

RC column

Introduction

This tutorial shows a design of RC column of a hexagonal cross-section. With 200mm depth of the section and 2000mm length, the column is subjected to axial compressive force and biaxial bending. The actions in the ultimate limit state are: N_x = 400kN, M_y = 2.33kNm and M_z = 5.46kNm. The stress limitation (serviceability limit state) should be checked for N_x = 350kN and M_y = 2.00kNm. The strength class of the concrete is C30/37 X0 and steel grade B500 is used for reinforcement.

Starting a new project

The following screen appears after running program "Concrete":

1 FIN EC - Concrete (Untitled.c3e)			×
Ele Edit Massimputs Data Option	s ∐elp		
Add section	General project	deta	
Add member	📌 Edit	Date : 19. 9. 2016	~
X Renove			
Project			~
	Standard		
	ef gåt	Standard EN 1992-1-1/Czech Rep. Concrete capacity - basic load continuation : $1 \text{ C} = 1,500$ Reinforcement capacity - saccidential load continuation : $1 \text{ C} = 1,150$ Concrete capacity - accidential load continuation : $1 \text{ C} = 1,200$ Reinforcement capacity - accidential load continuation : $1 \text{ C} = 1,200$ Modulus of elasticity of concrete : $1 \text{ Re} = 1,200$ Concrete compressive strength : $1 \text{ Geo} = 1,000$ Plain concrete tende strength : $1 \text{ Geo} = 0,000$ Plain concrete tende strength : $1 \text{ Geo} = 0,000$ Plain concrete tende strength : $1 \text{ Geo} = 0,000$	د >
	Calculation opti Check bar s Check of de Calculate or Program option Calculate co Calculate co Use wized f	ons parting saling only informative (does not set check result) adv width only after exceeding concrete tensile strength a and minisure occentricity considered in the direction of the bending moment vector; Minimum eccentricity considered before budding 6 for new input	
🛨 Up 🔰 Đown			
Check al			
EN 1992-1-1/Czech Rep.			

The start screen of the "Concrete" program

The program provides opportunity to calculate a unlimited number of partial tasks per project. There are two task types supported by the software: "Section" and "Member". The type "Section" is suitable for easy verification of RC cross-sections, the type "Member" is usually used for a verification of the structures, which were created in the programs "Fin 2D" and "Fin 3D". We will use the type "Section" for our analysis. The start screen contains a part "General project data", where the job name, description and other project identification data can be entered. After clicking the "Edit" button, we first enter the job name and other project details:

General project o	lata			×
Job name:	Testing project	Author:	John Smith	~
Part:	Columns	Date:	19. 9. 2016	
Description:		Project ID:		
Client:	Concrete Structures Ltd.	Archive ID:		
Comment:				^
				~
				ancel



"General project data" dialog box

These data can be displayed in the header or footer of the final documentation.

Prior to commencing any work, it is advisable to save the job. This can be done either using "**I**" button, or in the main menu clicking on "**File**" – "**Save As**", or using the "**Ctrl+S**" shortcut.

🍞 Save As								×
$\leftrightarrow \rightarrow \land \uparrow$	« Fin	e > FIN EC 2017	Examples	~	Ū	Search FIN EC 20)17 Examples	P
Organise 👻 Ne	w folde	r					•== •	?
 This PC Desktop Documents Downloads Music Pictures Videos 	^	Name	^ 1.c3e		1	Date modified 19. 9. 2016 11:11	Size 2 Ki	В
File name: Save as type:	✓ Colum Files C	nn C1.c3e oncrete (*.c3e)						~
∧ Hide Folders						Save	Cancel	

Saving the project

Now we can proceed to entering a new task by clicking the "Add Section" button in the upper part of the program's tree menu.



Adding a new section

The following dialog box appears, in which we can enter the section's name ("**Column**") into the "**Section description**" field, confirming by clicking the "**OK**" button.



Add Section		×
Create new section		
O Copy existing section	n	
Section description:	Column C1	
Check type:	👺 3D	
	€ <u>o</u> k	⊠ <u>C</u> ancel

Dialog box for adding a new section

A new item has been generated in the tree menu, representing the new added section (**"Column C1**"). The program has now automatically selected this item; therefore we can directly proceed to entering the section parameters.

1 FIN EC - Concrete (CAUse	es\Public\D	acuments/Fine/FIN	EC 2017 Example	s)Column C1.							
Ele Edit Massimputs De	ta <u>O</u> ption	s <u>H</u> elp									
Add section		Check type:	B =	📲 Edi	t -	ULS check : SLS check :	NO CHE	ECK.			
Add member						Interaction d	lagran 30	Interaction diagram M	Ny-Mit Interaction	dagram N-I	
× Remove									E N :	0,00	140
Gaburna C1		- Section, Material, Member type Soli Section 1 no inter-	Reinforcement schland •	Sector	Þ 0	Loads E G D	Name	and on N Bit My	[k]M_2 (H[V_2 10 V_2)	ी भिल्म अ T (अब) प	tilas
		Longitudinal reinf. Transverse reinf. Environment :	: no input : no input X0	Denforce	nent 1922 rforcement						~
		Include reinf, in	compression			+ édd	-0 ² 5.61	X Baueros	III Import	E In	dotal
		Imperfection, Bud	ding			Resulta					
		Calculate buck	91 167 400 Y 🔽 Calculate [10] [10]	to a constant	[r] Buddro [r] [r]	Cross-sec Cross-sec Reinforce	tion form is r tion material ment materia	not set. Cannot run ca is not set. Cannot ru al is not set. Cannot ru	ikulation! n.calculation! an.calculation!		4
+ Up + Dove		Calculate grads	width only at uppe	s botton surta	÷						
Check all		IL Haxinun allowe	ed oradi: width		[1011]						4
FM 1002-1-1/Crach Ren											

Main screen for "Section" task type

Section, Material, Reinforcement

At first it is necessary to enter the basic geometrical and material characteristics of the section in the "**Section**, **Material**, **Reinforcement**" frame. We select the function of the member in the structure from the "**Member type**" drop-down list. Available types are "**beam**", "**slab**", "**column**" or "**wall**".

Member type	Select kind	Section
Section :	beam slab	Materia
Concrete : Longitudinal	column 📈	Reinfor

Choice of member type

In our example we select member type "**column**". This selection affects the analysis and verifications of the reinforcement arrangement.



As a hexagon is not included in the library of pre-defined cross-sections, we need to use the "D" button to define geometry of a general polygon. The polygon shape can be defined graphically or numerically in the window "Cross-section editor". The shape of cross-section is defined by six points, which have to be defined in the correct order. We can define each of the points numerically by clicking the "+" button, located in the bar in the left upper corner above the table of points.

C	cross-section	description	
name	Co	lumn	
comment			
	Cross-sec	tion geometry	30000

Button for input of points

A new window with input lines for enetring coordinates appears. We specify the coordinates of the forst point *[-0,058;0,100]* and insert the point by the button "**Add**".

Point position	
Y:	-58,0 fx [mm]
Z:	100,0 fx [mm]

Input of particular points of the polygon

We will specify positions of following points [-0,058;0,100], [0,058;0,100], [0,115;0,000], [0,058;-0,100], [-0,058;-0,00], [-0,00], [-0,00], [-0,00], [-0,00], [-0,00], [-0,00], [-0,00], [-0,00]

The geometry of the cross-section appears on the right side of the dialog box; we can change the points coordinates either directly in the table on the left side or graphically on the right side. We close the dialog box by clicking the "**OK**" button.



Cross-section	on editor - C	oncrete, gener	al po	olygo	n																×
Cr	ross-section d	escription			•		•	Ð I	04			0	+	£	<u>r</u> t	•			F	E)	Image: A start and a start
name	Colu	mn		_	1000				-	_											_
comment							-200	.0 		-100,0		0,0			100,0			20	0,0 		mm
• • •				0.0							1			2							
	Cross-secti	on geometry		° :							Ĭ			1							
	Y [mm]	Z [mm]									/			1							
> 1	-58,0	100,0	\sim							/	·				1						
2	58,0	100,0																			
3	115,0	0,0		8-						e S					Ę	5					
4	58,0	-100,0		:						1					/						
5	-58,0	-100,0													/						
6	-115,0	0,0												1	/						
				-100,0	N A	۰Y					ē			4							
			~	Sna	ap to g	rid [Grid	d ste	P	5,0	[mm]										
Informatio	n															Ŀ	OK	(X	Can	cel

Geometry of the polygon

We proceed with defining the material properties in "**Materials**" dialog box which is run by clicking the "**Material**" button in the "**Section**, **Materials**, **Reinforcement**" fram. Assuming the column is located inside of the structure we select "**X0**" for "**Exposure class**", as the column is not in contact with outside environment. Subsequently we define the material properties of concrete and longitudinal and transversal reinforcement. We can select standardized materials from the library of pre-defined materials clicking the "**Catalogue**" button at relevant lines.

Materials			×
Environment:	xo	+	Edit
Concrete:	No input	Catalogue	User defined
Longitudinal reinf.:	No input	Catalogue	User defined
Shear reinf.:	No input	Catalogue	User defined
- Indicative strength d	ass	10-	
Aeration > 4%			
C12/15 (EN 1992-1-	1)		
C12/15 (EN 206-1/Z4	4)		
Ductility class of longi	tudinal reinforcement	○ A ●	a ⊖c
		<u> </u>	🔀 <u>C</u> ancel

Window "Materials"

For concrete we select the strength class "C30/37" and close the dialog box by clicking "OK".



Materials catalogue - Concrete	×
C 12/15 C 16/20 C 20/25 C 25/30	
C 30/37	N
C 35/45 C 40/50 C 45/55 C 50/60 C 55/67 C 60/75 C 70/85 C 80/95 C 90/105	N
Information	C OK Cancel

Choice of strength class for concrete

Proceeding to definition of the steel properties, we select the grade "**B500**" for both longitudinal and transversal reinforcement and close the dialog box by clicking "**OK**".

Materials catalogue - Long	gitudinal reinf.	×
10505 (R) 10425 (V) KARI wire (W) Nets (SZ) B420		
B500	N	
B550	13	
Information	🗹 ОК	X Cancel

Steel grade selection

After returning to the "**Materials**" dialog box we can check the summary of the selected materials and confirm whether the selected class of concrete fulfils requirements on the "**Indicative strength class**" given by the selected exposure class. We exit the window "**Materials**" by clicking "**OK**".

Materials			×
Environment:	xo	4	Edit
Concrete:	C 30/37	Catalogue	User defined
Longitudinal reinf.:	B500	Catalogue	User defined
Shear reinf.:	B500	Catalogue	User defined
- Indicative strength da	ass		
Aeration > 4%			
C12/15 ⇒ strength d	ass pass (EN 1992-1-1)		
$C12/15 \Rightarrow strength data$	ass pass (EN 206-1/Z4)		
Ductility class of longit	udinal reinforcement	⊖ a (● B	⊖c
		€ ōk	X Cancel

Indicative strength class check in "Materials" window



Loads

After defining the geometry of the section and material properties, we can proceed either with defining the reinforcement or loads. In our example we first define a load case because then we are able to check the results of the reinforcement assessment during its definition. To create a load case, we click the "Add" button located under the "Loads" table.

Shear reinforcement		
	+ Add	↓ <u>E</u> dit
	- Results	

Button for insertion of new loads

In the "**New load**" window we select a "**Combination type**". This type should be selected according to the type of combination, which was used for determination of forces and moments. This input affects the type of verification. The following options are available:

Basic design (ULS)	•Forces and bending moments have been obtained from the basic combination for persistent and transient design situations according to EN 1990, equations <i>6.10</i> resp. <i>6.10a</i> and <i>6.10b</i> . These loads are used for basic assessment of cross-section's capacity in the ultimate limit state.
Accidental design (ULS)	•Forces and bending moments have been obtained from the combination for accidental design situations according to EN 1990, equation <i>6.11</i> . These loads are used for assessment of cross-section's capacity in accidental design situations in the ultimate limit state (partial safety and material factors for accidental design situations are used).
Characteristic (SLS)	•Forces and bending moments have been obtained from the characteristic combination according to EN 1990, equation <i>6.14</i> . These loads are used for assessment of the stress limitation (serviceability limit state).
Quasi- permanent (SLS)	•Forces and bending moments have been obtained from the quasi-permanent combination according to EN 1990, equation <i>6.16</i> . These loads are used for assessment of the crack widths in the serviceability limit state.

Subsequently we enter the forces and bending moments acting on the cross-section; in our example the axial force is N= -400kN (negative value denotes compression) and the bending moments are M_v = 2,33kNm and M_z = 5,46kNm.

Also, we should enter the **"Load duration coefficient"**, i.e. the ratio of quasi-permanent and total loads for calculation of the creep coefficient. If the exact value is not available, we can leave the conservative value of *1.00*, which means that the total load is considered quasi-permanent in the calculations. The new load is confirmed by clicking the **"Add"** button and **"Cancel"** to exit the dialog box.



Load							
Load 1	-						
Combination type :		basic design (ULS)					
Forces calculated using 2nd order 1		basic design (ULS)					
		characteristic (SLS) quasi-permanent (SLS)					
Axial force:	N =	-400,00	[kN]	N > 0 : tension ; N < 0 : compress			
Bending moment:	M _y =	2,33	[kNm]	$M_y > 0$: bottom fibres in tension			
Bending moment:	M _z =	5,40	[kNm]	$M_z > 0$: left fibres in tension			
Shear force:	V _z =	0,00	[kN]	V _z :↓↑			
Shear force:	V _y =	0,00	[kN]	$v_{\gamma} \colon \leftrightarrow$			
Torsional moment:	T =	0,00	[kNm]				
Load duration coefficient							
Load duration coefficient:		1,000	[-]				
Represents ratio of quasi- from 0 to 1; 1 means that coefficient	permanent (: quasi-perma	SLS) and desig nent and desig	in (ULS) in load a	load by bending moment, values rang re equal; used for calculation of cree			

Input of new load

To define the loads for the serviceability limit state, analogical procedure is used as for the ultimate limit state loads. To assess the stress limitation, we select "Combination type - characteristic (SLS)" and enter the relevant axial compressive force N = -350kN and the bending moment $M_v = 2.00kNm$. After input confirmation by the button "Add", we will exit the window by using the button "Cancel".

As result, a table summarizing all defined load cases is generated in the dialog box.

	I- 1								1
P A Name	2nd orc	N [KN]	V _z [kN]	V _y [kN]	M _y [kNm]	M _z [kNm]	T [kNm]	Utilization	
1 Load 1 - basic design (ULS)		-400,00	0,00	0,00	2,33	5,46	0,00		
2 Load 2 - quasi-permanent (SL	5)	-350,00			2,33	5,46			
2 Load 2 - quasi-permanent (SL	5)	-350,00			2,33	5,46			_
2 Load 2 - quasi-permanent (SL	5)	-350,00			2,33	5,46			
Add	5)	-350,00			2,33	5,46		In a	letail

The number of loads is not limited in the software. The input can be done also in a batch using text or *.csv file (button "Import").

Reinforcement

After returning to the main dialog box, we can proceed to defining the longitudinal and transverse reinforcement. We open the dialog box "Edit reinforcement sector" for longitudinal reinforcement definition by clicking on the "Reinforcement" button in the "Section, Material, Reinforcement" frame. The upper part of the window contains an option to select a calculation method of the cover. We keep the method ""



In this dialog box, we can define the reinforcement either numerically in the table on the left-hand side or graphically in the

right part of the dialog box.We can also take advantage of a simplified approach by clicking "Bear and then the "Generate" button in the left lower corner of the dialog box. In our example we will use this option.

🕞 Edit reinforcement sector	
General reinf.	
	🖲 🕙 😑 🔄 16,00 [mm] 🔍 🔍 🖓 🕂 🛄 💭 🔀 🏹 [+ 📫 📥
	+z
[] []	ĺ. I I I I I I I I I I I I I I I I I I I
	>+y
T	
<u>A</u> dd	
Cover	Information on reinf.
Minimum cover Minimum cover	Use the same coordinate system as for cross-section
Min cover and stirrups	
Stirrups diameter: [mm]	
Cover: 20,0 [mm]	
Generate Import Drawing	Utilization by bending : NO CALCULATION

Simplified definition of reinforcement

We can now easily define required longitudinal reinforcement using *16mm* bars in each corner of the cross-section, entering 3 layers with appropriate covers:



Edit rein	forcement									×
Cover										
Mini	mum cover			Stirrups	diameter:	10,00	[mm]	Minim	num cover	
 Min Cus 	cover and sti tom cover	rrups		Cover:		30,0	[mm]			
Upper	reinforcemen	t								
	Diameter [mm]	Type Input	Distance [mm]	Count [-]	Cov Autom.	er [mm]	A _s [mm ²]			
V 1	16,00	Number 💌		2		30,0	402,1	^		
V 2	16,00	Number 💌		2		90,0	402,1]		$\langle \circ \rangle$
3]	A _{s,tot} . [mm ²]	
4] + [804,2	
Bottom	n reinforcemer	nt								
	Diameter	Туре	Distance	Count	Cov	er	As			
	[mm]	Input	[mm]	[-]	Autom.	[mm]	[mm ²]	. —		
V 1	16,00	Number 💌		2		30,0	402,1	^		
2]		Reinf. positioning
3]	As,tot. [mm ²]	Generate identical bar spacing
4] + [402,1	Bars as much on edge as possible
Inform	ation on reinf									
Chec Colur $\rho_s =$ $\rho_s =$	c k of min an nn (total reinf 0,0349 ≥ ρ ₅ 0,0349 ≤ ρ ₅	d max reinfo forcement): .,min = 0,002 .,max = 0,04	⇒ Pass ⇒ Pass ⇒ Pass	vel						*
Utilizatio	n by bending	: 34,1 %	PASS							

Definition of reinforcement

After defining reinforcement we can immediately check in the lower part of the dialog box that the area of the reinforcement is sufficient and passes design criteria with 34.1% utilization by bending. In the section "Information on reinforcement" we can also confirm that the detailing requirements given by the code are satisfied. Finally we need to check if the covers are correctly defined. Having a column with stirrups, the option "Min cover and stirrups" is selected in the "Cover" section of the dialog box. The program will calculate the minimum required cover of the longitudinal reinforcement as sum of the minimum cover given by the code and the diameter of the stirrups. The calculation can be checked in the dialog box opened by clicking "Minimum cover" button:

Reinforcement cover			— ×						
Exposure class									
Exposure class >>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>	ко	E	Edit						
Indicative strength class	$C12/15 \Rightarrow$ strength class p	ass							
Structure class									
Class :	S4		•						
Residential, civil and other common structures, industrial structures, structures for mining, reservoirs, wather management									
Lifetime > 80 years	Lifetime > 100	years							
Slab geometry	Special quality	control							
Resulting structural class:	53								
Other infl.									
Abrasion class : No abrasi	on		•						
Uneven surface		0,0	[mm]						
Additive safety element	Δc _{dur,γ}	0,0	[mm]						
Stainless steel	ΔCdur,st	0,0	[mm]						
Additional protection	ΔCdur,add	0,0	[mm]						
Allowance in design for deviat	ion ∆c _{dev}	10,0	[mm]						
Ground:	epared	i soil							
Minimal cover									
- mox(20, 20 + 10) = max	(20 / 10) – 50 mm								

Minimum cover calculation

As it is not necessary to change the settings of cover calculation, we can exit the dialog box by clicking "**OK**" and return to the dialog box for longitudinal reinforcement definition. To check whether the geometry of the longitudinal reinforcement satisfies requirements on minimum covers, we can run the assessment by clicking "**Check of cover**".



Covers check result

The check returned a positive result, therefore we can return to the main dialog box by clicking "OK".



Shear reinforcement

We can proceed to defining the shear reinforcement by clicking the eponymous button in the main screen.

We specify 10mm diameter boundary stirrups with 150mm spacing:

Boundary stirrups Diameter d : Spacing s :				
Diameter d : Spacing s :				
Spacing s :	10,00 [mm	1	ł	
	150,0 [mm	1		
Torsion : Co	nsired only to s	hear resistance		
Ratio of stirrup area used for	torsion resistan	ce:	[%]	
Ties, inner stirrups vertic	al	Ties, inner stirrups horiz	ontal	
Stirrups same as boundary		Stirrups same as boundar	y.	
Diameter d :	[mm] Diameter d :	[mm]	
Spacing s :	(mm] Spacing s :	[mm]	
Count of shears:	[-]	Count of shears:	[-]	
	B			
Bent-up bars vertical		Bent-up bars horizontal	~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~	
Diameter d :	[mm]] Diameter d :	[mm]	
Pitch a:	[°]	Pitch a :	[1]	
Count of shears:	[-]	Count of shears:	[-]	230,0
As row of bent-up bars		☐ As row of bent-up bars		
Spacing s :	[mm] Spacing s :	[mm]	
<u> </u>	1	<u>An</u>		
Inner lever arm		Angle of compression struts		
Define by calculation		 Iterate 		
🗩 Define as	×	d 🔘 User defined	["]	
Information				

Shear reinforcement definition

Buckling

The next step is defining the buckling parameters. Firstly we need to tick **"Calculate buck. Y/Z"** boxes followed by entering the nominal lengths of the column for both directions, based on which the effective buckling lengths will be calculated. For a column simply supported on both ends, the effective buckling lengths equal to the nominal. Defining

different boundary conditions can be done by clicking the "Be buttons for each direction.

Imperfection, b	uckl., crack	s			
Add imperfe	=	2,000	[m]		
🔽 Calculate bu	ck. Y 🔽 Ca	lculate	e buck. Z	🖉 <u>D</u> etails	-
Length Y :	2,000	[m]	🔛 L _{0y} =	2,000	[m]
Length Z :	2,000	[m]	L _{Oz} =	2,000	[m]



Defined buckling parameters

After defining the buckling parameters, two distinct areas of capacity are shown in the interaction diagram: thin dashed line denotes capacity of the member without influence of buckling and thick line denotes the capacity reduced by buckling effect. To check position of a defined load case in the "Interaction diagram" we need to enter the relevant axial force level; in our example N= -400kN. Program will create a cut through the interaction diagram on this level showing the position of the defined load case:



Defining a cut through the interaction diagram

Because we completed definition of all parameters, it is recommended to save the job by clicking "In the toolbar or using shortcut "**Ctrl+S**". The actual state of the job during work may not be identical to the one saved on disk; this is indicated by "*" in the program window header. In such case it is advisable to save the job.



Indication of non-saved job state

Indication of non-saved job stateAs all structural requirements have been checked and satisfied during the definition of parameters and as the main dialog box indicates that the section passes design checks in both ultimate and serviceability limit states, the job can be considered finished.





Assessed section in Concrete 3D program

Outputs

When the job is finished and saved, we can proceed to composing the output documentation. First we print out a concise single page output which summarizes all input data and design checks results. Composition of this output is run by

clicking "File" and "Graphic output" or "



Print and export document			
Document Edit View Page			
🖬 🥩 🖺 🗋 Document : Text output	· Scher	ve : color 🔹 🕫 🗘	ା 🔹 🏚 🔄 ସ୍ 🗰 🛅 🛅
Editor			
Editor	4 [B1/2	10 44 (21 0 x 22 7 cm)	<section-header><section-header><section-header><section-header><section-header><section-header><section-header><section-header><section-header><section-header></section-header></section-header></section-header></section-header></section-header></section-header></section-header></section-header></section-header></section-header>
a second meterics is seconds	Carl and	and been dealed a card, card	

Graphical output of Concrete 3D program

We can print the document directly by clicking " button or save it on disk as *.pdf or *.rtf file by clicking " button. We use the second option and save the file on disk. In the dialog box "**Save as**" we can enter the file name and select the destination folder.

😿 Save as								×
Mistni disk (C:)) 🕨 Users 🕨 Veřejné 🕨	 Veřejné dokumenty 	Fine FIN EC v3 Pril	klady	• 69 Pi	ohledat: FIN EC v3	Příklady	٩
Uspořádat 👻 Nová složka						8E	E 🕶 🛛 (0
 Dropbox Naposledy navštíven Stažené soubory Plocha Plocha Dokumenty Hudba Obrázky Subversion Videa Domácí skupina Počítač 	Název položky		Datum změny Hledání neodpovíd	Typ lají žádné položky.	Velikost			
Název souboru: Column - s Uložit jako typ: Adobe PDF	ummary document (*.pdf)							•
🔿 Skrýt složky						Uložit	Storno]



Apart from this concise document, we can also compose a detail text output by clicking the " selecting "**File**" and "**Text/Graphic output**" in the main menu. However, as we are still in the print and export document dialog box we can change the document type to text output directly in the toolbar's "**Document**" drop-down list.



Change of output type

After switching to the "**Text output**" mode we can set in the "**Editor**" which parts of the assessment will be included in the output and how detailed the output shall be.



Printing options of text outputs

Program will immediately re-generate the output to reflect each change made in the settings in the tree on the left-hand side. Once the output contains all required information, we can again save the document on disk.





Generated text output

Completing the outputs generation, our work is done.