USER MANUAL

FEM-Design



StruSoft *Structural Software Design* Visit the StruSoft website for company and FEM-Design information at *http://www.strusoft.com*.

> **User Manual** Copyright © 2010 by StruSoft, all rights reserved.

Trademarks

FEM-Design is a registered trademark of StruSoft. IFC is a trademark of the International Alliance for Interoperability. All other trademarks are the property of their respective holders.

INTRODUCTION

Thank you for choosing FEM-Design.

This user manual provides a detailed coverage of FEM-Design principles and operation by covering the basic concepts, input, analysis and design calculations and result documentation.

We hope you will enjoy using the program and its features and possibilities. We wish you success.

StruSoft, the developers

Legend



Pay attention / Note

Useful hint

Example



t Italic words marked in cyan are linked to their definition.

ABOUT FEM-DESIGN

FEM-Design offers a powerful, yet easy to use package that can be tailored to suite the exact analysis and design needs of the structural engineers using finite element method.

FEM-Design runs on Microsoft Windows ® 7, Vista and XP operation systems.

Single elements or a complete building, made from any number of materials and structural elements can be analyzed with ease related to Eurocode 2, Eurocode 3 and Eurocode 5.

FEM-Design Modules

The FEM-Design software system is a group of modules developed for different 2 and 3 dimensional structural problems, but it provides additional special features such as 2D drawing services and 3D solid modeling too.

The following tables summarize the design and auxiliary modules with their features and functionality.

 \bigtriangleup The list of available modules depends on what license you have.

Plate	↓↓↓ Wall	Plane Strain	3D Frame	3D Structure	PreDesign	
Problem description						
Slabs, slab-	Plane stress	Structure with	Two and three-	3D structures	Pre-calculation	
systems or other	problems, shear	long extension	dimensional	include	for reactions and	
2D elements	and load bearing	and uniform	frames and	combinations of	internal forces of	
loaded	walls, elements	section such as	trusses with	shell elements,	load bearing	
perpendicularly	loaded in their	retaining walls,	arbitrary loads	slabs, walls, bars,	walls and	
to their planes	planes	pipes etc.		beams and	columns of 3D	
				columns with	structures.	
	~	6		arbitrary loads		
Analysis						
Linear	Linear	Linear	Linear	Linear	Linear	
Non-linear	Non-linear	Non-linear	Non-linear	Non-linear	Non-linear	
Dynamic	Dynamic	Dynamic	Dynamic	Dynamic	Cracked-section	
Cracked-section	Cracked-section		Imperfections	Imperfections		
			Second order	Second order		
			Stability	Stability		
			Seismic	Seismic		
			Cracked-section	Cracked-section		
		RC D	esign	<u>.</u>		
RC slab	RC wall		RC bar	RC shell		
RC beam				RC bar		
Punching				Punching		
	ſ	Steel I	Design	T	ſ	
Steel beam			Steel bar	Steel bar		
				Steel bar-shell		
		Timber	Design			
Timber slab	Timber panel		Timber bar	Timber panel		
Timber beam				Timber bar		
		File fo	ormat	1	-	
.pla	.wal	.pls	.frm	.str	.prd	

Table: The Design Modules

Starting Program

FEM-Design modules can be started from dialogs appear after choosing one of the following ways:

- Click Start button of Windows status bar and select All Programs > FEM-Design...
- Click *Windows Taskbar* > *FEM-Design System Tray* (1), if it is installed by default.
- Double click the short-cut **D**, if it is placed on the Desktop.

Starting a design module (or a new project) offers codes for structure design. Code selection influences:

- the range of available modeling materials,
- automatic combination of load groups, and
- the method and result of foundation, reinforcement, steel and timber design.

General options	R
Code	
Code independent	
🙋 Eurocode	
🙋 Eurocode (NA: British)	
🙋 Eurocode (NA: Danish)	
🙋 Eurocode (NA: Finnish)	
🙋 Eurocode (NA: German)	
🙋 Eurocode (NA: Norwegian)	
🙋 Eurocode (NA: Swedish)	
✓ Query code at start time	
OK Cancel)

Figure: Design codes

The current code of a project can be modified any time during input process (structure and load definition) at *Settings* > *All...* > *FEM* > *Code*. Changing a code will modified all materials and design parameters according to the selected new code by using an editable conversion table.

BASIC CONCEPTS

User Interface

Each module has a similar user interface and contains the following parts:

- Menu bar
- Toolbars
- Tabmenus
- Status bar
- Application window
- Tool palette
- Dialog



Figure: FEM-Design user interface

Menu Bar

The *Menu bar* contains file operations (*File*), drawing (*Draw*) and editing commands (*Edit*), assistant tools (*Tools*), settings (*Settings*), views (*View*), window-system (*Window*) and user-guides (*Help*).

Menu commands having icons next to their names can be organized in *Toolbars* around the *Application window*. Some commands can be executed by hotkey displayed behind the command name.

Toolbars

The Menu bar commands can be grouped in toolbars with their icons and placed next to Application window.

By default, the *Standard* and the *View* toolbars are displayed and the other toolbars are hidden. If you rightclick anywhere on the menu or on one of the displayed toolbars, a list appears, where toolbars can be set visible or hidden.

tomize i ooibar							\checkmark	System
vailable toolbar buttons:			Current toolbar buttons:		Close	12	\checkmark	Draw
Separator	*		Display thickness		Reast		\checkmark	Modify
Zoom in			Separator		neset	IP.	\checkmark	View
Zoom out		Add ->	🗐 Wireframe		Help		\checkmark	Others
Zoom enlarge	=		Hidden line					Customizo
Zoom reduce		<- Remove	📁 Shading		· · · · · · · · · · · · · · · · · · ·		ļ	
Animate zoom			Shading with edges	=	Move Up	IK-	~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~	
Pan	-		Separator	-	Move Down	Ь	$ \leq $	J-AI
	P.		*	•		K	~	

Figure: Available Toolbars and the Customize tool

Customize... allows you to edit the command content of the toolbar, on which the right-click is used.

A toolbar can be moved into new position by dragging it with its "thick vertical line" start.

Tabmenus

Tabmenus contain the main functionalities and their order displays the recommended structural design workflow.

Tabmenu Function		
Structure Tools to define axis and storey system, structural elements, supports and connecti		
Loads Tools to define load cases, loads, load combinations and load groups		
Finite elements Tools to define finite elements and finite element mesh		
Analysis	Analysis Tools to run analysis for the current project and to display results	
RC design Tools to run automatic and manual design for concrete elements and to display re-		
Steel design Tools to run automatic and manual design for steel bars and to display results		
Timber design	Tools to run automatic and manual design for timber elements and to display results	

Table: Tabmenu types

By default, *Tabmenus* have different *Object layer* settings to protect their elements from the others. That means, for example, structural elements (defined at *Structure* tab) are protected against load editing (*Load* tab) although they can selected for load definition. Of course, the available layers of tabs can be customized (Status bar > *Current layer*).

The optional module *Documentation* can be launched from the *Tabmenus* bar.

Status Bar

The *Status bar* is situated under the *Application window* and separated into the following parts.



Figure: The parts of the Status bar

- Co-ordinate box

It shows the exact coordinates of the crosshair cursor in the current *co-ordinate system*. Coordinates can be also given for point/direction definition in a dialog appears after clicking on the *Co-ordinate* box (see *Entering Co-ordinates*).

- "Current layer" button

Clicking the button opens a dialog contains the *layer-system* of the current project. All layer operations like defining new, renaming, color-assignment, showing/hiding and deleting are available in the layer-system dialog. The "*Current layer*" button displays the name of the current *drawing layer*.

- "Current style" button

This button shows the line type currently available for drawing lines, arc, circles, edges etc. (*Draw* menu). Clicking the button opens a dialog, where new line type can be defined or line types can be edited.

- Point/Direction editor

Editor tools provide reference points and lines by using existing lines and points during element definition, drawing and editing.

- Object snap tools

Here you can set the snapping distance and turn object snaps on and off.

- "Current color" button

Here you can set the drawing color independently from the color of the current drawing layer. This color is used for numeric values on result figures too.

"OK" and "Cancel" buttons

These tools approve or reject the current operation.

- Command line

Here you can directly communicate with the program for example by typing coordinate values in it. In the command line coordinates can be defined in several ways (see *Entering Co-ordinates*). *Command line* also displays additional messages to show the steps of the current command.

Application Window

Two main windows types are available in FEM-Design:

- Graphical window

It works as a drawing board and displays the model defined in the current project.

- Detailed result window

Analysis and design results with detailed background calculation formulas (code references), figures and tables can be displayed by single elements or by design groups in separate windows. Quick navigation is powered with zooming details.

An arbitrary number of windows can be opened (*Window* menu) to show for example a model in different views at the same time. The list of the currently available windows is shown at the bottom of the windows (tabs) and in the *Window* menu. You can swap between the windows by clicking on their name tabs or by choosing the requested one in the *Window* menu. Applied windows can be arranged by *Window* > *Arrange*. Right-clicking on a window tab, the window's name can be edited.



Tool Palette and Dialog

In most cases, a command/tool has own palette (*Tool palette*) that contains its definition and setting tools.

Beam	E
_ ₹ ? 1] † †	
	Beam/Column properties
	A.1 General Section End conditions
Default settings	Identifier (.position number)
5	Enter identifier of bar (max 50 characters). The position number is generated automatically when "Tools", "Refresh numbering" function is called. Initially it is set to 0.

Figure: An example for tool palette (Beam command in 3D Structure module)

The parts of a tool palette are:

- Toolbar
 - It gives the editing modes and their additional tools. The main editing modes are:
 - Define: creates new objects according to its current settings ("*Default settings*"). See more *Element Definition*.

Properties: asks and/or changes the properties of selected object(s). See more Object Properties.

- "Default settings"

If this button has 3D shape, then you can set all default parameters for the new object. Otherwise, it only shows the symbol of the function.

- Main settings

The frequently changed parameters of the new object (Define) can be set directly in tool palette.

You can usually use a dialog to modify the settings, the properties of a command; it contains text and number fields where you can set parameter values.

The main difference between dialogs and *Tool palettes* is that you can work beside an opened tool palette, but cannot at a dialog.

There are three main switch types at a dialog or a tool palette: check boxes, radio and chain buttons.

In case chain button exists and it is "Active" (pushed in), any changes made to the related edit box will be transferred to the next edit box automatically. It gives a quick definition of defining surface elements with constant thickness, loads with constant intensities etc.

Surface load	
_! ? 🗆 🗖 i	
q1 [kN/m2] 10.000	Surface load
q2 [kN/m2] 10.000	🧏 ? 🗆 🔟 i
q3 [kN/m2] 10.000 💙	q1 [kN/m2] 10.000
	q2 [kN/m2] 10.000
	q3 [kN/m2] 10.000 ▼

Figure: Function of chain buttons

Program Settings

Clicking *Settings* > *All* opens a dialog with all available settings of the program and the current project.

Settings	General	
General Directories Cursor Cursor Calculation Solid model Display Window	Startup Check for update at start Check for upgrade at start Auto load last drawing Graphic engine OpenGL OpenGL OpenGL - safe mode DirectX - safe mode DirectX - safe mode Check at beat hardware acceleration Remarks >	System Language English Timesave [min] . 15 Undo steps 20 View Enable XP / Vista / W7 style Window background . Hilight colour Filter palette Display information pop-up Animate view changes
Save as default		OK Canc

Figure: Project and program settings

The settings are valid for the current project, but they can be extended for later projects by saving them as default settings. OK closes the dialog and validates the settings for the current project. Save as default set the "selected" project settings available for next projects and new files.

"Selected" means that *Save as default* works only for the branch belongs to the list/folder title selected by the cursor. For example, apply Save as default for "Environment" to save the General, the Directories and the Cursor settings as default, or select "Settings" and click Save as default to store all current settings as default values.

The range of available settings depends on the currently opened *Tabmenu*. For example, all setting are available at launched "Structure" tabmenu, but the settings are reduced with e.g. finite element (Mesh) settings in case of activated "Analysis" tabmenu.

The program stores project and default settings in the *fem.ini* file can be found in the "FEMData" folder of the installed program.

New program release can use the default settings of the previous release, if you keep them after the first running of the new release in a warning message dialog.

Environment

General settings enable the user to adjust the main program settings affect on startup, system, rendering and display style.

- Check for update at start _ By default, the program informs the user if a new official release of the current FEM-Design version is available.
- Check for upgrade at start

It sends a message if a newer commercial FEM-Design version is released. In this case, the newer version can be downloaded, but it can be run only in demo mode till renewing the current license agreement (please contact your local distributor).

- Auto load last drawing

Checking this box opens the last project at program start-up.

- Language

By default, the user interface is displayed in the installation language. Ask your local distributor about the available languages in your country.

- Timesave [min]

It defines the time period in minute for automatic project saving. Read more at *Data safety*.

- Undo steps

The number of executable undo steps can be set in the field. But do not forget, that there are some complex calculation processes (analysis, mesh preparing etc.) *Undo* does not work on.

- Graphic engine

The suitable graphic engine can be set here by choosing "DirectX" or "OpenGL" according to the video card.

If you have any rendering problem with the default graphic engine, apply the other one. In case of further troubles, download the latest driver of your graphic card.

"*Safe mode*" is developed to protect the program from possible video drive crashes. If no graphic problem is detected with the current driver, uncheck this box to take the full advantage of the new powerful graphic engine and to reach the maximum rendering speed.

- Enable XP/Vista style

By default, the user interface is developed in the height of Microsoft Windows Vista and XP fashion. Unchecking the box a simpler interface will be available after restarting the program.

- Window background

The color of the drawing background (graphical windows) can be set here. The default and suggested color is white. Objects having the same color with the background are always displayed in inverse color.

- Display information pop-up

If there is no running command, moving the cursor over an element displays an *Information pop-up* with the element's main properties. Depending on the currently used working mode (*Structure*, *Loads* etc.), pop-up is available for different elements (structural elements, loads etc.) only. For example, the pop-up displays the ID, the material, the thickness, the alignment and orthotropic features for *Plates* in *Structure* mode.



Figure: Information pop-up examples

Directories settings allow the user to specify the location of FEM-Design files.

- Temporary

The program stores temporary files during calculations and mesh generations in the given folder, which will be automatically removed, if the operations are done successfully. Remained temporary files can be erased from the *Temporary* folder after closing the currently opened project. The changes made in *Temporary* directory will affect after the program is restarted.

- Working

The default favorite folder can be set here for the open and save commands.

The style of cross-hair cursor can be set in *Cursor* dialog. It can be full-sized in graphical windows or custom-sized (*Limited*) by the scroll tool.

Drawing

Title dialog allows the user to specify the content of drawing title block (*Draw* > *Title information table*) and the header title of printing (*File* > *Print*). *Project, Description, Designer, Signature* and *Comments* attributes are autotexts. That means, modifying their content updates all titles used in the current project. These five title attributes can be used in documentation and reinforcement list templates with the help of *Field* tool (*Draw* menu).

Title information	Settings	
Geometry Strusoft	Settings Settings Settings Settings Tite Tite Tite Scale Settings Faram A Text D Sold model D Daplay C D Daplay Window	Title Project Concrete Structure Description Geometry Designer Sirusoft Signature Comments
	Save as default	OK Cancel

Figure: Title information table

Scale has effect in displaying texts, line types and wall hatches in graphic windows. Of course, *printing scale* can be set independently from drawing scale.

Param settings allow the user to specify the default initial settings for drawing (*Draw* menu commands) in the current or later projects (*Save as default*). Line type, pen width and drawing color can be edited independently from the default values with the *Status bar* tools "*Current style*", "*Current layer*" and "*Current color*".

Text settings enable the user to set the default font style and sizes for general text items.

	Text setting effect
Drawing Title block	yes
Structural element ID and label	yes
Design data label	yes
Load label	yes
Mass label	yes
Support ID and label	yes

Default settings of <i>Draw</i> > <i>Text</i>	yes
Default settings of numeric result values	yes
Draw > Dimension	no
Grid axis label (<i>Structure</i> > <i>Axis</i>)	no
Storey label (<i>Structure</i> > <i>Storey</i>)	no
Finite element and node ID	no
(Settings > Display > Mesh)	

Table: List of elements on which Text setting has effect or not

Dimension, Axis and *Storey* tools and the finite element display have individual text settings from the general *Text* settings.

EQ3

Script solves the character problems of different languages. For example character "ő" appears as "õ" in "*Western*", but as correct "ő" in "*Central European*" script.

Size is defined in real values (mm), so text size varies on drawings by modifying its Scale.

The *Width* factor 1.0 refers to the normal character width, so factor smaller/greater than 1.0 results a condensed/extended text.

For "italic" style use *Slant* between 10 or 15 degrees.

FEM

FEM includes settings influence structural behavior, finite element mesh generation and calculation.

Code displays the current and available design code for the current project. Code influences:

- the range of available modeling materials,
- automatic combination of load groups, and
- the method and result of reinforcement, steel and timber design.

Changing the current code erases the materials, the design parameters and the results in the project, so these properties have to be redefined to run valid calculations later.

Read more details from other FEM settings in the connected topics:

- "Rigid" values

The default value of "infinite rigidity" can be set for *supports, connections* and *fictitious bars* by types.

- **Mesh and Calculation** Mesh and peak smoothing settings of *automatic finite element mesh generation*. Perform gives additional features to *data safety*.
- **Result** *Options* define the calculation sections of bars and affect on *detailed results* of bars.

Display

The current and default display settings of the drawing elements, numbers, *structural objects*, *loads* and *finite elements/mesh* can be set here.

ettings	Beam, Column and	<mark>∞</mark> I Truss
Drawing FEM Solid model Display Display Local systems Plate and Wall Support Support Connections Mesh Mass Local Support Support Support Wesh Wesh Wesh Wesh Window	Display labels including Dip Position number Material Section name	Graphical options Image: Constraint option opting opting option option option optin
< □ III ► Save as default		OK Cancel

Figure: Display settings (of Beams, Columns and Trusses)

Window

Co-ordinate systems include the display settings of the *Global* and the *User-defined* (UCS) *co-ordinate system* symbols. The symbols are shown in the defined size in working windows, but they can be hidden by unchecking the *Display co-ordinate systems* box.

Grid defines the distribution settings of the visible *Construction grid*. The grid can be hidden by unchecking the *Display grid* box. The grid lines can be set to be parallel with the axis directions of the *Global (Ground)* or *UCS* system.

All default settings of **Object Snap tools** are grouped in Snap dialog.

Units setting offers various unit types for modeling (length, angle, force, mass and cross-sectional data) and for results. Stress and displacement units can be set independently of the length and force units.

Settings	Units
Drawing FEM Sold model Display Window Co-ordinate systems Grid Snap Ints	Units Length
Save as default	

Figure: Available units

Data Safety

Time saving and protection system ensures more the data safety of working files.

Time Save

FEM-Design automatically saves copy of the working file by given time period.

The saving time period can be set in minutes at *Settings* > *All* > *Environment* > *General* > *System* > *Timesave*.

Working on a project file called e.g. "steel construction.str" in the *3D Structure* module with a 15minute time save period, an extra file will be generated and updated by the time period (0, 15, 30min ...) with special extension and in a hidden folder:

Rules of time save:

- In case of a program crash, the last saved copy opens automatically in the right module.
- If you successfully exit from FEM-Design after saving the current project, its time save copy will be erased from the recovery folder.

The location of time save files cannot be modified for data safety.

Additional automatic savings can be set before and after calculations at *Settings* > *FEM* > *Calculation* > *Perform*, but automatic savings overwrite the original project file.

Recovery files

Recovery file command in File menu is for managing time-save files which are generated after some unexpected events. Its tool-window:

Recovery				
File	Date	Size 💽	Version:	11.00.000
untitled_16.str	11/18/2011	463 KB	User name:	femtester
untitled_17.str	11/18/2011	413 KB	File created:	11/21/2011/10:48 AM
untitled_18.str	11/18/2011	0 KB	Last modification:	11/21/2011/11:18 AM
untitled_19.str	11/18/2011	0 KB		
untitled_2.str	10/4/2011/	0 KB		
untitled_20.str	11/18/2011	0 KB		
untitled_21.str	11/18/2011	0 KB		
untitled_22.str	11/18/2011	0 KB		
untitled_23.str	11/18/2011	0 KB		
untitled_24.str	11/21/2011	570 KB		
untitled_25.str	11/23/2011	0 KB		
untitled_26.str	11/23/2011	0 KB		
untitled_27.str	11/29/2011	0 KB 😑		
untitled_28.str	11/29/2011	0 KB 📃		
untitled_29.str	11/29/2011	0 КВ 🛄		
untitled_3.str	10/13/2011	0 KB	· · · · · · · · · · · · · · · · · · ·	•
untitled_30.str	11/29/2011	0 KB		
untitled_31.str	11/29/2011	0 KB		
untitled_32.str	11/29/2011	0 KB 👝		
untitled 33.str	11/29/2011	🛛 КВ. 💌		
Delete				Open drawing Close

Figure: Recovery file dialog

You can open the desired version and delete the useless ones, and you can also set the sorting criterion, by clicking on File / Date / Size.

Protected Work File

All working files are locked not to be modified, overwritten by another user or process. Starting a project, a file having the project name with *.lck* extension is generated automatically, which guarantees the protection till closing the working file.

Opening a protected file sends the warning message: "*The file is locked by another user or process.*" You are allowed to open and modify the content of a protected file, but changes can be saved only under a new name (*File > Save as*).

Element Types

Two main types of elements are available in FEM-Design: *Drawing elements* and (structural) *Objects*. These element types are stored in different *Layer-systems* and have different kind of property and display settings.

Drawing Elements

Туре	Command	Property and Display Settings
Point	• Draw > <i>Point</i>	Settings > Drawing > Param, "Current layer"
Line	Draw > <i>Line</i>	Settings > Display > Drawing elements, "Current style", "Current layer"
Rectangle	Draw > <i>Rectangle</i>	Settings > Display > Drawing elements, "Current style", "Current layer"
Arc	C Draw > Arc	Settings > Display > Drawing elements, "Current style", "Current layer"
Circle	• Draw > Circle	"Current style", "Current layer"
Region	Draw > <i>Region</i>	Settings > Display > Drawing elements, "Current style", "Current layer"
Solid	Draw > Solid	Settings > Display > Drawing elements, "Current style", "Current layer"
Text	$\mathbf{A}_{\mathrm{Draw}} > Text$	<i>Text</i> tool > <i>Default settings</i> ,
Field	□ Draw > <i>Field</i>	<i>Field</i> tool > <i>Default settings</i>
Dimension	Draw > <i>Dimension</i>	Dimension tool > Default settings, "Current style"
Title information table	Draw > <i>Title information table</i>	Text tool > Default settings, "Current style", Settings > Title, "Current layer" (! Object layer)

Table: Drawing elements

Objects

Туре	Command	Property and Display Settings
Axis	Structure Axis	Axis tool > Default settings, "Current layer"
Storey	Structure, OStorey	Storey tool > Default settings
Soil	Structure, Soil	Settings > Display >Soil and foundation, Soil tool > Default settings
Borehole	Structure, Borehole	Settings > Display >Soil and foundation, Borehole tool > Default settings
Isolated foundation	Structure, Isolated foundation	Settings > Display >Soil and foundation, Isolated foundation tool > Default settings
Wall foundation	Structure, Wall foundation	Settings > Display >Soil and foundation, Wall foundation tool > Default settings
Foundation	Structure, 🖾 Foundation slab	Settings > Display >Soil and foundation, Foundation slab

slab		tool > <i>Default settings</i>
Beam	Structure, Ream	Settings > Display > Drawing elements and Beam and Column, "Current style", "Current layer", Beam tool > Default settings
Column		Settings > Display > Drawing elements and Beam and Column, "Current layer", Column tool > Default settings
Intermediate section	Structure, Intermediate section	Intermediate section tool > Default settings Drawing elements and Beam and Column
Apex	Structure, 📿 Apex	Apex tool > Default settings
Truss member	Structure, N Truss member	Settings > Display > Drawing elements and Beam and Column, "Current style", "Current layer", Truss member tool > Default settings
Plate	Structure, Plate	Settings > Display > Drawing elements and Plate and Wall, "Current layer", Plate tool > Default settings
Wall	Structure, Wall	Settings > Display > Drawing elements and Plate and Wall, "Current layer", Wall tool > Default settings
Profiled panel	Structure, for Profiled panel	Settings > Display > Drawing elements and Plate and Wall, "Current layer", Profiled panel tool > Default settings
Timber panel	Structure,	Settings > Display > Drawing elements and Plate and Wall, Timber panel tool > Default settings
Support	Structure, \$ \$2, \$ \$2, \$, Supports	Settings > Display > Drawing elements and Support, Settings > FEM > "Rigid" values, Support tools > Default settings, "Current layer"
Connection	Structure,	Settings > Display > Drawing elements and Connections, Settings > FEM > "Rigid" values, Connection tools > Default settings, "Current layer"
Fictitious bar	Structure, I , Fictitious bar	Settings > Display > Drawing elements, "Current layer" Settings > FEM > "Rigid" values, Fictitious bar tool > Default settings
Load	$ Loads, \qquad \downarrow $	Settings > Display > Load, "Current layer", Load tools > Default settings,
Finite element	Finite elements, Finite element tools	Settings > Display > Mesh, "Current layer", Settings > FEM > Mesh and Calculation
RC bar	RC design , <i>F</i>	Settings > Display > Design, "Current layer", Auto design and Manual design tools > Default settings
Steel bar stiffener	Steel design , , , , , , , , , , , , , , , , , , ,	"Current layer"

Table: Objects

Layers

The FEM-Design layer-system helps you to work in a well-organized way in drawings and in models while constructing and documenting. The layers can be considered as transparent papers put on top of each other will seem to be one drawing. All layers can be reached in a dialog appears by pressing the *Current layer* button.

Layer Types

Regarding the *element types* there are two main layer types: "*Drawing*" and "*Objects*".

Layers Dawing Objects Perforcement Univer C C C C C C C C C C C C C	Status Status Protected Ø. Active Colour. Pen Make current New Pensme Delete	Lyers	Satua Hdden hotecad clover. Pen. Select al Copyota Satiol system	
del 1	Cose	Snow	Statical system Load cases Cose Gose T G Snap + ² :: □,	< 1/2 ⊕ ± ⊾ × Ō

Figure: "Drawing" and "Objects" layers

"*Drawing*" layer contains all drawing elements defined by the *Draw* menu commands or generated from *DWG and DXF imports*. *Drawing* layers can be renamed and erased, and their colors and states can be edited. Only the default "0" layer cannot be deleted and edited. At the same time only one drawing layer called "Current" can be used for drawing. The name of the "Current" layer is appears on the *Current layer* button. The "*DIM*" layer is a special drawing layer for the dimensions. It is automatically created by using the *Dimension* command (*Draw* menu).

"*Objects*" layers are built-in layers for objects like plate, wall, column, supports, loads etc. Each object type has an own layer (e.g. walls are displayed on the "*Wall*" object layer). *Objects* layers are not allowed to be deleted and renamed, but their states and the display colors and pen width of objects can be edited.

Layer Status

Layer Status	Symbol	Function
Hidden	Ð	The layer content is hidden in graphic windows
Protected	6	The layer content is visible, but protected from editing
Active		The layer content is visible and editable in graphic windows

Table: Layer status

Editing Layers

Tool	Function	Available for layer type
Make current	Sets selected layer to be the current one	Drawing
New	Creates new layer in the available layer list	Drawing
Rename	Modifies the name of the selected layer	Drawing
Delete	Deletes the selected layer from the layer list	Drawing
	Δ Delete function cannot be undone	
	All elements will be erased of a deleted layer	
Clear	Removes all elements of the selected layer	Drawing
	Δ Clear function cannot be undone	
Color	Modifies the current color of the selected layer	Drawing, Objects
	Defines the current drawing color	Drawing
	Defines the display color of Objects	Objects
	Color can be modified by elements with	

	Edit > Change appearance	
Pen	Sets the pen width assigned to the selected layer	Drawing, Objects
	Defines the current drawing pen width	Drawing
	Defines the display pen width of Objects	Objects
	Pen width can be modified by elements with	
	Edit > Change appearance	
Select all	Quick selection of all layers	Objects
Deselect all	Deselecting currently selected layers	Objects
Objects	Quick selection structural object layers	Objects
Statical system	Quick selection of Supports, Modeling tools and	Objects
	finite element/mesh layers	
Load cases	Quick selection of load (cases) layers	Objects

Table: Editing layers

The full content of an *Objects* layer can be erased with *Tools* > *Delete all*. The objects deleted with *Delete all* can be restored with *Edit* > *Undo*. *Delete all* erases the objects of protected layers too!

Co-ordinate Systems

Various co-ordinate systems are available in FEM-Design with different function, properties and display settings.

	Global co-ordinate system	User-defined co-ordinate System (UCS)	Local co-ordinate System
Function	Definition of co-ordinates	Definition of co-ordinates	-
	Definition of directions	Definition of directions	Definition of directions
	Interpretation of results	-	Interpretation of results
	-	-	Definition of profiles
Туре	Right-handed Cartesian	Right-handed Cartesian	Each structural object has got
	_	_	Its own right-handed Cartesian
			co-ordinate system
Axes	X, Y, Z	X, Y, (Z)	x', y', z'
Symbol			
Symbol color	Fixed 3 colors,	Fixed 2 colors,	User-defined 3 colors,
	X = green	X = cyan	Settings > All >
	Y = red	Y = orange	Display > Local systems
	Z = blue	_	
Position	Fixed	User-defined:	Fixed for an object
		- View > UCS	
		- <i>linsert</i> moves the origin to the	
		position of the cross-hair cursor	
		- Shift + insert moves origin	
		back to the origin of the Global	
		system	
Direction	Fixed	User-defined:	Changeable:
		- View > UCS	Edit > Change direction
		Shift	C C
		- 🗘 + 🛄 resets the	
		directions to the original state	
Show/Hide	Settings > All > Window >	Settings > All > Window >	By element types:
	Co-ordinate systems	Co-ordinate systems	Settings > All > Display
Symbol size	Settings > All > Window >	Settings > All > Window >	By elements types:
	Co-ordinate systems	Co-ordinate systems	Settings > All > Display

Table: Co-ordinate systems

User-defined co-ordinate system (UCS) is developed in order to easily define coordinates and directions in a 2D user-defined *working plane*.

Point Definition with Co-ordinates

Points can be defined with their co-ordinates in *Descartes* or *Cylindrical co-ordinate systems*.



Figure: Co-ordinate system types for point definition

Point co-ordinates can be defined in various unit types available at *Settings > Units* (*Length* and *Angle*).

The crosshair cursor position can be displayed on the *Status bar* in the *Descartes* or in the *Cylindrical* co-ordinate system by clicking the symbol of the *Co-ordinate box*.

Descartes Orthogonal Co-ordinate System

(Cartesian co-ordinate system)

The individual co-ordinates represent the distance of the point from the origin of the orthogonal co-ordinate system (*absolute*) or from a previously defined point (*relative*) measured along X, Y and Z axes. *Cartesian co-ordinate system* is available for co-ordinate definition both in the *Global co-ordinate* and the UCS systems.

Cylindrical Co-ordinate System

(Polar co-ordinate system)

Point is defined by three components: an angle (a1) and two distances (d1 and d2). The a1 and d1 coordinates define the point position in the XY plane of the *Global co-ordinate* and the *UCS systems*, d2 sets the distance from the XY plane. The angle and the distances can be given from the origin of the co-ordinate systems (*absolute*) or from a previously defined point (*relative*).

Co-ordinate Definition Modes

Point co-ordinates can be defined by typing them in the *Command line* or the *Co-ordinate box* of the *Status Bar*, or just by mouse-clicking combined with the *Object snap tools* in the drawing area or in special points.

Absolute co-ordinate definition

Co-ordinates of a point are defined as distances/positions from a co-ordinate system origin.



Figure: Absolute co-ordinates given in Command line and Co-ordinate box

	Descartes Orthogonal System		Cylindrical co-ordinate system	
	Global co-ordinate	UCS	Global co-ordinate	UCS
	system	system	system	system
		20)	
Command	-	X_Y or X,Y	-	P_a1_d1
line				
		Example:		Example:
		Point (LB, RB = Place point) : 4 2		Point (LB, RB = Place point) : P 45 5
		Point (LB, RB = Place point) : 4,2		
Co-ordinate	Co-ordinates	Co-ordinates	Co-ordinates	Co-ordinates
box				
	X [m]4	X [m] 4	a1 [deg] 45	a1 [deg]45
	Y [m]2	Y [m]2	d1 [m]	d1 [m] 5
	Model Descartes	UCS Descartes	Model Cylindrical	UCS Cylindrical
		L, · L, ·		L, · & ·
	Origin	Origin	Origin	Origin
		1 7	1	\uparrow_2
		3E)	D 1 1 1
Command	-	X_Y_Z or X,Y,Z	-	P_a1_d1_d2
line		F 1		T 1
	Example:			Example:
		Fuint (LB, RB = Place point) : 4 2 5		Point(LB,KB = Place point) : P 45 5 2
		Point (LB, RB = Place point) : 4,2,5		



Table: Absolute co-ordinate definition modes

Rules of co-ordinate definition:

- Co-ordinates given in *Command line* are valid only in *UCS*. But, if the *UCS* and *Global system* are in the same position (same origin and same axis directions), the defined point will be in the similar position in both two systems.
- Decimal point has to be defined as ". " and not ",".
- If you do not give Z value in the 3D modules, its value will be automatically zero.

Relative co-ordinate definition

There are three ways to define a point with given distance from another point (called "relative coordinates").

Relative coordinates can be defined both in the *Descartes* and the *Cylindrical* co-ordinate systems and both in the *Global* and the *UCS* systems.

Relative (A): Distance from the last defined point

Distance from the last point can be defined in the *Command line* or with the *Co-ordinate box*. This mode is useful while creating line/polyline/polygon/surface with the distance from the start/previous point.

Steps	Command Line	Co-ordinate box	
1	Define the start/previous point		
2	2D		
	Type the distance co-ordinates as: R_X_Y or @X,Y	Launch the <i>Co-ordinate box</i> dialog, change <i>Origin</i> to Last point, and give the distance co-ordinates.	
	The second s		
	Example: Beam, Define, Straight line, End point (LB, BB = Place point) : B.6.4	Example:	
		Co-ordinates	
	Beam - Define - Straight line - End point (LB, RB = Place point) : [@6,-4	X [m]	
	3D		
	Type the distance co-ordinates as: R_X_Y_Z or @X,Y,Z	Launch the <i>Co-ordinate box</i> dialog, change <i>Origin</i> to Last point, and give the distance co-ordinates.	

Example: Beam - Define - Straight line - End point (LB, RB = Place point) : R 6 -4 3.5 Beam - Define - Straight line - End point (LB, RB = Place point) : @6,-4,3.5	Last point
	Co-ordinates X [m] 9 Y [m] 4 Z [m] 3.5 UCS Descartes Î Î Î Y Last point Y X

Table: Relative (A) co-ordinate definition modes



Figure: Meaning of Relative (A) way in case of beam and plate definition

Relative (B): Distance from a selected point

With the help of function key and the *Co-ordinate dialog box*, a point can be defined from the point on which the cross-hair cursor is left. The steps of definition:

- 1 Leave the cursor above the required point found by one of the *Object snap tools*, which you would like to define the distance from.
- 2 Click function key.
- 3 Set the distance co-ordinates in the *Co-ordinate dialog* according to a selected co-ordinate system.



Figure: Column definition with the distance from a slab corner or from a beam endpoint (Relative (B) way)

Relative (C): Point defined on line/edge with the distance from the end point

With the help of function key and the *Co-ordinate dialog box*, a point can be placed accurately on a line/edge defined with a given distance from the closer end point of the line/edge. The steps of definition:

- Leave the cursor above the required line/edge found and next to its end point you would like to define the distance from. The → "Nearest" Object snap tool helps you to find the line/edge.
- 2 Click function key.
- 3 Set the distance (d) from the closer end-point you left the cursor in the appeared dialog box.



Figure: Column definition on a beam by using the Relative (C) definition mode

Calculator

If you press while defining numeric values in a field of a dialog box, the Windows *Calculator* comes up. It is automatically filled with the content of the edit box. When you close the *Calculator*, the value calculated or typed in it will be written into the numeric field. It can be used with both *Normal* and *Scientific* views of the *Calculator*.

3D plate	Close
t1 [m] 0.20000 →	Press C 0.20000
t2 [m] 0.20000 1 t3 [m] 0.20000	Instruction Degrees Radians Dight Inv Hyp Reckname Reckname Reckname
	Ave dms Ego MR Ego Bigid V 3 (m)
	sum sin x'y log MS Type 0.3 s cos x'3 nt M+ t + - ht

Figure: Calculator used in numeric fields of dialog boxes

Working Plane

The default working plane – where the points and coordinates are defined – is the following by different *FEM-Design modules*.

		I
Design m	Section Editor	
Global XY plane	Global YZ plane	
Gravity di		
Global Z direction	Global Y direction	

Table: The default working plane by FEM-Design Modules

As the *User-Coordinate system* defines the working plane, its custom position can be set by the UCS definition tools. The \bigcirc *UCS* command (*View* menu or toolbar) gives different definition ways depending on the applied FEM-Design module.

User Co-ordinate System 🛛 🛛			
Move origin			

UCS definition tools	Function	Note
A	Defines vertical working plane by the	Available only in the 3D modules
Axis	global Z axis and a selected axis	
	Defines horizontal working plane in	Available only in the 3D modules
Storey	the plane of a selected <i>storey</i>	
r7	Defines working plane in the	Available in all modules, but it has not
Object plane	plane/reference plane of a selected	effect on the default working plane in 2D
	drawing/structural region	modules, where only the position of the
		origin changes
\diamond	Defines working plane with 3 given	Available in all modules, but it is useful in
3 points	points in their common plane	3D modules, where the working plane can
		be set in arbitrary 3D position with 3
		arbitrary points
∠	Sets the working plane in the global XY	Available in all modules, but it has not
\hookrightarrow Global XY plane	plane	effect on the default working plane in 2D

		modules, where only the position of the origin changes
Global XZ plane	Sets the working plane in the global XZ plane	Available only in the 3D modules
Global YZ plane	Sets the working plane in the global YZ plane	Available only in the 3D modules
↓ Origin	Move the current working plane parallel with its original position into a given point	Available in all modules, but it has not effect on the default working plane in 2D modules, where only the position of the origin changes

Table: UCS - working plane definition tools (in Design modules)



Figure: Working plane set into one of the global coordinate planes



Figure: Custom-defined working planes

It is very important to set correct position (plane) for the structural model. For example, although the default working plane is the Global XY plane in the 3D design modules, place the model of a 2D frame in a vertical (Global Z) plane with the help of a vertical working plane, because the gravity direction is always the Global Z axis direction.



Figure: Correct model (plane) definition of a 2D frame done in the 3D Frame module

The program automatically sets the working plane into the reference plane of a planar element (plate, wall, surface support, surface load etc.), if you use the *Hole* tool to cut *hole in a structural element* or a *load*.



Figure: Hole definition automatically sets the working plane into the plane of the attached object

Grid Systems

The program offers a number of aid tools finding specific points and directions while drawing and editing. These are the *Grid system* and the *Object snap tools*.

Two grid systems are available in the program: Construction grid and Snap grid.

Construction grid

By default, *Construction grid* is the grid visible in the *working plane*. Of course, it can be hidden at the program settings (*Settings > All > Window > Grid*). That grid sets the characteristic sizes of your project.



Figure: Construction grid and its settings

You can snap to the raster points of the *Construction grid* permanently by activating the **** "Raster" Object snap tool** and temporarily by holding down function key. By default, the grid directions are parallel with the **UCS** axis directions, so you can rotate the grid together with the UCS.

Snap grid

Although *Snap grid* is an invisible grid, you can join to its raster points permanently (*Raster tool*) or temporarily as written before at *Construction grid*. The distribution of the grid can be set at *Object snap tools* or at *Settings* > *All* > *Window* > *Snap*. The grid distribution can be set as default for the project/program in the latter setting dialog.



Figure: Snap grid and its settings

The X and Y directions are always parallel with the *UCS* axes. Settings affect the *Snap grid* (*Raster* dialog box):

- Dot cursor

If it is active, a small black dot shows the grid points at applied *Raster* snap.

- Sensitivity

It sets the sensitivity area of the raster snap for both the grids and other snapping tools. The sensitivity is constant while zooming.

Object Snap Tools

The *Object snap tools* finds and joins to special points of drawing elements and objects. The cursor form changes to the symbol of the actually used *Object snap tool*. A snap tool can be activated permanently by clicking its icon on the *Status bar* or checking its box in the *Raster* dialog box. The dialog shows the hotkeys of the snap tools activate the current tool temporarily while holding down the key.

Object snap tool		Function	Hotkey	Example
Select closest/ Farthest point	+ ⁺ +	Two states: finds the closest or the farthest point/element from the overlapped ones	-	
Raster	••	Activates the Snap grip and finds its points	F1	

Orthogonal	Þ.	Defines straight directions parallel with the UCS axes Available for line-type elements	F2	L. I. I. I.
End point	ľ	Joins to point, corner point and endpoint	F3	
Middle	1/2	Joins to line/edge midpoint	F4	
Center	۲	Joins to the center of circle/ arc/circular surface	F5	
Nearest	<u>+</u>	Finds line/edge with its points It joins to line/edge with the closest point to the cross-hair	F6	
Perpendicular	P	Locates the point on an element that forms a perpendicular line to it from the last set point	F7	
Intersection	×	Finds the nearest intersection point of two elements	F8	
Tangent	σ	Locates the point on a circular/ curved element that forms a tangential line from the last set point	F9	

Table: Object snap tools

Settings affect to the *Object snap tools* (*Raster* dialog box):

- Dot cursor

If it is active, a small black dot shows the points found by the applied *Raster* tool.

- Symbol cursor

If it is active, the symbol of the current *Object snap tool* when the cursor finds the similar snap point.

- Sensitivity

It sets the sensitivity area of the raster snap for both the grids and other snapping tools. The sensitivity is constant while zooming. Too high *Sensitivity* value makes more difficult to find a point from the neighbors close to it.

Input Devices

The following figure summarizes the main keyboard and mouse functions. Of course, these functions, which depend on the current working mode (drawing, editing, selection, documentation or no command is running), are mentioned at the related topics of this user manual.



Figure: Keyboard and mouse functions

From the above displayed buttons, the following ones have special functionalities during drawing/editing:

F1 F9	Activate temporarily the Object snap functions.
F11	Defines point on a line or edge by giving distance from the closest end point (<i>Relative</i> (C)).
F12	Inserts a relative point from the position of the crosshair cursor (<i>Relative</i> (B)).
insert	Moves UCS to the position of the crosshair cursor.
Shift 순 + insert	Moves UCS back to the <i>Global co-ordinate system</i> position (original state).
Esc	Quits from the current command.
PgUp	Opens the properties dialog (<i>Default settings</i>) of the current command.
Enter	Confirms data input / repeats the last command / finishes multi-selection.
Space	Restarts the steps of the current command.
Backspace	Goes back to previous step in a multi-step command.

Some commands can be accessed by hotkeys. See those commands in the *Menu bar* or in the *Toolbars*.

Selections

Mouse and keyboard selection

FEM-Design offers various selection modes to select objects for requiring and modifying their properties and for editing. If selection is possible, the available selection modes appear in the *Command line*.

Wall - Properties (LB = Box; RB = Object; Shift + RB = Select type; Ctrl+A = Select all; Ctrl = Multiply selection) :

Use use to restart selection, if you make mistakes while selecting.

Box selection

Rectangular selection box can be defined with two points placed with the \bigcirc mouse button. Depending on the box definition, the box selects elements:

- that are fully in the defined rectangular area
- → the end corner of the box is on the right from the start one,
- that are fully or partly (intersected) in the defined area \rightarrow
- the end corner of the box is on the left from the start one.



Figure: Box selection modes

Object selection

Use \bigcirc on an object to select it directly. If you click on a common part of more objects, the first drawn one will be selected.

Selection of same type elements

It is a quick selection to select visible elements defined with the same command (e.g. selection of all walls created by the *Wall* command). Just hold down and select on element (e.g. one wall element) with \Im , and all same type elements will be selected (e.g. all walls) independently their properties differ or not.

Select all

Depending on the current command, all visible elements (e.g. for the *Edit* > *Move* command) or all visible same type elements (e.g. for the *Properties* tool) will be selected by using $\begin{bmatrix} a \\ a \end{bmatrix}$ and $\begin{bmatrix} A \\ b \end{bmatrix}$ together.

Multiple-selection

In some situations elements cannot be selected by one selection mode or with one selection box. With $\overset{\text{or}}{=}$ the previous selection ways can be combined. Multiple selections can be finished with $\overset{\text{even}}{=}$.

Filter selection

Filter is a pop-up window in the workspace, which helps to select the requested objects easily for all kind of editing and modifying functions. When clicking on it, the following appears:

Filter	x
- Storey	<u>^</u>
Storey 2 (+6.000)	
Storey 3 (+9.000)	Ξ
Beam	
Column	
Analytical element Point support	
- Point support group + - 8	
Line support	
⊡. Identifier	
	-
Reset	
OK Continue Cancel	

With the cursor pointing on Point support group "+", "-" and "&" buttons appear, enabling the user to add or remove point support group elements. If you want to select most, but not all of the point support groups, you should click "+", and then "Continue" for deselecting the few individual supports not needed for the selection.

The "&" button works as an "AND" boolean operator: If you want to select the columns on the second storey, you have to click "+" at column and then "&" at storey 2.

After closing the Filter palette, it can be switched on again by a right click on any toolbar, or on the main menu zone. Another way is available through Settings>All...>Environment>General>View where filter palette can switched on and off:

View
🗹 Enable XP / Vista / W7 style
Window background .
Hilight colour
Filter palette
Display information pop-up
Animate view changes
OK Cancel

User defined filter in Tools menu is a function for saving some frequently used selections. If you define a filter of your own, it will appear at the bottom of the list. When defining, a name has to be given and then desired objects have to be selected. A window appears "Filter has been created." After this, if needed, with properties "?", the name of the block can be modified; with "+" you can add other members, and with "-", you can remove undesired ones.

It also can be exploded with

Filter		X
Firstfilter		OK Cancel Explode
Name	Firstfilter	

At first (like in the figure above) "OK" is grey, but after clicking on Explode button "OK" becomes active. Don't forget to validate the explosion with "OK", before closing this window.

Point/Direction Editors

The program offers tools for defining special points and directions while drawing and editing. You can reach them from the *Status bar*.

Object snap tool		Function	Example	
Multi steps		Fixes reference points defined by the following tools	Define a new point with $X=Y=1m$ distances from the virtual intersection of two predefined lines: Launch Point (Draw menu), and then activate both (\times) and i . Define the virtual intersection of the two lines and place the cursor close to it. Press $[12]$ and type 1 [m] for X and Y in the Co-ordinate dialog box. Pressing on Status bar defines the new point.	Line O7 New point Intersection 1m X 1m
Virtual intersections of lines	;-	Edits the intersection of two lines crossed virtually each other	Start a new line from the virtual intersection of two predefined lines: Launch Line (Draw menu), then activate i, and select the two lines one-by-one. The new line starts from the virtual intersection.	Line O1 Intersection ×
Divider point	+	Edits a point on a straight-line with a	Draw a new line from the quarter point of another	

	given ratio:	line	Start point
	ratio = the distance of	Launch Line (Draw	Start point
	the divider and start	menu) and then activate	, In the second s
	point divided by the	interta), una unen acavate	Divider point
	section length	Define the start	No.
	section length	and end point of the line	ALIO.
		(section), and set the	3
		ration to 0.25 (from the	> End point
		start point). The new line	
		starts from the quarter	
		point of the defined	
		section.	
Parallel with line	Defines direction	Draw a new line parallel	
\rightarrow	parallel with a line	with another line starting	End point
	•	from a point:	· settle
		Launch Line (Draw	allel
		menu), and define the	1. Par
		start point of the new line	ine
		in the point. Activate	New Start point Line
		\rightarrow and coloci the line	
		Click the point again and	
		finally define the ord	
		point of the new line	
Porpondicular	Defines direction	Drazy a nezu line	
from line	perpendicular from a	perpendicular from a fixed	
nom me	line	point of an arc:	Arc End point
	mite	Launch Line (Draw	line
		menu), and define the	Start point (nerpendicular)
		start point of the new line	(
		in the fixed point.	\mathbf{N}
		Activate and select	
		neint again and finally	
		define the end point of	
		the new line	
Tangent from line	Defines tangent	Drazn a nezn line tangent	
Tangent Hom line O	direction from an arc	with an arc and starts from	End point
		a fixed point of the arc.	
		Launch Line (Draw	New.
		menu), and define the	- dipertine
		start point of the new line	
		in the fixed point.	Start point
		d	
		Activate and select	
		ne arc. Click the fixed	1
		define the end maint of	
		the new line	
		uie new line.	

Table: Point and direction editors

Navigation

Thanks to the improved video engine, the advantage of all modern displaying technologies such as live pan, live rotation, live zoom, special display modes and transparency (see the next chapter) is available in FEM-Design.



The suitable graphic engine can be set at *Settings > All > Environment > General > Graphic engine*.

Pan

Panning view on the drawing area can be done in different ways. "Panning" modifies the current view only and not the position of elements.

First start *Animate pan* from the *View* menu or *View* toolbar, then press *O* or *O* and drag the mouse to pan the current view. If you are in panning mode, you cannot edit the model and the currently running command is also paused. Click to exit panning mode and return to editing mode.

However, while in editing mode, you can temporarily simulate *Animate pan* by pressing the O to pan the view. Release the mouse button to stop panning.

With the command *Pan* (*View* menu) the current view can be moved with a displacement vector defined by two points.

Zoom

Different tools allow you to zoom on the current view.

Live zoom

First start *Animate zoom* from the *View* menu or *View* toolbar, then press or and finally move the mouse up to zoom in or move it down to zoom out. If you are in zooming mode, you cannot

edit the model and the currently running command is also paused. Click to exit zooming mode and return to editing mode.

However, while in editing mode, you can temporarily simulate *Animate zoom* by scrolling Off forward to *zoom in* or backward to *zoom out*.

Other zooming tools of View menu

Zooming tools		Function
Zoom margin	53	Fits the view to all visible elements and leaves a 5% additional margin around it
Zoom in	Ð	Enlarges the view area defined with a box and fits it to the current window size
Zoom out	Q	Displays the current view in the defined box area
Zoom enlarge	\odot	Zooms in to the 133% of the actual size
Zoom reduce	Q	Zooms out to the 75% of the actual size

Table: Zooming tools
Orbit

View rotation can be done in different ways. "Orbit" modifies the current view only and not the position of elements.

First start *Orbit* from the *View* menu or *View* toolbar, then press Or or and drag the mouse to turn the model around its center point. If you are in rotation mode, you cannot edit the model and the currently running command is also paused. Click to exit rotation mode and return to editing mode.

However, while in editing mode, you can temporarily simulate *Orbit* by pressing \square together with Orbit to orbit the model. Release the keys to stop rotating.



Figure: Temporary Orbit

Rotation view can be set in a dialog box too with the command 3D Rotation (View menu).



Figure: Rotation by dialog box

Views

Besides of navigation tools, specific built-in 2D and 3D views can be used to set the required project view. All views can be stored with given names (user-defined views).

Built-in views

Just click one of the following view buttons to set a specific 2D view or a general 3D view:



User defined views and Home view

To return frequently to a view of the current project, you can save it as "*Home view*" (*View > Home view > Save as*). *Choose View > Home view > Return* to set this view just with one-click. *Home view* is saved with the project.

Current view can be also saved with names in a dialog appeared by clicking (View > User defined views) or using (++). Define a name and click *Add* to store the view. If you have more than one user defined views, *Select* displays preview of them. To use a stored view as the current one, click *Select and exit*. The list does not contain the *Home view*. User defined views can be saved with the 2D and 3D View commands too.

 \square specifies a section view, which contains objects within a given range (*Tolerance*), while \square defines a space view perpendicular to a given plane. The section/plane can be defined with the following tools:

2D view	3D view
I the □ L the the	
Name Custom2D Tolerance [m] 0.20000	3D Name Custom 3D

Figure: 2D and 3D View commands

Object plane

The section/view plane is parallel with a selected region (*Plate, Wall* etc.) plane. The final direction can be set by moving the cursor at the selected storey. The view direction is always the opposite of the local Z' direction of the selected surface object.

3 points

The section/view plane is defined by adding three points. The first point defines the origin, then the second point defines the X direction together with the first one, and finally the third point defines the Y directional extension of the plane. The last point also defines the view direction (an arrow shows it).

User Co-ordinate System

The section/view plane is set to the X-Y plane of the **UCS**. The view direction is always the opposite of the Z axis of the *UCS*.

With this tool, you can define the position of the section/view plane in a dialog box. At 3D View, Dialog opens 3D Rotation.

	3D Rotation		×
2D view	Plane Bottom	West East	South North
+X view (cross section) Distance from origin [m] 0.00000			

Figure: Section/view plane definitions in dialogs

Swapping among views

Moving one step back from the current view, zoom operations etc., click (View > Previous view) or use $A^{tt} + P$. To go to the next view, click (View > Next view) or use $A^{tt} + N$.

Select view

With this tool, you can select some specific views (Storey view, Axis view and User defined view).



Figure: Example for using Select view command



Figure: List of selectable views

After choosing Storey 2 you can see all the elements belonging to the current level, but for example, if you have high columns going through the whole building, they will belong only to the level where they have

been created. You can return to previous view by clicking on the icon again and then selecting "Return to 3D view".

You can also obtain cross-sections of the whole building. By choosing axis no. 1. (which is the global X axis) in the example above, this is what you can see:



Figure: Cross-section view of the building

The green dots emphasized with the blue arrows in the figure above indicate the truss elements pointing out of the plane determined by the current axis and global Z.

Display Modes

Thanks to the graphic engine, the following display modes (*View* menu or *View* toolbar) are available:

Wireframe

 \square or $\square + \square$ displays the model with all edges and lines drawn, but with no surfaces drawn.

Hidden line

 \Box or A^{R} + \Box displays the model with all edges and lines drawn except those covered by surfaces.

Shading

or A = 1 displays the model with all surfaces shaded according to their assigned layer color. A default (non-editable) light source provides illumination for shading. All non-building elements such as drawing elements (lines, regions, texts etc.), loads, supports and the grid-system (if not hidden) are also displayed in shading mode.

Shading with edges

 \bigcirc or \square + \square displays the non-occluded edges of the model in addition to *Shading* mode.



Figure: The meanings of the view modes

Display thickness

is a quick tool developed to real-time display or hide 3D solid representation of structural objects (*Plate, Wall* etc.). It's a switch that can be turned on/off in the *View* menu/toolbar or by using Att + D.

	ON	OFF
Beam, Column and Truss member	3D solid	Reference line
Plate and Wall	3D solid	Reference plane

Table: The visual effects of "Display thickness" switch status on element types



Figure: The statuses of "Display Thickness"

Hiding Element

The hide and show functions are merged in one command called *Hide/show objects*, which is available in the *View* menu/toolbar or by using Att + H. To hide elements use the *Hide* tool for selecting them. The selected elements are displayed as transparent till closing the command and then they disappear on screen. Resuming the command, the transparent objects represent the previously hidden objects. Apply the *Show* tool to display hidden elements again by selecting them. Click *All* (!) to hide or show immediately (without any selection) every project elements depending on the *Hide* or the *Show* tool is the active.

Hide all stories of the building except the first story and the ground floor.



Figure: Hide tool and box selection

Unhide the second and third stories.



Figure: Show tool and box selection

Transparency

Transparency can be set for all element types such as structural objects, drawing elements, loads, supports etc. Start the *Transparency* command from the *View* menu or *View* toolbar or with $A_{t}^{t} + \frac{T}{L}$. Set the transparency value in the *Set transparency* dialog and then select elements you would like to display as transparent.



Figure: Example for Transparency

To reset transparency settings to opaque for all elements, set the transparency value to 0% and click *All* (!).

At solid (*Display thickness* switch is on) and *Hidden line/Shaded* representation of plates/walls, the finite element mesh can be displayed, if transparency is given for the host surface element.



Figure: Display finite elements in solid and shaded view

To set the transparency of all walls to 50% in a project, select one wall with 2 + 3.



Figure: Transparency command combined with "Select type" selection

Axis

Axes (Axis objects) have multifunction in FEM-Design. They can be used to find special points and to define different kind of planes:

Intersections of axes

Figure: Intersections may define column base points

Working plane



Figure: Working plane defined on an axis by UCS

View planes (2D View and 3D View commands)

Project planes (used in the *model transfer between the FEM-Design modules*)

Wall module can open an axis plane defined in the *3D Structure* model and imports the walls attached to the view.

New axis or axis-system can be defined with the Axis command of the structure tabmenu. Both constant and variable distribution of axes can be defined in the Axis tool palette.

Definition steps

- 1. Set the required horizontal *working plane* (UCS), where you would like to place axis/axes.
- 2. Use the *Define* tool. Define the distribution in the *Multiple axes description* field. If distance is not defined, you can place one axis object as a line on the *UCS* working plane.

Constant spacing: (<i>n</i> -1) <i>xd</i>	where n means the numbers of axes, x is the multiplication sign and d means the distances between the axes.
Variable spacing: (n1-1)xd1,(n2-1)xd2,	where ni means the numbers of axes per different spacing, x is the multiplication sign and di means the different distances.



Figure: Axis-system with constant and variable spacing

- 3. Label type (*Number* or *Letter*), starting character (*No/Letter*) and *Prefix* can be defined for the new axis/axis-system in the *Axis* tool palette.
- 4. Display settings of axis symbols, colors and label can be set at *Default settings*.
- 5. An axis or the start axis of a system can be easily defined as a line with 2 points in the working plane. The distribution orientation can be changed by clicking \bigcirc before the definition of the axis end point. The program places all axes on the *Axes object layer*.

Optional steps:

- 6. The numbering of an axis or one direction of an axis-system can be modified with the *Renumbering* tool. Define new (start) numbering value in *No/Letter* field, and then select the axis you would like to renumber.
- 7. Display settings of the axis symbols can be modified with the *Properties* tool.

Storey

Stories (Storey objects) have multifunction in the 2 *3D Frame*, *3D Structure* and *PreDesign* modules. They can be used to define different kind of planes:

- Working plane



Figure: Working plane defined on a storey by UCS

Project planes (used in the model transfer between the FEM-Design modules)

Plate module can open storey defined in the *3D Structure* model and imports the slabs, the supports and the loads attached to the loaded storey.

New storey or storey-system can be defined with the Storey command of the Structure tabmenu. Both constant and variable distribution of stories can be defined manually in the *Storey* dialog box.

rizo oun	ntal size of building [m] d level [m]	50.0 1.50	x 30.0	
ottor	nmost level [m]	<mark>-6.00</mark>		
#	Storey	Height [m]	Level [m]	^
1	Storey 1	3.000	-3.000	
2	Storey 2	3.000	0.000	
3	Storey 3	3.000	3.000	E
4	Storey 4	3.000	6.000	
5	Storey 5	3.000	9.000	
6	Storey 6	3.000	12.000	
7	Storey 7	3.000	15.000	1
				-

Figure: Storey dialog box

Definition steps

1. Set the *Horizontal size of the building*. This rectangular size will be the floor plane size of the storey object.

The horizontal size and the summa height (see later), which define the whole model size, effect the *automatic wind load* calculation.

2. Set the height of the *Ground level*.

This data will also effect to the *automatic wind load* calculation, because the load values depend on the distance from the ground level.

- 3. Set the height of the *Bottommost level* that will be the lowest level.
- 4. To define the first storey, set a name for it in the *Storey* cell of the first row, and then give the *Height* of the storey. Then, define further required stories in the same way. You can also edit the level of a storey that automatically updates the storey height.



Figure: Bottommost, ground and further defined storeys

- 5. Set the display settings (symbol and label size of the level dimensions) at *Properties*.
- 6. Clicking *OK* generates and displays the storey-system in the project. The program places all stories on the *Stories object layer*.

Creating a storey-system automatically generates view planes by stories. Fast navigation among those storeys can be done with the keyboard or by using *Select view*:



Figure: Navigation by Storey

Find

"Find" can be used for finding structural and analytical elements with a given ID (A.1), some finite elements

, nodes , blocks
 , or user defined filters
 You can scroll down all the members of the current type.

Find	×
A1 🕀 🔁 🕾 🝸	
ID Press RET C.1 C.1.1 C.2 C.2.1 P.1	

You can scroll down all the members of the current type.

STRUCTURE DEFINITION

A model consists of structural objects, *loads* and *finite elements*. This chapter summarizes the definition ways, the properties and the features of the structural objects.

Depending on the current *FEM-Design Module* (license you have), the available Object types are different. Although the structural objects are real 3 dimensional objects, they are 1D members and 2D planar elements (because of the finite element method) having sectional (thickness, profile etc.) and material properties. So, they can be defined as lines or regions. Some elements like point supports can be defined as points.

Туре	Modules where available	Definition mode	Position	Material	Profile	Load- bearing
Soil	÷	Region	Horizontal	Arbitrary	-	Arbitrary
⁽⁽¹⁾ Borehole	→]↓ → Ţ	Point	-	-	-	-
Isolated foundation	•]] •	Regular shape/Solid	Horizontal	Concrete	Arbitrary	Arbitrary
Wall foundation	14	Line	Horizontal	Concrete	Arbitrary	Arbitrary
Foundation slab		Region	Horizontal	Concrete	Arbitrary	Arbitrary
Beam		Line	Horizontal Arbitrary	Arbitrary	Arbitrary	Arbitrary
Column		Point	Martinal	A .1.1.	A.1.1.1	(Support)
	i 🕅 👔	Point/Line	Vertical	Arbitrary	Arbitrary	Arbitrary
Truss <u>member</u>	1	Line	Arbitrary	Arbitrary	Arbitrary	Axial
Intermediate section	11 🔊 🗾	Point	-	Arbitrary	Arbitrary	-
Apex		Line	Vertical/Horizontal	Timber	Rectangle	Arbitrary
Plate		Region	Horizontal Arbitrary	Arbitrary	Constant/ variable thickness	Vertical Axial
Wall		Line	5			(Support)
		Region	Vertical	Arbitrary	variable thickness	Planar
		Line				Arbitrary
Profiled panel	1	Line/Region	Vertical/Horizontal	Concrete	Constant	Arbitrary
Timber panel	•]] 4 •]	Line/Region	Vertical/Horizontal	Timber	Constant	Arbitrary
\$ \$2		D	Vertical			
Point support	🛗 主 🍘 📮 🚹	Point	Arbitrary	-	-	-
1 12		Lino	Vertical			
Line support	🛗 🍺 🍘 🚺	LIIIe	Arbitrary	-	-	-
Surface	*	Region	Vertical			
support (group)		region	Arbitrary		-	
Point-point		Line	Horizontal			
connection	🛗 🏦 🍘 📮 🔒	LITE	Arbitrary		-	-

Line-line connection		Lines	Horizontal Arbitrary	-	-	-
l Fictitious bar		Line	Horizontal Arbitrary	-	-	-
Shell model	*]]4 *]	Regions	Arbitrary	Steel	Constant/ variable web height	Arbitrary

Table: Structural Objects and their main properties

The commands for defining structural objects can be started from the **Structure Tabmenu**. Each command has a **Tool palette** with the customizable element properties (cross-sections, materials, stiffness values etc.) and the definition tools of the element geometry and position (direction).

Properties

Tool palette contains all customizable structural properties. The main properties can be set directly in the tool palette and all properties can be set in the dialog opens by clicking on the *Default settings* button. The settings dialog and fields keep the last set property values by element types (beams, columns, plates etc.)

Identifier (.position number) Identifier (.position number) Enter identifier of bar (max 50 characters). The position number is generated automatically when "Tools", "Refresh numbering" function is called. Initially it is set to 0. Rotation Alpha [deg] Alpha is measured from default direction of y axis. The default direction is usually horizontal and in case of vertical elements points to global y direction.	
direction of y axis. The default direction is usually horizontal and in case of vertical elements points to global y direction.	

Figure: Setting part of Tool palette

Cross-sections

FEM-Design offers the possibility to add any cross-section type/shape (*Section*) to beams, columns and truss members.

To avoid design failures of concrete, steel and timber bars (section type and material do not fit), the program checks the section type - *material* compatibility while setting the properties. The program sends an error message when it finds incompatibility definition at closing the settings dialog. But, the so-called *General material* (defined by the user) can be added to each section types.

The *Section* tabpage of the settings dialog contains predefined cross-sections. Unique profiles and shapes can be define by their parameters (parametric shapes) in *New*> *Size* or with the **I** *FEM-Design Section Editor*. The following table sums the available cross-section types.

Туре	Description	Section library	Compatible material
Standard steel profiles	Not editable, built-in steel profiles depending on the applied national code	Section library Concrete sections Steel sections ⊕ HE-A → 100 → 1200 → 140 → 160 → 180 2	Steel (or General)
Often used concrete profiles	Not editable, built-in square, rectangular and circular concrete profiles	Section library ← Grade ← Retangle ← 120x250 ← 120x250 ← 120x250 ← 120x500 ← 120	Concrete (or General)
Often used timber profiles	Not editable, built-in square timber profiles	Section library ₂ Concrete sections Stell sections Stell sections Imber sections Timber sections Imber sections Gluam -42x90 -42x180 -42x25	Timber (or General)
Common parametric concrete shapes	Predefined concrete shapes to create required profiles by defining the shape parameters	H [mm] 500.0 B [mm] 250.0	Concrete (or General)
Common parametric rolled steel shapes	Predefined rolled steel shapes to create required profiles by defining the shape parameters	h (mm) 300.0 h (mm) 100.0 ts (mm) 10.0 ts (mm) 15.0 r (mm) 5.0 r 2 (mm) 5.0	Steel (or General)
Common parametric cold-formed steel shapes	Predefined cold- formed steel shapes to create required profiles by defining the shape parameters	h [mm] 300.0 b [mm] 100.0 t [mm] 100.0 t [mm] 5.0	Steel (or General)
Common parametric welded steel shapes	Predefined welded steel shapes to create required profiles by defining the shape parameters		Steel (or General)

Common parametric timber shapes	Predefined timber shapes to create required profiles by defining the shape parameters	H (mn) 1000 h (mn) 200 h (mn) 200 h (mn) 200 B (mn) 600 b (mn) 100 b (mn) 100	Timber (or General)
------------------------------------	---	---	---------------------

Table: Section types

Defining a new (e.g. parametric) section in the settings dialog, the program calculates automatically the parameters need for later analysis and design. Each bar element has a local coordinate-system which y and z axes define the plane of its cross-section.

Parameter	Meaning	Example
А	Area	
Р	Perimeter	
A/P	Area/Perimeter	
Yg, Zg	Center of gravity	
Ys, Zs	Shear center position	
Iy, Iz	Moment of inertia	
Wy, Wz	Section modulus	
ez max, ey max	Maximum distance of extreme fiber	
ez min, ey min	Minimum distance of extreme fiber	
iy, iz	Radius of inertia	
Sy, Sz	Maximum statical moment	
It	Torsion moment of inertia	
Wt	Torsion section modulus	
Igamma	Warping parameter	G y
Iyz	Centroidal product of inertia	
z omega	Wagner warping parameter	
alpha1, alpha2	Angle of principal direction	ys Ž
I1, I2	Principal moment of inertia	
W1 min, W2 min	Principal minimum section modulus	2
W1 max, W2 max	Principal maximum section modulus	ev max ev min
e2 max, e1 max	Maximum distance of extreme fiber	
e2 min, e1 min	Minimum distance of extreme fiber	,
i1, i2	Radius of principal inertia	
S1, S2	Principal maximum statical moment	
So1, So2	Principal statical moment of half area	
c1, c2	Plastic/elastic moment capacity	
Rho 1, Rho 2	Principal shear factor	
z2, z1	Wagner parameter	

Table: Sectional characteristics

A cross-section is stored in the following tree structure: *group > type > size*. New library items can be defined or previous ones can be edited (renamed, modified or deleted) by the following settings options.



Figure: Section library structure

Cross-sections used in the project and defined as parametric profiles are grouped in *Used sections* library, but they are available only in the current project.

Section library
Steel sections
 Concrete sections
Timber sections
I User-defined
Used sections
🚊 Steel, rolled
. E · O
T
⊕ UP
. Steel, cold-formed
Steel, welded
- Timber

Figure: "Used sections" library

You can Import and Export the Section Library to share the content of the Library.

Materials

The *Material* tabpage of the settings dialog contains predefined materials according to the current national standard.

A model may include mixture of elements with different materials, and analysis can be done for the complete model, but design can be done for elements having proper materials.

Material	Design
Soil	Foundation design
Concrete	RC design
Steel	Steel design
Timber	Timber design
General	- (only for analysis)
Bar steel	RC design

Table: Available and design materials

Material library stores the available materials by material type groups. Materials used in the project are grouped in *Used materials* library, but they are available only in the current project.

late properties	×
A.1 General ↓↓↓ Material Library □ Concrete □ C12/15 □ C16/20 □ C00/25 □	Application data Ultimate Acc./Seis. Gamma M01.00 1.00
	Gamma M1 1.00 1.00 Gamma M2 1.25 1.00
 → Steel → 5 235 → S 275 → S 355 → S 420 → S 450 	•
New Modify Delete	te Import Export OK Cancel

Table: Material library

Above the standard materials, user can define concrete, steel, timber and general materials. The new materials are also stored in the *Material library* in the proper material type group. To define a new (e.g. concrete) material, select the required material group name (e.g. *Concrete*), click *New* and set the required material properties starting with the material name first.

Com and			S	steel			
	le –	Library material				Timbe	r
ray material immersion immersion (-dc (Umm2) immersion (-dc (Umm2)) is (-dc (U	12.0 1.60 1.10 0.00175 0.00350 27000 0.20000 2.55 0.0000100 Cancel	tame Vidi strength - f,6k, for normal statesses - 1 [b]/mx.] t < 16 mm. 15 < et c = 40 mm. 40 < t <= 63 mm. 40 < t <= 63 mm. 61 < t <= 80 mm. 100 < t <= 100 mm. 100 < t <= 100 mm. 100 < t <= 200 mm. 100 < t <= 200 mm. 200 < t <= 200 mm. 200 < t <= 400 mm.	235 235 215 215 215 215 215 215 215 215 215	Ultimate tensis for nominal bit 3 <= t 40 < t 100 < t 150 < t 250 < t Modulus of el Shear modulu Posson's ratio Mass density Thermal coeff	Librory material Name Quality according to EN 14081-1 f m.0k, (Nimm2) 14.0 f m.0k, (Nimm2) 16.0 f m.0k, (Nimm2) 2.00 f v.k (Nimm2) 3.00	Type E.O.rean (Mmr2) E.S.O.rean (Mmr2) E.G.O.S.Num G.G.O.S.Num Han G.G.O.S.Num Han No k.Rythol Pho mean for Poor Coe	State State State State State State Y material State S density -rho ([htt]] 7.450 Autor of eastroty - ED ([htt]] 7.2000 State of eastroty - ED ([htt]] 2.2000 State of eastroty - Aph ([htt]] 1.2000
	any material any material any material any mpressive strength - fck [V/mn2] and stren - sector cu3 bit 050 [V/mn2] did stren - sector cu3 cost modular of electory - Ecn [V/mn2] assorb indio - na assorb indio - na assorb indio - na astrongen - na	Contracted are	CONNECTED Classy material are	Liney material Liney material are	CONTROL CONTROL Classy material are	Concerner Ubary material Lessy material are material me	Concernent Library material Mail me

Table: Material types

For concrete structures, the partial safety factors γ_c and γ_s can be calculated automatically from Eurocode 2 reductions. Click *Safety factor calculator* next to the safety factors (*Default settings* > *Material* > *Application data*), check the required reduction box and press *OK*.

Library	App	lication data				
Concrete			Ultimate	Acc./Seis		
C16/20	E Gami	ma c	1.50	1.20		Reductions
C20/25	Gam	ma s	1.15	1.00		A 2 1 Reduction based on quality.
	Gam	ma cE		1.20	>	control and reduced deviation
C35/45	Alpha	acc		1.00		A.2.2 Reduction based on using
	Alob			1.00		reduced or measured geometrical data in design
C50/60	Alph	a ct		1.00		A.2.1, A.2.2 Variation of the concre
🖻 Steel	Envir	ronment		X0 -		strength is less than 10%
S 235	Cree	p coefficient		0.000		A.2.3 Reduction based on
S 355	Shrin	nkage [‰]		0.000		assesment of concrete strength in finished structure
S 420	Redu	uction for dynamic	analysis	1.00		

Figure: Safety factor calculator

For reinforced concrete structure the user has the possibility to reduce the element stiffness in order to model the cracking's effect in eigenfrequency calculation.



Figure: Stiffness reduction for dynamic analysis

The properties of the current (selected in the list) material can be edited with the Modify tool.

Customized material database can be shared between projects and users with the Export and Import tools. Click Export to save all materials of the current project in a named database file (.fdlmat or .struxml). To load an exported material database to a project, just apply Import and browse for it.

Library	Application data	
C12/15	Ultimate Acc./Seis.	
C16/20	≡ Gamma M0 1.00 1.00	
	Gamma M1 1.00 1.00	
C25/30		
C35/45	Gamma M2 1.25	
C45/55	File name: custom_material.fdlmat	
E-Steel	Save as type: Material library, (*.fdlmat)	
S 235	Material library, (*.fdlmat)	
S 275	StruSoft XML (*.struxml)	
S 355		
S 450	-	

Figure: Material export/import

Just one click on *Quantity estimation* (*Tools* menu), and a fast process collects all structural elements of the current project with their applied materials, material qualities, identifiers, main geometric parameters (e.g. profiles), quantities etc.



Modifying the national standard for a model the program shows a dialog where you can convert the material property of the structural elements and the reinforcing steel, so the materials will automatically change after switching the code.

-			
lype	Eurocode (NA: Hungarian)	Eurocode (NA: Norwegian)	1
Steel	5 235 (ty = 235)	S 235 (ty = 235)	
Concrete	C20/25 (fyck = 20.00)	C20/25 (fyck = 20.00)	
			-
•			

Figure: Material conversion dialog

Information Pop-up

Moving the cursor over a structural object, an *Information pop-up* appears with its main properties. For example, the pop-up displays the ID, the material, the thickness, the alignment and orthotropic features for *Plates*.



Figure: Information pop-up example

Information pop-up appears only for elements on visible layers and if there is not running command.

Pop-up is enabled by default. To unable it, uncheck the *Display information pop-up* at *Settings* > *All* > *Environment* > *General* >*View*.

"Properties" Tool

With the *Properties* tool of a tool palette, the properties of a selected object or objects can be inquired and edited in dialog format (similar to *Default settings*).



Figure: Modifying object properties (e.g. cross-section of a selected beam)

Display Settings of Structural Elements

The display properties of the structural elements can be set at the *Settings* > *All*... > *Display*.

The available options depend on the current FEM-Design module.

Settings	Shell	
Drawing Calculation Display Drawing element Coal system Soil and Foundation Design data Window	Display labels including Di D Eccentricity Material E2 / E1 Alpha (orthotropic angle) Thickness	Graphical options Display local system Size [m] 1.0000 Hatch [mm] 1.5000 Display wall base line

Figure: Settings options affect on the appearance of the structural elements

- Drawing elements

With the *Show end points of line* option you can show (or hide) the insertion points of planar structural objects, their holes and bar objects (beams, columns etc.). These points are visible in all *display modes*, but the *Wireframe* mode without *displaying the elements' thickness* gives the clearest appearance of them.



Figure: Insertion points displayed in Wireframe mode

E Pr

By showing the insertion points you can also detect the unnecessary (for example accidentally) defined points, which may cause "too refined" finite element mesh in planar structural elements. The unnecessary points can be deleted with editing tools (*Edit* menu) which can be applied for region elements (such as plate, wall regions etc.). For example, stretch the unnecessary points outside the host region, and then cut the stretched region part(s) with the Split tool of the *Modify region* (*Edit* menu) or with the *Hole tool* of the structural planar object.



Figure: Unnecessary definition points detected visually

With the *Fill-up with color* option the planar objects (plate, wall, drawing regions) can be displayed with their reference plane as colored surface or with their contour lines only. Applying the fill colors is useful to display holes/openings clearly. The filling color of an element is the default color of the *Object layer* includes the element. It can modified at the layer settings by element types (Plate, Wall etc.) or with the *Color* option of the *Change properties* tool (*Edit > Properties*) by elements (independently from their types).



Figure: Planar objects displayed with their colored surface or without it in Wireframe mode

- Display labels

Information of structural elements (such as ID, position number, material properties, section names etc.) can be displayed on the screen by element types in *Wireframe display mode*.



Figure: Info labels displayed on screen by element types

- Numbers

This setting affects on the numeric values of the displayed labels (see the previous option).

- Display local system

The *local co-ordinate system* of the structural elements can be displayed with the *Display local system* option by element types. The size of the local system symbol can be set at the *Size* option. The color of the local system axes can be set at *Local systems* setting. The default colors are: green for the local x' axis, red for the local y' axis and blue for the local z' axis.



Figure: Local co-ordinate system displayed on screen by element types

- Special display settings of walls

With the *Display wall base line* option (*3D Structure* and *Predesign* modules only) the bottom end of the walls can be displayed as a hatched surface.

By inactivating the *Display wall height* option (*Plate* module only) only the reference support line (as hatched surface) can be displayed without showing the height-extension that is out from the plates' calculation 2D plane.



Figure: Special display options of walls

- Special display settings of bar elements

The *Display section shape* option shows the applied section of the bar elements as a colored symbol in the middle of the bar reference lines. The size (*Scale*), the filling (*Fill*) and contour (*Border*) colors are customizable. Although, section shape symbols are visible in most of the different display modes, their "best" display mode is the *Wireframe* mode without *displaying the elements' thickness*.

The *Display connections* option shows the end connection property (see *Beam, Column* or *Fictitious bar connection* settings) of the bar elements. Only the free end motion components can be displayed as arrows, where a simple arrow shows a motion component by its direction and a double-headed arrow shows the axis direction of a rotation component. The fix (rigid) end connection components are never displayed. The color of an end motion component equals with the color of the proper axis of the host bar element's local system (see before, *Display local system*). The size of the symbols can be set at the *Size* option.



Figure: Special display options of bar elements

Layer, color and pen width

All elements are placed (and grouped) on *Object layers* according to their type. So, for example, columns are on the "Columns" layer and the walls are on the "Walls" layer. The default color and pen width of elements' contours/reference lines are represented by their host layers. For example, by default, walls are red, if the color of the "Walls" layer is also red.



Figure: Layer-system of structural objects

ΞQIZ

The default color of an object can be modified independently from their types with the *Color* option of the *Change properties* tool (*Edit* > *Properties*).

Geometry

The definition modes and the available shapes of structural elements' reference line or reference plane depend on:

- the structural type: 1D Member or Planar Object, and
- the current FEM-Design module.

The Tool palette of a structural element contains only the available modes. The next table summarizes the geometry possibilities by structural elements.

Туре	Modules where available	Definition mode	Geometry
Soil	14	Reference region	Rectangular
			• Circular
			Delygonal
			Pick lines
			Pick existing region
Borehole		Reference point	- (Insertion point)
Isolated foundation		Regular shape	Rectangular
			• Circular
			Delygonal
			Pick lines
			Pick existing region
		Reference solid	Pick existing solid
Wall foundation	÷ 84	Line	Straight line
			• Arc by center, start and end points
			Arc by 3 points
			Arc by start, end point and tangent
			Line by selection
Foundation slab		Region	Rectangular
			Circular
			Polygonal
			Pick lines
			Pick existing region
Column		Reference point	- (Insertion point)
	i	Reference line	• Line by insertion point
			I Vertical line
			E Select axes
			Select point
			Line by selection

Truss member	Reference line	Straight line
Intermediate section	Reference point	- (Insertion point)
Apex	Reference beams	-
Plate	Reference region	 Rectangular Circular Polygonal Pick lines Pick existing region
Wall	Reference line	Straight line Arc by center, start and end points Arc by 3 points Arc by start, end point and tangent Line by selection
	Reference region	 Rectangular Circular Polygonal Pick lines Pick existing region
	Reference region	 Straight line Arc by center, start and end points Arc by 3 points Arc by start, end point and tangent Line by selection
Profiled panel	Reference region	 Use as plate: Rectangular Circular Polygonal Pick lines Pick existing region
	Reference region	Use as wall: Straight line Arc by center, start and end points Arc by 3 points

		Arc by start, end point and tangent
		Line by selection
Timber panel	 Reference region	Use as plate: Rectangular Circular Polygonal Pick lines Pick existing region
	Reference region	Use as wall: Straight line Arc by center, start and end points Arc by 3 points Arc by start, end point and tangent Line by selection
Point support	Reference point	- (Insertion point)
Line support	Reference line	 Straight line Arc by center, start and end points Arc by 3 points Arc by start, end point and tangent Line by selection
E Surface support	Reference region	 Rectangular Circular Polygonal Pick lines Pick existing region
Point-point connection	Reference points	- (Insertion points)
Line-line connection	Reference lines	 Straight line Arc by center, start and end points Arc by 3 points Arc by start, end point and tangent Line by selection
Fictitious bar	Reference line	Straight line Arc by center, start and end points Arc by 3 points Arc by start, end point and tangent Line by selection
	Reference line	Č.
P		

Cover Wall type Cover Slab or roof type		Reference region	 Straight line Arc by center, start and end points Arc by 3 points Arc by start, end point and tangent Line by selection Rectangular Circular Polygonal Pick lines Pick existing region
Building cover	in the second s	Reference plane shape	-

Table: Structural Objects and their geometry definition

🖌 Straight line

The steps of a straight line definition:

- 1. Define the start point of the line by giving coordinates or mouse-clicking.
- 2. Define the end point of the line by giving coordinates or mouse-clicking.



Figure: Some examples for defining structural objects with Straight line

Although Walls are planar objects with reference line, they are vertical and defined by their base reference line only in the *FEM-Design Plate* and *3D Structure* Modules. The final geometry of the reference region is set by the wall height. The next figure shows the differences of height measuring between the *Plate* and *3D Structure* Modules. Of course, the height defines the position of reference region of curved Walls too.



Figure: Height direction of Walls in Plate and 3D Structure Modules

In 3D Structure Module, the rectangle shape of the wall can be edited by the *Modify region* > *Split region* tool and other editing tools (*Edit* menu). Also the *Hole* tool of Wall tool palette can be used to edit the reference regions.



Figure: Some examples for additional editing of Wall region

• Arc by center, start and end points

The steps of an arc definition with its center, start and end points:

- 1. Define the center point of the arc by giving coordinates or \bigcirc mouse-clicking.
- 2. Define the start point of the arc by giving coordinates or \bigcirc mouse-clicking.

3. Set the drawing direction (clockwise or counterclockwise) with O mouse-clicking. Define the end point of the arc by giving coordinates or O mouse-clicking, or set the central angle (4.) by giving its value. Circle can be defined by angle 360°.



Figure: Some examples for defining structural objects with Arc by center, start and end points



Figure: Drawing direction and angle definition

Arc by 3 points

The steps of an arc definition with its three points:

- 1. Define the start point of the arc by giving coordinates or *mouse-clicking*.
- 2. Define the end point of the arc by giving coordinates or \bigcirc mouse-clicking.

3. Define the third, peripheral point of the arc by giving coordinates or \bigcirc mouse-clicking.



Figure: Some examples for defining structural objects with Arc by 3 points

• Arc by start, end point and tangent

The steps of an arc definition with its start, end point and tangent:

- 1. Define the start point of the arc by giving coordinates or \bigcirc mouse-clicking.
- 2. Define the end point of the arc by giving coordinates or \bigcirc mouse-clicking.
- 3. Set the tangent side with O mouse-clicking. Define the tangent direction from the start point with a third point (e.g. a point on a tangentially connected line) by giving coordinates or O mouse-clicking.



Figure: Some examples for defining structural objects with Arc by 3 points



Figure: Although same definition points are defined, the tangent side is different

Line by selection

The step of a reference line (straight or arc) definition by selecting a previously defined line: Select lines or region (drawing or structural object) edges define the requested shape of the reference line with one of the *selection modes*.



Figure: An example for defining Wall by selecting all lines



Figure: Defining Beams and Walls on Plate edges (1.) and Line supports on Wall edges (2.)

For *Columns* in FEM-Design 3D Modules, only vertical lines or edges can be selected.

 \mathbb{A}


Figure: Column definitions by selecting vertical lines/edges

Line by insertion point and height

As a simple definition of Columns in **3D** *Structure*, **3D** *Structure*, **3D** *Frame* and **1** *PreDesign* modules define the position of the Column with its insertion point and height. The positive or negative value of the Height sets the measuring direction of the height.



Figure: Column definition with one-click in 3D modules

E Select axes

This definition mode is for defining *Columns* in the intersections of selected axes. Select axes with one of the *selection modes*.



Figure: An example for defining columns with selecting axes

Vertical line

This is a special tool for defining Column with two points. FEM-Design Columns can be only vertical, so the second point is not needed to be on the column axis, because its vertical distance from the start (first) point defines the height of the column. This Column definition tool is useful, when you do not know the height value of the new column, but points and lines defines the column's vertical extension (height) can be easily found. Use *Object Snap* tools to find point defines indirectly the column's end point.



Figure: Examples for defining vertical columns with two points

Rectangular

The steps of a rectangular region definition:

I. Rectangle's edges parallel with the UCS:

- 1. Define the point of the first corner by giving coordinates or mouse-clicking.
- 2. Define the point of the end corner by giving coordinates or mouse-clicking.



Figure: Defining rectangular Plate and Surface support

- II. Rectangle's edges not parallel with UCS:
 - 1. Click one or two times to define the rectangle's first edge's direction
 - a. If you click ______ one time, the direction is can be defined from the global coordinate system's origin,
 - b. If you click one times, the direction is defined in an arbitrary direction.
 - 2. Define the arbitrary direction of the rectangle's one line.
 - 3. Define the point of the first corner by giving coordinates or mouse-clicking.
 - 4. Define the point of the end corner by giving coordinates or mouse-clicking.



Figure: Defining rectangular plate



The geometry of rectangular regions as well as other (later mentioned) region shapes can be edited by the *Modify region* > *Split region* tool and other editing tools (*Edit* menu). Also the *Hole* tool of planar objects' definition command can be used to edit the reference regions.



Figure: An edited rectangular Plate region

Arbitrarily positioned rectangle regions (e.g. plate) can be defined. The following picture shows how to define two different leaning rectangles between two points by moving the cursor on different paths.



Figure: Definition of two different leaning rectangles

OCircular

The steps of a circular region definition:

- 1. Define the center point by giving coordinates or mouse-clicking.
- 2. Define the radius by giving its value or a point on the circle (with coordinates or mouse-clicking).



Figure: Defining circular Plate and Surface supports

Delygonal

The steps of a polygonal region definition:

- 1. Define the points of the polygon vertexes by giving coordinates or \bigcirc mouse-clicking.
- 2. Close the polygon with *mouse-clicking or* key.



Figure: Defining polygonal Plate and Surface support

- Pick lines

With this method, previously defined or imported (DWG/DXF) drawing elements can be used as sketches of structural region shapes. The step of definition:

Select a closed line defines the requested shape of the reference region with mouse-clicking.



Figure: Defining Plate and Surface support by using close contours

In case of line junctions, more than one line has to be selected to make clear the continuity of the requested closed contour.



Figure: Selection of more lines to define the right path for the closed shape

□Pick existing region

The step of a reference region definition by selecting a previously defined object or drawing region: select the region(s) defines the requested shape of the reference region(s) with one of the *selection modes*.



Figure: An example for placing Surface support under a Plate by selecting its region

E Br

With *Pick existing region*, shell (*Plate*) elements can be easily place on the entire surface or some surface components of a *Solid* body (*Draw* > *Solid*).



Figure: Defining shell (Plate) regions by picking the surface regions of a Solid body

Hole

Holes, openings and cuttings can be added to reference regions (Plate, Wall and Surface support) with the *Hole* tool. The following geometries can be used for holes:

The steps of a hole definition:

- 1. Select the host region with mouse-clicking. Clicking a region places the *UCS* into the region plane, so giving hole coordinates needs only X and Y values from the UCS origin.
- 2. Define the geometry of the hole with one of the following geometry modes:
 - Rectangular
 - **⊙** Circular
 - Delygonal





Figure: Examples for holes



Figure: Hole tool can be used to edit region geometries as cuttings

Holes can be easily copy inside a region or among regions with same reference plane position with the *Copy command* (*Edit* menu). It is recommended to display only the regions' reference plane (inactive *Display thickness* option) to easily select the contour of the hole you would like to copy. To set the distances/new positions, the *UCS* has to be in the plane of the host region(s).



Figure: Copying holes in a Wall

Direction

Numerous objects need direction settings for their positioning (bar elements) or their components definition (supports). The next table summarizes the direction possibilities by structural elements.

Туре	Modules where available	Direction for	Direction Modes
Soil	*1•	-	-
Borehole		-	-
Isolated foundation	* T a	Solid position: Direction of local x' axis	 ∠ Predefined direction ∠ Local system
Wall foundation		-	-
Foundation slab		-	-
Ream Beam		Cross-section position: Direction of local y' axis	 Predefined direction Parallel with line Perpendicular to plane
Column		Cross-section position: Direction of local y' axis	 Predefined direction Parallel with line Perpendicular to plane
Truss member		Cross-section position: Direction of local y' axis	Predefined direction Parallel with line Perpendicular to plane
Foint support		Rotation direction	The Predefined direction
		Motion component direction	1 Parallel with line
	r 1	Component direction	▲ Perpendicular to plane
Point support		Component direction	- (Predefined direction)
		Component direction (reference system)	 └→ Predefined direction └→ Local system
Line support		Rotation direction	T Predefined direction
		Motion component direction	† Parallel with line
		Component direction	Perpendicular to plane
Line support		Component direction	- (Predefined direction)
		Component direction (reference system)	 Predefined direction Parallel with line Perpendicular to

			plane
Point-noint		Component direction	C Predefined
connection	ò		direction
			🖾 Local system
E Line-line		Component direction	C. Predefined
connection	ň		direction
			🖾 Local system

Table: Structural Objects and their direction settings

† Predefined direction

With this option an axis/a plane of the *Global* or the *User-defined* (*UCS*) co-ordinate system can be set for the required direction (e.g. the local y' axis of a beam profile). The direction can be chosen from the drop-down list attached to the *Predefined direction* option. The available directions depend on the applied FEM-Design Module (e.g. *Plate, 3D Structure* etc.).

Beam	Symbol	Meaning of direction	System
		Parallel with XY plane	Global
		Parallel with YZ plane	Global
	Ĩ Î Î	Parallel with XZ plane	Global
Column Image: Column (Column) Image: Column (Column) <t< th=""><th> ₩ </th><th>Parallel with UCS (XY plane)</th><th>UCS</th></t<>	 ₩ 	Parallel with UCS (XY plane)	UCS
	 <=>	Perpendicular to UCS (XY plane)	UCS
Point support	íl, íl,	Parallel with global X axis	Global
	F [,] N	Parallel with global Y axis	Global
		Parallel with global Z axis	Global
	\rightarrow	Parallel with X axis of UCS	UCS
		Parallel with Y axis of UCS	UCS

N N		
~ _¬	Parallel with Z axis of UCS	UCS

Table: The available axis directions to set the new direction



Figure: Examples for Beam profile positions

At supports, additional direction setting can be chosen from the definition tool palette:

"Positive direction": The orientation is the same with the selected axis orientation;

"Negative direction": The orientation is the opposite of the selected axis orientation.



Figure: Examples of Point support position

For some support and connection types, only the reference system can be selected as "predefined direction". That means the direction of the components (x', y' and z') are valid in the selected co-ordinate system (Global or UCS).



Figure: Support components in different co-ordinate systems

† Parallel with line

With this option, the required direction (e.g. the local y' axis of a column profile) can be defined manually with its start and end points. The new direction is set for all objects till stops the repetition of object insertion and restarts with another direction definition.



Figure: The local y' axis of column profiles is parallel with the defined direction

1 Perpendicular to plane/line

With this option, the required direction (e.g. the local y' axis of a beam/column profile) will be perpendicular to a defined plane/line. The plane can be given with three points and the line with two points (start and end points). In case of the perpendicular plane, the third point defines the final orthogonal direction, which the new direction will be parallel with. The new direction is set for all objects till stops the repetition of object insertion and restarts with another direction definition.



Figure: The local y' axis of beam profiles is perpendicular to the defined plane



Figure: The local y' axis of column profiles is perpendicular to the defined line

🖾 Local system

When the *Global* or *User-defined* (*UCS*) co-ordinate system is not enough as a predefined system for component directions (*Point support, Point-point connection* and *Line-line connection*), a custom local system can be defined. Tree points declare the system and the 4th point places the new object:

1st point: system origin,

2nd point: defines the x' axis direction,

3rd point: defines the y' axis direction.

The z' axis of the system always perpendicular to the x'y' plane.

The new system directions are set for all objects till stops the repetition of object insertion and restarts with another system definition.



Figure: Point support group set by a special local system



Figure: Typical example for special direction (Point support group)

Change Direction

Any previously set direction can be modified by the editing commands (*Edit* menu): *Change direction* and *Rotate*.

Change direction uses the *Predefined direction, Parallel with line* and *Perpendicular to plane* direction definition tools.



Figure: Beam position (local y' axis) is modified with Change direction

Rotate edits a selected direction or the main direction of a selected system with giving new direction points or the rotation angle. Rotation works around a given point or an axis.



Figure: Rotate modifies the main direction (x') of the Point support group

1D Members

This chapter summarizes all features and properties of a 1D member such as *Borehole, Isolated foundation, Wall foundation, Beam, Column, Truss member* and *Fictitious bar*.

Borehole

Borehole property	Description
Modules where available	
Geometry	Straight

Table: Borehole properties

Definition steps

- 1. If needed (and available in the current FEM-Design module), set a proper position for the *working plane*. In the 3D design modules, respect that the gravity direction is always the Global Z axis direction.
- 2. Start ⁽¹⁾ Borehole command from ^(Structure) tabmenu and choose ⁽¹⁾ Define.
- 3. Place a borehole. There are two way to place a borehole:
 - If "Use closest bore hole data" is checked, the borehole properties will be the same as the nearest borehole.

Borehole		×
\bigcirc	Use closest bore hole data	

• If "Use closest bore hole data" is unchecked properties of the new *Borehole* can be set at *Default settings*:

Borehole	x
A.1 General Strata	
Identifier (.position number)	BH
Ground level [m]	0.0000
Foundation level [m]	0.0000
Water level [m]	0.0000

- Identifier (General)

The program automatically generates it, but you can define custom value. Identifier (ID and Position) number can be displayed in model view (*Display settings*).

- Ground level (General)
- Foundation level (General)

This level is where the user places the foundation object.

- Water level (General)

Water level has effect to the design calculations. Soil and foundations which are under the water level, FD calculates with modified density.

- Strata

Optional steps:

- 1. Modify the borehole properties with the *Properties* tool of the *Borehole* tool palette.
- 2. Set the display settings of beams at *Settings* > *All* > *Display* > *Soil and Foundation*.



3. The boreholes are stored on "Soil" *Object layers*. At layer settings, the default color and pen width can be set for all beams.

With boreholes the user can define inclined level strata by setting the strata levels.



Isolated foundation

Isolated foundation property	Description
Modules where available	
Geometry	Regular shape, Solid
End connection	Rigid and hinged, or custom defined by <i>Point-point connection</i>
Material	Concrete
Load direction	Arbitrary in
Load type	Point load (force and moment), Point support motion
Available analysis results	Displacement, stresses, stability and vibration shape in
Available design	Foundation design

Table: Isolated foundation properties

Definition steps

- 1. If needed, set a proper position for the *working plane*.
- 2. Start Isolated foundation command from Structure tabmenu and choose 🖳 Define.

Isolated foundation
A.1 General
Identifier (.position number)
Enter identifier (max 50 characters). The position number is generated automatically when "Tools", "Refresh numbering" function is called. Initially it is set to 0.
Analytical system Point support group
Bedding modulus [kV/m2/m]

- *Identifier* (General)

The program automatically generates it, but you can define custom value. Identifier (ID) number can be displayed in model view (*Display settings*).

- Analytical system (General)

Here the user can choose how to model the *Isolated foundation* depends on if "*Calculate soil as a solid element*" is chose or not. If "*Calculate soil as a solid element*" is not chosen in "*Code, Configuration*" dialog, user can choose if he/she wants to model the *Isolated foundation* by line support group or by surface support group. In this case bedding modulus can be set in *Default settings* / *General* dialog.

"Calculate soil as a solid element" is unchecked

ical systemPoint support group	Analytical system
	Point-point connection
Point support group	Surface support group
"Calculate soil as a sol	lid element" is checked
Analytical system	Supported by soil
Bedding modulus [ktv/m2/m]
	Point-point connection Surface - surface connection

- Material

inaly

Only concrete material can be set for isolated foundation Analysis.

- 4. Choose a *geometry* definition method.
- 5. Define the Wall foundation in the model view based on the chosen geometry method.

Optional steps:

- 6. Modify the isolated foundation properties with the *Properties* tool of the *Isoltated foundation* tool palette.
- 7. Set the display settings of isolated foundations at *Settings > All > Display > Beam, Column and Truss.*



8. The Isolated foundations are stored on "Foundation" and if they modeled with *Surface support group* their supports will be stored at the "Support" *Object layers*. The color and pen width settings by selected beam elements can be modified by *Edit* > *Properties* > *Change appearance*.

Wall foundation

Wall foundation	
property	Description
Modules where available	
Axis position	Arbitrary (horizontal, vertical and skew)
Geometry	Straight and curved
Cross-section	Arbitrary profile
Eccentricity	Available
End connection	Rigid and hinged, or custom defined by <i>Point-point connection</i>
Material	Concrete
Load direction	Arbitrary in 🛄
	Point load (force and moment), Line load (force and moment), Line temperature
Load type	variation load, Line stress load, Line support motion
Available analysis results	Displacement, stresses, stability and vibration shape in
Available design	Foundation design

Table: Wall foundation properties

Definition steps

- 1. If needed (and available in the current FEM-Design module), set a proper position for the *working plane*.
- 2. Start Wall foundation command from Structure tabmenu and choose 🛃 Define.
- 3. Set the properties of the new *Wall foundation* at Default settings:

W	Vall foundation
	A.1 General Section End conditions
	Identifier (.position number)
	Enter identifier (max 50 characters). The position number is generated automatically when "Tools", "Refresh numbering" function is called. Initially it is set to 0.
	Analytical system
	Bedding modulus [ktv/m2/m] 10000

- Identifier (General)

The program automatically generates it, but you can define custom value. Identifier (ID) number can be displayed in model view (*Display settings*).

- Analytical system (General)

Here the user can choose how to model the *Wall foundation* depends on if "*Calculate soil as a solid element*" is chose or not. If "*Calculate soil as a solid element*" is not chosen in "*Code, Configuration*" dialog, user can choose if he/she wants to model the Wall foundation by line support group or by surface support group. In this case bedding modulus can be set in *Default settings / General* dialog.

"Calculate soil as a solid element" is unchecked



"Calculate soil as a solid element" is checked



- Section

Choose a profile from the Section library or define a new cross-section by parameters. The applied section name can be displayed in model view (*Display settings*).

- Connections

The end connections can be set under the *Connections* tab similarly to *Beams*. Only the released motion (arrow) and rotation (two-headed arrow) components are displayed in the model view (*Display settings*). Rigid connections can be customized to hinged or semi-rigid (spring) with additional *Point-point connection*.

The end connections (in the beam start and end) with joined objects can be set to rigid or hinged by motion and rotation components. The component directions are valid in the beam's *local co-ordinate system*.

Inactive component means a fixed motion/rotation, while a checked component behaves as free motion/rotation in the assigned direction.



Only the released motion (arrow) and rotation (two-headed arrow) components are displayed in the model view (*Display settings*).

- Eccentricity

Eccentricity has effect in calculation and it can be turn on/off with *Consider eccentricity in calculation* option. The default position (center of gravity) of the beam axis (local x' axis) can be set in the aspect of analysis/design calculation or accurate display.

The axis position (symbolized with yellow point in the section) can be set with its coordinates in the local y'z' plane, or can be placed into special section points (e.g. plate corners) by dragging the yellow point into the requested position.



Two type of eccentricity option can be chosen:

First type ("End releases applied at the ends of the theoretical axis") is typically used for modeling RC bars which work together with concrete slabs (slab normal forces transfers to the bar).

The second type ("End releases applied at the ends of the gravity (physical axis)") is for modeling single plate and bars which are not working together.

- Material

Only concrete material can be set for isolated foundation Analysis and Design.

- 4. Choose a *geometry* definition method for the wall foundation reference line (axis).
- 5. Define the Wall foundation in the model view based on the chosen geometry method.

Optional steps:

- 6. Modify the geometry with the *Edit* menu commands valid for line elements.
- 7. Modify the bar properties with the *Properties* tool of the *Wall foundation* tool palette.
- 8. Set the display settings of beams at *Settings* > *All* > *Display* > *Soil and Foundation*.



9. The Wall foundations are stored on "Foundation" and if they modeled with *Surface support group* their supports will be stored at the "Support" *Object layers*. The color and pen width settings by selected beam elements can be modified by *Edit* > *Properties* > *Change appearance*.

Beam

Beam property	Description		
Modules where available			
Axis position	Horizontal in		
	Arbitrary (horizontal, vertical and skew) in 🛄 🝘 💼		
Geometry	Straight and curved		
Cross-section	Arbitrary profile		
Eccentricity	Available		
End connection	Rigid and hinged, or custom defined by <i>Point-point connection</i>		
Material	Steel, concrete, timber and general		
Load direction	Vertical in		
	Arbitrary in 🛄 🚳 🧴		
Load type	All point and line load		
Available analysis results	Displacement, internal forces, stresses, stability and vibration shape in 🚈 🛺		
Available design	Steel design in 🖆 🔃 🚳		
	RC design in 🚰 💷 🕅		
	Timber design in 🚰 🛺 🕅		
Alternative	Steel beam can be modeled/designed as the set of shells (<i>Shell model</i>)		



Typical 2D and 3D frame structures build by beams (only) can be fast defined with the so-called *Wizard* tool. Fast means, that in a dialog you can set the beam sizes and positions by parameters. Besides the geometry, Wizard adds loads and supports to the beam-system.

Definition steps

- 1. If needed (and available in the current FEM-Design module), set a proper position for the *working plane*. In the 3D design modules, respect that the gravity direction is always the Global Z axis direction.
- 2. Start Beam command from Structure tabmenu and choose Define.



3. Set the properties of the new beam at *Default settings*:

B	Beam/Column properties 🛛 🔀			
	A.1 General Section End conditions			
Identifier (.position number) B				
Enter identifier of bar (max 50 characters). The position number is generated automatically when "Tools", "Refresh numbering" function is called. Initially it is set to 0.				

- Identifier (General)

The program automatically generates it, but you can define custom value. Identifier (ID and Position) number can be displayed in model view (*Display settings*).

- Section

Choose a beam profile from the Section library or define a new cross-section by parameters. The applied section name can be displayed in model view (*Display settings*).

- Connections

In ¹, the end connections are fully rigid.

In 3D modules, the end connections can be set under the *Connections* tab similarly to *Beams*. Only the released motion (arrow) and rotation (two-headed arrow) components are displayed in the model view (*Display settings*). Rigid connections can be customized to hinged or semi-rigid (spring) with additional *Point-point connection*.

The end connections (in the beam start and end) with joined objects can be set to rigid or hinged by motion and rotation components. The component directions are valid in the beam's *local co-ordinate system*.

Inactive component means a fixed motion/rotation, while a checked component behaves as free motion/rotation in the assigned direction.

Releases	Releases	Releases	Releases
e,x'	e,×'	e,×'	✓ e,×'
e,y'	e,y'	🔲 e, y'	💌 e,y'
e,z'	e,z'	e,z'	
🗌 phi,×'	🔲 phi,×'	🔲 phi,×'	💽 phi,×'
🔲 phi, y'	💽 phi, y'	💽 phi,y'	💽 phi, y'
🗌 phi,z'	🔽 phi,z'	phi,z'	🔽 phi, z'
Fully rigid	Hinged in 3D	Hinged in 2D	Free end

The local x' axis always points the beam end, so the start and end point position can be followed from the beam's local co-ordinate system.



Only the released motion (arrow) and rotation (two-headed arrow) components are displayed in the model view (*Display settings*).

The connection settings is not available in *b*, the end connections are always fully rigid. Rigid connections can be customized to hinged or semi-rigid (spring) with additional *Point-point connection*.

- Eccentricity

Eccentricity has effect in calculation and it can be turn on/off with *Consider eccentricity in calculation* option. The default position (center of gravity) of the beam axis (local x' axis) can be set in the aspect of analysis/design calculation or accurate display.

The axis position (symbolized with yellow point in the section) can be set with its coordinates in the local y'z' plane, or can be placed into special section points (e.g. plate corners) by dragging the yellow point into the requested position.



Default: "center of gravity"

Position modified by the mouse

Two type of eccentricity option can be chosen:

First type ("End releases applied at the ends of the theoretical axis") is typically used for modeling RC bars which work together with concrete slabs (slab normal forces transfers to the bar).



Figure: Modeling of a "ribbed concrete slab"

The second type ("End releases applied at the ends of the gravity (physical axis)") is for modeling single plate and bars which are not working together.



Figure: Modeling of a steel beam under slab in 3D Structure module

- Material

Any type of materials can be set for beam *Analysis*, but design can be run for concrete, steel and timber beams only. The applied material name can be displayed in model view (*Display settings*).

4. Define the *direction* of the cross-section y' axis (only in 3D modules).

The y' axis cannot be modified in the $\stackrel{\text{less}}{=}$ module, so it is always parallel with the Global X-Y (working).

- 5. Choose a *geometry* definition method for the beam reference line (axis).
- 6. Define the beam in the model view based on the chosen geometry method.

Optional steps:

- 7. Modify the geometry with the *Edit* menu commands valid for line elements.
- 8. Modify the bar properties with the *Properties* tool of the *Beam* tool palette.
- 9. Set the display settings of beams at *Settings* > *All* > *Display* > *Beam, Column and Truss.*



10. The beams are stored on "Beams" *Object layers*. At layer settings, the default color and pen width can be set for all beams. The color and pen width settings by selected beam elements can be modified by *Edit* > *Properties* > *Change appearance*.

Column

Column property	Description	
Modules where available		
Function	Point support in	
	Column in 🛄 🚳 🧴	
Position	Always vertical	
Geometry	Straight	
Cross-section	Arbitrary profile	
Eccentricity	Available	
End connection	Rigid and hinged, or custom defined by <i>Point-point connection</i>	
Material	Steel, concrete, timber and general	
Load direction	Arbitrary in 🛄 🚳 🛅	
Load type	All point and line load in 🛄 🚳 🧴	

Available analysis results	Reaction forces in	
	Displacement, internal forces, stresses, stability and vibration shape in 🛄 🕅 直	
Available design	Steel design in 🚑 🚳	
	<i>RC design</i> in 📰 🗺 (Column punching is available in 📩	
	Timber design in 📑 🗐	
Alternative	Steel column can be modeled/designed as the set of shells (Shell model)	

Table: Column properties

Definition steps

1. If needed (and available in the current FEM-Design module), set a proper position for the *working plane*.



3. Set the properties of the new column at *Default settings*:

Column proper	ties		
A.1 General	Section	Material	
Identifier (.pos	ition no.)	d	
y' direction	z' direction	Placement	Beam/Column properties
OFixed	O Fixed	O Above slab	
 Hinged 	📀 Hinged	💽 Under slab	A.1 General Section End conditions
	<u> </u>	h[m] 3.0000	Identifier (.position number) B Enter identifier of bar (max 50 characters). The position number is generated automatically when "Tools", "Refresh numbering" function is
,,,,,, +	,,,,,, +		called. Initially it is set to 0.
○ Fixed	O Fixed		
⊙ Hinged	 Hinged 	Allow uplift	

- *Identifier* (General)

The program automatically generates it, but you can define custom value. Identifier (ID and Position) number can be displayed in model view (*Display settings*).

- Allow uplift (General)

Available in ² only. "Uplift" behavior can be modeled by activating this option. It works in case of tensional reaction force.



In 3D modules, "uplift" can be modeled with non-linear *Point support* settings.

- Section

Choose a column profile from the Section library or define a new cross-section by parameters. The applied section name can be displayed in model view (*Display settings*).

- Connections

In *[10]*, the support conditions (*General*) of the top and bottom ends can be set to fully rigid or fully hinged. Because, *Columns* are point supports further point support is not needed to define at *Column* end.

In 3D modules, the end connections can be set under the *Connections* tab similarly to *Beams*. Only the released motion (arrow) and rotation (two-headed arrow) components are displayed in the model view (*Display settings*). Rigid connections can be customized to hinged or semi-rigid (spring) with additional *Point-point connection*. The local x' axis always points the column end, so the start and end point position can be followed from the column's local co-ordinate system.

- Eccentricity

In 3D modules, eccentricity can be set under the *Eccentricity* tab similarly to *Beams*. The default position (center of gravity) of the column axis (local x' axis) can be set in the aspect of analysis/design calculation or accurate display.



Figure: Modeling of a column connected to a beam

- Material

Any type of materials can be set for column *Analysis*, but design can be run for concrete, steel and timber columns only. The applied material name can be displayed in model view (*Display settings*).

- 4. Define the *direction* of the cross-section y' axis.
- 5. Choose a method for insertion point definition (*geometry*).
- 6. Place the column in the model view based on the chosen geometry method.

Optional steps:

- 7. Modify the bar properties with the *Properties* tool of the *Column* tool palette.
- 8. Set the *display settings* of columns at *Settings* > *All* > *Display* > *Beam, Column and Truss* similarly to *Beams*.

- 9. The columns are stored on "Columns" *Object layers*. At layer settings, the default color and pen width can be set for all columns. The color and pen width settings by selected column elements can be modified by *Edit* > *Properties* > *Change appearance*.
- 10. In Plate module, the **I** Info tool displays the support stiffness (motion and rotation) values of the column ends valid in the column *local co-ordinate system*.



Figure: Support stiffness of Columns in Plate module

11uss menuber

Truss member property	Description		
Modules where available			
Function	It models bars with limited compression and tension-only bars like cables, pull rods,		
	diagonal braces etc.		
Axis position	Arbitrary (horizontal, vertical and skew)		
Geometry	Straight		
Cross-section	Arbitrary profile		
Connection	- Always hinged (cannot be modified)		
	- It can be connected to other elements only at its ends		
	- It does not intersect other elements cross it visually		
Material	Steel, concrete, timber and general		
Load direction	Axial		
Load type	- Dead load (will be distributed to the end nodes)		
	- Uniform line temperature variation		
	- Shrinkage		
	- Line stress load		
	- Point mass (will be distributed to the end nodes)		
Analysis	- Displacement, normal force and stress		
	- It never participate in global stability (no buckling function)		
	- The 1 st and the 2 nd order analysis lead same results		
Available design	Steel design in 🛄 🕅		
	RC design in 🔛 🚳		
	Timber design in 🛄 🕅		



Definition steps

- 1. If needed (and available in the current FEM-Design module), set a proper position for the *working plane*. In the 3D design modules, respect that the gravity direction is always the Global Z axis direction.
- 2. Start V *Truss member* command from Structure tabmenu and choose *Define*.



3. Set the properties of the new truss element at \bigcirc *Default settings*:

Truss properties	×
A.1 General I Section Haterial	
Identifier (.position number)	
Enter identifier of bar (max 50 characters). The position number is generated automatically when "Tools", "Refresh numbering" functi called. Initially it is set to 0.	ion is
Behavior	
Limited compression	
Max. compression capacity [kN]	

- Identifier (General)

The program automatically generates it, but you can define custom value. Identifier (ID and Position) number can be displayed in model view (*Display settings*).

- *Behavior* (General)

Load-bearing function can be set by checking the *Limited compression* and by defining maximum capacity value for the compression.



Tension-only bar can be modeled by giving 0 to *Max. compression capacity*.

- Section

Choose a profile from the Section library or define a new cross-section by parameters. The applied section name can be displayed in model view (*Display settings*).

- Material

Any type of materials can be set for beam *Analysis*, but design can be run for concrete, steel and timber elements only. The applied material name can be displayed in model view (*Display settings*).

- 4. Define the *direction* of the cross-section y' axis.
- 5. Choose a *geometry* definition method for the truss element reference line (axis).
- 6. Define the element in the model view based on the chosen geometry method.

Optional steps:

- 7. Modify the bar properties with the *Properties* tool of the *Truss member* tool palette.
- 8. Set the *display settings* of truss elements at *Settings* > *All* > *Display* > *Beam, Column and Truss* similarly to *Beams*.
- 9. The truss elements are stored on "Truss members" *Object layers*. At layer settings, the default color and pen width can be set for all truss elements. The color and pen width settings by selected truss elements can be modified by *Edit* > *Properties* > *Change appearance*.

Fictitious Bar

Fictitious bar property	Description
Modules where available	
Function	Bar elements with given stiffness values
Axis position	Horizontal in 💒
	Arbitrary (horizontal, vertical and skew) in 🛄 🗖 🧴
Geometry	Straight and curved
End connection	Rigid and hinged, or custom defined by <i>Point-point connection</i>
Stiffness	Editable (but, material and cross-section definition is not available)
Load direction	Vertical in
	Arbitrary in 🛄 🚳 直
Load type	All point and line load
Available analysis results	Displacement and internal forces

Table: Fictitious bar properties



Figure: Fictitious bars connect plate elements

The program automatically generates "rigid" fictitious bars in the end points of "bars" modeled as a shell model (end edges of shell regions; see *Shell model*).

Definition steps

- 1. If needed (and available in the current FEM-Design module), set a proper position for the *working plane*.
- 2. Start J Fictitious bar command from Structure Tabmenu and choose Define.

Fictitious bar 🛛 🛛			
👤 ? [† †l 土 🖊 👁 🗢 🖵			

3. Set the properties of the new bar at *Default settings*:

F	ictitious bar		$\log_2 z' \rightarrow 2^{nd}$ axis
	A.1 General End conditions Identifier (.position number) Enter identifier of bar (max 50 characters). The position number is generated automatically when "Tools", "Refresh numbering" function is called. Initially it is set to 0.	local y'→1* axis	local x'
	Stiffness	,	

- *Stiffness* (General)

EA, GI_t, EI₁ and EI₂ stiffness values can be assigned to bars. No definition of material and crosssection is needed. The local y' axis represents the 1^{st} principal axis, and the local z' axis defines the 2^{nd} principal axis.

The value of rigid stiffness ("infinite" rigidity) can be defined and set as project default at *Settings* > *Calculation*> "*Rigid*" values > *Fictitious bar*.

- Connections

In 📶, the end connections are fully rigid.

In 3D modules, the end connections can be set under the *Connections* tab similarly to *Beams*. Only the released motion (arrow) and rotation (two-headed arrow) components are displayed in the model view (*Display settings*). Rigid connections can be customized to hinged or semi-rigid (spring) with additional *Point-point connection*.

4. Define the *direction* of the cross-section 1st principal axis (local y' axis; only in 3D modules).

The y' axis cannot be modified in the 2 module, so it is always parallel with the Global X-Y (working).

- 5. Choose a *geometry* definition method for the bar reference line (axis).
- 6. Define the bar in the model view based on the chosen geometry method.

Optional steps:

- 7. Modify the geometry with the *Edit* menu commands valid for line elements.
- 8. Modify the bar properties with the *Properties* tool of the *Fictitious bar* tool palette.
- 9. The fictitious bars are stored on "Modeling tools" *Object layers*. At layer settings, the default color and pen width can be set for all fictitious bars.

Intermediate section

Intermediate section property	Description
Modules where available	
Axis position	Horizontal in 💒
	Arbitrary (horizontal, vertical and skew) in 🗾 🕅
Cross-section	Arbitary
Eccentricity	Available
Material	Steel, concrete, timber and general
Available analysis results	Displacement, internal forces, stresses, stability and vibration shape in 🚰 🎞 🗖

Definition steps

- 1. If needed (and available in the current FEM-Design module), set a proper position for *working plane*. In all modules, respect that the gravity direction is always the Global Z axis direction.
- 2. Start *Intermediate section* from *Structure* tabmenu and choose *Perfine*.



3. Set the properties of the new Intermediate section at *Default setting*:



Section

Choose a beam profile from the Section library or define a new cross-section by parameters. The applied section name can be displayed in model view (Display settings).

Intermediate section can be used with the same section shape and fabrication technology. Insertion point

The axis default position is on the gravity center (symbolized with yellow point in the section), but it can be set with its coordinate in the local y'z' plane, or can be placed into special section points (e.g. plate corners) by clicking the yellow point into the requested position.



Default: "center of gravity"

Position modified by distances Position modified by the mouse 4. Define the direction of the cross-section y' axis (only in 3D modules).

The y' axis cannot be modified in the 🗾 module, so it is always parallel with the Global X-Y (working) plane.

Define the Intermediate section placement. 5.



Figure: Defining an Intermediate section

Optional steps:

6. Modify the *Intermediate section* with the *Properties*.

7. The *Intermediate section* is part of the bar element so its layer and setting can be found like in *Beam, Column or Truss.*

Apex

Apex property	Description
Modules where available	
Axis position	Arbitrary (horizontal, vertical and skew) in 荘 🕅
Cross-section	Rectangular
Eccentricity	Disable
Material	Steel, concrete, timber and general
Load direction	Arbitrary in 🛺 🕅
Load type	All point and line load
Available analysis results	Displacement, internal forces, stresses, stability and vibration shape in 🎽 🛺 🗖
Available design	Timber design in 🛄 🕅

Definition steps

_

- If needed (and available in the current FEM-Design module), set a proper position for *working plane*. In all modules, respect that the gravity direction is always the Global Z axis direction.
 - Start *Intermediate section* from *Structure* tabmenu and choose *Define*.



- Set the properties of the new Intermediate section at



Rounding radius (r)

The rounding radius can be invidually set. The default value is 1.00 m and rounding the inner edge. For rounding the outer edge check the *Round outer edge* option on. The following figure shows the difference between rounding the inner or both edges.

Default setting:



Figure: Rounding inner and both edge

- Select the neighboring bar element.

A Defining the correct way an apex, the following criteria has to meet:

- The neighboring bars cross-section has to be **Rectangular**
- Their Y axes are parallel and point into the same direction
- Their material has to be the same
- Apex zones with their connected bars are only one physical element but it has more analytical element.



Figure: Describing analytical and physical elements

Planar Objects

This chapter summarizes all features and properties of planar objects: *Foundation slab, Plate, Wall, Shell model elements, Timber panel* and *Profiled panel*.

Foundation slab

Foundation slab property	Description
Modules where available	
Reference plane position	Horizontal
Geometry	Any shape
Thickness	Constant or variable
Hole	Available
Eccentricity	Available for display and calculation
Edge connection	Rigid elastic (spring) linear or non-linear
Material	Concrete
----------------------------	---
Orthotropic feature	Available
Load direction	Vertical
	Point load (force and moment), Line load (force and moment), Surface load,
	Surface temperature variation load, Surface stress load, Surface support motion
Load type	
Available analysis results	Displacement, internal forces, stresses, stability and vibration shape
Available design	Foundation design !!!

Table: Foundation slab properties

Definition steps

- 1. If needed, set a proper position for the *working plane*.
- 2. Start Foundation slab command from Structure tabmenu and choose 🚽 Define.

Foundation	slab 🖉	
	t [m] 1.0000 Editing plane [m] 0.000	
	Alignment	
	Edges ["Rigid" 🔻	

3. Set the properties of the new plate at \square *Default settings*:

Foundation slab	X
A.1 General	
Identifier (.position number)	
Geometry	Eccentricity
t [m] 1.0000	‡e
Orthotropy	
E2	Alignment → •
E2/E1 1.0000	Consider eccentricity in calculation
Alpha [°]	Consider eccentricity caused by cracking in cracked section analysis
Bedding	
Bedding modulus [kN/m2/m]	10000
	OK Cancel

- Identifier (General)

The program automatically generates it, but you can define custom value. Identifier (ID) number can be displayed in model view (*Display settings*).

- Alignment and Eccentricity (General)

With alignment the position of the reference (definition) plane can be set to top, center or bottom.

Eccentricity (e) adds "virtual" distance between the reference plane and the calculation (middle!) plane if "*Consider eccentricity in calculation*" option is unchecked, otherwise the eccentricity has effects in the calculation. This option is useful for displaying regions with different thickness values.

Alignment and eccentricity have no effect in calculations if *"Consider eccentricity in calculation* is checked off. They are only to display plate *"solid"* with its defined position.

Eccentricity value can be displayed in model view (Display settings).

The "*Consider eccentricity caused by cracking in cracked section analysis*" option is for Reinforced concrete slabs. With this option modeling the eccentricity changes after cracking is available. The following figure shows the eccentricity changes.



Figure: Explaining cracked and uncracked analysis

- Orthotropic features (General)

The ratio between the E1 and E2 elasticity modules in the main directions can be set here. So, if the value is 1, the new plate will be isotropic, otherwise it is orthotropic. The orthotropy direction can be set with the angle between E1 direction and the plate's local x' axis (*Alpha*).

- Material

Only concrete material can be set for Analysis and Design.

4. Choose a *geometry* definition method for the plate reference plane (set by *Alignment, Eccentricity* and *Editing plane*).

Defining Foundation slab with *Editing plane*. By this the user can choose which plane he wants to

define for Foundation slab. The ______ in the bottom right side of the tool window shows the

"definition plane" with blue line. This can be set manually Editing plane [m] ... 0.500 or with some specific plane (top, middle or bottom) by clicking in the picture.

This has no effect on the alignment.



After selecting definition mode, default settings and editing plane geometry can be defined.



4. Define the geometry



5. Define the plate region in the model view based on the chosen geometry method. In 3D Structure module only horizontal position can be set. All region definition points have to be in the same plane. UCS may help to find the requested position of the new plate region.

Optional steps:

6. Modify the default edge connections with *L*. Just select the plane and then its edge(s) to set connection conditions, and finally set the requested motion and rotation settings.

Motion-Rotation / Compression-Tension

The x', y' and z' are valid in the connection's local system displayed in the preview and as a symbol in the middle of the edges in the model view. Choose "Rigid" or "Free" from the drop-down lists, or type stiffness values (spring connection).

Active Ink button assigns the stiffness value typed in the *Compression* field to the *Tension* field. Inactive icon lets to define stiffness values separately.

Predefined types

Click to set all motion and rotation components to "rigid". The result will be a "totally rigid" connection.

Click do set all motion components to "rigid" and all rotation components to free (allow rotation around all directions). The result will be a hinged connection.

Click to set all components to free. This tool gives the possibility to virtually connect independently moving elements.

Behavior

Click to assign the stiffness value defined in the *Compression* field to the *Tension* field by components.

Click + not to allow tension for all connection components.

Click *here to allow compression for all connection components.*

Setup "Rigid"

With this option the value of *"infinite" rigidity* can be defined by motion and/or rotation for selected connections.

For all edge connections, the value of "infinite" rigidity can be defined and set as project default at *Settings* > *Calculation*> "*Rigid*" values > *Connections*.

7. Add *hole* with tool.

- 8. Modify the reference plane geometry with the *Edit* menu commands valid for region elements.
- 9. Modify the *Foundation slab* properties with the *Properties* tool of the *Foundation slab* tool palette.
- 10. Set the display settings of plates at *Settings* > *Display* > *Soil and Foundation*.
- 11. The Foundation slabs are stored on "Foundation" and if they modeled with *Surface support group* their supports will be stored at the "Support" *Object layers*. The color and pen width settings by selected plate elements can be modified by *Edit* > *Properties* > *Change appearance*.

Plate

Plate property	Description	
Modules where available		
Reference plane position	Horizontal in	
	Arbitrary (horizontal, vertical and skew) in 🛄 值	
Geometry	Any shape	
Thickness	Constant or variable	
Hole	Available	
Eccentricity	Only for display	
Edge connection	Rigid, hinged, elastic (spring) linear or non-linear	
Material	Steel, concrete, timber and general	
Orthotropic feature	Available	
Load direction	Vertical in 🞽	
	Arbitrary in 🛄 🧴	
Load type	All point, line and surface load	
Available analysis results	Displacement, internal forces, stresses, stability and vibration shape in 🚰 🎞	
Available design	RC design in 🛃 🛄	
	<i>Timber design</i> in	

Table: Plate properties

Definition steps

- 1. If needed (and available in the current FEM-Design module), set a proper position for the *working plane*.
- 2. Start W *Plate* command from Structure tabmenu and choose V *Define*.

3D plate		2
. ? 🗉	■ 🔲 🗖 🖬 🗖 🔿 🖿 🖥 🗖	
	t1 [m] 0.20000 1 Alignment	
	t2 [m] 0.20000 🕽 Edges Rigid 💌	
	t3 [m] 0.20000	

3. Set the properties of the new plate at Default settings:

Plate properties	×
A.1 General Material	
Identifier (.position number)	2
Geometry	Eccentricity
t1 [m] 0.20000 1 t1 [m] t2 [m] 0.20000 1 t3 [m] t3 [m] 0.20000 1	Alignment
Orthotropy	e [m] 0.00000
	Consider eccentricity in calculation Consider eccentricity caused by cracking in cracked section analysis
E2/E1 1.0000 Alpha [deg] 0.00000	
	OK Cancel

- *Identifier* (General)

The program automatically generates it, but you can define custom value. Identifier (ID and Position) number can be displayed in model view (*Display settings*).

- Alignment and Eccentricity (General)

With alignment the position of the reference (definition) plane can be set to top, center or bottom.

Eccentricity (e) adds "virtual" distance between the reference plane and the calculation (middle!) plane if "*Consider eccentricity in calculation*" option is unchecked, otherwise the eccentricity has effects in the calculation. This option is useful for displaying regions with different thickness values.

Alignment and eccentricity have no effect in calculations if *"Consider eccentricity in calculation* is checked off. They are only to display plate *"solid"* with its defined position.

Eccentricity value can be displayed in model view (*Display settings*).

The "*Consider eccentricity caused by cracking in cracked section analysis*" option is for Reinforced concrete slabs. With this option modeling the eccentricity changes after cracking is available. The following figure shows the eccentricity changes.



Figure: Explaining cracked and uncracked analysis

- Thickness (General)

By default, only plate with constant thickness (t1=t2=t3) can be set at *Default settings*. Use the *Variable thickness* tool to define different t values that declare the top/bottom surface as a linearly variable plane.

- Orthotropic features (General)

The ratio between the E1 and E2 elasticity modules in the main directions can be set here. So, if the value is 1, the new plate will be isotropic, otherwise it is orthotropic. The orthotropy direction can be set with the angle between E1 direction and the plate's local x' axis (*Alpha*).



In *Plate* module the local x' axis is always parallel with the global X axis.

- Material

Any type of materials can be set for plate *Analysis*, but design can be run for concrete only. The applied material name can be displayed in model view (*Display settings*).

4. Set the default connections for all plate edges to rigid or hinged with the tool palette's *Edges* option.



Figure: Default edge connections



Use the *Edge connection* tool to modify the default connections by the selected edges of a predefined plate. Hinged, rigid, semi-rigid (spring), free connections can be set with linear and non-linear behavior.

- 5. Choose a *geometry* definition method for the plate reference plane (set by *Alignment* and *Eccentricity*).
- 6. Define the plate region in the model view based on the chosen geometry method. In kinetic module the slab plane can be horizontal only, while in 3D modules any skew position can be set. All region definition points have to be in the same plane. UCS may help to find the requested position of the new plate region.



Figure: Different working planes for plate definition

Optional steps:

7. Modify default thickness (Variable thickness)

Constant slab thickness can be fast modified with the *Properties* tool. Just select the plate(s) and set a new value under General Tab.

Constant thickness can be changed to variable, or reverse, with the **Variable** thickness tool.

Steps of defining plate with variable thickness:

- i. Inactivate the \square link buttons to allow giving different values in the *t* fields.
- ii. Type the *t*1, *t*2 and *t*3 according to the current length unit.

3D plate	3D plate	X
.! ? • 💶 🖸 🖬 🔄	1? 🗆 🗖 🗖 🖬	
t1 [m] 0.20000 D Alignment → ✓	t1 [m] 0.40700] 1 Alignment	2
t2 [m] 0.20000 1 Edges Hinged 🗸	t2 [m] 1.2030 2 Edges Hinged	-
t3 [m] 0.20000	13 m20 3	

- iii. Select the plate you would like to modify.
- iv. Define three points in the plane of the selected plate's reference plane, where *t*1, *t*2 and *t*3 will be measured. The measurement of the thickness values depends on the position of the reference plane (*Alignment* and *Eccentricity*).



Figure: Align the thickness of the middle slab to its neighbors



Figure: Align circular slab thickness to a variable height wall

The measurement of the thickness values depends on the position of the reference plane (*Alignment* and *Eccentricity*).



/!\



Figure: Edit thickness by using other objects' height

8. Modify the default edge connections with **L**. Just select the plane and then its edge(s) to set connection conditions, and finally set the requested motion and rotation settings.

2 Select 1 Select plate	Edge connection 3 Set connection	on conditions	
3D plate I [] ? I [] [] I [] </th <th>A.1 General Data Motions [kN/m] Compression Tension K.x' 1.000e+07 $+$ 1.000e+07 \times K.y' 1.000e+07 $+$ 1.000e+07 \times K.z' 1.000e+07 $+$ 1.000e+07 \times Predefined types $\frac{1}{mm}$ $\frac{1}{mm}$ $\frac{1}{mm}$ Local system \overbrace Keeps direction along line $iong line$</th> <th>Rotations [kNm/deg] Compression Cx' 0.000e+00 Cy' 0.000e+00 C2' 0.000e+00 x Behavior y y z' x'</th> <th></th>	A.1 General Data Motions [kN/m] Compression Tension K.x' 1.000e+07 $+$ 1.000e+07 \times K.y' 1.000e+07 $+$ 1.000e+07 \times K.z' 1.000e+07 $+$ 1.000e+07 \times Predefined types $\frac{1}{mm}$ $\frac{1}{mm}$ $\frac{1}{mm}$ Local system \overbrace Keeps direction along line $iong line$	Rotations [kNm/deg] Compression Cx' 0.000e+00 Cy' 0.000e+00 C2' 0.000e+00 x Behavior y y z' x'	

Figure: Modification of two slab edges

Motion-Rotation / Compression-Tension

The x', y' and z' are valid in the connection's local system displayed in the preview and as a symbol in the middle of the edges in the model view. Choose "Rigid" or "Free" from the drop-down lists, or type stiffness values (spring connection).

Active Hink button assigns the stiffness value typed in the *Compression* field to the *Tension* field. Inactive icon lets to define stiffness values separately.

Predefined types

Click to set all motion and rotation components to "rigid". The result will be a "totally rigid" connection.

Click to set all motion components to "rigid" and all rotation components to free (allow rotation around all directions). The result will be a hinged connection.

Click <u>to set all components to free.</u> This tool gives the possibility to virtually connect independently moving elements.



Figure: Edge connection types and their displacement effect

Behavior

Click to assign the stiffness value defined in the *Compression* field to the *Tension* field by components.

Click *t* not to allow tension for all connection components.

Click *here to allow compression for all connection components.*

Setup "Rigid"

With this option the value of *"infinite" rigidity* can be defined by motion and/or rotation for selected connections.

For all edge connections, the value of "infinite" rigidity can be defined and set as project default at *Settings* > *Calculation*> "*Rigid*" values > *Connections*.

EQ3

As a real example, *Edge connection* gives the possibility to model prefab concrete elements too. Define the objects with the *Plate/3D plate* commands, set the thickness value of the regions to substitute the real cross-section with the holes, and finally add "hinged" edge

connection ($C_x = 0$) to one of the region at the common edge with the other one (see the next figure).



Figure: Modeling slab elements connected to each other (hollow-core prefab slabs)

9. Add *hole* with \Box tool.

In 3D modules, select a plate region and define hole in it or cut it by using a geometry mode. Selecting a plate displays the *UCS* in the reference plane of the plate.



Figure: Hole and cuttings

In *Plate* module, just define hole or cutting by using a geometry mode in the required plate reference plane. Selection of the host plate is needed, if the hole region intersects more than one regions.

- 10. Modify the reference plane geometry with the *Edit* menu commands valid for region elements.
- 11. Modify the plate properties with the *Properties* tool of the *Plate* tool palette.
- 12. Information (the volume of the plate solid and the position of its centre of gravity) about a selected plate can be inquired with the **1** *Info* tool. A drawing point can be placed in the plate's centre of gravity, if needed.
- 13. Set the display settings of plates at *Settings* > *All* > *Display* > *Plate and Wall*.



14. The plates are stored on "Plates" *Object layers*. At layer settings, the default color and pen width can be set for all plate regions. The color and pen width settings by selected plate elements can be modified by *Edit* > *Properties* > *Change appearance*.

Wall

Wall property	Description		
Modules where available			
Function	Line support in		
	Shear or load-bearing wall region		
	Unit section of long-extensional object (tunnels, dams, retaining walls et.)		
	Structural wall in 📑 🧴		
Reference plane position	Vertical in 🖆 🛺 🗴		
	Global X-Y plane 🗰 進		
Geometry	Any shape		
Thickness	Constant or variable		
Hole	Available		
Eccentricity	Only for display		
Edge connection	Rigid, hinged, elastic (spring) linear or non-linear		
Material	Steel, concrete, timber and general		
Orthotropic feature	Available in 🗰 🏂 📑 🗴		
Load direction	Arbitrary in 🛄 🧴		
	Planar, global X-Y plane 🚻 達		
Load type	All point, line load and surface load in 🔛 🏂 🏣 👔		
Available analysis results	Reaction forces in		
	Displacement, internal forces and stresses		
	Displacement, internal forces, stresses, stability and vibration shape in 🔛 崖 🎞		
Available design	<i>RC design</i> in the second seco		
	<i>Timber design</i> in		

Table: Wall properties



Figure: Wall function by FEM-Design modules

Definition steps

The wall definition depends on what module is used for modeling:

- Wall is a line support, so its base line has to be defined with a given height and constant/variable thickness as a straight or curved line.
- The only modeling object is the Wall, which is a region with constant or variable thickness, so its reference plane has to be defined as a rectangle, circular, polygonal or arbitrary shaped region. Holes can be added to it.
- The only modeling object is the Wall, which is a region with unit (=1) thickness, so its reference plane has to be defined as a rectangle, circular, polygonal or arbitrary shaped region. Holes can be added to it.
- Wall is a region in vertical position, so the base line has to be defined with given height and constant/variable thickness as a straight or curved line. Holes can be added to it.
- 1. If needed (and available in the current FEM-Design module), set a proper position for the *working plane*.
- 2. Start Wall command from Structure tabmenu and choose 🛃 Define.

wall	Wall	wall		
1 mi 0.20000 ↓ → ▼ 12 (mi 0.20000 ↓ ↓ ▼ 13 (mi 0.20000 ↓ ↓ ▼			h [m] 3.0000 t1 [m] 0 Placement Above t2 [m] 0	1.20000] Alignment 1.20000 Edges Rigid Alpha [deg] 10.000
	₹2 (m) (0.15000 ♥ ?	■ 1 □ ○ 1: □ .	<u>.</u>	

3. Set the properties of the new wall at *Default settings*:

Wall properties		
A.1 General Material	Wall properties	
Identifier (.position no.)	A.1 General I Material	
Image: Statical system Statical system Alignment . Image: Statical system O Hinged O Under slab Alignment . Image: Statical system Fixed Alove slab Image: Statical system O Fixed Alignment . Image: Statical system O Fixed Alignment . Image: Statical system O Fixed Alignment . Image: Statical system O Hinged O Under slab Alignment .	Identifier (.position number) Identifier (.position number) Geometry Eccentricity t112 t1 [m] t2 [m] 0.20000 Orthotropy 0.20000 If t112 t1 [m] 0.20000 Image: the image	
	OK Cancel	

- *Identifier* (General)

The program automatically generates it, but you can define custom value. Identifier (ID and Position) number can be displayed in model view (*Display settings*).

- Height (h) (General)

In the height can be set in the dialog, but in 3D modules only in the Wall command's tool palette.



- In 3D modules, the sign (positive or negative) of the Height value defines the location of the Wall (*Straight line*).
- In *the position of the wall support can be set according to slab position: "Above slab" or "Under slab". It also defines the location of the base line.*
- *Alignment and Eccentricity* (General)

With alignment the position of the reference base line or the reference region can be set to left/front, center or right/back.

Eccentricity (e) is handled the same was as in 3D Plate.

Eccentricity value can be displayed in model view (*Display settings*).



While modeling, think about the reference plane/reference line connections with each other. Connecting edged "perfectly" results correct and "nice" finite element mesh. If they intersect each other with small distances, the outcome of mesh generation may be very small finite elements or unexpected mesh regions. The next figure shows an example for possible (recommended or not) wall connection to slab edge.



Figure: Recommended modeling of plate-wall edge connection (no calculation differences in case 1. and 2.)

- Thickness (General)

In **E**, the thickness values (t1 in the start point and t2 in the end point) can be set directly in the settings dialog. The measurement of the thickness depends on the *Alignment and Eccentricity* settings.



Figure: Variable thickness and eccentricity

In $\underbrace{1}$, the thickness value is fixed. It is always 1m, so only region(s) with unit thickness (section) can be defined.

In \underbrace{III} , the thickness value can be constant or variable like plate regions. Because walls are regions in the *Wall* module like plates in *Plate* module, the definition way of constant and variable thickness is the same. In the dialog, only constant thickness (t1=t2=t3) can be set, but with Variable thickness tool (same with Plate's *Variable thickness* tool) t values can be modified separately to define wall regions with variable thickness.

- Orthotropic features (General)

This option is available in \square \square \square modules. The ratio between the E1 and E2 elasticity modules in the main directions can be set here. So, if the value is 1, the new wall will be isotropic, otherwise it is orthotropic. The orthotropy direction can be set with the angle between E1 direction and the wall's local x' axis (*Alpha*).

In $\underbrace{1}{12}$ modules the local x' axis is always parallel with the global X axis.

- Statical System (Connections) (General)

In *the support conditions (General)* of the top and bottom wall ends can be set to fully rigid or fully hinged. Because, *Walls* are line supports further line/point support is not needed to define at *Wall* end.

- Allow uplift (General)

Available in ² only. "Uplift" behavior can be modeled by activating this option. It works in case of tensional reaction force.

In 3D modules, "uplift" can be modeled with non-linear *Line support* settings.

- Material

Any type of materials can be set for wall *Analysis*, but design can be run for concrete and timber walls only depending on the current FEM-Design module. The applied material name can be displayed in model view (*Display settings*).

3. In 🔜 🔟 modules, the default connections for all wall edges can be set to rigid or hinged with the tool palette's *Edges* option.





Figure: Default edge connections

Use the *Edge connection* tool to modify the default connections by the selected edges of a predefined wall. Hinged, rigid, semi-rigid (spring), free connections can be set with linear and non-linear behavior.



If a wall connect to slab at a common edge and you would like to define hinged connection between them, only one edge connection (for plate or wall) has to be defined hinged and the

other has to be kept as rigid. So, applying hinged edge connection for both plate and wall at their linked edge is not allowed to avoid instability in finite element modeling. The figure shows the possibilities to model the same hinged connection in two ways.



Figure: Modeling the same hinged wall-plate edge connection in two ways

The previous rule is also true for wall support in module: if, you would like to model hinged wall support at a plate edge, define hinged statical connection for wall (top or bottom depending on the wall position) and set the plate edge connection to rigid, or, define rigid wall support and hinged edge for plate.

4. Choose a *geometry* definition method for the wall's reference line/plane (set by *Alignment* and *Eccentricity*) and create the wall with co-ordinates or by picking points in the drawing area.

In 🛗 🔳 modules, the walls are regions in the Global XY working plane.

In these wall modules the dead-load is orientated in Global Y direction, although the dead-load is orientation in Global Z direction in all other modules.



Figure: Wall region with holes, loads and supports in Wall and Plane Strain modules

In module, wall is a line support, so it has to be defined with its base/reference line. Although walls are planar objects in modules, they can be defined with their reference lines, because of the fixed vertical position. The lines can be straight or "curved". The "curved" means real arc and circle in module, while it means the combination of straight lines (wall sections) in the 3D modules. In the second case the "curved walls" are built up by planar region members (curved finite elements are not available in FEM-Design), and the resolution of the curve approximation can be set by the peripheral angle of the sections (*Alpha*).



Figure: "Curved" wall definition in 3D modules

In the 3D modules, if you would like to connect a plate to "curved" walls at the edges, define polygonal slab instead of curved-edge slab. In case of curved plate, small regions are cut by the connected wall, and that results finite elements with very small angle.



Figure: "Curved" wall definition in 3D modules

Optional steps:

5. Modify default thickness (Variable thickness)

Constant wall thickness can be fast modified with the *Properties* tool. Just select the wall(s) and set a new value under General Tab. In in modules, you can define variable thickness with t1 and t2 values too.

In *modules, constant thickness can be changed to variable (linear distribution), or reverse, with the* Variable thickness tool.

Defining wall with *variable thickness* is similar as done in case of a Plate.

- 6. Modify the default edge connections with . Just select the wall and then its edge(s) to set connection conditions (rigid, hinged, free or semi-rigid (spring)), and finally set the requested motion and rotation settings. *See the definition steps and possibilities at Plate.*
- 7. Add *hole* with tool (*see hole definition in plates*).

Except of *module* where wall is support, holes and cuttings can be defined in walls.

In 3D modules, select a wall region and define hole in it or cut it by using a geometry mode. Selecting a wall displays the **UCS** in the reference plane of the wall.

In in modules, just define hole or cutting by using a geometry mode in the required wall reference plane. Selection of the host wall is needed, if the hole region intersects more than one regions.

- 8. Modify the reference line/plane geometry with the *Edit* menu commands valid for lines/region elements.
- 9. Modify the wall properties with the *Properties* tool of the *Wall* tool palette.
- 10. In *Plate* module, the *Info* tool displays the support stiffness (motion and rotation) values of the wall ends (start and end point) valid in the wall *local co-ordinate system*.

In the other design modules, information (the volume of the wall solid and the position of its centre of gravity) about a selected wall can be inquired with the **I** *Info* tool. A drawing point can be placed in the wall's centre of gravity, if needed.

- 11. Set the display settings of walls at *Settings* > *All* > *Display* > *Plate and Wall* (*see Plate display settings*).
- 12. The walls are stored on "Walls" *Object layers*. At layer settings, the default color and pen width can be set for all plate regions. The color and pen width settings by selected plate elements can be modified by *Edit* > *Properties* > *Change appearance*.

Shell Model

In 3D *Structure* module, straight and arc bars (*Beams* and *Columns*) can be modeled more accurately for stability, utilization check (against flexural, torsional, lateral torsional and local buckling) and design according to their EC by modeling them with planar shell members.

Definition steps

1. Press *Analytical model / Shell model* option of *Beam* or *Column* toolwindow and choose the bar. By default, selecting a bar element generates a complex 3D model built by a group of steel plates derived from the geometry of the selected bar's profile. All converted plates inherit the original bar element's steel material.



Figure: Straight bars with constant cross-section

After converting a bar to shell model, the program automatically defines "rigid" *Fictitious bar* objects at the ends of the shell model to guarantee continuous connection to other objects line 1D members, point supports and point loads. The stiffness of these fictive elements can be modified with the *Properties* tool of *Fictitious bar* command.



Depending on, *Beam*, *Column* or *Truss member* was converted to shell model, the model can be hidden together with the *Beam*, *Column* or *Truss member* object layer.

Timber Panel

Quick tool to model timber slabs and wall panels is optionally available for later analysis in 3D *Structure* and 1 *PreDesign* modules and timber design in 1.

Definition steps

- 1. If needed (and available in the current FEM-Design module), set a proper position for the *working plane*.
- 2. Click to define a new timber panel.

- 3. Choose the panel function: 🗇 Use as plate or 💷 Use as wall.
- 4. Set the panel settings under *Default settings*.



Figure: Timber command and settings

- *Identifier* (General)

The program automatically generates it, but you can define custom value. Identifier (ID and Position) number can be displayed in model view (*Display settings*).

- *Alignment and Eccentricity* (General)

With *Alignment* the position of the reference (definition) plane can be set to *Top/Left*, *Center* or *Bottom/Right*.

Eccentricity adds "virtual" distance between the reference plane and the calculation (middle!) plane. This option is useful for displaying regions with different thickness values.

Eccentricity (e) is handled the same was as in *3D Plate*. Eccentricity value can be displayed in model view (*Display settings*).

- *Type* (Panel)

Predefined panel types are available, but it can be expanded manually (*Edit library*) with new laminated, fiber boards, plywood etc. Design factors, stiffness and strength values can be edited for predefined and new panel types. Panel types can be imported or exported between users like the FEM-Design *materials*.

- 5. Set the width (w) of the selected panel type on the tool palette.
- 6. Set the default edge connection to hinged or rigid at *Edges* of the tool palette.
- 7. Define the timber plate/wall based on the *geometry* of the reference plane/base line. For plate panels, the stiff direction has to be defined first. (For walls, the stiff direction is always perpendicular to the wall base line.) The distribution of panels is described with an anchor point.



Figure: Timber plate definition

Optional steps:

- 2. The panel type of selected timber panels can be modified subsequently with the *Properties* tool.
- 3. Modify the default edge connections of the unique panel or panel group with **L**. Just select the plate/wall panel and then its edge(s) to set connection conditions at Border (DOF of the panel group) or Panel tab (rigid, hinged, free or semi-rigid (spring)), and finally set the requested motion and rotation settings. *See the definition steps and possibilities at Plate*.



Figure: Panel edge connections

- 4. Openings and cuttings can be added to timber panel elements with *Edit* > *Region operations* > *Split regions*.
- 5. Set the display settings of timber panels at *Settings > All > Display > Plate and Wall* (*see Plate display settings*).
- 6. The panels are stored on "Plates" *Object layers*. At layer settings, the default color and pen width can be set for all panel regions. The color and pen width settings by selected panel elements can be modified by *Edit* > *Properties* > *Change appearance*.

Profiled panel

Quick tool to model prefabricated slabs and wall panels is optionally available for later analysis in 3D *Structure* module and the designing in 3D.

Definition steps

- 8. If needed (and available in the current FEM-Design module), set a proper position for the *working plane*.
- 9. Click for to define a new timber panel.
- 10. Choose the panel function: 🕅 Use as plate or 🔟 Use as wall.
- 11. Set the panel settings under *Default settings*.
 - *Identifier* (General)

The program automatically generates it, but you can define custom value. Identifier (ID and Position) number can be displayed in model view (*Display settings*).

- Alignment and Eccentricity (General) With Alignment the position of the reference (definition) plane can be set to Top/Left, Center or Bottom/Right.
 Eccentricity (e) is handled the same was as in 3D Plate .
 Eccentricity value can be displayed in model view (Display settings).
- Section (Panel)
 Predefined panel types are available in Section library in *Hollow Core*. Cross-section types can be imported or exported between users like the FEM-Design *materials*.
- 12. Set the panel group's edge connection in *Border* tab.
- 13. Set the unique panel's edge connection in *Panel* tab. The following figure shows the difference between the panel group and the unique panel border.



14. Define the profiled plate/wall based on the *geometry* of the reference plane/base line. For plate panels, the stiff direction has to be defined first. (For walls, the stiff direction is always perpendicular to the wall base line.) The distribution of panels is described with an anchor point. Defining is similar to *Timber panel definition*.

Optional steps:

- 7. The panel type of selected profiled panels can be modified subsequently with the *Properties* tool.
- 8. Modify the default edge connections of the unique panel or panel group with . Just select the plate/wall panel and then its edge(s) to set connection conditions at Border (DOF of the panel group) or Panel tab (rigid, hinged, free or semi-rigid (spring)), and finally set the requested motion and rotation settings. *See the definition steps and possibilities at Plate*.

- 9. Openings and cuttings can be added to timber panel elements with *Edit* > *Region operations* > *Split regions*.
- 10. Set the display settings of timber panels at *Settings* > *All* > *Display* > *Plate and Wall* (*see Plate display settings*).
- 11. The panels are stored on "Plates" *Object layers*. At layer settings, the default color and pen width can be set for all panel regions. The color and pen width settings by selected panel elements can be modified by *Edit* > *Properties* > *Change appearance*.
- 12. Modifying the anchor point it's possible with *Base line* command:



13. With *Swap panel* two panel placement can be change which one panel is a truncated panel.



15. *Cut panel* will change the placement of the truncated panel.



Supports and Connections

This chapter summarizes the possibilities and properties of the available supports and connections.

The connection with the ground (model space) can be modeled with *Supports*, while the link among structural elements can be defined with *Connections*. All support/connection components can be set to:

- "infinite" rigid: blocked motion/rotation,
- "free": released motion/rotation, or
- semi-rigid: given stiffness value (spring) against motion/rotation.

Non-linear behaviors, independent compression and tension settings are also available for all motion and rotation components.



Edge connection is a special tool of structural object definitions. Although it was described at planar objects (like *Plate*), the following connection settings and functions are also valid for *Edge connections*.

Support motion loads have to be assigned to supports, so supports are requested input parameters for motion-type loads.

Always try to create stable structural model by defining correct support and connection conditions. From unstable (kinematically indeterminate) structure equilibrium error can be resulted. In that case the program sends a warning message at the end of the calculations, and you can determine the location of the problems by checking the equilibrium (*Analysis* > *Equilibrium*), the displacement/buckling and/vibration shape results of the structure. It may also happen that the model is so incorrect that calculations stops with error messages and without results.

"Infinite" Rigidity

The value of "infinite" rigidity can be defined and set as project default by support/connection types and components at *Settings* > *All* > *FEM*. Go through the *Motion* and *Rotation* components by support and connections types and set the requested stiffness value that model blocked absolute/relative displacement.



The "rigid" values can be set to default for further projects, if select "*Rigid*" values in the Settings tree and click Save as default.



Figure: "Rigid" values

The "rigid" value of

The "rigid" value can be set by support/connection/fictitious bar element inside its settings dialog (*Setup* "*Rigid*") independently of the project default.

Point support group	X
Motions [kN/m] Compression Tension K _x ' 1.000e+10 + 1.000e+10 × Ky' 1.000e+10 + 1.000e+10 × Kz' 1.000e+10 + 1.000e+10 ×	Rotations [kNm/rad] Compression Criminal Complexity Criminal Complexity Criminal Complexity Criminal Complexity Compression Criminal Complexity Criminal Complexity Criminal Complexity Complexity Criminal Complexity Criminal Complexity Criminal Complexity Criminal Complexity Complexity Complexity Criminal Complexity Complexity Criminal Complexity Criminal Complexity Criminal Complexity Criminal Complexity Criminal Complexity Criminal Complexity
Predefined types	Behavior
Point support Motions [kN/m] Rotations [kNm/rad] 0K Cancel	

Figure: "Rigid" value set by support objects

Properties (Non-Linear Behaviors)

Compression and tension behavior of supports and connections can be set separately and by components.

Point motion support	Line connection	- Internet - Laure	×
A.1 General Data Spring [4V/m] Compression Tension K 1.000e+10 V Tension Rigid" Free Setup "Rigid" > OK Cancel	A.1 General ▲★ Data Type Custom Motions [dV/m/m] Compression Compression Tere Ky' 1.000e+07 • 0 1. Ky' 1.000e+07 • 0 1. Predefined types Imm Imm Imm Local system	▼ Rotations [04m/m/*] nsion Compression 000e+07 Cx ¹ 000e+07 Cy ¹ 000e+07 Cx ² 000e+07 Cx ² 1.745e+05 Intraface Behavior Detach Interface position (proportion)	+05 V +05 V +05 V
Point support group	Direct Direct along	on changes	2
A.1 General	n/°] sion Tension +08 • • 1.745e +08 • +08 • • 1.745e +08 • +08 • • 1.745e +08 • 	OK Edge connection A.1 General Type Motions [kV/m/m] Compression Kx' 1.000e407 V 1.000e407 Ky' 1.000e407 Ky' 1.000e407 V Predefined types Imme Local system	Cancel X Rotations [klm/m/*] Compression Tension Cx1 1.745e+05 • 1.745e+05 • Cy1 1.745e+05 • 1.745e+05 • Cy1 1.745e+05 • 1.745e+05 • Cy1 1.745e+05 • 1.745e+05 • Detach No •
		Setup "Rigid" >	Z Y Z CK Cancel

Figure: The parts of typical support/connection settings dialogs

Stiffness fields by compression and tension

"Rigid" or *Free* can be chosen from the pop-ups, or stiffness value (spring support/connection) can be typed in the fields.

<u>E</u>OX

Active icon assigns the stiffness value typed in the *Compression* field to the *Tension* field. Inactive icon lets to define the stiffness values separately.

- Predefined types

Click to set all motion and rotation components to "rigid". The result will be a totally rigid support/connection.

Click to set all rotation components to "free" (allow rotation around all directions). The result will be a hinged support/connection.

Click to set all components to free. This tool gives the possibility to virtually connect independently moving elements.

- Behavior

(Not available for *single Supports*)

Click \swarrow to assign the stiffness value defined in the *Compression* field to the *Tension* field by components. Click \nrightarrow not to allow tension for all support/connection components. "Uplift" behavior can be modeled with + or by typing 0 in all *Tension* fields. "Free" (0) rotation components are recommended for the "uplift" case.

Click H not to allow compression for all support/connection components.

E Cr

"Uplift" can be modeled both in 2D and 3D design modules by defining compression-only *support* / *connection* (tension = 0 (free)) and by checking the *Consider non-linear behavior of supports, trusses and connections* box at *Calculate> Analysis*.



Figure: Allow uplift in Plate and 3D Structure modules

Supports

Support types

The following table summarizes the available support types and their main features in the different FEM-Design Modules.

Column and **Wall** are developed to model single point and line support in Plate module. These types of support are not mentioned in the *Support* chapter.

Point support property	Description
Modules where available	
Motion component direction	Parallel with Global Z axis in
	Arbitrary direction in Global XY plane in
	Arbitrary direction in 🛄 🕅

Rotation component direction	Arbitrary direction in Clobal XX plane in
Rotation component unection	
Carrier	Arbitrary direction in 🛀 🍱
Available analysis results	- (insertion point) Reaction forces and moments
Lood can be assigned	Point summert motion load
Ve de les sub ans sere ils bla	
Modules where available	
Motion component direction	Parallel with Global Z axis in
	Arbitrary direction in Global XY plane in 🛄 🛎
	Arbitrary direction in 📲 🗐 🔟
Rotation component direction	Arbitrary direction in Global XY plane in
	Not available in 🛗 進
	Arbitrary direction in 🛺 🕅 🧴
Geometry	- (insertion point)
Available analysis results	Reaction forces and moments
Load can be assigned	Point support motion load
Line support property	
Modules where available	
Motion component direction	Parallel with Global Z axis in
I I I I I I I I I I I I I I I I I I I	Arbitrary direction in Global XX plane in
	Arbitrary direction in Clour Ar plate in
Rotation component direction	Arbitrary direction in Global XY plane in
	Not available in 🛄 🚨
	Arbitrary direction in
Geometry Available analysis results	Straight or curved reference line
Load can be assigned	Line support motion load
Line support group property	
Modules where available	
Motion component	Parallel with Global Z axis in
	Arbitrary direction in Global XY plane in 🛄 🔎
	Arbitrary direction in 🐺 🔟
Rotation component	Arbitrary direction in Global XY plane in
	Not available in 🛄 🚢
	Arbitrary direction in 📲 🔟
Geometry Available analysis results	Straight or curved reference line
Load can be assigned	Line support motion load
Surface support group property	
Modules where available	
Motion component direction	Parallel with Global Z axis in
	Arbitrary direction in 🔛 💷
Rotation component direction	Not available 🖆 🚑 🔟
Geometry	Arbitrary region shape; holes can be placed into it
Available analysis results	Keaction forces

	Load can be assigned	Surface support motion load]
--	----------------------	-----------------------------	---

Table: Properties by Support types

Single Support versus Support group

Single Support defines only one component in one step, whilst Support group places a group of more components in one step. Although single Supports can be combined and work as a group support, it is recommended to define multi-support conditions with Support group, because all support components can

be set or modified later (with the *Properties* tool) inside one settings dialog.



Figure: A fully rigid support modeled by six single Support components or one Support group

Special color-system differentiates *single Supports* from the *Support group* components. Whilst a single *Support* uses the color of the "Supports" *Object layer*, the components of a *Support group* are displayed in the colors equivalent with the support's *Local system* (*Settings* > *All* > *Display* > *Support* and *Local system*).



Figure: Color settings of support types

Definition steps

While support definition, keep in mind not to create unstable situation! For example in 3D modules, model a hinged support under a wall with hinged line support added to a wall at its rigid edge or with rigid line support added to a wall at its hinged wall. In case of unstable situation the program sends a warning message during calculation.



Figure: Modeling same wall support condition in two ways

- 1. If needed (and available in the current FEM-Design module), set a proper position for the *working plane*.
- 2. Start the required support command from Structure tabmenu and choose $\square Define$.
- 3. In case of single *Support*, choose the displacement type you would like to be define:
 - 🕴 Motion
 - Rotation
- 4. Set the support *properties* as mentioned before in the applied support's *Default settings* dialog. See the settings possibilities and options at *Properties* (*Non-Linear Behaviors*).

In the settings dialog of a curved line support, the support direction has to set fixed or variable along the support's reference line.



Figure: Possible support directions in case of curved reference line

- 5. Define the required support component *direction* for the single *Support* or set the reference system for the *Support group*.
- 6. Choose a *geometry* definition method for the support's reference point/line/region.
- 7. Define the support in the model view based on the chosen geometry method.

Optional steps:

8. Multiple supports can be defined in one step by Point Supports and Point Support Groups instead of placing them one-by-one.



Figure: Definition of multiple supports (Point Supports)

- 9. Modify the geometry with the *Edit* menu commands valid for line/region elements.
- 10. Modify the support properties with the *Properties* tool of the support's tool palette.
- 11. Set the display settings of supports at *Settings* > *All* > *Display* > *Support*.



"Label" represents the supports' ID number generated by analysis/design calculation or by clicking *Refresh numbering* (*Tools* menu).

Rotation support is displayed with symbol strongly different from motion representation.



Figure: Typical symbols by support types

Support groups can also be separated from the *single Supports* by displaying their *Local system*.



Figure: Local system separates a one-component Support group from a same directional single Support

12. The supports are stored on "Supports" *Object layers*. At layer settings, the default color of the single *Supports* and pen width of all support types can be set. The color and the symbol line type (e.g. dotted) of selected supports can be modified by *Edit* > *Properties* > *Change appearance*.

Connection Objects

Connection types

The following table summarizes the available "connection model objects" and their main features in the different FEM-Design Modules.

C *Edge connection* is not a "connection object". It is a definition tool for boundary connection conditions, which is available for *Plate/Wall*, *Profiled panel* and *Timber Panel* elements. But, the *non-linear behavior and component settings* are valid for edge connections introduced for the supports and below mentioned connection elements. See the definition modes and possibilities of an *Edge connection at Plate*.

The end connection property of the bar elements are also not "connection objects". You can read more about them at the *Beam, Column* and *Fictitious bar connection* settings.

Point-point connection property	Description
Modules where available	
Motion component direction	Parallel with Global Z axis in
	Arbitrary direction in Global XY plane in
	Arbitrary direction in 🛺 🕅 直
Rotation component direction	Arbitrary direction in Global XY plane in
	Not available in 🛗 進
	Arbitrary direction in 🛄 🕅 💼
Geometry	- (insertion points)
Available analysis results	Connection forces and moments
Line-line connection property	
Modules where available	
Motion component	Parallel with Global Z axis in
	Arbitrary direction in Global XY plane in 🚻 進
	Arbitrary direction in 🛺 🕅 直
Rotation component	Arbitrary direction in Global XY plane in
	Not available in 🟥 進
	Arbitrary direction in 🛄 🕅 直
Geometry	Straight or curved reference lines
Available analysis results	Connection forces and moments

Table: Properties by Connection types

Point-point connection

With *Point-point connection*, different connections can be defined between crossed or connected bars. It is very useful in 3D modules. Some typical examples:

- Bracing (called St Andrew's cross)

A hinged connection between two channels intersecting each other: free rotation around the axis perpendicular to the plane defined by the two channels. (It is not necessary to define the eccentricity; the next figure is schematic.)



- Independent bars

Defining two bars crossed each other with their structural lines but having no connection in real state is a typical problem of modeling. Now, it can be solved easily by creating point-point connection in the bars' intersection with all components set to "free".



- Semi-rigid connection between members

At the *Connections* settings of members (*Beam* and *Column* > *Default settings*) only fully rigid or different type of hinged end connections can be defined. But, *Point-point connection* command gives the possibility to model any other connection types like semi-rigid connection with given stiffness values for rotation. The stiffness values can be set according to the applied standard instructions.



"Real state" modeling

Real eccentric position (e.g. bars connected with their web plates) without eccentricity settings can be also modeled with *Point-point connection*.



Definition steps

_

1. Start Foint-point connection command from Structure tabmenu and choose Z Define.

Point-point connection	

2. Set the properties of the new connection at *Default settings*. See the settings possibilities and options at *Properties (Non-Linear Behaviors)*.



Applying a connection different from rigid state, the *Interface position* (*r*) is very important. $(0 \le r \le 1$.)

r = distance between connection position and start point of definition / distance between start and end points of connection

The next figure introduces the meaning of the interface position (r) (it has no effect if the connection points are in the same position!):



- 3. Set the system for the *direction* of the connection components.
- 4. Select the first object you would like to connect.
- 5. Define the point of the first object you would like to start the connection.
- 6. Define the point of the second object you would like to end the connection. If the connection points between the two objects (crossed each other) are in the same position, just click O to define the connection end point.

Optional steps:

7. Modify the connection properties with the *Properties* tool of the tool palette.

- 8. Add members to a selected connection or remove members from it with 😁 *Edit connected objects*.
 - The next figure shows the steps of adding a new member (beam) to a predefined point-point connection. First, select the connection you would like to modify and then choose the new object you would like to add to the selected connection. To remove a member of a connection, do the same steps with



The next figure shows you the difference between the static-systems of the initial and modified states at all rigid bar end connections and defined hinged point-point connection.



9. Set the display settings of connections at *Settings* > *All* > *Display* > *Connections*.


"Label" represents the connections' ID number generated by analysis/design calculation or by clicking *Refresh numbering* (*Tools* menu).

10. The connections are stored on "Modeling tools" *Object layers*. At layer settings, the default color defines the connection label color.

Each connection element has its own local system. The connection symbol (darker colors, thicker lines etc.) shows the objects having one of the associated above mentioned connection types.

The following figure shows an example how the program makes different symbols depending on what object was selected during the selection of the connected points. But note, that although the definition ways are differ at the column-plate connections, if the properties (directions, components and stiffness values) are the same at both point-point connections (here in the example, hinged connection = all rotation components are free), the static-systems are also the same.



Line-line connection

With the *Line-line connection* command, different connections can be defined between two regions having different positions in 3D (so not having common parts in the geometry model). Some typical examples:

Hinged connection between regions having a gap

Line-line connection A.1 General Etc. Data Motions (M/m/m) Compression Tension Kg/1.1000e+07 M.1.1000e+07 M.1.1000e+07	Rotations [kNm/m/deg] Compression Tension Cx1 1.745e405 v 1 1.745e405 v
Ky' 1.000e+07 ♥ ♦ 1.000e+07 ♥ Kz' 1.000e+07 ♥ ♦ 1.000e+07 ♥ Predefined types	Cy 1.745e+05 v + 1.745e+05 v Cy 1.745e+05 v + 1.745e+05 v Behavior
	* + +
Local system Direction changes along line	Interface position (proportion)

- Slabs on different level

Instead of connecting two slabs with a *Wall* element, the connection can be modeled with the *Lineline connection* tool (if the internal stress and other results are not required in the wall).



Definition steps

1. Start *Line-line connection* command from *Structure* tabmenu and choose *Define*.



2. Set the properties of the new connection at *Default settings*. See the settings possibilities and options at *Properties (Non-Linear Behaviors)*.

na lina cannactian						
ine-time connection	\					
A.1 General Z Data						
Motions [kN/m/m]	Rotations [kNm/m/deg]					
Compression Tension Kx' 1.000e+07 • 1.000e+07 •	Compression Tension Cx' 1.745e+05 💌 🕈 1.745e+05 💌					
Ky 1.000e+07 ♥ + 1.000e+07 ♥	Cy 1.745e+05 • + 1.745e+05 •					
K2 1.0008+07 V 7 1.0008+07 V	(2 1.7458+05 V V 1.7458+05 V					
Predefined types	Behavior					
	* + +					
Local system	Interface position (proportion)					
Direction changes along line	rs 0.500 1 re 0.500					
Setup "Rigid" >	OK Cancel					

Applying a connection different from rigid state, the *Interface position* (r) is very important. $(0 \le r \le 1$.).

r = *distance between connection position and start point of definition / distance between start and end points of connection*

The figure shows two different interface positions at hinged line-line connection between two plate regions: in the first example, the hinged line (the place of the interface) is on the edge of the left plate ($r_s=r_e=0$), and in the second example the interface position is in the middle ($r_s=r_e=0.5$). In both examples, the left plate is supported rigidly and the right one has simple support against to Z-directional motion. The M_x moment diagrams show zero moments where the interface positions are placed.



In the settings dialog of a curved line support, the support direction has to set fixed or variable along the support's reference line.



- 3. Set the system for the *direction* of the connection components.
- 4. Choose a *geometry* definition method for the first connection object's reference line (axis).
- 5. Select the first object you would like to connect.
- 6. Define the first connection line object in the model view based on the chosen geometry method.
- 7. Choose a *geometry* definition method for the second connection object's reference line (axis).
- 8. Define the second connection line object in the model view based on the chosen geometry method.

Optional steps:

- 9. Modify the connection properties with the *Properties* tool of the tool palette.
- 10. Add members to a selected connection or remove members from it with $\stackrel{\clubsuit}{\longrightarrow} Edit \ connected \ objects.$
- 11. Set the display settings of connections at *Settings > All > Display > Connections*. See the possibilities at *Point-point connection*.
- 12. The connections are stored on "Modeling tools" *Object layers*. At layer settings, the default color defines the connection label color.

Rigidity type

This option is used by *Point support group, Line support group, Surface support group, Edge connection* in **3D** *Plate* and *Wall* structural objects, *Point-point connection, Line-line connection,* border and *panel* connection of *Timber* and *Profiled panels*. With rigidity type the user can define and load specific predefined spring constants.

If the rigidity type is *Custom* the user can define the spring constants as in previous versions. To define a type the user has to choose the *Edit library*... option.

Point support group	×		
A.1 General Data			
Type Custom Motions [kN/m] Custom Library Rotatic	ons [kNm/°]		
Compression In Compression Kx' 1.000e+10 Int Cx' Ky' 1.000e+10 Excel types Cy'	ompression Tension 1.745e+08 + 1.745e 1.745e+08 + 1.745e	nt library	
Kz' 1.000e+10 ▼ + 1.000e+10 ▼ Cz'	11 11 11 11 11 11 11 11 11 11 11 11 11	Motions [kN/m]	Rotations [khm/º]
Predefined types Behavi	th	Kx' 1.000e+10 \checkmark \checkmark 1.000e+10 \checkmark Ky' 1.000e+10 \checkmark \checkmark 1.000e+10 \checkmark K2' 1.000e+10 \checkmark \checkmark 1.000e+10 \checkmark	Cx² 0.000e+00 ◆ 0.000e+00 ▼ Cy² 0.000e+00 ◆ 0.000e+00 ▼ Cy² 0.000e+00 ◆ 0.000e+00 ▼
Setup "Rigid" >		Predefined types	Behavior Detach No 💌
	New	Modify Delete Import Export	Close

After defining rigidity types, the defined spring constants will be registered in the Library and it can be loaded in the future.



There are some objects that use the same rigidity type library:

- Line – line connection, Edge connection (3D Plate, Wall), border and panel connection of Timber panels and Profiled panels

Nonlinear connection

The nonlinear behavior of supports and connections (*Point support, Point support group, Line support, Line support group, Surface support, 3D* Plates and Walls *Edge connection, border* and *panel* connection of *Timber panels* and *Profiled panels*) can be controlled by one signed component. It means, if the force in the connection / support happen to act in this selected direction, all spring constants will be set to 0, so called it is detached. This option can be set in the dialog, where the spring coefficients are set. If in the *Detach* list anything is selected (e.g. z' tension) the corresponding spring constant will be automatically set to 0.

1. Select detach component

2. Component is disabled

Point support group	×) [Point support group	×
A.1 General Data Type Custom ▼ Motors [kV/m] Tenson ▼ Conversion Tenson ▼ Ky ² 1.000e+10 ▼ 1.000e+10 ▼ Ky ² 1.000e+10 ▼ 1.000e+10 ▼ Ky ² 1.000e+10 ▼ 1.000e+10 ▼	Rotations [kHm/9] Compression Cx 1.745e+408 * 1.745e+408 * Cy 1.745e+408 * 1.745e+408 * Cy 1.745e+408 * 1.745e+408 * Cy 1.745e+408 * 1.745e+408 *	No x' comp. x' tens. y' comp. v' tens.	A.1 General Data Type Custom • Motions [kV/m] Tension • Kor 1.000e+10 • 1.000e+10 • Kor 1.000e+10 • 1.000e+10 • Kor 1.000e+10 • 1.000e+10 • Kor 1.000e+10 • 0.000e+00 •	Rotations [khm/5] Tension Compression ▼ ● 1.745e+08 ▼ Cy 1.745e+08 ▼ ● 1.745e+08 ▼ Cy 1.745e+08 ▼ ● 1.745e+08 ▼
Predefined types	Behavior Detach	z' comp. z' tens.	Predefined types	Behavior Detach 2' tens. 🔹

If in Calculations dialog the "*Consider non-linear behavior of supports, trusses and connections*" option is checked the calculation method is the following: If in the connection where the detach behavior is defined (e.g. z' tension) the connection force is tension in the given direction (z') in an iteration step all of the other spring constant will be set to 0. If in any later iteration step the connection force will be compression, the spring constants will be set to the previously defined values.

Calculations	X
Calculations Analysis Load cases Imperfections Load combinations Stability analysis Bigenfrequencies Seismic analysis	Analysis Finite element types Fine elements with 9/6/3 nodes. Longer calculation time. Non linear effects in load combination calculations Consider non-linear behavior of setup Consider cracking of RC elements Setup Internal forces Peak smoothing
Recalculation	OK Cancel

There is a big difference between setting any of the components to zero or selecting a detach behavior. If the user sets manually a component to zero, in the analysis the connection or support will not have rigidity in that direction.

Cover

Cover

Cover property	Description
Modules where available	
Geometry	Any shape
Load direction	Arbitrary in 🕂
Load type	Surface load
Available analysis results	-

Table: Cover properties

Definition steps

1. If needed, set a proper position for the *working plane*.

2. Start Cover command from Structure tabmenu and choose 🖳 Define.



Figure: Cover toolwindow

3. Set the cover type if it is slab/roof type or wall type



Figure: Cover types

 4. Choose a *geometry* definition method for the cover's plane. Note: In case of arc geometry in wall covers, you must set the maximum angle between neighboring

covers at Alpha [9] 10.00, since arced walls are replaced with straight sections.

5. Set the load bearing directions with \blacksquare .



Figure: Setting the load direction (monodirection)

6. Defining the supporting structures with After defining the supporting structures, the equidistant lines will appear with the load distribution zones.



Building cover

With this command frequently required shapes such as flat roof, ridge roof, or inclined roof can be very quickly defined.

1 1 1		
Support	Modeling tool Cover	
Building co	ver 🛛	\leq
	h'[m] 1.000 h[m] 4.000	\succ
		\geq
\prec 7		

Figure: Starting Building cover command

Flat roof: and its height parameters:	h'[m] 1.000 h [m] 6.000
No negative value is accepted for either h	or h'.
Lean-to: and its height parameters: No negative value is accepted for either h	h' [m] 1.000 h [m] 6.000 or h'.
Ridge-roof: and its height parameter	h' [m] 1.000 h [m] 6.000

Negative values is accepted for h'.

Keep in mind that in EC, the total height is signed by h which is h+h' in the program's notation. This is important in wind calculation according to EC which requires the reference height.

After setting the height parameters you can create the layout rectangle. In the picture below you can see the three types with their assumed stiff direction.

🔚 FEM-Design 11.0 - 3D Structure - Untitled 📃 🗟 🔀																						
File Edi	t Draw M	vlodify Tools	Settings	View Wi	ndow Help																	- 8 >
Structure	Loads	Finite elements	Analysis	RC design	n Steel design	Timber	design															
- (• 1	- 🗖 t		l ≲†		000	67765	\$	\$∠.	\$	<u>ک</u>	\$	\$ \$	₹ î								
4	Datum		Bar		Shell	← Prefab	bricate 🕨	•		Support		4	Mode	ling tool	I Cor	over					Reinf	Docu
	× × × × × × × × × × × × × × × × × × ×																					
Building c	over · Define	 Direction of h 	rst edge - Sta	art point (LB :	= Mace poin						12.2	34 M	61.300 m	0.000	m Layers		$+ \rightarrow \top ($	Snap	+	L /2	⊕ <u>+</u> ⊡	0

All building covers are monodirectional by default except the upper surface of flat-roof. (They can be changed on<u>e</u> by one after definition.)

By clicking on ? (Properties) you can change the ID and the height parameters of the building cover:

Building cover
Identifier (.position number) CO1
Enter identifier (max 50 characters). The position number is generated automatically when "Tools", "Refresh numbering" function is called. Initially it is set to 0.
h' [m] 1.000 h [m] 4.000
OK Cancel
/ ¥

You can explode the building cover with \square , and then edit the covers one-by one.

3D Members

This chapter summarizes all features and properties of a 3D member such as Soil !!! (Link).

Soil



Soil's *ground level, foundation level, water level,* and strata top level can be different in each boreholes belonging to the soil. Boreholes are created in the corners of the soil region by default, but the user can create more if necessary (see 1.2). Soil's limit depth is constant.



The soil material details can be set individually:



User can choose if the he/she wishes to calculate soil as solid elements in analysis or just wants to consider it in foundation design calculation. In the Settings/Calculation/Code, Configuration dialog this option can be set.

Settings	Code, configuration
Calculation Calcu	Code Code independent Eurocode Eurocode (NA: Hungarian) Eurocode (NA: Danish) Eurocode (NA: Swedish) Eurocode (NA: Swedish) Eurocode (NA: Norwegian) Eurocode (NA: Polish) Eurocode (NA: Polish) Eurocode (NA: Seman) Eurocode (NA: British) Soil Image: Calculate soil as solid elementi Show this dialog at startup
Save as default	OK Cance

The soil can be loaded only with *Point load*, *Line load*, *Surface load* (not with moments).

Concentrated mass cannot be defined in the soil.

SECTION EDITOR

FEM-Design *Section Editor* gives the possibility to define, edit and store profiles for the bar elements (beam, column and truss element).

The functions of the Section Editor:

- drawing new sections,
- definition of new sections by parameters,
- definition of complex sections by mixing drawing and standard profiles,
- editing cross-sections,
- calculation of sectional data, and
- profile registration and organizing.

Section editor can be started directly from design modules that work with bar elements from the Tools menu.



Figure: Section Editor as a built-in tool of design modules

Section Editor can be started as an independent FEM-Design application by clicking the *Section Editor* icon in the *"FEM-Design tools"* folder of

- Start > All Programs > FEM-Design...,
- Windows Taskbar > FEM-Design System Tray (if installed),
- short-cut 🗭 of *Desktop* (if installed).

Section Editor module has only one *tabmenu* called ^{Cross-section} that contains all the section definition tools, but the FEM-Design drawing, editing, displaying etc. commands are also available on the *Section Editor* user-interface.



Figure: Section Editor user-interface

Sections are two-dimensional region elements defined in the Y-Z plane of the *Global* (and *UCS*) coordinatesystem. Section Editor uses the Y-Z system for defining and calculating profiles, because the sections are perpendicular to the x' axis (parallel with the y'-z' plane) of the *local coordinate system* of bar elements.

Cross-Section Definition

New cross-section can be defined by *drawing section* regions with arbitrary geometry, or by giving parameter values of a *parametric section* available in the built-in *Section library*, or by combining the two methods to build *complex sections*.

Drawing a Section



Figure: Section defined as polygon

With Pick lines tool, section can be generated by one-clicking drawing lines and arcs that define closed region. Drawing elements can be defined previously with the *Draw* menu commands or they can be imported from *DWG/DXF files*. With Pick existing region tool, section can be generated by one-clicking drawing region defined by the *Region* command of the *Draw* menu.



Figure: Section created by selecting closed contour

With the **Hole** tool, openings and cuttings can be inserted to section regions with any shape defined by the previously listed region definition modes.



Figure: Section edited by Hole

E Cor

You can read about the definition steps and functions of different region and hole geometries at the similar tools ("blue color-linked") of *Structure definition*.

Section region shapes, corner points and side lines can be modified with the *Modify* menu tools.



Figure: Section modified with Edit tools

Parametric Section

Although, quick definition of parametric profiles available in the settings dialogs (*Cross-sections*) of *Column*, *Beam* and *Truss member*, parametric sections can be inserted for editing and using them to create *complex sections*.

Use *Insert section into drawing* command to browse from parametric sections as well as standard and user-defined profiles, and to insert one or more of the required section with chosen geometry and sizes into the drawing area.

Although, you may choose for example a timber profile for editing and register as another fabrication type (for example steel, concrete etc.), we suggest you to choose profile type that will suit for later design. But, always you decide the available design type of the section defined by *Section Editor* when *registering* it with its *Fabrication technology* into *Section library*.

Definition steps

1. Choose a parametric section shape in *Section library* or select a predefined (standard or registered user-defined) profile from *New* >/*Size*... option.



Figure: Dialog for defining parametric sections or for choosing predefined sections

- 2. In case of parametric profiles, set the required geometric parameters. Each shape has a preview with the meaning of the parameters.
- 3. Set the insertion point at *Insertion point* tab, where the placement can be defined. Default setting for placement is the center of gravity, but it can be changed by clicking on the specific points (green points) or set by y' and z' coordinates.
- 4. Click *OK* to start inserting the defined or selected profile into the drawing area.



Figure: Definition of insertion point position

- 5. Place the section with its insertion point on the drawing area. Click Or or to insert the section in the origin of the global Y-Z system.
- 6. Further rotation can be set with the direction of the section's z axis. Accurate angle can be typed in the *Command line* or click or to use the direction set in the *Reference point*... dialog.

Complex Section

• Only homogenic complex profiles can be defined in the *Section Editor*, because one material can be assigned to it later in the design modules when defining beams, columns and bars.

At complex sections, intersecting or overlapping among section regions are not allowed. If there is any problem, the program stops the *sectional data calculation* with an error message.

As read in the previous chapter, quick definition of complex profile can be done by the repetition of section insertion with different positions and directions.



Figure: Complex profile built by parametric profiles

Of course, combining the section drawing and insertion modes various complex cross-sections can be defined.



Figure: Mixture of definition modes

Cross-Sectional Data

Before calculating the data of the defined section, it is recommended to place the section's centre of gravity to the global system origin. With the *Query* command of *Tool* menu, you can easily find the center of gravity point of the defined section region and insert a drawing point.





Query finds the center of gravity point of complex sections, if you merge the connected region parts with the *Union* tool of the *Modify region* command (*Modify* menu).



Figure: One complex region after merging region parts

The following settings (*Settings* menu) effect on sectional data calculation:

Settings			
Settings Settings Settings Calculation Options Cm Display result Settings Window	A Area P Perimeter A/P Area / Perimeter A/P Area / Perimeter Yg, Zg Center of gravity Ys, Zs Shear center I/J.2 Moment of inertia Wy, Wz Section modulus ez max, ey max Max. distance of extreme fibre win, ey min Min. distance of extreme fibre Wit Torsion section modulus Igamma Warging parameter Iyze Argie of principal direction Ini, W2 min Principal min. section modulus Vorming Parameter Principal mix. section modulus VT min, W2 min Principal mix. section modulus W1 max, W2 max Principal mix. section modulus W1 max, W2 max Principal mix. section modulus w2 mix. e1 max Max distance of extreme fibre win e1 min Min distance of extreme fibre	Settings Settings Settings Clowing Calculation Display Settings Display Window	Options Ink the touching regions automatically at the next calculation This writcher has effect for the followings: Perimeter (P): Area / Perimeter (A/P): Shear center (Ys, Zs): Shear center (Ys, Zs): Torsion modulus (Wt): Torsion section modulus (Wt): Wagner parameter (z1, z2, z gamma) Overwrite the user modified sectional data at the next calculation
Save as default	<u> </u>	Save as default	OK Cancel

Figure: Calculation settings

- Display result

Set data types you want to display after calculation in the *Sectional data window*. The data list contains all available data on which multiply-selection such as and and . (See the meanings of the sectional data at *Sections*.) All data can be selected by the *Select all* button.

- Calculation options

The *Link the touching regions automatically at the next calculation* option handles the connected regions as one complex region in calculations. The option influences the value of the following parameters: perimeter (*P*), area/perimeter (*A*/*P*), shearing center (*Ys*, *Zs*), shearing factors (*Rho y, Rho z, Rho yz, Rho 1, Rho 2*), torsion moment of inertia (*It*), torsion section modulus (*Wt*), warping parameter (*Igamma*) and Wagner parameters (*z*1, *z*2, *z gamma*).

To calculate the sectional data set at *Monitored data* click *Calculate* on the cross-section tabmenu. The results are automatically displayed in the *Sectional data window*.



Figure: Calculated sectional data

If you make modifications on the section geometry, you have to rerun the sectional data calculation with *Calculate.*

As a modeling feature, a calculated cross-sectional data can be manually modified/overwritten manually with arbitrary given value. With this possibility the user can create virtual cross-section with known data. Double clicking on a data, custom value can be set for it in the *Modify sectional data* dialog. Modified data is signed with ":" instead of "=" in the *Sectional data window*. The original value can be reset for the current data with the dialog's *Calculate* option, or you can reset all data with the *Overwrite the user modified sectional data at the next calculation option* (*Settings > Calculation options*).



Figure: Customizable sectional data

Documentation of Cross-Section

With the Insert report command, the table of the calculated sectional data can be placed on the drawing area based on the *Display result* setting.



Figure: Sectional data inserted to drawing area

If you make modifications on the section geometry, you have to rerun the sectional data calculation with *Calculate* and the content of the table refreshes automatically.

The table is stored on the *Sectional data* object layer. The color of the table can be modified together with the layer's color. More display changes such as the modification of the text and border style can be done with the *Properties* > *Change properties* tool (*Edit* menu).

The defined section is lying on the *Cross-section* object layer, so the default blue color of section regions can be modified together with the layer's color.

Insert section into library - Section Library

To add a new section having valid calculations to the *Section library*, click Register and set the required settings such as the name of the storing folder (*Library* name) and the *Type* and *Size* IDs. Also set the fabrication technology effect on the later available design calculations. To store the section press *OK*.

Before registering the current section click *Calculate*.

Туре	\mathbf{X}
Fabrication technology Rolled steel 1.	
Location	_
Library Section library 2. Group 3. Type 4. Size 5. OK 6. Cancel	Section library Concrete sections Steel sections Timber sections Hollow Core Used sections

Figure: New section registered to Section library

Fabrication technology	Design
Rolled steel	Steel bar design
Cold-formed steel	Steel bar design
Weld steel	Steel bar design
Concrete	RC bar design
Timber	Timber bar design
Generic	- (only for analysis)

Table: Fabrication technology and the assigned design type

The program automatically detects the shape type of the sections and shows it as a symbol. It is important for the later design calculations.

The content of the *Section library* can be edited (renaming, deleting etc.) with the *Library* command.

Edit section library		$\overline{\mathbf{X}}$
i i IPE	^	z
- 100	_	I I I
- 140		1 G <mark>S, y</mark>
	Ш	U U
240 270		2
300		A = 2009 mm2 P = 622.5 mm
		A/P = 3.227 mm
450	~	(Yg = 0.0000 mm)
New > Modify Delete		Import Export
		OK Cancel

Figure: Library and content editing

A registered section can be added to beam and column elements by choosing it in the *Section* tabpage of the element settings dialog (*Default settings*).



Figure: User-defined section used for bar elements (beams)

LOADS

After defining the *structure* and *support* elements, the next step is adding loads to the statical system. This chapter sums all load possibilities, load definition (directions and geometry) and combination modes. *Mass* definition required for dynamic calculation is also introduced in this chapter. Let's click on to reach all load definition commands.

Load Types

Depending on the current *FEM-Design Module* (license you have), the available type of load commands is different. Loads can be defined with their insertion point, action line or action region.

Туре	Modules where available	Definition mode	Direction	Intensity
Structural dead load		(Automatic)	Global Z-axis	-
Soil dead load	ļ.	(Automatic)	Global Z-axis	-
Point load (force and/or moment)		Point	Horizontal Arbitrary	Constant
Line load (force and/or moment)		Line	Horizontal Arbitrary	Constant/Variable
Surface load (force)		Region	Horizontal Arbitrary	Constant/Variable
Line temperature variation load		Line	Fixed Arbitrary	Constant/Linearly variable Constant/Variable
Surface temperature variation load		Region	Fixed Arbitrary	Constant/Linearly variable Constant/Linearly variable
→ Line stress load		Line	Fixed	Constant/Variable
Surface stress load		Region	Fixed Arbitrary Arbitrary	Constant/Variable
Point support motion (motion and/or rotation)		Point	Support direction	Constant
Line support motion (motion and/or rotation)		Line	Support direction	Constant Constant/Variable
Surface support motion (motion)		Region	Horizontal Support direction	Constant/Variable
Shrinkage		(Automatic)	-	-
Seismic load	M 1	(Automatic)	-	-

Wind	(Automatic)	Horizontal	Constant
⚠ Snow	(Automatic)	Vertical	Constant
	(Automatic)	Horizontal	Constant

Table: Load types and their main properties

Load Direction

Most of the load objects need direction settings. The next table summarizes only the editable direction possibilities by load types. Other load types have fixed direction (for example in Plate module, Force direction is always perpendicular to the calculation plane, so it is parallel with the global Z direction).

Туре	Modules where available	Direction for	Direction Modes
Point load		Moment direction	† Predefined direction
		Force direction	† Parallel with line
	M 1. 1	Force/Moment direction	Perpendicular to plane
Line load		Moment direction	† Predefined direction
		Force direction	† Parallel with line
		Force/Moment direction	Perpendicular to plane
			U Object's local system
I Surface load	*** 12 🐺 🔒	Force direction	† Predefined direction
			† Parallel with line
			Perpendicular to plane
			LObject's local system
Line temperature		Direction	† Predefined direction
variation load			† Parallel with line
			Perpendicular to plane
			Object's local system
→+ Line stress load		Direction	Predefined direction
			Parallel with line
			Perpendicular to plane
	.+ +++ (* → .	Direction	Cobject's local system
Surface stress load		Direction	T Predefined direction
			Parallel with line
			Perpendicular to plane
	4	Potation direction	U Object's local system
Point support motion load		Motion direction	T Predefined direction
motion iouu		Motion (Retation direction	Parallel with line
		wouldny kotation direction	Perpendicular to plane
:	4	Potation direction	Cobject's local system
Line support		Motion direction	Predefined direction
motion iouu		Motion /Rotation direction	Parallel with line
		wouldn/ Kotation direction	Perpendicular to plane
	→	Mation diverties	└─ Object's local system
I Surface support	→ <u>]</u> ↓	Notion direction	T Predefined direction

motion load	† Parallel with line
	Perpendicular to plane
	Upject's local system

Table: Load types and their directions

† Predefined direction

With this option an axis/a plane of the *Global* or the *User-defined* (*UCS*) co-ordinate system can be set for the required load direction. The direction can be chosen from the drop-down list attached to the *Predefined direction* option. The available directions depend on the applied FEM-Design Module (e.g. *Plate, 3D Structure* etc.).



Symbol	Meaning of direction	System
ĨI,	Parallel with XY plane	Global
<u>آ</u>	Parallel with YZ plane	Global
	Parallel with XZ plane	Global
$\langle \mathbf{z} \rangle$	Parallel with UCS (XY plane)	UCS
$\langle \overline{\mathbf{x}} \rangle$	Perpendicular to UCS (XY plane)	UCS
	Parallel with global X axis	Global
\mathbf{z}	Parallel with global Y axis	Global
	Parallel with global Z axis	Global
\downarrow	Parallel with X axis of UCS	UCS
\sim	Parallel with Y axis of UCS	UCS
₹	Parallel with Z axis of UCS	UCS

Table: The available directions to set the new load direction



Figure: Examples of temperature loads placed on bar elements

In some cases, additional direction setting can be chosen from the definition tool palette:

 \Rightarrow "Positive direction":

The orientation is the same with the selected axis orientation;

→ "Negative direction":

The orientation is the opposite of the selected axis orientation.



Figure: Examples of Point load direction

†| Parallel with line

With this option, the required load direction can be defined manually with its start and end points.



Figure: The load direction is parallel with the defined direction

Perpendicular to plane/line

With this option, the required load direction will be perpendicular to a defined plane/line. The plane can be given with three points and the line with two points (start and end points). In case of the perpendicular plane, the third point defines the final orthogonal direction, which the new direction will be parallel with.



Figure: The line load direction is perpendicular to the defined plane

└─Object's local system

If the load direction has to be set in the local co-ordinate system of the "assigned" structural object (beam, column, plate, wall and support), the fastest definition of load direction is to use the *Objects' local system* option. This feature is available for *Line, Surface, Line temperature variation, Line stress, Surface stress, Point* support motion, *Line support motion* and *Surface support motion load* in the 3D modules.

Using *Object's local system*, the geometry definition of the line or surface load is skipped, because one click on the assigned structural object is enough after setting the requested local system axis direction.

 \wedge

In case of surface loads, only load with constant intensity can be defined with the tool *Select objects to load in local system*. But, you can change intensity values (linear distribution) with the *Variable intensity* tool of the applied load command as a next step.



One-click definition of constant surface loads perpendicular (e.g. wind) to planar objects (e.g. shells).



Figure: Fast definition of surface load by using object's local system

One-click definition of line load (constant or variable) perpendicular (e.g. wind load) to a beam reference line.



Figure: Fast definition of line load by using object's local system

One-click definition of point support motion parallel with a component direction of a point support.



Figure: Support motion added to a point support component

 \bigtriangleup Only the same type support can be selected for a support motion:

Support motion load type	Support type
Point support motion load	Point support
Line support motion load	Line support
Surface support motion load	Surface support

Table: Support motion load types and their proper support types

Change Direction

Any previously set direction can be modified by the modifying commands (*Modify* menu): *Change direction* and *Rotate*.

Change direction uses the *Predefined direction, Parallel with line* and *Perpendicular to plane* direction definition tools.

Rotate edits a selected direction or the main direction of a selected system with giving new direction points or the rotation angle. Rotation works around a given point or an axis.

Naturally, you cannot modify fix directions. For example in the *Plate module* the point, line and surface forces are always vertical (perpendicular to the calculation plane of the plates).

Load Geometry

The definition modes and the available shape of the loads' action line/plane depend on:

- the load type: point, linear and surface load, and
- the current FEM-Design module.

The Tool palette of a load command contains only the available modes. The next table summarizes the geometry possibilities by the load types.

Туре	Modules where available	Definition mode	Geometry
Point load		Reference point	Edit point (give an insertion point)
Line load		Reference line	Straight line Arc by center, start and end points Arc by 3 points Arc by start, end point and tangent Line by selection
I Surface load		Reference region	 Rectangular Circular Polygonal Pick lines Pick existing region
Line temperature variation load		Reference line	Straight line Arc by center, start and end points Arc by 3 points Arc by start, end point and tangent Line by selection
Surface temperature variation load		Reference region	 Rectangular Circular Polygonal Pick lines Pick existing region
★ Line stress load		Reference line	Straight line

		• Arc by center, start and end points • Arc by 3 points
		- Arc by start, end point and tangent Line by selection
Surface stress load	Reference region	Rectangular
		○ Circular
		Polygonal
		Pick lines
		Pick existing region
Point support	Reference point	Edit point (give an insertion point)
motion load		Select point
Line support	Reference line	Straight line
motion load		• Arc by center, start and end points
		Arc by 3 points
		Arc by start, end point and tangent
		Line by selection
Surface support	Reference region	🗖 Rectangular
motion load		⊙ Circular
		D Polygonal
		Pick lines
		Pick existing region

Table: Loads and their geometry definition

If you define a load that does not acting on a structural element, a warning message appears during the calculation. Continuing the calculation does not take the "unconnected" load into consideration.

It follows from the previous fact, that if the documentation does not need "perfect geometry" for loads in some cases, you can spare time with neglecting some editing steps.



Table: Uncut, but unnecessary load parts

The units of the loads can be set at the *Settings* > *Units*.

Straight line

The steps of a straight line definition:

- 3. Define the start point of the line by giving coordinates or mouse-clicking.
- 4. Define the end point of the line by giving coordinates or mouse-clicking.



Figure: Some examples for defining line-type loads with Straight line

In 3D modules, the "curved" walls are modeled as planar shells instead of curved ones, so their base lines are straight segments and not real arcs. So, if you would like to place line loads on the reference contour of 3D walls, use the *Straight* or the *Line by selection* tool instead of one of the arc definition tools (see later). Otherwise, the misplaced load will not be taken into consideration in calculations.



Figure: Incorrect and correct ways to define line-type loads on "curved" wall edges in the 3D modules

•Arc by center, start and end points

The steps of an arc definition with its center, start and end points:

- 4. Define the center point of the arc by giving coordinates or \heartsuit mouse-clicking.
 - Use the Center Object Snap tool, if you would like to define the center point of the line load in another center point of an arc. (See the next figure, where the (1*) step means that the center point of the curved slab edge is selected for the load's center point.)
- 5. Define the start point of the arc by giving coordinates or \bigcirc mouse-clicking.
- 6. Set the drawing direction (clockwise or counterclockwise) with \bigotimes mouse-clicking. Define the end point of the arc by giving coordinates or \bigotimes mouse-clicking, or set the central angle (4.) by giving its value. Circle can be defined by angle 360°.



Figure: An example for defining line-type loads with Arc by center, start and end points



Figure: Drawing direction and angle definition

Arc by 3 points

The steps of an arc definition with its three points:

- 4. Define the start point of the arc by giving coordinates or \bigcirc mouse-clicking.
- 5. Define the end point of the arc by giving coordinates or O mouse-clicking.
- 6. Define the third, peripheral point of the arc by giving coordinates or \bigcirc mouse-clicking.



Figure: Some examples for defining curved line load with Arc by 3 points

Arc by start, end point and tangent

The steps of an arc definition with its start, end point and tangent:

- 4. Define the start point of the arc by giving coordinates or \heartsuit mouse-clicking.
- 5. Define the end point of the arc by giving coordinates or O mouse-clicking.
- 6. Set the tangent side with *mouse-clicking*. Define the tangent direction from the start point with a third point (e.g. a point on a tangentially connected line) by giving coordinates or *mouse-clicking*.



Figure: Definition of a curved line load tangent to a beam



Figure: Although same definition points are defined, the tangent side is different

Select point / Line by selection / Pick existing region

By selecting previously defined points, lines and regions, loads can be inserted very fast, in one step:

- **Point-type loads**: select drawing points or *point supports* (in case of *Point support motion load*) with one of the *selection modes*.
- Line-type loads: select drawing lines, reference line of 1D *member structural elements*, region (drawing or *structural object*) edges or line supports (in case of *Line support motion load*) with one of the *selection modes*. The length of the loads' action lines will be equal to the selected line elements.
- **Surface-type loads**: select drawing regions, drawing solid surfaces, *Planar objects* or surface supports (in case of *Surface support motion load*) with one of the *selection modes*. The size of the loads' action surface will be equal to the selected region elements.



Because only the loads/load parts located on structural elements will be considered in calculations, these "definition by selection" modes are the easiest technique to define loads with perfect accuracy.



Figure: Examples for defining line-type loads by selecting beams


Figure: Adding line loads to the members (beams) of a frame structure



Figure: Defining line loads on structural element edges

In 3D modules, the "curved" walls are modeled as planar shells instead of curved ones, so their base lines are straight segments and not real arcs. So, if you would like to place line loads on the reference contour of 3D walls, use the *Line by selection* or the *Straight* tool instead of one of the arc definition tools. Otherwise, the misplaced load will not be taken into consideration in calculations.



Figure: Defining line loads on structural element edges

-OZ

With *Pick existing region*, surface loads can be easily place on 3D shell elements (plates and walls) or drawing solid surfaces (defined by the *Draw* > *Solid* command).

Point/Line/Surface support motion loads will be considered in the calculations, if they are placed into supports, so it is recommended to select the reference point/line/surface of previously defined supports in case of motion load definition.



Figure: Defining motion loads in supports

Rectangular

The steps of defining a rectangular surface load:

- 5. Define the point of the first corner by giving coordinates or mouse-clicking.
- 6. Define the point of the end corner by giving coordinates or mouse-clicking.



Figure: Defining a rectangular surface-type load



The geometry of rectangular regions as well as other (later mentioned) region shapes can be edited by the *Modify region* > *Split region* tool and other editing tools (*Edit* menu). Also the *Hole* tool of surface loads' definition command can be used to edit the reference regions.

Circular

The steps of a circular region definition:

- 1. Define the center point by giving coordinates or mouse-clicking.
- 2. Define the radius by giving its value or a point on the circle (with coordinates or mouse-clicking).



Figure: Defining circular surface loads on a plate above columns and terrace part

🖿 Polygonal

The steps of a polygonal region definition:

- 3. Define the points of the polygon vertexes by giving coordinates or \bigotimes mouse-clicking.
- 4. Close the polygon with \bigotimes^{p} mouse-clicking or key.



Figure: Defining polygonal surface load

Pick lines

With this method, surface-type loads can be placed on **closed contours** defined by

- previously defined drawing lines,
- the edges of a previous defined drawing region, a plate, a wall (only in 3D modules) or surface support, or
- imported (DWG/DXF) drawing lines that can be used as sketches of surface load shapes.

It is a one-click definition mode: select a closed contour defines the requested shape of the surface load with mouse-clicking.



Figure: Defining surface loads by using close contours

In case of line junctions, more than one line/edge has to be selected to make clear the continuity of the requested closed contour.



Figure: Selection of more lines to define the right path for the closed shape

"Holes" in Surface Loads

Holes and cuttings can be added to surface loads with the *Hole* tool. The following geometries can be used for holes:

The steps of a hole definition:

- 3. Select the surface load with mouse-clicking. Clicking a region places the *UCS* into the region plane, so giving hole coordinates needs only X and Y values from the UCS origin.
- 4. Define the geometry of the hole or cutting with one of the following geometry modes:



Figure: Editing a surface load (previously defined by Pick existing region) with the Hole tools

Holes can be easily copy inside a surface load with the *Copy command* (*Modify* menu). To set the distances/new positions, the *UCS* has to be in the plane of the host region(s).

Load Cases

Loads in FEM-Design are represented with *Load cases*. A Load case has a name and physically contains one or more load objects.

Special loads (invisible loads) like dead load, shrinkage and seismic effect can only defined with Load cases. For timber elements *duration classes* can be set with the *Load cases* command.

Although a Load case can be assigned later to loads, the first recommended step of load definition is the load case classification (load case list).

No	Name	Туре	Duration dass (EN 1995 1-1)		ОК
1	Wind load X	Ordinary	Permanent		Cancel
2	Wind load Y	Ordinary	Permanent		
3	Soil dead load	+Soil dead load	Permanent		Import / Export >
4	Deviation X	Ordinary	Permanent		
5	Deviation Y	Ordinary	Permanent		Insert
6 Shrinkage	+Shrinkage	Permanent	=		
7	Snow load	Ordinary	Permanent		Delete
8	Imposed load	Ordinary	Permanent		Delete all
9	Dead load	+Struc. dead load	Permanent		
10	Seismic load	+Seis load, Fx+Mx	Permanent		
				-	
				-	

Figure: Load cases

Definition steps

- 1. Start *Load cases* command from *Loads* tabmenu.
- 2. Define Load case name in the *Name* column.
- 3. Set the *Type* of the new load case:

Туре	Туре	Modules where
+Dead load V Ordinary	Ordinary	
+Dead load +Shrinkage	- Deed lood	
+Seismic, sway, x, + +Seismic, sway, x, -	+Dead load	
+Seismic, sway, y, +	+Shrinkage	
+Seismic, torsional, x, +	+Seismic	
+Seismic, torsional, y, + +Seismic, torsional, y, + +Seismic, torsional, y, -		

Table: Load case types by FEM-Design modules

- "Ordinary" means that no additional effect will be assigned to the load case
 - "+Structural Dead load" means that the weight of all structural elements, "+Soil Dead load" means the weight of soil, which is calculated automatically, will be assigned to the load case as an invisible load. So, a "Dead load"-type load case contains automatic dead-load and can contain further manually defined loads (e.g. "dead load" of the non-load-bearing (non-core) parts of a composite slab).

Because "+ Structural Dead load" and "+Soil Dead load" type includes the dead load (calculated from the geometry and material) of all structural elements, define only one dead load type load case in one project. If you define more than one "+ Structural Dead load" and "+Soil Dead load" type load cases never group them in the same Load combination!

In all modules automatic dead load works in the global Z direction, except for the *Wall* and *Plane Strain* modules, where the dead load direction is parallel with the global Y.

- "+*Shrinkage*" means that free *shrinkage strain* behavior will be considered as a load effect in concrete design. The shrinkage strain value can be set at the material properties of concrete structural elements.
- "+*Seismic*..." means that sway and torsional effect components calculated from *Seismic calculations* will be considered as a load effect in analysis and design calculations.
- 4. In case of *Timber design* only, set a load-duration class for a load case according to the regulations of Eurocode 5 (EN 1995-1-1:2004). The load-duration classes are characterized by the effect of constant load acting for a certain period of time in the life of the structure. For a variable action the appropriate class shall be determined on the basis of an estimate of the typical variation of the load with time.

Actions shall be assigned to one of the load-duration classes given for strength and stiffness calculations.

Load-duration class Order of accumulated duration		Examples of loading
	of characteristic load	
Permanent	More than 10 years	Dead load
Long-term	6 months – 10 years	Storage
Medium-term	1 week – 6 months	Imposed floor load, snow
Short-term	less than one week	Snow, wind
Instantaneous		Wind, accidental load

Table: Load-duration classes and examples of load-duration assignment (EN 1995-1-1:2004)



Since climatic loads (snow, wind) vary between countries, the assignment of load-duration classes may be specified in the National annex.

Optional steps:

- 5. New load case can be inserted to the Load case list in the required row position with *Insert load case*. Just click in the *Name* field you would like to insert the new load case and press the *Insert load case* button. In the pop-up dialog, set the name, the type and the duration class of the new load case.
- 6. Load cases can be removed from the Load case list with *Delete load case*. Just click a field of the load case you would like to delete and press the *Delete load case* button.
- 7. The load case list can be set as default for next project by clicking the *Save as default* button.

After finishing the load case definition, a load case can be assigned to a load in two modes: by choosing a case directly from the Loads tabmenu (drop-down list) or in the *Default settings* dialog of the current load command.



Figure: Load case selection for load objects

A color is assigned to a load case after its definition. That color represents the color appearance of the loads included in a load case. The default load case color is red, but you can set different colors by load cases at the *layer settings*.



Figure: The color-system of Load cases represents the load appearance

Load Definition by Types

The commands for defining loads can be started from the Loads *Tabmenu*.

To show the mostly used load commands and to hide the rarely used ones; you can organize the icons in expanded or compact form.

Expanded mode	compact mot
± <u>↓</u> ↓	±
 Force 	▶ For

In compact mode, you see the icon of the last used command. If you click on the command's symbol, the program opens its tool palette. You can reach the other same type load by clicking on the symbol.

Each command has a *Tool palette* with the customizable load properties (load value/intensity, host load case etc.) and the definition tools of the load geometry and position (direction).

Structural dead load

The weight of the load-bearing structural elements are calculated automatically by the program and is considered in the calculations, if "+*Structural dead load" type load case* is defined and it is added to the load combinations or load groups. So, the so-called "auto dead-load" is calculated from the geometry and material-dependant density value.

The direction of the dead-load is vertical, but the vertical direction differs in the different FEM-Design modules.



Table: The structural dead-load direction by FEM-Design Modules

Definition steps

- 1. Start the *Load case command,* and define a "+Structural dead load" type load-case with a name (such as "Dead load"). So, the "auto dead-load" is allowed on load-case level.
- 2. Define loads (*Point-, Line-* and *Surface loads*) under the load-case defined in the previous, 1st step, if additional dead-loads coming from invisible not-load-bearing parts (non-load-bearing layers of composite structures) are requested.
- 3. Add the load case defined in the 1st step to load combinations/groups depending on the later design.

The auto dead-load is always invisible. Only the additional, manually defined dead-loads can be displayed (as point/line/surface load) by activating the layer of the host load-case.

Soil dead load

The weight of the soil is calculated automatically by the program and is considered in the calculations, if it is added to a "+*Soil dead load*" *type load case* and then load combinations or load groups. So, the so-called "auto dead-load" is calculated from the geometry and material-dependent density value.

The direction of the dead-load is vertical, but the vertical direction differs in the different FEM-Design modules.



Table: The soil dead-load direction by FEM-Design Modules

Definition steps

- 1. Start the *Load case command*, and define a "+Soil dead load" type load-case with a name (such as "Dead load"). So, the "auto dead-load" is allowed on load-case level.
- 2. Define loads under the load-case defined in the previous, 1st step.

The soil can be loaded only with *Point load, Line load, Surface load* (not with moments).

3. Add the load case defined in the 1st step to load combinations/groups depending on the later design.

The auto dead-load is always invisible. Only the additional, manually defined dead-loads can be displayed (as point/line/surface load) by activating the layer of the host load-case.

Shrinkage Strain

Free shrinkage strain behavior can be modeled as a load effect in reinforced concrete design and check.

Shrinkage effect is calculated as a specific rotation. The following formulas give the example of calculation for reinforced concrete slabs:



 $N_X = E_c A_c \varepsilon_{cs}$ The specific normal force causing the given shrinkage value (ε_{cs} [‰]) in the concrete zone of the section is (here in X direction).

 $z_s = \frac{n \cdot S_s}{A_c + n \cdot A_s} \qquad \text{Th}$

The position change of centre of gravity considering reinforcement bars (here X-

direction; see dashed line), where $n = \frac{E_s}{E_c}$ and S_s is the statical moment of (here) X-directional bars around the Y axis of the calculation plane.

- $M_Y = N_X \cdot z_s$ The moment around the Y axis of the calculation plane from N_x because of the position change of centre of gravity.
- $\kappa_Y = \frac{M_Y}{E_c I_Y}$ The specific rotation (curvature) from M_y for 1 meter wide section, where

$$I_Y = I_{c,Y} + n \cdot I_{s,Y} - z_s^2 (A_c + n \cdot A_s).$$

Definition steps

1. Set the shrinkage strain (ε_{cs}) value at the material properties of concrete structural elements.

Plate properties	
A.1 General Material	
Library	Application data
Concrete 	Gamma c
	Gamma cE 1.20
	Gamma s 1.15 >
- C35/45	Alfa cc 1.00
C40/50 C45/55	Alfa ct 1.00
C50/60	Environment X0 💌
Steel	Creep factor
5 275	Shrinkage [%] 0.25
	Don't forget to setup creep factor!
New Modify Delete	Import Export
	OK Cancel

Figure: Shrinkage value of concrete materials



Shrinkage effect is taken into consideration in RC design and check of concrete elements, which have *applied reinforcement*.

2. Start the **Load case command**, and define one "+Shrinkage" type load-case with a name (such as "Shrinkage effect"). That means, the automatically calculated shrinkage effect is allowed on load-case level.

No	Name	Туре	Duration class (EN 1995 1-1)	-	ОК
1	Structural dead load	+Struc. dead load	Permanent		Cancel
2	Soil dead load	+Soil dead load	Permanent		Save as defai
3	Live load 01	Ordinary	Permanent		Jave as delad
4	Live load 02	Ordinary	Permanent		
5	Shrinkage	+Shrinkage	Permanent		
6	Wind X	Ordinary	Permanent		
7	Wind Y	Ordinary	Permanent		
8	Snow	Ordinary	Permanent		
9	Seismic X+	+Seismic, sway, x, +	Permanent		
					Insert load case
					Delete load ca

Figure: Shrinkage effect added to a load case

3. Add the load case defined in the 2nd step to load combinations/groups depending on the later RC design.

Point Load

Depending on the current FEM-Design module, point force and/or moment can be defined easily by giving just the load insertion point and direction. The load properties and definition tools are available on the tool palette of the *Point load*.

Point load	E.	3
	<u>↓</u> ⊙ † † ± • □	
	F [kN] 10.000	
¥.		

Figure: Tool palette of Point load

Definition steps

1. Select the load case from the load case drop-down list from Loads tabmenu, which you would like to add the new point load(s) to.

If you have not defined a load case yet, you can also define and set at the *Default settings* by giving a load case name.



Figure: Load case definition

The load will be displayed on the layer assigned to the selected load case and in the layer's color.

- 2. Choose the load type: Force or OMMoment (FEM-Design module dependent).
- Set the load value according to the current unit (that can be set at *Settings > Units*) in the tool palette or at
 Default settings.
- 4. Define the *direction* of the force or moment axis (FEM-Design module dependent).
- 5. Choose a method for the insertion point definition (*geometry*).
- 6. Place the point load in the model view based on the chosen geometry method (5th step).

Optional steps:

- 7. Modify the load properties (the host load case, load value) with the *Properties* tool of the *Point load* tool palette.
- 8. Modify the load direction with the editing tools (*Editing Loads*).
- 9. Set the display settings of the load (*Load Display Settings*).

Deviation load (Automatic)

Deviation load (as point loads) can be placed automatically in the centre of gravity points of plates located on stories.

Automatic deviation load generation needs *storey-system* in the project!

Definition steps

1. Entering to "Load" mode, the program displays the so-called "main directions" of the current project. These main X' and Y' load directions are used by the automatic deviation and *wind load* definition. By default, the main X and Y load directions are parallel with the *Global* X and Y axis directions. Set the required load directions for the deviation (and wind) loads with the *Change direction* command (*Modify* menu).

The symbol of the main load directions are stored on "*Main directions*" **Object layers**.

2. Start the Deviation load command from Loads tabmenu.

The generated point loads will be inserted in a load cases called "*Deviation X*" and "*Deviation Y*". If there are no same named load cases in the project, the program automatically defines them for the automatic deviation load components.

- 3. Select the story you would like to add deviation load. Choose the required direction X or Y based on the main directions set in the 1st step. Define the required *alpha* parameters according to the current standard. The program calculates the point deviation load value in the selected storey from the dead load act in it, the storey height and the number of vertical members contributing to the horizontal force. Clicking *Generate* creates deviation load in the centre of the gravity of the selected storey slab with the given load direction.
- 4. Repeat the previous step to generate deviation load in the other storey slabs. Click *Close* to exit from the dialog box after finishing the load definitions.

Line Load

Depending on the current FEM-Design module, constant or variable line force/moment can be defined easily by giving the load shape, direction and position. The load properties and definition tools are available on the tool palette of the *Line load*.

Line load	×
	$\begin{array}{c c} & & & \\ \hline \\ \hline$

Figure: Tool palette of Line load

Definition steps

1. Select the load case from the load case drop-down list from *Loads* tabmenu, which you would like to add the new line load(s) to.

If you have not defined a load case yet, you can also define and set at the *Default settings* by giving a load case name.

Line load			
🖳 ? 📑 💽	Line distributed	l force	×
q1 (kN/ q2 (kN/	Load case	q1 [kN/m] 10.000 q2 [kN/m] 10.000 . Live load 02 Intensity meant along action line. (e.g. dead load) Direction is constant along action line. (e.g. snow load, dead load)	OK Cancel

Figure: Load case definition

The load will be displayed on the layer assigned to the selected load case and in the layer's color.

- 2. Choose the load type: \checkmark *Force* or \bigcirc *Moment* (FEM-Design module dependent).
- 3. Set the intensity values in the start (*q*1) and end point (*q*2) of the new load according to the current unit (that can be set at *Settings* > *Units*) in the tool palette or at \bigcirc *Default settings*. If you inactivate the arrow () next to the *q*1 field, you can type intensity value in *q*2 different from *q*1.

- 4. Define the *direction* of the force or moment axis (FEM-Design module dependent).
- 5. Choose a method for the action line definition (*geometry*).
- 6. Set the "intensity mode", that means the connection between applied intensity and the action line at
 - *Default settings,* if the new load will be in skew position (not vertical or horizontal):
 - **"Intensity meant along action line"** The intensity will be meant along the action line, so the resultant will be calculated from the intensity and the length of the action line. Apply this option for dead load-type loads.
 - "Intensity meant perpendicular to direction of load"
 - The intensity will be meant perpendicular to the load direction, so the resultant will be calculated from the intensity and the length of the horizontal projection of the action line. Apply this option for example to model snow load.



Figure: Intensity difference at skew bars

7. Set the "direction mode", that means the connection between applied load direction and the action

line at — *Default settings,* if the new load will be a curved line load:

"Direction is constant along action line"

The load direction set in the 4th step will be constant along the action line.

- "Direction varies along action line"

The load direction will vary along the action line, so the characteristic direction set in the 4th step will be taken into consideration in the middle point of the curved action line.



Figure: Intensity difference at skew bars

8. Place the line load in the model view based on the chosen geometry method (5th step).

With the special tool called *Object's local system* (\checkmark) you can add line load directly to a selected bar element in a direction set parallel with the one of the local system axis of the bar element. So, with this tool, you can merge 4th, 5th and 8th steps to one step, if the required load direction is equal to the direction of the assigned bar element's local axis.

Optional steps:

- 9. Modify the load properties (the host load case, intensity values, "intensity" and "direction mode") with the *Properties* tool of the *Line load* tool palette.
- 10. Modify the load direction with the editing tools (*Editing Loads*).
- 11. Set the display settings of the load (*Load Display Settings*).

Wind Load (Automatic) – Regular building

Two types of wind load can be generate in FEM-Design, *Regular building* and *Generic building* wind load.



Regular building wind load generation needs *storey-system* in the project!

Definition steps

1. Entering to "Load" mode, the program displays the so-called "main directions" of the current project. These main X' and Y' load directions are used by the automatic wind and *deviation load* definition. By default, the main X and Y load directions are parallel with the *Global* X and Y axis directions. Set the required load directions for the wind (and deviation) loads with the *Change direction* command (*Modify* menu).

The symbol of the main load directions are stored on *"Main directions"* **Object layers**.

2. Start the *Wind load* command from *Loads* tabmenu.

The generated wind line loads will be inserted in a load cases called "*Wind load X*" and "*Wind load Y*". If there are no same named load cases in the project, the program automatically defines them for the automatic wind load components.

- 3. Set the *wind speed* and *terrain type* in the settings dialog according to the building environment. The program calculates the applied wind load values by the levels of the current project's storey-system from the given parameters and the model size set at the *Storey* command (horizontal building size and story level heights). The calculated intensity values are displayed in table format (right part of the dialog).
- 4. Set the wind load X and Y directions same (+) or converse (-) with/to the *main X and Y direction system*.
- 5. Clicking *OK* generates line loads in the external edges of plates that are located on stories surfaces/region parts.

Surface Load

Depending on the current FEM-Design module, constant or variable surface load (force) can be defined easily by giving the load shape, direction and position. The load properties and definition tools are available on the tool palette of the *Surface load*.

Surface load		×
🛃 ? 😐 ,	1 1 1 ± t □ ○ □ □ 	
q1	[kN/m2] 10.000 1 🕨 1 🖉 🗸	
q2	[kN/m2] 10.000	
ga ga	[kN/m2] 10.000 →+ -←	

Figure: Tool palette of Surface load

Constant Surface Load



Figure: Constant surface load on a slab and on a wall

Definition steps

1. Select the load case from the load case drop-down list from *Loads* tabmenu, which you would like to add the new surface load(s) to.

If you have not defined a load case yet, you can also define and set at the *Default settings* by giving a load case name.



Figure: Load case definition

The load will be displayed on the layer assigned to the selected load case and in the layer's color.

- 2. Set the intensity value in the *q1* field according to the current unit (that can be set at *Settings > Units*) in the tool palette or at *Default settings*. You may also choose intensity from the *predefined load values* from the list appears after clicking on the *b* button.
- 3. Define the load *direction* (FEM-Design module dependent).
- 4. Choose a method for the action surface definition (*geometry*).

- 5. In the 3D modules only, you can set the "intensity mode", that means the connection between applied intensity and the action plane at *Default settings*, if the new load will be in skew position (not vertical or horizontal):
 - "Intensity meant along action plane"

The intensity will be meant along the action plane, so the resultant will be calculated from the intensity and the area of the action surface. Apply this option for dead load-type loads.

- "Intensity meant perpendicular to direction of load"

The intensity will be meant perpendicular to the load direction, so the resultant will be calculated from the intensity and the area of the horizontal projection of the action surface. Apply this option for example to model snow load.



Figure: Intensity difference at skew bars

6. Place the surface load in the model view based on the chosen geometry method (4th step).

With the special tool called *Object's local system* (\checkmark) you can add surface load directly to a selected planar object in a direction set parallel with the one of the local system axis of the planar object. So, with this tool, you can merge 3rd, 4th and 6th steps to one step, if the required load direction is equal to the direction of the assigned planar object's local axis.

Variable Surface Load

Linearly variable surface loads can be defined with *Surface load* command. Three intensity values define the linearly variable load as a "skew plane".



Figure: Variable surface loads

Definition steps

- 1. Define first a constant surface load with the required direction and geometry as written at *Constant Surface Load*.
- 2. Use the *Variable intensity* tool of the *Surface load* command.
- 3. Set the new intensity values (q1, q2 and q3). If you inactivate the arrows () next to the value fields, you can type different values by the q fields.
- 4. Select the constant surface load that you would like to modify being variable.
- 5. Give the position of the intensity values. You may click outside the surface load's action plane too.

Optional steps for constant/variable surface loads

- 1. Place a hole/opening in a surface load. Use the Place tool of the *Surface load*.
- 2. Modify the load properties (the host load case, intensity values, "intensity mode") with the *Properties* tool of the *Surface load* tool palette.
- 3. Modify the load direction with the editing tools (*Editing Loads*).
- 4. Information (the resultant force value and the position of its action point) about a selected surface load can be inquired with the *Info* tool. A drawing point can be placed in the resultant's action point, if needed.

Surface load info	$\overline{\mathbf{X}}$
Resultant [kN] 1006.2	Close
Position of resultant	
×[m] 10.667	
Y [m] 5.0000	
Z [m] 4.6667	Insert point

Figure: The resultant force value and position calculated by Info tool

5. Set the display settings of the load (*Load Display Settings*).

Wind load (Automatic) - Generic building

Generic building's wind load (as surface loads) can be placed automatically on covers.

1. <u>Set the type</u> of the cover according to EC: External wall , Flat roof , Lean-to or Ridge-roof

	_	
Default settings		

Figure: Generic building wind load dialog

In all four cases a **green arrow plays a crucial role in correct load generation**. In Lean-to and Ridge-roof the arrows are generated automatically by FD, but for flat roof and external wall, the user has to define it. Details can be found in the sections below.

The importance of the green arrows can be understood at the bottom-left figure in the four settings windows of the four cover type, showing the wind direction terminolgy of EC. In the standard, if the deviation of wind speed is in the range of +45° and -45°, it is taken as 0°. In the range of 45°-135°, it's taken to be 90°, between 135°-225°, to be 180°, and in the last interval to be 270°. By this assumption, no other direction can be imagined as the four cases in the window below, which shows the Default Settings (the button shown in the figure above) for external wall first, and the others next.

External wall

After having chosen, this is what you get in Default settings:



Figure: External wall dialog

The figure at bottom-left shows the direction terminology. For 0° , you can scroll down the typical cases mentioned by EC:



Figure: External wall - setting multiplication factor for predefined wall positions

By choosing the one valid for the current cover, the multiplication factor will be adjusted to the right value. For more sophisticated cases you have the possibility to define the multiplication factor manually.

If the wind direction is in the plane of the wall, some more highly loaded zones (A and B, see EC 1-4 for details) are applied by EC at the leading edge of the wall for taking the effect of turbulence into account. Followed by zone C, a less loaded domain. If no turbulence can be imagined at the leading edge, you can cancel out the creation of zone A and B (The legend in bottom left helps in understanding the directions.):

90°	270°	
 Turbulence at front edge (A, B zones are applied) 	 Turbulence at front edge (A, B zones are applied) 	
Mult. factor 1.00	Mult. factor 1.00	
↓ 180° ↓ 180°		
270° 90°		
to. to.		

Figure: Wind load direction terminology

EC allows the division of the load domain into horizontal stripes, so you may take into account the height dependence of wind velocity by the application of several horizontal stripes:

Load strips
n i

Figure: Wind load vertical distribution

In case of external wall, you have to select the cover by clicking on it, and than select a point at the external side of the building. The green label, and also the arrow will appear, but the arrow will be perpendicular to the plane of the cover. If you have a composed building like in the case below, you can unify several wall covers by pushing Ctrl, and selecting the members:



Figure: Selecting External wall

If you have selected one element, you can immediately define the green arrow by picking an external point, in case of more elements you have to end the selection by 'Enter', and then point an external point outside the walls:



Figure: Defining wall's external point

The wind load generated according to EC will consider the unified green rectangle as one surface.

fter classified all the walls, verify all arrows' pointing out from the building in order to obtain well directed wind loads on the walls.

Flat roof

In case of Flat roof , the following appears:



In the bottom left, the figure shows the **notation of the directions** with respect to the green arrow. The user can switch on or off the creation of the turbulence zones like in the case of *External wall*, and also can change the multiplication factors manually.

If an attic wall is at the edge of the cover, the height is required, as it has influence on the values of the load.

After choosing the settings, you have to click on the cover, and then, you have to give the main direction for wind generation:



Figure: Defining the main direction for wind generation

After the definition of the direction a green label will appear with an arrow in the middle of the cover. If possible, you should choose a main direction of the building that coincides with axis X, for obtaining the correct distribution in the different wind load cases.

ightarrow An incorrectly defined direction can lead to false wind load distribution.

You can modify its direction in Modify>Change direction. (If the label or the arrow is missing after creation, try another graphic engine in Settings>All>Environment>General>Graphic Engine)

Lean-to

In case of Lean-to

to , the dialog window is as follows:

Lean-to		
0°	180°	ОК
¥1±1 ×		Cancel
✓ Turbulence at front edge (F, G, J zones are applied)	▼ Turbulence at front edge (F, G, J zones are applied)	
Suction factor 1.00	Mult. factor 1.00	
Pressure factor 1.00		
90°	270°	
✓ Turbulence at front edge (F, G, J zones are applied)	✓ Turbulence at front edge (F, G, J zones are applied)	
Mult. factor 1.00	Mult. factor 1.00	
1180° 1180°		
270°		
† 0° † 0°		

Figure: Lean-to dialog

Likely to former cases, the notation of EC appears at bottom-left corner. If you have a cover that is not parallel with the load coordinate-system, the wind load directions will fit the cover's direction. The different factors can be set eiter by scrolling down the symbol:



Figure: Lean-to - setting multiplication factor for predefined wall positions

or, by setting them manually.

If required, turbulence zones can be switched off:

90°	270°		
✓ Turbulence at front edge (F, G, J zones are applied)	✓ Turbulence at front edge (F, G, J zones are applied)		
Mult. factor 1.00	Mult. factor 1.00		
Figure: Turbulence zone application			

Ridge roof

The pop up window in case of Ridge roof:

Ridge roof 🛛 🔀		
0° Turbulence at front edge (F, G, J zones are applied) Mult. factor	180° Turbulence at front edge (F, G, J zones are applied) Mult. factor	OK Cancel
90°	270°	
Turbulence at front edge (F, G, J zones are applied) Mult. factor	Turbulence at front edge (F, G, J zones are applied) Mult. factor	
↓180° ↓180° 270° ↓180° 0° ↓0°		

Figure: Ridge roof dialog

Likely to former cases, the notation of EC appears at bottom-left corner. If you have a cover that is not parallel with the load coordinate-system, the wind load directions will fit the cover's direction. The different factors can be set eiter by srcolling down the symbol:

Ridge roof		
	180°	OK
2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2 2		
ge iied)		
anc		

Figure: Ridge roof - setting multiplication factor for predefined wall positions

or, by setting them manually.

If required, turbulence zones can be switched off.

90°	270°
 Turbulence at front edge (F, G, J zones are applied) 	✓ Turbulence at front edge (F, G, J zones are applied)
Mult. factor 1.00	Mult. factor 1.00

Figure: Turbulence zone application

2. *Generate* the wind load

When clicking on the symbol *the wind generation tool window appears:*

W	/ind loa	ad	×
			e
	Wind s	peed [m/s] 26.0	ОК
	Buildin	g height [m] 4.00	
	Terrair	n type:	
	O 0.	Sea or coastal area exposed to the open sea	
	○ I.	Lakes or flat and horizontal area with negligible vegetation and withou obstacles	
	⊙ II.	Area with low vegetation such as grass and isolated obstacles (trees, buildings) with separations of at least 20 obstacle heights	
	 III Area with regular cover of vegetation or buildings or with isolated obstacles with separations of maximum 20 obstacle heights (such as villages, suburban terrian, permament forest) 		
	OIV.	Area in which at least 15 % of the surface is covered with buildings and their avarage height exceeds 15 m	

Figure: Wind load generation dialog

In this window you have to classify the building according to EC, and give the basic wind velocity for load generation.



After clicking OK, the wind loads are generated in 16 different load cases:

Figure: Automatically generated wind load (WL,Y-,1)

(For proportional displaying of loads click: Settings>All>Display>Load>Display proportionally) In the example above, WL Y-, 1. is active which corresponds to a wind marked by the orange arrow.

It is parallel with the loaded wall, and as it is in the range of 45°-135°, according to the load's coordinate system, it is stored in WL, Y-, 1. For each (four) directions, generally 4 different cases can be imagined in EC (because of variation of suction and pressure), but they not necessarily vary: in the case above, all four versions of WL, Y-, are the same for the loaded external wall.

According to EC, protuding parts of roofs should be loaded at underside with the pressure applied on the wall below. This is not taken into account by the wind loading module of FD 11.0.

If the building have a frequently used shape, the program generates the different load domains and the values will be applied on them automatically.

Snow Load (Automatic)

Snow load (as surface loads) can be placed automatically on the open, external plates and plate parts.

Definition steps

1. Start the \bigtriangleup Snow load command from *Loads* tabmenu.

The generated snow surface loads will be inserted in a load case called "*Snow load*". If there is no "Snow load"-named load case in the project, the program automatically defines it for the automatic snow load.

- 2. Set the characteristic snow load value in the settings dialog according to the current code rules. The program calculates the applied snow load value (intensity) from the characteristic value and the built-in form, exponation and thermal factors of the Eurocode.
- 3. Clicking *OK* generates surface loads on the open plate surfaces/region parts.

Thermal Load

Depending on the current FEM-Design module, uniform or non-uniform temperature variation can be added to the structural elements as loads.

FEM-Design Module	Temperature Variation	Load Command
	Non-uniform temperature in Beams	Line temperature variation
	Non-uniform temperature in Plates	Surface temperature variation
	Uniform temperature in Walls	Surface temperature variation
	Uniform/Non-uniform temperature in bar elements	Line temperature variation
	Uniform/Non-uniform temperature in planar elements	Surface temperature variation
1 A	Uniform/Non-uniform temperature in bar elements	Line temperature variation

Table: Temperature load types by FEM-Design modules

Line Temperature variation

- Non-Uniform Temperature Variation in Beams

In Plate module, non-uniform temperature variation can be defined along beam action line or a part of it. The absolute value of the temperature has to be the same in the top and bottom cords: $|dt_{top}| = |dt_{bottom}|$

So the meaning of the temperature variation is the following: $|dt| = \pm |dt| dt$

 $|dt| = \pm |dt_{top} - dt_{bottom}| = \pm 2 \times |dt_{top}|$



The temperature variation can be linearly variable along the action line, so it can be different in the start and end point of the action line.

- Uniform/Non-Uniform Temperature Variation in Bar Elements

In 3D *Structure* and 2D *Frame* modules, uniform or non-uniform temperature variation can be defined along beam and column action lines or a part of them. The temperature values can be different in the top (*t*) and bottom (*t'*) cords.



The temperature variation can be linearly variable along the action line, so it can be different in the start and end point of the action line.

Depending on the current FEM-Design module, temperature load can be easily added to bar elements. The load properties and definition tools are available on the tool palette of the *Line temperature variation load*.

Line temperature variation lo	Line temperature variation load
dt1 ["C] . 20.000] dt2 ["C] . 20.000	r1 [rc]. 10.000 12 [rc]. 10.000 r1 [rc]. 10.000 12 [rc]. 10.000

Figure: Tool palettes of Line temperature variation load

Definition steps

1. Select the load case from the load case drop-down list from Loads tabmenu, which you would like to add the new load(s) to.

If you have not defined a load case yet, you can also define and set at the *Default settings* by giving a load case name.

The load will be displayed on the layer assigned to the selected load case and in the layer's color.

2. Set the temperature values at the start and end point of the "load" action line to the current temperature unit (that can be set at *Settings > Units*) in the tool palette or at *Default settings*. If you inactivate the arrow () next to the temperature fields, you can type different temperature values in the "1" and "2" fields.



Figure: Settings dialogs (the meaning of the "t" values)

- 3. Define the *direction* of the plane you would like to place the thermal load (only in 3D modules).
- 4. Add the thermal to its belonging bar element by selecting it with .

Or, choose a method for the action line definition (*geometry*) and define the thermal load. Use this method, if you would like to place the load on a part of the bar element.

With the special tool called *Object's local system* (\checkmark , only in 3D modules) you can set the load's plane direction directly to a selected axis of the bar element's *local system* in the tool palette and by the clicking on the bar element. So, with this tool, you can merge 3rd and 4th steps to one step, if the direction of the thermal load plane is equal to a required axis direction of the assigned bar element's local system.



Figure: Different ways of thermal load definition

Optional steps:

- 5. Modify the load properties (the host load case and temperature values with the *Properties* tool of the *Line temperature variation load* tool palette.
- 6. Modify the direction of the thermal load plane with the editing tools (*Editing Loads*).
- 7. Set the display settings of the thermal load (*Load Display Settings*).

Surface Temperature variation

- Non-Uniform Temperature Variation in Plates

In *Plate* module, non-uniform temperature variation can be defined in plate region or a part of it. The absolute value of the temperature has to be the same in the top and bottom cords: $|dt_{top}| = |dt_{bottom}|$

So the meaning of the temperature variation is the following: $|dt| = \pm |dt_{top}-dt_{bottom}| = \pm 2 x |dt_{top}|$



The temperature variation can be linearly variable along the action plane. Temperature value in 3 region points defines the linearly variable temperature in a plate object.

Uniform Temperature Variation in Walls

In *Wall* module and *Plane Strain* modules, uniform temperature load can be defined in wall region or a part of it:

 $t_{si} = t_{top} = t_{bottom}$



The temperature variation can be linearly variable along the action plane. Temperature value in 3 region points defines the linearly variable temperature in a wall object.

Uniform/Non-Uniform Temperature Variation in Planar Elements

In 🚰 3D Structure module, uniform or non-uniform temperature variation can be defined in plate, wall and shell regions or a part of them. The temperature values can be different in the top (t') and bottom (*t*) cords.

t3 +2	6"	Non-unifo	rm temperature
		t3 t2	
ti [C] 10.000	t1'[C] 10.000	[
12 [C] 10.000		t1 [C] 10.000	t1'[C]20 1
10 000		t2 [C] 10.000 🛛	t2'[C]20 🛛 🕽
		t3 [C] 10.000	t3' [C]20
Uniform to	emperature		

The temperature variation can be linearly variable along the action plane. Temperature value in 3 region points defines the linearly variable temperature in a plate, wall or shell object.

Depending on the current FEM-Design module, temperature load can be easily added to planar objects. The load properties and definition tools are available on the tool palette of the *Surface support motion load*.



Figure: Tool palettes of Surface temperature variation load

Definition steps

1. Select the load case from the load case drop-down list from *Loads* tabmenu, which you would like to add the new load(s) to.

If you have not defined a load case yet, you can also define and set at the Default settings by giving a load case name.

The load will be displayed on the layer assigned to the selected load case and in the layer's color.

2. First, define temperature variation constant along the load action plane. Set the motion value in the "t1" fields according to the current temperature unit (that can be set at *Settings > Units*) in the tool palette or at Default settings.

The local *z'* axis of the surface element (plate, wall) points to the "top" surface temperature (t') in 3D Structure module.

Surface temperature variation le	oad 📌 📃 🔯	Surface temperatu	re variation load 🛛 🛛 🔀
	OK Cancel	t3 t2	
dt1 [C] 20.000	Surface temperature variation l	t1 [C] 10.000	ם t1'[C] 10.000 ב
dt2 [C] 20.000	2	t2 [C] 10.000	₽ ¹ 2'[C] 10.000 ₽
	ts3 ts2	t3 [C] 10.000	t3'[C] 10.000
Load case		Load case	~
	ts1 [C] 10.000 1		
	ts2 [C] 10.000 1	14	
	ts3 [C] 10.000	E	
	Load case		

Figure: Settings dialogs (the meaning of the "t" values)

3. Add the thermal load to its belonging surface element by selecting it with \Box .

Or, choose a method for the action surface definition (*geometry*) and define the surface thermal load. Use this method, if you would like to place the load on a part of the surface element. In this case set the *working plane* into the element's plane.



Figure: Different ways of surface thermal load definition

Optional steps:

4. Use the \checkmark *Variable intensity* tool, if you would like to define variable surface thermal load. Set the new temperature values (dt1/t1/ts1, dt2/t2/ts2 and dt3/t3/ts3). If you inactivate the arrows () next to the value fields, you can type different values by the *t* fields. Select the constant surface

temperature load that you would like to modify being variable. Give the position of the three temperature values. You may click outside the surface load's action plane too.

- 5. Place a hole/opening in a surface thermal load. Use the *Hole* tool of the *Surface temperature variation load*.
- 6. Modify the load properties (the host load case and temperature values with the *Properties* tool of the *Surface support motion load* tool palette.
- 7. Set the display settings of the thermal load (*Load Display Settings*).

Initial Internal Load

Post-stress can be added to bars (*Initial internal line load*) and structural regions (*Initial internal surface load*) as load. Constant moment and normal force values in a given direction (e.g. only in one direction) may be defined by points of line and planar structural objects.

The initial internal load commands are useful for defining post-tensioning loads in concrete bars and slabs. Post-stress is also good for steel bars at welded connections. The definition steps are the same as introduced at *Line* or *Surface temperature variation load* commands.



Figure: Initial internal force (compression) in a stiffener truss member

Support Motion

Predefined displacement (motion and/or rotation depending on the current FEM-Design module) can be assigned to point, line and surface supports.

Only the same type support motion can be assigned to a support.

Support motion load type	Support type
Point support motion load	Point support

Line support motion load	1 Line support
I Surface support motion load	I Surface support

Table: Support motion load types and their proper support types

If a support motion load is not belonging to a support, the program sends you a warning message during calculation, that that load will be ignored in the calculations.

In *Plate module* the *Column* and *Wall* elements are supports, so *Point support motion load* can be assigned to *Columns* and *Line support motion load* to *Walls*.

Point Support Motion

Depending on the current FEM-Design module, predefined point motion/rotation can be easily added to point supports by giving displacement direction and position. The load properties and definition tools are available on the tool palette of the *Point support motion load*.



Figure: Tool palette of Point support motion load

Definition steps

1. Select the load case from the load case drop-down list from *Loads* tabmenu, which you would like to add the new load(s) to.

If you have not defined a load case yet, you can also define and set at the *Default settings* by giving a load case name.

The load will be displayed on the layer assigned to the selected load case and in the layer's color.

- 2. Choose the motion type: \downarrow *Motion* or \bigcirc *Rotation* (FEM-Design module dependent).
- 3. Set the displacement values in the *e* or *phi* of the new load according to the current unit (that can be set at *Settings* > *Units*) in the tool palette or at *Default settings*.
- 4. Define the *direction* of the motion or the rotation axis (FEM-Design module dependent).
- 5. Add the point motion to its belonging point support by clicking the support's insertion point ($\stackrel{\times}{}$) or by selecting the point support ($\stackrel{\frown}{}$).

Optional steps:

- 6. Modify the load properties (the host load case and displacement values with the *Properties* tool of the *Point support motion load* tool palette.
- 7. Modify the displacement direction with the editing tools (*Editing Loads*).
- 8. Set the display settings of the load (*Load Display Settings*).

Line Support Motion

Depending on the current FEM-Design module, predefined motion/rotation can be easily added to line supports by giving displacement direction and position. The load properties and definition tools are available on the tool palette of the *Line support motion load*.

Line support motion load	×
] ? ! 💿 🕇 tl ± t∠ 🖉 🗘 Q 🔲 🗌	
e1 (m) 0.010000 e2 (m) 0.010000 =	

Figure: Tool palette of Line support motion load

Definition steps

1. Select the load case from the load case drop-down list from *Loads* tabmenu, which you would like to add the new load(s) to.

If you have not defined a load case yet, you can also define and set at the *Default settings* by giving a load case name.

The load will be displayed on the layer assigned to the selected load case and in the layer's color.

- 2. Choose the motion type: \checkmark *Motion* or PROTATION *Rotation* (FEM-Design module dependent).
- 3. Set the displacement values at the start and end point of the "load" action line: e1 and e2 or phi1 and

phi2 according to the current unit (that can be set at *Settings* > *Units*) in the tool palette or at \square *Default settings*. If you inactivate the arrow (\square) next to the *e1* or *phi1* field, you can type intensity value in *e2* or *phi2* different from *e1* or *phi1*.

- 4. Define the *direction* of the motion or the rotation axis (FEM-Design module dependent).
- 5. Set the "direction mode", that means the connection between applied load direction and the action

line at — *Default settings,* if the new load will be a curved line load:

"Direction is constant along action line"

The load direction set in the 4th step will be constant along the action line.

- "Direction varies along action line"

The load direction will vary along the action line, so the characteristic direction set in the 4th step will be taken into consideration in the middle point of the curved action line.

6. Add the motion to its belonging line support by selecting the line support (

Or, choose a method for the action line definition (*geometry*) and define the line load. Use this method, if you would like to place the load on a part of the support's action line.

With the special tool called *Object's local system* () you can add motion load directly to a selected component of a support/more supports just by setting the required component direction (an axis of the support's local coordination system) in the tool palette and by the clicking on the support(s). So, with this tool, you can merge 4th and 6th steps to one step, if the required displacement direction is equal to the direction of the assigned support component.



Figure: Different ways of line support motion load definition

Optional steps:

- 7. Modify the load properties (the host load case and displacement values with the *Properties* tool of the *Point support motion load* tool palette.
- 8. Modify the displacement direction with the editing tools (*Editing Loads*).
- 9. Set the display settings of the load (*Load Display Settings*).

Surface Support Motion

Depending on the current FEM-Design module, predefined motion can easily be added to surface support by giving displacement direction and position. The load properties and definition tools are available on the tool palette of the **Surface support motion load**.

Surface support motion load	
	e1 [m] 0.010000 1 12
1	e2[m]
	e3[m]0.010000

Figure: Tool palette of Surface support motion load

Definition steps

1. Select the load case from the load case drop-down list from *Loads* tabmenu, which you would like to add the new load(s) to.

If you have not defined a load case yet, you can also define and set at the *Default settings* by giving a load case name.

The load will be displayed on the layer assigned to the selected load case and in the layer's color.

2. First, define a constant predefined displacement. Set the motion value in the *e1* field according to the current unit (that can be set at *Settings* > *Units*) in the tool palette or at *Default settings*.
- 3. Define the *direction* of the motion (FEM-Design module dependent).
- 4. Add the motion to its belonging surface support by selecting the surface support (\Box).

Or, choose a method for the action surface definition (*geometry*) and define the surface load. Use this method, if you would like to place the load on a part of the support's action surface.

With the special tool called *Object's local system* (\checkmark) you can add motion load directly to a selected component of a support/more supports just by setting the required component direction (an axis of the support's local coordination system) in the tool palette and by the clicking on the support(s). So, with this tool, you can merge 3rd and 4th steps to one step, if the required displacement direction is equal to the direction of the assigned support component.



Figure: Different ways of surface support motion load definition

Optional steps:

5. Use the Variable intensity tool, if you would like to define variable surface motion assign to a surface support or a part of it. Set the new displacement values (*e1*, *e2* and *e3*). If you inactivate the arrows () next to the value fields, you can type different values by the *e* fields. Select the constant surface load that you would like to modify being variable. Give the position of the three motion values. You may click outside the surface load's action plane too.



Figure: Variable surface support motion

- 6. Place a hole/opening in a surface load. Use the Place tool of the *Surface support motion load*.
- 7. Modify the load properties (the host load case and displacement values with the *Properties* tool of the *Surface support motion load* tool palette.
- 8. Modify the displacement direction with the editing tools (*Editing Loads*).
- 9. Set the display settings of the load (*Load Display Settings*).

Mass

For dynamic, vibration shape and seismic effect calculations (especially for frame structures), masses can be defined optionally as point loads or automatic conversion of load cases to mass.

Masses are not participants of *load cases, load combinations* and *load groups*.

Concentrated Mass

Definition steps

- 1. Start the Mass command from Loads tabmenu.
- 2. Set the mass value according to the current unit (that can be set at *Settings > Units*).
- 3. Place the mass just by clicking in the required point. The mass will be displayed on the *Masses* object layer assigned in green color by default.

Optional steps:

- 4. Modify the mass value with the *Properties* tool of the *Mass* tool palette.
- 5. Set the display settings of masses (*Load Display Settings*).



Figure: Definition of concentrated mass

Load Case-Mass Conversion

With the *Load case-mass conversion* command of *Loads* tabmenu, you may convert load cases to masses by assigning factor values to them. All conversions are done just before dynamic calculation.

Typing 1 into the *Factor* field means that the loads (of the load case) with their original value will be converted to mass. Typing 0 value removes predefined conversion of a load case (as a result you will get the "--" symbol again).

Factor	Name	~	ОК
1.000	Auto dead load		
1.000	Dead load of roof		Cancel
[Live load 01		
	Live load 02		
	Snow		
	WindX		
	WindY		

Figure: Mass definition by conversion of loads (load cases)

The following point, line and surface force types can be converted to masses:

Plate module: force with global -Z direction: mz = -Fz/g

Wall and *Plain Strain modules*: force with global -Y direction: mx = my = -Fy/g

3D Frame and 3D Structure modules: force with global -Z direction: mx = my = mz = -Fz/g

Moments, support motions, thermal loads cannot be converted to masses.

Seismic Load

Seismic loads are taken into account according to the Response Spectrum Analysis method of Eurocode 8.

Structure type, some parameters and response spectra have to be defined with *Seismic load* 4 command.

In order to define seismic load in Seismic load dialog the user has to select the structure type under Structure information tab / Structure type:

- Eurocode: Building structure and Non-building structure.

Eurocode

- Turkish seismic code: New building, Existing building, Non-building structure.



Figure: Select structure type in Seismic load dialog panel

If **Building structure** is chosen, storeys have to be defined in order to get Storey seismic sensitivity (P-delta (θ)) coefficient and Storey shear forces.

If **Non-building structure** is chosen, these results are not available.

Further parameters can be set in Seismic load dialog panel:

- *ksi* is the viscous damping ratio (generally 5%), which is used in *Modal analysis* when the summation of the effect of the same direction vibration shapes is carried out by the CQC (Complete Quadratic Combination).
- *qd* is the displacement behavior factor, assumed equal to *q* unless otherwise specified.
- *Beta* is the lower bound factor for the horizontal design spectrum. In *Modal analysis* the displacements and stresses are multiplied with $\frac{\beta \cdot V_{stat}}{V_{modal}}$ if the following expression is false: $\frac{V_{modal}}{V_{stat}} < \beta$.

Required spectrums can be defined either by using standard spectra (automatic) or by manual definition (unique).



The applied/defined spectra and its properties can be added to documentation as figures and tables by clicking on *Add to documentation*.

Automatic Design Spectrum Definition ("Standard")

Clicking on *Standard* generates the horizontal and vertical spectra based on settings and parameters required by Eurocode 8.



The horizontal spectrum is always necessary. The vertical spectrum is necessary when the vertical affect taken into account.

- Horizontal spectra

Seismic code: Turkish



Figure: Parameters and automatic generation of horizontal spectra

Data of horizontal design spectra (based on Eurocode 8):

- *Type* (of spectra: 1 and 2) and *Ground* type (A, B, C, D and E) specify the values of *S*, *TB*, *TC* and *TD*;
- *ag* is the design ground acceleration;
- *S* is the soil factor;
- *q* is the behavior factor that depends on material and type of the structure;
- *beta* is the lower bound factor for the horizontal design spectrum.
- Vertical spectra

Seismic load Horizontal spectra Vertical spectra Type 1 v agv/ag 0.900 S S 1.00 TB TC [s] 0.150 TD TD [s] 1.00 q deta 0.200 D	Sd [m/s2] 0.050, 1.125 0.150, 1.125 0.000, 0.300	1.000, 0.169
Standard Unique 4	add to documentation	OK Cancel

Figure: Parameters and automatic generation of vertical spectra

The vertical design spectrum is derived from the horizontal spectrum using the *agv* (vertical design ground acceleration)/ ag multiplicator based on Eurocode 8.

Custom Design Spectrum Definition ("Unique")

User can define custom horizontal and vertical spectra in table format or by editing spectra graphically. Switch to *Unique* mode, and then add *T* and *Sd* values to the table cells or place points in the Sd-T(s) curve by clicking in the required points of the figure.



Figure: Custom spectra definition

Unique spectra can be stored in files and exported with Save. Spectra can be imported with Load.

Import/Export of load cases, load groups and load combinations

Load cases, Load groups and Load combinations can be imported from or exported to native FEM-Design or text format.



Figure: Import/Export of load cases, load groups and load combinations

-	Export		
FEM-Design specific file	FEM-Design	Clipboard	Another program
.fdlcase .fdlcomb .fdlgroup	Load I I I I I I I I I I I I I I I I I I I	Paste	Ctrl + V Ctrl + C etc.
	Import -		

Figure: Importing from and exporting to another programs

- Save as default

FD saves the load cases / groups / combinations and they will be loaded at every new model.

- Save

Load cases can be saved to a specific .ldcase / .ldgroup / .ldcomb type file.

- Load

The user has the possibility to load previously saved .ldcase / .ldgroup / .ldcomb file.

- Copy

Load cases / groups / combinations can be copied to Clipboard with Copy command to use them in another program (e.g. MS Excel).

- Paste

Load cases / groups / combinations can be imported through Clipboard with Paste command.

Predefined Load Values

You can browse from predefined intensity values when clicking on the \blacktriangleright button of the Surface load command's tool palette. Just select a value from the drop-down list and it will be added in the proper *q* field.

Surface load				
🖳 ? 🗖 🖬 🖬 🏌 tl 土 tz				
q1 [kN/m2] 10.000 ↓ ▶	Ref.	Description	Value [kN/m2]	
q3 [kN/m2] 10.000	A.1	Areas for domestic and residential activities (Floors)	2.0	
	A.2	Areas for domestic and residential activities (Stairs)	2.0	
	A.3	Areas for domestic and residential activities (Balconies)	2.5	~
		ОК	Cancel	

Figure: Predefined intensity value added to q field (Surface load)

Load Display Settings

The display properties of the loads can be set at the *Settings* > *All*... > *Display* > *Load*.

The available options depend on the current FEM-Design module.

Settings		X
Settings Environment Drawing FEM Solid model Display Converting elements Support Support Connections Mass Local Design Window	Load Graphical options Display label Display proportionally Hatch dist. [mm] 20.000 Point load Size/Scale [m] 1.0000 Line load Size/Scale [m] 1.0000	Temperature variation Size/Scale [m] 1.0000 Stress load n-Size/Scale [m] 1.0000 m-Size/Scale [m] 1.0000 Support motion Size/Scale [m] 1.0000
Save as default		OK Cancel

Figure: Settings options affect on the appearance of the loads

- Display label

The load values can be displayed on the screen in *Wireframe display mode*. The default font size and style can be set at *Settings* > *Text* settings.



Figure: Load labels displayed according to Text settings



The position of the load labels can be modified with *Edit* > *Move*.

- Display proportionally

By default, this option is inactive. That means all loads are displayed according the *Size* [m] set by load types. So, for example, point loads having different force values are displayed with same size arrow symbols with the height set by *Size*.

Activating the *Display proportionally* option the loads will be displayed with their values multiplied with the *Scale* value set by load types. So, if the *Scale* value is 1.0 for all load types, the loads are

displayed with their real values converted to meter units. For example, 5kN point force is displayed 5m-height symbol in case of 1.0 *Scale* value; but modifying the Scale value to 2.0 displays the 5kN force with 10m-height symbol.



Figure: Examples for different Size/Scale values and Display proportionally option

Hatch distance

_

Hatch distance sets the density of the hatches of the surface loads' action plane.

Layer, color and pen width

All loads are placed (and grouped) on *Object layers* according to their host load case. Color and pen width are assigned to each load case. After defining a load case the program automatically set red color for it by default. The default color together with the pen width of the load symbol contours can be modified with the *Color* and *Pen* tools.

By default, the layer of the current load case is active and the others are hidden. Of course, you can activate, hide or protect load case layers as you want.



Editing Loads

Copying Loads

The *Copy load case* command (*Loads* menu) gives an easy way to copy all defined loads of a load case (*Source*) into another load case (*Destination*). With the *Multiplication factor* the values of copied loads can be increased or decreased proportionately.



Figure: Copying loads of a load case to another one

Loads can be copied inside a load case with the *Copy* command (*Modify* menu).

Modifying Load Values

Load properties (such as the load values) can be easily modified with the *Properties* tool of the proper load definition command.

With the *Multiply load* command, you can modify values of selected loads with a given multiple factor.

Modifying Load Directions

Depending on the current FEM-Design module, the direction of predefined loads can be modified with the *Change direction, Mirror* and *Rotate* command of the *Edit* menu.

Modifying Load Positions

The position of Loads can be modified with the *Move* command (*Modify* menu).

Modifying the Geometries of Action Lines and Surfaces

The editing tools valid for region elements can be used to modify the geometry of surface loads' action plane. These *Edit* menu commands are for example *Modify region, Stretch, Curve, Elbow, Scale, Chamfer, Fillet* etc. The *Hole* tool of surface load commands can also be used to cut parts from a surface load.

The editing tools valid for lines and arcs can be used to modify the geometry of line loads' action line. These *Edit* menu commands are for example *Stretch*, *Curve*, *Elbow*, *Split*, *Trim*, *Extend*, *Break* etc.

Combination of Loads/Load cases

Load cases (and so their load contents) can be combined manually with given load factor multipliers (*Load Combination*) or the program finds the most unfavorable combinations of the load cases grouped in different types (permanent, temporary, accidental etc.).

Load Combination

Load combination lets you combine load cases by multiplying them with given load factors.

Four types of load combinations can be defined: load combinations for **ultimate (U)**, **accidental (U**_a), **seismic (U**_s) and for **serviceability (S) limit state**. Although *Analysis* calculations can be done for all types, the recommended functions of the types are the followings:

- Ultimate limit state-type load combination (U)

Define U-type load combinations for strength and stability calculations. All design calculations (except for crack width of RC Design) are done only for U combinations.

- Ultimate limit state-type (accidental) load combination (Ua) This type of load combination is like Ultimate limit state, but the difference is in the safety factors.
- **Ultimate limit state-type (seismic) load combination (Us)** This type of load combination is like Ultimate limit state, but the difference is in the safety factors.
- Serviceability limit state-type load combination (S)

Define S-type load combinations to calculate displacement and to design RC elements for crack width combinations.

1 U.S.X U 1.35 Dead load (+Dead load) Inos Cancel 1.05 Imposed load (Ordnary) 1.00 Snow (Ordnary) Import / Export 1.00 Wind X (Ordnary) 1.00 Wind X (Ordnary) Import / Export 2 Ust.S Us 1.00 Dead load (+Dead load) Import / Export 2 Ust.S Us 1.00 Dead load (+Dead load) Import / Export 3 SLS S 1.00 Dead load (+Dead load) Import / Export 3 SLS S 1.00 Dead load (+Dead load) Import / Export 1 1.00 Sesinic max. Import / Export Import / Export Import / Export 3 SLS S 1.00 Dead load (+Dead load) Import / Export Import / Export 1 1.00 Snow (Ordinary) Import / Export Import / Export Import / Export 1 1.00 Snow (Ordinary) Import / Export Import / Export Import / Export 1 1.00 <th>No</th> <th>Name</th> <th>Type</th> <th>Factor</th> <th>Included load cases</th> <th>*</th> <th>OK</th>	No	Name	Type	Factor	Included load cases	*	OK
1.05 Imposed load (Ordinary) 1.00 Snow (Ordinary) 1.00 Snow (Ordinary) 1.00 Deviation (Ordinary) 1.00 Deviation (Ordinary) 1.00 Deviation (Ordinary) 1.00 Import / Export 1.00 Deviation (Ordinary) 0.70 Wind X 0.70 Wind X 1.00 Sesmic max. 1.00 Snow (Ordinary) 0.70 Wind X 0.70 Wind X (Ordinary) 0.70 Sinow (Ordinary) 0.70 Wind X (Ordinary) 0.70 User 0.70 User 0.70 Wind X (Ordinary) 0.70 User 0.70 User 0.70 User 0.70 User 0.7	1	ULS X	U	1.35	Dead load (+Dead load)		
1.00 Srow (Ordinary) 1.00 Wind X (Ordinary) 1.00 Wind X (Ordinary) 1.00 Deviation (Ordinary) 1.00 Deviation (Ordinary) 0.00 Imposed load (Ordinary) 0.00 Imposed load (Ordinary) 0.00 Sow (Ordinary) 0.00 Sow (Ordinary) 0.00 Sessini max. 3 SLS S 1.00 Sessini max. 1.00 Sow (Ordinary) 0.70 Wind X (Ordinary)				1.05	Imposed load (Ordinary)		Cancel
1.00 Wind X (Ordinary) Import / Export 1.00 Deviation (Ordinary) Import / Export 2 Usl.S Us 1.00 0.20 Snow (Ordinary) 0.20 Snow (Ordinary) 0.20 Snow (Ordinary) 0.20 Simic max. 3 SLS S 1.00 Dead load (+Dead load) 1.00 Snow (Ordinary) 0.20 Snow (Ordinary) 0.70 Wind X Insert Insert 1.00 Snow (Ordinary) 0.20 Oppy 0.70 Wind X (Ordinary) 0.70 Wind X (Ordinary) 0.70 Wind X (Ordinary) Opey Opetete 0.70 Wind X (Ordinary) Opey Delete 0.70 Wind X (Ordinary) Opey Delete 0.70 Wind X (Ordinary) Opey Delete 0.70 Wind X (Ordinary) Insert Insert 0.70 Wind X (Ordinary) Insert Insert 0.70 Wind X (Ordinary) Insert				1.00	Snow (Ordinary)		r
1.00 Deviation (Ordinary) 2 UsLS Us 1.00 Dead load (+Dead load) 1.00 Deviation (Ordinary) 0.20 Snow (Ordinary) 0.20 Snow (Ordinary) 0.70 Wind X 1.00 Dead load (+Dead load) Generate 3 SLS 5 1.00 Dead load (+Dead load) Insert 1.00 Down (Ordinary) 0.70 Wind X (Ordinary) Delete 0.70 Wind X (Ordinary) 0.70 Wind X (Ordinary) Delete 0.70 Wind X (Ordinary) 0.70 Wind X (Ordinary) Delete 0.70 Wind X (Ordinary) 1.00 Deviation (Ordinary) Delete				1.00	Wind X (Ordinary)		Import / Export
2 Us 1.00 Deadload (Verlaar)) 1.00 Imposed load (Ordinary) 0.20 Sown (Ordinary) 0.20 Sown (Ordinary) 0.20 Sown (Ordinary) 0.20 Somn (Ordinary) 0.20 Somn (Ordinary) 0.20 Somn (Ordinary) 0.20 Somn (Ordinary) Insert				1.00	Deviation (Ordinary)		
1.00 Imposed load (Ordinary) 0.20 Snow (Ordinary) 0.70 Wind X 1.00 Seismic max. 3 SLS S 1.00 0.70 Wind X 1.00 Seismic max. 1.00 Snow (Ordinary) 0.70 Wind X (Ordinary) 0.70 Unot deviation (Ordinary) <	2	UsLS	Us	1.00	Dead load (+Dead load)		
0.20 Snow (Ordinary) 0.20 Snow (Ordinary) Load combination 0.70 Wind X 1.00 Semic max. Generate 3 SLS S 1.00 Dead load (+Dead load) Insert 0.70 Wind X (Ordinary) 0.70 Wind X (Ordinary) Delete Delete 0.70 Wind X (Ordinary) 0.70 Wind X (Ordinary) Delete Delete 0.70 Wind X (Ordinary) 0.70 Wind X (Ordinary) Delete Delete 0.70 Wind X (Ordinary) 0.70 Wind X (Ordinary) Delete Delete 0.70 Wind X (Ordinary) 0.70 Wind X (Ordinary) Delete Delete 0.70 Wind X (Ordinary) 0.70 Wind X (Ordinary) Delete Delete 0.70 Wind X (Ordinary) 0.70 Wind X (Ordinary) Delete Delete 0.70 Wind X (Ordinary) 0.70 Wind X (Ordinary) Delete Delete				1.00	Imposed load (Ordinary)		
0.70 Wind X 6 <td< td=""><td></td><td></td><td></td><td>0.20</td><td>Snow (Ordinary)</td><td></td><td>Load combinatio</td></td<>				0.20	Snow (Ordinary)		Load combinatio
1.00 Seismic max. Insert 3 SLS S 1.00 Dead load (Head load) 0 D0 Wind X (Ordinary) Copy 0.70 Wind X (Ordinary) Delete 1.00 Deviation (Ordinary) Delete all 1.00 Load case Insert				0.70	Wind X		Generate
3 SLS S 1.00 Dead load (+Dead load) Insert 1.00 Snow (Ordnary) 0.70 Wind X (Ordnary) Delete 1.00 Deviation (Ordnary) 0.00 Deviation (Ordnary) Delete 1.00 Deviation (Ordnary) 1.00 Deviation (Ordnary) Delete				1.00	Seismic max.	-	Generate
1.00 Snow (Ordinary) Copy 0.70 Wind X (Ordinary) Delete 1.00 Deviation (Ordinary) Delete all	3	SLS	S	1.00	Dead load (+Dead load)	-	Insert
0.70 Wind X (Ordinary) 1.00 Deviation (Ordinary) Delete all Load case Insert New				1.00	Snow (Ordinary)		Copy
1.00 Deviation (Ordinary)				0.70	Wind X (Ordinary)		
Delete al Load case Insert New				1.00	Deviation (Ordinary)		Delete
Load case Insert New							Delete all
Insert							Load case
New							Insert
							New

Figure: Load combination definition

Load combination and Load case definition functions are shown on the right side of the *Load combinations* dialog box.

lo	Name	Type	Factor	Included load cases	^	OK	
1 gravity lc		U	1.00	Dead load		Grand	
			0.30	Live load 1		Cancel	
					- 11	Import / Export >	
					- 11	Import/ Import/	
					10		
						Load combination	
					- 11	Generate	
					Ξ	Insert	Load combina
					-11		Loud comona
						Сору	group
						Delete	
						Delete all	
					-11		
					-11	Load case	
					-11	Tesert	Teedees
					-11	Diserc	Load case
						New	

Figure: Load combination definition functions

Under *Load combination* group, there are five functions:

- Generate

FD can automatically generate Load combinations from the Load cases assigned to the Load groups, according to Eurocode 1990 – Chapter 6.4.3.

1. In *Load combination* dialog click *Generate* button.

1. Select load cases from load groups



Figure: Generate load combinations dialog

- 2. Select *load cases* from the *Load groups*.
- 3. Select *Load combination* type to generate and their properties (if there is any).

The selected *Load combinations* will be generated if there is at least one *load group* of that type. E.g. if the user wants to generate Accidental load combination, but there is no Accidental Load group, or in that Load group there is no Load case, the program will not create any accidental Load combination.

- 4. Set the *General options* parameters.
- Insert

With this function the user can add a new Load combination. After clicking on *Insert* button, the parameters can be set in the dialog.

- Copy

With this function the user can *copy* an existing combination with load cases and factors.

- Delete

The user can *delete* a selected load combination.

- Delete all

All the load cases can be deleted with *Delete all* function.

Under *Load case* group, there are three functions:

- Insert

An existing load case can be inserted to a load group.

- New

The user has the opportunity to define load cases in *Load conbinations* dialog. After defining a load case in this dialog, the new load case will be added to *Load case* list (the user can see it in Load cases dialog).



5. The new load case is defined!

No	Name	Туре	Duration class (EN 1995 1-1)	ОК
1 St	uctural DL	+Struc. dead load	Permanent	Cancel
				Import / Export
				Insert
				Delete
		++		Delete all

Figure: Load case definition in Load combinations dialog

Remove

This function removes the selected load case from the current load group

Definition steps of new load combinations

- 1. Type the name of the new combination in the *Name* cell.
- 2. Set the load combination type by choosing U (ultimate), U_a (accidental), U_s (seismic) or S (serviceability) from the *Type* drop-down list.
- 3. Select a load case that you would like to add to the combination from the *Included load cases* dropdown list, which contains all load cases predefined in the current project.
- 4. Type a load factor for the load case chosen in the previous step in the Factor cell.
- 5. Repeat the 3rd and 4th step in the following rows, if you would like to add more than one load case to the combination.



If you would like to add more than one load case to the current load combination with the same factor in one step, use the *Insert case(s)* option of the *Load combinations* dialog.

6. Define the next load combination by repeating the previous steps.



If you would like to define a load combination with a similar content (load cases with factors) of another one, apply the *Copy comb*. option for the source load combination and define the destination combination by defining a new load combination. *Copy comb*. option can also add a load combination content to another predefined one.

1. Select the combin you want to copy	ation	Factor Included load cases		
2 U.S S	Us	1.05 Imposed load 1.05 Imposed load 1.00 Snow 1.00 Deviation 1.00 Snow 0.20 Snow 0.70 Wind X	Cancel Import / Export > Load combination Generate Insert Insert	
Copy combination Destination load combinets Copied U.S ULS S 3. Type	e the name	of the	Copy Delete Delete all Load case Insert New Remove	2. Click on "Copy" button
Load combinations	CK Cancel	1.35 Dead load 1.05 Imposed load 1.00 Snow 1.00 Wind X 1.00 Deviation 1.00 Deviation	OK Cancel Import / Export >	
3 Copied ULS 4. The Load com	u bination is	1.00 Imposed load 0.20 Snow 0.20 Wind X 1.35 Dead load 1.05 Imposed load 1.00 Snow 1.00 Deviation	Load combination Generate Insert Copy Delete Delete all	
with the load of	cases		Load case Insert New Remove	

Figure: Creation of a new Load combination copy

Optional steps

- 7. Load combination can be renamed by typing a new name in its proper *Name* cell.
- 8. Load combinations can be imported from a pre-defined database and exported for next and later projects by clicking on *Import/Export>* in *Load combinations* dialog box.

Load Group

Load cases can be grouped (Load groups command) by their action type (permanent, temporary, accidental etc.). The program calculates (if required) the critical values of analysis and design results from the most unfavorable combinations of "grouped" load cases.



 Δ Non-linear calculations such as 2nd order analysis, stability and cracked-section analysis cannot be done for load groups.

No	Loa	d group		Included load cases	A	ОК
1	Permanent 01 (Permaner	nt, 1.00, 1.35,	1.00, 1.00, 0	Dead load (+Dead load)		Carral
2	Permanent 02 (Permanent, 1.00, 1.35, 1.00, 1.00, 0			Roof weight		Cancel
3	Wind (Temporary, 1.50,	0.70, 0.50, 0.2	20)	Wind X		
				Wind Y		Import / Export
	Name Snow					Combination methol
	Type Temporary	•	Tempor	ary 🔻		🔘 EC0 6. 10.a, b
	Safety factor	1.50	Perman Stress	ent	н	Load group
	Psi 0	0.700	Tempor Acciden	ary tal		Insert
	Psi 1	0.500	Seismic			Delete
	Psi 2	0.200				Delete all
						Load case
						[Insert
					-	New
						Remove

Figure: Load group definition

Combination method can be chosen in the Load groups dialog box. The results of the analysis are calculated based on one of the two methods of Eurocodes (EC0 6.10 or EC0 6.10.a,b).

Load group and Load case definition functions are shown on the right side of the *Load groups* dialog box.

.oad	groups			×	
No	Load group	Included load cases		ОК	
1	Dead load (Permanent, 1.00, 1.35, 1.00, 1.00)	Soil dead load	-	Grand	
		Structural DL	-	Cancel	
2	Live load (Permanent, 1.00, 1.35, 1.00, 1.00)	Live load 1			
		Live load 2		Import / Export >	
3	Shrinkage (Stress, 1.00, 1.30)	Shrinkage			
				Load group	т 1
				Insert	Load group
			_	Delete	groun
			_	Delete all	group
			_	Delete all	
				Load case	
			_	Insert	Load case
				New	group
				Remove	0 1
			-		
			-		

Figure: Load group definition functions

Under *Load group* group, there are three functions:

- Insert

With this function the user can add a new *Load group*. After clicking on Insert button, the parameters can be set in the dialog.

- Delete

This function deletes a *Load group*.

- Delete all

This function deletes all defined load groups.

Under *Load case* group, there are three functions:

- Insert

An existing load case can be inserted to a load group.

- New

The user has the opportunity to define load cases in *Load groups* dialog. After defining a load case in this dialog, the new load case will be added to *Load case* list (the user can see it in Load cases dialog).



5. The new load case defined!



Figure: Load case definition in Load groups dialog

- Remove

This function removes the selected load case from the current load group.

Definition steps of new load groups

- 1. Click on an empty *Load group* cell. Define the name of the new load group and set its type. Set the required factors of the group according to the selected type and the applied standard.
- 2. Add a load case to the group by selecting it from the *Included load cases* drop-down list. To add more than one cases to a group, use the *Insert case(s)* option.



"*Permanent*"-type load groups defined will be present in all load combinations.

If more than one load cases are assigned to a load group, they will never be simultaneously present! So, it is recommended to set permanent-type load cases (e.g. automatic dead-load, roof weight etc.) in own "*Permanent*"-type groups to avoid the loss one of them.

3. Define the next load group by repeating the previous steps.

Optional steps

- 4. A load group can be edited by double clicking on its *Load group* cell.
- 5. Load groups can be imported from a pre-defined database and exported for next and later projects by clicking on *Import/Export>* in *Load groups* dialog box.

Let's see an example, how partial loading of slab can be modeled with load groups. Just place the load cases represented the loading statuses into one *"Temporary"*-type load group.



Figure: Partial loading of a slab

Moving load is not available in FEM-Design, but it can be modeled together with load cases (load steps) and load groups ("Temporary"-type) for example in a beam case.



Figure: Moving loads on a beam

MODEL AND DATA CONNECTIONS

There are numerous ways to get draft and initial model geometry for later design in FEM-Design.

With the so-called *Wizard* tool, typical frame and roof structures can be modeled quickly by giving the parameters of the required geometry, support and load, and the program automatically generates the real 3D model.

Models can be shared between the 2D and the 3D modeling of *FEM-Design modules*. For example, a slab of a multi-storey building (modeled in *3D Structure* module) can be design separately from the other slabs in the *Plate* module.

FEM-Design supports the major CAD drawing exchange formats (*DWG/DXF*), so imported drawings can be used as referenced draft for building up the structural models.

FEM-Design is able to import 3D models from other structural and architectural programs in two ways:

- FEM-Design supports the import of the most popular 3D model exchange data format called *IFC* (*Industry Foundation Classes*). The model-based data import gives the possibility to use the models designed by other disciplines (such as architectural) as an initial model or to show them as a reference.
- StruSoft develops *direct 3D model exchange links* with ArchiCAD (architectural discipline) and the major structural applications: Tekla Structures and Revit Structure. Add-ons build in these applications export native FEM-Design elements generated from 3D models.

Frame Wizard

Wizard helps you to model typical frame and roof structures with *beam* elements, *point supports* and *line loads*. With geometry parameters, you can define one frame/roof stand (planar model) or a multi-frame/roof system with 3D extension. Line loads such as automatic and additional dead-loads, snow, wind and live load can be added to the beam components.



Definition steps

15. Start the Wizard command from the Tool menu.



Figure: Typical frame and roof structures as the range of Wizard

16. Choose the required frame/root type in the Wizards dialog box.

17. According to the selected structure type, set the geometry parameters. Click *Preview* to check the defined geometry.





Figure: Settings dialogs

Meanings of some main parameters:Number:Number of the frame standingsc/c:Distance between the frame standings

18. If load definition is available for the selected type, define the additional dead load (roof weight), snow, wind and live load based on the figure shows the meaning and position of line loads (intensity).



Save option (of *Wizard*) lets you save all parameters of the current wizard template. The exported wizard settings can be imported for later project with the *Wizard's Load* tool.



Defining "Dead load" (additional, roof weight) line load generates the *automatic dead load* too as a separate *load case*.

19. Clicking *OK* finishes the *Wizard* and generates the 3D model with loads and supports (depending on the selected type).



Figure: Roof structure example

- 20. The generated beams have no profiles and materials. So, *Wizard* generates only beam axes without properties. Add properties to beams with the *Beam* command's *Properties* tool.
- 21. Wizard sets up hinged and rigid supports (as displayed by wizard settings) from single support components. So, for example a fully rigid point support is modeled by 3 motion-type and 3 rotation-type single point supports. You can modify the support conditions by the *Point support* command's *Properties* tool.



Figure: Complex (rigid) support modeled by single Point supports

Optional steps:

- 22. Add more *structural elements* (plate, wall, beam, column etc.) with the commands of the *Structure* tabmenu.
- 23. Edit the generated loads or add new ones with the commands of the Loads tabmenu.

Model Exchange between FEM-Design Modules

Different FEM-Design modules are able to share models between each other. The next table shows the possible connections between modules.

		Source Module (.file extension)						
		2D mo	odels					
		(.pla)	(.wal)	(.frm)	(.str)	(.prd)		
ıle					(1)	(1)		
tion Modu			\mathbf{r}		(2)	(2) (2)		
					(4)			
estina	→ ↓ ↓	(3.1)	(3.2)	(5)				
Ď	1	(3.1)	(3.2)	(5)				

Table: Model connections between FEM-Design modules

The main featured connections from the above possibilities:

- Slab-system imported from 3D Structure/PreDesign to Plate module (1)
 - Slabs or slab-system of a building (created by *3D Structure* or *PreDesign*) *storey* together with connected walls, columns, beams and loads can be imported into *Plate* module.



Only the loads directly placed on the slab elements can be imported to *Plate*. The loads arrive from other stories or connected elements have to be added manually to the slab regions as *Point/Line/Surface loads*.

Steps of import:

- 1. Click *Open* (*File* menu) in the *Plate* module. Select the *.str* (*3D Structure*) or .prd (*PreDesign*) which contents you would like to import to *Plate*.
- 2. Select the *Storey*, which slabs (plates) you would like to load as a new project.

The import requires *storey-system* from the file selected by *Open*.

- 3. Allow or forbid the import of columns and walls connected above/below the slabs of the selected storey. Beams and loads connected to related slabs will be automatically imported.
- 4. Set the end conditions (fixed or hinged) of the walls (=wall supports) which you would like to import.



5. Click *OK* to start the import.

Figure: Import of slabs with connected elements into a Plate project

- Wall-system imported from 3D Structure/PreDesign to Wall module (2)

Wall regions situate on a vertical plane defined by an *axis* of a building (created by *3D Structure* or *PreDesign*) can be imported into *Wall* module together with connected supports and loads.



Only the loads directly placed on the wall elements can be imported to *Wall*. The loads arrive from other connected elements have to be added manually to the wall regions as *Point/Line loads*.

Steps of import:

- 1. Click *Open* (*File* menu) in the *Wall* module. Select the *.str* (*3D Structure*) or .prd (*PreDesign*) which contents you would like to import to *Wall*.
- 2. Select the Axis, which wall regions you would like to load as a new project.

The import requires *axis-system* from the file selected by *Open*.

3. Click *OK* to start the import.



Figure: Import of walls with connected elements into a Wall project

- Single plate/wall import to 3D Structure (3.1 and 3.2)

Slabs/Walls with their supports and loads created and designed in the Plate / Wall module can be imported into the 3D Structure module. Multi-storey building can be defined by copying the opened elements, and run global stability analysis and design for the final model.

Steps of import:

- 1. Click *Open (File* menu) in the 3D *Structure* module. Select the *.pla (Plate)* or *.wal (Wall)* which contents you would like to import to *Plate*.
- 2. In case of curved walls, set their conversion to planar wall regions. Curved wall element is not available in the 3D structure modules. The value of the approximation can be set in the *tmax* field. If the "originally curved" walls are connected to curved plate edges, the program also modifies the curved plate edges to polyline.
- 3. Click *OK* to start the import.

Optional steps:

- 4. Copy (*Modify* > *Copy*) plate regions with their wall/column supports and loads, if required.
- 5. Only one *Plate/Wall* file can be imported (with *Open*) in a 3D project. But, you can open different files in separate 3D projects, and later you can merge their contents into one with the *Copy* and *Paste* commands and set their real position with the *Move* command (*Edit* menu).
- 6. In *Plate* module, the *Wall* and *Column* elements are supports. So, do not forget to add supports to the bottom end of the columns and walls of the lowest storey.



Figure: Import of a slab (system) into a 3D structure project

- Frame part exported from 3D Structure to 3D Frame (4)

Frame parts (beams, columns and truss members) of a multi-storey building (created in *Structure*) can be imported into the *3D Frame* module by filtering out the shell, plate, wall elements and their related surface supports and loads. The import process is very simple: just apply *Open* (*File* menu) for the *.str* (*3D Structure*) files in *3D Frame*.



Figure: Import of frame parts into the 3D Frame module

- Frame import to 3D Structure (5)

Initial analysis and design can be done for only the load-bearing frame parts in the 3D *Frame* module, then the bar elements can be imported into the 3D *Structure* modules to complete them

with planar element such as plates, walls and their supports and loads. The import process is very simple: just apply *Open (File* menu) for the *.frm (3D Frame)* files in *PreDesign* or *3D Structure*.

Opening PreDesign model in 3D Structure and 3D Frame is not allowed since version 12.0, as PreDesigh has totally different database and not possible to keep compatibility with other 3D modules.

CAD Drawing Import/Export

A DWG or DXF file generally contains drawing objects (points, lines, dimensions, texts etc.), so it can be used as a reference draft displaying a floor plan, a section or an elevation. Each *FEM-Design module* is able to import (and also export) DWG and DXF.

All elements, so both structural model and drawing elements, can be converted and exported to DWG/DXF format. The objects are automatically converted to lines with their 3D position and placed onto separate layers by object types. Also, the symbols and info labels are broken to text and line parts.

Import

The steps of opening a DWG/DXF file:

- 1. Open the DWG/DXF file with *File > Open*.
- 2. Set the import settings in the upcoming dialog:

Layer colour and name	Visible	Locked	Empty	One conversion		
0	Yes	No	No	1 unit = 1 mm		
RED	Yes	No	Yes	I unit = 1 cm		
GREEN	Yes	No	Yes	1 unit = 1 dm		
BLUE	Yes	No	Yes	1 unit = 1 m		
YELLOW	Yes	No	Yes	1 unit = 1 inch		
MAGENTA	Yes	No	Yes	Custom, 1 m =		
CYAN	Yes	No	Yes	1.0 units		
BLACK	Yes	No	Yes			
WHITE	Yes	No	Yes	Custom, 1 unit =		
FEM-COLUMN	Yes	No	No	1.0 mm		
FEM-3DPLATE	Yes	No	No			
FEM-3DWALL	Yes	No	No	 Original colours 		
FEM-COVER	Yes	No	Yes			
FEM-INFO	Yes	No	Yes	Halftone colours		
SYS-AXES	Yes	No	No	Grevscale		
SYS-STORIES	Yes	No	No	One colour:		
SYS-VIEWS	Yes	No	Yes			
FEM-MODELING	Yes	No	Yes			
FEM-PDREINF	Yes	No	Yes	Cheuryisible layers ealy		
DIM	Yes	No	Yes	Show visible layers only		
FEM-BEAM	Yes	No	No	Do not show empty layer		
FEM-TRUSS	Yes	No	Yes	Resolve XRefs		
FEM-RESNUM	Yes	No	Yes			
FEM-RESULT	Yes	No	No			

Figure: DWG import settings

- *Unit*: Set the same drawing length unit with the unit-system saved in the file by the exporting application. Most of the DWG-compatible applications use "mm" while defining and exporting drawings.
- *Scale*: Set the drawing scale of the opened file by typing the required value. Scale affects the display of texts, line types and hatches.
- Ignore points: Check this box to filter out drawing points, so not to allow their import.
- *Set to 2D*: Check this box, if you would like to merge all drawing elements (independently their level position) in the XY plane of the *Global co-ordinate system*. It is recommended only, if the points of the imported drawing place in the same or near the same (small geometry inaccuracy in Z direction) plane.
- *Ignore empty layers*: Check this box not to import empty layers that do not contain drawing elements. Unchecked box merges all layers stored in the DWG/DXF file.

The imported layers will be converted to *Drawing layers*.

- *Merge points*: Checking this box corrects the inaccuracies of the imported drawing in the given merging distance and deletes the point duplications.
- 3. Hide the unnecessary drawing layers to make clear the points and lines you would like to use for later model definition.



Figure: Active imported drawing layers

With *Modify* > *Change appearance* you can query the host drawing layer of a selected element.

4. Create the structural model based on the referenced (opened) drawing.

ZQZ

Apply the proper *Object Snap Tools* to find the required main points, endpoints, intersections, lines etc. for the input (reference points and lines) of the *model elements*.

Set the size of the elements according to the drawing elements after inquiring/measuring positions, directions and sizes with *Tools* > *Query*, if it is needed.

Pay attention to pick proper/correct lines when defining reference lines and insertion points for the structural objects. For example, think on the wall position while connecting it to a plate edge. Not always following the real state (defined by the DWG drawing) is the optimized way to model elements.



Figure: Reference line (and finite element mesh) definition based on the drawing

Rotate the position of the drawing, if it is needed. For example set a section/elevation drawing to the *Global XZ* or *YZ* plane in 3D modules.



Figure: Rotation of a drawing (definition of an elevation)

In most cases the drawing elements of the merged DWG/DXF file are placed far from the origin of the *global co-ordinate system*, so click *View* > *Zoom margin* to show the entire drawing after opening it.



Only one DWG/DXF file can be opened at a time. But, you can merge more than one drawing (e.g. floor plans of different stories) into a project as the following hint:

- 1. Open the first drawing as a new FEM-Design project.
- 2. Open the second/other drawing as a separate FEM-Design project. (More than one copies of a FEM-Design module can run in the same time.
- 3. Apply *Edit* > *Copy* for the required elements of the second/other drawing to send them to Clipboard.
- 4. Use *Edit* > *Paste* to merge the drawing elements (from the Clipboard) into the first project. Set the real position of the drawing insertion (point), so for example, place the 2nd floor elements in the correct 2nd level position.

Export

The steps of saving a DWG/DXF file:

- 1. Click *File* > *Save as,* and choose the proper DWG/DXF file.
- 2. Set the export settings in the upcoming dialog:

ayer colour and name	Visible	Locked	Empty	Unit conversion
0	Yes	No	Yes	1 unit = 1 mm
RED	Yes	No	Yes	I unit = 1 cm
GREEN	Yes	No	Yes	1 unit = 1 dm
BLUE	Yes	No	Yes	1 unit = 1 m
YELLOW	Yes	No	Yes	1 unit = 1 inch
MAGENTA	Yes	No	Yes	Custom, 1 m =
CYAN	Yes	No	Yes	1.0 units
BLACK	Yes	No	Yes	Carter funit
WHITE	Yes	No	Yes	Custom, 1 unit =
FEM-COLUMN	Yes	Yes	No	1.0 mm
FEM-3DPLATE	Yes	Yes	No	
FEM-3DWALL	Yes	Yes	No	Colour modes
FEM-COVER	Yes	Yes	No	Original colours
FEM-INFO	Yes	Yes	Yes	Halftone colours
SYS-AXES	Yes	Yes	Yes	Greyscale
SYS-STORIES	Yes	Yes	No	One colour:
SYS-VIEWS	Yes	Yes	Yes	
FEM-MODELING	Yes	Yes	Yes	
FEM-PDREINF	Yes	Yes	Yes	Show visible layers only
DIM	Yes	No	Yes	
FEM-BEAM	Yes	Yes	No	bo not snow empty laye
FEM-TRUSS	Yes	Yes	No	
FEM-RESNUM	Yes	No	Yes	
FEM-RESULT	Yes	Yes	Yes	
				AutoCAD 2010/2011/20: *

Figure: DWG export settings

- *Unit:* Set the drawing length unit of the exported elements to the type required by the other application which wants to open the DWG/DXF file.
- *Arc resolution*: Curved elements, arcs and circles will be exported as connected straight lines (polyline). The accuracy of the line-conversion approximation can be set here. Arc resolution is equal the angle closed by the tangent of the connected lines. This setting effects the number of the generated lines too.
- *Convert text*: The option converts the special Unicode characters (such as ő, ű etc.) to readable ANSI-compatible characters (such as o, u etc.). So, this option is recommended to be used in case of text, labels etc. Non-used option may cause unreadable symbols in the application opens the exported DWG/DXF file.

All elements (beams, columns... supports, loads, drawing elements, labels, etc.) will be generated to lines and texts, and they will be placed onto separate drawing layers by their types. All DWG/DXF-compatible applications are able to filter elements by layers during or after the import process.



Figure: Exported DWG in 2D and 3D views

Strusoft XML

Models can be imported or exported between structural engineering programs (e.g. Tekla, Revit) in a special Strusoft XML format (struxml). Exporting from and importing to FEM-Design 3D Structure module by struxml format can be done by *Save as* or *Open* commands respectively.

Characteristic of the second s



IFC Import

FEM-Design gives the possibility to import 3D models saved in IFC files from other architectural and structural programs. Some of the most important features of the FEM-Design's IFC import:

- Compatible import formats: IFC2x version 2 and 3
- Material conversion table and style
- Object-filtering
- Connections of misplaced objects (point-point and line-line connection)

An IFC model contains elements according to the mapping rule of model elements to IFC element types. For example, a column is represented as *lfcColumn* in the IFC scheme by default. Not only the geometry, but numerous properties like material, profiles etc. are assigned to one element.

The IFC model has to contain **Architectural View**. The static system is converted from the architectural/structural model elements (IFC architectural representation), so *FEM-Design* creates native structural objects such as *Plates*, *Walls*, *Beams* and *Columns*.



Figure: Generation of real FEM-Design objects

Of course, the result of element import depends on the FEM-Design module where the process is done:

Module	Architectural View
Plate	One storey only: - Plates and Beams - Walls and Columns as supports
🗖 3D Frame	Beams and Columns only
3D Structure PreDesign	Plates, Walls, Beams and Columns

Table: Imported IFC elements by "views" and FEM-Design modules

The steps of opening an IFC file:

- 1. Load the IFC file with *File> Open*.
- A tree lists the IFC model content by buildings and their storeys. In the *Architectural* views, objects 2. can be selected for the import.

"Irregular" label represents the element types/geometries, which cannot be converted to FEM-Design structural elements.



Plate module only imports plates situate on the same storey together with supports under them. Most of the architectural software places the "support elements" (like walls, columns etc.) on one storey below the plates' host storey, so the requested elements can be selected from separate storeys. In case of 🛄 3D Structure models, the entire model can be imported independently from the storey system.



Plate-system with under supports

Figure: Differences between modules used for the model import

3. Set the main properties of the selected elements.

General properties	1		Statical system		
Name SW - 004	Layer filter			Тор	Bottom
Unique identifier 3As Ob9eX2cPnWpUo7cfxJ	Concrete Block 150			• Fixed	• Fixed
	Layer name	Thickness [mm]		🔘 Hinged	○ Hinged
	Concrete Block	150.00		Uplift allowed	
Material mapping					
IFC material 🧱 Concrete Block 150	Total thickness o Thickness of load-be	layerset: 150.00 i aring core: 150.00 i	nm		
FEM-Design material . 🧱 C20/25		2			
Select material Mapping	g table				

Figure: Main properties of elements

- *General properties*: it displays the ID name of the elements and the IFC global unique identifier (GUID). Description may contain the profile name of the elements depending on the exporter application.
- *Material mapping*: it displays the material of the selected element (IFC material) and the default FEM-Design material (available in the current standard) assigned to the imported IFC material. Of course, you can modify and set the required material that will model the imported one. Material can be set with *Select material* by elements or based on the material *Mapping table*. Of

course, the material can be modified later with the element's $\stackrel{\checkmark}{\rightarrow}$ *Properties* tool.

The conversion can be done on different model levels, so for the entire structure, by storeys (e.g. 1st floor), by object type (e.g. all slabs), and for the type (e.g. slabs) of the selected objects.

- Select material

A dialog appears with the built-in (available in the current standard) and user-defined concrete, steel and timber materials of FEM-Design. New concrete, steel and general materials can be created in the dialog too.

- Mapping table

You can associate standard materials (as the applied code) to one or more materials of the imported model (= "FC Material") in dialogue format. It works for the complete model, so there is no difference that you select an object or a storey or a building. There are two independent conversion maps, one for the architectural and one for the structural model.

Select one (or more) material in the "IFC Material" column, then press *Assign material* and finally choose proper FEM-Design material available in the current code. Here, new materials can also be created (*New*). Clicking *OK* returns to the material conversion table that shows the selected conversion: the selected material appears in the "FEM-Design Material" column. Material mapping table can be saved and exported for others with the *Export* option, or previously saved conversion rules can be loaded with *Import*.

- *Layer filter*: the core of the composite layered materials can be set here.
- *Statical system*: in Plate module, the end connections (fixed or hinged) of selected columns and walls (supports) can be set as a default value. Of course, the end connections can be modified later with the element's *Properties* tool.
- 4. Clicking *OK* loads the *IFC* model. The loading process may take a long time depending on the complexity and the size of the imported model.

The program generates rigid *connection objects* (*Point-point* and/or *Line-line connection*) between the misplaced elements, which solid surfaces are touching each other, but the generated calculation planes are not. The method depends on the types of the "connected" objects:

- If a bar element connects to an object, the program applies *Point-point connection*.
- If only planar elements connect to each other, *Line-line connections* are assigned to them.



Figure: Line-line connection generated between the plate and the touching wall elements

You can modify the connection stiffness with the *Properties* tool of the *Point-point* and *Line-line connection* commands.

Direct Data Links

StruSoft develops direct links with the architectural software ArchiCAD and the structural applications Tekla Structures and Revit Structure. So, thanks to external tools, ArchiCAD can exports the entire architectural or a simplified structural model or its part, and the two structural management/editor applications can export the analysis model generated from the structural model.

Direct link means that add-ons embedded in the previous applications generates automatically FEM-Design *Plate, Wall, Column* and *Beam* elements from their native 3D elements, so further conversion is not needed inside FEM-Design while opening the model files.

A

The external StruSoft applications together with their user manuals are available on the *www.fem-design.com* (*Downloads* menu) by the versions of ArchiCAD, Tekla Structures and Revit Structure.

The ArchiCAD Add-On exports *.fdx* files compatible with the FEM-Design 3D *Structure* or *flate*.

The Revit Structure Add-In exports *.r2f* files compatible with the FEM-Design *3D Structure*.

Tekla Structures sends analysis model (*Analysis > Analysis & Design Models*) directly to FEM-Design 3D Structure, so both FEM-Design and Tekla Structures licenses can be installed on the same computer.

The next table summarizes the FEM-Design elements matched to the elements of the related applications by the external tools in the special exchange file formats.

FEM-Design	ArchiCAD	Revit Structure	Tekla Structures	
Pean	Beam	- Beam	- Concrete Beam	
r v Deum		- Truss	- Concrete Polybeam	
		- Brace	- Steel Beam	
		- Beam System	- Steel Polybeam	
			- Steel Curved Beam	
Column	Column	Structural Column	- Concrete Column	
• • Column		(straight only)	- Steel Column	
Diata	- Slab	- Structural Floor	- Concrete Slab	
r rule	- Roof	- Isolated Foundation	- Steel Contour Plate	

		- Wall Foundation - Foundation Slab	
Wall	Wall	Structural Wall	Concrete Panel
	(Straight, Curved, Trapezoid)	(straight only)	

Table: Elements by applications can be imported as FEM-Design elements

Material and profile mapping is recommended at all direct link connection:

- In case of Revit Structure, the Add-In gives the possibility to map the Revit Structure profiles and materials with the FEM-Design ones.
- In case of Tekla Structures, the material and profile mappings can be done in editable (e.g. with Notepad) text files. The factory default templates can be found in the FEM-Design's "FORMATS" library: "TeklaFEM-DesignMaterialMapping.cnv" and "TeklaFEM-DesignProfileMapping.cnv".
- In case of ArchiCAD, opening the *.fdx* file in the 3D *Structure* or *Plate* module, a setting dialog comes up with the following settings:

Control parameters			$\overline{\mathbf{X}}$		Control parameter	s		
Storey Composites Defaults			_		Storey Composites	Defaults		
Selected item	Source file	Control parameters		×	Plate	Undefined		
No. Name D. Ground Floor	Source : Ai	Composites Defaults			Beam	Undefined		
	Version : 1	Composites / Fills X roof, conversion	Layer Roof Tile Air Space	- 0.0350[m] - 0.0250[m]	Material	Undefined	Arrange beam to reference	e line
	Origin : C:N	× Lightweight Concrete × Masonry Block × Structural Concrete	X Foreground Air Space Batt Insulation	- 0.0010[m] - 0.0500[m] - 0.0750[m]	Column	Undefined	Wall	defined
	12\Default Project :		Plaster	- 0.0125[m]	Min. size [m]	0.10000	Min. thickness [m] 0.1	0000
	Author :				y' direction	z' direction	Connection	
The slabs on the selected	Revision :		Colorb for the sharebox		 Fixed Hinged 	 Fixed Hinged 	 Fixed Hinged 	Wall
connected objects will be loaded.	Keywords :	Use Dontuse	Select for structure	Clear from structure		Ĩ	Crea from	att Min. thickness [m] 0.10000
	Notes :						- Arct refe	re Create support from ArchiCAD's
	- I				Fixed Hinged	Fixed Hinged	Fixed Fixed Hinged	reference line.
						0		Accuracy of arc wall [m]
							OK	A A
				OK Court				K
	L			UK Cancel				

Storey (Plate module)

In ArchiCAD, users work by storeys. The Plate module's "Selected item" lists all available storeys. Choose the required one, and the program will import the plates and its connected objects like beams and column/wall supports. In case of 3D Structure, storeys can be not selected, the module opens the entire model.

Composites

"Composites/Fills" lists the materials (ArchiCAD Cut Fills) together with the multi-layered composite structures defined in ArchiCAD. For composites, the core part (that thickness will be considered in FEM-Design) can be set in this dialog tab.

Defaults

- Here you can set the "Material" mapping by element types.
- Minimum size/thickness value gives the limit of the columns/walls can be imported. For example, giving 0.1m min. thickness for walls means that the partition walls with smaller thickness (e.g. 9cm) will be not loaded while import.
- Fixed and hinged end "Connections" for columns and walls (Plate module only) can be set with the radio buttons.
- Unchecked "Arrange beam to reference line" option will be place beams in their original position as defined in ArchiCAD; but the calculation line will be their (middle) axis. The checked state derives same position of the beam's calculation line with its reference line in ArchiCAD.
- "Create support from ArchiCAD's reference line" has the same function with the previous beam setting, but it effects on walls.

- "Accuracy of arc wall" generates straight wall segments from a curved ArchiCAD wall in the 3D Structure module (because curved wall can be defined in 3D modules). The maximum deviation (a) from the arc can be set in meter unit.

FINITE ELEMENT MESH

Finite Element Method is used in calculation engine as it is also visible in the name of program (\underline{F} inite \underline{E} lement \underline{M} ethod = FEM). This chapter introduces the main concepts, features and functions of the built-in finite element method.

Element Types

Depending on the applied FEM-Design module, the engine uses the following line and 2D (rectangular and triangular) finite elements.




Table: Finite element types by FEM-Design modules

In the 3D modules, you can choose between "standard" and "accurate" 2D element types. With standard elements you can run 4-times faster but less accurate analysis than with the fine elements. In case of bar elements, the program assigns 2-node line elements to each beam and column element by default:

- 1 piece when choosing "standard" elements, and
- 2 pieces when choosing "accurate" mode.



The program divides curved beam objects by several line elements in number depends on the central angle of the arc.

The element type "standard" or "accurate" can be set in the calculation dialog (*Analysis* > *Calculate* > *Analysis* > *Finite element types*).



Figure: Finite element types by FEM-Design modules

Modification on the geometry of a structural object causes the deletion of its finite elements.

Mesh Generation

FEM-Design offers a fully automatic finite element mesh generation by using optimized (factory default) or custom mesh settings. Of course, the generated mesh can be modified with special easy-to-use *edit and modify functions*.

Fully automatism means that the program generates the mesh with elements having average element size optimized for the structure and its environment (supports and loads). The process can contain automatic element refinement and *peak smoothing* algorithm according to the settings.

Automatic mesh generation can be done according to the *mesh settings*:

- Before calculations

Click *Prepare* in the *Finite elements* tabmenu. You can see and check the finite element suggested. The mesh will be visible by activating the *Surface elements* layer automatically. If you do not edit the mesh and modify the structural model, the later calculations will use the mesh generated by *Prepare*, so no further mesh generation will be done.

- As the result of calculations

If the program does not find previously generated finite element mesh, running calculations (analysis or design) by *Calculate* generates it automatically. The mesh can be displayed by activating the *Surface elements* layer in the current structural or result view mode or just by returning to Finite elements mode.



Figure: Automatic finite element mesh generation

Mesh Settings

The settings of the automatic mesh generation are available only in Finite elements mode and at *Settings> All...> FEM > Mesh* and *Calculation*.



Figure: Settings affect automatic mesh generation (Prepare)

"General" settings

Merge objects - options	Auto peak smoothing region around
Distance [m] 0.050000	Beam end points Column end points Truss end points Plate intersection end points Wall intersection end points Point supports line support end points
Merge objects - modes Manual start Auto / manual start Inform about volume of merged objects at auto	Surface support break points Point connections Line connection end points Surface connection break points Fictitious bar end points r f*v r = t/2 + f * v
Load handling	

Figure: General settings of mesh generation

- Merge objects

The program merges *fixed points, fixed lines, supports, beams, columns* and *walls* (only in the *Plate* module) to *plate* and/or *wall* regions (the border of the regions). It is decided randomly which objects will be let in their place or removed. The objects shorter than a merging *distance* will be deleted. The objects being at the same place (covering) and having same properties will be deleted except one.

The program also merges columns and/or beams together. It is decided randomly which line elements will be let in their place or removed. The bars shorter than the merging distance will be deleted. After that, the supports fit to the bars. The objects being at the same place (covering) and having same properties will be deleted except one.

The program merges the loads to the geometry created in the first step. Those loads that have not been merged by the previous way will be merged together. It is decided randomly which ones will be let in their place or removed. Line loads shorter than the merging distance will be deleted. If the *Avoid load multiplication* option (see later) is active, the loads having the same position and same properties will be deleted except one.

$\underline{\mathbb{A}}$

The merge process may modify the original shape of the objects to a simpler geometry; but a simple figure cannot be changed for a more complicated one. For example the program does not fit a line load having straight action line to a curved edge although merging distance requires that.

This version of object merge cannot merge the plate or wall regions to themselves or to each other. The user has to pay attention to the correctness of these objects.

Merge object function can be launched either manually in *Finite elements* tab \bigcirc or automatically before mesh generation, if *Auto / manual start* option is selected.

Distance sets the maximal investigation zone between elements, so if the objects are closer than the defined distance, they will be merged. The suggested distance value is 3 to 5cm for engineering problems.



Figure: Correction of misplaced elements (Object merge)

Activating *Merge peak smoothing region* and *Merge load* allows the object merge to work for peak smoothing regions and loads.

Using *Avoid load multiplication* deletes the loads having the same position, geometry and properties (load case host and value) by keeping only one copy.

- Load handling

By default, load positions are independent from the finite element mesh; so for example, it is not necessary to place point loads into finite element nodes and vice versa. Although it is recommended to place loads (especially the concentrated loads with high value) into nodes, it is not necessary. *Adjust mesh to load positions* automatically places finite element nodes in the action points, on the action line and region border of the loads depending on their types (point, line and surface load), so the mesh follows the load position and geometry.



Figure: Load handling in mesh generation

- Auto peak smoothing region around...

To solve the result *singularity problems* above supports and other critical points, the program may run *peak smoothing algorithm* around the listed elements. Activating an element in the list, the program automatically creates *peak smoothing region* around it.

"Elements" settings



Calculated average element size

By default, the program automatically calculates the optimal average size of the 2D finite elements considering the size, the geometry, the environment etc. of the structural elements. So, you do not need to give an initial value for it. The automatic calculation and the element size depend on the following settings options.



Element sizes can be set manually for all model regions or by regions with the Average element size command. If you modify the default "Automatic" value for a planar structural element (wall or plate) to a given value, the automatic element size calculation will be skipped for that region, and the given size will be used for that.



Figure: The Average element size command and the "Automatic" option

Using *Region by region*, the program optimizes the element size by model regions. In this case, the regions will contain meshes generated by different average element sizes. This option is recommended to use in case regions (e.g. having openings and holes) need to be refined (more dense mesh).

Using Consider all regions together, the one optimal average element size will be determined for all model regions having "Automatic" size setting. This option is suggested for structural models contain regions with nearly same geometry and size parameters.



Figure: Calculation modes of average size of 2D elements

In the Scale figure, the optimal average element size can be reduced or increased with a given ratio set by the scroll button. The grey mesh shows the recommended optimal size, whilst the blue one shows the modified custom size. Double clicking the *Scale* figure resets the element size to the optimal average element size (1:1 ratio).



Figure: The average element size will be four times bigger than the optimal size

The *Correct according to the minimum division numbers* option modifies the average element size of the 2D elements, if the minimum division number (*automatic* or *custom*) of the boundary lines and edges requires that. This option is recommended for generating uniform finite element meshes. Skipping this option causes dense mesh near edges where the minimum division numbers are predefined.

Using the *According to the peak smoothing regions* option considers the *peak smoothing* settings of model elements in the calculation of the average element size.

- Line element parameters

The default minimum number of the line elements can be set here for the bar elements. The meaning of the default division number depends on the applied element type: *standard or accurate*. For example n=2 value sets (minimum) 2 finite elements for a whole bar (if neighboring elements connect to it only in its endpoints) or a continuous part of it (in case of joined or intersected neighbors) in case *standard* element type and 4 elements at *accurate* element type.



Figure: Meaning of Division number in case of Standard and Accurate element types

At curved beams, alpha parameter sets the minimum division number: minimum division number = central angle of the curved beam / alpha.

 \mathbb{A}

Division number can be set manually for all bar elements or by line elements with the *Division number* command.

-QZ

For imperfection (of steel bars), stability and dynamic calculations, it is suggested to set the default *n* value to more than 1; 4-5 division number is the recommended minimum for *n*.

"Functions" settings

Functions					
Generate surface mesh	Check surface mesh				
Refine locally where needed	Geometry				
Max. step 6	Min. angle [deg] 10.000				
Warn about reaching max. step	Max. angle [deg] 170.00				
Reduce average element	Max. side ratio 8.0000				
size if neccessary	✓ Overlap & cut				
Smooth surface mesh	🗹 Topology & gap				
Steps 3					

- Automatic refinement in surface mesh (Generate surface mesh)

The *Refine locally where needed* option – as an iteration process - eliminates distorted elements, which may normally be derived from accidental geometric errors (if *object merge* is not used). So, the option makes the finite element mesh denser at the locations where needed. (Deactivation of this option can be used in case of searching geometric errors.) The *Max. step* (recommended value is 6) defines the number of the iteration step of refining. The iteration will end when generated elements have the required side-ratio or the steps of the iteration reach their maximal value. If *Max. step* is not enough for the optimal refining, a warning message informs you the number of iteration steps is not enough and there are critical geometry errors (if the *Warn about reaching max. step* option is active).

The automatic refinement may cause too dense mesh at special geometries (e.g. at highly depressed regions), so in that case, it would be more practical to reduce the average element size with the *Reduce average element size if necessary* option.

- Smooth surface mesh

Smooth procedure calculates the optimal coordinates of the corner nodes of elements. It is recommended after splitting or merging elements. The best mesh can be achieved with the iterative use of the *Rebuild* and the *Smooth* commands. Smoothing of a mesh is executed with iteration technique: the procedure places the nodes of the triangle elements in such a way, that the area of the triangles will be balanced. The number of the smoothing steps can be set in the *Steps* field.

- Check surface mesh

The *Check surface mesh* option lets the program to check the geometry of the mesh after automatic mesh generation. The mesh can be controlled with respect to unsuitable geometry, overlaps and topology. This means that mesh errors, produced by an automatic or manual mesh generation can be easily found. If the program finds defective geometric elements in the model, it sends a warning message (error list) and displays the position of the mesh errors.

The *Geometry* tool checks the geometry of finite elements, such as the angles of the elements and the ratio of the largest and smallest sides (*Max. side ratio*).

The *Overlap & cut* tool checks overlapping and intersecting finite elements, which can be caused for example by copying or moving regions together with their finite element meshes.

The *Topology & gap* tool checks the topology of the finite elements and finds possible gaps. A typical topological error, when for example a corner node of an element lies on a side edge of another element. This problem can easily arise manually by using the *Split* command incorrectly.



Figure: Topologic errors (unconnected nodes)

Topological errors can be easily solved by the *Rebuild* and *Smooth* commands and quick algorithms.

"Prepare" settings

Prepare				
Regenerate surface mesh autom Refine surface mesh after reger Beams Columns Free edges Region borders Point supports	atically on the changed regions rerate around Point-point connections Inter-line connections Point loads Unre loads Unre loads			
Surface support edges	after refine			
Smooth surface mesh after refine				
Check the surface mesh				

- Regenerate surface mesh automatically on the changed regions

Using this option, the program will regenerate the mesh at any geometrical changes of region elements and will generate mesh on the regions having no mesh.

If you switch off the *Regenerate surface mesh automatically*... option, the program will not generate mesh on the regions modified geometrically and will send an error message. In this case you cannot start calculation until you generate mesh on those regions. And, if you would like to generate mesh manually with the *edit functions*, also inactivate this option.

From the element list, you can choose element types for automatic mesh refinement around them.

The *Optimal rebuild surface mesh after refine* option rebuilds the mesh automatically after refine. It is recommended to use this option, because elements having non-optimal side-ratio (after refining) will be removed. This procedure will also create more optimal quadrates from the triangles.

If you have objects to refine, the *Smooth surface mesh after refine* option will smooth the mesh after generation, refine and rebuild procedures.

- Check the surface mesh

If this option is activated, the program will automatically check the mesh on regions having valid mesh and are not necessary to be regenerated. The mesh of a region is valid, if the geometry of the region has not been modified since the last generation.

Peak Smoothing

Singularity problem

As an effect of the mesh refinement the calculated results are converging to the theoretical solution. The problem is that at certain places we get infinite inner forces according to the theory, so the inner forces increase each time by refining the mesh. These places could be: point supports, end points of edge supports, vertices of surface supports, end points of beams and columns, end points of intersection lines of adjoining surfaces, point loads, end points of line loads, vertices of surface loads etc.

In practice, usually, the singularity problem occurs at supports because they heavily influence the inner forces (e.g. negative moments) in ratio.

Possible solutions

There are three known possibilities to solve the above-mentioned problem:

- Choosing optimal finite element size at singularity places

FEM-Design aids that with several built-in tools such as automatic element size adjustment, automations in mesh generation, automatic local densification etc. It is evident, that choosing optimal element size cannot be perfect, because we should have to know the appropriate values to which we adjust the average element size in advance. The functions used today for automatic element size calculation and generation are providing values with adequate precision in most cases, but it is obvious that they cannot guarantee that in any case.

- More realistic and precise model definition

Point and line loads/supports with action surface (only action points and lines) do not exist in real life. So, if you model all point/line loads and supports as surface loads/supports, then you can cease the problem derived from the singularity. This opportunity is available in FEM-Design, because the user can apply *surface supports* and *loads* with any directions and any geometry of action surface.

- Peak smoothing

Singularities always cause only local disturbance in the inner forces, they do not influence the inner forces at adequately short distance from the location of singularities. The "adequately short distance" is defined by the national standards. In the zones causing substantial changes three solutions can occur according to the codes: the peak can be cut (1), or it can be approached with a linear function (2), or a constant value may be set above the substantial area (3). In the last solution, the capacity of the inner force figure above the area may become equal with the capacity of the original figure. The last solution is the safest one, so it is accepted by every standard.

The peak smoothing algorithm is available (**for internal forces and stresses in planar elements**) in every FEM-Design modules work with planar objects.

Peak smoothing region

The program defines peak smoothing regions to solve the possible singularity problems. Basically, these regions are the active zones in the environment of the singularity, where the inner forces change substantially as a result of mesh refinement.

Peak smoothing regions can be generated automatically by the mesh generator or calculation processes. Automatic generation always results circular peak smoothing regions with centre points placed in the location of the singularity. The radius of a circular smoothing region depends on the geometry of singularity locations.



Peak smoothing regions with any arbitrary shape can be defined manually with the *Peak smoothing region* command. That command is able to edit predefined (automatically or manually) peak smoothing regions.

Automatic generation of peak smoothing regions can be set and controlled at the *General* settings of mesh generation (*Settings> All...> FEM> Mesh> General*). At *Peak smoothing region around...* option you can set the places (depending on the current module) where you want the program to create circular peak smoothing regions. The radius of the circular regions is calculated from the following formula:



Figure: Settings of automatic peak smoothing generation

$r = t/2 + f^* v,$

where:

t is the characteristic geometric parameter of the object that causes singularity:

- 0 value in case of supports (point, line and surface),
- the diameter of circle circumscribed of a cross-section, if bar elements connect to planar elements,
- the thickness of the plate or wall, if the peak smoothing region generated in plate/wall connection; *v* is the thickness of the planar element (plate, wall) in the considered place;

f is a factor can be set manually. The default value is 0.5, which means 45 degrees angle of projection starts from the connection (singularity cause) and ends in the calculation plane of the related planar element (see the figure before).



Figure: Examples for peak smoothing regions by different element-plate connection

Peak smoothing algorithm

The steps of the peak smoothing algorithm are the followings during calculations (inner forces):

1. The program creates peak smoothing regions and/or checks the predefined active zones.



Figure: Generation of peak smoothing region at column-plate/wall connections

2. Allow peak smoothing algorithm for internal force and stress calculations. It is not enough to generate peak smoothing regions, so you have to confirm the smoothing process in the *Calculate* dialog before starting any analysis (and design) calculations.

Calculations	$\overline{\mathbf{X}}$
Calculations Analysis Analysis Analysis Ded cases Imperfections Load combinations Maximum of load grou Stability analysis	Analysis Finite element types Standard elements with 4/3/2 nodes. Short calculation time.
Eigenfrequencies	 d→b Non linear effects in load combination calculations ✓ Consider non-linear behavior of supports, trusses and connections Consider cracking of RC elements
< <u> </u>	Internal forces
Recalculation	OK Cancel

Figure: Peak smoothing algorithm set for analysis calculations

3. The program calculates a constant value for cutting the peaks according to volume calculations of inner diagrams above the peak smoothing regions. That means, the volume at the final constant result value (*Volume (smooth)*) is equal with the volume derived from the peak (singularity) value (*Volume (peak)*) above the same peak smoothing region. Let's see the next figure.



Figure: Peak smoothing algorithm (modified inner force diagram)



Figure: Internal force "graph" and "section" diagrams after using peak smoothing algorithm



Although peak smoothing is available for internal and stress calculations of planar elements, you can solve the singularity peak problem of line reactions and line connection forces. The program calculates the average value of the reaction and connection forces by finite element. That means, line reactions and connection forces can be displayed with constant (average) value by element (*Distribution > Constant by element*). In this case, you can easily place *numeric values* onto the



Figure: Singularity of line reaction force solved by simple display technique

Edit Functions

There are numerous tools in the Finite elements tabmenu allow you to edit the finite element mesh generated automatically.

A short summary of the edit functions:

- adding additional nodes to the mesh (*Fixed point*),
- fixing lines inside the mesh (*Fixed line*),
- editing peak smoothing regions or definition of new ones (*Peak smoothing region*),
- fixing the node numbers on fixed lines (Division number),
- modifying the element number on bar elements (Division number),
- modifying the average element size of 2D finite elements (Average element size),
- refining mesh manually (*Refine*),
- splitting elements to modify finite element geometries (*Split*),
- modifying node positions (*Move node*),
- deletion of mesh regions (*Delete*).

After editing finite element mesh, you can do automatic mesh *object merge, rebuild, smooth, check* etc. processes with special tools of the Finite elements tabmenu. Mesh generation can be done by planar regions with the *Generate* tool.

Adding nodes to mesh

With **Fixed** *point*, nodes can be manually added to a predefined mesh in given points.

Click *Prepare* to update the mesh according to the new nodes. Depending on *mesh settings*, the program automatically does refine, smooth and check processes to create an optimal mesh, if it is possible.



Figure: Adding nodes to a mesh

Fixed points defined in the model can be displayed with the Geometrical system object layer.

Fixing lines of mesh

With Fixed line, lines (straight and curved) can be fixed in a mesh to generate nodes on it.

Click Prepare to update the mesh according to the new nodes. The program defines nodes in the line endpoints and some points on the lines. Depending on *mesh settings*, the program automatically does refine, smooth and check processes to create an optimal mesh, if it is possible. Further node distribution of the fixed line element can be influenced by the *Division number* command.



Figure: Adding lines to a mesh

Fixed points defined in the model can be displayed with the *Geometrical system* object layer. In brackets, the number of line elements can be seen defined by the *Division number* command. If "-" is displayed, there is no restriction on the element number.



Fixed lines display the intersection lines of structural regions (e.g. intersections of slabs and walls) too. This is very useful for defining holes/openings that connect accurately to finite element meshes (see the next figure).



Figure: Fixed lines used for editing regions

Editing peak smoothing regions

With Peak smoothing region, the predefined peak smoothing regions can be edited (status, shape and size modification), or new ones can be defined manually.

The following functions can be done with the *Peak smoothing region* command:

- Modifying the geometry of predefined peak smoothing region

Automatic generation of peak smoothing region always results circular geometry of regions. If you would like to create regions with custom shape, draw a new one with the *Define* tool and the different shape tools. If the new region covers an automatic one (and it is active, see later), the program will use the new region instead of the automatic one for solving the singularity.



Figure: Manually drawn peak smoothing regions with custom shape

Sometimes, you need to correct the geometry of automatic regions to avoid the creation of incorrect or too dense finite element mesh (nodes are generated in the intersections and on the edges of peak smoothing regions).



Figure: Geometry modification effect on finite element mesh generation

Creation of new peak smoothing regions manually With the *Define* tool, totally new peak smoothing regions can be drawn with custom shapes, too.

- Modifying the status of a predefined peak smoothing region

Apply the *Properties* tool and the *Inactive* option for peak smoothing regions to modify their status from active to inactive. Inactive region will not be considered in mesh generations and in

calculations too. To reset an inactive status of a region to an active one, apply the *Properties* tool together with the *Active* option for the region.



Figure: Difference between active and inactive peak smoothing regions

Peak smoothing regions defined in the model are stored in the *Peak smoothing regions* object layer.

Click *Prepare* to update the mesh according to the new and modified peak smoothing regions. Depending on *mesh settings*, the program automatically does refine, smooth and check processes to create an optimal mesh, if it is possible.

It is not enough to generate/define peak smoothing regions, so you have to confirm the smoothing process in the *Calculate* dialog before starting any analysis (and design) calculations.

Division number

Division number sets:

- the number of line finite elements of bar elements (columns and beams),
- the number of 2D element-edges on *fixed lines* and structural region edges.

In case of bar elements, the *Division number* command can modify their default element number (value in brackets) set at *"Elements" setting*.



In 3D modules, the meaning of division number depends on the element type sets at calculations. For example division number 2 sets 2 line elements for *standard* element type and 4 line elements for *accurate* element type. The line elements together with their node numbers can only be displayed after analysis (or design) calculations (see *display settings*).



Figure: The meaning of division number for bar elements

In case of fixed lines and structural region edges, the *Division number* command can fix the **minimum number** of 2D elements (sides) will be connected to the lines/edges by the mesh generator. "-" label in brackets shows no number-restriction for lines/edges.



Figure: The meaning of division number for fixed lines and edges

Defined division numbers together with fixed lines, edges and structural lines of bar elements are displayed with the *Geometrical system* object layer (red by default).

Modifying average element size

The Average element size command modifies the optimized element size(s) of 2D elements calculated by the program according to *mesh settings*.

Applying the command for a structural region (wall, plate etc.), a dialog shows with "*Automatic*" label, that the optimal element size calculation is the active by default for the region. Typing an arbitrary element size (in *a* field) overwrites the usage of the optimal size. Of course, the final average element size generates by the mixture of the value set by *Average element size* and the effect of *mesh settings*. Inactivating all mesh generation automatism (refine, smooth and check processes together with "*Elements*" settings), the *a* value will be the average element size for the related structural regions.

Clicking *Calculate automatically* button shows the optimized average element size used when "*Automatic*" is set for the related object region(s).



Figure: "Automatic" and custom average element size

Click E Prepare or Generate (for the region only) to update the mesh according to the average element size setting.

Refine mesh manually

A predefined finite element mesh can be refined manually. Apply the *Refine* command to refine mesh parts by picking structural objects connect to the meshed region or by clicking directly finite elements.

- Refine mesh around objects

Activate first the *Mark element mode*. Pick the checkbox of object types (e.g. *Region border*), which you would like to allow refining the mesh around. Select objects (e.g. Walls) assigned to allowed object types on the drawing area with *Pick object*, or click *Mark elements around all objects* to refine mesh around all objects belongs to the checked object types. Click to start the refine process.



Figure: Refined finite element mesh around columns

- Refine 2D elements

Activate first the \square *Mark element mode*. Select elements of a mesh on the drawing area with \square *Pick element,* or click \square *Mark all elements* to refine all finite elements of the project. Click \square to start the refine process.



Figure: Refined finite elements (element-selection)

To deselect previously selected elements (before clicking to start refine), switch to *Unmark element mode* and do the same steps mentioned for selection.

Splitting elements

2D finite elements can be refined according to splitting patterns, too. Start the Split command, then chose a cut pattern, and finally select an element which you would like to apply the selected cut mode. Each pattern has a short description about its usage for rectangular and triangular 2D finite elements. At some splitting patterns, red "+" shows the click position in an element to get the requested splitting shapes.



Figure: Element refinement manually with Split

Modifying node positions

The position of mesh nodes can be modified with the *Move node* command. Select node(s) you would like to move, and define the displacement vector with a start- and endpoint. The program sends an error message, if it finds nodes being out of legal moving range.



Figure: Modifying node positions

Deletion of mesh regions

With the Delete command, predefined mesh of selected planar object (regions) can be deleted. Just select the required region(s) and the program erases the mesh.

If you modify the geometry of an object region (e.g. by inserting a hole, by stretching one of region corners etc.), mesh assigned to it will be automatically deleted.

Object merge

The *object merge process* can be run any time you want by clicking the *Object merge* command button.

Rebuild

After using manual tools (mentioned before) to edit the mesh generated automatically before, it is recommended to rebuild the mesh. The process rebuilds the current mesh of selected object regions according to the global optimum without the movement of mesh nodes.

The steps of the rebuild process are:

- 1. The program builds a mesh from triangular elements using the principles of Delaunay triangular technique.
- 2. It converts these triangles to quadrates corresponding to the global optimum of the mesh.

If the *Rebuild* command results drastic changes in the mesh, *smoothing* algorithm is also recommended. The best, the most optimal finite element meshes can be achieved with the iterative usage of the two commands.

Smooth

Smooth process calculates the optimal coordinates of the corner element nodes. Smoothing the mesh is highly recommended after *splitting* or *merging* elements. If the mesh is modified drastically after *rebuild*, use smoothing for the model. The best, the most optimal finite element meshes can be achieved with the iterative use of the *rebuild* and the smoothing.

Just apply the *Smooth* command for selected mesh regions. Mesh smoothing is executed with iteration technique: the procedure places the nodes of the triangular elements in such a way, that the area of the triangles will be balanced. The number of the smoothing iteration steps can be set at *"Functions > Smooth surface mesh" settings*.



Undo cannot restore the mesh state before using *Smooth*.

Check

After using manual tools (mentioned before) to edit the mesh generated automatically before, it is recommended to check the mesh. It can be done according to *"Functions > Check surface mesh" settings* any time you want by clicking the Check command button.



Figure: Topological errors found by Check

Generate (vs. Prepare)

Automatic mesh generation can be done by planar object region with the Generate command. (Prepare runs automatic mesh generation for the entire structural model.) Generate takes into consideration the manual editing functions (such as Fixed point, Fixed line, Peak smoothing, Division number and Average element size) and the Mesh settings excluding the Check, the Smooth and the Prepare settings. Generate is also recommended to find defective geometric finite elements by checking (generating mesh) regions by regions.



Figure: Mesh generation by regions (Generate)

Mesh generation is an iteration process, of which goal is to generate optimal and eurhythmic mesh by the given or automatically calculated average element size. The main and general steps of the generation process are the following:

- 1. Defining the node positions of elements.
- 2. Creating triangle elements by nodes.
- 3. Converting triangle elements to mixed quadrate mesh where it is possible.
- 4. Optimization of node coordinates (e.g. smoothing the mesh).
- 5. Definition of middle points on element sides.

Renumbering and Display Settings

Renumbering

Mesh generator automatically add numbers for finite elements and nodes. If you edit the mesh (e.g. adding new nodes and elements, mesh refinement etc.), you can rerun the renumbering process with the *Refresh numbering* command of the *Tools* menu.

Display settings

The display style of node symbols and *peak smoothing regions*, the numbering of nodes and finite elements can be set at *Settings > All... > Display > Mesh*.

Settings		×
Settings Environment Daving Environment Environment Environment Environment Environment Display Drawing elements Display Drawing elements Display Drawing elements Display Display Display Display 	Mesh Nodes None Circle Circle Circle Size [mm]	Peak smoothing regions Hatch dist. [mm] 2.5000
Save as default		OK Cancel

Figure: Display settings of mesh items

- 2D finite elements

Although 2D finite elements are displayed automatically after mesh generation (if the *Surface elements* object layer is active), the element numbers (*Number*) can be displayed only after *analysis* or *design* calculations. The nodes can be displayed with circle or cross symbols any time, but with numbers (*Number*) after analysis and design calculations.



Figure: Finite elements displayed with numbers and nodes

- Line finite elements

Line finite elements together with their numbers (*Number*) can be displayed only after *analysis* or *design* calculations, if the *Line elements* object layer is active. The nodes can be displayed with circle, cross symbols or with numbers (*Number*).

The color of the elements, nodes and peak smoothing regions depends on the color of their own object layer.

Error Handling

Warning and error messages assist you when problems appear in mesh creation functions, during analysis or design calculations. But, how you can find the position of these cautions to solve them later?

The program points the geometrical and mesh errors, the load misplacements and any other problems in the model, and it collects their coordinates (in the *Global coordinate-system*) in an error/warning dialog.



Figure: Geometrical errors detected by the average element size calculation

ANALYSIS

The calculation type "Analysis" means main static and dynamic calculations for the current model. Depending on the current FEM-Design module you can do different calculations: displacement, internal forces, stresses, stability, imperfections, stability analysis, eigenfrequencies and/or seismic analysis. Some extra settings such as cracked-section analysis, non-linear behavior etc. are also available for certain modules.

Analysis type/settings				F	→ ↓ ↓	1
Analysis for load cases		$\langle \rangle$	\Diamond		\diamond	
Analysis for load combinations	\mathbf{i}	\mathbf{r}	\Diamond		\mathbf{i}	$\langle \mathbf{x} \rangle$
Analysis for maximum of load groups	\mathbf{i}	\mathbf{r}	\Diamond		\Diamond	
Imperfections					\diamond	
Second order analysis					\mathbf{i}	
Stability analysis					$\langle \rangle$	
Eigenfrequencies		$\langle \rangle$	\Diamond		\diamond	
Seismic analysis					\diamond	
Non-linear behavior	\mathbf{i}	\diamond	\Diamond		\diamond	
Cracked-section analysis			\diamond			
Peak-smoothing algorithm						

Table: Analysis features by FEM-Design Modules

Analysis can be done independently from any design calculations by entering to Analysis tabmenu and clicking *Calculate* command, or together with *designs* (RC, Steel or Timber) with the same command.

Structure Loads Finite elements Analysi	s RC design Steel design Timber design	Structure Load	ls Finite elements Analysis	s RC design Steel design Timber design
🔲 + 1↓ · 🖞 ነም		+	↑↓ - 😭 1.00	
Calet Manage results	Quick change Equilib	Calcu	Manage results	Quick change
Colculations	$\overline{\mathbf{X}}$	Calcula	ations	
Calculations	Analysis		Calculations	Analysis
✓ Load cases Imperfections Load combinations Maximum of load grot Stability analysis Eigenfrequencies Seismic analysis	Finite element types Image: Construct of the system of the syst		Load cases Imperfections Load combinations Maximum of load grou Stability analysis Eigenfrequencies Seimic analysis Design calculations	Finite element types Standard elements with 4/3/2 nodes. Short calculation time. Non linear effects in load combination calculations Consider non-linear behavior of supports, trusses and connections Consider cracking of RC elements
	Internal forces Peak smoothing			Internal forces Peak smoothing
Recalculation	OK Cancel	Rec	calculation	OK Cancel

Figure: Analysis calculations

Analysis settings contain general and calculation-type settings. This chapter summarizes these settings and their effect result. Clicking *OK* runs Analysis according to the settings and selected calculation types. Other chapters introduce the *display of results* and their *documentation* (such as listing results in tables).

General Analysis Settings

Finite Element Types

In the 3D modules, you can choose between "standard" and "accurate" 2D *element types*. With standard elements you can run 4-times faster but less accurate analysis than with the fine elements.

Non-Linear Behavior

Non-linear behavior of supports (e.g. uplift), connections and truss members (e.g. tension-only) can be considered in analysis calculations (for load-combinations, imperfections and stability).



Figure: Non-linear behavior chosen for Analysis

E CAR

"*Uplift*" can be modeled both in 2D and 3D design modules by defining compression-only *support/connection* (tension = 0 (free)) and by checking the *Consider non-linear behavior of supports, trusses and connections* box at *Calculate> Analysis*.

Possible divergence can be issued in calculations (iteration algorithm) by setting the non-linear behavior wrong separately by the components.

There is a possibility for the user to set the maximum iteration number of nonlinear calculation in *Calculation* dialog.

Calculations		
Calculations Calculations Calculations Calculations Cod cases mperfections Cod combinations Cod	Analysis Finite element types Fine elements with 9/6/3 nodes. Longer calculation time. Consider non-linear behavior of supports, trusses and connections Consider non-linear behavior of Setup Setup	Non-linear calculation
Recalculation	Internal forces Peak smoothing OK Cancel	

Second Order Analysis

In the and and modules, 2nd order theory can be applied for load combination calculations of 3D structures. The 2nd order analysis considers the placement of the loads that changes together with the displacement, so it results additional moments derived from the new load positions.

To allow the 2^{nd} order analysis for load combinations, just activate the *Second order analysis* checkbox at *Calculations* > *Analysis* > *Load combinations*.

Calculations	
Calculations Analysis Load cases Imperfections V Load combinations	Load combinations
Stability analysis	Load combination Rigd. Calc. U - Ultimate01 Image: Comparison of the second sec
Recalculation	OK Cancel

Figure: Second order theory chosen for load combinations



The 2nd order analysis is recommended to be done together with *imperfection* calculation. In the previous dialog, choose load combinations which you would like to apply the 2nd order theory for, and give the number of imperfection shape (simultaneous or previous calculation for imperfection is needed) you would like to consider for the 2nd order analysis.



If the loads are too large when using second order analysis, the program stops the calculations with an error message.

Cracked-Section Analysis

Cracked-section analysis means that the displacement of RC plates, walls, columns and beams can be calculated based on their cracked state and designed reinforcement.



Figure: Iteration steps of cracked-section analysis

Just, check Consider cracking of RC elements box at Calculate> Analysis.

Í	Calculations		3
	Calculations Calc	Finite element types Image: Consider non-linear behavior of supports, trusses and connections Image: Consider cracking of RC elements Internal forces Image: Peak smoothing	Cracked section analysis One load step in % of the total load
	Recalculation	OK Cancel]

Figure: The "Consider cracking for RC elements" option

Cracked-section analysis will be done for each load-combination of the project. It is recommended to design applied reinforcement, before considering cracking of RC elements.

Click *Setup* to influence iteration process:

- One load step in % of the total load (= number of load steps):
 For example, 20% means 5 load steps (= 100/20[%]). Less percentage generates more steps and more running time.
- **Maximum iteration number:** The value must be in range 1 and 100.
- Allowed displacement error [%]: Iteration ends, when the relative displacement error becomes less than the allowed value.

Peak Smoothing

To solve *singularity problem* in analysis results (internal forces), it is not enough to create *peak smoothing regions* in the finite element mesh. You have to allow the use of the peak smoothing algorithm in the calculations. Without that permission, peak smoothing regions cause only mesh refinements (densifications) around objects.



Analysis for Load Cases and Combinations

Analysis calculations can be done by load case and/or load combination. The next table summarizes the results available for load cases and load combinations by FEM-Design modules.

Calculations	
Calculations Analysis Analysis Analysis Doad combinations Maximum of load gro Stabily analysis Eigenfrequencies Seismic analysis	Load cases
Recalculation	OK Cancel

Figure: Starting analysis for load cases and/or load combinations

Analysis result					→ → ↓ ↓	i
Displacements	(Plate/Beam)	(Wall)	(Wall)	Ì		(Wall/Column)
Reactions			\mathbf{k}		\mathbf{i}	\diamond
Connection forces			\diamond		\Diamond	\diamond
Bar internal forces	(Beam)					(Column)
Shell internal forces	(Plate)	(Wall)	(Wall)			(Wall)
Bar stresses	(Beam)					(Column)
Shell stresses	(Plate)	(Wall)	(Wall)			(Wall)

Table: Basic analysis results by FEM-Design Modules

Displacements

Depending on the current FEM-Design module, the program calculates and displays the model displacement from linear or non-linear (for RC elements: *cracked-section analysis*) analysis. For bar elements, the motion and rotation components can be displayed separately (*Detailed result*) by direction (*local axis*).

In *Plate*, the displacement is calculated for the plate regions and beam elements, and the motion is parallel with the global Z direction, so perpendicular to the plate regions. Only reactions can be asked for columns (point reaction) and walls (line reaction).

In *Wall* and *Plane Strain*, the motion is parallel with the calculation plane of the wall regions.

In *PreDesign*, although the 3D model contains all types of elements, displacement is calculated for the vertical elements such as columns and walls.

Displacement results are recommended to be asked for *serviceability load combinations*.

Reactions

Depending on the support types, the program calculates the reaction forces and/or moments in the *supports* by direction component, their resultants and the resultant at the support's center of gravity of line and surface supports.

The *Plate* module calculates reactions in columns and walls too above the point/line and surface supports.

The available result components:

Fx' / Fy' / Fz'	- reaction force in the local $x'/y'/z'$ axis of the support (<i>group</i>);
Fr	 resultant of the reaction force components (support group);
F	- reaction force of the <i>single support</i> ;
Mx' / My' / Mz'	- reaction moment around the local $x'/y'/z'$ axis of the support (group);
Mr	- resultant of the reaction moment components (support
group);	
M	- reaction moment of the <i>single support</i> .

Connection Forces

Similarly to reactions, the program calculates the forces and/or moments in the connection objects (*Edge connection, Point-point connection* and/or *Line-line connection*) by direction component and their resultants.

The available result components:

Fx' / Fy' / Fz'	- connection force in the local $x'/y'/z'$ axis of the connection;
F	- resultant of the connection force components;
Mx' / My' / Mz'	- connection moment around the local $x'/y'/z'$ axis of the connection;
М	- resultant of the connection moment components.



The figure shows an example for displaying connection forces at the three connection types. The Fy' is displayed at the line (line-line and edge) connections, and the Fx' and Fz' for the point-point connection. (The color of a result component (e.g. Fy') is the same with the color of the local axis (e.g. y') associated to the component direction). On the next figure shows the resultants.



Figure: The resultants (force and moment) for edge connection and line-line connection forces

Bar Internal Forces

The program calculates internal forces and/or moments in the bar elements depending on the applied FEM-Design module.

The available result components:

Ν	- normal (axial) force (local x' axis of the bar element);
Ty' / Tz'	- shear force in the local y'/z' axis direction of the bar
element);	
Mt	- torsion moment (around the local x' axis of the bar
element);	
My' / Mz'	- bending moment around the local y'/z' axis of the bar element.

Truss members bear only normal forces (N).

The *Plate* module calculates internal forces only for beams. Columns are point supports. In *PreDesign*, although the 3D model contains all types of elements, internal forces are calculated in columns.

Shell Internal Forces

Depending on the current FEM-Design module, the program calculates internal forces and/or moments in the planar structural elements

The <i>Plate</i> module c	alculates internal forces in the <i>plate</i> regions and in the <i>Global coordinate system</i> :
Mx'/My'	- bending moment around the global Y/X axis ;
Mx'y'	- torsion moment;
Tx' / Ty'	- shear force for the global X / Y normal and in the Z direction;
M1 / M2	- principal moments;
M1/M2	- principal moment directions.

Although Analysis calculations give results for the global Descartes system, internal forces can be asked and displayed in arbitrary (reinforcement) directions by checking *design forces* in case of RC design.

The *Wall* module calculates internal forces in the *wall* regions and in the *Global coordinate system*:

	0
Nx' / Ny'	- normal force in the global X / Y direction;
Nx'y'	- shear force in the global X-Y plane;
N1 / N2	- principal normal forces;
N1/N2	- principal normal directions.

The *Plane Strain* module calculates only the *shear stresses* in the *wall* regions and in the *Global* coordinate system.

The 📴 3D Structure module calculates internal forces and moments in the planar object regions (plate and wall) in their local coordinate system:

M x' / M y'	- bending moment around the local y' / x' axis of the region element;
Mx'y'	- torsion moment;
Nx' / Ny'	- normal force in the local x' / y' axis of the region element;
Nx'y'	- membrane shear force;
Vx' / Vy'	- shear force for the local x' / y' normal and in z' direction;
M1 / M2	- principal moments;
M1/M2	 principal moment directions;
N1 / N2	- principal normal force;
N1/N2	- principal normal directions.

The *PreDesign* module calculates the previous internal forces and moments but only walls.

Bar Stresses

FEM-Design calculates the normal stress in bar elements (beams, columns and/or truss members) with the following meaning:

Sigma x'(max)	 maximal normal stress (tension);
Sigma x'(min)	- minimal normal stress (compression).



The Plate module calculates stresses only in beams. Columns are point supports.

In *PreDesign*, although the 3D model contains all types of elements, stresses are calculated in columns.

Shell Stresses

The program calculates stresses in the top, bottom and middle (so called "membrane") planes of the planar elements. The meaning of top and bottom side depends on the position (*Plate* module) or the *local coordinate system* (3D modules) of a region element.



Figure: The meaning of planes depending on region position

So, depending on the current FEM-Design module, we get results from the following:

Sigma x', top / membrane / bottom	- normal stress from <i>Nx</i> ′ in top/membrane/bottom plane;
Sigma y', top / membrane / bottom	- normal stress from <i>Ny</i> in top/membrane/bottom plane;
<i>Tau x'y', top / membrane / bottom</i>	- shear stress from $Nx'y'$ in top/membrane/bottom plane;
Tau x'z'	- shear stress (x' normal and z' direction);
Tau y'z'	- shear stress (y' normal and z' direction);
Sigma vm, top / membrane / bottom	 von Mises stress in top/membrane/bottom plane;
Sigma 1/Sigma 2, top / membrane / bottom	- principal stresses and directions in top/membrane/
	bottom plane.

The x', y' and z' directions are valid in the global coordinate system at *Plate* and in the local coordinate system of planar elements in the 3D modules.

In *PreDesign*, although the 3D model contains all types of elements, stresses are calculated in walls.

In *Wall* and *Plane Strain*, stresses are calculated only in the membrane plane.

Equilibrium Check

The program automatically checks the equilibrium of the analysis calculations. Statical equation is written to the origin [0; 0; 0] of the *Global Coordinate System*. It compares the sum of the reactions and the sum of applied loads. Equilibriums can be asked by load case and load combination.

Just click the *Equilibrium* icon (in Analysis or *Design* mode), choose a load case or load combination to see the equilibrium check results.

Equilibrium	$\overline{\Delta}$				\square
Check O Load cases O Load combin	nations	Load cases / cr ULS1 ULS2 SLS1 SLS2 SLS2	ombinations	Close	
Component Fx [kN] Fy [kN] Fz [kN] Mx [kNm] My [kNm] Mz [kNm]	Loads 0.00000 0.00000 -11630 -2181246 3866318 0.00000	Reactions 0.00000 0.00000 11630 2181246 -3866318 0.00000	Error [%]		
	\downarrow				

Figure: Equilibrium check of Analysis calculations

If equilibrium error derives from an analysis calculation, the error will be appeared in percentage in the Error column by equation types (force (F) and moment (M)) and directions (x, y and z directions of the global coordinate system). "Error" shows the differences between the resultants of the queried loads and the calculated reactions.

Analysis for Maximum of Load Combinations and Groups

Choosing *Load combinations* for Analysis automatically generates results for the maximum of load combinations too.

If you define *Load groups* and choose *Maximum of load groups* for Analysis, FEM-Design calculates maximum/minimum results (in all finite element nodes) from the most unfavorable combinations of the load groups according to the applied code.

So, maximum and simultaneous results of *displacements, reactions, connection forces*, internal forces (*bar* and/or *shell*) and stresses (*bar* and/or *shell*) can be calculated for maximum of load combinations and groups.

The symbol "+" and "-" sign the direction of the maximal value in the valid systems: local or global coordinate systems (depend on the current FEM-Design module). Some examples for the meaning of "+" and "-":

Displacement:

Displacement	•
ez'(+)	- maximal uplift in global z' direction in 🞽 Plate,
	- maximum motion in the positive direction of element's local system in 3D modules;
ez'(-)	- maximal depression in global z' direction in 2 <i>Plate,</i>
	- maximum motion in the negative direction of element's local system in 3D modules;

Internal forces:

Mx'(+) - maximal bending moment around the y' axis (global/local) in positive direction (= same direction with the axis direction).

Mx' (-) - maximal bending moment around the y' axis (global/local) in negative direction (= opposite direction to the axis direction).

The next figure shows the meaning of simultaneous results.

Display result		2	
Shell internal forces	Mx'+	Display result	Mx'+
- My'- - Mxy'+ - Mxy'- Nx'+ - Nx'- Ny'+ - Ny'-	Mx My Mx' Mx' Ny' Ny' Nx' Tx' Ty'z'	- My'+ - My'- - Mxy'+ - Nx'y'- - Nx'+ - Nx'- - Ny'+ - Ny'-	Simultenous Mx' My' My' Mxy' Nxy' Tx2' Tx2' Ty2'
Maximal Mx'		Nx' belongs to n	aximal Mx'

Figure: The meaning of simultaneous results belong to a maximal value

Combination of *Load cases* that gives the maximum analysis results in *Maximum of load groups* can be listed in tables. Just use the *List* command (*Tools* menu) for the *Maximum of load groups* result data.



Figure: Combination of load cases for maximum of load groups results

Imperfections

Imperfection calculation is run only for steel bar elements of the structure. Users can add imperfections to a structure in two ways:

- Imperfection modeled by defining loads (manual)
 Place for example horizontal point and line loads on a multi-storey building to model imperfection manually.
- Imperfection calculation according to the formula EC3: 1-1 (automatic)

For load combinations, the program can calculates the probable imperfect shapes in real dimensions from the mode shapes (get from *stability analysis*) according to Eurocode. *Second order analysis* must be run by using imperfection. To do automatic imperfection calculations, activate *Imperfections*

and set the required number of the imperfect shapes (*Rqd.* cell) for the load combination which you would like to run imperfection for.

Calculations			×
Calculations	Imperfections		
	Shapes		
✓ Load combinations Maximum of load grou	Load combination	Rqd.	Calc. 🔼
Stability analysis	U - Ultimate 01	2	0
Eigenfrequencies	U - Ultimate 02	0	0
Seismic analysis	S - Serviceability	0	0
	· · · · · · · · · · · · · · · · · · ·		
<			<u> </u>
Recalculation	OK.		Cancel

Figure: Imperfection calculation by load combination

For the automatic imperfection calculation you got the buckling shape of the structure with real size in real dimension. Critical parameter assigned to a buckling shape is also available with the following meaning:

critical parameter = critical buckling force/actual load

or in other words:

if the critical parameter is bigger than 1, the structure or a part of it is sufficient to perform the stability analysis; if it is smaller it is not.

If the critical parameters differ a lot between the buckling lengths, the first buckling shape is the critical. If the critical parameter values are close to each other, it is your decision what structural part you check by its shape.

The factor defines the real imperfect shape, so:

imperfect shape in real dimension = factor * buckling shape

Eurocode (NA: British) code: Imperfections - Ultimate 01 - 1.shape - [mm]



Figure: Automatic imperfect shape calculation


Before imperfection calculation, it is recommended to set minimum 4-5 *division numbers* (finite elements) for bars.

Stability Analysis

In 3D modules, global stability of the structure can be analyzed automatically if it is requested. Similarly to *Imperfections*, the program calculates buckling shapes together with their critical parameters for selected load combinations.

To do stability analysis, activate *Stability analysis* and set the required number of the buckling shapes (*Rqd.* cell) for the load combination which you would ask stability results for.

Calculations	Stability analysis			
Imperrections	Ducking snapes			
Load combinations	Load combination	Rqd.	Calc.	^
Stability analysis	U - Ultimate 01	3	0	
Eigenfrequencies	U - Ultimate 02	3	0	
Seismic analysis	S - Serviceability	3	0	
Recalculation	С		Canc	

Figure: Stability analysis by load combination

As result, you got the buckling shape(s) of the structure with unit dimension. The greatest displacement value of the buckling shape is 1 and the others are the ratio of that.

Critical parameter assigned to a buckling shape is also available with the following meaning:

critical parameter = critical buckling force/actual load

or in other words:

if the critical parameter is bigger than 1, the structure or a part of it is sufficient to perform the stability analysis; if it is smaller it is not.

Eurocode (NA: British) code: Stability analysis - Ultimate 01 - 1.shape (230.426) - [mm]

Results Analysis Cat cases Analysis Analys	Stability analysis
Buckling shapes Load combination Rqd. C U · Ultimate 01 3 U · Ultimate 02 3 S · Serviceability 3	alc

Figure: Buckling shape calculation

Before stability analysis, it is recommended to set minimum 4-5 *division numbers* (finite elements) for bars.

Eigenfrequencies

FEM-Design can do dynamic analysis by calculating vibration shapes of the structural model and the belonging eigenfrequencies and free vibration time values (periodic time).

To do dynamic calculation, activate *Eigenfrequencies* and just set the required number of the vibration shapes (*Number of shapes* cell), the active masses in X, Y or Z direction and the *Top of substructure*.

Calculations	
Calculations Analysis Load cases Imperfections Load combinations Load combinations Stability analysis V Bgerfrequencies Seismic analysis	Eigenfrequencies Number of shapes Onsider masses in eigen frequency calculation in the following global direction(s): Image: Ima
۰ III >	
Recalculation	OK Cancel

Figure: Dynamic calculation

Dynamic calculation requires *masses* to be predefined.

Seismic analysis needs the eigenfrequencies calculations.



Results of Eigienfrequencies calculation:

Masses

- mass matrix of point masses and/or masses calculated from load cases converted into finite element nodes are displayed by green circle.
 - mass center of masses is displayed by a red circle.

Vibration shape - vibration shape and associated eigeinfrequency (Frequency) and periodic time (Period).



Figure: Results of dynamic calculations



Before dynamic analysis, it is recommended to set minimum 4-5 division numbers (finite elements) for bars.



To get the whole structure's mass centre position set the level of the *Top of the substructure* a bit under the structure.



By setting "Top of the substructure" the foundation mass can be neglected in the eigenfrequency calculation, so the total mass contribution in Modal analysis can reach $\geq 90\%$.

Seismic Analysis

Methods

In the 🍘 and 🛄 modules, seismic calculation offers several methods to the users according to the chosen code: Eurocode 8 or Turkish seismic code.

Seismic analysis based on Eurocode 8 offers the following methods:

Lateral force method / Equivalent static load method _ This method can be used to calculate the seismic effect in horizontal plan, x' and/or y' direction. The main point is to calculate "base shear force" taking into account the base vibration period and design spectrum in x' or y' direction, which is distributed into those nodes of the structure where there are nodal masses. The "base shear force" formula is taken from *EC-8* 4.3.3.2.2(1)*P*. The "base shear force" is nothing else than the total seismic force of inertia that acts between the ground and the structure, and it can be distributed in two ways:

• Linear shape method (Static, linear shape)

The distribution of the "base shear force" happens according to a simplified fundamental mode shape, which is approximated by horizontal displacements that increased linearly along the height.

- Mode shape method (Static, mode shape)
- Modal response spectrum analysis ("Modal analysis")

Seismic analysis based on Turkish seismic code offers the following methods:

- Equivalent seismic load method (similar to Linear shape method)
- **Mode superposition method** (similar to "Modal analysis")

See the detailed description and the applied theory of all calculation methods in the *Theory book*. This guide introduces only the user interface and the steps of seismic analysis.

Steps of Seismic Calculation

The suggested steps of seismic calculation are the followings:

1. Mass definition

To calculate the seismic effect, it is necessary to know the vibration shapes and corresponding periods (except the *Static, linear shape* method). To perform dynamic calculations, it is necessary to define mass distribution which can be defined as *concentrated mass* or *load case-mass conversion*.

2. Design spectrum definition

The program contains predefined *design spectra* according to current stanard, but you can also define your own spectra. Use the command *Seismic load* (*Loads* menu).

3. Dynamic calculation

Dynamic calculation should be done before performing seismic calculation, which gives sufficient vibration shapes of the structure. Although setup for the seismic calculation can be done at any time, but the seismic calculation could be performed only after *Eigenfrequency* calculation. Run dynamic calculation under *Analysis* by setting the required number of vibration shapes.

 $\sum_{i=1}^{N}$

It is suggested to set the finite element number bigger than 1 at bars (*Finite elements/ Division number*).

4. Settings of seismic calculation

A national code always provides which seismic calculation method has to be performed for different structure, where and when it should be performed and what other effects to be considered (e.g. torsional effect, P- Δ effect). *FEM-Design* provides three types of calculation methods for Eurocode 8 and two types for Turkish seismic code, which can be set in the "*Seismic analysis, Setup*"/"*Method*" dialog.

Calculations						
Calculation Calculation Calculation Calculation Calculation Calculations Calculations Calculations Maximum of load grou	Sei	ismic analysis , Method Options	setup Moda	l analys	sis	
Seismic analysis Seismic analysis Design		1. Static, linear shape	No No	Alfa [deg] T[s] 0.495	0.000 mx'[%] 58.2	Set my"[%] 19.4
Load combinations Maximum of load grou Checking		Ħ	2	0.478	19.5 0.0	58.5 0.0
Load combinations Maximum of load grou		2. Static, mode shape	5	0.205	0.0	0.0
<u> </u>		3. Modal	7 Summati SRSS	0.097 on rule by di $6: E_E =$	$\frac{0.6}{\sqrt{\sum E_{j}^{2}}}$	1.9 2 Bi

Figure: Settings of seismic analysis

Static, linear shape (EC8); Equivalent seismic load method (Turkish seismic code)

As a matter of fact, eigenfrequency calculation is not necessary for this method, because giving the base period time in x' and y' directions (Tx' and Ty') is enough for the calculation. Practically, eigenfrequency calculation performs before setting this data, but these data can be defined using experimental formulas as well. Investigation can be done in x' or y' direction, or both together.

ethod Option	s					
Taraz	Static	, linear s	hape			
Ħ	<u>∧</u> ×	Alfa [º]	0.000	1		
Static,	Lambda	Lambda x'		Lambda y'	1.00	
linear shape	U Tx		. 1.00] ту		1.00
H	No	T[s]	mx'[%]	my'[%]		Selart

You may set the calculation direction to be performed by selecting the desired direction. To set the desired x'-y' direction, you should give *Alfa* (α is the angle between the global *X* and *x'*; see *Direction of the horizontal effect*). $\alpha = 0.0$ means x'-y' directions coincide with global *X*-Y directions.

This method is unusable, if the whole foundation is not in same plane or the horizontal foundation is elastic. In these cases, *Static, mode shape* or *Modal analysis* should be used.

Static, mode shape (EC8)

In this method the distribution of "base shear force" happens according to fundamental mode shapes (base vibration shapes).

Options	04-41				
E r	Static, mode sr				
	Alld [*]	1.00	Lambda v	Auto	1.00
Static, linear shape			combodd y	*	Salact
ITTTT					Clear
				3	Lical
Static,					
H					
Madal					
analysis				-	

The table shows how to select the base vibration shapes. It contains all mode shapes (*No*), the vibration time (*T*(*s*)) and effective masses of the mode shapes in x' and y' directions (mx'(%)) and my'(%)). The effective masses are given in a relative form to the total or reduced mass of the structure. The reduced mass means the total mass above the foundation or above the rigid basement. The value of the effective mass refers to how the mode shape responds to a ground motion direction, so the effective mass shows the participation weight of the mode shape.

Select (or double-click on it) one mode shape in x' or/and y' direction(s) (mx' / my'). (Yellow field color shows the activation.)



It is recommended to select that mode shape which gives the largest effective mass as the fundamental mode shape.



The calculation of "base shear force" is performed according to the total mass of the structure and not to the effective mass.

- Modal analysis; Mode superposition method (Turkish seismic code)

The essence of the method is the calculation of the structural response for different ground motions by the sufficient summation of more vibration shapes. Method gives possibility to take into account full x', y' and z (=global Z) direction investigation.

In the table, more vibration mode shape could be selected in x', y' and z directions if necessary. The last row of the table shows that how large is the sum of the considered effective masses compared to the total or reduced mass of the structure in a given ground motion direction.



According to *EC8*, sum of the effective mass of the choosen mode shapes (at least in horizontal direction) should reach 90% of total mass. Additionally every mode shape has to be taken into account where effective mass is larger than 5%.



If the sum of the effective mass is much smaller than 90%, eigenfrequency calculation should be done for more shapes in order to reach 90%.

Lots of mode shapes should be ensured to reach the 90% of total mass in vertical direction. It is highly recommended to check the national code, whether it is necessary to examine the vertical effect or not.

The mode shapes which have small effective mass may be neglected, because their effect in result is very small, but calculation time increases.

Signed results

The results of seismic analysis can be signed results. This is relevant in Modal analysis (in Turkish seismic code in Mode superposition) where the Equivalent seismic loads are calculated from the Vibration shapes and summarized with SRSS or CQC combination rule, where the sign of the forces are lost. In order to get signed results, the user has to select dominant shapes in the two perpendicular directions (x' and y') and check the *Signed results* combo box, so the seismic forces will have the signs of the dominant shape's sign in x' and y' direction.



Figure: Settings of modal analysis (in Eurocode)

Summation rule by directions

According to the *EC8*, the summation rule in the individual directions can be selected. In all other codes always the *SRSS* rule is used for summation (there is no choice). Read more about *SRSS* and *CQC* summation rules in *Theory book*. If the *Automatic* is selected, the rule selection procedure is as follow:



- Direction of the horizontal effect
- Always three directions are investigated (if more than one mode shape is selected in a column), where all mode shape is independent from each other or not.
- If at least one dependent situation exists in a direction, the program automatically uses the *CQC* rule for all mode shape in that direction, otherwise *SRSS* rule is used.

Codes generally speak about seismic calculation in *X*-*Y* directions. These directions give the maximum effect, if the mass and elastic properties of the structure ensure that the calculated mode shapes lay in *X*-*Z* or *Y*-*Z* plane. But it is not always achieved in practice.

To achieve the unfavorable direction, where the results of ground motion are maximum, the program gives the possibility to set x'-y' direction for the seismic horizontal effect (*Alpha*). The program suggests the *Alfa* value, if you click on *Auto* button. It finds the most unfavorable direction, where any of the mx' and my' is zero and the other is maximum in the same row (same shape). But, there is a rule: the direction can be ensured only for one mode shape, so the program selects the row where the effective mass is the maximum. If manually definition is chosen, give an angle for *Alfa* and press the button *Set*.

T

On the left hand side figure you can see a badly adjusted x'-y' direction (*Alpha* = 0). Appling *Auto* button, the program arranges the direction for the 58.5% effective mass my' and correct it to 78%.

Seismic analysis	, setup	Seismic analysis, setup						Seismic analysis, setup						
Method Options	Method Options Initial "Wrong" direction							Method Options		Aft	ter "A	uto"		
	Moda	analys	is	_	_				Moda	analys	sis	_		
	<mark>∕×</mark> ×′	Alfa [deg]		_)>	Auto	E ff			X	Alfa [deg]	30.0	Se	t Auto	Eff. mass >
Static,	No	T[s] 🍃	mx'[%]	mu'[%]	mz[%] 🔨	Sele		Static,	No	T[s] 🌙	mx"[%]	mu'[%]	mz[%] 🔥	Select
linear shape	1	0.495	58.2	19.4	0.0			linear shape	1	0.495	77.7	0.0	0.0	Clear
777	2	0.478	19.5	58.5	0.0	Lice		111	2	0.470	0.0	78.0	0.0	
\square	3	0.302	0.0	U.U	0.0				3	0.302	0.0	U.U	0.0	
<u> </u>	4	0.206	0.0	0.0	10.4				4	0.206	0.0	0.0	10.4	
Static,	5	0.205	0.0	0.0	62.3			Static,	5	0.205	0.0	0.0	62.3	

Figure: Settings of Alpha

Effective mass

The modal effective masses can be compared to the total mass or reduced mass at *Eff. mass*:



In *FEM-Design "Reduced mass"* means the difference between the total mass of the structure and the basement mass. The basement mass is the sum of all masses which lay on the foundation level (set at *Seismic load/ Others*).

EC8 defines the total mass without basement (*Reduced mass*). The effective masses are generally compared to the *Reduced mass*, but this is not valid for the massive basement with elastic foundation. If the above mentioned situation is the case, it might happen that the sum of the effective masses of a column is larger than the 100%.

It is uninterested in the calculation point of view, that effective masses are compared to the total or the reduced mass, because these values are given in percentage and only gives information, that which mode shape is the fundamental or which shapes are dominant in a given direction.

At Options, more calculation properties can be set:



Combination rule

The combination rule of effects in the x', y' and maybe z directions can be set here. You can choose from two possibilities.

Signed results

The seismic results are added to the static results component by component (see in Modal analysis).

Torsional effect

The program gives the possibility to take into account the accidental mass distribution of the structure by the calculation of the torsional effect. This means that from the horizontal seismic forces a Z directional torsional moment can be calculated. See the detailed description of these effects in *Theory book*.

The *Options* menu is different in Turkish Seismic code. Selection of *Signed results* is recommended.

Method Options		
Signed result		
Gravity load combination f	or PBD	
		Ŧ

Figure: Seismic calculation options in Turkish seismic code



5. Seismic calculation

After choosing a calculation method and setting its properties, activate first *Seismic analysis* under *Analysis* and then press *OK*.

The Results

Besides *displacements, reactions, connection forces* and *internal forces*, the program calculates the *Equivalent loads* and the "*Base shear force*". Results can be displayed by vibration shape (selected at calculation settings) from sums by direction and from the total sum (*Seismic max*). If equivalent loads are displayed, also the "base shear force" appears on screen (in grey color). Torsional moment effect on the whole structure can also be displayed, if torsional effect was taken into consideration during calculation.



Figure: Results of Seismic analysis

- In case of Eurocode code, if the Structure type is Building (Loads/Seismic load):
 - Base/Level shear forces (for levels),
 - Second oder effects (P-delta) (for storeys).
- In case of Seismic code: Turkish, if the Structure type is Building (Loads/Seismic load):
 - Torsional Irregularity Amplifying factor (for storeys),
 - Base/Level shear forces (for levels),
 - Second order effects (P-delta) (for storeys).



Because of the square combination rule, the results summed by direction (Sum, x', Sum, y', Sum, z) and the total sum (Seismic max) give only positive values, so absolute maximums. Also, because of combination rule, the displacement components and the internal forces in one point are not simultaneous results.



Seismic results will be added to Seismic max. load case, if the seismic loads are Unsinged. The seismic results will be added to the static results with their sign.

 \triangle

In case of some special structures signed results may be incorrect if the user doesn't choose the dominant shape properly. We suggest using Seismic max. results in this case.

Seismic analysis results (Equivalent loads, Displacements, Reactions, Connection forces, Bar internal forces, Shell internal forces) are also rearranged by the 5 cases of earthquake effect.

e sults ⊷ Analysis Ė∵ Eigenfrequencies	Equivalent	oads	
···· Wasses ···· Vibration shapes ···· Seismic analysis	Shape, dir.	F [kN]	-
Equivalent loads	1.shape, Fx+Mx	18	
Reactions	1.shape, Fx-Mx	18	E
- Connection forces	4.shape, Fx+Mx	107	
	4.shape, Fx-Mx	107	_
. Shell internal forces	6.shape, Fx+Mx	42	
	6.shape, Fx-Mx	42	
	9.shape, Fx+Mx	31	
	9.shape, Fx-Mx	31	
	Sum, Fx+Mx	120	
	Sum, Fx-Mx	120	+

Figure: Display results of Seismic analysis

Summary of Static and Seismic Effects

Seismic effect can be combined with static loads in two ways:

- By defining special *load cases* which contain equivalent seismic loads as real static loads,
- By adding the maximum seismic effect to load combinations or load groups.

Seismic loads as load cases

The x' and y' directional loads (also torsional moments) equivalent to the horizontal ground motion can be converted to load cases. "*Seis load,..."-type load cases* behave as static loads: they can be combined, they can be added to load groups and combinations, and they can be taken into consideration at stability, imperfection and design calculations. As you see in the list of load case types, the seismic effects can be considered with positive and/or negative sign.

No	Name	Туре	Duration class (EN 1995 1-1)	*	ОК
1	Wind load X	Ordinary	Permanent		Cancel
2	Wind load Y	Ordinary	Permanent		
3	Deviation X	Ordinary	Permanent		Import / Export >
4	Deviation Y	Ordinary	Permanent		
5	Snow load	Ordinary	Permanent		Insert
6	Imposed load	Ordinary	Permanent	E	Insert
7	Dead load	+Struc. dead load	Permanent		Delete
8	Seismic load	Ordinary	 Permanent 		Delete all
		Ordinary +Struc. dead load +Soil dead load +Shikage +Seis load, Fx-Mx +Seis load, Fx-Mx +Seis load, Fx-Mx			

Figure: Seismic effect added as load case

Seismic effect in load combinations

According to Eurocode 1998-1-1, 4.3.2 Chapter and TS-Eartquake 2.7.3.2 and 2.8.2.2, the masses are shifted to + 5% and - 5% of the building size perpendicular to earthquake direction, so the seismic analysis results are separated to 5 cases, Fx + Mx, Fx – Mx, Fy + My, Fy – My, Fz. In Load combinations dialog these 5 cases and Seismic max. load case can be added to Load combinations.

Start the command *Load combinations* (*Loads* menu). Apply *Insert* under *Load case* on a predefined or new load combination, choose a case, define a load factor and press *OK*.

2 ULS1y	U 1.00 Dead load Cancel 1.00 Imposed load Cancel Import / Export 1.30 Wind load X Import / Export Import / Export U Factor 1.00 Load combination U Snow load Load combination
2 ULS1y	U Factor
2 ULS1y	0.70 Snow load 1.30 Wind load X 1.00 Deviation X U Factor
2 ULS1y	1.30 Wind load X 1.00 Deviation X U Factor 1.000 Snow load Imposed load Load combinat
2 ULS1y	1.00 Deviation X U Factor Snow load U Load combinat
2 ULS1y	U Factor
	Factor
	Snow load Load combination
	I Imposed load
	Dead load Generate
	Seismic load
3 ULS2X	Seis res, Fx+Mx
	Seis res, Fx-Mx Seis res, Fv+Mx
-	Seis res, Fy-M Delete
-	Delata all
4 ULS2y	U OK Cancel Delete all
	1.30 Snow load
	0.70 Wind Icad Y Insert
	1.00 Deviation Y
	IVEW
	- Remove
combinations Name T	Type Factor Induded load cases A OK
	U 1.00 Dead load
1 ULS1x	1.00 Imposed load Cancel
1 ULS1x	1.00 Inposed load
1 ULS1x	0.70 Snow load
1 ULS1x	0.70 Snow load 1.30 Wind load X
1 ULS1x	0.70 Snow Ivad 1.30 Wind kad X 1.00 Seis res, Fx+Mx
	1.00 Impose load 0.70 Snow load 1.30 Wind load X 1.00 Sets res, Fx+Mx 1.00 Deviation X
1 ULS1x	1.00 Imposed load 0.70 Snow Ibad 1.30 Wind load X 1.00 Sels res, Fx+Mx 1.00 Deviation X U 1.00 Dead load
1 ULS1x	
1 ULS1x	1.00 Imposed load 1.30 Wind load X 1.00 Sels res, Fx +Mx 1.00 Devator X U 1.00 Imposed load 1.00 Imposed load 0.70 Snow load Generate
1 ULS1x	1.00 Imposed load 1.30 Wind load X 1.00 Sels res, Fx +Mx 1.00 Devation X U 1.00 Imposed load 1.00 Imposed load 0.70 Snow load 1.30 Wind load Y
1 ULS1x	1.00 Imposed load 1.30 Wind load X 1.00 Selis res, Fx +Mx 1.00 Deviation X U 1.00 Imposed load 1.30 Wind load Y 1.30 Wind load Y 1.00 Deviation Y
1 ULS1x 4	1.00 Imposed load 1.30 Wind load X 1.30 Wind load X 1.00 Deviation X U 1.00 Deviation X U 1.00 Imposed load 1.30 Wind load Y 1.30 Wind load Y 1.00 Deviation Y U 1.00 Deviation Y

Figure: Adding seismic load case in Load combination dialog box

Fx + Mx means that the earthquake force is parallel to x' direction, and the masses are shifted to result positive Mz' moments.

M3

To add -Fx -Mx, -Fy-My or -Fz earthquake effects to a load combination, the Factor of the Load case (Fx + Mx, Fy + My, Fz) has to be negative.

Seismic effect as load group

The above mentioned seismic effects can also be added to groups in all codes. Define a group as "Seismic".

No	Load group	Included load cases	-	OK
1	Permanent (Permanent, 0.90, 1.10, 1.00, 1.35, 0.85)	Dead load		
2	Live (Temporary, 1.35, 0.70, 0.50, 0.30)	Imposed load		Cancel
3	Seismic (Seismic, 1.00, Load cases)	Seismic max.		Import / Export >
	Name Type			Combination metho
	Safety factor 1.00			EC0 6.10 EC0 6.10.a, b
	Calculate maximum of		- =	Load group
	Seismic calculation results Output User defined load cases			Insert
				Delete
				Delete all
			_	Load case
				Insert
	OK Cancel			New
		- 2	-	Remove

Figure: Maximum seismic effect defined as load group

- In case of Unsigned seismic analysis:
 - The program automatically takes the "*(Seismic max)*" into consideration with +/- values in the generation of the most unfavorable results (all components of the seismic results are added to the components of static results with the sign of static component)
- In case of Signed seismic analysis:
 - If Seismic calculation results option is selected, the seismic results are added to the static results component by component
 - If User defined load cases option is selected, the program combines the static "+Seis load..." type load cases with the static loads.

Investigate

If a warning message appears during calculation (e.g. Load mismatch or Finite element mesh problem) there is a possibility to check and fix the error by navigating in the model with *Investigate*.

The following pictures show a badly defined load and how can the user check and fix the error with *Investigate* function.



DESIGN

Depending on the current FEM-Design module you can do different design calculations for concrete, steel and timber model elements. This chapter summarizes the design possibilities and results by design type.

Design Load

All design calculation works with load combinations and load groups.

Calculations	$\overline{\mathbf{X}}$
Calculations Analysis Analysis Load cases Imperfections Load combinations Maximum of load grou Stabily analysis Eigenfrequencies Seismic analysis Design calculations	Design calculations Calculation Auto design all structural elements Calculation Calculations Based on analysis of Load combinations Load combinations Load groups
< · · · · · · · · · · · · · · · · · · ·	
Recalculation	OK Cancel



Figure: Design calculation settings

Although analysis can be run for load combinations and load groups in the same time, design calculation results exist only either for the load combinations or the maximum of load groups.

The *Shell design* can be done only for load combination, because it based on stability results.

Serviceability load combination is considered only in RC bar and shell design, other design is done just for ultimate load combinations.

Buckling Length Factors

Buckling length is claimed input data of all bar element design (RC/steel/timber columns, beams). The buckling length is determined from the bar length and a factor depends on bar end connections. For typical end conditions (hinged, fixed, cantilever etc.), proposed *beta* factors are available for flexural buckling.



Figure: Proposed beta factors for flexural buckling

Buckling length of a *Truss member* element is equal to its length, so buckling length of trusses cannot be modified.

The default *beta* factor is 1.0 for both flexural and torsional buckling.

The defined buckling length fundamentally influences the design results.

Design Groups

Elements having certain common properties can be assigned to a *design group*, and the members in one group will be designed in the same way. For example, uniform reinforcement (longitudinal bars and stirrups) will be calculated for RC bars assigned to one design group, or the same profile will be designed for each steel bar of a design group, etc.

General terms to be a design group member:

- Same design element type (beam, column, truss member, plate, wall or timber panel)
- Same material
- Same geometry (sizes, profiles etc.)
 - Same design parameters (e.g. base reinforcement)

Depending on specific cases/element types, additional conditions have to be materialized:

- Same support conditions
- Same end/edge connections
- Same load, etc.

In design mode, click 💷 to group elements under a name and color.

Design group	×
	lame A

Figure: Design group tools

Design group tool provides the following group operations:

- ____ creates a new group,

- edits the properties (name and color) of a selected group,
- + adds selected elements to the current group,
- removes selected element from the group,
- **!** is available only in *Surface* and *Punching reinforcement* design. A group member has to be set as the "*Master*" that carries visually the representative applied reinforcement designed uniformly for the group. Only the "Master's" applied reinforcement is editable; symbolic reinforcement is displayed for the other group members.

explodes/deletes a selected group.

The next figures give examples of grouping same type steel bars, concrete slabs and beams.



Figure: Some design groups of a steel frame



Figure: Design groups of concrete slabs and beams

From Auto Design till Final Design

FEM-Design covers the whole RC, steel and timber design process by starting with automatic predesign (socalled "*Auto design*") for the entire structure and/or by elements/*design groups*, then by continuing with detailed *manual design* of the domain structural components and by ending with the final design (so-called "Check") of the entire structure.



Figure: From predesign of components till final global design

Although analysis can be done in one step for the entire structure independently of the element types (concrete, steel and timber), design calculations (and so the design modules) can be run separately by the element types (e.g. one after the other).

- Apply design changes and recalculate: one-click validation of design changes in the current entire model together with analysis and design recalculation according to the new states. But, leaving Design mode, the design changes are always updated in the current model.
- Delete all applied quantity: restores the model state previous to the applied design according to following settings dialog:



Auto Design

Based on internal forces, utilization check and initial design settings, FEM-Design finds quickly and automatically the most suitable applied quantity depending on the design category and element type. The so-called "*Auto design*" makes a *proposal* for reinforcement distribution, for the applied profile, for the panel type, etc.

Auto design can be done for the whole project with the *Calculate* > *Design calculations* > *Auto design all* structural elements. The fast algorithm uses the default calculation and design parameters of the elements and search for the most suitable applied quantity according to the utilization. *Auto design* automatically runs *Analysis* calculations (for *Load combinations* or *Load groups*), for *Stability* etc. depending on the design type: e.g. *Steel bar design* or *Shell design*.



Auto design can be applied for single elements and/or design groups only (without running design for the complete model).

The next table summarizes the initial design parameters and the results of *Auto design* by design categories and element types.

Category	Element type	Design parameter	Auto design
RC design	Bar reinforcement	Steel quality, bar diameter	Bar and stirrup distribution
		and profile, concrete cover	(numbers of bars, spacing)
	Surface reinforcement	Steel quality, bar spacing,	Bar distribution, applied
		shape settings of the	reinforcement regions and
		reinforcement regions,	area, and bar diameter
		possible bar diameters	
	Punching reinforcement	Steel quality, range of	Bar and stirrup sizes and
		diameters, distribution	distribution
		shape (bended bar, circular	
		stirrup, open stirrup)	
Steel design	Steel bar	Range of cross-sections	Suitable cross-section
	Shell model	Range of plate thicknesses	Suitable plate thickness
Timber design	Timber bar	Range of cross-sections	Suitable cross-section
	Timber panel	Range of panel types	Suitable panel type

Table: Input parameters and the results of element-based Auto design

Auto design gives summary tables, which display both the initial design parameters and the recommended (applied) quantities optimized to maximum utilization. All utilization details done by the Eurocode 2, 3 or 5 regulations can be also displayed.

Fast redesign can be started from the table by modifying the design parameters.

design		Util	izati	ion								
J	Calculation			Group	Desig	in param	eters	Total w [t]	veight	Max [%]	Min [%]	^
\mathbb{Z}	Load combinations		B	lars C	d = 16,	8, 8,			0.084	82	82	
			/ B	ars B	d = 16,	8, 8,			0.204	93	93	
	🗹 Display table		B	lars A	d = 16,	8, 8,			0.124	96	96	≡
			B	.10	d = 16,	8, 8,			0.042	82	82	
			B	.9	d = 16,	8, 8,			0.124	96	96	
			/ B	.8	d = 16,	8, 8,			0.042	82	82	-
			B	.6	d = 16,	8, 8,			0.124	96	96	~
				Bar		Max [%]	SEC [%]	ST [%]	C [%]	T [%]	CW [%]	•
			/ В.	.4		93	62	93	11	0	0	
			ИВ.	.1		93	62	93	11	0	0	
		Ŀ	_									
			_									~
		F	aran	meters Desi	gn	Dele	te			< Hid	le detail	s

Figure: Summary table of Auto design (Bar reinforcement)

If the program is not able to find suitable parameters from the given/available design parameter range, it sends a warning message and marks the problematic elements/groups.

Leaving *Design* mode applies all design changes by updating the current model in order to be considered in a next recalculation. It also invalidates previous analysis results.

In RC modules, *Auto design* results applied reinforcement that is considered in case of cracked-section analysis.

Manual Design

Following *Auto design*, "on-the-fly" fine tuning of applied quantities can be done by elements and design groups. In RC design, *Manual design* completes the applied reinforcement editing task for both concrete surface and bar elements.

Category	Element type	Possible initial data	Manual design
RC design	Bar reinforcement	Reinforcement come from	Applied reinforcement
_		Auto design	(longitudinal bars and stirrups)
	Surface reinforcement	Reinforcement come from	Applied reinforcement
		Auto design	(top/bottom/middle RC)
	Punching reinforcement	Reinforcement come from	Applied reinforcement
		Auto design	(bended bar, circular stirrup,
			open stirrup)
Steel design	Steel bar		Applied cross-section
	Shell model		Applied plate thickness
Timber design	Timber bar		Applied cross-section
	Timber panel		Applied panel type

T

Table: Input parameters and the results of Manual design

The next figure gives an example for the reinforcement design of a concrete beam from starting with *Auto design* and finishing with detailing (*Manual design*).



Figure: Combined RC design

Leaving *Design* mode applies all design changes by updating the current model in order to be considered in a next recalculation. It also invalidates previous analysis results.

In RC modules, *Manual design* results applied reinforcement that is considered in case of cracked-section analysis.

Detailed Result

Utilization results with detailed background calculation formulas (together with Eurocode references), figures and tables can be displayed by single elements or by design groups. Quick navigation is powered with zooming details.



Figure: Detailed result

Of course, *Detailed result* is also available for the analysis (displacement, internal forces, and stresses) of bar elements.

RC DESIGN

Following the analysis calculation, the applied reinforcement (longitudinal bars, stirrups etc.) can be designed automatically and refined manually for concrete beams, columns, truss members, slabs, walls and shell elements. The applied reinforcement can be considered in cracked-section analysis.

The table summarizes the available RC design features and its analysis-related effect (cracked-section analysis) by FEM-Design module.

Design element type	Design feature				→ ↓ ↓
Bar reinforcement	Auto design				
	Manual design				
	Cracked-section analysis			\diamond	\diamond
Surface reinforcement	Auto design	\diamond	\diamond		
	Manual design		\diamond		\diamond
	Cracked-section analysis		\diamond		\diamond
Punching reinforcement	Auto design	\diamond			
	Manual design	\diamond			\diamond
	Cracked-section analysis	\diamond			\diamond

Table: RC design features by FEM-Design module

Global *Auto (RC) design (Calculate > Design calculations > Auto design all structural elements)* gives applied reinforcement for all concrete elements based on their initial design calculation parameters, *Auto design > Parameters* and optimized to their internal forces and detailed utilization calculations. With *Manual design* you can fine-tune the reinforcement by elements and/or design groups. You can do quick *Auto design* by elements and design groups only instead of global design. Of course, any number of design cycles is executable, so the global *Auto design* can be combined with both previous and additional element-based *Auto designs*.

No.	Global RC design	Element-based RC design	Combined RC design
1	🖼 Design calculation parameters	📟 Global Analysis	📓 Design calculation parameters
2	💷 Design group	E Design calculation parameters	💷 Design group
3	🗲 🤋 Auto design > Parameters	🔤 Design group	差 ? Auto design > Parameters
4	📟 Global Auto design	差 ? Auto design > Parameters	🔤 Global Auto design
5	<i>Manual design</i> by elements	<i>I</i> Auto design by elements	差 🤋 Auto design > Parameters
6	🖎 Apply design changes	<i>Manual design</i> by elements	I Auto design by elements
7	📟 Global <i>Check</i>	🛯 Apply design changes	Manual design by elements
8	📟 Cracked-section analysis	📟 Global <i>Check</i>	🕺 Apply design changes
9	Documentation	📟 Cracked-section analysis	🔤 Global Check
10		Documentation	Cracked-section analysis
11			Documentation

Table: Recommended steps by design alternatives

Bar Reinforcement

Bar reinforcement design needs internal forces from *Analysis* calculations applied for *Load combinations* or *Load-groups*, \mathcal{D}_{ℓ_0} *Buckling length* and initial design settings defined by \mathcal{D} *Design calculation parameters* and 2nd order calculation method by \mathcal{D} *Configuration*.



Figure: Design calculation parameters

RC design configuration							
2nd order effect calculation method							
Nominal stiffness							
Nominal curvature							
OK Cancel							

Figure: Design calculation parameters

The maximal section distance defines the approximate position of the design sections and so the available sections for detailed results.

Auto Design

Global *Auto design* gives utilization results and suitable reinforcement distribution for all concrete bars of the current project.



The applied design parameters can be displayed on screen by showing the "*RC bar, design parameters*" object layer, or click Design tool of the \swarrow Auto design to show the parameters together with a detailed utilization table. Utilization as colored figure (color palette) can be displayed by selecting *New result* > *RC bar* > *Utilization*, and clicking the *Numeric value* tool summarizes the list of maximum utilizations by elements in a dialog.



Figure: Global Auto design and utilization results

Element-based *Auto design* finds the most suitable position and distribution of longitudinal bars and stirrups for selected unique or grouped members only according to their user-defined design parameters. Initial reinforcement (steel quality, diameter, profile, concrete cover) and design (aggregate, vibration) settings can be set for the concrete members/groups with the *Parameters* tool of *Auto design*.



Figure: Design parameters

To run element-based design for the load combinations or the maximum of load groups, select the required members and/or group with the *Auto design* command and click *Design* tool. The quick process results applied reinforcement and their checked utilization. Check the *Display table* box to have a look at the overall design results.

uto design			_				ι	Jtiliz	ation			_					
	Calculation								Group	Desi	gn param	eters	Total v [t	veight	Мах [%]	Min [%]	
47	Load combinatio	ins 💌						\checkmark	B.3	d = 16,	8, 8,			0.124	96	96	5
/ (Display table	`						\checkmark	B.6	d = 16,	8, 8,			0.124	96	96	5
								1	B.8	d = 16,	8, 8,			0.042	82	82	2
	_							1	B.9	d = 16,	8, 8,			0.124	96	96	3
a made								1	B.10	d = 16,	8, 8,			0.042	82	82	2
UCIII	zation							V	Bars A	d = 16,	8, 8,			0.124	96	96	5
	Group	Design parameters	Total weight [1]	Max [%]	Min [%]	<u>^</u>		V	Bars B	d = 16,	8, 8,			0.204	93	93	3
	B.3	d = 16, 8, 8,	0.124	96	96			\checkmark	Bars C	d = 16, 8, 8,			0.084		82	82	2 🗸
	B.6	d = 16, 8, 8,	0.124	96	96				Bar		Max	SEC	ST	r I	т	rw	
, i v	B.8	d = 16, 8, 8,	0.042	82	82				ba		[%]	[%]	[%]	[%]	[%]	[%]	
J	B.9	d = 16, 8, 8,	0.124	96	96			\checkmark	B.3		96	83	96	12	0	0	J
	B.10	d = 16, 8, 8,	0.042	82	82												
, i v	Bars A	d = 16, 8, 8,	0.124	96	96												
J	Bars B	d = 16, 8, 8,	0.204	93	93												
J J	Bars C	d = 16, 8, 8,	0.084	82	82												
P	'arameters De	isign Delete		Shov	v details	Ŋ	H										
								Pa	rameters De	esign	Dele	te			< Hie	de deta	ails

Figure: Quick summary of Auto design results

The upper table shows the design efficiency and the maximal utilization of the designed single members and groups based on the given design parameters. The bottom table (*"Show details"*) displays the utilization details of the bar or the members of the group selected in the upper table.

	Meaning	Note
\checkmark	Suitable reinforcement is available	
•	Suitable reinforcement is not available	Modify bar profile, material or RC
		design parameters
Group	ID of a single bar or a group name	
Design parameters	Applied design parameters	
Total weight	Total weight of applied reinforcement	
Max	Max. utilization of a single bar or the	
	significant member of a group	
Min	Max. utilization of the less significant	
	group member	
Bar	ID of a single bar or a group member	
SEC	Section utilization	According to Eurocode 2: 6.1
ST	Stirrup utilization	According to Eurocode 2: 6.2 and 6.3
С	Concrete utilization	According to Eurocode 2: 6.2 and 6.3
Т	Utilization of torsional reinforcement	According to Eurocode 2: 6.3
CW	Utilization for crack width	According to Eurocode 2: 7.3

Table: The meaning of symbols, design parameters and utilization results

Quick redesign can be done inside the *Utilization* table:

- 1 Select a bar or a design group in the upper table.
- 2 Modify the design parameters of the select element under *Parameters*.
- 3 Click Design.

Applied reinforcement generated by *Auto design* can be displayed with:

- *Detailed result* applied for the displayed utilization result (*New result > RC bar > Utilization*) of a single bar or a group member.
- *Manual design* applied for a single bar or a group member to edit the design reinforcement and/or add further longitudinal bars and stirrups.

Manual Design

Manual design opens a new window in the current project and gives tools to define new (applied) reinforcement in concrete beams, columns and bars, or to modify/edit the reinforcement generated by *Auto design*. The drawing area is divided into two view windows:

Cross-section

It shows the cross-section of the current concrete bar. The definition of new longitudinal bars (sectional position) and stirrups (shape) starts in this window. The position of the cross-section (section view) can be set by moving the section marker 4 in *3D view*.

- 3D view

It shows the side view of the concrete bar by default. But, any 2D and 3D view can be set with the *View* menu commands (e.g. general 3D view with *View* > *Space view*). The start and end pont (and so the length) of longitudinal bars and the position (the distribution) of the stirrups can be defined here.



Figure: Working windows of Manual design

You can choose the required window by clicking inside it or its title.

The *Longitudinal bar* tool defines new bars in given insertion points. Set the main properties of the new bar on the tool palette or all properties under *Default settings*.



Figure: Definition tools and settings of Longitudinal bar

Use one of the following tools to place the new bar in the *Cross-section* view:

1. Clicking in *Cross-section*, the new bar will be placed with its center point.

Align the new bar to a line/edge by select one in *Cross-section*. Move the mouse to set the bar's relative position to the selected line/edge: the center point/upper/bottom/left/right surface will be on the line/edge.



Figure: Placement of a bar aligned to an inner edge of a stirrup

2. Align the new bar to a corner defined by two lines/edges. The bar will be tangent for the first and then the second given line.



Figure: Placement of a bar aligned to a corner of a stirrup

3. Define the group of longitudinal bars by set the number of the horizontal (nc), vertical (nr) bar numbers and the distance between the rows (c)



Figure: Placement of bar group

The steel bar length can be defined manually by giving the bar's start and end point in 3D view.
Bar length is defined by two points
Longitudinal bar



Figure: Steel bar length definition

The *Stirrup* tool defines new stirrups with given shapes. Set the main properties of the new stirrup bars on the tool palette or all properties under *Default settings*.



Figure: Definition tools and settings of Stirrup

Choose the contour geometry of the new bar and define the bar's relative position to the contour with *Alignment*, which also defines the final stirrup shape with the *Cover* value in the *Cross-section* view. In the final step, the distribution of the stirrups based on the *Distance* value is defined with a start and an end point in the *3D view*.



- reinforcement(s) with the *Properties* tool of *Longitudinal bar* and *Stirrup*.
- ✓ To exit from *Manual design* with validating the new and modified reinforcement bars and stirrups, click *OK*.
- **To exit from** *Manual design* without accepting the defined and modified reinforcement, click *Cancel*.

Detailed Result

Utilization of RC bars can be displayed in the following cases:

- After global *Auto design*, you can display utilization of all concrete bars calculated from the suggested applied reinforcement.
- When running element-based *Auto design*, utilization can be displayed by designed elements.
- After *Manual design*, element-based *Check* displays utilization for selected elements.
- After global Check done for all bar elements having final applied reinforcement.

No.	Global Auto design	Element-based Auto des.	Element-based Check	Global Check
1	🔲 Calculate>Design	🗲 Auto design	差 Auto design and/or	🗲 Auto design
	calculation>Auto	_	👤 Manual design	and/or
	design all structural			! Manual design
	elements			
2	🛨 New result>RC	🗯 New result>RC bar	le Check	🖎 Apply changes
	bar			
3			া New result>RC bar	폐 Calculate>Design
				calculation>Auto
				design all structural
				elements
4				📌 New result>RC bar

Table: Steps of displaying RC bar utilization by different design cases

Utilization displayed with *New result* appears for all designed bars. The utilization components for a bar/design group can be displayed with *Detailed result*.

Detailed result opens a new windows in the current project after selecting a bar/group member, which display:

Applied reinforcement

The figure gives the distribution of the applied reinforcemen.



Figure: Longitudinal bar and stirrups data

- Cross-section data

The figure gives the cross-section datas: height, width, area, moment of inertias.

Cross-section



Figure: Cross-section datas

- Material properties

In Materials section, the program shows the material properties of the applied concrete's and reinforcement, e.g. compression strength (f_{ck}), the mean tensile strength (f_{ctm}), mean Young modulus (E_{cm}).

Materials							
C25/30				B420A			
f _{ek}	=	25.00	N/mm ²	$f_{vd} = f_{vwd}$	=	365.22	N/mm ²
f _{etm}	=	2.60	N/mm ²	Ė₅ .	=	200000.00	N/mm ²
f _{сtк,0.05}	=	1.80	N/mm ²	$\epsilon_{yd} = f_{yd}/E_s$	=	0.00183	N/mm ²
Ecm	=	31000.00	N/mm ²	ε _{ud}	=	0.02250	
acc	=	1.00	N/mm ²				
α _{ct}	=	1.00	N/mm [≠]				
Yo	=	1.50					
Ys	=	1.15					
ϕ_{ef}	=	0.00	2				
$f_{cd} = \alpha_{cc} f_{dk} / \gamma_{c}$	=	16.67	N/mm ²				
$f_{\text{ctd}} = \alpha_{\text{ct}} f_{\text{ctk}} / \gamma_{\text{o}}$	=	1.20	N/mm ²				
$E_{od} = E_{om} / \gamma_{o}$	=	20666.67	N/mm ⁺				
ε _{c2}	=	0.00200	(Table 3.1)				
ε _{cu2}	=	0.00350	(Table 3.1)				
		Figu	re: Material	properties			

- Calculation formulas

This contains the EC-3 formulas for checking RC bar elements, the substitutions in the formulas and the calculation table for each cross section. In the table, the amount of cross-section calculations can be set in the Display options.

Concrete utilization for shear and torsion (Part 1.1: 6.2, 6.3)

```
\begin{split} \Theta &= 45^{\circ} \\ \sigma_{qp} &= \min\left(\frac{N_{Bd}}{A_{\odot}}, 0.2f_{cd}\right) \\ \sigma_{cw} &= 1.0 + \frac{\sigma_{qp}}{f_{cd}}, \text{ if } \sigma_{qp} < 0.25f_{cd} \quad (6.11.aN) \\ \sigma_{cw} &= 1.25, \text{ if } 0.25 \leq \sigma_{qp} < 0.5f_{cd} \quad (6.11.bN) \\ \sigma_{cw} &= 2.5 \left(1.0 + \frac{\sigma_{qp}}{f_{cd}}\right), \text{ if } \sigma_{qp} \geq 0.5f_{cd} \quad (6.11.cN) \\ \gamma_{1} &= 0.6, \text{ if } f_{ak} < 60 \text{ MPa} \\ \gamma_{1} &= \max(0.9 - f_{ak}/200, 0.5), \text{ if } f_{ck} \geq 60 \text{ MPa} \\ \gamma_{edmax} &= \frac{\sigma_{cw}}{cot(0) + tan(\Theta)} \quad (6.9) \\ \gamma &= 0.6 \left(1 - \frac{f_{ak}}{25U}\right) \quad (6.6.N) \\ T_{edmax} &= 2v\alpha_{cw}f_{cd}A_{k}t_{r}sin(\Theta)cos(\Theta) \quad (6.30) \\ \text{Utilization: } ma\left(\frac{T_{Ed}}{T_{edmax}} + \frac{\sqrt{V_{Edx}}}{\sqrt{V_{edmaxy}}}, \frac{T_{Ed}}{T_{edmax}} + \frac{\sqrt{V_{Edx}}}{\sqrt{V_{edmaxy}}}\right) \quad (6.29) \end{split}
```

Figure: Formulae

Detailed calculation tables _

Calculation details and final values are collected in tables sorted by checking types and under utilization graphs by default. Column number depends on the number of calculation sections or the table settings defined with **Display options**.

Sections	5	19	23
LC	а	а	а
N _{Ed} [KN]	0.00	0.00	0.00
$ee_{Ed,y}[kN]$	0.00	0.00	0.00
$\vee_{Ed,z}[kN]$	23.58	23.58	37.06
T _{Ed} [kNm]	0.00	0.00	0.00
σ _{cp} [N/mm²]	0.00	0.00	0.00
A _{sl} [mm ²]	402	402	1206
d _y [mm]	84	84	84
К _у [-]	2.00	2.00	2.00
b _{w.y} [mm]	350	350	350
Р _{1. у} [-]	0.01368	0.01368	0.02000
V _{min.y} [N/mm ²]	0.49	0.49	0.49
∨ _{Rd.c.y} [kN]	22.90	22.90	25.99
$(A_{sw,y}/s)f_{ywd}$ [kN/mm]	91.79	91.79	163.18
z _y [mm]	76	76	76
$\forall_{Rd,s,y}[kN]$	22.90	22.90	25.99
$(\vee_{Ed,y}\!$	0.00	0.00	0.00
d _z [mm]	314	314	314
k _z [-]	1.80	1.80	1.80
b _{w,z} [mm]	120	120	120
ρ _{1, z} [-]	0.01067	0.01067	0.02000
V _{min,z} [N/mm ²]	0.42	0.42	0.42

Figure: Detailed calculation table

Stress-strain graphs _

Different colors display the strain (red) and the concrete stress (cyan) curves grouped by Section utilization (ultimate limit states) and Crack width(serviceability limit states). Also the compressed concrete zones are shown with cyan fills in the calculation sections. The number of displayed sections can be set with *Display options*.



Utilization graphs _

Section (Eurocode2: 6.1), Stirrup (6.2, 6.3), Concrete (6.2, 6.3), Torsional reinforcement (6.3) and Crack width (7.3) utilization graphs together with a Summary graph are displayed with legends by default. Numeric values can be inquired in the calculation sections (*Design calculation parameters* sets the maximum distance of sections).



Detailed result *Tabmenu* contains the following tools and settings:

- Selection of element to display

You can choose a unique or a design group member from the drop-down lists to display its detailed results mentioned before. Each row displays the ID and the maximum utilization of a member. In case of a design group, "Maximum" means the significant member having the maximum utilization.

Detailed result			
Unique members	~	B.6	~
Bars A	96%	B.5	82%
Bars B	93%	B.7	82%
Bars C	82%	Maximum	82%
Onique members	96%	-	
		1 B.8	82%
		B.10	82%
		B.6	96%
		B.9	96%
		B.3	96%

Figure: Selection of a unique or a group member

- Selection of design load

Depending on RC design was done for load combinations or load groups, a load combination or the maximum or a significant component of load groups can be selected for detailed results. Each row displays the name of the load combination/load group component and its utilization effect. "Maximum" means the significant load combination or component of load groups.

Detailed result				
Unique members	▼ B.6	~	Maximum	~
			U - load comb01	85%
			U - load comb02	99%
			S - load comb03	20%
			Maximum	99%

Figure: Selection from design loads

Auto design

Quick *Auto design* can be done for the currently displayed unique/group member. Its design parameters can be set/modified in the appearing dialog, and then clicking *OK* starts RC design that updates all detailed result figures and tables.

📕 Manual design

Manual design can be launched directly for the currently displayed unique/group member. Returning from reinforcement editing updates all detailed result figures and tables.

Display options

The content and the appearance of the detailed result can be set with *Display options*. For tables or stress-strain graphs, you can choose all, maximum and characteristic calculation sections to display.

Display options	X
Table of content Reinforcement Cross-section Materials Section utilization Stirup utilization Concrete utilization Torsional reinforcement Crack width Summary	Content options Hide all unrelevant verifications Hide details Hide graphs Hide tables Hide formulae Display results in characteristic sections. Appearance characteristic all Scale (1:?) Font Sideview
Settings >	Close

Figure: Display options of Detailed result

🤳 Go to

Navigate in the *RC bar detailed result window* by selecting the required design type in the dropdown list. It is useful when you are in zoomed view.

Click *Tools > Add view to document* to place all figures and tables or specified details only into *Documentation*.

Surface Reinforcement

Surface reinforcement design needs internal forces from *Analysis* calculations applied for *Load combinations* or *Load-groups* and initial reinforcement properties (direction, shape, steel quality, diameter and concrete cover) defined by Design calculation parameters.



Figure: Initial reinforcement settings

The reinforcement shape (*Straight/Centric*) and the bar directions (x'/y'/r/t) set by *Design calculation parameters* will be fixed parameters in *Auto* and *Manual design*, so only they can be edited/modified only with *Design calculation parameters*.

Although all design results can be calculated for all reinforcement types, *Auto design* does not work for the *Centric* reinforcement! Only *Manual design* can be used to define and edit the required centric reinforcement area.

Auto Design

Global *Auto design* gives design force and required reinforcement results and suitable applied reinforcement for all concrete surfaces of the current project. Furthermore, *Auto design* calculates missing reinforcement and crack width based on the determined applied reinforcement.



Figure: Global Auto design and its result

Element-based *Auto design* finds the most suitable top and bottom reinforcement for selected concrete plate, wall and shell elements or element groups only according to their internal forces and design parameters. Initial settings of the base net and additional reinforcement by positions (*Bottom face/Top*

face) and directions (x'/y'/r/t) can be set with the **Parameters** tool of Auto design.



Figure: Design parameters

The minimum concrete cover together with bar directions is derived from *Design calculation parameters* settings.

To run element-based design for the load combinations or the maximum of load groups, select the required

members and/or group with the *Auto design* command and click *Design* tool. The quick process finds the suitable bar diameter for the additional reinforcement from the defined diameter range, calculates the bars' utilization and distributes top and bottom reinforcement where required.



The placed bars can be displayed by their directions and positions with the navigator icons of the *Auto design* tool palette.

The generated applied reinforcement is also visible in *Manual design*, where additional reinforcement can be defined or the current state can be edited.

In case of design groups, the applied reinforcement appears only at the *Master group member*.

Check the tool palette's *Display table* box to have a look at the overall utilization results given for the designed surface elements or design groups.



Figure: Quick summary of Auto design results

The upper table shows the design efficiency and the maximal utilization of the designed single elements and groups based on the given design parameters. The bottom table displays the utilization details of the surface element or the elements of the group selected in the upper table.

	Meaning
\checkmark	Suitable reinforcement is available
	Suitable reinforcement is not available
	Note: Modify the element thickness, material or RC design parameters.
Group	ID of a single element or a group name
Design parameters	Main applied design parameters
Total weight	Total weight of applied reinforcement
Max	Max. utilization of a single element or the significant member of a group
Min	Max. utilization of the less significant group member
Shell	ID of a single element or a group member
RBX	Utilization of bottom x'/r reinforcement
RBY	Utilization of bottom y'/t reinforcement
RTX	Utilization of top x'/r reinforcement
RTY	Utilization of top y'/t reinforcement
CWB	Utilization for crack width on the bottom face
CWT	Utilization for crack width on the top face

Table: The meaning of symbols, design parameters and utilization results

Quick redesign can be done inside the *Utilization* table:

- 1 Select a surface element or a design group in the upper table.
- 2 Modify the design parameters of the current elements under *Parameters*.

3 If the selected surface elements already contain applied reinforcement defined by Manual design or

earlier *Auto design*, you can delete it by activating on the *Auto design* tool palette.

4 Click Design.

Manual Design

Manual design gives tools to define new (applied) reinforcement in concrete plates, walls and shells, or to modify/redefine the reinforcement generated by *Auto design*.





E C

If needed, applied reinforcement defined earlier by *Auto design* or earlier *Manual design* can be deleted with *Edit* > *Erase*. The geometry of the applied reinforcement regions can be edited with the region-related *Edit* commands such as *Region operations*.

To define new reinforcement in a surface element, first choose the same reinforcement shape (*Straight/Centric*) set earlier for the surface element with *Design calculation parameters*.

Choose the geometry of the new surface reinforcement.

Choose the reinforcement layer with

t	1
+	••

Set the position (*Bottom face/Top face*) of the new surface reinforcement.

Set the direction (x'/r/y'/t) of the new surface reinforcement.

All parameter of the new reinforcement bars can be set under *Default settings* or the main parameters on the *Manual design* tool palette.

According to the geometry place the surface reinforcement with its required points. The new surface reinforcement will be situated parallel with the plane of the host element with the defined concrete cover thickness.



Figure: Additional reinforcement defined with Manual design

U In case of design groups, the applied reinforcement of the *group Master* is editable only.

- You can modify the properties (quality, diameter, spacing, cover) of previously defined surface reinforcement(s) with the *Properties* tool of *Manual design*.
- Running global (*Calculate > Design calculations > Check*) or element-based *Check* the program gives *Applied reinforcement* result, so the applied area of surface elements can be displayed with *Color palette*, *Contour lines*, *Graph* or *Sections mode*.



Figure: Applied reinforcement area displayed with Graph, Color palette, Contour lines, Sections and Bi-direction mode

Shear capacity

FD calculates RC shells (*3D Plate, Wall*) shear capacity and their results can be seen from *New result / RC Design / Shear capacity.* These results can be displayed in *Graph, Contour Line, Color palette* and *Section* format.


Punching Reinforcement

Punching check and design can be done according to Eurocode 2 both in FEM-Design Plate and 3D *Structure* modules. It is recommended to perform after *Surface reinforcement design*, because surface reinforcement influences punching calculations.

The punching regions are automatically defined by the program at the plate – coumn intersections and you can define them manually.

Punching design/check can be done at any point where a punching region is exists. Design and check can be done by these punching zones or by their *design groups*.

The program does not generate punching objects for columns that are connected to more than one slab at the same level.

Initial properties are needed to be set for punching zones. Use Calculation parameters to define the followings:



Figure:Setting $\boldsymbol{\beta}$ values for punching objects

- Beta (β)

According to Eurocode 2, β coefficient is taken into account as the effect of any eccentricity of loading. Its value depends on column position. The program offers default standard values for column positions: for corner (β =1.5), for edge (β =1.4) and inside columns (β =1.15). These standard

values have to de added manually by selected punching zones or by their groups. If you neglect β values, choosing *Calculate automatically* defines them during design/check process and gives accurate results for β coefficients (see *Detailed results*).



Calculation parameters are displayed on the model view by the *Punching, calculation parameters* object layer.

Auto Design

Global *Auto design* gives utilization results for punching. Auto design applies the *initial punching reinforcement settings*.

Punching design always deletes the existing reinforcement and generates new one.

Defining punching regions

When clicking on the $^{\pm\pm}$, the dialog appears:



Figure: Punching region toolwindow

First the origin of the local system should be specified, which is also the calculation point for punching. The punching force is taken from the finite element corner node closest to the local system's origin.

ZQZ

Before analysis, the user should place a fixed point in Finite element tab, where a punching calculation is planned to be performed, because in this way it will be surely a corner node, leading to higher precision in punching calculation.



If you pick a punching force calculation point away from the region the following appears:

FEM-Des	FEM-Design - Question 🛛 🕅					
2	Origin of local system is outside of the convex wrap of punching region. Do you want to continue with this region?					
	<u>Y</u> es <u>N</u> o					

Punching utilization results can be displayed for the entire model with *New result > RC punching > Utilization*. Maximum utilization in table format can be also shown with *Numeric value*. Detailed utilization

results can be asked by unique or grouped punching zones with \swarrow (see later). Different colors display the adequacy of the checked punching zones.



Figure: Global Auto design and its result

Element-based *Auto design* checks the utilizations against punching and finds required reinforcement for selected zones and/or design groups of zones. Initial settings can be set for required reinforcement by bended bar, circularly placed stirrup and open stirrup types with the *Parameters* tool of *Auto design*.



Figure: Design parameters by punching reinforcement type

The symbols of the design parameters are stored on the Punching, design parameters object layer.

To run element-based design for the load combinations or the maximum of load groups, select the required zones and/or groups with the *Auto design* command and click *Design* tool. The quick process runs detailed utilization and finds the suitable bar diameters for the additional reinforcement.

Check the tool palette's *Display table* box to have a look at the overall utilization results.



Figure: Quick summary of Auto design results



The upper table shows the design efficiency and the maximal utilization of the designed punching zones based on the given design parameters. The bottom table displays the utilization details of elements or design members.

	Meaning
\checkmark	RC column is suitable for punching without or with shear reinforcement
	Suitable shear reinforcement is not available
	Note: Modify the slabs' surface reinforcement or the design parameters.
Group	ID of a unique punching zone or design group
Design parameters	The suitable bar diameter and other design parameters as the reinforcement type
Total weight	- The "-" symbol displays that no shear reinforcement is applied
	- Total weight of applied punching reinforcement
Max	Max. utilization of a unique zone or the significant zone of a group
Min	Max. utilization of the less significant group member
Punching	ID of a unique column or a design group member
CC	Utilization for concrete compress
CS	Utilization for concrete shear
RS	Utilization for reinforcement shear

Table: The meaning of symbols, design parameters and utilization results

Quick redesign can be done inside the *Utilization* table:

- 1 Select a punching zone or a design group in the upper table.
- 2 Modify the design parameters of the current elements under *Parameters*.
- 3 Click Design.



Detailed result summarizes the applied formulas by the design modes and display the applied shear reinforcement (if needed) drawing too.

Manual Design

____1

Manual design gives tools to define new (applied) punching reinforcement in concrete plates, or to modify/redefine the reinforcement generated by *Auto design*.

The rules of new reinforcement definition, if predefined reinforcement already exists from Auto design:

- new same-type reinforcement will be added to applied reinforcement,
- new different type reinforcement always overwrites the previously defined one, and
- new "open stirrup"-type reinforcement always overwrites the previously defined reinforcement even if it was "open stirrup".



First choose the type of the new reinforcement. If you would like to modify/refine a previously defined (generated by *Auto design* or defined by *Manual design* in an earlier phase) reinforcement, you

also need to set the required type from the drop-down list of *Manual design*. The *Properties* tool of *Manual design* can be used for modifying actions.

The tools of Manual design depends on the selected reinforcement type.

Bended bar

First set the reinforcement properties under *Default settings*, and choose the bar direction according to the local system directions of the related column. Then select the punching zone you would like to be reinforced (with totally new or additional reinforcement) and place the new bars.



Figure: New bended bar

Stirrup, circular

First set the reinforcement properties under *Default settings*, then select the punching zone you would like to be reinforced, and finally define the circular form with its inner radius.



Figure: New stirrups in circular shape

Stirrup, open

First set the reinforcement properties under Default settings, then select a geometrical shape (e.g. rectangular, circular, polygonal) for the stirrup position. Select the punching zone you would like to be reinforced, and finally define the shape in the model view.



OK

Figure: New open stirrups

In case of design groups, the applied reinforcement of the *group Master* is editable only. Other group members have symbolic reinforcement figure.

Running global (Calculate > Design calculations > Check) or element-based Check the program gives utilization result for the applied punching reinforcement.

Detailed Result

Utilization of punching reinforcement can be displayed in the following cases:

- After global *Auto design*, you can display utilization calculated from the suggested applied reinforcement.
- When running element-based Auto design, utilization can be displayed by designed elements.
- After Manual design, element-based Check displays utilization for selected zones.
- Global Check done for final applied reinforcement.

No.	Global Auto design	Element-based Auto des.	Element-based Check	Global Check
1	🔲 Calculate>Design	🗲 Auto design	差 Auto design and/or	🗲 Auto design
	calculation>Auto	_	! Manual design	and/or
	design all structural			📕 Manual design
	elements			
2	🛨 New result >	া New result >	S Check	🖎 Apply changes
	RC punching	RC punching		
3			🛨 New result >	폐 Calculate>Design
			RC punching	calculation>Check all
				structural elements
4				+ New result >
				RC punching

Table: Steps of displaying punching utilization by different design cases

Utilization displayed with *New result* appears for all designed punching zones. The utilization components with calculation formulas and the applied reinforcement (if needed) can be displayed with *Detailed result*.

Detailed result opens a new window in the current project after selecting a unique zone or a member zone of a design group, which display:

- Design parameters and applied reinforcement

The figure displays the calculation perimeters, the design parameters and the applied reinforcement calculated by *Auto design* (if required) or defined in *Manual design*.



Figure: Calculation perimeters, design parameters and applied reinforcement

- Detailed calculation formulas

Calculation details and final values are collected by checking types: *Concrete compression resistance* (Eurocode2: Part 1.1: 6.4.3), *Shear reinforcement resistance* (Part 1.1: 6.4.3) and *Concrete shear resistance* (Part 1.1: 6.4.3). The proper results are displayed in green, while the red result warnings you to repeat design. The content of the utilization checks depends on *Display options*. Not relevant checks can also be hidden.



Figure: Utilization formulas and tables

- Summary graph

Summary graph is displayed with legend by default.

Sun	nmary Utiliza	tion [%]					
100							
80	_	at 1	at 2	at 3	at 4	_	
60	ressio	shear	shear	shear	shear		
40	comp	ment	ment	ment	ment:	shear	
20	crete	force	force	force	lforce	crete	
0	Col	Reir	Rei	Rei	Reir	S S	



Detailed result *Tabmenu* contains the following tools and settings:

Selection of zones checked for punching

You can choose a unique zone or a design group zone member from the drop-down lists to display its detailed results mentioned before. Each row displays the ID and the maximum utilization of a member. In case of a design group, "Maximum" means the significant member having the maximum utilization.

- Selection of design load

Depending on punching design was done for load combinations or load groups, a load combination or the maximum or a significant component of load groups can be selected for detailed results. Each row displays the name of the load combination/load group component and its utilization effect. "Maximum" means the significant load combination or component of load groups.

Detailed result			
Unique members 🛛 🙀	C.10	~	Maximum 😭
	C.13	94%	
Unique membre	C.7	57%	
Cilique member	C.6	95%	Design load
or design group	C.15	111%	selection
selection	C.10	96%	
	C.9	3%	
	C.4	70%	
	C.14	98%	
	C.12	130%	
	C.8	40%	
	C.1	93%	Unique
	C.11	93%	member
	C.2	94%	or oroug member
	C.3	83%	or group member
	IC.5	89%	

Figure: Selection from checked elements and design loads

Auto design

Quick *Auto design* can be done for the currently displayed unique/grouped zone. Its design parameters can be set/modified in the appearing dialog, and then clicking *OK* starts punching design that updates all detailed result figures and tables.

- 🚽 Display options

The content and the appearance of the detailed result can be set with *Display options*.

isplay options	\mathbf{X}
Table of content Input data Concrete compression Concrete schear Concrete schear Concrete schear Summary	
Content options	
Hide all unrelevant verifica	tions
Hide details	
Appearance	
Font	AaBbYyZz ∙
Figure scale 1	.00
Settings >	Close

Figure: Display options of Detailed result

🤳 Go to

Navigate in the *Detailed result* window by selecting the required design type in the drop-down list. It is useful when you are in zoomed view.

Click *Tools* > *Add view to document* to place all figures and tables or specified details only into *Documentation*.

STEEL DESIGN

Fast auto design and check are available to find the most suitable steel bar profiles. All section shapes and classes - including class 4 profiles (slender sections) - can be checked for utilization. When modeling steel bars with 3D steel shell components, an auto design finds the proper thickness of the shell elements.

The table summarizes the available steel design features by FEM-Design module.

Design element type	Design feature			→ ↓
Steel Bar	Auto design	\diamond	Ì	Ì
	Manual design	Ì	\diamond	Ì

Shell Model	Auto design		\diamond
	Manual design		

Table: Steel design features by FEM-Design module

Steel Bar

Global *Auto (steel bar) design (Calculate > Design calculations > Auto design all structural elements)* finds the most suitable cross-section (from the profile range set at *Auto design > Parameters*) for all steel bars (columns, beams and truss members) based on their buckling length, stiffeners, calculation parameters, internal forces and detailed utilization calculations. With *Manual design* you can run quick utilization check for given profiles by bar elements and/or design groups. You can also do quick *Auto design* by elements and design groups only instead of global design. Of course, any number of design cycles is executable, so the global *Auto design* can be combined with both previous and additional element-based *Auto designs*.

No.	Global steel bar design	Element-based steel design	Combined steel design
1	Dr. 🗉 🚔 Calculation	📟 Global Analysis	📭 🖬 🚔 Calculation
	parameters		parameters
2	💷 Design group	De II 🖙 Calculation	💷 Design group
		parameters	
3	🗲 🤋 Auto design > Parameters	🔎 Design group	🗲 🤋 Auto design > Parameters
4	📟 Global Auto design	🕖 ? Auto design > Parameters	📟 Global Auto design
5	Documentation	差 ! Auto design by elements	🗲 🤋 Auto design > Parameters
6		Manual design by elements	<i>I</i> Auto design by elements
7		🖎 Apply design changes	<i>Manual design</i> by elements
8		📟 Global <i>Check</i>	🖎 Apply design changes
9		Documentation	📟 Global <i>Check</i>
10			Documentation

Table: Recommended steps by design alternatives

Initial Calculation and Design Parameters

All bar design calculations needs internal forces from *Analysis* calculations applied for *Load combinations* or *Load-groups*, $\int_{a}^{b} Buckling length$ and initial design settings defined by E Design calculation parameters.



- Stiffeners can be added manually to steel bars at any time during design (e.g. before *Auto design*, or between *Auto design* and final *Check*). Stiffeners can be defined element by element, but they can be copied among bar elements. Stiffener definition tools are:
- time creates given number of evenly distributed stiffeners on a defined length section,
- creates stiffeners by given distance on a defined length section (in case the multiple of the distance does not fit in the section, the program distributes stiffeners with equal parts at the ends),
- Here defines stiffeners one by one in given position,
- ^m copies stiffeners of a selected bar to other bars with the same length.

You can increase the number of stiffeners in more steps, if you inactivate *(Do not delete the original)*.



Figure: Stiffeners

Auto Design

Global *Auto design* gives utilization results and suitable profiles for all steel bars of the current project.



Figure: Global Auto design and utilization result

The recommended profile names can be displayed on screen by showing the "*Steel bar, applied quantity*" object layer, or click **!** *Design* tool of the *Auto design* and the parameters together with utilization results are available in table format. Utilization as colored figure (color palette) can be displayed by selecting *New result* > *Steel bar* > *Utilization*.



Variable cross-section steel bars can't be designed, just checked.

Applied profiles are displayed in blue in the *Utilization* table, if they are assigned to the steel bars during design, otherwise black color represents the original/initial profiles.

Element-based *Auto design* finds the most suitable profile of steel columns, beams and bars for selected unique or grouped members only from a range of available profiles defined by *Parameters*.



Figure: Range of available profiles for design

To run element-based design for the load combinations or the maximum of load groups, select the required members and/or group with the *Auto design* command and click *Design* tool. The quick process results

recommended profiles and their utilization. Check the Display table box to have a look at the overall design results (see the figure before).

The upper table shows the design efficiency and the maximal utilization of the designed single members and groups based on the given design parameters. The bottom table displays the utilization details of the bar or the members of the group selected in the upper table.

	Meaning	Note
\checkmark	Suitable profile is available	
•	Suitable profile is not available	Modify the range of available profiles
		or steel materials
Group	ID of a single bar or a group name	
Design parameters	The defined range of available profiles	
Applied profile	Profile currently assigned to the bar	
Max	Max. utilization of a single bar or the	
	significant member of a group	
Min	Max. utilization of the less significant	
	group member	
Bar	ID of a single bar or a group member	
RCS	Resistance of cross-section: the maximum	According to Eurocode 3:
	utilization from all strength calculations	6.2.3 - 6.2.10
FB	Utilization for flexural buckling	According to Eurocode 3: 6.3.1
TFB	Utilization for torsional-flexural buckling	According to Eurocode 3: 6.3.1
LTB	Utilization for lateral torsional buckling	According to Eurocode 3: 6.3.2.4
SB	Utilization for shear buckling	According to Eurocode 3: 1-5: 5
IA	Interaction	According to Eurocode 3: 6.3.3
	(between normal force and bending)	

Table: The meaning of symbols, design parameters and utilization results

Quick redesign can be done inside the *Utilization* table:

- Select a bar or a design group in the upper table. 1
- 2 Modify the range of available profiles for the select element under Parameters.
- 3 Click Design.

Manual Design

With Manual design quick utilization check can be done for given steel profile and for selected steel beam, column, bar or design group only. Just, choose the load type (load combination or load group) and a profile name from the drop-down lists and select a bar, bars or group, and program displays detailed utilization results in table format.



m M The meaning of the utilization components, the table content and features are the same as written before at Auto design. The program use the chosen profile for all selected bar elements.



Figure: Quick check by Manual design

Detailed Result

Utilization of steel bars can be displayed in the following cases:

- After global *Auto design*, you can display utilization of all steel bars checked for the recommended profiles.
- When running element-based *Auto design*, utilization can be displayed by designed elements.
- After Manual design, element-based Check displays utilization for selected elements.
- After global *Check* done for all bar elements having final cross-section.

No.	Global Auto design	Element-based Auto des.	Element-based Check	Global Check
1	📠 Calculate>Design	🗲 Auto design	差 Auto design and/or	🗲 Auto design
	calculation>Auto	_	🖳 Manual design	and/or
	design all structural			🤳 Manual design
	elements			
2	🛨 New result>	🛨 New result>Steel bar	Scheck	🖎 Apply changes
	Steel bar			
3			🛨 New result>Steel bar	폐 Calculate>Design
				calculation>Auto
				design all structural
				elements
4				🛨 New result>
				Steel bar

Table: Steps of displaying steel bar utilization by different design cases

Utilization displayed with *New result* appears for all designed bars. The utilization components for a bar/design group can be displayed with *Detailed result*.

Detailed result opens a new window in the current project after selecting a bar/group member, which displays:

- Input data

The figure displays the applied steel cross-section with its main calculation and material parameters.

C.4 Maximum of load combinations



Figure: Applied cross-section

- Detailed calculation formulas

Calculation details and final values are collected by checking types: *Shear resistance* (Eurocode3: 1-1: 6.2.6, 6.2.8), *Torsional resistance* (1-1: 6.2.7), *Shear stress* (1-1: 6.2.6), *Normal stress* (1-1: 6.2.1), *Normal capacity* (1-1: 6.2.1), *Flexural buckling* (1-1: 6.3.1), *Torsional-flexural buckling* (1-1: 6.3.1), *Lateral-torsional buckling* (1-1: 6.3.2.4), *Interaction between normal force and bending* (1-1: 6.3.3) and *Shear buckling* (1-5: 5). The proper results are displayed in green, while the red result warnings you to repeat design with new bar properties. The content of the utilization checks depends on *Display options*. Not relevant checks can also be hidden.



Figure: Utilization checks and formulas

- Summary graph

Summary graph is displayed with legend by default. Numeric values can be inquired in the calculation sections (set by *Design calculation parameters*).



Figure: Utilization summary

Detailed result *Tabmenu* contains the following tools and settings:

- Selection of element to display

You can choose a unique or a design group member from the drop-down lists to display its detailed results mentioned before. Each row displays the ID and the maximum utilization of a member. In case of a design group, "Maximum" means each check is displayed for the significant member having the maximum utilization for that check.



Figure: Selection of a unique or a group member

- Selection of design load

Depending on steel design was done for load combinations or load groups, a load combination or the maximum or a significant component of load groups can be selected for detailed results. Each row displays the name of the load combination/load group component and its utilization effect. "Maximum" means each check is displayed for the significant load combination or component of load groups having the maximum utilization for that check.

Detailed result				_
Unique members	💌 B.25	~	Maximum	~
			Load comb 01	93%
			Load comb 02	74%
			Maximum	93%

Figure: Selection from design loads

Auto design

Quick *Auto design* can be done for the currently displayed unique/group member. Its design parameters can be set/modified in the appearing dialog, and then clicking *OK* starts steel design that updates all detailed result figures and formulas.

📕 Manual design

Manual design can be launched directly for the currently displayed unique member / group.

Display options

The content and the appearance of the detailed result can be set with *Display options*. You can show only the final equation without details of the different checks (*Hide details*).

Display options 🛛 🕅			
Table of content			
🗹 İnput data 🔷			
Shear resistance, y-y			
Shear resistance, z-z			
✓ Torsional resistance - Part 1-1:6.2.7			
Shear stress			
Vormal stress			
Vormal capacity			
Flexural buckling,1			
Content options			
Hide all unrelevant verifications			
Hide details			
Appearance			
Font AaBbYyZz •			
Figure scale			
Settings > Close			

Figure: Display options of Detailed result

🤳 Go to

Navigate in the detailed result window by selecting the required design type in the dropdown list. It is useful when you are in zoomed view.



Click *Tools > Add view to document* to place all figures, fomulas and summary table or specified details only into *Documentation*.

Shell Model

In FEM-Design, a steel bar can be modeled as a real 3D element defined from steel plates.

To convert a bar element to 3D shells apply the *Steel bar, shell model* tool of the **Structure** *Tabmenu*.

Global *Auto (Shell model) design (Calculate > Design calculations > Auto design all structural elements)* finds the most suitable thickness (from a thickness range set at *Auto design > Parameters*) for all steel plates of the shell model based on internal forces, stability check and detailed utilization calculations. With *Manual design* you can run quick utilization check of given thickness values for selected shell models or their design groups. You can also do quick *Auto design* by shell elements and design groups only instead of global design. Of course, any number of design cycles is executable, so the global *Auto design* can be combined with both previous and additional element-based *Auto designs*.

No.	Global steel bar design	Element-based steel design	Combined steel design
1	🛣 Structure>Steel bar,shell	🛣 Structure>Steel bar,shell	🛣 Structure>Steel bar,shell model
	model	model	
2	💷 Design group	📟 Global Stability analysis	💷 Design group
3	🗲 ? Auto design > Parameters	🔤 Design group	🗲 ? Design Parameters
4	📟 Global Auto design	🗲 ? Auto design > Parameters	📟 Global Auto design
5	Documentation	🗲 ! Auto design by elements	🗲 🤋 Auto design > Parameters
6		Manual design by elements	🗲 ! Auto design by elements
7		🛯 Apply design changes	Manual design by elements
8		📟 Global <i>Check</i>	🛯 Apply design changes
9		Documentation	📟 Global <i>Check</i>
10			Documentation

Table: Recommended steps by design alternatives

Stability analysis is required to get utilization check of steel bar-shell models. Global *Auto design* automatically runs stability analysis for the entire structure.

Auto Design

Global *Auto design* gives utilization results and suitable thickness for all steel plates of shell models.



Figure: Global Auto design and utilization result

The recommended thickness values can be displayed on screen by showing the "Steel bar, shell model,

applied quantity" object layer, or click *Design* tool of the *Auto design* and the thickness values together with utilization results are available in table format. Utilization as colored figure (color palette) can be displayed by selecting *New result* > *Steel bar, shell model* > *Utilization*.

Applied thickness values are displayed in blue in the *Utilization* table, if they are assigned to the steel shell parts during design, otherwise black color represents the original/initial thickness.

Element-based *Auto design* finds the most suitable thickness of steel plates for selected unique or grouped shell models only from a range of available thicknesses defined by *Parameters*.



Figure: Range of available thickness values for design

To run element-based design for the load combinations, select the required shell element and/or element group with the *Auto design* command and click *Design* tool. The quick process results recommended

group with the *Auto design* command and click *Design* tool. The quick process results recommended thickness values and the utilization of the shell model elements. Check the *Display table* box to have a look at the overall design results (see before).

The upper table shows the design efficiency and the maximal utilization of the designed single element and groups based on the found thickness values. The bottom table displays the utilization details of the shell model element or the members of a group selected in the upper table.

	Meaning	
~	Suitable thickness is available	
0	Suitable thickness is not available	
-	(Modify the range of available thicknesses or steel materials.)	
Group	ID of a single element or a group name	
Design parameters	The defined range of available thicknesses	
Applied thicknesses	Thickness currently assigned to the shells	
Max	Max. utilization of a model element or the significant	
	member of a group	
Min	Max. utilization of the less significant group member	
Bar	ID of the designed single bar or a group member	
Stress	Stress utilization	
FBS	Utilization for flexural buckling around the stiff axis	
FBW	Utilization for flexural buckling around the weak axis	
ТВ	Utilization for torsional buckling	
LTB	Utilization for lateral torsional buckling	
LB	Utilization for local buckling	

Table: The meaning of symbols and utilization results

Quick redesign can be done inside the *Utilization* table:

- 1 Select a bar or a design group in the upper table.
- 2 Modify the range of available plate thicknesses for the select element under *Parameters*.
- 3 Click Design.

Manual Design

With *Manual design* quick utilization check can be done for custom (also different) thickness values of selected steel plates (or their groups). Just select steel plate(s) and set a thickness value in the appeared dialog. Run *Check* calculation to run and display utilization check for the modified plates according to their new thickness values.

The meaning of the utilization components, the table content and features are the same as written before at *Auto design*.



Figure: Quick check by Manual design

Detailed Result

Utilization of steel bar-shell models can be displayed in the following cases:

- After global *Auto design*, you can display utilization of all steel shell models.
- When running element-based *Auto design*, utilization can be displayed by designed elements.
- After Manual design, element-based Check displays utilization for selected elements.
- After global *Check* done for all bar-shell elements having final plate thicknesses.

No.	Global Auto design	Element-based Auto des.	Element-based Check	Global Check
1	🔤 Calculate>Design	差 Auto design	差 Auto design and/or	🗲 Auto design
	calculation>Auto	_	I Manual design	and/or
	design all structural			Manual design
	elements			_
2	+ New result>	🛨 New result>	S Check	🖎 Apply changes
	Steel bar, shell model	Steel bar, shell model		
3			🛨 New result>	🔜 Calculate>Design
			Steel bar, shell model	calculation>Auto
				design all structural
				elements
4				+ New result>
				Steel bar, shell model

Table: Steps of displaying steel bar-shell model utilization by different design cases

Utilization displayed with *New result* appears for all designed bar-shell models. The utilization components for an element/design group can be displayed with *Detailed result*.

Detailed result opens two new windows in the current project after selecting a shell model or group member, which display:

- **Applied shell thicknesses** (*Detailed result* window) A list displays the applied thicknesses by the steel plate components.
- **Detailed calculation formulas** (*Detailed result* window) Calculation details and final values are collected by checking types: *Stresses, Flexural buckling, stiff/weak direction, Torsional buckling, Lateral torsional buckling* and *Local buckling*. The proper results are displayed in green, while the red result warnings you to repeat design with new thickness

values. The content of the utilization checks depends on *Display options* and *Buckling mode* (see later). Not relevant checks can also be hidden.



Figure: Utilization checks and formulas

- **Summary graph** (*Detailed result* window) *Summary* graph is displayed with legend by default.



Figure: Utilization summary

- **Buckling mode** (*Buckling mode* window)

Based on stability analysis, the calculated shapes of the bar-shell model can be displayed in 3D view. Just select the required shape from the *Shape* drop-down list of the navigator panel and the program automatically shows the deformed shape in the current *display mode*. *Buckling mode* can be also chosen for the selected shape that affects on the design check content of the *Detailed result* window. You can add numeric values to the deformed shape or run buckling animation.



Figure: Buckling shape

Detailed result Tabmenu contains the following tools and settings for the Detailed result window:

- Selection of element to display

You can choose a unique or a design group member from the drop-down lists to display its detailed results mentioned before. Each row displays the ID and the maximum utilization of a member. In case of a design group, "Maximum" means the significant member having the maximum utilization.

- Selection of design load

A load combination can be selected for detailed results. Each row displays the name of the available load combination and its utilization effect. "Maximum" means the significant load combination.

F Auto design

Quick *Auto design* can be done for the currently displayed unique/group member. Set the thickness of the steel plate components, and then click *OK* to start design that updates all detailed result figures and tables.

Display options

The content and the appearance of the detailed result can be set with *Display options*.

🤳 Go to

Navigate in the *Detailed result* window by selecting the required design type in the drop-down list. It is useful when you are in zoomed view.

Click *Tools* > *Add view to document* to place all calculation and check formulas into *Documentation*.

TIMBER DESIGN

Fast auto design and check are available to find the most suitable timber bar cross-sections and panel types.

The table summarizes the available timber design features by FEM-Design module.

Design element type	Design feature			→ ↓
Timber Bar	Auto design			
	Manual design	Ì	$\langle \rangle$	\diamond
Timber Panel	Auto design			
	Manual design			\diamond

Table: Timber design features by FEM-Design module

Load-Duration Classes

Based on Eurocode 5 (EN 1995-1-1:2004), load-duration classes can be taken into account in the *Timber design*.

The load-duration classes are characterized by the effect of constant load acting for a certain period of time in the life of the structure. For a variable action the appropriate class shall be determined on the basis of an estimate of the typical variation of the load with time.

Actions shall be assigned to one of the load-duration classes given for strength and stiffness calculations.

Load-duration class	Order of accumulated duration of characteristic load	Examples of loading	
Permanent	More than 10 years	Dead load	
Long-term	6 months – 10 years	Storage	
Medium-term	1 week – 6 months	Imposed floor load, snow	
Short-term	less than one week	Snow, wind	
Instantaneous		Wind, accidental load	

Table: Load-duration classes and examples of load-duration assignment (EN 1995-1-1:2004)

Since climatic loads (snow, wind) vary between countries, the assignment of load-duration classes may be specified in the National annex.

Loads can be assigned to load-duration classes with the *Load cases* command in the *Loads Tabmenu*.

No	Name	Туре	Duration class (EN 1995 1-1)	-	ок
1	Dead load	+Dead load	Permanent	Ξ	Cancel
2	Storage	Ordinary	Long-term		Save as defaul
3	Snow	Ordinary	Short-term		
4	Wind	Ordinary	Short-term	~	
			Permanent Long-term Medium-term Short-term		

Figure: Load cases and load-duration classes

Timber Bar

Global *Auto (timber bar) design (Calculate > Design calculations > Auto design all structural elements)* finds the most suitable cross-section (from the profile range set at *Auto design > Parameters*) for all timber bars (columns, beams and truss members) based on their initial calculation parameters, internal forces and detailed utilization calculations. With *Manual design* you can run quick utilization check for given profiles by bar elements and/or design groups. You can also do quick *Auto design* by elements and design groups only instead of global design. Of course, any number of design cycles is executable, so the global *Auto design* can be combined with both previous and additional element-based *Auto designs*.

No.	Global timber bar design	Element-based timber design	Combined timber design
1	De 😭 Calculation parameters	📟 Global Analysis	Dr. 😭 Calculation parameters
2	💷 Design group	De Calculation parameters	💷 Design group
3	🗲 🤋 Auto design > Parameters	🔤 Design group	🗲 🤋 Design Parameters
4	📟 Global Auto design	差 🤋 Auto design > Parameters	📟 Global Auto design
5	Documentation	Muto design by elements	🗲 🤋 Auto design > Parameters
6		<i>Manual design</i> by elements	🗲 ! Auto design by elements
7		🛯 Apply design changes	Manual design by elements
8		📟 Global <i>Check</i>	🖎 Apply design changes
9		Documentation	📟 Global <i>Check</i>
10			Documentation

Table: Recommended steps by design alternatives

Initial Calculation and Design Parameters

All bar design calculations needs internal forces from *Analysis* calculations applied for *Load combinations* or *Load-groups*, $\mathcal{D}_{\mathfrak{c}}$ *Buckling length* and initial design settings defined by E Design calculation parameters.



Figure: Design calculation parameters

Auto Design

Global *Auto design* gives utilization results and suitable profiles for all timber bars of the current project from a defined range of cross-sections.



Figure: Global Auto design and utilization result

The recommended profile names can be displayed on screen by showing the *"Timber bar, applied quantity"* object layer, or click *Design* tool of the *Auto design* and the parameters together with utilization results are available in table format. Utilization as colored figure (color palette) can be displayed by selecting *New result* > *Timber bar* > *Utilization*.

Applied profiles are displayed in blue in the *Utilization* table, if they are assigned to the timber bars during design, otherwise black color represents the original/initial profiles.

Element-based Auto design finds the most suitable profile of timber columns, beams and bars for

selected unique or grouped members only from a range of available profiles defined by *Parameters*.



Figure: Range of available profiles for design

To run element-based design for the load combinations or the maximum of load groups, select the required

members and/or group with the *Auto design* command and click *Design* tool. The quick process results recommended profiles and their utilization. Check the *Display table* box to have a look at the overall design results (see the figure before).

The upper table shows the design efficiency and the maximal utilization of the designed single members and groups based on the given design parameters. The bottom table displays the utilization details of the bar or the members of the group selected in the upper table.

	Meaning	Note
\checkmark	Suitable profile is available	
•	Suitable profile is not available	Modify the range of available profiles
		or timber materials
Group	ID of a single bar or a group name	
Design parameters	The defined range of available profiles	
Applied profile	Profile currently assigned to the bar	
Max	Max. utilization of a single bar or the	
	significant member of a group	
Min	Max. utilization of the less significant	
	group member	
Bar	ID of a single bar or a group member	
Т	Utilization for combined bending and	According to Eurocode 5: 6.2.3
	axial tension	
С	Utilization for combined bending and	According to Eurocode 5: 6.1.4, 6.2.4
	axial compression	
S	Utilization for combined shear and torsion	According to Eurocode 5: 6.1.7, 6.1.8
FB1	Utilization for flexural buckling around	According to Eurocode 5: 6.3.2
	axis 1	
FB2	Utilization for flexural buckling around	According to Eurocode 5: 6.3.2
	axis 2	
LTB	Utilization for lateral torsional buckling	According to Eurocode 5: 6.3.3

Table: The meaning of symbols, design parameters and utilization results

Quick redesign can be done inside the *Utilization* table:

- 1 Select a bar or a design group in the upper table.
- 2 Modify the range of available profiles for the select element under *Parameters*.
- 3 Click Design.

Manual Design

With *Manual design* quick utilization check can be done for given timber profile and for selected timber beam, column, bar or design group only. Just, choose the load type (load combination or load group) and a profile name from the drop-down lists and select a bar, bars or group, and program displays detailed utilization results in table format.



Figure: Quick check by Manual design

The meaning of the utilization components, the table content and features are the same as written before at *Auto design*. The program use the chosen profile for all selected bar elements.

Detailed Result

Utilization of timber bars can be displayed in the following cases:

- After global *Auto design*, you can display utilization of all timber bars checked for the recommended profiles.
- When running element-based *Auto design*, utilization can be displayed by designed elements.
- After Manual design, element-based Check displays utilization for selected elements.
- After global *Check* done for all bar elements having final cross-section.

No.	Global Auto design	Element-based Auto des.	Element-based Check	Global Check
1	🔤 Calculate>Design	差 Auto design	差 Auto design and/or	🗲 Auto design
	calculation>Auto	_	! Manual design	and/or
	design all structural			! Manual design
	elements			
2	🛨 New result>	া New result>Timber bar	Scheck	🖎 Apply changes
	Timber bar			
3			🛨 New result>	폐 Calculate>Design
			Timber bar	calculation>Auto
				design all structural
				elements
4				+ New result>
				Timber bar

Table: Steps of displaying timber bar utilization by different design cases

Utilization displayed with *New result* appears for all designed bars. The utilization components for a bar/design group can be displayed with *Detailed result*.

Detailed result opens a new window in the current project after selecting a bar/group member, which displays:

Applied cross-section

The figure displays the applied timber cross-section with its main calculation parameters.

Group E, Maximum of group members Maximum of load combinations			
Rectangle 140x360 C22 (Solid), Service class: 1, γ_M = 1.30, k_{sys} = 1.00			
	$\begin{array}{rcrcr} A &=& 50400 \mbox{ mm}^2 \\ W_1 &=& 3.024e+06 \mbox{ mm}^3 \\ W_2 &=& 1.176e+06 \mbox{ mm}^3 \\ i_1 &=& 104 \mbox{ mm} \\ i_2 &=& 40 \mbox{ mm} \\ i_2 &=& 8.232e+07 \mbox{ mm}^4 \\ i_4 &=& 2.486e+08 \mbox{ mm}^4 \end{array}$	$\begin{array}{rcl} {\sf E}_{0.05} &=& 6700 {\sf N/mm}^2 \\ {\sf G}_{0.05} &=& 419 {\sf N/mm}^2 \\ {\sf f}_{0.0k} &=& 13.00 {\sf N/mm}^2 \\ {\sf f}_{0.0k} &=& 20.00 {\sf N/mm}^2 \\ {\sf f}_{m.1k} &=& 22.00 {\sf N/mm}^2 \\ {\sf f}_{m.2k} &=& 22.31 {\sf N/mm}^2 \\ {\sf f}_{vk} &=& 2.40 {\sf N/mm}^2 \end{array}$	

Figure: Applied cross-section

- Detailed calculation formulas

Calculation details and final values are collected by checking types: *Combined bending and axial tension* (Eurocode5: 6.2.3), *Combined bending and axial compression* (6.1.4, 6.2.4), *Combined shear and torsion* (6.1.7, 6.1.8), *Flexural buckling around axis* 1 and 2 (6.3.2) and *Lateral torsional buckling* (6.3.3). The proper results are displayed in green, while the red result warnings you to repeat design with new bar properties. The content of the utilization checks depends on *Display options*. Not relevant checks can also be hidden.

- Summary graph

Summary graph is displayed with legend by default. Numeric values can be inquired in the calculation sections (set by *Design calculation parameters*).

Group E, Maximum of group members Maximum of load combinations



Figure: Utilization checks, formulas and summary

Detailed result *Tabmenu* contains the following tools and settings:

- Selection of element to display

You can choose a unique or a design group member from the drop-down lists to display its detailed results mentioned before. Each row displays the ID and the maximum utilization of a member. In case of a design group, "Maximum" means the significant member having the maximum utilization.

- Selection of design load

Depending on timber design was done for load combinations or load groups, a load combination or the maximum or a significant component of load groups can be selected for detailed results. Each row displays the name of the load combination/load group component and its utilization effect. "Maximum" means the significant load combination or component of load groups.

Auto design

Quick *Auto design* can be done for the currently displayed unique/group member. Its design parameters can be set/modified in the appearing dialog, and then clicking *OK* starts timber design that updates all detailed result figures and formulas.

📕 Manual design

Manual design can be launched directly for the currently displayed unique/group member.

Display options

The content and the appearance of the detailed result can be set with *Display options*. You can show only the final equation without details of the different checks (*Hide details*).

Display options 🛛 🔀				
Table of content				
🗹 Input data				
Combined bending and axial tension - 6.2.3				
Combined bending and axial compression - 6.1.4, 6.2.4				
Combined shear and torsion - 6.1.7, 6.1.8				
Flexural buckling around axis 1 - 6.3.2				
Flexural buckling around axis 2 · 6.3.2				
Lateral torsional buckling - 6.3.3				
Summary				
Content options				
Hide all unrelevant verifications				
Hide details				
Appearance				
FontAaBbYyZz •				
Figure scale				
Settings > Close				

Figure: Display options of Detailed result

Go to

Navigate in the detailed result window by selecting the required design type in the dropdown list. It is useful when you are in zoomed view.



Click Tools > Add view to document to place all figures, fomulas and summary table or specified details only into Documentation.

Timber Panel

Global Auto (timber panel) design (Calculate > Design calculations > Auto design all structural elements) finds the most suitable type (from panel type range set at Auto design > Parameters) for all timber panels (plates and walls) based on internal forces and detailed utilization calculations. With Manual design you can run quick utilization check for given panel types by elements and/or design groups. You can also do quick Auto design by elements and design groups only instead of global design. Of course, any number of design cycles is executable, so the global Auto design can be combined with both previous and additional element-based Auto designs.



To define a timber panel (slab or wall) read more at *Timber Panel Element*.

No.	Global timber panel design	Element-based timber design	Combined timber design
1	🔤 Design group	📟 Global Analysis	💷 Design group
2	差 🤋 Auto design > Parameters	💷 Design group	🗲 🤋 Design Parameters
3	🔤 Global Auto design	差 ? Auto design > Parameters	📟 Global Auto design
4	Documentation	<i>I</i> Auto design by elements	🗲 🤋 Auto design > Parameters
5		<i>Manual design</i> by elements	Muto design by elements
6		🔍 Apply design changes	Manual design by elements
7		📟 Global <i>Check</i>	🖎 Apply design changes
8		Documentation	📟 Global <i>Check</i>
9			Documentation

Table: Recommended steps by design alternatives

Auto Design

Global *Auto design* gives utilization results and suitable panel type for all timber panels of the current project from a defined range of panel types.



Figure: Global Auto design and utilization result



The recommended panel type names can be displayed on screen by showing the *"Timber panel, applied quantity"* object layer, or click *Design* tool of the *Auto design* and the parameters together with utilization results are available in table format. Utilization as colored figure (color palette) can be displayed by selecting *New result* > *Timber panel* > *Utilization*.

Applied panel types are displayed in blue in the *Utilization* table, if they are assigned to the timber panels during design, otherwise black color represents the original/initial types.

Element-based *Auto design* finds the most suitable type of timber panels for selected unique or grouped members only from a range of available types defined by *Parameters*.



Figure: Range of available timber panels for design

To run element-based design for the load combinations or the maximum of load groups, select the required members and/or group with the *Auto design* command and click *Design* tool. The quick process results recommended panel types and their utilization. Check the *Display table* box to have a look at the overall design results (see before).

The upper table shows the design efficiency and the maximal utilization of the designed single panel element and groups based on internal forces. The bottom table displays the utilization details of the panel or the panel members of the group selected in the upper table.

	Meaning	Note
\checkmark	Suitable panel is available	
•	Suitable panel is not available	Modify the range of available panels or timber materials
Group	ID of a single panel or a group name	
Design parameters	The defined range of available panels	
Applied type	Panel type currently assigned to the panel	
Max	Max. utilization of a single panel or the	
	significant member of a group	
Min	Max. utilization of the less significant	
	group member	
Panel	ID of a single panel or a group member	
Sx+ / Sy+	Utilization for x/y-directional tension	According to Eurocode 5: 6.2.3
Sx- / Sy-	Utilization for x/y-directional compression	According to Eurocode 5: 6.1.4, 6.2.4
Тху	Utilization for shear, xy	According to Eurocode 5: 6.1.7
Tx / Ty	Utilization for shear, xz /yz	According to Eurocode 5: 6.1.7

Table: The meaning of symbols, design parameters and utilization results

Quick redesign can be done inside the *Utilization* table:

- 1 Select a panel or a design group in the upper table.
- 2 Modify the range of available panel types for the select element under *Parameters*.
- 3 Click Design.

Manual Design

With *Manual design* quick utilization check can be done for given types of selected timber panels or their design groups only. Just, choose the load type (load combination or load group) and a type name from the drop-down lists and select a panel or panel group, and program displays detailed utilization results in table format.

 Λ

The meaning of the utilization components, the table content and features are the same as written before at *Auto design*. The program use the chosen type for all selected panels.



Figure: Quick check by Manual design

Detailed Result

Utilization of timber panels can be displayed in the following cases:

- After global Auto design, you can display utilization of all timber panels checked for the recommended panel types.
- When running element-based Auto design, utilization can be displayed by designed elements. _
- After Manual design, element-based Check displays utilization for selected elements.
- After global *Check* done for all panel elements having final thickness (type). _

No.	Global Auto design	Element-based Auto des.	Element-based Check	Global Check
1	🔤 Calculate>Design	差 Auto design	差 Auto design and/or	🗲 Auto design
	calculation>Auto	_	! Manual design	and/or
	design all structural			👤 Manual design
	elements			
2	🛨 New result>	া New result>	Scheck	🖎 Apply changes
	Timber panel	Timber panel		
3			🛨 New result>	🔲 Calculate>Design
			Timber panel	calculation>Auto
				design all structural
				elements
4				🛨 New result>
				Timber panel

Table: Steps of displaying timber panel utilization by different design cases

Utilization displayed with New result appears for all designed panels. The utilization components for a panel/design group can be displayed with *Detailed result*.

Detailed result opens a new window in the current project after selecting a panel/group member, which displays:

Applied panel type

A list displays the applied timber panel type with its main calculation parameters.

Group Shell A, Maximum of group members Maximum of load combinations

CLT 178

Service class: 1, γ _M = 1.30, k _{sys} = 1.00							
t	=	178	mm _	f _{m.0.k}	=	14.00	N/mm ²
E _{0,mean}	=	5500.00	N/mm ²	f _{m,90,k}	=	10.30	N/mm ²
E _{90.mean}	=	4100.00	N/mm ²	£.0.k	=	9.80	N/mm ²
E _{0,t}	=	4100.00	N/mm ²	f.,90,k	=	13.20	N/mm ²
E _{90.t}	=	5500.00	N/mm ²	f _{o.0.k}	=	9.80	N/mm ²
E _{0,c}	=	4100.00	N/mm ²	f _{c,90,k}	=	13.20	N/mm ²
E _{90,c}	=	5500.00	N/mm ²	f _{v,0,k}	=	2.00	N/mm ²
Go	=	110.00	N/mm ²	f _{v.90.k}	=	2.00	N/mm ²
G ₉₀	=	110.00	N/mm ²	f _{r.k}	=	2.00	N/mm ²

Figure: Applied timber panel type

Detailed calculation formulas

Calculation details and final values are collected by checking types: Tension and bending (Eurocode5: 6.2.3), Compression and bending (6.1.4, 6.2.4), Shear, xy (6.1.7) and Shear, xz and yz (6.1.7). The proper results are displayed in green, while the red result warnings you to repeat design with new panel types. The content of the utilization checks depends on *Display options*. Not relevant checks can also be hidden.

Summary graph _

Summary graph is displayed with legend by default.

Group Shell A, Maximum of group members Maximum of load combinations

Tension and bending, x - 6.2.3 Panel: TB.22 , LC: Load comb02 (k_{mod} = 0.70), Coordinates [m]:{ 16.13; 3.00; 4.00} $\frac{\sigma_{t,0,d}}{f_{rd,x}} + \frac{\sigma_{m,x}}{f_{rd,x}} = \frac{0.01}{5.28} + \frac{6.32}{7.54} = 0.84 \le 1 \quad (6.17) - OK$ Tension and bending, y - 6.2.3 Panel: TB.22 , LC: Load comb02 (k_{mod} = 0.70), Coordinates [m]:{ 15.75; 0.75; 1.00} $\frac{\sigma_{t,90,d}}{f_{td,y}} + \frac{\sigma_{m,y}}{f_{md,y}} = \frac{0.03}{7.11} + \frac{0.41}{5.55} = 0.08 \le 1 \quad (6.17) - OK$ Compression and bending, x - 6.1.4, 6.2.4 Panel: TB.22 , LC: Load comb02 (k_{mod} = 0.70), Coordinates [m]:{ 16.50; 3.00; 4.00} $\frac{\sigma_{o,0,d}}{f_{od,v}} = \frac{0.02}{5.28} = 0.00 \le 1 \quad (6.2) - OK$ f_{od,x} $\left(\frac{\sigma_{o,0,d}}{f_{od,x}}\right)^2 + \frac{\sigma_{m,x}}{f_{md,x}} = \left(\frac{0.02}{5.28}\right)^2 + \frac{6.46}{7.54} = 0.86 \le 1 \quad (6.19) - OK$ Compression and bending, y - 6.1.4, 6.2.4 Panel: TB.22 , LC: Load comb02 ($k_{\rm mod}$ = 0.70), Coordinates [m]:{ 15.38; 1.00; 1.33} $\frac{\sigma_{c,90,d}}{f} = \frac{0.06}{7.11} = 0.01 \le 1 \quad (6.3) - OK$ fed w $\left(\frac{\sigma_{o,90,d}}{f_{od,y}}\right)^2 + \frac{\sigma_{m,y}}{f_{md,y}} = \left(\frac{0.06}{7.11}\right)^2 + \frac{0.43}{5.55} = 0.08 \le 1 \qquad (6.19) \ \text{- OK}$ Shear, xy-6.1.7 Panel: TB.24, LC: Load comb02 (k_{mod} = 0.70), Coordinates [m]:{ 16.50; 0.50; 0.67} $\frac{T_{xy}}{1000 \text{ t } f_{r,d}} = \frac{82365.41}{1000 \cdot 178 \cdot 1.08} = 0.43 \le 1 \qquad (6.13) \ \text{- OK}$ Shear. xz - 6.1.7 Panel: TB.21 , LC: Load comb02 (k_{mod} = 0.70), Coordinates [m]:{ 15.00; 1.25; 1.67} $\frac{T_{xz}}{1000 \text{ t } f_{v,d,x}} = \frac{40867.52}{1000 \cdot 178 \cdot 1.08} = 0.21 \le 1 \quad (6.13) - OK$ Shear, vz - 6.1.7 Panel: TB.24 , LC: Load comb02 (k_{mod} = 0.70), Coordinates [m]:{ 15.00; 1.00; 1.33} $\frac{T_{yz}}{1000 \text{ t } f_{v,d,y}} = \frac{15076.34}{1000 \cdot 178 \cdot 1.08} = 0.08 \le 1 \quad (6.13) - OK$



Figure: Utilization checks, formulas and summary

Detailed result *Tabmenu* contains the following tools and settings:

- Selection of element to display

You can choose a unique or a design group member from the drop-down lists to display its detailed results mentioned before. Each row displays the ID and the maximum utilization of a member. In case of a design group, "Maximum" means the significant member having the maximum utilization.

- Selection of design load

Depending on timber design was done for load combinations or load groups, a load combination or the maximum or a significant component of load groups can be selected for detailed results. Each row displays the name of the load combination/load group component and its utilization effect. "Maximum" means the significant load combination or component of load groups.

륮 🛛 Auto design

Quick *Auto design* can be done for the currently displayed unique/group member. Its design parameters can be set/modified in the appearing dialog, and then clicking *OK* starts timber design that updates all detailed result figures and formulas.

📕 Manual design

Manual design can be launched directly for the currently displayed unique/group member.

Display options

The content and the appearance of the detailed result can be set with *Display options*. You can show only the final equation without details of the different checks (*Hide details*).

Display options 🛛 🛛 🛛 🛛					
Table of content					
🗹 Input data 📃					
✓ Tension,x - 6.2.3.					
Tension,y - 6.2.3.					
Compression, x - 6.1.4, 6.2.4					
Compression, y - 6.1.4, 6.2.4					
✓ Shear, xy- 6.1.7					
✓ Shear, xz - 6.1.7					
Snear, yz - 6.1.7					
Content options					
Hide all unrelevant verifications					
Hide details					
Appearance					
FontAaBbYyZz •					
Figure scale					
Settings > Close					

Figure: Display options of Detailed result

🤳 Go to

Navigate in the detailed result window by selecting the required design type in the dropdown list. It is useful when you are in zoomed view.

Click *Tools* > *Add view to document* to place all figures, fomulas and summary table or specified details only into *Documentation*.

FOUNDATION DESIGN

Fast check are available to find the most suitable foundation.



In Configuration dialog, the user can set design options, like *Design Approach*, soil's *Safety Factors*, the load combination's *Limit state* and *Settlements parameters*.

		Foundation design configuration			
Foundation design configuration	— X —	Design approach Safety factors Load of No Name 1 Dead Load	Type Limit state STR U GEO 1		
Design approach Safety factors Load combinations Settlements		2 load combination 2	U STR GEO 2		
A1 + M1 + R1 and A2 + M2 + R1	Foundation design configuration				
© 2 A1+M1+R2	Design approach Safety factors load combinat	tions Settlements	Limit state	Foundation design configuration	X
3 (A1* + A2+) + M2 + R3	Type of load combination .	Ultimate Accidental	definition	Design approach Safety factors Load combin	nations Settlements
	Soil properties R	Resistances Seismic		Thete may limit [9] 0.382	Î Î Î Î I
Design approaches	M1 M2 Gamma cu 1.00 1.40	R1 R2 R3 Gamma Rv 1.00 1.40 1.00		Alpha max limit [*] 0.382	Alpha Delta s
U U II	Gamma c 1.00 1.25	Gamma Rh 1.00 1.10 1.00		Delta s max limit [m] 0.0500	
	Gamma phi 1.00 1.25 p	Permanent actions	OK Cancel	V L/Delta max limit [-] 300	
	Gamma gamma 1.00 1.00	A1 A2 Unfavourable		V Omega max limit [°] 0.382	Delta
				E Beta max limit [°] 0.382	Omega
OK	y factors		Settlement		
		OK Cancel		parameters	
	<u>[</u>		J		OK Cancel

In Foundation design only *Check* is available, *Manual design* or *Auto design* not.

In *Design approach* tab, the user can select the design approach which influences the combination of effects. For more details, see Eurocode 1997-1:2006 Chapter 2.4.7.3.4.

In *Safety factor* tab, the user can set the safety factors of soil properties, resistance and permanent actions.

In the *Load combination* tab, the user can set the limit states for each load combinations according to the selected *Design approach*.

In the *Settlements* tab, the user can define the limit values of the relative displacements and rotations. For more details, see Eurocode 1997 H Annex.

By Design calculation parameter command the foundation's calculation parameters can be set: 1. Isolated foundation:

- settlement limit value
 - preparation method (Cast in-situ or Precast)



- 2. <u>Wall foundation</u>:
 - o settlement limit value
 - maximum distance of calculated sections

3. Set properties



2. Select foundation
• Put additional check points for settlement calculation. These points automatically generated at the slab region's vertices and at the joint of columns and foundation slab.



Design calculations can be done by *Check* command or in *Calculation dialog*. Under *Check* command the user can check either the objects or display the relative settlements.



Check objects

	Group	Total weig [t]	iht		Max. [%]	Min. [%]	-
•	F.2			22.6	122	122	-
•	F.4			22.6	122	122	
•	F.1			22.6	122	122	۲
•	F.3			22.6	122	122	-
		ltem	Max. [%]	BE [%]	SL [%]	SE [%]	
•	F.2		122	4	0	122	
-							
							-
Check Delete < Hide details							

Check relative settlements

Settle	ments	1						
Load	combination	SLS	•	·				
	Value	Point 1.	Point	t 2.	Max.	Limit	Util.	*
\checkmark	Theta [°]	F.1-1 (-61.1)	F.5-1 (-60	.9)	0.001	0.382	0.2 %	
\checkmark	Alpha [º]	F.1-1 (-61.1)	F.5-1 (-60	.9)	0.001	0.382	0.2 %	
	Delta s [mm]	-	-		-	-	-	
\checkmark	L/Delta [-]	F.1-1 (-61.1)	F.5-1 (-60	.9)	401045	300	0.1%	
\checkmark	Omega [°]	-	-		0.001	0.382	0.2 %	
	Beta [°]	-	-		-	-	-	-
_								_
	Point 1.	Point 2.	Theta [°]	Alpha [º]	Delta s [mm]	L/Delta [-]	Beta [°]	Â
$\mathbf{\nabla}$	F.1-1 (-61.1)	F.5-1 (-60.9)	-0.001	0.001	-	401045	-	-
\checkmark		F.3-1 (-60.9)	-0.001	0.001	-	402003	-	
\checkmark		F.2-1 (-61.1)	0.000	0.001	-	428321	-	
1		F.4-1 (-61.1)	0.000	0.001	-	429360	-	
1	F.2-1 (-61.1)	F.1-1 (-61.1)	0.000	0.000	-	428321	-	
		E 0 4 (c0 0)	0.004	0.004				Ψ.

For foundation slabs only relative settlements can be calculated with SLS combination. Due to this the design calculations for foundation slabs will not run the ULS combinations.

After displaying the *Foundation utilization* results, foundation design calculations can be seen by *Detailed results* command.



When choosing *Display options* the following dialog appears.

Go to... **Command helps the user to navigate in detailed results.**

Display options	×
Table of content	
Geometry	
Material	
Sarety factors Bearing	
Sliding	
Settlements	
Summary	
Content options	
🔲 Hide all unrelevant verific	ations
Hide details	
Appearance	
Font	AaBbYyZz •
Figure scale	
Settings >	Close

1. Click	2. Sel	ect the par	agraph
_		Top of page Geometry Material	
		Satety factors Bearing Sliding Settlements	
		Summary Reset view	

PERFORMANCE BASED DESIGN

Model definition and calculation

Performance Based Design is implemented in FD 13 according to Turkish code. This way the performance level of existing RC buildings can be calculated from linear analysis. To calculate Performance level of an existing reinforced concrete building, follow these steps:

- 1. Set the code to "Eurocode (Seismic code: Turkish)".
- 2. Build the model in Structure tab.

 \bigwedge The columns and walls have to be defined from storey to storey.

- 3. Add the loads in Loads tab.
 - a. Define Gravity load combination.
 - b. Define masses from loads in Load case mass conversion dialog.
 - c. Define the Seismic Load properties:
 - i. Set the Structure type to Existing building, and also the Information level in Structure information tab.
 - ii. Define the design spectra in Spectra tab.



There are Standard and Unique spectra. Standard spectra are calculated according to the current seismic code. In order to define non-standard spectra the use the Unique option.

- 4. Run analysis in Analysis tab
 - a. Calculate Load combinations in Calculation dialog.
 - b. Calculate Eigenfrequencies in Calculation dialog.
 - c. Calculate Seismic analysis (*Equivalent seismic load*) in Calculation dialog.
- 5. Define reinforcement for RC elements in RC Design tab.
- 6. In Performance based design tab
 - a. Set the Calculation parameters for beams, columns and walls:

Calculation parameter	×
ក្រា	

Figure: Setting the calculation parameters

• Beams: both ends are **Confined** and/or **Shear strengthened** and/or **Secondary beams**



Figure: Calculation parameters for beams

• Columns / Walls: both ends are **Confined** and/or **Shear strengthened**

Ca	Iculation parameter
	Confined
	OK Cancel

Figure: Calculation parameters for columns and walls

b. Start the Performance based design calculation by selecting the *Design calculations* under *Calculations* dialog.

Calculations	Design calculations
Load cases	Calculation
Load combinations Maximum of load grou Stability analysis Significantly analysis Selemic analysis	Auto design all structural elements Oreck all structural elements Based on analysis of
Design calculations	Load combinations *
• III.	

Figure: Design calculations

Results

For Performance based design the following results are available to display and list:

Results for display:

- Damage mode/zone:
 - By member
 - By member and joints
 - By member and storey drift
 - By all
- Damage factor
- Relative storey drift ratio
- Performance level

Results — Analysis	By member	
Load cases Load combinations Load combinations Maximum of load combinations Generating the second se	Seismic load direction +Fx+Mx +Fx+Mx -Fx+Mx +Fy+My +Fy+My -Fy+My -Fy+My -Fy-My	

Figure: Display results

Results to list:

- Input data
 - Beam longitudinal reinforcement
 - Column longitudinal reinforcement
 - Wall longitudinal reinforcement
 - Beam strirrup reinforcement
 - Column stirrup reinforcement
 - Wall stirrup reinforcement
- Results
 - Beams, fracture mode
 - Beams, damage zone by member
 - Beams, damage zone, summary
 - Strong column check
 - Columns, fracture mode
 - Columns, damage zone by damage factor
 - Columns, damage zone by drift
 - Column-beam joint shear check
 - Columns, damage zone
 - Wall fracture mode
 - Walls, damage zone
 - Building performance (Fx +Mx, Fx Mx, -Fx + Mx, -Fx Mx, Fy+My, Fy-My, -Fy+My, -Fy-My, summary)



DISPLAY RESULT

Different modes can be used for displaying *Analysis* and *Design* results. This chapter summarizes the skills and settings of display results on the screen. Of course the result figures can be printed out or can be inserted to documents (see *Documentation*).

Display Techniques

Finishing analysis and design calculations, the results are available in the list of the *New* result command. Just click the command icon and browse from the results available from previously done calculations.



Figure: Quick way for displaying results

Select a result data, then a display technique (*Display options*) for it, and finally, clicking *OK* displays the result figure based on the default or last-used settings of the applied display technique.

	Availa	ble for
Display option	Bar elements	Planar elements
Graph		
Contour Lines		
Color Palette		
Sections		

Table: General display options

If display options are not available for the selected result data, it will be displayed in one way. Some result types have special display modes.

	Avail	able for
Display option	Bar elements	Planar elements
Reactions/Connections	All type of suppo	rts and connections
Bi-directional results		Principal internal forces, required
		reinforcement shown
		simultaneously in all directions
Principal directions		Directions of principal internal
		forces
Crack width		Result of crack width calculation
		(RC Design)
Punching results		Punching check results (RC Design)

Table: Special display options

For bar elements, the currently displayed result (e.g. displacement, internal forces, utilization for RC/steel/timber design etc.) can be detailed by selected bar with the *Detailed result* command.

EQ3

The scale / color distribution of the current figure can be modified fast with *Quick editing tools* or with the *Display option*.

Graph

By choosing *Graph* display option, the result values (e.g. displacement, internal forces etc.) will measured and displayed in their calculation point (finite element nodes).



Figure: Displacement of a slab in different views and display modes



Figure: Moment and shear diagrams of beam elements

The settings of the currently displayed graph can be modified with the *Display options* tool.



Figure: Display settings of graphs (planar and linear)

The following options can be modified and set for a graph:

- Scale factor

It sets the display scale of the graph (enlarge (>1) or reduce (<1)). *Scaling value,* which is the absolute or relative maximum of the current result type, helps you in setting the scale factor.

- Reset absolute vs. Reset relative

Pressing the buttons fits the *Scale factor* to the *absolute or relative maximum* of the same type results on the current structure.

- Graph color

The color of the result and the positive/negative values (available for bar elements only) can be set by clicking the "colored line" button and by browsing requested colors from a palette. The button shows the actual color set for the displayed graph.

Step (available for bar elements only)
It is the distances between the points in which you can ask and display *numeric values*.

Contour Lines

Contour lines are a set of color lines. Each line connects points having the same result value and displays them with one color. It is similar to the contours of a topographical map which connect contiguous points of the same altitude.



Figure: Contour lines

Contour lines display technique is developed to display results of plates, walls, shells etc. Two types of contour lines are available: "*Continuous*" and "*Discrete*".

"*Continuous*"-type contour lines contain colored line by equal step, while lines can be defined in arbitrary distribution with the "*Discrete*". You can define as many colors as you want for the contour lines.



Figure: Continuous and discrete contour lines

The distribution and color settings of the currently displayed contour lines can be modified with the *Display options* tool:

"Palette" settings

Choose the required contour palette type: continuous or discrete.

The active option displays the meanings of the color (palette symbol) on the screen next to the result. *Size* (%) defined the size of the palette symbol, if it is displayed on screen.

In case of "Continuous" palette, *Step* sets the distance between the neighboring contour lines and so the color distribution.

Pen width sets the thickness of the contour lines on the screen.

"Value" settings



Defining a color and a value in the available numeric field, the *Insert* option inserts a new line into the palette with the given color line and at the given value. This option is mainly developed to define the "*Discrete*" palette.

⊠

Delete option can be used to remove the unnecessary color line and its value currently selected in the palette with its \triangleleft symbol. It is equivalent with the "drag and drop out" of the line with its \triangleleft symbol.

Modify option moves the contour line, which is currently selected in the palette, to the given value and/or changes its color with a color set under the option.

- "View" settings

Zoom functions to enlarge or reduce the range of the palette. The last status will be used for the palette symbol, if it is displayed by \square .

Reset absolute vs. Reset relative

Pressing the buttons fits the top and bottom limits of the palette adjust to the *absolute or relative maximum* of the current result.

Color Palette

Color palette is a set of colored regions. Each colored regions are represent values in a given interval. It is similar to *Contour line*, but not only the region borders but their inner areas are colored.



Figure: Color palette used for displaying results for planar and bar elements

Color palette display technique is developed to display results of both planar and bar elements. Two types of color palettes are available: "*Continuous*" and "*Discrete*".

"*Continuous*"-type contains colored zones by equal step, while the interval of the colored regions can be defined in arbitrary distribution with the "*Discrete*". You can define as many colors as you want for the color palettes.



Figure: Continuous and discrete color palette for planar objects

The distribution and color settings of the currently displayed color palette can be modified with the *Display options* tool:

"Palette" settings

Choose the required color palette type: continuous or discrete.

The active option displays the meanings of the color (palette symbol) on the screen next to the result. *Size* (%) defined the size of the palette symbol, if it is displayed on screen.

In case of "Continuous" palette, *Step* sets the interval of the colored zones and so the color distribution.

"Value" settings



Defining a color and a value in the available numeric field, the *Insert* option inserts a new colored zone into the palette with the given color line and at the given bottom value. This option is mainly developed to define the "*Discrete*" palette.

Delete option can be used to remove the unnecessary colored zone and its value currently selected in the palette with its \triangleleft symbol of the bottom value. It is equivalent with the "drag and drop out" of the zone with the \triangleleft symbol.

₽

Modify option moves the colored zone, which is currently selected in the palette, to the position defined by the given bottom value and/or changes its color with a color set under the option.

"View" settings

Zoom functions to enlarge or reduce the range of the palette. The last status will be used for the palette symbol, if it is displayed by \Box .

- Reset absolute vs. Reset relative

Pressing the buttons fits the top and bottom limits of the palette adjust to the *absolute or relative maximum* of the current result.

Color palette display mode also gives the possibility to show displacement (=deformed shape) of surface/bar elements simultaneously the displayed result shown by color palette.



Apply *Deformed shape* for the color palette of Mx' plate result (*Analysis/ Plate internal forces*). First display Mx' with *Color palette*, and then open the palette's setting with the Mx' *Display options* tool. Open the *Others* tab of the color palette dialog, and mark the *Display with the deformed shape* checkbox.

Surface, colour palette 🛛 🛛 🛛 🛛	
Palette Others	
Display with deformed shape	
Scale factor	Display options
Scaling value	
Reset absolute Reset relative	
Colour	
Pen width for bars 2	
Display settings	
Settings > OK Cancel	

Figure: Color palette result displayed with deformed shape (planar elements)

Deformed shape can be also shown together with a color palette result (My') of bars.



Figure: Color palette result displayed with deformed shape (bar elements)

Sections

Results of planar objects, which are able to be shown with *Graph* mode, can be displayed in given section lines.



Figure: Result displayed in sections

After defining sections, the program fixes the section positions, so the next result - selected for displaying with *Sections* - will be displayed in the same section lines. For example, you can easily and fast check the same parts of a slab, wall etc. for different internal forces, loads or load combinations.

Ð

Display an analysis result (e.g. Mx') and one of its derivatives in design (e.g. *Required reinforcement, bottom, x*') in the same section positions. Select the first result type with "*Sections*" display option, and define sections with their start and end points. Choosing the next result (e.g. *Rqd. Reinforcement, e.g. Rqd. Reinforcement, the same section position and explanation of the section position and end points.*



Figure: Section positions are stored for next results

To modify the color and hatch settings of the displayed sections, click *Display options* and open the settings dialog with the *extension* to be setting option.

	Sections
Edit sections	Scale factor 0.0104
_ 🖉 🖉	Scaling value 195.42060 kNm/m
	Reset absolute Reset relative
	Positive values Negative values Step [m]

Figure: Section settings

- Scale factor

It sets the display scale of the sections (enlarge (>1) or reduce (<1)). *Scaling value*, which is the absolute or relative maximum of the current result type, helps you in setting the scale factor.

- Reset absolute vs. Reset relative

Pressing the buttons fits the *Scale factor* to the *absolute or relative maximum* of the same type results on the current structure.

- Positive/Negative values

The color of the positive/negative values can be set by clicking the "colored line" button and by browsing requested colors from a palette. The buttons show the actual color.

- Step

It is the distances between the points in which you can ask and display *numeric values* and the displayed hatch lines.

Additional sections with the current settings can be added to the view with the \square *Define* tool. Unnecessary sections can be removed with the *Delete section* tool by just their selection.

Display Reaction and Connection Forces

Reaction and connection forces can be displayed in *supports* and *connection* objects by type, by component, with group resultants, with the combination of them or the *resultants* at the middle of the support/connection.



Figure: Reaction forces and moments in a Wall and Column support (Plate module)



Figure: Reultants of the line support group

The color of a reaction/connection force component is represented by the proper direction component (x', y' or z') of the support's/connection's local coordinate system. The color of the local system axes (directions) can be set at *Settings* > *All* > *Display* > *Local systems*.

To check and modify the settings of the displayed reaction/connection results, click Display options.

		Support/Connection types
	Reactions	X
	Point Line Surface Resultant	
Components	Group components Single	Others
components	↑ Fx' Fy' Fz' Fz'	Distribution
	ハx' マ My' ロ Mz' マ M ハ	Hatch dist. [m] 0.200
Group	Group resultant	Special settings
resultant	↑ F = □ Fx' + □ Fy' + □ Fz'	Special settings
	M = Mx' + My' + Mz'	
Scale	Scale	
	Force [kN/m] 0.0248 94.5	
	Moment [kNm/m] 0.0621 29.6	
	Reset absolute Reset relative	
	Settings >	OK Cancel

Figure: Display options of reactions and connection forces

Support/Connection types

First, choose from the available support/connection types (point/line/ surface) you would like to display. Of course, the results of all types can be displayed simultaneously.

Component

Choose component(s) (force/moment and directions) you would like to display in case of single/group *support* or connection.

Resultant

The force (F) and/or moment (M) resultant of group support reactions or connection forces can be displayed. Resultants appear with dashed lines on the screen



Quick selection buttons let you hide all or show selected components on the screen.

Scale

It sets the display scale of the result diagrams (enlarge (>1) or reduce (<1)). The scaling value (shown in the grey fields), which is the absolute or relative maximum of the current result type, helps you in setting the scale factor. Pressing the *Reset absolute* or *Reset relative* buttons fits the *Scale factor* to the *absolute or relative maximum* of the same type results on the current project.

Distribution

Different displaying modes can be chosen for line reactions/connection forces:

- Original

It displays the calculated result values.

Linear

It displays linear mean value of the result by support/connection element, which is calculated by the area-weight method.

- Constant

It displays constant mean value of the result by support/connection element, which is calculated by the area-weight method.

- Constant by element

To solve *singularity peaks* at line reactions and connections, the program calculates the average value (constant mean value) of the reaction and connection forces by element.



In this case, you can easily insert *numeric values* by the constant section (*Numeric value* > *Find all local maximum/minimum*).



Figure: Line reactions displayed with "Constant by element"



Figure: The theory of the average calculation by element (Constant by element)

Bi-Directional Results

In some cases (e.g. initial forces, reinforcement results etc.) two result components can be displayed together, in one diagram. At these results, numerical values are also displayed besides the directions.



Figure: Required bottom reinforcement displayed with directions and numeric values

The color and text settings of the currently displayed bi-directional result can be modified with the *Display options* tool.

Numeric results
Decimals
Size of pointer [m] 0.20000
Direction x'
Direction y'
AaBbYyZz •
Settings > OK Cancel

Figure: Display settings of bi-directional results

Principal Directions

The direction of principal internal forces and stresses can be displayed in two ways:

- "One size"

The symbols of the principal directions are shown with uniform-length lines. The symbol line of the first principal direction is longer than the second ones.

- "Proportional – scale"

The length of a symbol line represents the size of the principal force/stress. The display scale factor can be customized according to the absolute/relative maximum of the result.



Figure: One-sized or proportional-scaled principal results

The display type and the colors of the principal directions can be set and modified with the *Display options* tool.

Principal directions	
One size [m]	1.30
Proportional - scale	0.00728
Scaling value 243	.38120 kNm/m
Reset absolute R	eset relative
Colour if 1 > 2	
Colour if 2 > 1	
Pen width [mm]	2 🚔
Settings > OK	Cancel

Figure: Display settings of principal results

Crack Width

Crack width results of RC slabs and bars can be displayed with crack lines. The lines represent the direction of the cracks and their pen width the size of the crack width.



Figure: Crack width of RC slab

The length, width and color of the crack lines can be modified with the *Display options* tool.

Cı	ack width 🛛 🔀
	Length scale 9.33 Width scale 0.933
	Scaling value
	Result colour
(Settings > OK Cancel

Figure: Display settings of crack width result

Design Utilization Results

Global *Auto design* and *Check* calculations give utilization results for planar and bar elements by the applied design type.

Utilization results are colored figures (color palette), and summarize the list of maximum utilizations by elements in a table dialog with the help of the *Numeric value* tool.

Read more about the utilization results by design type: *RC design, Steel design* and *Timber design*.



Figure: RC bar design and utilization result



Figure: RC punching design and utilization result

Detailed Bar Result

After displaying bar results of the entire model such as the internal forces, displacements and design utilization results, more detailed results can be displayed in separate windows by bar with the *Q Detailed result* tool. For example, you can see diagrams of all internal force components of a selected beam in a new window separately from the entire model.



Figure: Internal force diagrams by a single bar

The display type and the colors of the detailed bar results can be set and modified with the *Display options* tool. The content of the settings dialog depends on the bar result.

Single bar - display options 🛛 🔀
Common scale
OK Cancel

Figure: Display settings of bar internal forces

- Diagram type

Common scale displays the same-type result diagrams (e.g. My, Mz and Mt are moment diagrams) in common scale. The program automatically sets the scale value to the *absolute maximum* of the current result.

Scale by figures displays diagrams in their best scale. The scale values, the upper and bottom limits will be set by figure according to its own *absolute maximum* value (e.g. the scales of My and Mz differ from each other).

- Positive/Negative values

The color of the positive/negative values can be set by clicking the "colored line" button and by browsing requested colors from a palette. The buttons show the actual color.

The hatch line dense by the figures is the same with the "display steps" of the original bar result, on which the *Detailed result* tool was used.

Utilization results with detailed background calculation formulas (together with Eurocode references), figures and tables can be displayed by single elements or by design groups. Quick navigation is powered with zooming details. Read more about *Detailed results* by design type: *RC design, Steel design* and *Timber design*.



Figure: Detailed design result (RC bar design)

"Absolute" vs. "Relative" Maximum Value

The *Absolute maximum* and *Relative maximum* options of the display settings search the maximum values of a result component in all diagrams independently the negative and positive signs. *Relative maximum* finds the maximum value of the current diagram with its absolute value for the current load case/combination/group. *Absolute maximum* finds the maximum value of the same type results (e.g. shear force diagrams) with its absolute value for all load cases/combinations/groups.

Let's see an example to understand the meaning of "absolute" and "relative" maximum values.

Let's display the Tz shear force diagram of a frame structure in case of two different load combinations. The next table summarizes the positive and negative maximum values of both diagrams.

	Maximum values								
Display option	Positive	Negative	Relative	Absolute					
Load combination 01	9.37	-9.42	9.42	10.20					
Load combination 02	10.20	-8.67	10.20	10.20					

Table: Maximum values of the same-type result diagrams (Tz)



Figure: The values found by Reset absolute/relative

Style Templates

Display settings (scale, colors, steps etc.) can be stored in files (so-called styles) by display technique or by figure. The saving and the loading facilities can be reached at the *Display options* tool. The styles are stored in files with different file extensions by display technique. The files can be swapped between users, so companies can create standard display setting for their own documentation style.



Save as default

It saves the current display settings as the default (without file name!) settings of the applied display technique. If you do not use a style (see later), the program always offer the last saved default settings when you apply the same technique inside one example or in further ones.

Load default

It restores the default display settings after any change.

- Save

The color settings of the current dialog can be saved as a style in a template file. The file extension depends on the current display technique.

- Save with scale

All display settings (scale, steps, colors etc.) of the current dialog can be saved as a named style in a template file.

- Load

User-defined styles can be loaded from a file browser.

The following example helps in understanding the difference between *Save* and *Save with scale* after loading styles created by them. The color palette settings of an *Mx* internal force diagram was saved with *Save* and *Save with scale*. The figure shows the settings of *My* also displayed with color palette after loading (*Load*) the two saved styles. After loading "*Save*"-style, the palette contains the same colors but the scale is optimized for *My* values. Loading "Save with scale"-style results the same scale together with the same color information with *Mx*.



Figure: Save vs. Save with scale

Numeric Values

Tool Tip

Rolling your mouse over a result picture will display the numeric value in the current position and in the proper metrical unit as a tool tip. Tool tip works for all types of *display techniques*.



Figure: Numeric value displayed by tool tip

Value Labeling

Result diagrams can be completed by displaying the calculated numeric values. Use the $\stackrel{1.00}{+}$ *Numeric value* tool to add labels to the currently displayed result figure.

Numeric value	1	Plate/wall results
1.00 +	Numeric value	×
Bar results	1.00 +	

Figure: Tool palettes of Numeric value

Depending on the currently displayed result figure and the object type – you would like to ask numeric values – the tool palette of *Numeric value* contains different definition tools. With the *Define* tool, new value labels can be placed on the current figure.

The color of the numeric values can be set with the *"Current color" button* of the *Status bar*. Of course, you can place values with different colors on a result figure. For example the maximum values can be displayed in red while the others with black.

	1.00	
Set the text settings of the numeric values at	+	Default settings.

Numeric valu	le			X
Font T r Arial			~	ОК
Script	Western		~	Cancel
Size [mm]		6.0000		
Width		1.0000		
Slant [deg]		0.00000)	
AaBbY	уZz			
Decimal places		2		

Figure: Numeric value settings

The next figures give examples for the tools of *Numeric value*.

Use $\stackrel{2.15}{+}$ Select to place numeric values in clicking points of the displayed result. With \land tool, result maximum, minimum and absolute maximum/minimum can be found for selected object(s). With \land tool, all result limits (maximum and/or minimum) can be displayed for selected object(s).



Figure: Arbitrary and maximum values (bar displacement)



Figure: Arbitrary and maximum values (plate internal forces)



Figure: Display numeric values (with Select) on a Contour palette result

In case of bar displacement, points can be selected for displaying numeric values on bars (*Original shape*) or on the *deformed shape* of the bars.



Figure: Numeric values displayed on the original and deformed shapes

Bar displacement can be asked with its real size (*Vector size*) or by its component (parallel with the global/local x'/y' or a given direction).



Figure: Numeric values by displacement component

The display settings of predefined numeric values can be modified by the *Properties* tool.

The position of the text labels can be modified independently from their insertion (clicking) point with *Move* (*Edit* menu).

Editing Results

The settings of the currently displayed result can be modified with the *Display options* tool any time. The content of the setting possibilities (colors, scale, palettes etc.) depends on the *display technique* used for the visible result.



Above *Display options*, quick change tools give fast and real-time modification in scale/color distribution (step) of the current result figure and in size and decimal numbers of displayed numeric values. Quick change tools work for all type of *display techniques*. The next figures show some examples.



Figure: Modifying diagram scale

Scale increases/decreases the scale (e.g. graph, section) or the step distribution (e.g. contour lines, color palette) of the currently displayed result figure with 10% percent. Use the "+" or "-" key of the keyboard's numeric pad as the fast key of *Increase scale* or *Reduce scale*.



Using *Display options* or *Scale* to modify the current diagram causes the loss of the numeric values; so defining *numeric values* is suggested after finishing the scale setting of the current figure.



Figure: Modifying the text size of numeric values

Font size modifies the font size of the numeric values with 10% of the initial size.



Figure: Modifying the decimal numbers of numeric values

Browsing Results

A one-click browser (*Quick selection*) is developed for changing between result figures displayed earlier on the screen. By clicking on " \checkmark " symbol next to \frown *Select result*, a list appears with titles of results displayed so far. Just click on the result you wish to display again. The content of the list can be organized with *Select result*: hide current figure or delete items.



Figure: Quick or dialog browsing among previously displayed results

Animation

The currently displayed results can be animated with the *Animate result* tool. Not only for bar results (such as dynamic and stability results, displacement, internal force/stress diagrams etc.), but all result display types of planar elements (such as displacement, internal forces, RC results, utilizations etc.) can be animated. That means graph points are moving, the colors and color intervals are pulsing etc. Clicking *Animate result* starts the animation after some seconds (depending on the model size) by continuously displaying the linear interpolation statuses between the initial and the final states of the current result figure. Press any key or click by mouse the stop animation.



Figure: Animation of a frame displacement

DOCUMENTATION

This chapter summarizes the documentation possibilities of FEM-Design projects, models and results. It introduces the printing and *listing* (summary tables) functions and the automatic documentation based on templates (*Documentation module*).

Printing

The full content or a selection part of the screen (*Drawing area*) can be printed out with the *Plot* command (*File* menu). Just click with \bigcirc to send the entire screen to print preview, or define a selection box with \bigcirc to set it content as the plotting input.



Figure: Plotting screen (entire or selected part)

The printing settings are the following:

- Settings dialog style

With *Dialog* and *Preview* buttons you can switch between the dialog box and the full screen mode of the plotting settings.

- Printer setup

The goal printer can be chosen from the available (installed or network) printers and plotters. PDF creator (not included by default) can be also set here to save the screen content as PDF. The printer or PDF settings are available at *Properties*.

- Page setup

The size of the paper and the margins together with the paper orientation can be set here in a separate dialog. Title label can be added to the print area. Check the box "*Info label*" to add title label to the bottom right corner of the print area. Set the page number and set the text size and the width of the label at *Setup*. The label contains custom text set by *Settings* > *Title* and autotext such as the file name and date modified.



Figure: Title label as header

If you would like to print out only the content of the *Title*, place it on the drawing area with *Draw* > *Title information table*, add it to the print area selection and inactivate the *Info label* option at *Page setup*.



Figure: Title label as drawing element

- Color print Switch between grayscale and color (seen on screen) printing.
- Pen simulation

Different pen width values can be assigned to colors at *Setup*.

- **Print border** Rectangle border fit to paper margins (*Page setup*) can be shown and set for printing out.
- Print only selected area

If this check box is active, only the selected area (in case of selection) will be printed otherwise the its environment (the whole content of the drawing area).



Figure: Printing selection

- Scale

Set the printing scale manually or select one from the list. Dashed border marks objects (selection size) to print and helps in setting the scale. *Fit scale to margins* option sets the biggest scale of the selected print area according to the paper size, margins and the origin (see later). *Fit text size to scale* option sets the scale of the texts according to the drawing scale (1:?) or keeps the original size of the text in case of its inactive status.



Figure: Printing text size

- Origin

The position of the selected printing area can be set with its distance from the origin. *Horizontal* defines the distance of the bottom-left corner of the printing area from the left margin and *Vertical* sets the distance of the bottom-left of the printing area from the bottom margin. *Center* buttons fit the printing area to the middle of the page area bordered by the margins. The printing area can be dragged by the mouse also on the symbolic white page.



Figure: Printing area position

- Simple preview

It is generally a problem to plot big models; setting the final position of drawings on the paper takes time. By activating *Simple preview*, the program shows only the frame of the selected printing area to reduce the generation time of the drawing when setting its position (*Origin*) and *Scale*.



Figure: Simple preview of the printing area

- Save as default

It saves the paper and color printing settings as the default plotting settings for later use and next projects.

Click Print to start printing. A small dialog appears, in which you may cancel/stop printing.

Quantity Estimation

Just one click on *Quantity estimation* (*Tools* menu), and a fast process collects all structural elements of the current project and lists their materials, identifiers, main geometric parameters (e.g. profiles/thickness), quantities etc.

									Quantity e	stimation							X
Quantity esti	mation												17773	_			
Concrete									Concrete III Reinforcement I Steel								
Storen Turne General									Store	y Type	Identifier	Quality	Section/ Thickness	Unit weight [t/m (m2)]	Total length[m]/ Total area[m2]	Total weight [t]	Painted area (m2)
Storey									0	Beam	B.42	C22	RECT 300.0x	0.184	6.0	1.107	5.4
1	Beam	Storev	Type	Ide	Quantity (Standton			0	Beam	B.43	C22	RECT 300.0x	0.184	6.0	1.107	5.4
1	Beam				0.00			T	0	Beam	B.44	C22	RECT 300.0x	0.184	18.0	3.321	16.2
1	Beam	1	Beam	B.1	<u></u>	ncrete	Reinforcemen	<u> </u>	1	Beam	B.45	C22	RECT 300.0x	0.184	18.0	3.321	16.2
2	Beam	1	Beam	B.2					1	Wall	TB.1	CLT 72	PL72	0.031	7.2	0.223	15.2
2	Beam	1	Beam	B.3	Store	y Type	Identifier	Qualit	1	Wall	TB.2	CLT 72	PL72	0.031	7.2	0.223	15.2
2	Beam	2	Beam	B.4	0	Ream	B 17	S 275	2	Wall	TB.3	CLT 72	PL72	0.031	7.2	0.223	15.2
2	Beam	2	Beam	B.5	0	Beam	D.11	C DEE	2	Wall	TB.4	CLT 72	PL72	0.031	7.2	0.223	15.2
2	Beam	2	Beam	B.6	0	Beam	D.10	C DEE	2	Wall	TB.5	CLT 72	PL72	0.031	7.2	0.223	15.2
2	Beam	2	Beam	B.7	1	Baam	0.10	C 265	2	Wall	TB.6	CLT 72	PL72	0.031	7.2	0.223	15.2
2	Beam	2	Beam	B.8	1	Deam	0.20	0.075	2	Wall	TB.7	CLT 72	PL72	0.031	7.2	0.223	15.2
TOTAL		2	Beam	B.9		Dean	D.21	5 275	2	Wall	TB.8	CLT 72	PL72	0.031	7.2	0.223	15.2
		2	Beam	B.10	-	Deam	D.22	0.275	2	\u/all	TB 9	CLT 72	PI 72	0.031	72	0.223	15.2 🞽
		TOTAL			2	Deam	B.23	5 275	Add to	locumentation	1					Option	is List
Add to door	umontation				2	Beam	B.24	5 275)					L	
Add to doo	unientation				2	Column	L.1	5 300									
		Add to docu	mentation		2	Column	L.2	5 300									Close
					2	Column	L.3	5 355	LUE D. COO	-		5.0	0.005		5.0		
					2	Lolumn	L.4	5 355	HE-B 200	0.1	513	5.0	3.065		5.8		
						11 olumn	15	5 455	HE R 200		513	511	31065		58		
			_	_	Add to	documentation	J		Options List								
														_			
															Close		

Figure: Quantity estimation
Reinforcement quantities are displayed after *RC design* only.

Categorization by priorities and summary by certain properties are also available in the quantity estimation tables (*Options*).

Quantity estimation options 🛛 🛛 🔀	Quantity estimation options
Sort Summarize	Sort Summarize
1st priority Storew Ascending	Summarize items that has the same
2nd priority	Type
(none) (none) Storey	Name Quality
3 Type Identifier Duality Volume Formwork	
4th priority	
(none) OAscending Clear all	
OK Cancel	OK Cancel

Figure: Categorization and summary features

The currently displayed table can be attached to *Documentation* by clicking *Add to documentation*.

The results of quantity estimation can be printed out directly as tables with the *List* option. (Other summary tables such as structural, load, finite element input and analysis/design result outputs of the current project can be also listed here. You may also use the **List** command of the *Tools* menu any time.)

Tables 	^	Quantity estimation
Coads Finite elements Quantity estimation Analysis Load combinations Maximum of load groups Stability analysis Sitability analysis Sitability analysis Seismic analysis Red cesign Input data Load combinations Maximum of load groups Steel design		Quantity estimation, Concrete Quantity estimation, Steel Quantity estimation, Timber Quantity estimation, Timber Glipboard File Table Qptions Save settings Reset tables List data

Figure: Listing quantity estimation results

Listing

All tables of input and output data can be collected and printed out with the command *List* (*Tools* menu). Just select the required data type (e.g. *Loads*) from the "*Tables*" tree and its required data (e.g. *Load cases* and *Load combinations;* multi-selection works) from the list next to the *Tables* tree, and finally set the listing output mode from the following:

List tables Tables Structure	^ Loads		
Loads Add Combinations Add Combinations	Load cases Load comile Point loads Line loads Surface loads	Send to Cipboard Dipboard File File Table Options Save setti Reset tab List dat	

Figure: Listing procedure and modes

- Clipboard

Clicking *List data* sends the table of the selected item to the *Microsoft Office Clipboard*, which allows you to copy the table and text items from FEM-Design and paste them into another document such as Microsoft Excel, Word etc.

- Printer

Send to	
Printer	~
Page setup	
Printer setup	

List data sends the selected data table directly to the printer according to *Page* and *Printer setup*. *Page setup* offers the same possibilities mentioned at *Printing*.

- File



List data saves a *TXT* file with a name and into a location defined at >. The exported TXT file contains only numerical data without titles, text, table borders etc., so it is a good input (matrix) for mathematical software.

The table settings can be set for all export modes at *Table...*:

ĺ		Table									
	6										
	5	AaBbYyZz ►	Quantity es Storey	Type	nber Identifier	Quality	Section/	Unit weight 1	otal length[m]	Total weight	Painted area
Text parameters		Units	abcdefg abcdefg abcdefg	abcdefg abcdefg abcdefg	abcdefg abcdefg abcdefg	abcdefg abcdefg abcdefg	abcdefg abcdefg abcdefg	123.457 123.457 123.457	123.457 123.457 123.457	123.457 123.457 123.457	[m2] 123.457 123.457 123.457
Font OK	I	Length m		and a g	and a g	axazy	and a g	113.137	1442.527	140.00	163.537
Script Western	I	Angle rad									
Size [mm] 3.0000	I	Mass t									
Width 1.0000	I	Sectional data mm									
Slant [rad]	I	Displacement mm									
AaBbYyZz	l	Stress N/mm2 💌									
	J	Column properties									
		Title Storey									
		Hide									
		Width [mm] 15.0									
		Scientific (exponent) form									
		Decimals									
			Zoom	table	Zoon	n page w	iidth Z	oom page al		Show hid	den columns
										ок (Cancel

Figure: Customizable table settings

The following properties can be set:

- a common text style (font type and size) for table titles, text and values;
- the unit of the values by element type (*Length, Force, Displacement* etc.);
- the column width that can be defined by the Width option or by stretching the right side of the selected column with mouse;
- decimal number of numeric data; and
- scientific form of numeric data.

Unnecessary columns of a table can be hidden by the *Hide* option. The hidden columns can be temporarily visible in the preview with *Show hidden columns*.

At *Options* (depending on the data type), the calculation points (nodes) can be set where you would like to ask and list (result) values.

List tables			
List tables Tables Tables Stucture Loads Finite elements Quantity estimation Cables Load cases Load cases Load combinations Max. of combinations Eigenfrequencies RC design Input data Load combinations Maximum fload groups Steel design Input data Load combinations Maximum fload groups Dister data Load combinations	Load combinations Result Equilibrium Nodal displacements Shells, Displacements Font group reaction forces Line group reaction forces Shells, internal forces Shells, inter	Send to Printer Page setup Printer setup In center of element In vertices Reset tables	OK Cancel
Max. of combinations Maximum of load groups	¥	List data Close	

Figure: Calculation points for listing

Save settings stores the current list settings as the default settings for later use and next projects.

After modifications in settings, *Reset tables* restores all setting parameters to default setting (set before with *Save settings*).

Documentation Module

Although the *Plot* and *List* commands give quick report data possibilities, you can create multi-page documentation with the *Documentation* module. Titles, figures, texts, tables, images, headers, footers, table of content etc. can be placed in custom style into the complete documentation. *Documentation* module can be run in all FEM-Design modules (depending on your license).

Click to enter *Documentation* mode.



Figure: Starting Document module

The next chapters introduce the possibilities and the proposed steps for defining complete set of design documentation.

Main Commands

The commands and their functions:



Figure: Documentation tools

Section

Section defines documentation parts having different properties (="sections") such as page size, orientation, margin sizes and associated printer.

Define a section for the cover page with a color image as *A*4 page size and with a color associated printer, and another section for the other pages as *A*3 size and with grey-scale printer for texts and figures with black lines.

- Title block

This is a command for creating and registering new labels, headers, footers and frames, and for inserting stored title templates into the document from the built-in or user-defined title block library. You can

apply drawing and auto text tools to design your private label, title and frame style and then register it with *Title block* command for current and future use.

Table of contents

It inserts a table of contents into the current document. It contains the list of chapters (see the next command) and their page numbers. If you define a new chapter or make modifications, the program refreshes this list.

- Chapter

Titles of main and sub-chapters can be inserted into the document with item numbers and names.

- Page break

Defining a page break generates a new page with the same properties of the current section.

- Graphical window

The command lets you to insert (working) windows in which you can display different figures (geometry, loads, mesh, analysis and design results, etc.) into the current document.

A window object defined by Graphical window is not a static figure with constant and fixed content. It is a working window, where you can use the layer systems, display settings and view commands (zoom and rotate operations, plane, side and space views, shade and wireframe display modes, etc.) to set the required appearance of the model, the loads, the supports, the results, etc.

- Table

It inserts tables of input and output data (e.g. list of load cases and load combinations). The function and the table settings are similar with the possibilities of the *List* command.

- Draw box

The tool inserts a working window into the document where you can create your own drawing with the commands of the *Draw* and *Edit* menus. Drawing layer-system is also available in a Draw box.

- Text box

It is a simple word processor for creating texts, notes into the document. The size of the text box, the font styles and text alignment can be set manually.

- Image

Image files (e.g. jpg, bmp etc.) can be inserted to the document with this command.

Control Panel

The control panel enables you to easy move between different parts of the current document, to set the properties of the inserted objects, and to print and create templates for further documentation.

Documentati	on
	Section 1
1 Initial data	
1.1 Geometry	
1.2 Supports	
1.3 Loads	
2 Analysis	
3 Results	
	Section 2
Navig	jation
lte	ms
Cut	Сору
Paste	Delete
Properties	
Docu	iment
Reset	Print
remplates >	

(Documentation map)

The list contains the defined title boxes, sections, graphic boxes, images, drawings and texts (hereafter called "objects"). You can activate the selected list object by double clicking on its name.

Navigation

You can navigate between pages by Page up, Page down, Home and End buttons. Clicking Go to switches to the object selected in the list (the same as double clicking the selected object in the *List*).

Titles added to documentation elements provide easy navigation through defined *text boxes* and figures (*Graphical window*, *Image* and *Draw box*).

- Items

With *Cut, Copy* and *Paste* tools you can modify the position of selected list items (multi-selection also works by using the mouse together with and/or keys). You can move a selected list object into a new position by holding down the way and moving the mouse into the required row of the list. To erase one or more selected objects click on *Delete*. With *Properties* the settings (sizes, styles, contents, etc.) of the selected object can be modified. The variability of objects depends on the type of object (drawing, text, image, section, etc.).

- Document

With *Reset*, you can delete all contents of the current documentation and "open" a new, empty document with the default settings. At *Templates* you can save the current document structure and settings under a name as a template. You can also load a predefined one to create fast documentation of your current project (see later *Templates*). The *Print* command lets you print out the whole document or a part of it.

Steps to Create Your Own Documentation

This chapter shows the proposed steps of document creation via an easy example (in Plate module).

For a faster documentation process, we recommend that you start documentation after finishing model generation and calculations and not before them.

1. Document title

Set the main properties of the document by selecting *Untitled* in the list and click on *Properties* at *Items*. Give a name for your document (in *Title*) and set the font settings by chapter orders (levels). Under *Utilities* you can save the settings as default or as a named style, so you can then load styles for future projects.



Figure: Document title

2. Cover page (Text box and Image - Section 1.)

Set the properties of *Section 1*. by activating it in the list and by clicking on *Properties*.

Design docume	entation section 1=	
	Section	
Navigat	Paper A4 Width 210 Height 237 Margins Left 15 Right 15 Top 60 Bottom 19	Page preview OK Cancel Associated printer: Adobe PDF
Paste	Delete	٨

Figure: Cover page

- Set the paper size and orientation (e.g. *A4* and portrait).
- Choose a printer for the current section. You can set different printers, plotters and PDF creators per sections. It is very typical that the text parts are wanted to print as A4 and the figures etc. as A3, etc.
- Set the margin, from which the texts, figures, images, etc. will start. For example, the main title will start 60mm from the top of the page.

Write the title of the document on the first page with *Text box*. The text will start from the upper margin.

Design documentation Section 1. Text box (Design Documentation)	60mm from the top margin Design Documentation
Text box Vidth [mm] 188 > = Text Design Documentation C Propetities Propetities Propetities	AaBbYyZz ↓ Cancel Font OK Taxial OK Script Western Size (rmn) 12 Width 1.0000 Slart [deg] 0.00000

Figure: Cover page with title

Also a subtitle can be created by re-running *Text box*. Empty rows and new lines can be defined with the *+ + key-combination*.

Place an image file (e.g. a picture about the model) with the *Image* command. To open an image file, browse for it at > option of *File name*. Set the position of the picture from the side margins at *Alignment*. At *Size* you can define arbitrary size or special size (*Page width*/*Full page*) by choosing from the list under > symbol. Apply *Preserve ratio* to avoid figures from distortion during size definition.

Image OK OK Display title Font Cancel Width (nm) 170 > Height (nm) 147 V Preserve ratio Alignment File name	Design documentation Hotel Moonlight Ground & 1st floor
File name C:\Documents and Settings\a	a sector and a sector a sector a sector a sector a sector a

Figure: Cover image



The settings for an inserted object (text box, image, etc.) may be modified by selecting it in the list and by pressing on the *Properties* option. It works the same as with the initial dialog of the object.

3. Table of contents (Section 2.)

Create a new section (leave your selection on the last item (here the image)) with *Section* and set the value of the top margin, which the table of content will start from (e.g. 30mm). Click on *Table of contents* command and an empty "table" appears on the *Section* 2 page. The table will be refilled automatically when defining *chapters*.



Figure: Inserted (empty) table of contents

4. Header and footer (Title block - Section 3.)

Create a new section (leave your selection on the last item (here "Table of contents")) with Section and set top and bottom margin values that define the available area for further text, figures and tables (30mm in this example). Use the commands from the *Draw* menu to prepare the appearance (frames and texts) of headers and footers. Apply e.g. *Line* and *Field* commands (*Auto-text*).



Figure: Header/footer defined by Line and Field commands

Where you see "?" symbol at an auto-text, the content can be defined at *Settings* > *Title*.

After finishing drawing, use the *Register* tool of **D** *Title block* to save your "header and footer" as builtin label parts. Type an identifier ("header&footer" in this example) for the title in *Name* field, select the lines and texts by window selection and give an insertion point (bottom left corner of the page in this example).



Figure: Header/footer defined by Line and Field commands

After registration, delete the drawings and text from the document by *Erase* (*Edit* menu)!

Insert the defined and registered title block into the document with the *Insert* tool of *Title block*. Choose the saved "header&footer" from the *Name* list, and then place the insertion point of the selected title block on the bottom left corner of the page. As result, the "header&footer" will be repeated on every page in the current section (*Section 3.*).

Title block	
Name borito_A4	
2. Insertion	

Figure: Insertion of title block (header & footer)

5. Chapters (Section 3.)

Place your first chapter with the ^{1.2.1} *Chapter* command. Give a name in *Title* field for the current chapter ("Geometry" in this example).

Design documentation Section 1. Text box (Design Documentation) Text box (Hotel Moonlight) Cover Image (Image) Section 2. Contents	Hotel Moonlight	Ground & 1st floor	StruSoft AB
Chapter Title: 1 Geometry 1 < 1.1			
	Close		

Figure: Insertion of chapter

For further chapters the program offers alternative numbering systems for the next main chapter ("2") or subchapter ("1.1"). It depends on which button is active: $1 \rightarrow 1.1$ Increase indent or $1 \leftarrow 1.1$ Decrease indent.

6. Figures (Graphical window - Section 3.)

In a chapter you may insert text (see *Text box*), images (see *Image*) and figures of the model geometry, loads, results, etc. In this example, first place a figure about the model geometry (plates and supports). Click *Graphical window*, choose *Structure*-type and set the display parameters of the image in the appeared dialogue box. You may give a name to the figure for displaying it (*Display title*, "Plane view"). A "1m scale bar" can be displayed inside the figure with *Display scale*. The size of the window can be set with a width and a height value or by choosing predefined sizes under > (*Page width/Full page*). The required figure position can be defined by activating one of the buttons for *Alignment* ("centre" in this example). The position of the title is the same as the activated *Alignment* of the window. Clicking *OK* inserts the new figure into the document.

Docu	mentation		
Ū			
	Untitled Section 1. Text box (Design Documershion) Text box (Hotel Moonlight) Section 2. Contents Section 3. 1 Geometry Plane view (Structure, Graphical with	Structure Loads Finite elements Analysis Rc design Steel design Steel design U Display tile V Display tile V Display scale V Plane view V Display scale V Pint border Width (mm) 180 V Example to the state of	

Figure: Insertion of figure

Other graphical windows can be defined by repeating the previous step. For example, place a side view of the structure, etc.

Clicking the window area activates it as a working window, so you can modify the view and position of the window content. Black thick frame shows what graphical window is the active.

For example, clicking the *Structure*-type graphical window, the display settings (*Settings* menu), the view commands (*View* menu) and the *layer system* (*Current layer*) may be used; but at an

Analysis (result)-type graphical window, you can browse from results (*New result* and *Select result*), displaying numeric values (*Numeric value*).



Figure: Available tools depend on the graphical window type

Each graphical window has its own layer system, main and display settings. *FEM-Design* also gives the possibility to define different units for displaying geometry and various results.

Click the white paper (outside from the active graphical window) to return back documentation.

/!\

Repeat the previous steps to define more *chapters* (2. Supports, 3. Loads, 3.1 Load cases, 3.2 Load combinations, etc.), and to place new figures (such as loads with a *Loads-type graphical window*).

There is no facility to set distances between texts, chapter title rows and figures, but with *Text box* command you can insert empty lines between objects into the document.

Another way to insert figures is, that the current model view can be placed onto the last page of documentation with the Add view to document command (*Tool* menu). Of course, the settings of the new graphical window can be modified later with the *Properties* tool of *Documentation*.



Figure: Quick export of current view as a figure into documentation

7. Tables (Section 3.)

Input and output data as tables can be added to the document with the *Table* command. The functions and tools of *Table* are same as described at *List*. Choose data to print out as a table from the available *Tables*, *Data* and *Result* lists. In this example, place a summarization table of load combinations (under chapter 3.2: Load combinations). First select *Load combinations* in *Data* list (*Tables/Loads*), second set the table settings at *Table* and *Options*, and finally click on *List data*. As a result the selected table appears after the current documentation object.



Figure: Quick export of current view as a figure into documentation

8. Result figures (Graphical window - Section 3.)

To document result figures, use Graphical window and select Analysis-, RC design-, Steel design- or Timber design (depending on the available calculation type and what result you would like to display). After insertion, click inside the graphical window and choose result types with New result and/or Select result.

Templates and Automatic Documentation

The documentation structure (cover, chapters, text and window positions) defined in *Documentation* module can be saved under a file name (*.DSC*) as a template for further projects. Click *Control Panel's Templates* > *Save as* tool.

The program automatically generates your complete documentation, if you apply a previously defined or built-in template. Use *Templates > Apply* when entering to *Documentation* mode after modeling and calculate a new project, and the program automatically generates complete documentation by refreshing based on the applied template (it updates the title label, table of content, page numbers, figures and tables etc. of the template).



A pictures inserted by *Image* are deleted in a new template!

If you load a template into a document having no calculated results, the *Analysis*, *RC design* or *Steel design*-type graphical windows contain only the structure, but they will be refreshed any time you have calculations.

If an applied template generates documentation with *tables* having no valid values, the empty tables can be hidden. Select the title of the document (first row) in the *Control panel*, then click *Properties* and inactivate the "*Display empty tables*" option in the dialogue that pops up.



Figure: Empty tables hidden by Properties

Printing

The completed document or only some parts of it can be printed out with *Control Panel's Print* tool or *File > Print document*.

Print document	X
Page range Image Image	OK Cancel
Options	

Figure: Printing setup

Set the *Page rage* as:

- All

The whole document (all pages from start to end) will be printed out.

- Selection

The objects selected in the *Control panel* (multi-selection also works) and their pages will be printed out.

Pages

This provides the possibility for printing by defined page. Enter page number and/or page ranges by separating with commas and by using "-" (an example: 1,10,15-20,25, etc.).



The printing process requires associated printer(s) by the sections of the current documentation. If a section has not got a defined printer (*Undefined*), you can add one by selecting the section name in the document list and by using the *Properties* tool (*Change* option).

Design documentation	Section	
Section 1. — Text box (Decim Documentation) Text box (Hotel Moonlight) Section 2. Contents 1 Geometry Plane viw (Structure, Graphical window) 2 Supports 3 Loads 3.1 Load cases 	Page preview A4 A4 A4 A4 A4 A4 A4 A4 A4 A	JK ncel
Navigation	Top 15 Bottom 15 Adobe PDF	
Items Cut Copy		
Properties Document		

Figure: Printer assigned to Section



If you assign a PDF creator program to all sections, you can save your documentation as a *PDF* file.

DRAWING AND EDITING TOOLS

Although FEM-Design is an analysis and design application, CAD drawing can be also done with its built-in drawing tools. Editing tools also available to edit, modify and multiply both structural and drawing elements. Drawings can be the draft of the later design model, but you may also export them in the popular *DWG and DXF* formats.

Drawing tools are grouped in the *Draw* menu, but they are directly available from the list appears when clicking $\overset{\frown}{\longrightarrow}$ key together with $\overset{\frown}{\bigcirc}$.

Editing tools are grouped in the *Edit* menu, but they are directly available from the list appears when clicking \mathfrak{O} .



Figure: Drawing and editing tools

Draw Menu Commands

The following rules are valid for the drawing commands and elements:

- Drawing elements are placed and stored in *Drawing layers*.
- The color of the current drawing layer is the default drawing color. Under "*Current color*" button of *Status bar*, you can customize the drawing color independently from the host layer's color.
- The style of the drawing lines and edges can be set with "*Current style*".
- Drawing and text styles together with display settings of drawing elements are available at Settings.
- Although, you may draw in 3D, in most cases, it is recommended to set the *user coordinate-system* in the drawing plane.
- To define drawing points, type coordinate and angle values in the *Command line* and/or *Coordinate box* or use the *Object snap tools* to pick requested special points.
- Command line guides you with the steps of drawing element definition.

The next table summarizes the available drawing tools, their functions and definition steps.

Command	Description	Definition modes	
Point	Defines a point based on the current drawing settings.	-	
Line	Defines a line or connected lines (open, closed and/or polyline) based on the current drawing settings.	Definition of start- and endpoints	
Rectangle	Defines a rectangle (closed contour) based on the current drawing settings.	Definition of two opposite corners	
Arc	Defines a circular arc based on the current drawing settings.	Arc by center, radius and angle Arc by 3 points Arc by start, end point and tangent	
• Circle	Defines a circle arc based on the current drawing settings.	 Circle by center and radius Circle by diameter Circle by 3 points 	
Region	Defines a drawing region with arbitrary shape. Also holes can be added to the region.	Rectangular Circular Polygonal Pick lines (close contour)	
Solid	Defines prismatic, centric, sphere, regular and general 3D solid surface. Generic solid can be generated with the help of arbitrary shapes and directrix. Solids can be also cutted in their heights.	 Prismatic Rectangular base (brick/cube) Circular base (cylinder) Polygonal base (prism) Pick lines (arbitrary base) Pick existing region (arbitrary base) Centric Rectangular base (pyramid) Circular base (cone) Polygonal base Polygonal base Polygonal base Pick lines (arbitrary base) Pick lines (arbitrary base) Pick lines (arbitrary base) Pick existing region (arbitrary base) Pick existing region (arbitrary base) Sphere Sphere Sphere Sphere Cube Cube Octahedron Cosahedron Cosahedron 	
A Text	Defines arbitrary text in custom style and any direction.	-	
# Field	Places autotext in custom style and any direction.	-	
^{₩→} Dimension	Defines length, angle and level dimensions in custom style.	 ← Linear (Length) ◇ Arc ⊘ Diameter 	

		 Radius Angle Level dimension
i Title information table	Places a table that displays title Information of the current project. The content of the table can be set by <i>Settings</i> > <i>Title</i> . The width of the information table is defined by the text size and style (<i>Settings</i> > <i>Text</i>). Table is placed in the Title information object layer. Layer color defines the table color too	-

Table: Draw menu tools

The following chapters give more details for some drawing items.

Solids

This chapter shows some example for available solid geometries and introduces how general solids can be defined with the help of shape contour and directrix.

There are no real curved surfaces in 3D modeling, so the program approximates the curved surfaces with polygonal planes. The value of approximation can be set by the Refracting angle option (Primary/Secondary). The next figure shows the meaning of refracting angle values for a double-

curved surface (defined by 🐸 General).



Figure: Curved surface approximation



Figure: Some examples for prismatic solids with definition steps



Figure: Full or truncated pyramid (Centric solid)



Figure: Some examples for centric solids with definition steps



Figure: Spheres defined in different coordinate-systems



Figure: Half and truncated spheres



Figure: Octa-, dodeca- and icosahedron

With *General* tool, general 3D solids can be defined with a directrix and a generation line (builds up the surface of the solid). Both of them can be a closed contour line or an arbitrary line (it can be also closed), but one by one. The only rule is that they must touch each other. Five tools can be used for defining the contour geometry and other five ones for defining the line geometry. The function of the contour line or the arbitrary line (and the generation line is the arbitrary line) or the *Directrix is contour line* (and the generation line is the solid depends on the *Perpendicular* and *Constant* options that define how (in what angle) the generation line will be transformed around the directrix:

- *Perpendicular*: the solid sections having the same shape with the generation line will be perpendicular to the directrix in its all points.
- *Constant*: the solid sections (inner surfaces) will be parallel with the generation line along the directrix.



Figure: Different combinations of directrix and generation line



Figure: Both directrix and generation line are closed contours

Text and Field

Custom (\triangle *Text* command) and automatic (*[#] Field* command) texts can be placed on the drawing area in arbitrary position. They can also be edited any time.



Figure: Text and Field commands with their settings

At *Text*, the custom text can be defined in the *Text* field. At Field, the content of autotext can be set at *Type*. Some autotext types (such as *Project* title, *Description*, *Designer*'s name, *Signature* and *Comments*) are defined by the user at *Settings* > *Title*, and the other ones automatically set by the current project settings (such as *File name*, *File date*, *Code name* etc.). Field is especially useful in *Documentation*, as you can see it from the range of autotext types.

The following text properties can be set for both text and field items:

- Font

Choose the required font type from available MS Windows offered types.

- Script

Script solves the character problems at different languages. An example for the Times New Roman font: character "ő" appears as "õ" in *Western*, but as correct "ő" in *Central European* script.

- Size

It is the height of the font in millimeters.

- Width

It is the width of the font. Its value that is lower than 1 defines narrower letters. Its value that is bigger than 1 results thicker letters.

- Slant

You can set the slant of the letters in degrees that closes angle with the vertical direction. The angle is measured clockwise. The standard italic font has about 10-15 degree slant.

- Alignment

In the *Alignment* figure you can set the position of the text insertion point. The hatched rectangle is the border of the text and the insertion point is in the intersection of the two black lines. Moving the black lines with the mouse offers nine possible positions for the insertion point.

To place the text/field on the drawing area, give the coordinates of the insertion point, and then define its direction. The \bigcirc mouse button sets the direction parallel ("horizontal") with the X direction of *UCS*.



Figure: Text position

Special characters can be defined in *Text* field by typing "%" before letters and numbers. These are the followings:

- %g = ° (degree symbol)
- %d = Ø (diameter symbol)
- %p = ± (plus/minus symbol)
- %% = %
- %2 = 2 (square sign)
- %3 = 3 (cube sign)

Dimension

With \bowtie *Dimension*, length, angle and level dimensions can be defined in custom style. The style of dimensions can be set by \bowtie *Default settings*.



Figure: Dimension settings

- Dimension line extension

Dimension line can be extended with *a* value.

- Level

The size of the 📌 *Level dimension* symbol can be set with *Size* value.

- Color and Pen width

You can choose a color (black by default) for all dimension items, and set a pen width for the lines.

- Layer

Drawing layer ("DIM" by default) can be chosen for the new dimension items. Of course, you can define and use different layers by position (external/internal), by direction (horizontal/vertical), by type (length/level dimension) for example.

- Extension line

Sizes, positions and display settings can be set for extension lines.

- Arrowheads

The style, the size and the pen width of the arrowheads can be set here.

- Text display properties

The text style, the decimal numbers and the measurement unit type (independently from the project unit settings) of the numeric values can be set here. Additional text can be inserted before and after the measured values with *Prefix* and *Suffix*. Also *special characters* can be used as dimension text.

- Horizontal / Vertical alignment

The alignment of text to dimension lines can be set by direction.

Dimension types

From the *Dimension* tool palette chose the required type of the new dimension.



Figure: Dimension types

Different directions can be set for *timear* type dimensions:

- Predefined direction

The direction of dimension lines can be set parallel with one of the axis directions of the *UCS* or the *global system*.



Figure: Dimensions parallel with global axes

- Parallel with line

The direction of dimension lines can be set parallel with a line defined by two points.



Figure: Dimensions parallel with a given line

- Perpendicular to plane

The direction of dimension lines can be set perpendicular to a given plane (e.g. the plane of a wall).



Figure: Dimensions perpendicular to a given plane



Figure: Dimensions of arcs and circles



Figure: Dimensions of angles



Figure: Level dimension

Modification in dimension

- Modify line dimension

With this tool you can add new (reference) points to a selected predefined linear dimension line. First select the required dimension line, and then place the new point.



Figure: New points added to dimension line

- Add dimension to base line

New dimension lines can be added to a baseline of an existing linear dimension line. This means, that the starting point of the new lines is at the selected baseline of the dimension line. First select the dimension line, then choose the side or baseline, and finally add the new points. The new lines will inherit all properties from the selected one. The location of the selected dimension line determines that the new lines will be above it or below it.



Figure: New dimension line assigned to a base line of an existing one

- Dimension line position

The position of a predefined dimension lines can be modified easily. Just select the required dimension line, and then define its new position.



Figure: Modification in dimension line positions

- Text content

The real measured values can be substitute with any custom text. Not only numbers but letters can be added as text content. Do not forget to hide unit system, if you want to display only the new text. First select the text value, and then type the required text in the *Text content* dialog. Clicking the *Original* button restores the measured dimension value and overwrites the custom, the user-modified text.



Figure: Modified text content

Edit and Modify Menu Commands

The *Edit* menu contains tools for editing and modifying drawing elements, structural elements, loads, reinforcement objects, result data, etc. The following table summarizes the editing tools, their function and examples.

Edit Command	Function	Example
Repeat command	Reruns the last used command.	
1	Undoes the last operation such	
Undo	as the editing operations	
	summarized in this list.	
	The maximum numbers of	
	undoable operations can be set	
	at Settings > Environment >	
	General > System > Undo steps.	
	Calculation operations and	
	previous commands cannot be	
	undone.	
Redo	Executes the previously undone	
_	command.	
Cut	Erases selected elements of the	
Cut	<i>Clickand in ander to Desta theme</i>	
	Labor to the drawing area Are	
	insertion point is required for	
	the later insertion (paste)	
	Places selected items on the	
Copy	<i>Clinhoard</i> It does not delete the	
	original objects. An insertion	
	point is required for the later	
	insertion (paste)	
(Fr	Inserts the FEM-Design-type	
- Paste	drawing contents of the	
	<i>Clinhoard</i> into the current	
	project's drawing area. The	
	elements are placed with the	
	coordinates of the insertion	
	point defined by <i>Cut</i> and <i>Copy</i> .	
	Paste together with Cut and	
	Copy is working between more	
	than one opened projects too.	
S Frank	Deletes selected objects from the	
- Erase	current project and so from the	
	drawing area.	
	Some special objects can be	
	deleted with their own eraser	
	tool:	
	- Holes of planar regional	
	elements (plates, walls etc.)	
	can be deleted with the	
	Remove note tool of Moaijy	
	Finite elements of a mach	
	- Finite elements of a mesh can	
	of the Finite elements to be only	
Copy bitmap	It exports the selected part of the	

	drawing area to the <i>Clipboard</i> as an image, and it can be inserted to other programs such as photo editors, MS Word, MS Excel, etc.	
Paste file	Inserts the whole geometrical content (drawing and structural elements, loads, finite element mesh) of a FEM-Design file into the current project. The insertion point of the copied content is the global system origin of the pasted file.	-
	The loaded file must be compatible with the current <i>FEM-Design module</i> .	
External reference	It imports DWG/DXF drawings as an image in the background	
Modify commands	Function Moves objects or their copies with a given distance (displacement vector) in the	Example
Сору	drawing area. Makes more than one copies of selected objects. Each copies can be moved with different distances. The base point of each copies are the same point of the selected elements.	
Mirror	Mirrors selected objects or make a mirrored copy of them. Elements or their copies can be mirrored to a given line or plane.	Mirror to Plane
Rotate	Rotates selected objects or make a rotated copy of them. Elements or their copies can be rotated around a given point or line with an accurate angle or a defined new position.	Rotate around point
Array > Rectangular	Multiplies selected objects as row (UCS y direction) and column (UCS x direction) distribution. The number of copies and the distance between the copies by direction are required.	1.5 1.5 1.5 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0

Array > Polar	Multiplies selected objects by rotating them around a centre point. Required data: direction of rotation (<i>Clockwise/Counter-</i> <i>clockwise</i>), the origin of the rotation, the rotation angle and the number of copies.	number of copies=5 120 Array/Polar Total no. Total no.
- Offset	Makes a copy of line-type objects (line, arc, region contour, beam, column etc.) that will be placed parallel with the original object with a given distance and on the defined side.	Offset side and distance
Stretch	Stretches line-type elements or the selected corners of closed polygons and region-type elements.	one corner-selection
Curve	Curve a line-type element or an edge of region-type elements. Clicking on a curved line straigntens that.	straight wall editing rectangular surface load editing slab edge editing
Elbow	Breaks a selected line or region edge, or in other word, it adds new point to lines and edges in new position.	straight wall editing slab edge editing
Scale	Scales objects, blocks, even the entire structure. A base point (will be the only point that will remain in its original place after scaling) and the scale factor are required for scaling	scale factor < 1

	Automatically splits selected line-type elements in their common intersections.	
+ - Trim	Cuts line-type objects along an edge defined by another object. The cutting edge may have a real or virtual intersetion with the selected line wanted to be trimmed.	Cutting edge
-+ Extend	Extends line-type objects to an edge defined by another object. The boundary edge may have a real or virtual intersetion with the selected line wanted to be extended.	boundary edge
Break	Cuts a section from line-type object and breaks it into two independent object. <i>Break</i> can be used to cut a line into two parts by double clicking in the same line point (zero section length).	two pieces 1, 2 2 1 x one piece
Chamfer	 Cuts the corner of a closed polygon and region. Connects two line-type objects with a skew line defined by the chamfer distances. 	Sm Danne Sm Second chamfer datance (m) 3 Scond chamfer datance (m) 9 0K Cancel
Fillet	 Rounds the corner of a closed polygon and region. Connects two line-type objects with an arc defined by the fillet radius. 	3m
Modify region	 Removes hole from region- type object. Splits region by cutting edge defined manually or by the intersection of another region. Reduces/increases region area with parallel offset of region edges. 	
	4. Swaps faces of wall/plate regions (mirrors local system).	

	 Does logical operations (union, substarct, intercection and excluded "OR") between regions. 	
Modify solid	 Removes hole from drawing solid. Splits solid by cutting region/plane defined manually or by the intersection of another solid. 	
	 Does logical operations (union, substarct, intercection and excluded "OR") between solids. 	
← Change direction	 Modifies the direction of the following object types: column and beam section position, support, load, parametric reinforcement, applied reinforcement bar. 	(opposite) global Z
Change appearance	 Changes display properties of drawing and structural objects such as color, line type, pen width, text style etc. Matches properties of an object with others. 	

Table: Edit menu tools

Quick modifications

	Click LB (default)	Click + hold LB	CTRL + Hold LB
Line's end point Node on surface edge	Stretch	Stretch	Drag a copy
Along point along line	Curve	Move	Drag a copy
Any point inside Surface	Move	Move	Drag a copy
Any point on screen, where there is nothing	Box selection		

At \bigcirc user can select another command in the Modify menu which is also available by \bigcirc .

• Ine's end point or node on surface edge starts 'Stretch' command or the last used command selected by the user for this case




On any point inside on surface starts 'Move' command



and holding at line's end point or surface edge's end node starts 'Stretch' command



• Or and holding at any point along line or any point inside surface starts 'Move' command



• Im +holding It any point in a structural or drawing element starts to drag a copy of the object.



• On an object pops up the available commands depends on where the user clicks.



• On none of the objects hit pops up 'Modify' commands.



Guide lines and Tracker

The Guide lines and Tracker appears when any commands started and assist the user in modeling and modifying elements in a much more effective way. Guide lines are displayed temporarily during modification or drawing to assist user to find desired point in a fast and easy way. There are two kinds of Guide lines **Straight** and **Circular**. The Color and Angle step can be set in the *Settings/Environment/Guide line and Tracker*. All features of Guide lines can be turned OFF/ON in its Setting dialog. There are three colors of Guide lines in FEM-Desig. Dark green line will be drawn if the Guide line is parallel with the X axis of UCS, orange line will be drawn if it is parallel with the Y axis of UCS and light green (as default color) line will be drawn for any other guide line.





	Stand	Click RB
Along line or	Parallel guide	Parallel guide
arc	line	line
Line's middle	Perpendicular	Perpendicular
or end point	guide line	guide line
Guide line	-	Remove that guide line

Guide lines can be defined on the following ways:

• It appears automatically after 1 second, if mouse stands over a line

This function can be turned off in its Settings by unchecking "Pop up guide line automatically if the mouse stays over a line for more than 1 second"

• Of on a line (straight, arc, circle or any object having these kind of geometry)

Virtual intersections can be found easily with Guide lines



• and drag mouse to define arbitrary guide line. If is pressed, the line is created with finite length (from start point to end point), otherwise it will have infinite length. Middle point of a rectangle can be found easily with Guide lines



Guide lines are drawn in **3D**, but Master Line (thick line) from which angle is measured, can only be placed in UCS or in a plane parallel to UCS.

• **Perpendicular** guide line appears if mouse cursor stands over the end or middle point of a line, otherwise **Parallel** guide line appears.



over guide line will remove that guide line.





- By Over snap-free places, guide line settings will pop up.
- Snapping on a Guide line + \bigcirc^{Shift} will stick the pointer to that guide line.



The **Tracker tooltip** shows **Angle**, **Distance**, **dx**, **dy** and **dz** from the last point. It pop up next to the cursor while drawing and the tracker will be the same color as the Guide line.



Modify region

Modify region command includes editing functions of drawing regions and region-type structural objects (plate, wall, surface load, surface support, surface reinforcement etc.).

- Remove hole
- Split region

It divides regions into several pieces by cutting edge that can be defined manually as custom polyline or by selecting predefined lines or by selecting a region intersects the original one.



Figure: Splitting region

- Perpendicular region offset

It increases or decreases region area parallel with region edges. Offset can be done for all region edges, for all external contour, or for the hole edges.



Figure: Perpendicular region offset

- Direction region offset

It increases or decreases region area along a defined direction. Offset can be done for all region edges, for all external contour, or for the hole edges.



Figure: Direction region offset

- Swap top and bottom faces

It exchanges the upper and lower surface of plates and the front and back sides of walls. It also mirrors objects with variable thicknesses.



Figure: Swap plate region surface (variable thickness)

- Logical operation: Union

It adds user-defined area to a selected region or unifies two predefined regions. The result region always inherits the properties of the goal region selected first.



Figure: Custom drawn part added to a (plate) region



Figure: Unifying two regions into one

- Logical operation: Substract

It cuts out a part of a region, which will be determined by another region intersects the original one. The result region always inherits the properties of the goal region selected first.



Figure: Substract of two intersected regions

- Logical operation: Intersection

It creates a new region from the intersection of two regions. The result region always inherits the properties of the goal region selected first.



Figure: Intersection of two regions

- Logical operation: Excluded "OR" (XOR)

It performs an algebral XOR operation on an existing region and a shape drawn to intersect it or two regions having common part. The common part will be deleted. The result region always inherits the properties of the goal region selected first.



Figure: Deleted intersection part of two regions

Modify solid

Solid operations command includes editing functions of drawing solids.

- Remove hole
- Split solid

It divides solids into several pieces by cutting plane that can be defined manually as custom plane or by selecting existing regions or by selecting a solid intersects the original one.



Figure: Splitting solid with predefined region and custom defined plane

- Logical operation: Union

It unifies one solid with another one(s). The result solid always inherits the properties of the goal solid selected first.



Figure: Unifying three solids into one

- Logical operation: Substract

It cuts out a part of a solid, which will be determined by another solid intersects the original one. The result solid always inherits the properties of the goal solid selected first.



Figure: Substract of solids

- Logical operation: Intersection

It creates a new solid from the common part of two solids. The result solid always inherits the properties of the goal solid selected first.



Figure: Intersection of two solids

- Logical operation: Excluded "OR" (XOR) It performs an algebral XOR operation on two selected solids. The common part will be deleted. The result solid always inherits the properties of the goal solid selected first.



Figure: Deleted intersection part of two solids

Change appearance

The *Change appearance* command changes several properties of objects according to user modifications.

- Change properties

The display properties of objects can be modified in one settings dialog:

- layer of drawing objects,
- line type of lines, edges and axes,
- text style of texts, tables, dimensions etc.

- color of selected objects (different color can be set from the color of the layer assigned to the object),
- pen width of objects.

	Change appearance	
	· · · · · · · · · · · · · · · · · · ·	OK
	Point style	
	Pen [mm] 1	
	Line colour	
	Fill mode	
	Fill colour	
	🖌 AaBbYyZz 🔹 🔸	
		And and a second and a second and a second a sec
	7	
<u> </u>	2	

Figure: Appearance modification

ACCESSORIES

Query

Starting in Tools menu helps to get geometrical informations in the model. You can get the point coordinates, length between two point, angle, region and solid parameters.

Point coordinates

Choosing a point with shows the point coordinates in orthogonal and cylindrical *Global co-ordinate system*.



Figure: Point coordinate

Length

Choosing two points with shows distance between the selected points and their projection to the coordinate axes.

Angle

Choosing two lines with shows the angle between the lines.

Region

Choosing or drawing a region with 🛄 shows area, perimeter, center of gravity (co-ordinates).

Solid

Choosing a solid with 💷 shows volume, perimeter, area and center of gravity (co-ordinates).

Block

Blocks can contain only drawing elements (*point, line, arc, circle, region, solid* and *text*) for editing these elements in one step.



Figure: Block toolwindow

Similarly to filter, for definition, you have to type the name first, then select the desired objects. If required, members can be added with the "+", and removed with the "-" buttons, and exploded with the explosion symbol at the right hand side. With properties "?", you can also change the name of the block.

For grouping structural elements or loads *User defined filter* function can be used.

Archive

This command deletes the result files for save free space in the hard drive. To see the result, you have to run calculation again.

External reference

With this command one can display a drawing of dwg or dxf format as an image in the background. The main advantage of external reference is that modifications made in the source file will be executed in the FEM-Design file each time it is reopened, or Refresh function called.



Figure: External reference toolwindow

After Clicking on Define you can specify the insertion coordinate system:



Figure: Insertion coordinate system of DWG/DXF drawing

With the third click, you specify the coordinate system's second axis. The second axis can define reverse polarity compared to the polarity of the referenced image and thus you can get a reverse image (with wrongly directed texts, etc.).

ΞÔЗ

If the origin in the dxf or dwg file is far away from the drawing, you may not find the image in FD's workspace. Click on "Zoom margin" in this case:



In the import window, you can set the conversion scale, select the desired layers, with different colour representations:

ayer colour and name	Visible	Locked	Empty	Unit conversion	
0	Yes	No	No	1 unit = 1 mm	
RED	Yes	No	Yes	I unit = 1 cm	
GREEN	Yes	No	Yes	① 1 unit = 1 dm	
BLUE	Yes	No	Yes	1 unit = 1 m	
YELLOW	Yes	No	Yes	1 unit = 1 inch	
MAGENTA	Yes	No	Yes	Custom, 1 m =	
CYAN	Yes	No	Yes	1.0 units	
BLACK	Yes	No	Yes	Outer turit	
WHITE	Yes	No	Yes	Custom, Tunit =	
FEM-COLUMN	Yes	No	No	1.0 mm	
FEM-3DPLATE	Yes	No	No	Colour modes	
FEM-3DWALL	Yes	No	No	Colour modes	
FEM-COVER	Yes	No	Yes	Original colours	
FEM-INFO	Yes	No	Yes	Halftone colours	
SYS-AXES	Yes	No	No	Greyscale	
SYS-STORIES	Yes	No	No	One colour:	
SYS-VIEWS	Yes	No	Yes	_	
FEM-MODELING	Yes	No	Yes		
FEM-PDREINF	Yes	No	Yes	Show visible lavers only	
DIM	Yes	No	Yes	Do not show ampty layor	
FEM-BEAM	Yes	No	No	Decelve VDefe	
FEM-TRUSS	Yes	No	Yes	Resolve XRets	
FEM-RESNUM	Yes	No	Yes		
FEM-RESULT	Yes	No	No		
				OK Cancel	

Figure: Import dialog

With the display option in the red box, you can turn the picture on and off.

External rel	ference 🛛 🛛
👤 ? 🛚	
Ø	File path C:\Documents and Settings\femtester\Desktop\ Display

Figure: Turning on the Display option

In "off" state, the file path is printed at the insertion point, and you can set its font parameters in the window below, which opens in default settings during definition, or when clicking on an existing external reference with active:

External reference 🛛 👔 👔	
File path	
C:\Documents and Settings\femtester\Desktop\BiboAndı	
Path font	
AaBbYyZz →	

If the source file has changed, you can refresh it, or refresh it with different parameters **a**.

Table of contents

INTRODUCTION	2
ABOUT FEM-DESIGN	3
FEM-Design Modules	3
STARTING PROGRAM	4
BASIC CONCEPTS	5
User Interface	5
Menu Bar	5
Toolbars	5
Tabmenus	6
Status Bar	6
Application Window	7
Tool Palette and Dialog	8
PROGRAM SETTINGS	8
Environment	9
Drawing	11
FEM	12
Display	12
Window	13
DATA SAFETY	13
Lime Save	14
Recovery files	14
FIENENT TYPE	14
Drawing Elements	15
Objects	15
LAVERS	16
Laver Types	16
Laver Status	17
Editing Lavers	17
CO-ORDINATE SYSTEMS	18
POINT DEFINITION WITH CO-ORDINATES	19
Descartes Orthogonal Co-ordinate System	19
Cylindrical Co-ordinate System	19
Co-ordinate Definition Modes	20
Absolute co-ordinate definition	20
Relative co-ordinate definition	21
Calculator	24
WORKING PLANE	24
GRID SYSTEMS	27
Construction grid	27
Snap grid	28
OBJECT SNAP TOOLS	28
INPUT DEVICES	30
SELECTIONS Box m / Duragmony Education	30
POINT/ DIRECTION EDITORS	33
NAVIGATION Dem	35
Pan Zeom	35
Zoolli	
VIEWS	30 27
DISPLAY MODES	57 40
HIDING FLEMENT	40 /1
TRANSPARENCY	41
Axis	43
	10

STOREY	45
FIND	47
STRUCTURE DEFINITION	49
PROPERTIES	50
Cross-sections	50
Materials	53
Information Pop-up	55
"Properties" Tool	56
Display Settings of Structural Elements	56
GEOMETRY	63
Hole	78
DIRECTION	80
1D MEMBERS	86
Borehole	86
Isolated foundation	88
Wall foundation	90
Beam	93
Column	97
Truss Member	100
Fictitious Bar	102
Intermediate section	104
Apex	106
PLANAR OBJECTS	107
Foundation slab	107
Plate	111
Wall	118
Shell Model	125
Timber Panel	126
Profiled panel	129
SUPPORTS AND CONNECTIONS	131
"Infinite" Rigidity	131
Properties (Non-Linear Behaviors)	132
Supports	134
Connection Objects	140
Rigidity type	147
Nonlinear connection	148
COVER	149
Cover	149
Building cover	151
3D MEMBERS	153
5011	153
SECTION EDITOR	156
CROSS-SECTION DEFINITION	157
Drawing a Section	157
Parametric Section	159
Complex Section	160
CROSS-SECTIONAL DATA	161
DOCUMENTATION OF CROSS-SECTION	164
INSERT SECTION INTO LIBRARY – SECTION LIBRARY	165
LOADS	167
LOAD TYPES	167
LOAD DIRECTION	168
LOAD GEOMETRY	173
"HOLES" IN SURFACE LOADS	184
LOAD CASES	185

LOAD DEFINITION BY TYPES	187
Structural dead load	188
Soil dead load	188
Shrinkage Strain	189
Point Load	190
Deviation load (Automatic)	191
Line Load	192
Wind Load (Automatic) - Regular building	194
Surface Load	191
Wind load (Automatic) - Conoric building	109
Snow Load (Automatic) - Generic bunding	206
Thermal Los A	200
	207
Initial Internal Load	212
Support Motion	212
Mass	217
Seismic Load	219
PREDEFINED LOAD VALUES	222
LOAD DISPLAY SETTINGS	222
EDITING LOADS	225
COMBINATION OF LOADS/LOAD CASES	225
Load Combination	225
Load Group	230
	225
MODEL AND DATA CONNECTIONS	223
FRAME WIZARD	234
MODEL EXCHANGE BETWEEN FEM-DESIGN MODULES	236
CAD DRAWING IMPORT/EXPORT	240
Import	240
Export	242
STRUSOFT XML	244
IFC IMPORT	245
DIRECT DATA LINKS	248
EINITE EI EMENT MESH	251
FINITE ELEMENT MEST	231
ELEMENT TYPES	251
MESH GENERATION	253
Mesh Settings	253
Peak Smoothing	260
EDIT FUNCTIONS	264
RENUMBERING AND DISPLAY SETTINGS	273
Error Handling	274
	276
ANAL 1515	276
GENERAL ANALYSIS SETTINGS	277
Finite Element Types	277
Non-Linear Behavior	277
Second Order Analysis	278
Cracked-Section Analysis	278
Peak Smoothing	279
ANALYSIS FOR LOAD CASES AND COMBINATIONS	280
Displacements	280
Reactions	200
Connection Forces	201
Bar Internal Forces	201
Dat Internal Forces	282
Snell Internal Forces	283
bar Stresses	283
Shell Stresses	284
Equilibrium Check	284

ANALYSIS FOR MAXIMUM OF LOAD COMBINATIONS AND GROUPS IMPERFECTIONS STABILITY ANALYSIS EIGENFREQUENCIES SEISMIC ANALYSIS Methods Steps of Seismic Calculation The Results Summary of Static and Seismic Effects INVESTIGATE	285 286 288 289 290 290 290 291 296 297 290
DESIGN	301
DESIGN LOAD BUCKLING LENGTH FACTORS DESIGN GROUPS FROM AUTO DESIGN TILL FINAL DESIGN Auto Design Manual Design Detailed Result	301 301 302 304 304 304 306 307
RC DESIGN	308
BAR REINFORCEMENT Auto Design Manual Design Detailed Result SURFACE REINFORCEMENT Auto Design Manual Design Shear capacity PUNCHING REINFORCEMENT Auto Design Manual Design Detailed Result STEEL DESIGN STEEL BAR Initial Calculation and Design Parameters Auto Design Manual Design Detailed Result SHELL MODEL Auto Design Manual Design Detailed Result	309 309 312 315 319 320 322 323 324 325 329 331 333 334 334 334 334 335 337 338 341 342 343 344
TIMBER DESIGN	347
LOAD-DURATION CLASSES TIMBER BAR Initial Calculation and Design Parameters Auto Design Manual Design Detailed Result TIMBER PANEL Auto Design Manual Design Detailed Result	347 348 348 348 350 351 353 354 355 356
FOUNDATION DESIGN	358

FOUNDATION DESIGN

DISPLAY RESULT	362
DISPLAY TECHNIQUES	366
Graph	367
Contour Lines	368
Color Palette	370
Sections	372
Display Reaction and Connection Forces	374
Bi-Directional Results	377
Principal Directions	378
Crack Width	379
Design Utilization Results	380
Detailed Bar Result	381
"ABSOLUTE" VS. "RELATIVE" MAXIMUM VALUE	382
STYLE TEMPLATES	383
NUMERIC VALUES	384
Tool Tip	384
Value Labeling	385
EDITING RESULTS	388
BROWSING RESULTS	389
ANIMATION	390
DOCUMENTATION	391
Printing	391
OUANTITY ESTIMATION	395
LISTING	396
DOCUMENTATION MODULE	399
Main Commands	399
Control Panel	400
Steps to Create Your Own Documentation	401
Templates and Automatic Documentation	408
Printing	409
DRAWING AND EDITING TOOLS	411
Draw Menu Commands	411
Solids	413
Text and Field	418
Dimension	419
EDIT AND MODIFY MENU COMMANDS	426
Quick modifications	431
Guide lines and Tracker	434
Modify region	438
Modify solid	442
Change appearance	444
ACCESSORIES	445
QUERY	445
BLOCK	446
Archive	446
EXTERNAL REFERENCE	446