

Software for civil engineers



## **Cross Section Analysis & Design**

# Worked Examples



## Contents

Introduction
Example 1 Reinforcement design of a beam section4
Example 2 Reinforcement design of a beam section with rebar size constraints
Example 3 Moment vs. Curvature Curve for specific axial force levels23
Example 4 Moment vs. Curvature Curve for specific reinforcement ratios
Example 5 Control of concrete cracking: Estimation of the stress at reinforcement (Serviceability Limit State)
Example 6 Control of concrete cracking: Estimate the stress at reinforcement (Serviceability Limit State) considering concrete tensile strength and using a parabolic concrete curve46
Example 7 Control of concrete cracking by reducing the maximum reinforcement stress49
Example 8 Reinforcement design for both Ultimate and Serviceability Limit States
Example 9 Check of user-defined reinforcement pattern in a composite cross section of a column
Example 10 Creation of interaction diagram for a composite cross section of a column76
Example 11 Cross sectional properties of a built-up steel section
Example 12 Creation of interaction diagram for a built-up steel section
Example 13 Estimation of maximum and minimum developed stresses on a built-up steel section under given external loads
Example 14 Check of a timber cross section114
Example 15 Steel jacketing for improvement of column strength and ductility





November 2012

## Introduction

Thank you for using ENGISSOL products. Our experienced scientific team can assure you that you will be provided with a robust, powerful and reliable application of high standards and guaranteed accurate results, regarding the analysis and design of any cross section (generic, built up of one or more geometric entities, composite etc.). ENGISSOL's know-how with respect to engineering application development is incomparable, a fact that is proven by thousands of satisfied end-users around the world.

Cross Section Analysis & Design performs all cross section calculation tasks. Among the program's capabilities are the calculation and illustration of Moment vs. Curvature graphs, interaction surfaces, stain distributions as well as stress contour plots. Moreover, reinforcement check or design options are also supported. The application fully complies with all major codes for reinforced concrete sections (AASHTO, UBC, AS 3600, IS 456, ACI 318, BS 8110, CSA A233, EC2, NZS 3101 and CP 65) but can also handle other user-defined materials (linear, bilinear, trilinear, parabolic or fully customized materials by providing a stress/strain curve).

Cross Section Analysis & Design covers efficiently a design engineer's everyday needs due to the program's simplicity and the wide range of supported cases.

This document contains a series of worked examples for Cross Section Analysis & Design which can be found in the application using the File -> Open sample project menu item. We have tried to include characteristic examples from an extended range of common cases in the field of concrete, steel and composite structures.

At ENGISSOL we are always looking for feedback on how to improve our products. You can contact us at engissol@engissol.com.



## **Example 1**

## **Reinforcement design of a beam section**

## Data

The bottom required reinforcement of a beam section 25 cm x 60 cm is to be estimated. Top reinforcement is  $3\phi12$ . Extra side reinforcement is  $2\phi12$ . Top and side reinforcement bars are constant and do not need to be designed.

Regulation: ACI 318

Concrete (cylinder) strength: 25 MPa

Yield stress of reinforcement steel: 500 MPa

Reinforcement cover to rebar center: 4 cm

Tensile concrete strength is ignored.

Stirrup type: Tied

Frame type: Intermediate

All remaining data (concrete stress block,  $\phi$  reduction factors, other coefficients etc.) to be taken according to ACI 318.

## **Load cases**

lc1:	N = 0	M <sub>y</sub> = -250 kNm	$M_z = 0$
lc2:	N = 0	M <sub>y</sub> = 20 kNm	$M_z = 0$
lc3:	N = 15 kN	M <sub>y</sub> = -45 kNm	$M_z = 0$
lc4:	N = -7 kN	M, = -65 kNm	M <sub>2</sub> = 0





## **Solution with Cross Section Analysis & Design**

First of all define the corresponding Reinforced Concrete Code, by clicking on the Project -> Reinforced Concrete Code menu item.

Reinforced Concrete Regulation
Reinforced concrete regulation
ACI 318 11 -
Stimup type Spirally Tied
Frame Type Ordinary, Intermediate Special
Reinforcement ratio limits <ul> <li>Default by code for columns</li> <li>User values</li> </ul>
User defined values Minimum ratio 0.0015 Maximum ratio 0.04
OK Cancel

**Selection of Reinforced Concrete Code** 

Selection of ACI 318 regulation and definition of Stirrup type (Tied) and Frame type (Ordinary, Intermediate)

Since the examined section does not belong to a column, we can select the "User values" option in the Reinforcement ratio limits box. We set a minimum ratio equal to 1.5‰ and keep the maximum value to 4%. Since the program is intended to handle generic cross sections, you should review the limits for reinforcement ratios, as they can differ depending on the frame elements which the examined section is assigned to. But in case of columns, you can keep the option "Default by code for columns".



#### **Definition of material properties**

Next, we are going to specify the material properties.

🐳 Concrete	? 🔀
Available concrete materials	Concrete
Default Concrete	Name Default Concrete Color
	Notes Default concrete material, compr. strength=15 MPa
	Concrete strength 15 MPa
	Confined
	User defined elasticity modulus
All available materials	🕂 Add new 🛛 🕂 From library 🕅 🗙 Delete
Default Bilinear Material Default Concrete Default Linear Material Default Parabolic Material Default Reinforcement Default Trililinear Material	
	OK Cancel

The Default Concrete material can be overridden by changing its properties, as shown in the form below.

💐 Concrete	2 🔀
Available concrete materials	Concrete         Name       C25         Color         Notes       Concrete, cylinder stress strength: 25 MPa         Concrete strength       25         MPa         Confined
All available materials C25 Default Bilinear Material Default Linear Material Default Parabolic Material Default Trililinear Material S500	Add new From library X Delete

Value "Concrete strength" has been set equal to 25 MPa. The concrete is not assumed confined and the elasticity modulus is to be calculated according to ACI 318, so we do not

Eng SSol



need to change anything else. The concrete defined here is now accessible through the name "C25". Alternatively we could click the "Add new" button in order to define a new concrete material.

The reinforcement can respectively be defined from the Material -> Reinforcement menu item.

💐 Reinforcement	? 🗙
Available reinforcement materials	Reinforcement         Name       S500         Color         Notes       Reinforcement material, Yield stress:500 MPa         Yield stress       500         MPa         Add new       From library
All available materials C25 Default Bilinear Material Default Linear Material Default Parabolic Material Default Trililinear Material S500	OK Cancel

We modify the Default Reinforcement material by specifying a new name (S500) and a yield stress of 500 MPa.

7





#### Drawing the geometry

We are now ready to draw the geometry of the cross section, by clicking the Draw -> Rectangle using dimensions. The center point of the rectangular section can be inserted by entering its coordinates (0,0) and then clicking the subtron, or just by clicking on the point (0,0) with the mouse. Then the Length and Width values should be set to 0.25 and 0.60 respectively. Finally, the C25 concrete should be selected for the rectangle we are drawing.



After we have clicked the "OK" button, the rectangle is shown in the drawing area.





Before drawing the rebars, we have to make sure that a proper rebar set is available for the design. This can be checked by selecting Project -> Available Rebars.

Hebars		l	? X
Name 06	Diameter 6	mm Moo	dify current
Rebars			
Nam	ne E	Diameter [mm]	A A
Φ6	6		
Φ8	8		-
Φ10	10		-
Φ12	12		
Φ14	14		
Φ16	16		
Φ18	18		
Φ20	20		
622	22		*
📌 Add new	/ From library	X Delete o	current
	OK Car	ncel	

The rebars shown above are available in the project. This means that we can draw rebars of these diameters and additionally the program will only choose from these rebar sizes when performing a reinforcement design.

So, we can now draw the reinforcement bars by clicking on the Draw -> Rebar Line menu item.



We can enter a value (0.04 m) for the reinforcement cover to the rebar center in the field, which becomes active when inserting reinforcement bars, as shown below. The program automatically draws help lines at the specified distance from the concrete rectangle edges and enables mouse snapping at their intersections.



To draw the bottom rebars, we click on both green circles successively.



Start and end point coordinates have been automatically filled in the corresponding form. The remaining properties of the bottom reinforcement can be specified, by selecting a rebar count (4), a material (S500) and a reinforcement size (ø20).

Eng SSOI <u>http://www.engissol.com/cross-section-analysis-design.html</u>



Note: The specified reinforcement ( $\emptyset$ 20) is not important at this point, since their diameter will be calculated by the program automatically and the required rebar will be assigned to them during the design procedure.

Rebar line							
Start point							
y 0.085 m z -0.26 m S2							
End point	et						
y 0.085 m z -0.26 m							
Offset direction							
Up  Down Offset 0 m							
Method s1 0 r	n						
Specify rebar count Rebar count 4     s2 0	n						
Specify distance Rebar distance 0.1 m							
Constant size for design Design parameters	Constant size for design Design parameters						
Grow factor 1							
Material S500    Reinforcement Φ20	•						
Split to single rebars							
OK Cancel							

After clicking the "OK" button, the lower rebars are shown in the drawing area.

We take the same steps in order to draw the top rebars. After clicking on the top points, we can define the corresponding properties, as below.

Rebar line					
Start point y -0.085 m z 0.26 m				\$2	
End point y 0.085 m z 0.26 m		S1	_	or	fset
Offset direction Up  Down Offset 0	m	-			
Method			s1	0	m
Specify rebar count Rebar co	unt 3		s2	0	m
Specify distance Rebar distance 0.1 m					
Constant size for design Design parameters Grow factor					
Material S500	<b>•</b>	Reinforceme	ent 🚺	D12	-
Split to single rebars					
ОК		Cancel			

Note that the rebar count is now set to 3 and the reinforcement size is  $\emptyset$ 12. Moreover, the option "Constant size for design" has been selected, so that the rebar sizes will not change

Eng SSol





during the design. After clicking the "OK" button, we can continue to draw the side reinforcement bars.



The rebar count is now set to 2 and both s1, s2 distances to 0.15 m. The option "Constant size for design" is still enabled since we do not want these rebars to be re-dimensioned.

Start point y -0.085 m z -0.26 m					
\$2					
End point y -0.085 m z 0.26 m					
Offset direction O Up  O Down Offset 0 m					
Method s1 0.15 m					
Specify rebar count Rebar count 2     s2 0.15     m					
Specify distance Rebar distance 0.1 m					
Constant size for design Design parameters Grow factor					
Material S500   Reinforcement Φ12					
Split to single rebars					
OK Cancel					

In order to draw the side reinforcement bars on the right, we can take advantage of the replicate option.





First of all, we click on the reinforcement line that has been defined before in order to select it.



The line appears now thicker with its related dimensions, which informs us that it is selected. In order to replicate it to the right side (16 cm to the right), we click on Edit -> Replicate menu item and on the "Linear" tab enter a dy distance equal to 0.17 m.

💐 Rep	licate			? 🗙	
Linear	Radial M	Airror			
Incre	ments —		-Increment data-		
dy	0.17	m	Number 1	_	
dz	0	m			
		]	L	)	
Delete original objects					
OK Cancel					

After clicking "OK", the side reinforcement on the right is shown correctly.







Note that the rebars have automatically been numbered. When referring to specific rebars (for example when reporting the reinforcement forces), these numbers will be used.





#### **Review of Analysis Parameters**

We need to specify the corresponding Analysis Parameters set by clicking Analysis -> Analysis Parameters. We will use the default "ULS" set. All parameters related to ACI 318 code can be found in this form. The concrete stress block is rectangular according to ACI 318, the tensile resistance of concrete is ignored and all remaining data have the default values according to the code. You can override these values by modifying any of the fields of the form.

💐 Analysis parameters	? 🗙
ULS Add new Analysis Parameters set Delete current set	
Name ULS Default Analysis Parameters Set for Ultimate Limit State	×
Reinforced concrete data Other Materials data	
Apply R/C code defaults	
Currice Data Amplification factors due to confinement	ור
Compression Part Default rectangular by code for ULS A rectangular stress block is only valid for resistance analysis!	
Selected R/C code       (Interaction, Reinforcement check and design)         Code specific values (ACI 318 05)         phi Tension controlled	
ρhi Compression controlled 0.65	
Tension Part	
No Tension	
Reinforcement Data Stress - Strain Curve Ignore compression part Stress - Strain Limits	51
Linear-Constant V 05 MPa	3
esu 0.02	
E. Compressive stress limit: 0 MPa	3
Esu Es	
OK Cancel	



-			
v	Worked	exampl	es

#### **Definition of load cases**

The load cases can be selected by clicking on the Analysis -> Reinforcement design -> Load cases menu item. The 4 load cases can be entered in the corresponding table. A new load case can be inserted by clicking the + button. We make sure that the assigned Analysis Parameters set to each load case is ULS.

💐 Loa	d cases fo	r reinforcem	ent design		?	
Load c	ases					
	Name	Axial force [kN]	Moment about y [kNm]	Moment about z [kNm]	Analy: Parame	sis ters
•	lc 1	0	-250	0	ULS	~
	lc 2	0	20	0	ULS	~
	lc 3	15	-45	0	ULS	~
	lc 4	-7	-65	0	ULS	~
	2			6		
1	< l				비비	
		OK	Cano	el		

#### Carry out the design

We just click Analysis -> Reinforcement design -> Analyze, to perform the reinforcement design procedure.

The program asks if it should assign the calculated rebars to the existing ones. We click on "Yes".



## Results

The results can be obtained from the Analysis -> Reinforcement design -> Show results menu item.

As we can see, the cross section is adequate and the bottom rebars have been chosen to be ø18. Remaining rebars (top and side reinforcement) did not change (ø12), as we previously checked the "Constant size for design" option. Furthermore, the calculated rebars have been assigned to the reinforcement bars we provided in the beginning.



Note: The program has calculated the required rebar sizes so that the provided reinforcement ratio lies between the limits specified in the Reinforced Concrete Regulation form.

Reinforcem	nent design re	esults						•	-	2
Design info				Res	ults for load c	ase: lc 1 –				
Cross sectio	n is adequa	te		Load	case Ic 1			•		
		_		Nee	led ratio 0.	011514	Needed ratio co	nsidering minimum	reinforcement 0	.011514
Minimum ratio	0.0015			Prov	ded ratio 0.	012064	Cross section	is adequate fo	r selected load	l case
laximum ratio	0.04									
Summary				Re	inforcement of	details				
										Þ
Load	Needed ratio	Placed ratio	Notes		y cord [m]	z cord [m]	As,needed [cm2]	Placed reinf.	As,placed [cm2]	*
	0.011514	0.012064	ok	1	-0.085	-0.26	2.3386	Φ18	2.5447	
CZ 5	0.2779E-3	6.0319E-3	ok	2	-0.028333	-0.26	2.3386	Φ18	2.5447	
C3 5	0.2//9E-3	6.0319E-3	ok	3	0.028333	-0.26	2.3386	Φ18	2.5447	
34 J	0.000TE-3	6.0319E-3	ок	4	0.085	-0.26	2.3386	Ф18	2.5447	≡
				5	-0.085	0.26	1.131	Φ12	1.131	
				6	0	0.26	1.131	Φ12	1.131	
				7	0.085	0.26	1.131	Φ12	1.131	
				8	-0.085	-0.11	1.131	Φ12	1.131	
				9	-0.085	0.11	1.131	Φ12	1.131	
						0.44	4 404			-





## Example 2

## Reinforcement design of a beam section with rebar size constraints

## Data

In this example, we are going to see how rebar size constraints can be used for designing reinforced concrete cross sections, so that the calculated rebars have a predefined area proportion, as shown in the picture below. The used cross section is similar to that defined in Example 1.



Demonstration of area constraint between rebars

Let's assume the following constraint: The area of each rebar 1, 2, 3 and 4 to the area of 7, 8 or 9 should be equal to 1.5. We suppose that side rebars are constant ø12, as in Example 1.

## **Load cases**

Same to these defined in Example 1.



## Solution with Cross Section Analysis & Design

#### **Opening a file from disk**

First of all we click on the File menu and select Open in order to open the file we created in Example 1.

#### Unlock the model

Afterwards, if the model is locked, we click on Edit -> Unlock model, in order to modify the geometry of the cross section.



In the popup window, we choose Yes to unlock the model.

Unlock	
2	Unlocking the file will result in the loss of all analysis results! Do you want to continue?
	<u>Y</u> es <u>N</u> o





#### **Application of constraints**

Next, we move the mouse over the reinforcement line at the top rebars (7, 8, 9) till the "rebar line" text appears.



After right-clicking, we select "Properties" from the menu, in order to view the properties of the top rebars.





In the corresponding form, we make sure that the option "Constant size for design" is unchecked and the "Grow factor" value is set to 1 and finally click OK.

Rebar line
Start point           y         -0.085         m         z         0.26         m
End point y 0.085 m z 0.26 m S1
Offset direction Up  Down Offset 0 m
Method s1 0 m
Specify rebar count Rebar count 3 s2 0 m
Specify distance Rebar distance 0.1 m
Constant size for design Design parameters Grow factor 1
Material S500    ■ Reinforcement Φ12
Split to single rebars
OK Cancel

Proceeding respectively for the bottom rebars, we set the "Grow factor" value to 1.5.

Rebar line	
Start point y -0.085 m z -0.26 m	
End point y 0.085 m z -0.26 m	S2 S1 Offset
Offset direction Up  Down Offset 0 m	
Method	s1 0 m
Specify rebar count Rebar count 4	s2 0 m
Specify distance Rebar distance 0.1	m
Constant size for design Design parame Grow factor	eters 1.5
Material S500 🗸	Reinforcement Φ18 -
Split to single rebars	
ОКС	Cancel

Note that the selected Reinforcement values are not important, since the reinforcement bars will be designed by the program.





#### Carry out the design

We just click Analysis -> Reinforcement design -> Analyze, to perform the reinforcement design procedure, as in Example 1.

The program asks if it should assign the calculated rebars to the existing ones. We click on "Yes".



#### Results

The results can be obtained from the Analysis -> Reinforcement design -> Show results menu item.

As we can see, the cross section is adequate and the bottom rebars have been chosen to be  $\emptyset$ 18. Top rebars are now of size  $\emptyset$ 14. But the calculated required area for reinforcement bars 1 to 4 and 7 to 9 conforms to the applied constraint, since 2.2966 cm<sup>2</sup> = 1.5 x 1.5311 cm<sup>2</sup>.

Remaining rebars (side reinforcement) did not change (ø12), as their "Constant size for design" option has been checked (in Example 1).

Note: The program has calculated the required rebar sizes so that the provided reinforcement ratio lies between the limits specified in the Reinforced Concrete Regulation form and also the applied constraint is valid. Furthermore, it should be noticed, that the constraint may not be precisely valid for the placed rebar areas as well, since the program chooses rebars from the available ones in project (Project -> Available Rebars menu item) under the condition that their area is greater than the required one.

Reinforcement design results				•			γ 2
Design info	Results f	for load ca	ase: lc 1				
Cross section is adequate	Load cas	se [lc 1			•		
	Needed r	ratio 0.0	)12202	Needed ratio co	nsidering minimum	reinforcement (	).012202
Minimum ratio	Provided	ratio 0.0	)12881	Cross section	is adequate fo	r selected loa	d case
Maximum ratio 0.04							
Summary	Reinfor	rcement d	etails				
							<b>P</b>
Load Needed ratio Placed ratio Notes	ID	y cord [m]	z cord [m]	As,needed [cm2]	Placed reinf.	As,placed [cm2]	^
C 1 0.012202 0.012881 ok	1 -0	0.085	-0.26	2.2966	Φ18	2.5447	
C 2 3.0159E-3 4.3354E-3 0K	2 -0	0.028333	-0.26	2.2966	Φ18	2.5447	
C 3 3.0103E-3 4.3304E-3 0K	3 0	928333	-0.26	2.2966	Φ18	2.5447	
C 4 3.4361E-3 4.3334E-3 0K	4 0	.085	-0.26	2.2966	Ф18	2.5447	Ξ
	5 -0	0.085	0.26	1.5311	Φ14	1.5394	
Validation of applied constraint:	6 0	)	0.26	1.5311	Φ14	1.5394	
$2.2966 \text{ cm}^2 = 1.5 \text{ x} 1.5311 \text{ cm}^2$	7 0	.085	0.26	1.5311	Φ14	1.5394	
	8 -0	0.085	-0.11	1.131	Φ12	1.131	
		0.085	0.11	1.131	Φ12	1.131	
	9 -6		1				





## Example 3

## Moment vs. Curvature Curve for specific axial force levels

## Data

The creation of Moments vs. Curvature curves using Cross Section Analysis & Design will be demonstrated in this example. The used cross section is similar to that defined in Example 2.





## Moment vs. Curvature diagrams data

The Moment vs. Curvature diagram for bending about Y axis is requested for the following axial force values:

- N = -1000 kN (compressive)
- N = -500 kN (compressive)
- N = 0 (no axial force)



## Solution with Cross Section Analysis & Design

## **Opening a file from disk**

First of all we click on the File menu and select Open in order to open the file we created in Example 1.

## Unlock the model

Afterwards, if the model is locked, we click on Edit -> Unlock model, in order to modify the geometry of the cross section.



In the popup window, we choose Yes to unlock the model.

Unlock	
2	Unlocking the file will result in the loss of all analysis results! Do you want to continue?
	<u>Y</u> es <u>N</u> o



#### **Review Analysis Parameters**

We click on Analysis -> Analysis Parameters menu item to view the existing Analysis Parameters sets. In the "ULS" set, the concrete stress distribution is chosen to be rectangular, which only applies to ultimate limit states and it has to be changed since the Moment vs. Curvature curve will be created for all possible stress states of the section, from the unstressed one to its failure.

Thus, we choose a Parabolic-Linear stress strain - curve for concrete compressive parts. All remaining values (factors, strains etc.) are now set according to ACI 318 regulation. By clicking OK the form is closed.

Hanalysis parameters			? <mark>x</mark>
ULS 🔸	Add new Analysis Paramet	ers set 🛛 🗶 Delete current set	
Name ULS Note	Default Analysis Parar	neters Set for Ultimate Limit State	۸ ٦
Reinforced concrete data Other Materials data			
Concrete Data	Apply R/C code defaults	Amplification factors due to	confinement
Compression Part		Concrete strength factor	1
Default rectangular by code for ULS	ecu -0.003	ec2 factor	1
Bilinear Parabolic-Constant	ec2 -0.002	ecu factor Code specific values (ACL3	1
Parabolic-Linear Eurocode 2-ULS Eurocode 2-SLS	c 0.85	phi Tension controlled	0.9
BS 81110-97	λ 1	phi Compression controlled	0.65
Tension Part			
No Tension 👻			
Reinforcement Data	ression part	Stereo Sterio Linzia	
Linear-Constant ▼		Tensile stress limit:	) MPa
E 199.95 GPa	o0	Tensile strain limit:	)
esu 0.02		Compressive stress limit:	) MPa
Es.	د ٤su ٤s	Compressive Strain limit:	)
	OK Ca	ncel	



## **Definition of load cases**

The load cases for Moment vs. Curvature analysis can be selected by clicking on the Analysis -> Moment Curvature -> Load cases menu item. By clicking the + button we can add a new load case. The Moment curvature type is set to "Moment about y", the Custom Angle (coordinate system rotation) is left to 0 and the Analysis Parameters item is changed to "ULS" so that we can use the parameters defined previously.

	📛 Loa	d cases for	Moment Curvature					? X
	Load	cases						
		Name	Moment curvature typ	be	Custom Angle [deg]	Analysis Parameters		Options
	1	lc 1	Moment about y	•	0	ULS	•	Options
l	+	×	ОК		Cancel			

In order to use specific axial forces for the analysis, we click on "Options" button and in the form that shows, we select the option "Alternate Axial Force".

👋 Options for N	Ioment Curvature	? ×
Load case name Interaction Type Variations No alteration Alternate Ax Atemate Re	Ic 1 Curvature y vs. Moment y ial Force einforcement Ratio	Axial force Axial force Add Modify Remove
Bilinearization of Second branch	f Moment-Curvature curve slope 0 Cancel	



Next we enter the axial force values successively in the corresponding field at the top right corner of the form and click the "Add" button, so that all axial forces have been filled, as shown below.

Coptions for Moment Curvature	? ×
Load case name Ic 1 Interaction Type Curvature y vs. Moment y Variations No alteration Alternate Axial Force Alternate Reinforcement Ratio Bilinearization of Moment-Curvature curve Second branch slope 0 OK Cancel	Axial force Axial force 0 kN Add Modify Remove 0 -500 -1000

We can also define the slope of the second branch of the equivalent bilinearized curve here, but we keep this value equal to 0. This means that the calculated bilinearized curve will be elastic-fully plastic. We click OK to close the form.

#### **Carry out the analysis**

We just click Analysis -> Moment Curvature -> Analyze, to perform the Moment Curvature analysis.





## Results

The results can be obtained from the Analysis -> Moment Curvature -> Show results menu item.

d case	la 1		_	Selected curve data										
ment or	lint		•	Axial force [kN]	0		-		Show strai	n/stress				
vature		м	kNm	Color	Evport current granh data		oh data	d	istibution for	current point				
Show e	equivalent	Bilinearized curve	e - Positive part		Capon	contra gra	Bilineariz	ed curve - Neg	ative part					
linear	ized curve	Pos. yield curvatur	re 8.6454E-3	Pos. yield moment	219.68	kNm	Neg. yield	curvature	-8.5298E-3	Neg. yield mome	nt -319.3	kNm		
		Pos. ultimate curva	ature 0.037714	Pos. ultimate moment	t 219.68	kNm	Neg. ultim	ate curvature	-0.027648	Neg. ultimate mo	ment -319.3	kNm		
						Mom	ent - Cur	vature Cur	ve (lc 1)					
		N=0 kN	N=-500 I	KN N=-100	0 kN									
	600 +		! ! ! !		! ·		· T		<u> </u>	· · · !		• <u>!</u> • •	· · · · · ·	
1	Ŧ		-											
	500 ± ·												• • • • • • • • • • • • • • • • • • • •	• • • • • • • • • • •
	ŧ													
	400 -		÷											
	ŧ		-							-				
	200 E		1	1							-			
	ŧ							///	<u>م ا</u>					
	200				• • • • • • • • • • • • •				· · · ·	· · · · ·		•••••		• • • • • • • • • • • •
	ŧ													
_	100 +		·····		• • • • • • • • • • • •		····· /·	. f. i, e		••••••			•••••	
E N	ŧ													
놑	o 🕇													
l l	Ŧ													
ž	-100 E						//							
	100					/	//							
	+													
	-200						7					:		
	ŧ		1			2/1				-				
	-300 +		•				· · · · · · · · · · · · · · · · · · ·				• • • • • • • • • • • • • • •			
	Ŧ													
	-400 +			····÷		/		•••••	· · · <del>:</del> · · · ·	· · · · · · · · · · · · · · · · · · ·	• • • • • • • • • • • • • • • • • • • •	····÷···	·····	
	ŧ				+									
	-500													
	ŧ													
	-600													
	-0.04	-0	0.03	-0.02	-0.01		0		0.01	0.02		0.03	0.04	
								Curvature						

The Moments vs. Curvature diagrams for the three specified axial force levels are shown in the form. We note that the equivalent bilinearized curves (same color, but dashed) have also been calculated. These can disappear by unchecking the "Show equivalent bilinearized curve".



## 🍷 Worked examples



The color of each curve can be controlled from the corresponding "Color" box after selecting the curve to modify by its axial force level.



## Section stress distribution for a specified point on Moments vs. Curvature curve

Assuming we would like to display the strain distribution and stress contour for a specific point on the diagram for the curve which refers to axial force level of -1000 kN.

Moment curvature results ? X Selected curve data
 Axial force [kN] Load case Ic 1 -1000 Current point Curvature -7.2366E-3 M -397.44 Show strain/stress distibution for current point Color kNm -500 Bilinearized curve - Pos. yield curvature 8.6454E-3 Pos. yield me 219.68 -8.5298E-3 Neg. yield moment -319.3 kNm Neg. yield curvature Pos. ultimate curvature 0.037714 Pos. ultimate moment 219.68 Neg. ultimate curvature -0.027648 Neg. ultimate moment -319.3 Ô ÷ Moment - Curvature Curve (lc 1) • N=0 kN N=-500 kN N=-1000 kN Q 600 à, \* 500 400 300 nent [kNm] Mom 600 -0.03 -0.02 -0.01 0.03 0.01 0.04 -0.04 Curvature Close

First of all we select this value from the related list as shown in the picture below.

Afterwards we move the mouse over the the graph .







By clicking on "Show strain/stress distribution for current point", a form is shown, which represents the following data:

- Neutral axis location
- Strain distribution
- Equivalent force couple
- Stress contour on the cross section area
- Reinforcement forces
- Stress information per material

The form is shown below.





Jan St	ain plane analysis results					? ×
-	un plane analysis results					
Load	case Deformed configuration at defin	ed moment 🔻				
÷	Neutral axis location and deformed cor	figuration			Neutral axis information	
Ð	-0.6 -0.55 -0.5 -0.45 -0.4 -0.35 -0.	3 -0.25 -0.2 -0.15 -0.1 -0.05 0 0.05 0.1 -0.15 -0.2 -2	(0.25 0.3 0.35 0.4 0.45 0.5 0.55 0.6 0.	65 0.7 0.75 <u>0.</u> 8 0.85		
a		1		[0]	Property Value	e Units
n.	0.25	•		0.25	N -1000.1	kN
-3-	0.2		1447.8 KN	0.2	My -399.45	kNm
R	0.15	0.2862 m		0.15	Mz 2.8388E-13	kNm
	0.1		0.18592 m	0.1	Eqv. tensile Force 447.66	kN
	0.05				Tensile force lever arm -0.24638	m
	0.05	NA		0.05	Eqv. compr. Force -1447.8	kN
	0	Î	l ↑	°-	Compr. force lever arm 0.19973	m
	-0.05		Λ	-0.05		
	-0.1		0.26018 m	-0.1	Reinforcement forces	
	-0.15	0.3138 m		-0.15		
	-0.2			-0.2	ID y cord z cord Force [m] [m] [kN]	[MPa] =
	-0.25			-0.25	1 -0.085 -0.26 101.70	399.65
			447.66 KN		2 -0.028333 -0.26 101.70	399.65
	-0.6 -0.55 -0.5 -0.45 -0.4 -0.35 -0.	3 -0.25 -0.2 -0.15 -0.1 -0.05 0 0.05 0.1 0.15 <b>2</b> ,29	088533(tensiens) 0.4 0.45 0.5 0.55 0.6 0.	65 0.7 0.75 0.8 0.85	3 0.028333 -0.26 101.70	399.65
	Stress distribution				4 0.085 -0.26 101.70	399.65
	7 -0.65 -0.6 -0.55 -0.5 -0.45 -0.4 -	0.35 -0.3 -0.25 -0.2 -0.15 -0.1 -0.05 0 0.05 0.1 (	0.15 0.2 0.25 0.3 0.35 0.4 0.45 0.5 0.5	5 0.6 0.65 0.7 0.75		
				[③] 412.78		
	-0.25			314.81		
	-0.2			0.2	Material stress information	
	0.15			216.83	Material C25	🗕 🗈 🗈
	-0.1	• •		0.1	Property Value	Units
	0.05			118.85	Max stress 0	MPa
	0			20.875	y cord of max stress -0.085	m
	-0.05			1.05	z cord of max stress -0.251	m
				-77.102	Min stress -25	MPa
	-0.1	•		-0.1	y cord of min stress 0.125	m
	-0.15			-175.08 0.15	z cord of min stress 0.28777	m
	-0.2			-273.06	L	
	-0.25			0.25	Analysis info Cross section is adequate	
	7 .0.6 .0.6 .0.7 .0.4 .0.4		115 0.7 0.75 0.3 0.75 0.4 0.45 0.5 0.5	-371.03		
					Close	
Axes	Grid Ruler Dim lines Labels R	einforcement rebar Nr.				



## **Example 4**

## Moment vs. Curvature Curve for specific reinforcement ratios

## Data

Aforementioned Example 3 will be extended here, in order to have Moment vs. Curvature diagrams calculated for different reinforcement ratios. The used cross section is similar to that defined in Example 3.

## Moment vs. Curvature diagrams data

The Moment vs. Curvature diagram for bending about Y axis is requested for the following reinforcement ratio values:

- 0.5%
- 1.0%
- 1.5%

## **Solution with Cross Section Analysis & Design**

## **Opening a file from disk**

First of all we click on the File menu and select Open in order to open the file we created in Example 3.







## Unlock the model

Afterwards, if the model is locked, we click on Edit -> Unlock model, in order to modify the geometry of the cross section.

🐳 Cross !	Section Analysis &	Design - [C	: Woc um	ents a	and Settings\mary\Desktop\1.csad*]	- 8 🔀
File Edit	it View Project	Materials C	Draw An	alysis	; Settings Help	
	Select		- • I			
	Assign Material to Sele	ected Figures				
in <sub>a</sub> i	Properties		¢	<b>b</b> 🗄		
1 1 1 1 1 1 1 1	Move					
Informa 💼	Сору		- 1		<u>□□₽₫000∆®\$Ľ I +∛Φ</u> #	
Proje 📀	Rotate			0.1		0.35 0.4 3 87
	Mirror			1		(i)
Unit 🐁	Replicate		- b	+	F Z 2 2	0
Deta	Delete			Ð		0.25 4
Rein 🛷	Change Dimension Lin	e Location		Q		
	Unlock Model	Ctrl	HU	14	A Fue	. = 💻
Defit 🔊	Undo	Ctr	I+Z	-40		
~	Redo	Ctr	I+Y		I E I I I I I I I I I I I I I I I I I I	2
Default Br	oncrete	Bein	forcement		-0.15	0.15
Default Bil	ilinear Material	Biline	Nar			
Default Lir	inear Material	Linea	м		<b>● ●</b>	0.1
Default Pa	arabolic Material	Para	bolic			
Default Tr	riilinear Material	Trilin	ear		E and the second s	
					z A	
Analysis p	parameters					
Name	Note Default Analysis Parameter	es ers Set for Liltin	nate Limit			
ULS S	State		in a fille		-0.05	-0.05
SLS L	Jerauk Analysis Paramete Limit State	ers sector serv	viceability			
- Load case	03				-0.15	-0,15
Name	Туре	1.	Analyzed			
le 1 C	Code based interaction				-0.2	-0.2
Ic1 R	R/C design					
102 H	R/C design				1 12 13 4	0.25
lo 4 B	R/C design		<b>I</b>			3
					a con also also also also also also also also	0.35 0.4
Axes Grid	Ruler Draw cross Us	ser points Dir	m lines Lab	oels R	Reinforcement reber Nr. Snap to grid Snap to points Snap to Mid points Delete after microring/rotating Grid distance 0.05 m	
Selected Figu	ures: 1, Model is locked,	Ctrl+U to unio	ock.		y= -0.127m , z= -0.323m	Units: Default Metric 👻

In the popup window, we choose Yes to unlock the model.

Unlock	
2	Unlocking the file will result in the loss of all analysis results! Do you want to continue?
	<u>Y</u> es <u>N</u> o



#### Definition of a new load case

We click on the Analysis -> Moment Curvature -> Load cases menu item and next on the button in order to insert a new load case "Ic 2". Load case "Ic 2" contains information regarding the reinforcement ratio levels used for the analysis. We change the Analysis Parameters to "ULS" and keep the remaining values. Afterwards, the different reinforcement ratio levels can be defined by clicking on the "Options" button of the "Ic 2" row.

 💐 Load cases for Moment Curvature								
-Load c	ases							
	Name	Moment curvature type		Moment curvature type Custom Analys Angle [deg] Parame		Analysis Parameters		Options
	lc 1	Moment about y	•	0	ULS	•	Options	
Þ	lc 2	Moment about y	•	0	ULS	•	Options	
+ 2	<	ОК		Cancel		C	1	

On the form following that shows, we choose the "Alternate Reinforcement Ratio" option and provide the reinforcement ratio values on the list on the right.

Hoptions for M	oment Curvature	? ×
Load case name Interaction Type Variations No alteration Alternate Axi a Alternate Rei Bilinearization of Second branch s	Curvature y vs. Moment y al Force inforcement Ratio Moment-Curvature curve slope 0	Reinforcement ratios         Reinforcement ratio         0.015         Add         Modify         Remove         0.015         0.01         0.005
ОК	Cancel	

We click OK to close the form.





#### Carry out the analysis

We just click Analysis -> Moment Curvature -> Analyze, to perform the Moment Curvature analysis.

#### Results

The results can be obtained from the Analysis -> Moment Curvature -> Show results menu item. On this form, we choose "Ic 2" in the Load case list at its top left corner in order to view the results of the load case defined previously.



The Moments vs. Curvature diagrams for the three specified reinforcement ratio levels are shown in the form. We note that the equivalent bilinearized curves (same color, but dashed) have also been calculated. These can disappear by unchecking the "Show equivalent bilinearized curve" box.






The color of each curve can be controlled from the corresponding "Color" box after selecting the curve to modify by its reinforcement ratio level.

Note: All reinforcement size constraints are respected when a reinforcement ratio is calculated. Thus, the side reinforcement bars are constant and bottom reinforcement area is 50% greater of the top ones (as they grow factor has been specified equal to 1.5).

37



#### Section stress distribution for a specified point on Moments vs. Curvature curve

Assuming we would like to display the strain distribution and stress contour for a specific point on the diagram for the curve which refers to reinforcement ratio level of 0.015.



First of all we select this value from the related list as shown in the picture below.

Afterwards we move the mouse over the the graph.







By clicking of "Show strain/stress distribution for current point", a form is shown, which represents the following data:

- Neutral axis location
- Strain distribution
- Equivalent force couple
- Stress contour on the cross section area
- Reinforcement forces/stresses
- Stress information per material

The form is shown below.





🥮 S	rain plane analysis results						? <mark>×</mark>
Loa	Case Deformed configuration at defined moment						
÷	Neutral axis location and deformed configuration				Neutral axis information		
Ð,	-0.6 -0.55 -0.5 -0.45 -0.4 -0.35 -0.3 -0.25 -0.2 -0	15 -0.1 -0.05 0 0.05 0.1 -1:6485E-3 (compres	55100) 0.4 0.45 0.5 0.55 0.6 0.65 0.7	0.75 0.8 0.85			
٩	-0.25	• • •	728.01 kN	0.25	Property	Value	Units
Là	0.15851 m		0.10103	0.2	N	-91.162	KN INNE
÷¢.	0.15 NA		0.10402 m	0.15	Mz	-321.00 2.6157E-13	kNm
	0.12		<b>Ť</b>	0.15	Equitensile Force	636.85	kN
	-0.1	T T		0.1	Tensile force lever arm	-0.22476	m
	0.05			0.05	Eqv. compr. Force	-728.01	kN
	0			0-	Compr. force lever arm	0.24551	m
	-0.05		0.36625 m	-0.05			
	0.1			-0.1	Reinforcement forces		
	-0.15			-0.15			
	-0.2			-0.2	ID y cord z co	d Force [kN]	Stress ^
	-0.25		636.85 kN	-0.25	1 -0.085 -0.26	127.23	500
	E J	$\Phi \Phi \Phi \Phi$			2 -0.028333 -0.26	127.23	500
	-0.6 -0.55 -0.5 -0.45 -0.4 -0.35 -0.3 -0.25 -0.2 -0	15 -0.1 -0.05 0 0.05 0.1 0.15 <b>1559 1553 (ten</b>	<sup>psi</sup> @93 0.4 0.45 0.5 0.55 0.6 0.65 0.7	0.75 0.8 0.85	3 0.028333 -0.26	127.23	500
	Stress distribution				4 0.085 -0.26	127 23	500 *
	7 -0.65 -0.6 -0.55 -0.5 -0.45 -0.4 -0.35 -0.3 -0.2	-0.2 -0.15 -0.1 -0.05 0 0.05 0.1 0.15 0.2	0.25 0.3 0.35 0.4 0.45 0.5 0.55 0.6	0.65 0.7 0.75			
	E		[@]	500			
	-0.25			404 62			
	E-0.2			0.2	Material stress information	n	
	0.15			309.23	Material C25		- 🗈 🗅
	-0.1	· · · · · · · · · · · · · · · · · · ·		0.1	Property	Value	Units
	-0.05			213.85	Max stress 0	N	/IPa
				118.46 0	y cord of max stress -0	.085 m	1
	-0.05			0.05	z cord of max stress -0	.251 m	1
	01			23.081	Min stress -2	4.228 N	4Pa
	E 0	• •		-0.1	y cord of min stress 0.	125 m	<u> </u>
	-0.15		-	/2.303 0.15	z cord of min stress 0.	3 п	1
	-0.2			167.69			
	-0.25			1.25	Analysis info Cross section is ade	quate	
	17 .0.65 .0.6 .0.55 .0.5 .0.45 .0.4 .0.35 .0.3 .0.7	-0.2 -0.15 -0.1 -0.05 0 0.05 0.1 0.15 0.2	0.25 0.3 0.35 0.4 0.45 0.5 0.55 0.6	263.07			
	Englandendendendendendend		uduuluuluuluuluuluuluulu			uose	
Axes	Grid Ruler Dim lines Labels Reinforcement	rebar Nr.					





# **Example 5**

# Control of concrete cracking: Estimation of the stress at reinforcement (Serviceability Limit State)

## Data

We will extend the file created in Example 2 in order to calculate the stress at each rebar under the loads defined below.

As many regulations propose, the effect of cracking can be controlled by considering a reduced reinforcement stress which is specified by other parameters, such as allowable crack width, rebar diameters, rebar spacing etc.

The concrete stress/strain curve is assumed to be linear.

## **Load cases**

lc1:	N = 0	M <sub>y</sub> = -120 kNm	$M_z = 0$
lc2:	N = 0	M <sub>y</sub> = 37 kNm	$M_z = 0$
lc3:	N = 20 kN	M <sub>y</sub> = -54 kNm	$M_z = 0$

# **Solution with Cross Section Analysis & Design**

### **Opening a file from disk**

First of all we click on the File menu and select Open in order to open the file we created in Example 2.







#### Unlock the model

Afterwards, if the model is locked, we click on Edit -> Unlock model, in order to modify the geometry of the cross section.

🗳 Cross	Section Analysis &	Design - [C:\Docum	ients a	nd Settings\mary\Desktop\1.csad*]	_ B X
File Edi	it View Project	Materials Draw A	nalysis	Settings Help	
	Select	•			
	Assign Material to Sel	lected Figures 🔹 🕨			
in <sub>a</sub>	Properties		û 🔥		
1 1 3	Move				
Informa 💼	Сору			<u>ㅂㅁ;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;;</u>	
Proje @	Rotate			علم دم علم بلم علم و علم بلم علم دم علم دم علم دم	0.3 0.35 0.4 3
	Mirror		1		(i) - (i)
Unit	Replicate		+		0
Deta	Delete		Đ.		0.25 - 4
Rein 🤝	Change Dimension Lir	ne Location	Q		
	Unlock Model	Ctrl+U	14		
Defit 🔊	Undo	Ctrl+Z	-4		
~	Redo	Ctrl+Y			
Default B	concrete leinforcement	Beinforcement		0.15	0.15
Default Bi	ilinear Material	Bilinear			
Default Li	inear Material	Linear			0.1
Default P	arabolic Material	Parabolic			1
Default T	minear Material	Infinear			0.05
				Z∱	3
			l		
Analysis p	parameters Net		1	• v	
In c C	Default Analysis Paramet	ers Set for Ultimate Limit			1
	State Default Analysis Parametr	ers Set for Serviceshiltu		-0.05	-0.05
SLS L	Limit State	or our for our needowy			1
-Load cas	105				-0,15
Name	Туре	Analyzed			1
le 1 0	Code based interaction			-0.2	-0.2
101 P	H/C design B/C design				
lc 3 F	R/C design				
lo 4 F	R/C design	<b></b>			
				$\begin{bmatrix} c_0 J_5 & -c_0 J_5 & -c_0 J_5 & -c_0 J_2 & -c_0 J_1 & -c_0 J_5 & 0 & -c_0 J_5 & -c_0 J_2 & -c_0 J_5 & -c_0$	03-035-04-1
Axes Grid	Ruler Draw cross U	ser points Dim lines La	abels R	enforcement rebar Nr. Snap to grid Snap to points Snap to Mid points Delete after mirroring/rotating Grid distance 0.05 m	
Selected Figu	ures: 1, Model is locked,	Ctrl+U to unlock.		y0.127m, z=-0	).323m Units: Default Metric +

In the popup window, we choose Yes to unlock the model.

Unlock	
2	Unlocking the file will result in the loss of all analysis results! Do you want to continue?
	<u>Y</u> es <u>N</u> o



#### **Review of Analysis Parameters**

We need to specify the corresponding Analysis Parameters set by clicking Analysis -> Analysis Parameters. We will use the default "SLS" set. The concrete stress block is linear and the corresponding strain values are defined according to ACI 318. Strength reduction factors  $\phi$  do not need to be changed since we are performing an analysis in Serviceability Limit State (stress calculation). The concrete tensile strength is ignored. You can override these values by modifying any of the fields of the form.

analysis parameters	8 ×
SLS   Add new Analysis Parameter	s set Delete current set
Name SLS Notes Default Analysis Parame	eters Set for Serviceability Limit State
Reinforced concrete data Other Materials data	_
Concrete Data	Amplification factors due to confinement
Stress Strain Curve Compression Part	Concrete strength factor 1
ecu -0.003	max axial strain factor 1
max axial strain -0.002	ecu factor 1 Code specific values (ACI 318 05)
$\lambda$ 1	phi Tension controlled
ε <sub>cu</sub> ε <sub>c</sub>	phi Compression controlled 1
Tension Part	
No Tension 💌	
Reinforcement Data Ignore compression part	Stress - Strain Limits
Linear-Constant	Tensile stress limit: 0 MPa
E 199.95 GPa fy	Tensile strain limit:
	Compressive stress limit: 0 MPa
	Compressive Strain limit: 0
OK	cel

EngiSSOI <u>http://www.engissol.com/cross-section-analysis-design.html</u>





#### **Definition of load cases**

We click on the Analysis -> Deformed configuration -> Load cases menu item and next on the + button in order to insert a new load case under the name "Ic 1". Afterwards we fill the values for loads and make sure that the "SLS" Analysis Parameters set is enabled. We can create the remaining load cases Ic 2 and Ic 3 likewise.

👋 Load	l cases for o	leformed conf	iguration		? X	
-Load ca	ases					
	Name	Axial force [kN]	Moment about y [kNm]	Moment about z [kNm]	Analysis Parameters	
•	lc 1	0	-120	0	SLS 🗸	•
	lc 2	0	37	0	SLS 🗸	
	lc 3	20	-54	0	SLS 🗸	
+ >	<	ОК	Canc	el	) () (	

We click OK to close the form.

#### Carry out the analysis

We just click Analysis -> Deformed configuration -> Analyze, to perform the analysis.



#### Results

The results can be obtained from the Analysis -> Deformed configuration -> Show results menu item.

On this form, we choose the "lc 1" load case at the top left corner, in order to view the results related to the first defined load case.

The following data can be found on this form.

- Neutral axis location
- Strain distribution
- Equivalent force couple
- Stress contour on the cross section area
- Reinforcement forces/stresses
- Stress information per material



As we can see, the maximum stress values at reinfordcement are reported as expected for rebars 1 to 4 (bottom layer reinforement) and have a value of 222.72 MPa.

By the above mentioned way, we can obtain the results for the remaining load cases "lc 2" and "lc 3" by changing the selected load case item at the top left corner of the results form.



# **Example 6**

# Control of concrete cracking: Estimate the stress at reinforcement (Serviceability Limit State) considering concrete tensile strength and using a parabolic concrete curve

## Data

We will modify the file created in Example 5 in order to re-estimate the reinforcement stresses, but considering the concrete tensile strength and using a Parabolic-Constant concrete curve for compression.

## **Load cases**

As defined in Example 5.

## **Solution with Cross Section Analysis & Design**

#### **Opening a file from disk**

First of all we click on the File menu and select Open in order to open the file we created in Example 5.

#### Unlock the model

Afterwards, if the model is locked, we click on Edit -> Unlock model, in order to modify the geometry of the cross section.





In the popup window, we choose Yes to unlock the model.



#### **Review of Analysis Parameters**

We need to modify the previously created Analysis Parameters set "SLS" by clicking Analysis -> Analysis Parameters and selecting "SLS" from the list at the top left corner.

Next, we select a Parabolic-Linear stress strain curve for compressive parts of concrete. Finally, in order to consider the tensile concrete strength, we select the "Linear and Drop to Zero" option in the tensile concrete data, as shown in the form below.

Hanalysis parameters		_	8 X
SLS   Name SLS	Add new Analysis Parameters Notes Default Analysis Parame	s set Delete current set	jtate
Reinforced concrete data Other Materia	als data Apply R/C code defaults		
Concrete Data		Amplification factors due to (	confinement
Stress Strain Curve Compression Part	•	Concrete strength factor	1
σ <sub>c</sub>	ecu -0.003	ec2 factor	1
	ec2 -0.002	ecu factor	1
Af <sub>c</sub>	c 0.85	-Code specific values (ACI 3	18 05)
cλfc		phi Tension controlled	1
ε <sub>c2</sub> ε <sub>cu</sub>	λ 1	phi Compression controlled	1
Tension Part			
Linear and Drop to Zero	•		
Reinforcement Data Stress - Strain Curve	re compression part	Stress - Strain Limits	
Linear-Constant		Tensile stress limit:	) MPa
esu 0.02		Tensile strain limit:	)
		Compressive stress limit:	MPa
¥	۲ <u>Ls</u> ٤su ٤s	Compressive Strain limit:	
	OK Cano	el	



#### **Definition of load cases**

The load cases do not need to be changed.

#### **Carry out the analysis**

We just click Analysis -> Deformed configuration -> Analyze, to perform the analysis.

#### Results

Similarly to Example 5, the results can be obtained from the Analysis -> Deformed configuration -> Show results menu item.



We notice that the stress at rebars 1 to 4 (bottom layer rebars) is now 208.39 MPa for load case "lc 1".



# **Example 7**

# Control of concrete cracking by reducing the maximum reinforcement stress

## Data

In this example, we will introduce the application of reinforcement stress limits in order to prevent the cracking of concrete. This option is available in the "Analysis Parameters" form. So, by applying the corresponding Analysis Parameters set to each analysis, the corresponding limits will be considered automatically.

In this example we will apply a tensile stress limit of 180 MPa.

## **Load cases**

As defined in Example 5.

# **Solution with Cross Section Analysis & Design**

#### **Opening a file from disk**

First of all we click on the File menu and select Open in order to open the file we created in Example 6.

#### Unlock the model

Afterwards, if the model is locked, we click on Edit -> Unlock model, in order to modify the geometry of the cross section.







In the popup window, we choose Yes to unlock the model.



#### **Review of Analysis Parameters**

We need to modify the previously created Analysis Parameters set "SLS" by clicking Analysis -> Analysis Parameters and selecting "SLS" from the list at the top left corner.

At the bottom right corner of the form, we activate the option "Tensile stress limit" and enter the value 180 MPa.

🐳 Analysis parameters			? ×
SLS - Add	new Analysis Parameter	s set X Delete current set	
Name SLS Notes	Default Analysis Parame	eters Set for Serviceability Limit	State
Reinforced concrete data Other Materials data			
Concrete Data Stress Strain Curve Compression Part Parabolic-Linear e λf <sub>c</sub> cλf <sub>c</sub> λ	cu -0.003 c2 -0.002 0.85 1	Amplification factors due to Concrete strength factor ec2 factor ecu factor Code specific values (ACI 3 phi Tension controlled phi Compression controlled	confinement 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1
Tension Part Linear and Drop to Zero ▼			
Reinforcement Data Stress - Strain Curve	ion part	Stress - Strain Limits	]
Linear-Constant	ß	Tensile stress limit:	180 MPa
E 199.95 GPa esu 0.02 fy		Tensile strain limit:	0 MPa
Es_	Esu Es	Compressive Strain limit:	D
	DK Cano	el	

Eng SSol



#### **Definition of load cases**

The load cases do not need to be changed.

#### **Carry out the analysis**

We just click Analysis -> Deformed configuration -> Analyze, to perform the analysis.

#### Results

Similarly to Example 6, the results can be obtained from the Analysis -> Deformed configuration -> Show results menu item.



The maximum tensile stress at reinforcement was 208.39 MPa in Example 5, where no reinforcement limit was applied. For the same loads (lc 1), we can now see that the cross section is now inadequate.





Cross section is adequate for the remaining load cases, where apparently the developed reinforcement stresses are smaller than the limit of 180 MPa.



Note: These limits are respected for other analysis types as well (such as Interaction diagram, Reinforcement design etc.) provided that such stress limits have been applied to the corresponding Analysis Parameters sets.





# Example 8

# Reinforcement design for both Ultimate and Serviceability Limit States

## Data

In this example, we will perform a reinforcement design for both Ultimate and Serviceability Limit States (ULS and SLS). Such procedure is very common when designing reinforced concrete beams, since both requirements, regarding ultimate strength and cracking due to service loads should be checked.

Cross Section Analysis & Design can easily handle similar cases by modifying the default Analysis Parameters sets ULS and SLS. If more design situations should be considered, we can create new Analysis Parameters sets.

We will start from the file created in Example 7. So materials and section geometry will not change.

Concre Reinforce	ete and ment data	Ultimate Limit State (ULS)	Serviceability Limit State (SLS)	
Stross/Strain	Compressive	Rectangular according to ACI	Barabalic Constant	
distribution	part	318	Farabolic-constant	
of concrete	Tensile part	Ignored (no tensile strength)	Yes (consider tensile	
orconcrete			resistance)	
Boinforcomor	t stross limits	No limits (uso full strongth)	Reduce tensile strength to	
Remorcemer	it stress limits	No mints (use full strength)	220 MPa to limit crack width	

The additional design data are as follows:

## **Load cases**

Moreover, the reinforcement design procedure will take place for the following loads:

#### Ultimate loads (ULS)

lc1-ULS	N = 0	M <sub>y</sub> = -140 kNm	$M_z = 0$
lc2-ULS	N = 0	M <sub>y</sub> = 55 kNm	$M_z = 0$
lc3-ULS	N = -10 kN	M <sub>y</sub> = -44 kNm	$M_z = 0$
Service loads (S	<u>LS)</u>		
lc1-SLS	N = 0	M <sub>y</sub> = 49 kNm	$M_z = 0$
lc2-SLS	N = -3 kN	M <sub>v</sub> = 24 kNm	$M_{7} = 0$



# Solution with Cross Section Analysis & Design

#### **Opening a file from disk**

First of all we click on the File menu and select Open in order to open the file we created in Example 7.

#### Unlock the model

Afterwards, if the model is locked, we click on Edit -> Unlock model, in order to modify the geometry of the cross section.



In the popup window, we choose Yes to unlock the model.

Unlock	
2	Unlocking the file will result in the loss of all analysis results! Do you want to continue?
	<u>Y</u> es <u>N</u> o



#### **Review of Analysis Parameters**

We need to modify the Analysis Parameters sets "ULS" and "SLS" to match with the design data provided above. This can be done by clicking Analysis -> Analysis Parameters.

First of all, we select "ULS" from the list at the top left corner.

💐 Analysis parameters	8 ×
ULS   Add new Analysis Parameters	set Delete current set
Name ULS Notes Default Analysis Parameter	ers Set for Ultimate Limit State
Reinforced concrete data Other Materials data	
Apply R/C code defaults Concrete Data Stress Strain Curve Compression Part Default rectangular by code for ULS Compression Part Compression Pa	Amplification factors due to confinement         Rectangular stress strain curve for         concrete cannot be applied to confined         concrete, thus confinement will be ignored         Code specific values (ACI 318 05)         phi Tension controlled       0.9         phi Compression controlled       0.65
Reinforcement Data Stress - Strain Curve Linear-Constant E 199.95 GPa	ress - Strain Limits ] Tensile stress limit: 0 MPa ] Tensile strain limit: 0
esu 0.02	Compressive stress limit:     0     MPa       Compressive Strain limit:     0
OK Cance	4

We make sure that the compressive stress strain part of concrete is set to "Default rectangular by code for ULS". Moreover, the "No Tension" should be selected for the tensile part of concrete. The Stress/Strain limits for reinforcement are not changed as shown in picture above.





Next, we select "SLS" from the list at the top left corner.

šLS 🔻	Add new Analysis Paramet	ers set 🛛 🗶 Delete current set	
ame SLS No	Default Analysis Para	meters Set for Serviceability Limit	State
leinforced concrete data Other Materials data	a		
Concrete Data Stress Strain Curve Compression Part Parabolic-Constant	Apply R/C code defaults           ecu         -0.003           ec2         -0.002           λ         1	Amplification factors due to Concrete strength factor ec2 factor ecu factor Code specific values (ACI 3	confinement 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1
Tension Part		phi Compression controlled	
Reinforcement Data Stress - Strain Curve Ignore com	pression part	Stress - Strain Limits	
Linear-Constant 🔹		Tensile stress limit:	220 MPa
E 199.95 GPa esu 0.02 fyp	· · · · · · · · · · · · · · · · · · ·	Tensile strain limit:     Compressive stress limit:	0 MPa
E.	ε <sub>su</sub> ε <sub>s</sub>	Compressive Strain limit:	0
(	ОК Са	ncel	

According to the given design data, we make sure that a Parabolic-Constant concrete curve has been selected. The tensile part should have the option "Linear and Drop to Zero" selected, in order to consider the tensile resistance of concrete. Finally, we have applied a tensile stress limit for reinforcement equal to 220 MPa.

We notice that the strength reduction factors are in this case equal to 1.0, since the program automatically adjusts them to be compatible to current design situation SLS. On the other hand, the default values by ACI 318 were applied for the ULS case. We click ok to close this form.





#### **Definition of load cases**

We click on the Analysis -> Reinforcement design -> Load cases menu item to enter the load cases.

	👋 Load	cases for re	inforcement	design		? ×			
	Load cases								
		Name	Axial force [kN]	Moment about y [kNm]	Moment about z [kNm]	Analysis Parameters			
		lc 1	0	-250	0	ULS 🔻			
		lc 2	0	20	0	ULS 🔻			
		lc 3	15	-45	0	ULS 🔻			
	Þ	lc 4	-7	-65	0	ULS 🔻			
		2							
l		Delete curre	ent load case	Canc					

First of all, we select all the rows and click the  $\times$  icon to delete them. Afterwards, we enter the load cases and assign the corresponding Analysis Parameters sets, as shown below.

	Name	Axial force [kN]	Moment about y [kNm]	Moment about z [kNm]	Analysis Parameter	5		
	Ic 1-ULS	0	-140	0	ULS	Ŧ		
	Ic 2-ULS	0	55	0	ULS	Ŧ		
	Ic 3-ULS	-10	-44	0	ULS	Ŧ		
	lc 1-SLS	0	49	0	SLS	Ŧ		
•	lc 2-SLS	-3	24	0	SLS	Ŧ		

#### Carry out the analysis

We just click Analysis -> Reinforcement design -> Analyze, to perform the analysis.



#### Results

The results can be obtained from the Analysis -> Reinforcement design -> Show results menu item.

-	Reinforce	ment design r	esults						•	9	2 📉 🗙
	Design info				Resu	ts for load c	ase: lc 1-U	LS			
1	Cross secti	on is adequa	te		Load	case Ic 1-l	JLS		-		
	Minimum ratio 0.0015				Neede	ed ratio 6.9	9713E-3	Needed ratio co	nsidering minimum	reinforcement 6	9713E-3
N	Maximum ratio 0.04 Provided ratio 7.602/E-3 Cross section is adequate for selected load case										
<b>-</b> :	Summary				Rein	forcement d	etails				
	Load	Needed ratio	Placed ratio	Notes	ID	y cord [m]	z cord [m]	As,needed [cm2]	Placed reinf.	As,placed [cm2]	-
		6.9/13E-3	7.6027E-3	ok	1	-0.085	-0.26	0.98885	Φ16	2.0106	
	IC 2-ULS	3.0109E-3	4.3304E-3		2	-0.028333	-0.26	0.98885	Φ16	2.0106	
		3.0159E-3	4.3304E-3	ok	3	0.028333	-0.26	0.98885	Φ16	2.0106	
		9.3219E-3	0.01064	ok	4	0.085	-0.26	0.98885	Φ16	2.0106	=
	IC 2-SLS	4.4922E-3	4.9218E-3	0K	5	-0.085	0.26	0.65923	Φ12	1.131	
					6	0	0.26	0.65923	Φ12	1.131	
					7	0.085	0.26	0.65923	Φ12	1.131	
					8	-0.085	-0.11	1.131	Φ12	1.131	
					9	-0.085	0.11	1.131	Φ12	1.131	
					10	0.085	-0.11	1 131	Φ12	1 131	Ŧ
				a	ose	]					

We note that the load case is adequate and we can see the placed reinforcement sizes.

Notice that the most unfavourable load case is Ic1-SLS, which refers to Serviceability Limit State. Although the loads for SLS are smaller than ULS, the reinforcement demand can apparently be greater, since the reinforcement stress was reduced to 220 MPa.

Furthermore, the reinforcement size constraints applied in Example 2 are considered in this design as well.

These can be verified for each load case. For instance for lc1-ULS, the needed reinforcement for rebars 1, 2, 3 and 4 is 0.99 cm<sup>2</sup>, whereas the reinforcement demand for rebars 5, 6 and 7 is 0.66 cm<sup>2</sup>. The size constraint is valid since 0.99 cm<sup>2</sup> =  $1.5 \times 0.66$  cm<sup>2</sup>.



# **Example 9**

# Check of user-defined reinforcement pattern in a composite cross section of a column

# Data

In this example we are going to create the geometry of a composite cross section, define rebars and check their capacity according to Eurocode 2.



Regulation: Eurocode 2

Concrete (cylinder) strength: 20 MPa

Yield stress of reinforcement steel: 500 MPa

Reinforcement cover to rebar center: 4 cm

Steel grade: Fe 360

Tensile concrete strength is ignored.

Stirrup type: Tied

All remaining data (safety factors, other coefficients etc.) to be taken according to Eurocode 2.





## Load cases

lc1:	N = -1000 kN	M <sub>y</sub> = 550 kNm	M <sub>z</sub> = 123
lc2:	N = -770 kN	M <sub>y</sub> = 140 kNm	M <sub>z</sub> = -365

# **Solution with Cross Section Analysis & Design**

First of all define the corresponding Reinforced Concrete Code, by clicking on the Project -> Reinforced Concrete Code menu item.

#### **Selection of Reinforced Concrete Code**

Reinforced Concrete Regulation								
Reinforced concrete regulation								
EUROCODE 2 2004								
Reinforcement ratio limits								
Default by code for columns								
O User values								
User defined values								
Minimum ratio 0.008								
Maximum ratio 0.04								
OK Cancel								

Selection of Eurocode 2 regulation

The reinforcement ratio limits are to be calculated according to Eurocode 2, so we keep the default option checked.



#### **Definition of material properties**

Next, we are going to specify the material properties.

💐 Concrete	? 🔀
Available concrete materials	Concrete
Default Concrete	Name Default Concrete Color
	Notes Default concrete material, compr. strength=15 MPa
	Concrete strength 15 MPa
	Confined
	User defined elasticity modulus
All available materials	🕂 Add new 🛛 🕂 From library 🔀 Delete
Default Bilinear Material Default Concrete Default Linear Material Default Parabolic Material Default Reinforcement Default Trililinear Material	
	OK Cancel

The Default Concrete material can be overridden by changing its properties, as shown in the form below.

Concrete	2 ×
Available concrete materials	Concrete
C20	Name C20 Color
	Notes Concret,compr. strength: 20 MPa
	Concrete strength 20 MPa
	Confined
	User defined elasticity modulus
All available materials	🕂 Add new 🛛 🕂 From library 🔀 Delete
C20 Default Bilinear Material Default Linear Material Default Parabolic Material Default Reinforcement Default Trililinear Material	
	OK Cancel

Value "Concrete strength" has been set equal to 20 MPa. The concrete is not assumed confined and the elasticity modulus is to be calculated according to Eurocode 2, so we do



not need to change anything else. The concrete defined here is now accessible through the name "C20". Alternatively we could click the "Add new" button in order to define a new concrete material.

The reinforcement can respectively be defined from the Material -> Reinforcement menu item.

Reinforcement	1000000	? x
Available reinforcement materials	Reinforcement	
S500	Name S500	Color
	Notes Reinforcement materi	ial, Yield stress: 500 MPa
	Yield stress 500 M	IPa
	🕂 Add new 🕹 From	library 🔀 Delete
All available materials		
C20 Default Bilinear Material Default Linear Material Default Parabolic Material Default Trililinear Material S500		
	ОК	Cancel

We modify the Default Reinforcement material by specifying a new name (S500) and a yield stress of 500 MPa.





Finally we need to define the grade of the steel Fe 360. This steel grade can be imported from the library of the program by clicking on the Materials -> Import from Library menu item. From the top list we select "Hot Rolled Steel – European (no hardening)" in order to view the materials belonging to this collection.

Material type Hot Rolled Steel - European (no hardeding) Hot Rolled Steel - Australian (with hardening) Hot Rolled Steel - Australian (no hardening) Hot Rolled Steel - British (with hardening) Hot Rolled Steel - British (no hardening)	
Hot Rolled Steel - Canadian (with hardening) Hot Rolled Steel - Canadian (no hardening) Hot Rolled Steel - Canadian (no hardening) Hot Rolled Steel - European (no hardening) Hot Rolled Steel - Indian (with hardening) Hot Rolled Steel - Indian (no hardening) Hot Rolled Steel - Indian (no hardening) Hot Rolled Steel - US (with hardening) Hot Rolled Steel - US (no hardening)	otes rolled steel rolled steel rolled steel ot rolled steel ot rolled steel
Cold Formed Steel - US Aluminum Timber - EN 338 Solid wood Timber - EN 1194 Glulam Timber - South African pine	Cancel

Then, we can select the first row (Fe 360) and click on "Insert selected only" button to use this material in our project.





Fe 360 steel has been imported in our project. We can view/change its properties by clicking on it from the list of the main form, as shown in the picture below.



y= 0.409m , z= -0.456m Units: Default Metric 🗸

Alternatively, as Fe 360 steel is as bilinear material, it can be edited from the Materials -> Bilinear Materials menu item.

🟺 Bilinear materials					? ×		
Available bilinear materials	Material	data		)	Stress-Strain curve		
Default Bilinear Material	Name	Fe 360			Stress-Strain curve		
Fe 360	Notes	Fe 360 Hot rolle	d steel				
	E1 pos	210	GPa		200		
	ey pos	0.001119	]		100		
	E2 pos	0	GPa				
	e max	0.2					
All available materials	E1 neg	210	GPa		-100		
Default Bilinear Material	ey neg	-0.001119	]		-200		
Default Parabolic Material	E2 neg	0	GPa	Is symmetric	-300		
Fe 360	e min	-0.2	]	✓ can take tension	-0.3 -0.2 -0.1 0 0.1 0.2 0.3 Strain		
3500	Color			✓ can take compression	Convert to editable		
	📃 Allo	w strain values o	utside the	e defined curve	custom defined material		
Add new X Delete							
				OK Cancel			

In this form, the stress/strain curve of Fe 360 hot rolled steel is represented. We do not need to change anything and click OK.





#### **Drawing the geometry**

We are now ready to draw the geometry of the cross section, by clicking the Draw -> Circle (2Points).

Circle (2 Points)					
Center point y 0 m z 0	m				
Point on circle y 0 m z 0	m 🞑				
Material C20 -					
Cancel					

First of all we make sure that material "C20" has been selected in the shown form.

Then, we can specify the center point of the circle by entering its coordinates (0,0) and then clicking the 🎑 button, or just by clicking on the point (0,0) with the mouse.

After that we need to define one point on the circle. This can be done by clicking at the point (0.30,0).









So, the circle is now defined and we can see it in the drawing area.



66





We can modify its properties by moving the mouse over it, right clicking and selecting "Properties" from the corresponding menu.



In this way, the "Circle" form is shown and we can view/change its data.

Circle (2 Points)
y m z 0 m
Point on circle y 0.3 m z 0 m
Material C20 💌
Hole OK Cancel





Afterwards, we can insert a hot rolled section from the library by clicking the Draw -> Standard Section menu item.

Standard section						
		Preview				
Sections from library	European steel shapes (mm)	ee				
Use custom section	Aluminum shapes (in) Aluminum shapes 6061-T6 (in)	Ι Ύ				
Section type	AISC shapes v3 (in) AISC shapes v13 (in)					
I wide flange	AISC13 (mm)					
Channel	AISCLRFD2 (in)	o <sup>CG</sup> o o				
© Tee	AISCLRFD3 (in) Australian and New Zealand steel shapes v8 (mm)					
Angle	British Standards steel shapes (mm) British Standards steel shapes 2006 (mm)					
─ Box	Chinese steel shapes (mm)					
Pipe	Indian steel shapes (mm) Indian steel shapes (mm) User defined sections					
- Section dimensions		Place at point				
Section height 0.3	9 m Fillet radius 0.027 m	У 0 m z 0 m				
Flange width 0.3	m	Materials C20 -				
Flange thickness 0.0	<b>19</b> m					
Web thickness 0.0	11 m	Insert section Close				

In this form, the collections of all sections of the library are represented. In order to insert a HEA-400 cross section, we select "European steel shapes" from the top list and then we choose the "HE400A" shape from the list below.

💐 Standard section		? ×
		Preview
<ul> <li>Sections from libra</li> </ul>	European steel shapes (mm)	• • • •
Use custom section	on dimensions	Ύ Ι
Section type	Select HE400A	
I wide flange		
Channel		0 <b>CG</b> 0
© Tee	Section name HE400A	
Angle		
© Box	The insertion point of the shape	
Pipe	is shown in red and can change	
- Section dimensions -	by clicking on other possible	Place at point
Section height	points indicated in blue.	У 0 m z 0 m
Flange width	.3 m	Materials C20
Flange thickness	.019 m	
Web thickness 0	. <b>011</b> m	Insert section Close

The insertion point of the shape is selected to coincide with its centroid.



We should not forget to change the material for this section to Fe 360, as illustrated below.

👋 Standard section									? X
<ul> <li>Sections from library</li> <li>Use custom section of</li> </ul>	European steel sl limensions	hapes (mm)		•	Preview	ę	ſ		P
Section type I wide flange	Select HE40	0A	•			_	CG		
<ul> <li>Channel</li> <li>Tee</li> <li>Angle</li> </ul>	Section name	HE400A				Ŭ			Ŭ
<ul><li>Box</li><li>Pipe</li></ul>						6			
Section dimensions Section height 0.39	m	Fillet radius	0.027	m	Place at У 0	point r	n z	0	m
Flange width 0.3	m				Materials	Fe 360 C20	1	- 4-1	•
Web thickness 0.01	1 m					Default B Default L Default P Default T Fe 360	inear Mate arabolic M inlinear Ma	rial aterial aterial	

We click OK and see that the steel shape is shown in the main drawing area.





Furthermore, we need to draw the rebars, which will have a circular pattern.

But first let's see if rebar sizes are defined. We select Project -> Available Rebars.

	Rebars			? X
Γ	Name 06	Diameter 6	mm Mc	odify current
	Rebars			
	Name	[	Diameter [mm]	
	Φ6	6		
	Φ8	8		-
	Φ10	10		-
	Φ12	12		
	Φ14	14		
	Φ16	16		
	Φ18	18		
	Φ20	20		
	Lan	22		
	👍 Add new	+ From library	X Delete	current
		OK Car	ncel	

The rebars shown above are available in the project. This means that we can draw rebars of these diameters and additionally the program will only choose from these rebar sizes when performing a reinforcement design.

Now we are ready to draw the reinforcement bars by clicking on the Draw -> Rebar Circle menu item.



We can enter a value (0.05 m) for the reinforcement cover to the rebar center in the field, which becomes active when inserting reinforcement bars, as shown below. The program automatically draws help points at the specified distance from the concrete edges and enables mouse snapping.



To draw the rebars, we click on (0,0) point and then on one of the available green points.

Rebar circle					
y 0 m z 0 m					
Other point y 0.25 m z 0 m	(+ · · + ·				
Offset direction Outside Olinside	offset				
Rebar count 8	Offset 0 m				
Constant size for design Design param Grow factor	eters 1				
Material S500    Reinforcement Φ20					
Split to single rebars					
OK Cancel					





We change the rebar count to 8 and also check that the reinforcement material is "S500" and choose a rebar size, ø20.

After clicking the "OK" button, the lower rebars are shown in the drawing area.






-	Worked	ovamn	00
-	VVUIKEU	exump	CO

#### **Review of Analysis Parameters**

We need to specify the corresponding Analysis Parameters set by clicking Analysis -> Analysis Parameters. We will use the default "ULS" set. All parameters related to Eurocode can be found in this form. The concrete stress block is rectangular according to Eurocode 2 parameters, the tensile resistance of concrete is ignored and all remaining data have the default values. You can override these values by modifying any of the field of the form.

Analysis parameters	? ×
ULS   Add new Analysis Parameters	set Delete current set
Name ULS Notes Default Analysis Parameter	ers Set for Ultimate Limit State
Reinforced concrete data Other Materials data	
Concrete Data Apply R/C code defaults Stress Strain Curve	Amplification factors due to confinement
Default rectangular by code for ULS       ✓         σ <sub>c</sub> All values as per selected R/C code	Rectangular stress strain curve for concrete cannot be applied to confined concrete, thus confinement will be ignored
λf <sub>c</sub> and design)	Code specific values (EUROCODE 2 2004) Design situation Accidental  Persistent/Transient
	gama c 1.5
Tension Part	acc 1
No Tension 🔻	act 1
Reinforcement Data Stress - Strain Curve Ignore compression part St	ress - Strain Limits
Unear-Constant ▼	Tensile stress limit: 0 MPa
E 200 GPa fy	Tensile strain limit: 0
esu u.uz	Compressive stress limit:     0     MPa       Compressive Strain limit:     0
OK Cance	8



•	Worked	example	PC
	<b>WOINCU</b>	слатр	

#### **Definition of load cases**

The load cases can be selected by clicking on the Analysis -> Reinforcement check -> Load cases menu item. The 2 load cases can be entered in the corresponding table. A new load case can be inserted by clicking the + button. We make sure that the assigned Analysis Parameters set to each load case is ULS.

📛 Load	cases for i	reinforcement	check	-	? ×
- Load ca	ases				
	Name	Axial force [kN]	Moment about y [kNm]	Moment about z [kNm]	Analysis Parameters
•	lc 1	-1000	550	123	ULS 🚽
	lc 2	-770	140	-365	ULS 🗸
+ >	<	ОК	Can	cel	È) 🗗 💼

#### **Reinforcement check**

We just click Analysis -> Reinforcement check -> Analyze, to perform the reinforcement design procedure.



#### **Results**

The results can be obtained from the Analysis -> Reinforcement check -> Show results menu item.

As we can see, the cross section is adequate for both load cases. This form reports the placed reinforcement ratio (9.42 %) which lies between the allowable limits (2% – 4%) calculated according to Eurocode 2 specifications.

Reinforcement ch	eck resu	sults ? X				
Reinforcement ratio pla	iced 9.4	).4185E-3				
Minimum	ratio 0.0	0.002				
Summary						
Load Capa	city ratio	Notes				
lc 1 0.616		ok				
lc 2 0.767		ok				
		Close				





# Example 10

# Creation of interaction diagram for a composite cross section of a column

## Data

We will use the file created in Example 9 and create the interaction diagram according to Eurocode 2.

All design data are to be taken from Example 9.

## **Interaction diagram data**

<u>Case 1:</u>	Moment about Y axis vs. Moment about Z axis for zero axial force
<u>Case 2:</u>	Moment about Y axis vs. Moment about Z axis for axial force values -1000 kN (compression) and 400 kN (tension)
<u>Case 3:</u>	Moment about Y axis vs. Axial force for reinforcement ratio values: 0.5%, 1.5% and 2.5%

## **Solution with Cross Section Analysis & Design**

### **Opening a file from disk**

First of all we click on the File menu and select Open in order to open the file we created in Example 9.





#### Unlock the model

Afterwards, if the model is locked, we click on Edit -> Unlock model, in order to modify the geometry of the cross section.



In the popup window, we choose Yes to unlock the model.





#### **Review of Analysis Parameters**

We do not need to modify the previously created Analysis Parameters set "ULS". We can view the assigned properties by clicking Analysis -> Analysis Parameters and selecting "ULS" from the list at the top left corner.

Analysis parameters		? <mark>x</mark>
ULS   Add new Analysis Parameters	set Delete current set	
Name ULS Notes Default Analysis Parameter	ters Set for Ultimate Limit State	* *
Reinforced concrete data Other Materials data		
Concrete Data	Amplification factors due to confine	ement
Stress Strain Curve Compression Part Default rectangular by code for ULS → arectangular stress block is only valid for resistance analysis! (Interaction, Beinforcement check	Rectangular stress strain curve for concrete cannot be applied to con concrete, thus confinement will be	fined ignored
λf <sub>c</sub> and design)	Code specific values (EUROCODE Design situation Carteria Accidental Persistent/Tra	E 2 2004) Insient
	gama c 1.5	
Tension Part	gama s 1.15	
No Tension 👻	act 1	
Reinforcement Data Stress - Strain Curve Ignore compression part St	tress - Strain Limits	
Linear-Constant	Tensile stress limit:	MPa
E 200 GPa esu 0.02 fyp	Tensile strain limit:	
Es Esu Es	Compressive stress limit: Compressive Strain limit:	MPa
ОК Сапсе	el	

We click OK, as we do not need to change anything.



#### **Definition of load cases**

The load cases can be selected by clicking on the Analysis -> Interaction (as per selected R/C code) -> Load cases menu item.

#### Case 1

We click on the + button to add a new load case. We change the name to "Case 1" and select "Moment about y – Moment about z" option. The assigned Analysis Parameters set should be "ULS".

📛 Load	cases for I	nteraction as per R/C code		1.4.4		? <mark>X</mark>
Load ca	ses					
	Name	Interaction type		Analysis Parameters		Options
•	Case 1	Moment about y - Moment about z	-	ULS	-	Options
+ 🗙		OK	ncel		Ŀ	) 🗈 🔒



Case 2

We click on the + button to add the next load case and change the name to "Case 2". We make sure that the option "Moment about y – Moment about z" is selected and the Analysis Parameters set is "ULS". Then we click on "Options" to specify the axial force levels for the interaction diagram.

👋 Options for In	teraction		? ×
Load case name Interaction Type Variations No alteration Alternate Axi Alternate Re	Case 2 Moment y vs. Moment z al Force inforcement Ratio	Axial force - Axial force Add	-1000 ON Modify Remove
ОК	Cancel		

On the form above, we select "Alternate Axial Force" in the "Variations" box on the left. Next, we enter at the top right corner the values -1000 and click on "Add" button. We repeat this step for the value 400.



In this way we request the interaction diagram to be produced for both specified axial force levels. We click OK to close the form.

Eng SSol



#### Case 3

As shown before, we click on the + button to add the last load case "Case 3". We make sure that the option "Axial force - Moment about y" is selected and the Analysis Parameters set is "ULS". Then we click on "Options" to specify the reinforcement ratio values for the interaction diagram.

We select the "Alternate Reinforcement Ratio" option and provide at the top right corner the values of 0.005, 0.015 and 0.025.

🏺 Options for Int	teraction	? ×
Load case name Case 3 Interaction Type Moment y vs. Axial Force Variations No alteration Alternate Axial Force Alternate Reinforcement Ratio		Reinforcement ratios         Reinforcement ratio         0.025         Add         Modify         Remove         0.025         0.015         0.005
ОК	Cancel	

We click ok to close the form.





The 3 load cases are now defined.

闄 Loa	ad cases for	Interaction as per R/C code		1.4.4		? X
Load	cases					
	Name	Interaction type		Analysis Parameters		Options
	Case 1	Moment about y - Moment about z	•	ULS	•	Options
	Case 2	Moment about y - Moment about z	-	ULS	•	Options
•	Case 3	Axial force - Moment about y	•	ULS	•	Options
+	×				Þ	) 🗅 💼
		OK	cel			

#### Carry out the analysis

We just click Analysis -> Interaction (as per selected R/C code) -> Analyze, to perform the analysis.



#### Results

The requested interaction diagrams can be obtained by clicking click Analysis -> Interaction (as per selected R/C code) -> Show results.

#### Interaction diagram for Case 1:

We choose "Case 1" from the list at the top right corner of the form. The interaction diagram for Moment about Y vs. Moment about Z is shown below.

		Selected curve data	
rrent point -196.57	kNm Mz -481.54 kNm	Color	
		Moment about y - Moment about z (Mrd,y - Mrdz as j	per EUROCODE 2 2004) - Case 1
800			
600 -	+		
400 -	-		
-	+		
[mNx] (z'p-w)			
Moment about	-		
	-		
-400 -			
-600 -			
-800 - -15	500 -1000		FRNm1
-400 - -600 - -800 - -15	000 -1000	-500 0 Moment about y (Mrd,y)	500 1000







#### Interaction diagram for Case 2:

By choosing "Case 2", appears the same interaction diagram, but for different axial force levels.







#### Interaction diagram for Case 3:

Respectively, by selecting "Case 3", we can display the interaction diagram in terms of Moment about Y vs. Axial force for three different reinforcement ratio values (0.5%, 1.5% and 2.5%).







# Example 11

# Cross sectional properties of a built-up steel section

## Data

In this example, we will create a built-up steel section and calculate its properties.

Steel grade: A36 Gr.36



## **Solution with Cross Section Analysis & Design**

### Setting Reinforced Concrete Code to None

Since the cross section does not have a concrete part, we can optionally set the Reinforced Concrete code to "None", in order to hide all data related to reinforced concrete analysis. We click on Project -> Reinforced Concrete code menu item and select "None" as below.

Reinforced Concrete Regulation
Reinforced concrete regulation
None
OK Cancel





#### **Definition of material properties**

Next, we are going to specify the material properties.

The A36 Gr.36 steel grade can be imported from the library of the program by clicking on the Materials -> Import from Library menu item. From the top list we select "Hot Rolled Steel – US (no hardening)" in order to view the materials belonging to this collection.

Haterials library	? ×
Material type         Hot Rolled Steel - US (no hardening)         Hot Rolled Steel - Australian (with hardening)         Hot Rolled Steel - Australian (no hardening)         Hot Rolled Steel - British (with hardening)         Hot Rolled Steel - British (no hardening)         Hot Rolled Steel - British (no hardening)         Hot Rolled Steel - British (no hardening)         Hot Rolled Steel - Canadian (no hardening)         Hot Rolled Steel - Canadian (no hardening)         Hot Rolled Steel - European (with hardening)         Hot Rolled Steel - European (no hardening)         Hot Rolled Steel - Indian (with hardening)         Hot Rolled Steel - Indian (no hardening)         Hot Rolled Steel - Indian (no hardening)         Hot Rolled Steel - US (with hardening)         Hot Rolled Steel - US (no hardening)         Hot Rolled Steel - US         Aluminum         Timber - EN 338 Solid wood         Timber - South African pine	Notes Hot rolled steel i0 Hot rolled steel rolled steel 2 Hot rolled steel 6 Hot rolled steel
Insert whole material set Insert selected only	Cancel







Then, we can select the first row (A36 Gr.36) and click on "Insert selected only" button to use this material in our project.

💐 Materials li	ibrary			? 🗙	
Material type Hot Rolled Si	Material type Hot Rolled Steel - US (no hardening)				
Available mate	enais imeUl	Туре	Notes		
A36 Gr.36	Bilinear		A36 Gr.36 Hot rolled steel		
A572 Gr.42	Bilinear		A572 Gr.42 Hot rolled stee	9	
A572 Gr.50	Bilinear		A572 Gr.50 Hot rolled stee	1	
A572 Gr.55	Bilinear		A572 Gr.55 Hot rolled stee		
A572 Gr.60	Bilinear		A572 Gr.60 Hot rolled stee	sl .	
A572 Gr.65	Bilinear		A572 Gr.65 Hot rolled stee	*	
A588 Gr.B	Bilinear		A588 Gr.B Hot rolled steel		
Insert	whole material set	Insert selected	d only Cancel		



88





A36 Gr.36 steel has been imported in our project. We can view/change its properties by clicking on it from the list of the main form, as shown in the picture below.



#### Drawing the geometry

First we will insert a standard wide flange section from the library by clicking the Draw -> Standard Section menu item. We select the AISC13 library in the form below.

		S X
		Preview
<ul> <li>Sections from library</li> </ul>	AISC13 (mm)	٩٩
Use custom section	Aluminum shapes (in) Aluminum shapes 6061-T6 (in) AISC shapes v3 (in)	Υ Υ
Section type	AISC shapes v13 (in)	
<ul> <li>I wide flange</li> </ul>	AISCI 3 (mm) AISCI RED1 (in)	
Channel	AISCLRFD2 (in)	o <sup>CO</sup> o
© Tee	AISCLRFD3 (in) Australian and New Zealand steel shapes v8 (mm) Patish Standarda steel shapes (mm)	
Angle	British Standards steel shapes (mm) British Standards steel shapes 2006 (mm)	
© Box	Chinese steel shapes (mm) European steel shapes (mm)	I I I I I I I I I I I I I I I I I I I
Pipe	Indian steel shapes (mm) User defined sections	
Section dimensions		Place at point
Section height 0.0	m Fillet radius 0.007 m	У 0 m z 0 m
Flange width 0.0	55 m	Materials A36 Gr.36
Flange thickness 0.0	047 m	
Web thickness 0.00	<b>136</b> m	Insert section Close





After selecting AISC13 shape family, all sections belonging to it, appear in the list below.

👋 Standard section		5 ×
Sections from library AISC1	13 (mm) 🔹	Preview
Use custom section dimensi     Section type		
<ul> <li>I wide flange</li> <li>Channel</li> </ul>	va ₩360X64	o cg o
Tee Sect	tion name W360X64	
<ul> <li>Angle</li> <li>Box</li> </ul>		
Pipe     Section dimensions		Place at point
Section height 0.348	m Fillet radius 0.0149 m	У 0 m z 0 m
Flange width 0.203	m	Materials A36 Gr.36
Hange thickness 0.0135	m	
0.00773		Insert section Close

We select a W360X64 cross section.

Handard section		5 ×
Sections from library	AISC13 (mm)	Preview
Use custom section of     Section type     I wide flange	Select W360X64	
<ul><li>○ Channel</li><li>○ Tee</li></ul>	Section name W360X64	o CG o
<ul><li>Angle</li><li>Box</li></ul>		
Pipe     Section dimensions	/	Place at point
Section height 0.34	<sup>8</sup> The insertion point of the shape	У 0 m z 0 m
Flange thickness	by clicking on other possible	laterials A36 Gr.36 🔹
Web thickness 0.00	points indicated in blue.	Insert section Close
L		

The insertion point of the shape is selected to coincide with its centroid.

We should not forget to check that the assigned material is A36 Gr.36.





We click OK and see that the steel shape is shown in the main drawing area.



Next, we will insert a Tee section. We click again on the Draw -> Standard Section menu item and select the AISC13 library. We click on Tee section type, select WT155X26 and change the "Place at point" y value to 0.2 m.

Standard section	8 ×
Sections from library AISC13 (mm)     Use custom section dimensions	Preview CG
Section type Select WT155X26	
Channel  Tee Section name WT155X26	o 0
⊘ Angle ⊘ Box	
© Pipe	o • o
Section dimensions Section height 0.159 m Fillet radius 0.0076 m	Place at point y 0.2 m z 0 m
Flange width 0.167 m	Materials A36 Gr.36
Flange thickness 0.0132 m	
	Insert section Close

Finally we click on "Insert section" button.



Now we need to rotate the Tee  $90^{\circ}$  clockwise. This can be done by right clicking on it and then selecting "Rotate".









The program asks for the center point of rotation, as the status bar shows.



93





So we click on point (0.2,-0.15). Then, in order to enter the base point for rotation, we click on point (0.2,0.1). No we can move the mouse to rotate the cross section interactively.





After aligning it horizontally, we can click anywhere with the mouse to fix the shape in this position. Following this action, we delete the initial section by right-clicking on it and selecting "Delete".



#Right click -> Properties to edit geometry

y= 0.206m , z= 0.0246m Units: Default Metric +





Now we are about to move the Tee section next to the W section.

First, we uncheck the "Snap to grid" option, from the toolbar at the bottom of the screen.



In this way, we disable the "Snap to nodes" function during editing geometry. On the other hand, the "Snap to points" option should be checked in order to enable snapping to the remaining characteristic points of the section parts except for the grid points.

In order to move the T section, we right-click on right and select "Move".



Then we click at the middle point of the Tee web and move it to the middle web point of the W section, as shown below.







By left-clicking on that point, the T section is fixed in this position.

Next we will mirror the T section about a vertical axis. This can be accomplished by right clicking on it and choosing "Mirror".

The program asks for the starting point of the mirror axis (see Status bar). We enable the snap option again by clicking on the "Snap to grid" item in the lower toolbar and successively click on points (0,-0.25) and (0,0.25).

👋 Cross Section Analysis & Design - 0 **X** File Edit View Project Materials Draw Analysis Settings Help 1 🗟 🗟 Ø 🚍 Information User grid points Project 3 ര 0 \* Units ÷ Default Metric 1 1 10 Ð, Reinforced Concrete Code ۹ Not specified 1 ~ Defined Materials ÷k Name 2 Туре A36 Gr.36 Bilinear Bilinear -6 Default Bilinear Material Default Linear Material Default Parabolic Material Default Trililinear Material Parabolic Trilinea alysis parameters Notes ameters Set for Ultimate Limit Name Default Analysis Parame ULS State Default Analysis Paramet Limit State ers Set for Ser LS Load cases Name Ic 1 Adva Туре Analyzed Aves Grid Ruler Draw cross User points Dim lines Labels Reinforcement rebar Nr. Snap to grid Snap to points Snap to Mid points Delet after mirroring/rotating Grid distance 0.05 m y= -0.243m , z= -0.189m Units: Default Metric +

The geometry is now defined and the drawing screen should look like this.

#### **Calculate sectional properties**

We just click Analysis -> Inertia Data -> Analyze, to have the sectional properties computed.





#### Results

The calculated data can be found by clicking on Analysis -> Inertia Data -> Show results menu item.







# Example 12

# Creation of interaction diagram for a built-up steel section

## Data

In this example we will calculate the interaction diagram of the built-up section created in Example 11.

## Interaction diagram data

- <u>Case 1:</u> Moment about Y axis vs. Axial force
- <u>Case 2:</u> Moment about Y axis vs. Moment about Z axis for axial force values -750 kN (compression) and 900 kN (tension)

## **Solution with Cross Section Analysis & Design**

#### **Opening a file from disk**

First of all we click on the File menu and select Open in order to open the file we created in Example 11.





#### Unlock the model

Afterwards, if the model is locked, we click on Edit -> Unlock model, in order to modify the geometry of the cross section.



In the popup window, we choose Yes to unlock the model.





v.	Worked	examn	lec
	<b>WORKCU</b>	слатр	<b>C</b> 3

#### **Review of Analysis Parameters**

In order to review the Analysis Parameters, we click on the Analysis -> Analysis Parameters menu item. The reinforced concrete related parameters are hidden, since no concrete regulation has been selected. We can select a material from the list on the left and modify the existing preferences.

Hanalysis parameters	? ×
ULS	Add new Analysis Parameters set     X Delete current set
Name ULS	Notes Default Analysis Parameters Set for Ultimate Limit State
Other Materials data	
A36 Gr.36 Default Bilinear Material Default Linear Material Default Parabolic Material Default Trililinear Material	Material data Safety factor Partial safety factor 1
	Use whole stress-strain curve as defined
	Stop when reaching maximum or minimum material stress
	Stop when reaching a tensile stress value of: 0 MPa
	Stop when reaching a compressivet stress value of: 0 MPa
	Stop when reaching a tensile strain of:
	Stop when reaching a compressive strain of: 0
	Stop when reaching a tensile strain value of: 0 on a fiber at distance: 0 m , from the fiber with the maximum tensile stain
	Stop when reaching a compressive strain value of: 0 on a fiber at distance: 0 m , from the fiber with the maximum compressive stain
	OK Cancel

We do not change anything and click OK.



#### **Definition of load cases**

We will use the advanced interaction analysis in this example. The other option for interaction analysis (as per selected R/C code) is available only when a reinforced concrete code has been specified.

The load cases can be selected by clicking on the Analysis -> Interaction (advanced) -> Load cases menu item.

#### Case 1

We click on the + button to add a new load case. We change the name to "Case 1" and select "Axial force - Moment about y" option. The assigned Analysis Parameters set should be "ULS".

👋 Loa	ad cases for	Advanced Interaction				? X
Load	cases					
	Name	Interaction type		Analysis Parameters		Options
1	lc 1	Axial force - Moment about y	-	ULS	-	Options
+	×	ОК	Cancel		Ē	🗅 💼





Case 2

We click on the + button to add the next load case and change the name to "Case 2". We make sure that the option "Moment about y – Moment about z" is selected and the Analysis Parameters set is "ULS". Then we click on "Options" to specify the axial force levels for the interaction diagram.

💐 Options for I	nteraction		? 🔀
Load case name Interaction Type Variations No alteration Alternate Axi	Case 2 Moment y vs. Moment z al Force inforcement Ratio	Axial force Axial force	·750 kN y Remove
ОК	Cancel		

On the above form, we select "Alternate Axial Force" in the "Variations" box on the left. Next, we enter at the top right corner the values -750 and click on "Add" button. We repeat this step for the value 900.

Load case name Case 2 Interaction Type Moment y vs. Moment z Variations No alteration Alternate Axial Force Alternate Reinforcement Ratio	xial force vial force 900 kN Add Modify Remove 00 750

In this way we request the interaction diagram to be produced for both specified axial force levels. We click OK to close the form.

Eng SSol





The 2 load cases are now defined.

	🟺 Load		? <mark>X</mark>						
	Load cases								
		Name	Interaction type		Analysis Parameters		Options		
		Case 1	Axial force - Moment about y	•	ULS	•	Options		
	1	Case 2	Moment about y - Moment about z	•	ULS	•	Options		
	+ >	<				P			
L				cel		C	opy selected v		

#### Carry out the analysis

We just click Analysis -> Interaction (advanced) -> Analyze, to perform the analysis.



#### Results

The requested interaction diagrams can be obtained by clicking click Analysis -> Interaction (advanced) -> Show results.

#### Interaction diagram for Case 1:

We choose "Case 1" from the list at the top right corner of the form. The interaction diagram for Moment about Y vs. Axial force is shown.







#### Interaction diagram for Case 2:

By choosing "Case 2", the corresponding interaction diagrams in terms of Moment about Y axis vs. Moment about Z are shown.





## Example 13

# Estimation of maximum and minimum developed stresses on a built-up steel section under given external loads

## Data

In this example we will calculate the stresses developed in the cross section discussed in Example 11 and create corresponding stress contours.

The partial safety factor for steel is assumed to be 1.0.

## **Load cases**

lc1:	N = 500 kN	M <sub>y</sub> = 150 kNm	M <sub>z</sub> = 100 kNm
lc2:	N = 0	M <sub>y</sub> = -250 kNm	M <sub>z</sub> = 50 kNm
lc3:	N = -200 kN	M <sub>y</sub> = 0 kNm	M <sub>z</sub> = -147 kNm

## **Solution with Cross Section Analysis & Design**

#### **Opening a file from disk**

First of all we click on the File menu and select Open in order to open the file we created in Example 11.

#### Unlock the model

Afterwards, if the model is locked, we click on Edit -> Unlock model, in order to modify the geometry of the cross section.

In the popup window, we choose Yes to unlock the model.

Unlock	
?	Unlocking the file will result in the loss of all analysis results! Do you want to continue?
	<u>Y</u> es <u>N</u> o


-	Markad	avere a la a	
	vvorkea	examples	

# **Review of Analysis Parameters**

The partial safety factor of the steel A36 Gr.36 is set to 1, so we do not need to change anything in the form below. We click OK to close the form.

Hanalysis parameters		? <mark>X</mark>
ULS	✓ Add new Analysis Parameters set	
Name ULS	Notes Default Analysis Parameters Set for Ultimate Limit State	4 7
Other Materials data		
A36 Gr.36 Default Bilinear Material Default Linear Material Default Parabolic Material	Material data Safety factor Partial safety factor 1	
Default Thillinear Material	☑ Use whole stress-strain curve as defined	
	Advanced parameters     Stop when reaching maximum or minimum material stress	
	Stop when reaching a tensile stress value of: 0 MPa	
	Stop when reaching a compressivet stress value of: 0 MPa	
	Stop when reaching a tensile strain of:	
	Stop when reaching a compressive strain of:	
	Stop when reaching a tensile strain value of: O on a fiber at distance: O m , from the fiber with the maximum tensile sta	in
	Stop when reaching a compressive strain value of: O on a fiber at distance: O m , from the fiber with the maximum compressi	ve stain
L	OK Cancel	



# **Definition of load cases**

First we will define the load cases by clicking on the Analysis -> Deformed configuration -> Load cases menu item.

The load cases can be defined by successively clicking on the + button. We have to make sure that the "ULS" Analysis Parameters set have been assigned to each load case.

Load cases for deformed configuration						
	Name	Axial force [kN]	Moment about y [kNm]	Moment about z [kNm]	Analysis Parameters	
•	lc 1	500	150	100	ULS 🔻	-
	lc 2	0	-250	50	ULS 🗸	-
	lc 3	-200	0	-147	ULS 🗸	·
+ 2	<	ОК	Canc	el	<u>)</u> (1	

# Carry out the analysis

We just click Analysis -> Deformed configuration -> Analyze, to perform the analysis.



#### Results

The requested interaction diagrams can be obtained by clicking click Analysis -> Interaction (advanced) -> Show results.

#### Results for load case lc1:

We choose "lc1" from the list at the top right corner of the form. The corresponding strain distribution and stress contour are shown. Moreover, information about the maximum/minimum stresses developed on the section can be found on the right.







# Results for load case lc2:

Respectively, by selecting "lc2", appears the corresponding information for the second load case.







# Results for load case Ic3:

Respectively, by selecting "lc3", appears the corresponding information for the second load case.



As we can see, the maximum stress is developed for load case "lc2" and has a value of 248.21 MPa. The minimum stress is -248.21 MPa for "lc2" as well.

Since this stress is equal to the yield of A36 Gr.36, we can conclude that some parts of the cross section have entered the plastic region.



# **Example 14**

# Check of a timber cross section

# Data

In this example, we are about to draw a timber cross section and check its adequacy under given external loads.

Wood grade: C20 solid wood, according to EN 338

Partial safety factor for solid timber: 1.3



# **Load cases**

lc1:	N = 5 kN	$M_y = 14 \text{ kNm}$	$M_z = 0 \text{ kNm}$
lc2:	N = 0	M <sub>y</sub> = -15 kNm	M <sub>z</sub> = 0 kNm
lc3:	N = -5 kN	$M_y = 4$	M <sub>z</sub> = 3 kNm



# Solution with Cross Section Analysis & Design

#### **Setting Reinforced Concrete Code to None**

Since the cross section does not have a concrete part, we can optionally set the Reinforced Concrete code to "None", in order to hide all data related to reinforced concrete analysis. We click on Project -> Reinforced Concrete code menu item and select "None" as below.

;	Reinforced Concrete Regulation
	Reinforced concrete regulation
	None
	OK Cancel

# **Definition of material properties**

Next, we are going to specify the material properties.

The C20 solid wood grade can be imported from the library of the program by clicking on the Materials -> Import from Library menu item. From the top list we select "Timber – EN 338 Solid wood" in order to view the materials belonging to this collection.

Hot Rolled Steel - Canadian Hot Rolled Steel - Europea Hot Rolled Steel - Europea Hot Rolled Steel - Indian (w Hot Rolled Steel - Indian (n	o hardening) n (with hardening) n (no hardening) n (with hardening) n (no hardeding) vith hardening) no hardening)		ites	
Hot Rolled Steel - Indian (w Hot Rolled Steel - Indian (n Hot Rolled Steel - US (with Hot Rolled Steel - US (no h - Cold Formed Steel Cold Formed Steel - US		- =		
Timber - EN 338 Solid woo Timber - EN 1194 Glulam - Timber - South African pine				
C35	Linear	C35 Timber	,	
C40	Linear	C40 Timber		
C45	Linear	C45 Timber		-
C50	Linear			



Then, we can select the property C20 and click on "Insert selected only" button to use this material in our project.

Materials library	00048	2	x
Material type			
Timber - EN 338 Solid wood		•	
Available materials			
NameUI	Туре	Notes	*
C14	Linear	C14 Timber	
C16	Linear	C16 Timber	
C18	Linear	C18 Timber	
C20	Linear	C20 Timber	=
C22	Linear	C22 Timber	
C24	Linear	C24 Timber	
C27	Linear	C27 Timber	
C30	Linear	C30 Timber	
C35	Linear	C35 Timber	
C40	Linear	C40 Timber	
C45	Linear	C45 Timber	
C50	Linear	C50 Timber	Ŧ
Insert whole material se	et Insert selec	ted only Cancel	





C20 wood has been imported in our project. We can view/change its properties by clicking on it from the list of the main form, as shown in the picture below.



#### Drawing the geometry

First we will draw the web of the Tee section. To do this, we click on Draw -> Rectangle using dimensions from the top menu.

The center point of the rectangular section can be inserted by entering its coordinates (0,0) and then clicking the subtron, or just by clicking on the point (0,0) with the mouse. Then the Length and Width values should be set to 0.08 and 0.15 respectively. Finally, the material C20 should be selected for the rectangle we are drawing.

Rectangle (using dimensions)					
Center point y 0 m z 0 m					
Dimensions Length 0.08 m Width 0.15 m					
Material C20 -					
Hole     OK     Cancel					







The web should now appear in the drawing area.



Next, we can draw the top flange by using the same command (Draw -> Rectangle using dimensions).

On the forms that shows, we enter the coordinates of the top flange center (0,0.105) and click the  $\swarrow$  button. Then we specify the values for length and width and click OK.

Rectangle (using dimensions)					
Center point y 0 m z 0.105 m					
Dimensions Length 0.2 m Width 0.06 m					
Material C20 💌					
Hole					
OK Cancel					







The cross section is shown in the drawing screen as follows.



Finally, we repeat the last action to draw the bottom flange.

Rectangle (using dimensions)					
Center point y 0 m z -0.105 m					
Dimensions Length 0.15 m Width 0.06 m					
Material C20 -					
Hole					
OK Cancel					





## After clicking OK, we can see the final geometry of the cross section in the drawing area.





# **Review of Analysis Parameters**

In order to review the Analysis Parameters, we click on the Analysis -> Analysis Parameters menu item. The reinforced concrete related parameters are hidden, since no concrete regulation has been selected.

While the "ULS" set is active, we select the C20 wood grade from the list on the left and change the partial safety factor to 1.3 as shown in the picture below.

💐 Analysis parameters	? ×
ULS	Add new Analysis Parameters set
Name ULS	Notes Default Analysis Parameters Set for Ultimate Limit State
Other Materials data	
C20 Default Bilinear Material Default Linear Material Default Parabolic Material Default Trililinear Material	Material data Safety factor Partial safety factor 1.3
	Use whole stress-strain curve as defined
	Stop when reaching maximum or minimum material stress
	Stop when reaching a tensile stress value of: 0 MPa
	Stop when reaching a compressivet stress value of: 0 MPa
	Stop when reaching a tensile strain of: 0
	Stop when reaching a compressive strain of:
	Stop when reaching a tensile strain value of: on a fiber at distance: 0 m , from the fiber with the maximum tensile stain
	Stop when reaching a compressive strain value of: 0 on a fiber at distance: 0 m , from the fiber with the maximum compressive stain
	OK Cancel

We click OK to close the form.



# **Definition of load cases**

We will first define the load cases by clicking on the Analysis -> Deformed configuration -> Load cases menu item.

The load cases can be defined by successively clicking on the + button. We have to make sure that the "ULS" Analysis Parameters set have been assigned to each load case.

	Load cases for deformed configuration						
ſ	Name	Axial force [kN]	Moment about y [kNm]	Moment about z [kNm]	Analysis Parameter	s	
	le 1	5	14	0	ULS	•	
	lc 2	0	-15	0	ULS	•	
	lc 3	-5	4	-3	ULS	•	
6		ОК	Canc	el	) <b>(</b>	ł	

# Carry out the analysis

We just click Analysis -> Deformed configuration -> Analyze, to perform the analysis.



## Results

The calculated data can be found by clicking on Analysis -> Deformed configuration -> Show results menu item.

# Results for load case lc1:

We choose "lc1" from the list at the top right corner of the form. The corresponding strain distribution and stress contour are shown. Moreover, information about the maximum/minimum stresses developed on the section can be found on the right.



Design for load case lc1:adequate

Maximum stress: 6.95 MPa at point (0.1,0.135)

Minimum stress: -7.89 MPa at point (-0.075,-0.135)





# Results for load case Ic2:

Respectively, by selecting "lc2", appears the corresponding information for the second load case.



# Design for load case lc2:adequate

Maximum stress: 8.43 MPa at point (-0.075,-0.135)

Minimum stress: -7.31 MPa at point (0.1,0.135)





# Results for load case Ic3:

Respectively, by selecting "lc3", appears the corresponding information for the second load case.

	eformed configuration results		l	8 X				
Loa	d case llc.3							
Netral axis location and deformed configuration								
		Neutral axis information		D D				
		Property	Value	Units				
n.	0.15	Strain at origin	-4.1291E-5					
-3-	L.1 0.1	Curvature	6.8308E-3					
T	0.05	NA angle	5.0331	rad				
		Eqv. tensile Force	29.937	kN				
		Tensile force lever arm	0.075851	m				
		Eqv. compr. Force	-34.938	kN				
		Compr. force lever arm	-0.052536	m				
	0.35							
		L						
	- 485 - 0.75 - 0.65 - 0.55 - 0.45 - 0.25 - 0.15 - 0.05 0 0.05 0 0.05 0 0.25 0 0.25 0.2 0.25 0.2 0.25 0.5 0.5 0.5 0.5 0.5 0.7 0.75 0.8 0.85 0.9 0.95 1 13	Material stress information	n					
	Stress distribution	Material C20	-					
		Property	Value	Units				
		Max stress 6.	56 MPa					
		y cord at max stress 0.	1 m					
	rain: 7.183E-4	z cord at max stress 0.	135 m					
	Stress: 5.2491 MPa 3.4252	Min stress -5	.9792 MPa					
	-0.05 D.05 -	y cord at min stress -0	.075 m					
	1.8578	z cord at min stress -0	.135 m					
	-1277 -							
	0.5							
	-2.8444							
	0.14.41180.1	Analysis info						
	-5.9792	Cross section is ade	quate					
	-0.3 -0.25 -0.2 -0.15 -0.1 -0.05 0 0.05 0.1 0.15 0.2 0.25 0.3		Close					
Axes	Grid Ruler Dim lines Labels Reinforcement rebar Nr.							

# Design for load case lc3:adequate

Maximum stress: 6.56 MPa at point (0.1,0.135)

Minimum stress: -5.98 MPa at point (-0.075,-0.135)



# Example 15

# Steel jacketing for improvement of column strength and ductility

# Data

In this example we are going to assess the behavior of a concrete column, after it is retrofitted with a 5mm thick metal jacket. So, we will create the Moment vs. Curvature curves before and after the jacket application.

Concrete grade: C16/20, according to Eurocode 2

Reinforcement grade: S400, according to Eurocode 2

Concrete stress/strain curve: according to Eurocode

Steel jacket grade: Fe 430, according to Eurocode 3 with a safety factor of 1.15

Confinement options for jacketed column

Compressive strength: 21 MPa

Compressive strain at maximum stress: -3‰

Ultimate strain:

-7‰







# **Solution with Cross Section Analysis & Design**

First of all define the corresponding Reinforced Concrete Code, by clicking on the Project -> Reinforced Concrete Code menu item.

Reinforced Concrete Regulation							
Reinforced concrete regulation							
EUROCODE 2 2004							
Reinforcement ratio limits							
Default by code for columns							
O User values							
User defined values							
Minimum ratio 0.008							
Maximum ratio 0.04							
OK Cancel							

Selection of Reinforced Concrete Code

Selection of Eurocode 2 regulation

# **Definition of material properties**

Eng SSol

Next, we are going to specify the material properties.

Concrete can be specified by selecting Materials -> Concrete

💐 Concrete	2 🔀
Available concrete materials	Concrete
Default Concrete	Name       Default Concrete       Color         Notes       Default concrete material, compr. strength=15 MPa         Concrete strength       15       MPa
	User defined elasticity modulus
All available materials Default Bilinear Material Default Concrete Default Linear Material Default Parabolic Material Default Reinforcement Default Trililinear Material	🕂 Add new 🕂 From library 🗙 Delete
	OK Cancel



We click on "From library" button to import an existing concrete material from the database of the program.

European	•
Available concrete materials	
NameUI	fc,cyl [MPa]
C12/15	12
C16/20	16
C20/25	20
C25/30	25
C30/37	30
C35/45	35
C40/50	40
C45/55	45
C50/60	50

After choosing the European standard, we select the C16/20 material and click the "Insert selected only" button.

📛 Concrete	
Available concrete materials C16/20 Default Concrete	Concrete Name C16/20 Color Notes
	Confined User defined elasticity modulus
All available materials C16/20 Default Bilinear Material Default Concrete Default Linear Material Default Parabolic Material Default Reinforcement Default Trililinear Material	Add new From library X Delete

Since the material has been defined, we click OK to close the form.



The reinforcement can respectively be defined from the Material -> Reinforcement menu item. As before, we click the "From library" button to select from the existing predefined reinforcement materials.

Reinforcement grade library	? <mark>×</mark>
Standard European	•
Available reinforcement materials	6/ [MP=1
S220	220
S250	250
S300	300
S400	400
S460	460
S500	500
Insert whole reinforcement material set	Insert selected only Cancel

We choose the S400 grade from the European standard and click on "Insert selected only".

*	Cross Section Analysis & Design



Reinforcement	? ×
Available reinforcement materials	Reinforcement
Default Reinforcement	Name S400 Color
5400	Notes
	Yield stress 400 MPa
	🕂 Add new 🛛 👫 From library
All available materials	
C16/20 Default Bilinear Material Default Concrete Default Linear Material Default Parabolic Material Default Reinforcement Default Trililinear Material S400	OK

The reinforcement grade S400 is now available in the project.





# Drawing the geometry

We are now ready to draw the geometry of the cross section, by clicking the Draw -> Rectangle using dimensions. The center point of the rectangular section can be inserted by entering its coordinates (0,0) and then clicking the button, or just by clicking on the point (0,0) with the mouse. Then the Length and Width values should both be set to 0.40. Finally, the C16/20 concrete should be selected for the rectangle we are drawing.









After we have clicked the "OK" button, the rectangle is shown in the drawing area.



Before drawing the rebars, we have to make sure that a proper rebar set is available for the design. This can be checked by selecting Project -> Available Rebars.

Name 🚾	Diameter 6	mm Mo	dify current
Rebars			
Name	e	Diameter [mm]	A
Φ6	6		
Φ8	8		_
Φ10	10		-
Φ12	12		
Ф14	14		
Ф16	16		
Φ18	18		
Φ20	20		
	22		
🕂 Add new	+ From library	X Delete	current



The rebars that are shown above are available in the project. This means that we can draw rebars of these diameters and additionally the program will only choose from these rebar sizes when performing a reinforcement design.

So, we can now draw the reinforcement bars by clicking on the Draw -> Rebar Rectangle menu item.

We can enter a value (0.04 m) for the reinforcement cover to the rebar center in the field, which becomes active when inserting reinforcement bars, as shown below. The program automatically draws help lines at the specified distance from the concrete rectangle edges and enables mouse snapping at their intersections.







To draw the bottom rebars, we click on the green points at the top left and at the bottom right successively.





Next, as shown on the form below, we choose to use  $\emptyset$ 20 and  $\emptyset$ 16 rebar sizes for the corner and remaining rebars respectively. The rebar count (apart from the rebars at the corners) horizontally and vertically is set to 2.

Rebai rectangie						
First comer						
y -0.16 m z 0.16 m						
Opposite comer						
y 0.16 m z -0.16 m						
Offset direction						
Place rebars at corners Offset 0 m						
Rebar count horizontally 2 Rebar count vertically 2						
Constant size for design Design parameters Grow factor for comer rebars						
Grow factor for other rebars 1						
Comer rebars reinforcement Φ20						
Other rebars reinforcement 016						
Material S400   Salary Split to single rebars						
OK Cancel						

Moreover, the S400 reinforcement material is selected.





After clicking the "OK" button, the rebars are shown in the drawing area.





#### Behavior assessment of the cross section

First we are going to assess the behavior of the cross section by calculating a Moment vs. Curvature diagram for different axial force levels.

#### **Review of Analysis Parameters**

Eng SSol

We need to specify the corresponding Analysis Parameters set by clicking Analysis -> Analysis Parameters. We will use the default "ULS" set.

Analysis parameters		8 ×
ULS   Add new Analysis Parameters	set X Delete current set	
Name ULS Notes Default Analysis Parameter	ers Set for Ultimate Limit State	A T
Reinforced concrete data Other Materials data		
Apply R/C code defaults	]	
Concrete Data	-Amplification factors due to	confinement
Compression Part	Concrete strength factor	1
Parabolic-Linear ecu -0.0035	ec2 factor	1
ec2 -0.002	ecu factor	1
λf <sub>c</sub> c 0.85	Code specific values (EURO	DCODE 2 2004)
λ 0.85	Accidental  Persiste	ent/Transient
ε <sub>c2</sub> ε <sub>c</sub> ε <sub>c</sub>	gama c	1.5
Tension Part	gama s	1.15
No Tracico -	асс	1
No Tension	act	1
Reinforcement Data Stress - Strain Curve	tress - Strain Limits	
Bilinear ▼	Tensile stress limit:	) MPa
E 200 GPa hrfy	Tensile strain limit:	D
esu 0.02 Tyd	Compressive stress limit:	) MPa
Hf 1.15	Compressive Strain limit:	D
OK Cance	el	

The option "Default rectangular by code for ULS" for the compressive parts of concrete stress/strain curve cannot be used, since we are about to perform a Moment Curvature analysis. "Default rectangular by code for ULS" option is only valid for designing or checking the capacity of a cross section.

So we choose the option "Parabolic-Linear". The strain data have already been set by the program to their default values according to Eurocode 2.

All parameters related to Eurocode 2 can be found in this form, including partial safety factors etc.



The reinforcement bars are chosen to have a bilinear behavior. A hardening of 1.15 is assumed.

Additionally, we check the "Ignore compression part" in order to ignore the contribution of the reinforcement to the sectional resistance, i.e. steel reinforcement will only be active if it is under tension. This assumption is common, especially when retrofitting damaged concrete elements, as the sparse stirrups will not provide a lateral support to vertical reinforcement bars. Consequently their compressive resistance cannot be used due to buckling.

We click OK to close this form.

# Definition of load cases

The load cases for Moment vs. Curvature analysis can be selected by clicking on the Analysis -> Moment Curvature -> Load cases menu item. By clicking the + button we can add a new load case. The Moment curvature type is set to "Moment about y", the Custom Angle (coordinate system rotation) remains 0 and the Analysis Parameters item is changed to "ULS" in order to use the parameters defined previously.

📛 Load	d cases for I	Moment Curvature					? x
Load c	ases						
	Name	Moment curvature typ	be	Custom Angle [deg]	Analysis Parameters		Options
1	lc 1	Moment about y	•	0	ULS	-	Options
+ ?	<		_			Ŀ	
		ОК		Cancel			





In order to use specific axial forces for the analysis, we click on "Options" button and as shown on the form below, we select the option "Alternate Axial Force".

Coptions for Moment Curvature				
Load case name Interaction Type Variations No alteration Alternate Axi Alternate Re	Ic 1 Curvature y vs. Moment y al Force inforcement Ratio	Axial force Axial force Add Mod	ify Remove	
Bilinearization of Second branch	Moment-Curvature curve slope 0 Cancel			

Next we enter the axial force values successively in the corresponding field at the top right corner of the form and click the "Add" button, so that all axial forces have been filled, as shown below.

Load case name Interaction Type   Interaction Type Curvature y vs. Moment y   Variations Add   No alteration   Attemate Axial Force   Altemate Reinforcement Ratio   Bilinearization of Moment-Curvature curve   Second branch slope

We can also define the slope of the second branch of the equivalent bilinearized curve here, but we keep this value equal to 0. This means that the calculated bilinearized curve will be elastic-fully plastic. We click OK to close the form.





# Carry out the analysis

We just click Analysis -> Moment Curvature -> Analyze, to perform the reinforcement design procedure.

# **Results**

The calculated Moment – Curvature curves for the specified axial force levels can be found by clicking Analysis -> Moment Curvature -> Show results.





Behavior assessment after the application of steel jacket

# Definition of a steel material

The steel grade of the jacket is assumed to be Fe 430. This steel grade can be imported from the library by clicking on the Materials -> Import from Library menu item. From the top list we select "Hot Rolled Steel – European (no hardening)" in order view the materials belonging to this collection.

Waterials library				
Material type Hot Rolled Steel - European (no hardeding)				
Hot Rolled Steel - Australian (with hardening) Hot Rolled Steel - Australian (no hardening) Hot Rolled Steel - British (with hardening) Hot Rolled Steel - British (no hardening) Hot Rolled Steel - Canadian (with hardening) Hot Rolled Steel - Canadian (no hardening)	Notes ot rolled steel			
Hot Rolled Steel - European (with hardening) - Hot Rolled Steel - European (no hardeding) Hot Rolled Steel - Indian (with hardening) Hot Rolled Steel - Indian (no hardening)	ot rolled steel			
Hot Rolled Steel - US (with hardening) Hot Rolled Steel - US (no hardening) Cold Formed Steel	Hot rolled steel Hot rolled steel			
Cold Formed Steel - US Aluminum Timber - EN 338 Solid wood Timber - EN 1194 Glulam Timber - South African pine				
Insert whole material set Insert selected only	Cancel			

Then, we can select the corresponding row (Fe 430) and click on "Insert selected only" button to use this material in our project.



# Drawing the steel jacket

Eng SSol

The simplest way to draw the steel jacket geometry is to use the Draw -> Polygon from line offset.

Polygon from line offset		
- Line points		
y [m] z [m]		
	Method	
	<b>∕</b> ,,,,	
	<b>1</b>	
	0111	
Material Fe 430	~	
Thickness 0.005 m	🗌 Hole	
OK Cancel		

In the form that appears, we select the third option in the box "Method". This will enable the drawing of the steel jacketing by just providing a path of points which are the 4 corners of the cross section. The specified thickness is set to 5 mm (0.005m) and current material is Fe 430.



After successively clicking on the 4 corners of the cross section, we right-click at any point and choose "Close polygon" from the menu as shown above.



The retrofitted column section has been now defined. We can continue by reviewing the used Analysis Parameters set "ULS".

# Informing the program that the concrete is confined

This can be easily carried out by clicking the Materials -> Concrete menu item and checking the "Confined" option for C16/20 concrete grade, as shown below.

💐 Concrete	? 🗙 ?
Available concrete materials	Concrete
C16/20 Default Concrete	Name       C16/20       Color         Notes
All available materials C16/20 Default Bilinear Material Default Concrete Default Linear Material Default Parabolic Material Default Reinforcement Default Trililinear Material Fe 430 S400	Add new From library X Delete




In this way, the program will handle the C16/20 concrete as defined, i.e. the confinement factors specified in Analysis Parameters will be applied to this material.

## **Review of Analysis Parameters**

The Analysis Parameters set "ULS" will be modified in order to simulate the effect of steel jacketing on the behavior of the section.

Since the existing concrete is assumed confined, we can define the related amplification factors for its strength, the strain that corresponds to its strength and its ultimate strain. These values can be calculated according to any regulation or theoretical method and can be applied in the program.



## Example of confined and unconfined concrete stress/strain curves

In this example we will assume that the confined concrete stress is 21 MPa, with a corresponding strain of -3‰ and its ultimate strain is -7‰.

Consequently, the amplification factors can be calculated as follows:

•	Concrete strength factor:	21 MPa / 16 MPa	=	1.31
•	ec2 factor:	3‰ / 2‰	=	1.5
•	ecu factor:	7‰ / 3.5‰	=	2





🖡 Analysis parameters		?					
ULS	Add new Analysis Paramet	ters set Delete current set					
Name ULS Default Analysis Parameters Set for Ultimate Limit State							
Reinforced concrete data Other Materials data							
	Apply R/C code defaults	s					
Concrete Data		Amplification factors due to confinement					
Compression Part		Concrete strength factor 1.31					
Parabolic-Linear	ecu -0.0035	ec2 factor 1.5					
	ec2 -0.002	ecu factor 2					
λf	0.85	Code specific values (EUROCODE 2 2004)					
cλfc		Design situation					
	ð 0.85	Accidental  Persistent/Transient					
Ec2	Ecu Ec	gama c 1.5					
C Tension Part		gama s 1.15					
No Tonsion		acc 1					
	•	act 1					
Reinforcement Data							
Stress - Strain Limits							
		Tensile stress limit: 0 MPa					
E 200 GPa	Nf Ty	Tensile strain limit: 0					
esu 0.02	fyg	Compressive stress limit: 0 MPa					
hf 1.15	Es	Compressive Strain limit:					
	E su Es						
OK Cancel							

All remaining data regarding concrete and reinforcement do not need to be modified.



Furthermore, in tab "Other Materials data", we can define a partial safety factor for the steel jacketing, which is taken as 1.15.

💐 Analysis parameters 🛛 👔 🔀							
ULS	Add new Analysis Parameters set Delete current set						
Name ULS Notes Default Analysis Parameters Set for Ultimate Limit State							
Default Bilinear Material Default Linear Material Default Parabolic Material Default Trililinear Material Fe 430	Material data Safety factor Partial safety factor Use whole stress-strain curve as defined						
	Advanced parameters           Stop when reaching maximum or minimum material stress           Stop when reaching a tensile stress value of:         0         MPa						
	Stop when reaching a compressivet stress value of: 0 MPa						
	Stop when reaching a tensile strain of:       0         Stop when reaching a compressive strain of:       0						
	Stop when reaching a tensile strain value of:       0         on a fiber at distance:       m       , from the fiber with the maximum tensile stain						
	Stop when reaching a compressive strain value of: O on a fiber at distance: M , from the fiber with the maximum compressive	: stain					
OK Cancel							

We click OK to close this form.

## Carry out the analysis of the retrofitted section

We just click Analysis -> Moment Curvature -> Analyze, to perform the reinforcement design procedure.



## **Results**

Eng SSol

The calculated Moment – Curvature curves for the specified axial force levels can be found by clicking Analysis -> Moment Curvature -> Show results.



As expected, after the application of the steel jacket, the cross section behavior in terms of stiffness, resistance and ductility has dramatically been improved.

By clicking the "Export current graph data", we can export the points of selected diagram to a text file and use them in other applications, such as Excel. In the following graph, the Moment vs. Curvature curves before and after the retrofit are shown for the same axial force load (-500 kN).

