This document gives a detailed summary of the new features and modifications of FEM-Design version 14. We hope you will enjoy using the program and its new tools and possibilities. We wish you success.

Strusoft, the developers

Legend



Pay attention / Note

Useful hint



Example

Clicking left mouse button



Clicking right mouse button

Clicking middle mouse button

WHAT'S NEW IN FEM-DESIGN 14

- Full 64-bit version (larger models can be calculated)
- List tables
 - for selected objects
 - using User-defined batches
 - to Excel by User-defined templates
- Single layer reinforcement for shells
- Fictitious shell
- Detailed result for line support group reaction and connection forces
- Local stability analysis
- Improved user interface
- Improved graphical performance

Table of contents

1. Str	UCTURE	5
1.1.	Material properties	5
1.2.	Shell objects	5
1.3.	Fictitious shell	6
1.4.	Profiled shell	10
1.4.1.	Physical and Analytical system	10
1.4.2.	Stiffness calculation	11
1.5.	Creep coefficient	13
1.6.	Reduction for stability analysis	14
1.7.	Edge connection / End point behaviour	15
2. Lo.	ADS	17
2.1.	Load groups	17
3. AN.	ALYSIS	
3.1.	Setup calculation by load combinations	
3.2.	Local stability results	
3.3.	Detailed result	
3.4.	Refreshing numeric values	
3.5.	Section result distribution for shells	
3.6.	Displaying support reactions and connection forces according to uplift	
4. Des	SIGN	
4.1.	Manual design	
5. RC	DESIGN	
5.1.	Single layer reinforcement	
6. Ste	EL DESIGN	
6.1.	Flexural buckling curve defined by the user	
6.2.	Convergence criteria for Class 4 steel sections	
7. Do	CUMENTATION	
7.1.	Options tab in table properties	
8. SYS	ТЕМ	
8.1.	Full 64 bit version	
8.2.	Increased graphical performance	
9. CA	D/User interface	
9.1.	Quick menu	
	-	
9.2.	Quick selection	
9.2. 9.3.	Quick selection	

9	.6.	Using Undo/Redo	36
9	.7.	Editing point	36
9	.8.	Display result	37
9	.9.	Cursors in "Draw" menu	38
9	.10.	Layer settings	38
10.	Т	OOLS	39
1	0.1.	List	39
1	0.2.	Quantity estimation	44
11.	S	ETTINGS	46
1	1.1.	Displaying element ID	46
1	1.2.	Scale options	46

1. STRUCTURE

1.1. Material properties

In FEM-Design 14 the Environmental class is not part of the Concrete/Application data dialog because it has no role any more in the calculations.

Library	Application data	Library	 Application data
Concrete - C12/15 - C16/20 - C20/25 - C20/25 - C20/37 - C30/37 - C30/37 - C30/37 - C30/37 - C30/35 - C45/55 - C50/60 - Steel - S 235 - S 235 - S 235 - S 420 - C 45 New Modify [□]	E Ultimate Acc./Seis. Gamma c 1.50 1.20 Gamma s 1.15 1.00 Gamma c 1.15 1.00 Gamma c 1.15 1.00 Gamma c 1.20 1.20 Alpha c 1.00 1.20 Alpha ct 1.00 1.00 Environment X0 Creep coefficient. Creep coefficient. 0.000 Shrinkage [%e] 0.000 Reduction for dynamic analysis 1.00 Delete Import Export	E - Concrete - (12/15 - (16/20 - C20/25 - (25/30) - (25/30) - (25/35) - (25/35) - (25/35) - (26/35) - (26/35)	E Ultimate Acc./Sels. Gamma c 1.50 1.20 Gamma s 1.15 1.00 Gamma c 1.15 1.00 Alpha cc 1.15 1.00 Alpha cc 1.00 0.000 Shrinkage [%s] 0.000 0.000 Shrinkage [%s] 1.00 1.00 Delete Import Export

1.2. Shell objects

In *Structure* tab the Shell objects are separated both for in-situ (*Plate* and *Wall*) and prefabricated (*Profiled panel* and *Timber panel*) elements.

With this reorganization it is easier to choose the required element type.



1.3. Fictitious shell

A new object, the Fictitious shell is added to the Structure tab.

Fietib	ous shell	POL	B 0		
	? C	DOL	BJ		
	Edges	Contract of the local division of the local			
		Rigid"	•		
	1				
ctitious shell					-×
		N 10 1 10 1			-
Identifier (.posit	on number]	anana ana			OK
Identifier (.posit	on number)	*****			OK Cancel
Identifier (.posit Membrane stiffi	ess matrix D [d	4m]	Physical properties		- OK Cancel
Membrane stiff	ess matrix D [kt	4m]	Physical properties Unit mass [t/m2]	[1.00	OK Cancel
Membrane stiffs 10000 Dxx 5000	ess matrix D [ld 5000 Dxy 10000 Dyy	4 m]	Physical properties Unit mass [t/m2] t1 [m]	1.00	OK Cancel
Membrane stiffs 10000 Dox 5000 0	ess matrix D [k 5000 Dxy 10000 Dyy 0	4m] 0 0 10000 DGay	Physical properties Unit mass [t/m2] t1 [m] t2 [m]	1.00 0.100 0.100	Cancel
Membrane stiffs 10000 Dox 5000 0 Flexural stiffset	ess matrix D [ld 5000 Dxy 10000 Dyy 0 s matrix K [ldim]	(m) 0 10000 DGay	Physical properties Unit mass [t/m2] t1 [m] t2 [m] Alpha 1 [1/°C]	1.00 0.100 0.100 0.0000100	Cancel
Membrane stiffi 10000 Dox 5000 0 Flexural stiffner 10000 Exe	ess matrix D [ld 5000 Diy 10000 Dyy 0 s matrix K [idim] 5000 Koy	(m) 0 100000 DGay	Physical properties Unit mass [t/m2] t1 [m] t2 [m] Alpha 1 [t/*C] Alpha 2 [1/*C]	1.00 0.100 0.100 0.0000100 0.0000100	Cancel
Membrane stiffs 10000 Dox 5000 0 Flexural stiffnes 10000 Kox 5000	ess matrix D [id 5000 Dxy 10000 Dyy 0 s matrix K [id4m] 5000 Kay	(m) 0 10000 DGay	Physical properties Unit mass [t/m2] t1 [m] t2 [m] Alpha 1 [1/ºC] Alpha 2 [1/ºC]	1.00 0.100 0.100 0.0000100 0.0000100	Cancel
Identifier (.posit Membrane stiffi 10000 Dox 5000 0 Plexural stiffnes 10000 Kox 5000 0	ess matrix D [d 5000 Diy 10000 Dyy 0 s matrix K [id4m] 5000 Kyy 0	(m) 0 100000 DGay 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0	Physical properties Unit mass [t/m2] t1 [m] t2 [m] Alpha 1 [t/*C] Alpha 2 [1/*C]	1.00 0.100 0.100 0.0000100 0.0000100	Cancel
Identifier (.posit Membrane stiffi 10000 Dox 5000 0 Plexural stiffnes 10000 Kox 5000 0	ess matrix D [k 5000 Diy 10000 Dyy 0 s matrix K [khm] 5000 Kyy 0	(m) 0 10000 DGay 0 0 10000 KGay	Physical properties Unit mass [t/m2] t1 [m] t2 [m] Alpha 1 [1/*C] Alpha 2 [1/*C] t1 t2	1.00 0.100 0.0000100 0.0000100	Cancel

In the tool-window the type of the edges (Rigid or Hinged) can be chosen.

In the Default settings dialog the User can define the stiffness matrices (Membrane stiffness matrix, Flexural stiffness matrix and Shear stiffness matrix) and physical properties (Unit mass, distances of the planes where thermal load acts from the center plane, coefficient of thermal expansion).

ZQZ.

Fictitious shell is ideal to model a section with *composite material*, where the designer may calculate appropriate stiffness and fill the D, K and H stiffness values.

2Q3

Also Fictitious shell is a powerful tool when calculating a section with *complex geometry* where the parameters are specified by the manufacturer of the structural element.

Stiffness of orthotropic shell element for fictitious shell can be calculated as follow:



Numerical example below will show how to fill-up the stiffness matrix.



Input Data

E=30GPa v=0.2 G=12.5GPa h=40cm a1=25cm b1=35cm a=1.5m b=1.0m

Membrane Stiffness

Dxx=	E*h*a1/a=	2000000 kN/m
Dxy=	Here lateral effect is negligible and assumed to be zero	0 kN/m
Dyy=	E*h*b1/b=	4200000 kN/m
DGxy=	$G^{h^{(1-(a-a1)^{(b-b1)}/(a^{b}))=}$	2291667 kN/m

Bending Stiffness

Kxx=	E*h*h/12*a1/a=	$26667 \ kNm^2/m$
Kxy=	v*E*h*h*h/12*(a1*b1)/(a*b)=	1867 kNm^2/m
Куу=	E*h*h/12*b1/b= E*h*h/12*(1-	56000 kNm^2/m
KGxy=	$v^*v^*a1^*b1/(a^*b))/(a^*b^*(a/(a1^*a1^*a1)+b/(b1^*b1^*b1)))=$	892 kNm^2/m

Shear Stiffness

Hxz=	5/6*G*h*a1/a=	694444 kN/m
Hyz=	5/6*G*h*b1/b=	1458333 kN/m

As it was mentioned above, if the D, K and H stiffness values are calculated and filled in, then it is possible to model the sections with complex geometry and composite materials with new fictitious shell object. Few section are show below which can be handled by the help of fictitious shell.



Corrugated steel + Concrete floor

9

1.4. Profiled shell

1.4.1. Physical and Analytical system

In Profiled (also Timber) shell dialog there is opportunity to select the physical model (in-situ or prefabricated), then analytical system of shell, if applicable. The analytical system can be *Continuous* or *Panel by panel*. This option is available in profiled plate and profiled wall.

Profiled plate Profiled wall Profiled wall Profi	lignment
A.1 General Section Anterial Border Panel Identifier (.position number) PP Model PP Model Eccentricity (display only) Physical model Profiled plate/wall Gap between panels [m] 0.000 Analytical model 0.000	Physical model
Transverse flexural stiffness factor Display stiffness > OK Cancel	Panel by panel

MZ MZ

Choosing continuous analytical system can be useful when modelling the whole structure e.g. in the preliminary design phase. The memory usage of calculation with continuous analytical system is less than panel by panel system due to the reduced number of edge connections.

The use of panel by panel analytical system is reasonable in the phase of detailed design. In this case the results of the calculation are more accurate.

ZQZ

In case of continuous analytical system, effect of the connections between the panels can be considered by the transverse flexural stiffness factor.

1.4.2. Stiffness calculation

Profiled shells are calculated as Fictitious shells (the stiffness matrices are calculated automatically, according to the tables below).



Section data:

 h_x , h_y : equivalent thickness in x and y direction

I_x, I_y: equivalent inertias in x and y direction

 ρ_x , ρ_y : equivalent shear factor in x and y direction

x direction: strong axis of the panel cross-section

y direction: is perpendicular to the x direction in the plane of the panel

Material data:

- E: Young modulus
- G: Shear modulus
- v: Poisson's ratio

Panel data:

q: transverse flexural stiffness factor (applicable only in Continuous analytical system)

Modifying any above mentioned data will affect the stiffness of shell. The stiffness of shell is calculated automatically. The value of the stiffness (derived from properties of the shell) can be checked by clicking on the "Display stiffness" button.

Profiled plate/wall		
A.1 General OC Section Material	Calculated properties of profiled panel	x
Identifier (.position number)	Calculation Load case and comb., ULS 🔻	Close
Model E	Membrane stiffness matrix D [kN/m] Physical properties	
Physical model	3099244 540459 0 Unit mass [t/m2] 0.245	
	540459 2356190 0 t1 [m] 0.0978	
Panel type identifier A	0 0 1091087 t2 [m] 0.102	
Gap between panels [m] 0.00300	Alpha 1 [1/°C]	
Analytical model	Alpha 2 [1/°C]	
Transverse flexural		
stiffness factor		
	Shear stiffness matrix H [kN/m]	
	525438 0	
<u> </u>	0 781076	
Display stiffness >		

As it will be mentioned later $(\underline{1.6})$, in version 14 there will be different types of Young moduli (except for steel material) in different types of calculations. This also affects the stiffness of shell in different types of calculations. These stiffness can easily be checked in "Display stiffness" dialog.

Calculated pr	operties of profiled panel	-		×
Calculation .	Load case and comb., ULS 🔻			Close
	Load case and comb., ULS			
Membrane s	tiffi Load comb., 2nd order, ULS	Physical properties		
30992	Load comb., 2nd order, SLS Dynamic analysis	Unit mass [t/m2]	0.245	
5404	59 2350190 0	t1 [m]	0.0978	
	0 0 1091087	t2 [m]	0.102	
		Alpha 1 [1/ºC]	0.0000100	

This powerful tool in version 14 gives possibility to automatically calculate stiffness of concrete shell having *complex geometry*. Some typical examples are shown below (sections are made in section-editor).



1.5. Creep coefficient

In Material settings dialog different creep coefficients can be specified for Serviceability Limit State (SLS) and Ultimate Limit State (ULS). It has some consequences in load case calculation and results:

- All load cases are calculated twice (first with the SLS Creep coefficient, than with the ULS Creep coefficient.
- The displayed displacements are the results of the SLS calculation.
- The displayed internal forces, reactions are the results of the ULS calculation.

ibrary	Application data
Concrete	Ultimate Acc./Seis.
C16/20	E Gamma c 1.50 1.20
C20/25	Gamma s 1.15 1.00
	Gamma cE 1.20 >
C35/45	Alpha cc
	Alpha et 100
C50/60	Alpha ct 1.00
Steel	Creep c. (SLS-ULS) 3.00 1.00
S 235	Shrinkage [‰] 0.000
- S 275	Reduction for dynamic analysis 1.00
	Reduction for stability analysis 1.00
New Modify	Delete Import Export

1.6. Reduction for stability analysis

In Material settings dialog the stiffness for stability analysis can be decreased. Taking a reduction factor into account is needed in those calculations where it is specified by the standards (e.g. at the cracked section analysis).



The following table and figures summarize the calculation of Young moduli used in different analysis types for concrete, steel and timber materials.

Analysis type		Concrete	Steel	Timber
Load cases,	ULS	$\frac{E_{cm}}{(1+\varphi_u)}$		E _{o,mean}
1st order load combination	SLS	$\frac{E_{cm}}{(1+\varphi_s)}$		$(1+k_{def})$
2nd order load combination,	ULS	$\frac{E_{cm}}{(1+\varphi_u)}\cdot\frac{1}{\gamma_{cE}}$	E_{k}	E _{o,mean}
Imperfection calculation	SLS	$\frac{E_{cm}}{(1+\varphi_s)} \cdot \frac{1}{\gamma_{cE}}$		γ_M
Stability analysis		$f_{stab} \cdot E_{cm}$		$\frac{E_{o,mean}}{(1+k_{def})}$
Eigenfrequency calculation Seismic analysis		$f_{din} \cdot E_{cm}$		E _{o,mean}



1.7. Edge connection / End point behaviour

In FD 14 for the end points of the edge connections two options are available:

- Separate end points from environment,
- Connect end point to environment.

In all Plate and Wall Toolwindows this setting is available by choosing 'Edge connection' then clicking on 'End point behaviour' button. The end point can be chosen in three steps by selecting the Plate or Wall, then the edge and finally the end point.



Using this function can solve problems like the one on the picture below:

On the right side structure all edge connection end points are connected to the environment which cases difference between the shear force on the wall and the sum of shear forces on the plate panels, which should be in balance according to common sense and this is exactly the case on the left side structure where edge connection end points are separated from the environment.



2. LOADS

2.1. Load groups

In *Load groups* dialog the User has the opportunity to choose one of the combination methods offered by Eurocode 0. Two methods of determining the combination of actions are allowed for the STR Ultimate Limit States.

No	Load group	Included load cases	-	ОК
				Cancel
				Import / Export >
				Combination metho © EC0 6.10 © EC0 6.10.a, b
			Ш	Load group
_				Insert
				Delete
				Delete all
				Load case
				Insert
			_	New
			-	Remove

The first approach is to use expression 6.10.

$$\sum_{j\geq 1} \gamma_{G,j} G_{k,j} + \gamma_p P'' + \gamma_{Q,1} Q_{k,1} + \sum_{i\geq 1} \gamma_{Q,i} \psi_{0,i} Q_{k,i}$$
(6.10.)

The second approach is to use the more onerous of expressions 6.10.a and 6.10.b.

$$\sum_{j\geq 1} \gamma_{G,j} G_{k,j} + \gamma_p P'' + \gamma_{Q,1} \psi_{0,1} Q_{k,1} + \sum_{i\geq 1} \gamma_{Q,i} \psi_{0,i} Q_{k,i}$$
(6.10.a)

$$\sum_{j\geq 1} \xi_{j} \gamma_{G,j} G_{k,j} + \gamma_{p} P'' + \gamma_{Q,1} Q_{k,1} + \sum_{i\geq 1} \gamma_{Q,i} \psi_{0,i} Q_{k,i}$$
(6.10.b)

The subtle attraction of this pair of expression derives from two important changes from 6.10.:

- The application of the ψ_0 factor to the leading variable action in expression 6.10.a (not applied in 6.10.)
- The introduction of a reduction factor ξ applied to the permanent actions in expression 6.10.b (not applied in 6.10.)



The whole expressions appear on the screen as the cursor is moved on the name of the combination method.

3. ANALYSIS

3.1. Setup calculation by load combinations

The calculation of the load combinations can be run with different options. They can be set in Calculations dialog by selecting the Load combination items and clicking \bigcirc on *Setup by load combinations*.

	Calculations	Load combinations				
	Load cases	Non-linear analysis				
	Load combinations Maximum of load groups	Maximum iteration number		30		
	Eigenfrequendes	Cracked section analysis				
		One load step in % of the total lo	ad	20		
		Maximum iteration number		20		
		Allowed displacement error [%] .		1		
	Recalculation	Setup by load combinations		Ca	ncel	
	Recalculator			Ca	licel	Ľ.,
Set	up load combinations					
Set /pe	up load combinations Load com 1.35"Dead load + 1.50"Wind x + 1.50"0.70"	abination Snow	NL	CS	2ND	IS
Set	up load combinations Load com 1.35°Dead load + 1.50°Wind x + 1.50°0,70° 1.35°Dead load + 1.50°0,70°Wind x + 1.50°	ibination Snow Snow	NL	CS X	2ND	IS
Set	up load combinations Load com 1.35"Dead load + 1.50"Wind x + 1.50"0.70" 1.35"Dead load + 1.50"0,70"Wind x + 1.50" 1.35"Dead load + 1.50"Wind y + 1.50"0,70"	bination Snow Snow Snow	NL	CS X	2ND X	IS 5
Set	up load combinations Load com 1.35"Dead load + 1.50"\Vind x + 1.50"0,70" 1.35"Dead load + 1.50"0,70"Vind x + 1.50" 1.35"Dead load + 1.50"\Vind y + 1.50"0,70" 1.35"Dead load + 1.50"0,70"Vind y + 1.50"	bination Snow Snow Snow Snow	NL	CS X	2ND X	IS
pe	Load combinations Load com 1.35°Dead load + 1.50°Wind x + 1.50°O.70° 1.35°Dead load + 1.50°O,70°Wind x + 1.50° 1.35°Dead load + 1.50°O,70°Wind y + 1.50° Dead load + 1.50°O,70°Wind y + 1.50° Dead load + 1.50°O,70°Snow	bination Snow Snow Snow Snow	NL	CS X X	2ND X	IS
Set /pe	Load combinations Load con 1.35"Dead load + 1.50"\Vind x + 1.50"\0.70" 1.35"Dead load + 1.50"\0.70"\Vind x + 1.50" 1.35"Dead load + 1.50"\0.70"\Vind y + 1.50" Dead load + 1.50"\0.70"\Vind y + 1.50" Dead load + 0.70"\Vind x + 5now Dead load + 0.70"\Vind x + 5now	bination Snow Snow Snow Snow	NL	CS X X X	X	IS
/pe	Load combinations Load con 1.35"Dead load + 1.50"Wind x + 1.50".07" 1.35"Dead load + 1.50"7,0"Wind x + 1.50" 1.35"Dead load + 1.50"7,0"Wind y + 1.50" Dead load + 1.50"7,0"Wind y + 1.50" Dead load + 0.70"Wind x + Snow Dead load + Wind y + 0.70"Snow Dead load + Wind y + 0.70"Snow Dead load - 0.70"Wind y + Snow	bination Snow Snow Snow Snow	NL	CS X X X X X X	X	IS 5
pe	Load combinations 1.35"Dead load + 1.50"Wind x + 1.50"0,70" 1.35"Dead load + 1.50"0,70"Wind x + 1.50" 1.35"Dead load + 1.50"Vind y + 1.50" 1.35"Dead load + 1.50"0,70"Wind y + 1.50" Dead load + 1.50"0,70"Wind y + 1.50" Dead load + 1.50"Vind y + 5.50" Dead load + 1.50"Vind y + 5.50" Dead load + 1.50"Wind y + 5.50" Dead load + 0.70"Wind y + 5.50" Dead load + 0.70"Wind y + 5.50"	bination Snow Snow Snow	X	CS X X X X X X	X	IS
pe	Load combinations Load com 1.35"Dead load + 1.50"Wind x + 1.50". 1.35"Dead load + 1.50"N/01 x + 1.50" 1.35"Dead load + 1.50"N/07"Wind x + 1.50" 1.35"Dead load + 1.50"N/07"Wind y + 1.50" Dead load + 0.70"Wind x + 50row Dead load + 0.70"Wind y + 50row Dead load + 0.70"Wind y + 50row	tbination Snow Snow Snow Snow	NL	CS X X X X X	X	IS
Set	Load combinations Load com 1.35"Dead load + 1.50"Wind x + 1.50" 1.35"Dead load + 1.50"\Vind x + 1.50" 1.35"Dead load + 1.50"\Vind y + 1.50" 1.35"Dead load + 1.50"\Vind y + 1.50" Dead load + 0.70"\Vind x + 5now Dead load + 0.70"\Vind y + 5now Dead load + 0.70"\Vind y + 5now Dead load + 0.70"\Vind y + 5now	bination Snow Snow Snow	NL	CS X X X X X	ZND X	IS
pe	up load combinations Load com 1.35"Dead load + 1.50"\/md x + 1.50"0,70" 1.35"Dead load + 1.50"0,70"/Wind x + 1.50" 1.35"Dead load + 1.50"0,70"\/Wind y + 1.50" Dead load + 1.50"\/Mo y + 1.50" Dead load + 1.50"\/Wind y + 5.00 Dead load + 0.70"\/Wind y + 5.00 Dead load + 0.70"\/Wind y + 5.00W Dead load + 0.70"\/Wind y + 5.00W	bination Snow Snow Snow	NL	CS X X X X X	ZND X X	15

The User has the opportunity to choose

- Non-linear calculation (NL),
- Cracked-section analysis (CS),
- Second order analysis (2ND),
- Imperfection calculation (IS, the selected shape will be taken into account in Second order analysis)

for each Load combinations.



For example, in practice it can be useful to set 2nd order analysis only for the ULS and Cracked-section analysis only for the SLS combinations.

3.2. Local stability results

In FD 14 local stability results can be displayed. After calculating the load combinations the Local stability results (*Overturning of walls* and *Sliding*) are available in Display results dialog.



- Overturning of walls: only those walls can be calculated which have at least one horizontal edge in the bottom and edge connection is defined for it. The result is expressed as a percentage:
 - 0% belongs to the case when the vertical force acts at the centre of bottom edges,
 - 100% belongs to the case when the vertical force acts at one of the corners,
 - 1000% belongs to the case when the resultant is outside the wall edge.



• Sliding of edge connections: the result is the ratio of the design force and the friction capacity. The friction factor can be set in the *edge connection* dialog.



Overturning of walls results are informative. Without accurate modelling it may lead to incorrect results!

Numerical example below will illustrate the *Local Stability*.



Geometry and Loads

Properties of edge connections



Non-linear calculation (which allows uplift) is recommended to get correct result for local stability.

Calculations		
Calculations Analysis Cad cases Coad combinations Cad com	Load combinations Non-linear analysis Maximum iteration number Cracked section analysis One load step in % of the total load Maximum iteration number Allowed displacement error [%] Setup by load combinations	Setup load combinations
	OK	



Displacement graph (as well as connection force) is the easiest way to check the uplift.

Overturning of wall

With the help of resultants of edge connections, wall's overturning can be examined as below.





Sliding of edge connections

Edge connection's sliding is calculated in each edge connection *separately* by comparing the x' component of the connection force as design force, and the limit force calculated by the y' components of the connection forces and the friction coefficient of the edge connection.



3.3. Detailed result

Detailed analytical results are available for two-dimensional analytical objects (e.g. bars, line support groups and connections) after clicking on \bigcirc button and by selecting the object.



In *Detailed result* window the User can manage the results by selecting the object, the calculation and other – calculation dependent – options.



The dialog of the display options is renewed, as the distribution of the selected result functions can be chosen (calculated, linear, constant or constant by element) for line support groups and connections.



3.4. Refreshing numeric values

The numeric values are refreshed automatically when the display options of a result are modified.

The following figure describes this feature on an example of a simple frame structure.



3.5. Section result distribution for shells

ŽČ

The distribution of the displayed section results can be chosen by the user. Besides the calculated distribution it can be linear or constant.

This function can be useful at checking the average in-plane shear force between profiled panels.



3.6. Displaying support reactions and connection forces according to uplift

In FD 14 displaying of line support reactions and connection forces has been improved in case of linear and constant distributions by considering which part of the support/connection is "uplifted".



4. DESIGN

4.1. Manual design

In FEM-Design 14 the manual design can be done without calculated analysis results in the following categories:

- RC design
- Steel design
- Timber design

This function is useful especially at the analysis of an existing building.

5. RC DESIGN

5.1. Single layer reinforcement

In RC design the User has the possibility to place single layer surface reinforcement. A Plate or Wall can be specified as single layer reinforced by defining "Single layer reinforcement" Calculation parameter for it.



In the Calculation parameters dialog after clicking on the single layer reinforcement and the default

settings button the calculation parameters can be set in the dialog. The User can define the followings:

- the quality and diameter of the reinforcement for both directions,
- the direction of the bottom layer,
- the distance of the reinforcement from the centreline,
- the allowed crack width on the bottom and on the top of the structure.

Defau	lt reinforcement				OK
Dir	Quality	Diam. [mm]	Bottom layer	-	Cancel
X'	B420A	10	Х		
y'	B420A	10		-	

If a shell has "Single layer reinforcement" Calculation parameter, its Design parameter can be modified only if "Single layer reinforcement" option is selected in Auto design/Parameters.

	jn						
1	÷ …						
9	Ţ						
	¥						
id plar	ase net						
Dir	Quality	Diam. [mm]	Space [mm]	Area [mm2/m]			
x'	B500A	10	150	524			-
y'	B500A	10	150	524			-
	Quality	Diam	Space	Area	Shane	Bound	
Addi Dir	Godiny	[mm]	[mm]	[mm2/m]	[mm]	[mm]	=
Addi Dir	1	8, 12, 16	150	?	Rect.	100	
Addi Dir x'	B500A		150	?	Rect.	100	-
Addi Dir x' y'	B500A B500A	8, 12, 16	150				

In case of Manual design, single layer reinforcement can be placed only on "Mid, x'/r" and "Mid, y'/t" layers.

Reinfo	rcemen	t layer	_	23
-‡	++	+	 1	+

Single and double layer reinforcements cannot be used in the same Plate or Wall element.

6. STEEL DESIGN

6.1. Flexural buckling curve defined by the user

User can specify the flexural buckling curves (EC3-1-1: 6.3.1.2) for each steel bar in Calculation parameters dialog or they can let the program to calculate it as in the previous versions by selecting "Auto" option.

[Ĵ፲·]ℓ₀ ∏ ∰ ☞ ≠ _₹
Calculation parameters
Maximal distance between calculated sections [m] 0.500
Flexural buckling curves
Stiff direction Weak direction b c Auto (d)
a0 a ross-section of Class 4 sections. b c
d Strong
Maximum number of iteration steps
OK Cancel

For steel bars with varying section the "Auto" option cannot display the automatically calculated curve, since it is determined during the design calculation.

When the section of a steel bar is modified, buckling curve options of the calculation parameter is reset to "Auto"

ightarrow When buckling curve is calculated automatically, applied section is considered, if it exist.

6.2. Convergence criteria for Class 4 steel sections

Convergence criteria and the maximum number of iteration steps can be set for effective cross-section calculation of Class 4 steel bar section in Calculation parameters dialog.

[] I ·] lo II 🔛 🚳 🔚 🛩 👤
Steel bar
Calculation parameters
Maximal distance between calculated sections [m] 0.500
Flexural buckling curves
Stiff direction Weak direction
Calculation of effective cross-section of Class 4 sections.
Convergence criteria
Weak Strong
Maximum number of iteration steps
OK Cancel

ΞÂ

In some cases the iteration for effective cross-section fails because of the too strong convergence criteria. In this case reducing its factor or increasing the number of the iteration steps may solve the problem.

7. DOCUMENTATION

7.1. Options tab in table properties

Options tab (if available) is added to the Table properties dialog in documentation.

Documentation Section 1. Load casess Cuarkty estimation. Concrete 1 (Loads) Analysis (Analysis)	Go to Cut Copy Paste Delete	able properties		
	Properties	Sort	Summarize	
	2. 🕅	1st priority	Summarize items that has the same	
		(frore)	Storey	
		2nd priority	Name	
		(none)	Quality	
		3rd priority	1	
		(none)		
		4th priority	1	
		(none) Ascending Descending		
		Clear all]	
				OK Cancel

8. SYSTEM

8.1. Full 64 bit version

As FD 14 is a full 64-bit software, all memory of the computer can be utilized. In previous 32-bit versions the memory usage was limited to 2 Gigabytes, which could cause out of memory error message when displaying results at large models. The analysis core was already 64-bit in earlier versions.

A 32-bit operating systems are not supported anymore!

8.2. Increased graphical performance

The graphical performance is increased and more development is expected in future versions. In FD 14 the DirectX graphic engine is faster than OpenGL on most of the computers. We recommend the User to check which graphic engine performs better on his computer.

s General	
ièneral Jirectones Dursor Dursor Dursor Dursor Dursor Question DirectX OpenGL DirectX OpenGL DirectX - safe mode OpenGL Disable all hardware acceleration Remarks >	System Language (R). English Basically three rendering modes can be chosen; OpenGL, DirectX or none of them. Almost every graphic card supports hardware acceleration for both OpenGL and DirectX. However - depending on the graphic card -, the rendering speed can be significantly different in these two modes. You may select the proper mode making a quick test in both modes: - Open a large model with many elements, - Start "Pan" or "Orbit" command and check the speed. Unfortunately, the drivers delivered with the graphic cards often having more or less problems. If you experience any trouble during rendering (e.g. the program crashes during "Orbit" function) follow the steps below to solve the problem: - Download and install the latest driver. - If latest driver is already installed and it is not working properly, switch to the other mode (e.g. from OpenGL to DirectX). - If this stil does not help switch to one of the safe modes. The safe modes.
as default	However, the rendering speed decreases. - If problem still exists, then select "Disable all hardware acceleration". This mode does not use any special feature of the driver, so the crashes caused by the driver can be avoided.



9. CAD/USER INTERFACE

9.1. Quick menu

By clicking right mouse button, a renewed quick menu appears. After clicking on an arbitrary element all the commands of quick menu are visible but those that are not available for the chosen element(s) are disabled.



3 new commands are added to the quick menu:

• Hide selected

For this command FD hides the selected elements

• Hide all others

For this command FD hides all the elements except the selected ones.

• Show all

For this command FD shows all the hidden elements.

9.2. Quick selection

Holding pressed and by using objects can be added to or subtract from the current selection. By pressing + A all the visible objects on the screen get selected.

9.3. Selection box, multiple selection

The selection box changed in FD 14:

• '+' sign is drawn if multiply selection is active (by pressing



- The selection box has different colours depending on the way of the selection:
 - a. By 'left-to-right' selection (the included objects get selected) the colour of the selection box is blue.



b. By 'right-to-left' selection (all intersected objects get selected) the colour of the selection box is green.



c. By area selection (e.g. to print the selected area) the colour of the selection box is grey.



Multiple selection can be ended by \bigcirc + \bigcirc (it has the same effect like pressing

- For pressing + I in the application window (without any active tool palette) the quick menu appears.
- When a tool palette (e.g. Beam tool palette or Point load tool palette) is active, for pressing + O the properties of the items belonging to the active window appear.

9.5. Using Esc in tools

Using in tools has changed in FD 14:

- Pressing 1st Esc breaks the current command (e.g. placing a structural element).
- For pressing 2nd Esc the active tool palette closes.

9.6. Using Undo/Redo

Undo/Redo command does not close the current action window only resets it.

9.7. Editing point

Cursor can be captured by for a not only to guide lines but to arbitrary lines or surfaces.

With the help of this feature the projection of a truss can be easily drawn on an arbitrary plane wall.



If the ray that is perpendicular to the plane of the screen and crosses the cursor intersects more than one plane, User can switch between the planes by pressing +



9.8. Display result

In FD 14 the labels of the colour palette changed. While in the previous version only the zero and the two extreme values were displayed, in FD 14 to all colour tones belongs a label.





 \bigtriangleup The dimension of the currently displayed result is located under result's title.

Calcul Manage results Quick change Equilib	

9.9. Cursors in "Draw" menu

Cursors in "Draw" menu and in the first two tabs (Structure and Loads) are drawn real time.



9.10. Layer settings

In Layers dialog changing the layer status is real-time, which means that the selected active (hidden) drawing or object layers become hidden (active) without closing the dialog.



10. TOOLS

10.1. List

In FD 14 List dialog has improved in several aspects.

After launching the List command, the User can select the **required objects** – using Filter and/or common graphical selection methods - before the List tables dialog appears. The whole database can be

listed by pressing button. Selection can be carried out in the List tables dialog as well using the drop-down menu in the top right corner. The User can choose the followings:

- All (all objects of the model),
- Current selection (objects selected before List tables appeared; this option is not available when all objects were selected (by pressing Enter) after launching the List command),
- Visible objects (objects visible in the Application window),
- User defined filter (objects belonging to pre-defined filters).

Listed data depend on the selected objects. By ticking 'Hide irrelevant tables' check box those tables which are irrelevant by the current selection disappear (otherwise they are shown in grey).

ist tables	Selecting obje	cts 🔍	
ist tables Tables Structure Loads Finite elements Quantity estimation Analysis Load cases Imperfections Load combinations Max. of combinations Stability analysis Eigenfrequencies Seismic analysis Foundation design Load combinations Max. of combinations Max. of combinations Generation design Load combinations Max. of combinations	Selecting obje in-dialog Structure	List objects Current selection Al Current selection Visible objects Filter 1 Filter 2 Printer setup Printer setup Hidding rrelevant table Format table	Data Concrete materials Steel materials Beams Columns
C C design I put data Load combinations Max. of combinations Maximum of load groups C Steal design Batch	Line supports Line support groups Surface support groups Point connections Line connections Surface connections	Options Settings > Add to batch List tables Delete tables Save batch Load batch	
< Hide batch		List batch Close	

In 'Send to...' drop-down menu the destination of the listed tables can be set. Beside Clipboard, Printer and Text file, two new destinations are available:



- Documentation (listed tables appear in the documentation),
- **Excel file** (listed tables can be exported to an Excel file)

File path				
C:\ProgramD	ata\StruSoft\ist	2.xlsx		>
Template path	1			
C:\Users\Cor	mputer Desktop	Template_3.xlt	x	>
Open file in	n Excel			
		_		

In List to Excel dialog besides the File path also a Template path can be set if the User has a predefined Excel-template (with .xltx, .xlt, or .xltm extension). The created Excel file can be opened immediately by ticking 'Open file in Excel' checkbox.

- Listing without a template file: creates an Excel file in which all the tables are located on separate spreadsheets.
- Listing with a template file: the User can create a template in which the locations of the required tables in the spreadsheet are marked with '\$\$' followed by the exact title of the table. Using a template allows the User to gain exactly the required data.

🐹 | 🔙 🧐 ד (🖻 ד | 🖛 FDSteelFrame - Microsoft Excel Fájl Kezdőlap Beszúrás Lap elrendezése Nézet Képletek Adatok Korrektúra Fejlesztőeszközök Team Calibri · 11 · A A ■ = = >>·· Sortöréssel több sorba Általános ▼ $\begin{array}{c|c} & & & \\ \hline \\ Beillesztés \end{array} \xrightarrow{\bullet} & F D \underline{A} \\ \hline \\ \hline \\ \end{array} \xrightarrow{\bullet} & \hline \\ \hline \\ \end{array} \xrightarrow{\bullet} \begin{array}{c} & & \\ \hline \\ \hline \\ \end{array} \xrightarrow{\bullet} \\ \hline \\ \hline \\ \end{array} \xrightarrow{\bullet} \begin{array}{c} & & \\ \hline \\ \hline \\ \end{array} \xrightarrow{\bullet} \\ \hline \\ \hline \\ \end{array} \xrightarrow{\bullet} \begin{array}{c} & & \\ \hline \\ \hline \\ \end{array} \xrightarrow{\bullet} \\ \hline \\ \hline \\ \end{array} \xrightarrow{\bullet} \begin{array}{c} & & \\ \hline \\ \hline \\ \end{array} \xrightarrow{\bullet} \\ \hline \\ \end{array} \xrightarrow{\bullet} \begin{array}{c} & & \\ \hline \\ \end{array} \xrightarrow{\bullet} \\ \hline \\ \end{array} \xrightarrow{\bullet} \begin{array}{c} & & \\ \hline \\ \end{array} \xrightarrow{\bullet} \\ \end{array} \xrightarrow{\bullet} \begin{array}{c} & & \\ \hline \\ \end{array} \xrightarrow{\bullet} \\ \end{array} \xrightarrow{\bullet} \begin{array}{c} & & \\ \hline \\ \end{array} \xrightarrow{\bullet} \\ \end{array} \xrightarrow{\bullet} \begin{array}{c} & & \\ \hline \\ \end{array} \xrightarrow{\bullet} \\ \end{array} \xrightarrow{\bullet} \begin{array}{c} & & \\ \hline \\ \end{array} \xrightarrow{\bullet} \\ \end{array} \xrightarrow{\bullet} \begin{array}{c} & & \\ \end{array} \xrightarrow{\bullet} \\ \end{array} \xrightarrow{\bullet} \begin{array}{c} & & \\ \end{array} \xrightarrow{\bullet} \\ \end{array} \xrightarrow{\bullet} \begin{array}{c} & & \\ \end{array} \xrightarrow{\bullet} \\ \end{array} \xrightarrow{\bullet} \begin{array}{c} & & \\ \end{array} \xrightarrow{\bullet} \\ \end{array} \xrightarrow{\bullet} \begin{array}{c} & & \\ \end{array} \xrightarrow{\bullet} \\ \end{array} \xrightarrow{\bullet} \begin{array}{c} & & \\ \end{array} \xrightarrow{\bullet} \\ \end{array} \xrightarrow{\bullet} \begin{array}{c} & & \\ \end{array} \xrightarrow{\bullet} \\ \end{array} \xrightarrow{\bullet} \begin{array}{c} & & \\ \end{array} \xrightarrow{\bullet} \\ \end{array} \xrightarrow{\bullet} \begin{array}{c} & & \\ \end{array} \xrightarrow{\bullet} \\ \end{array} \xrightarrow{\bullet} \begin{array}{c} & & \\ \end{array} \xrightarrow{\bullet} \end{array} \xrightarrow{\bullet} \end{array} \xrightarrow{\bullet} \begin{array}{c} & & \\ \end{array} \xrightarrow{\bullet} \end{array} \xrightarrow{\bullet} \end{array} \xrightarrow{\bullet} \begin{array}{c} & & \\ \end{array} \xrightarrow{\bullet} \end{array} \xrightarrow{\bullet} \begin{array}{c} & & \\ \end{array} \xrightarrow{\bullet} \end{array} \xrightarrow{\bullet} \end{array} \xrightarrow{\bullet} \begin{array}{c} & & \\ \end{array} \xrightarrow{\bullet} \end{array} \xrightarrow{\bullet} \begin{array}{c} & & \\ \end{array} \xrightarrow{\bullet} \end{array} \xrightarrow{\bullet} \begin{array}{c} & & \\ \end{array} \xrightarrow{\bullet} \end{array} \xrightarrow{\bullet} \end{array} \xrightarrow{\bullet} \end{array} \xrightarrow{\bullet} \begin{array}{c} & & \\ \end{array}$ \$ v % 000 00 00 Feltételes Formázás formázás v táblázatként 📰 🔄 🗐 🗊 🖬 Cellaegyesítés 🕶 Cellastílusok Vágólap Betűtípus Igazítá Stílusol f_x 131 B C G М N C Α D н К L E 1 \$\$Site and load information 2 3 3 4 5 6 7 7 8 9 10 10 11 12 13 14 15 16 \$\$Load cases 17 \$\$Load combinations \$\$Point loads 17 18 19 20 21 22 23 24 25 R + M Structure Loads Quantity estimation Displacement Reactions Internal forces Design 2 Kész

Template file:

Template-based exported file:

🗶 🛃 🧐 • (**	- -					FDSteelF	rame5 - Micro	osoft Excel	l.								- 6 8
Fájl Kezdő	ilap Beszúrás	s Lap elr	endezése	Képletek Adat	ok Korrektúra Nézet	Fejlesz	tőeszközök	Team								_ (] ∩	- @ X3
Beillesztés	Calibri F D A - Betŭ	• 11 • ₩ • 3	$A^* A^* \equiv$: :=::::::::::::::::::::::::::::::::::	Sortöréssel több sorba	Általáno S = 9	os % 000 % 8 Szám	* Felt	tételes Form názás v tábláz St	názás Cella atként + ílusok	astílusok *	Beszúrás Törlés Formátum Cellák	Σ Au Kite Z Tör	toSzum öltés + rlés +	Rendez és szűri	' A tés Keresés és * kijelőlé s	és s *
К34	v (n	fx	I														~
	A		В	С	D	E	F	G	н	1	J	К	L	м	N	0	4
1 Site and load	d information																
2 Value			Quantity														
3 Wind speed	[m/s]			26													
4 Characterist	ic snow load [k	(N/m2]		2													
5 Terrain type	(-)		11														
6 Extent [m]			50.00x30.00														
7 Building hei	ght (m)			0													
8 Building wid	ith x-dir [m]			50													
9 Building wid	tth y-dir [m]			30													
10 Ground leve	el [m]			0													
11																	
12																	
13																	
14																	
15																	
16 Load cases								Load co	mbinations						Point loa	.ds	
17 No.			Name	Туре	Duration class			No.	Name	Туре	Factor	Load cases			No.	F	M
18		1	dead	+Struc. dead	load Permanent				1 U1	Ultimate	1.3	5 dead+Struc. dead load			[-]	[kN]	[kNr
19		2	live	Ordinary	Permanent						1.	2 live				1 1	10
20		3	snow	Ordinary	Permanent						1.	1 snow			1	2 1	10
21		4	windX	Ordinary	Permanent				2 U2	Ultimate	1.3	5 dead+Struc. dead load				3 1	10
22		5	windY	Ordinary	Permanent						1.	2 live				\$ 2	10
23		6	shrinkage	+Shrinkage	Permanent						1.	1 windX					
24									3 U3	Ultimate	1.3	5 dead+Struc. dead load					
25											1.	2 live					
DC	ra Lorde /	Quantity and	imation / P	icolacement Res	Internal forces	Darion /1	2		D 4 D		4	1indV					
Kész 🎦	are coaus /	Quantity est		approximate / Rec	www.a	wayli (٦m	100% (=):		(+
					,									10 10			
🚺 Start 🛛 🔁 Beér	kez 🔀 Visual	🛛 📵 rej	telm 👓	W new_fe.	💌 📄 🔕 Skype	× src (ma 🔚 Wi	ndo	Total C	Notep	FEM-	De 🥶 Untitled 🗶 Mici	ro	EN	* 😼 🕃	/ @ 116	22:53



After listing to Excel without any template and examining the number of columns belonging to a table, it is easier to create a template.

2 batch files + 2 templates are delivered with the program (FDSteelFrame, FDRCStruct).

To avoid the time-consuming procedure of selecting the tables for listing one by one **Batch** is implemented for the tables which can be saved and reloaded, even in different models. This way all the

required tables can be listed by a few click. Tables can be added to the **Batch** by selecting the tables in the "Data" window and clicking on Add to batch button.

	Tables ·	Structure	List objects	
	Loads		Send to	
	Finite elements	Data 1. Select tables	Excel file	
	- Analysis	Concrete materials	Page setup	
	Load cases	Timber materials	Printer setun	
	Imperfections	Soils Techos accel turces	I Hida irrolavant	
	Max, of combinations	Timber panels	tables	
	Maximum of load groups	Storeys Example		
	Stability analysis	Columns		
	Eigenfrequencies	Trusses Plates		
	E Foundation design	Walls		
	Load combinations	Profiled panels Point supports	Format table	
	Max. of combinations	Point support groups	Ontions	
	E-RC design	Line supports		
	Load combinations	Surface support groups	Settings >	
	Max. of combinations	Line connections 2. Add to batc	Add to batch	
	Maximum of load groups	Surface connections	List tables	
	Patrick			
	Concrete materials			
	Steel materials Dat	a annears in hatch window		
tch	Columns	a appears in batter winder		
dow	Trusses		Delete tables	
	Walls		Save batch	Batch
	Profiled panels		Load batch	option
			List batch	
			List Datch	

The following options can be done with the batch:

- Delete tables (selected data will be deleted), •
- Save batch (the batch can be saved to a batch script file with .bsc extension),
- Load batch (the saved batches can be loaded and used in other models; those tables which are not relevant in other models are displayed in grey),
- List batch (all tables in batch window will be listed in accordance with the settings of 'List objects...' and 'Send to...' drop-down menus.

The batch window and options can be hidden by clicking on *<Hide batch* button.

In batch window more data can be selected by holding $rac{S}{2}$ pressed or with the help of $\begin{subarray}{c} \end{subarray}$ and Shin buttons. In a listed batch the order of the tables is the same as in the batch window. The order of the tables

in batch window can be rearranged with \square pressed and using \heartsuit .

A saved batch file with load cases (or load combinations, stability analysis etc.) can be used in other models when the names of the load cases (or load combinations, stability analysis etc.) are the same due to that the identification is based on the name. If all load cases (or load combinations, stability analysis etc.) are selected in the batch window and the batch file is loaded in another model, all the

load cases (or load combinations, stability analysis etc.) of the other model will be taken into account.

10.2. Quantity estimation

In *Quantity estimation* dialog diameter is added to reinforcement quantity estimation. From now in Options menu the items can be sorted or summarized by diameter as well as by the already existing parameters (Storey, Structure type, Name, Quality, Quantity).

Diameter is also added to the Table of reinforcement quantity estimation which is available after clicking on *List...* menu.



Reinforcement quantities are displayed after RC design only. The table can be easily added to the documentation with the *Add to documentation* button.

11. Settings

11.1. Displaying element ID

Displaying structural and analytical element ID's is separated in order to avoid duplicate labels on the screen. The required ID can be set by *Bar* and *Shell* elements in Display settings dialog.

Setting s Draying Draying Caldidation Display Vorawing element -2.15 Numbers -2.15 Numbers -2.15 Stall -37 Shell -35 Support -40 Cover -41 Mesh	Bar Display label including Structural element ID Analytical element ID Material Section name	Graphical options Graphical options Graphical options Graphical options Border Fill Display social system Size [m] Display connections Size [m] D	Shell Display label including Structural element ID Analytical element ID Panel type ID Eccentricity Material E2 / E1 Abba (orthotropic angle)	Graphical options Display local system Size [m] 1.0000 Hatch [mm] 1.5000 Display wall base line
Load Design data ⊕ Window		OK Cancel	Thickness	OK Cancel

11.2. Scale options

In different windows different scales can be defined by the User, which can be specially useful in documentation module.

Settings	
Settings Environment Drawing Calculation Display Window Grid Grid 	Scale 1 2 4 5 10 20 25 40 50 100 125 150 Scale Scale 50.000 This parameter has only effect in text size. Please don't change the scale if you are not advanced user.
Save as default	OK Cancel

E.g. by displaying the results of a reinforced concrete plate once it can be necessary to add a figure to the documentation in which the parameters of the reinforcement are easy to read. In other case the parameters can be decreased to display another result.

