimization

We now will use the optimization algorithm of Diamonds to find the most optimal cross-sections.

The optimization is based on the obtained percentages with the timber verification (see §5.2.7).

There are two optimization principles in Diamonds:

- The basic cross-section is provided in the **section library**. Diamonds will therefore search for the best suited cross-section in the library.
- The cross-section is built based on a **characteristic shape**. Diamonds will optimize by modifying either the height or width step by step defined by the user.

In this example the second method will be used, because the sections are based on a standard (rectangular shape). This first method is used in §5.1.8.

### Step 23: Cross-section optimization

To start the optimization, select the menu command 'Analysis – Optimization' of click on the button  $\frac{1}{2}$  in the icon bar. This window pops up:

Op	otimization ×
Optimization parameters Adjust cross-se	ections   Optimize user-defined cross-sections
Optimisation targets:	95 🚖 %
Optimize for:	I Resistance I Stability
Order sections by:	Resistance
Help < Previous	Next > Cancel QK

Opti	Optimization							
Optimization parameters Adjust cross-sections Optimize user-defined cross-sections								
Adjust cross-sections								
C Automatic adaptation								
How to replace cross-sections:								
Equal cross-sections remain equal af	ter optimisation							
All bars of same type have the same	section after op	timisation						
Help < Previous	Next >	<u>C</u> ancel	<u>о</u> к					
Opti	imization		×					
Optimization parameters Adjust cross-sect	tions Optimize (	user-defined cross	-sections					
This construction contains								
Define parameters for section type 'Solid rectangle':								
	rectangle':							
Adjust height	C Adjust t	hickness of web						
<ul> <li>Adjust height</li> <li>Adjust width</li> </ul>	C Adjust t	hickness of web hickness of flange	5					
<ul> <li>Adjust height</li> <li>Adjust width</li> <li>Adjust size in steps of:</li> </ul>	C Adjust t C Adjust t C Adjust t	hickness of web hickness of flange mm	5					
<ul> <li>Adjust height</li> <li>Adjust width</li> <li>Adjust size in steps of:</li> <li>Define minimum:</li> </ul>	C Adjust t C Adjust t	hickness of web hickness of flange mm mm	5					
<ul> <li>Adjust height</li> <li>Adjust width</li> <li>Adjust size in steps of:</li> <li>Define minimum:</li> <li>Define maximum:</li> </ul>	C Adjust t C Adjust t 10.0 10.0 500.0	hickness of web hickness of flange mm mm mm	5					

Fill in the settings like in the images here above.

### About the window 'Timber optimization'

- The **first tab** '**Optimization parameters**' contains a field with a percentage value. This is the value we should try to approach during the optimization.
- The second tab 'Adjust cross-sections' is present when the project contains cross-section coming from the section library. In this tab you can ask Diamonds to show you a summary of the optimization. If you do not ask the summary, Diamonds will adjust the cross-sections automatically.
- In the third tab 'Optimization used defined sections' specify the parameters for optimization of sections defined based on a type form.

When the optimization is completed, a dialog box appears with the summary of the optimization.

Optimisation report	×
<ul> <li>Bar number 9: Height changed from 180,0 to 100,0mm</li> <li>Bar number 10: Height changed from 180,0 to 100,0mm</li> <li>Bar number 11: Height changed from 180,0 to 100,0mm</li> <li>Bar number 12: Height changed from 180,0 to 100,0mm</li> <li>Bar number 13: Height changed from 180,0 to 100,0mm</li> <li>Bar number 14: Height changed from 180,0 to 100,0mm</li> <li>Bar number 15: Height changed from 180,0 to 100,0mm</li> <li>Bar number 15: Height changed from 180,0 to 100,0mm</li> <li>Bar number 15: Height changed from 180,0 to 100,0mm</li> <li>Bar number 16: Height changed from 180,0 to 100,0mm</li> <li>Bar number 17: Height changed from 180,0 to 100,0mm</li> </ul>	*
Number of bars to adjust: 36	<u>O</u> K

Diamonds proposes you to change some cross-sections. You can accept or ignore the changes by (un)checking the corresponding line.

Note: the optimization is only possible for constant cross sections. So the beams will not be optimized.

We don't make a report for this example. The report manager is explained in the second example in reinforcement (§3.2.8). The principle is the same.