SCIA ENGINEER

SCIA Engineer Basics

Terminology, layout, settings, basic working tools

Contacts

Belgium Headquarters

SCIA nv

Industrieweg 1007

B-3540 Herk-de-Stad

Tel: +32 13 55 17 75

E-mail: info@scia.net

Support Phone

CAE (SCIA Engineer)

Tel: +32 13 55 09 90

CAD (Allplan)

Tel: +32 13 55 09 80

Support E-mail:

support@scia.net

France

SCIA France sarl

Centre d'Affaires

16, place du Général de Gaulle

FR-59800 Lille

Tel.: +33 3.28.33.28.67

Fax: +33 3.28.33.28.69

E-mail: france@scia.net

Agence commerciale

8, Place des vins de france

FR-75012 Paris

Tel.: +33 3.28.33.28.67

Fax: +33 3.28.33.28.69

E-mail: france@scia.net

Brazil

SCIA do Brasil Software Ltda

Rua Dr. Luiz Migliano, 1986 - sala 702 , CEP

SP 05711-001 São Paulo

Tel.: +55 11 4314-5880

E-mail: brasil@scia.net

Germany

SCIA Software GmbH

Technologie Zentrum Dortmund, Emil-Figge-Strasse 76-80

D-44227 Dortmund

Tel.: +49 231/9742586

Fax: +49 231/9742587

E-mail: info@scia.de

Netherlands

SCIA Nederland B.V.

Wassenaarweg 40

NL-6843 NW ARNHEM

Tel.:+31 26 320 12 30

Fax.: +31 26 320 12 39

E-mail: info@scia.net

Switzerland

SCIA Swiss Office

Dürenbergstrasse 24

CH-3212 Gurmels

Tel.: +41 26 341 74 11

Fax: +41 26 341 74 13

E-mail: info@scia.ch

Czech Republic

SCIA CZ s.r.o. Praha

Evropská 2591/33d

160 00 Praha 6

Tel.: +420 226 205 600

Fax: +420 226 201 673

E-mail: info.praha@scia.cz

SCIA CZ s.r.o. Brno

Slavickova 827/1a

638 00 Brno

Tel.: +420 530 501 570

Fax: +420 226 201 673

Slovakia

SCIA SK s.r.o.

Murgašova 1298/16

SK-010 01 Žilina

Tel.: +421 415 003 070

Fax: +421 415 003 072

E-mail: info@scia.sk

E-mail: info.brno@scia.cz	
Austria SCIA Datenservice Ges.m.b.H. Dresdnerstrasse 68/2/6/9 A-1200 WIEN Tel.: +43 1 7433232-11 Fax: +43 1 7433232-20 E-mail: info@scia.at Support Tel.: +43 1 7433232-12 E-mail: support@scia.net	

Document created: 06 / 05 / 2018

SCIA Engineer 18.0

Getting started

Introduction

About program

Program mission

The SCIA Engineer software system has been designed and developed to provide structural engineers and designers with an efficient, comprehensive and robust tool.

Theoretical background

SCIA Engineer is a software system for a static and dynamic analysis of structures and their design to standards. It is grounded on the displacement-based finite element method.

SCIA Engineer does not work with finite elements directly but it exploits structural elements (referred to as members) on which the finite element mesh is automatically generated just before the calculation.

SCIA Engineer can be used to calculate and design structures consisting of 1D members (modelled by linear finite elements) and planar parts such as walls, plates, and curved slabs (modelled by 2D finite elements).

Types of calculation

SCIA Engineer comprises calculation modules for the following types of calculation:

1. linear static calculation (including some non-linear features),
2. geometrically non-linear calculation,
3. dynamic natural vibration calculation,
4. seismic calculation,
5. buckling analysis,
6. and much more.

Code checks

In addition to the calculation itself, SCIA Engineer enables the user to carry out the final design of a structure in accordance with appropriate technical standards.

The "Code Check library" of SCIA Engineer contains a multi-national set of technical standards for various material types, mainly for steel and concrete.

Proper and exhaustive use of program features assumes that a user is well accustomed to the principles of the finite element method, is familiar with appropriate technical standards and conventions, and is a skilled professional in the field of design and calculation of engineering structures.

About documentation

> We recommend undergoing a specialised training for SCIA Engineer organised for you by your local SCIA dealer before using the program for real work.

The documentation contains explanation of the program principles, theoretical background and operation and will provide the user with invaluable knowledge about the SCIA Engineer software.

Purpose and contents

This manual provides an in-depth coverage of SCIA Engineer main module functionality and covers the input, calculation and result-evaluation phases for both frame and shell structures.

Protection of SCIA Engineer

Protection of SCIA Applications

Introduction

This manual provide basic information about settings necessary for running of SCIA applications

This manual provides information about both floating (network) and standalone (dongles) protection.

Standalone protection

The biggest change with respect to the previous version is that starting from version 2010 the dongle must be accompanied with LIC file and in some cases also E2C file. Dongle itself is not sufficient for starting the SCIA Engineer.

This change gives us possibility to provide more flexible licensing solution including time limited licensing, editions etc…

To simplify as much as possible updating of dongles, working with dongle on more computers and other similar use cases it is possible to download LIC and E2C files online from the SCIA server. (see the chapter related to Protection setup)

> Compatibility with older versions: The same dongle can be used to run EPW, older version of SCIA Engineer (up to version 2009.0) and new version of SCIA Engineer (from version 2010.0).

Floating protection

SCIA Floating protection is based on FLEXnet technology of Flexera Software Company. This type of licensing stores commercial modules in “Trusted storage” on the Licence server. Those modules can be used (Checked out) by SCIA application online or borrowed (offline using of modules from licence server).

FLEXnet based protection fully replaces FlexLM based protection from version 2010.0.

> Compatibility with older versions: Older versions of SCIA Engineer (FlexLM protection) can be run with new FLEXnet licence server. In this case the modules are provided by the licence server only if the FlexID dongle is attached.

Trial, Student and Viewer modes

From the version 2011 it is possible to run SCIA Engineer without protection as SCIA Viewer. In this case SCIA Engineer has limited functionality. It is intended to be used for viewing of calculated structures only. In versions starting from 2010 the user can also use "TRIAL" or "STUDENT" modules which are also free but still require some modules.

Student version

- User can use majority of all SCIA Engineer functionality.
- User can edit and calculate a project
- Projects can be saved as *.esad only
- Available for free for registered users (students) only

Trial version

- User can use functionality listed in his licence file only
- User can edit and calculate a project
- Projects are saved into *.esa
- Provided by SCIA dealers to prospect customers

SCIA Viewer version

- Available from version 2011
- User can view existing model and document only
- User can not edit nor calculate a project
- If the project is saved with results then user can view precalculated checks
- Projects cannot be saved neither exported to any file format
- Available to anybody without any protection

Standalone protection

Standalone protection is facilitated by Sentinel Safenet dongles.

New user – activity on user's side

When a hardware lock (dongle) is used for the protection, the following procedure needs to be followed:

- Installation of the Sentinel Protection Installer
- Attaching the Sentinel Dongle
- Import LIC file (and E2C file in case od editions) in "Protection setup" utility

Note: All mentioned actions are usually done automatically during the installation. Following steps are described for special cases when user needs to perform them manually.

Installation of the Sentinel Protection

The Sentinel Protection Installer provides for communication between SCIA Engineer and the Sentinel dongle.

In the **Installation & Software menu** of the installation DVD choose the option **Install Sentinel Driver**. Follow the installation instructions on the screen.

The Sentinel Protection Installer can also be installed manually by executing the file **X:\Tools\Sentinel\Sentinel Protection Installer 7.* .exe** where X represents the DVD drive and * the version number.

> Note: When an older version of the Sentinel System Driver or the Sentinel Protection Installer has already been installed on the computer, this older version must first be de-installed through the Control Panel before installing the new Sentinel Protection Installer.

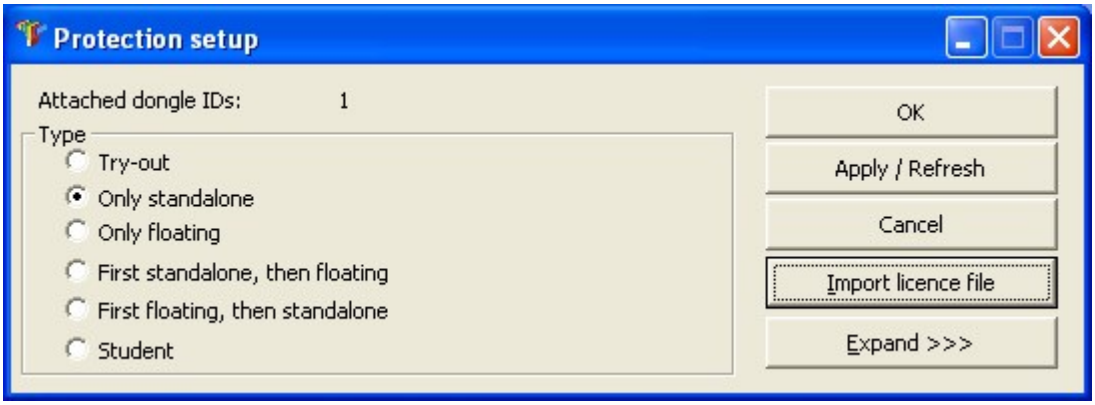
Attaching the Sentinel Dongle

The Sentinel hardware locks are available for USB ports. Attach the dongle to a free port of the computer.

Importing of new LIC file

Each dongle needs to be accompanied with the LockId.LIC file to provide licenses. To get this file on the user's computer is it necessary to do the following:

1. attach the dongle to computer
2. run Protection setup dialogue
3. press [Import licence file]



Then the Protection setup tries to connect to SCIA Activation server and download necessary LIC file. If the file cannot be downloaded from the internet it can be sent via email and then opened from the disk. The open file dialogue is opened automatically when the internet download fails.

The following message will appear after successful import of licence file and user can see the list of his commercial modules in the "Expand" mode of Protection setup dialog.

Note 1:Automatic import of LIC file from the SCIA protection server can be done with attached dongle only.

Note 2: Import of LIC file can be done only if the user is logged with the rights to write into with Licence file (see Setup / Options / Directories)

Note 3: It is possible to change the path to the folder where the licence file will be imported in Advanced setting dialogue (right click on the header of Protection setup dialogue).

Note 4: It is possible to disable automatic downloading of LIC file from the SCIA protection server in Advanced setting dialogue (right click on the header of Potection setup dialogue).

Checking the Sentinel Dongle

To make sure the Sentinel Dongle has been installed correctly the Sentinel SuperPro Medic can be used.

Installation of Sentinel SuperPro Medic

1. Using Windows Explorer go to X:\Tools\Sentinel\ where X represents the DVD drive.

2. Execute the file SuperproMedic.exe and follow the instructions on the screen.

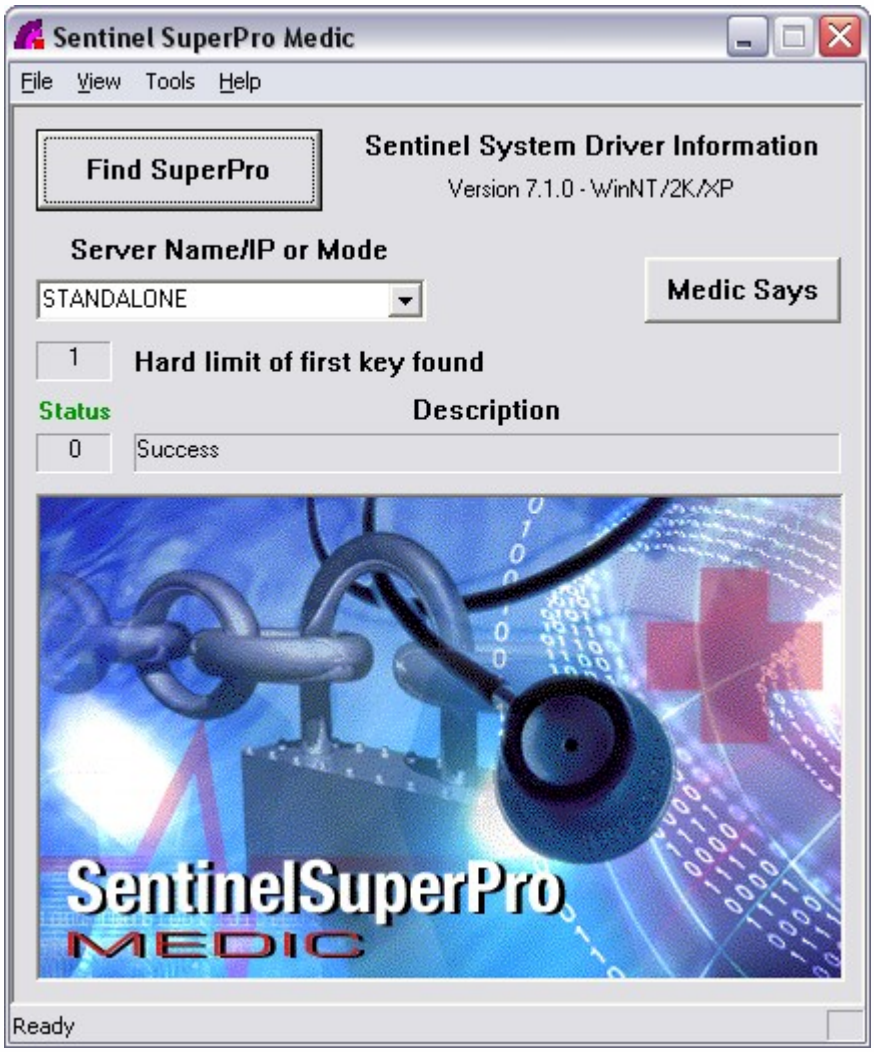
3. By default the SuperPro Medic is installed in the folder C:\Program Files\Rainbow Technologies\SuperPro\Medic

Checking the Sentinel Dongle

1. Using Windows Explorer go to C:\Program Files\Rainbow Technologies\SuperPro\Medic

2. Execute the file SuperproMedic.exe The SuperPro Medic appears on the screen.

3. In the upper right corner of the SuperPro Medic the version of the installed Sentinel Driver is shown. When no version is shown the Sentinel Protection Installer needs to be installed as described earlier.



4. In the field Server Name/IP or Mode choose the option STANDALONE

5. Next press [Find SuperPro] to search for correctly installed Sentinel dongles. A dialog box appears showing the number of keys found.

6. When no Sentinel dongles are found, press [Medic Says] to receive extra information in order to find the cause of the problem.

When the SuperPro Medic indicates that the Sentinel dongle has been found, then this means the dongle is attached correctly to a parallel or USB port and the Sentinel Protection Installer has been installed correctly.

Upgrade of dongle – existing user

After buying a new version of SCIA Engineer or new modules users needs to update his Licence file using [Import licence file] button in the Protection setup dialogue – see previous chapter.

Following message will appear after successful update of Licence file.

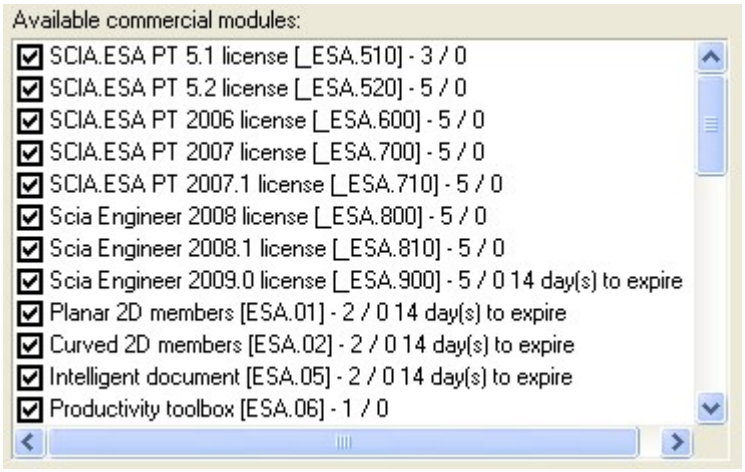
When the user uses some predefined sets, he will need to update modules in those sets (add new modules).

Time limited dongle

Time limited licence is available also for dongles.

It is possible to set different time limitation (different number of days) for each commercial module in the LIC file. The number of days is calculated from the moment of importing the LIC file by user in the protection setup utility. During the import of LIC file, the current date is written into the dongle (number of days is stored in the LIC file).

The number of remaining days (excluding the current day) for time limited module is displayed in the list of modules. Once the time limitation is exceeded, the licence is displayed as "Expired" and will not be available during start of SCIA Engineer.



Note 1: The date in the dongle is changed only if the counter of LIC file is higher that the counter in the dongle (to prevent prolonging of time limited version by repetitive Importing LIC file by the user).

Note 2: It is not necessary to send new dongle to the user to prolong is time limited version. Only performing of [Import licence file] from Protection setup is enough

Note 3: If there is "0 day(s) to expire" it means that this is the last day the module can be used.

Note 4: In case of using more dongle, the number of available days is compared with the date written in the dongle linked with the LIC file.

Using of more dongles together

User can use as many dongles as he can attach to the computer. The LockID numbers of all attached dongles are displayed at the header of protection setup utility.

In the list of available commercial modules is displayed the sum of modules from LIC files related to all attached dongles. Finally the sum of modules from LIC files related to all attached dongles can be used in SCIA application.

Note 1:**[Import LIC file] can be done also with more attached dongles but it is strongly recommended to do it with just one dongle attached.**

Note 2: It is possible to use more dongles with different time limitation. In case that some commercial module is in all LIC files, the one with longer time limitation is taken into account.

Floating (network) protection

This manual is focused on licensing features specific for SCIA License Server but it also contains basic information about managing of licence server. More detailed information about general managing of licence server can be found in Licence Administration Guide provided by Flexera Software.

Supported operating systems

Licence server can be installed on following operating systems

Platform Architecture	Processor Type	Operating System
Windows 32-bit	x86	• Windows Server 2008 • Windows Server 2003 • Windows 7 (Ultimate) • Windows Vista (Ultimate)
Windows 64-bit	x64	• Windows Server 2008 • Windows Server 2003 • Windows 7 (Ultimate) • Windows Vista (Ultimate)

Description of floating protection

SCIA floating protection is type of network protection. It is based on the FLEXnet technology of Flexera Software Company. Modules are stored in Trusted storage on the Licence server in case of this licensing. Those modules can be used online (checked out) or they can be borrowed and used offline. On the licence server there are stored all modules and editions.

The SCIA Licence Server can be installed on any computer in the client's network. This means that it does not have to be the domain or file server.

Each of modules on the SCIA License Server can have different amount of licenses, which can be started simultaneously. The user can determine which modules he wants to use (via Protection setup dialogue) and which modules will remain available on the server for other users.

FLEXnet protection fully replaces FlexLM protection in new versions (from 2010.0).

Managing of licence server is newly (from version 2012) done by lmadmin.exe. This manager has web based user interface which will be described later.

Components used in SCIA floating licensing

SCIA floating (network) licensing consists of following components:

- **Application (e.g. SCIA Engineer):** This application is a client for the SCIA License Server. Setting of protection is done in Protection setup (lockman.exe).
- **Trusted storage on End user's machine (Application TS):** It is Trusted storage on end user's machine. Here are stored modules borrowed from client's licence server. No user's action is needed to install and use application TS.
- **Vendor daemon:** It is the SCIA.exe file which is located on client's licence server machine. Vendor daemon is part of Licence server which communicates with application and administrates requests for modules. It is installed together with the SCIA License Server
- **Trusted storage on server machine (Server-side TS):** It is trusted storage on user's licence server machine. Here are stored modules which user has activated. Those modules are then enabled to user's application via SCIA daemon.
- **SCIA Activation Manager:** application located on client's licence server. It is used to activate (deactivate) modules on client's server from SCIA Activation server.
- **SCIA activation server:** This server is operated by SCIA. It is located on our machine and communicates directly with our database containing information about modules bought by our clients. It proceeds activation, return and other requests from client's Activation manager and send responses to them. The domain name of the activation server is: activation.scia-online.com

Trusted storage - overview

The trusted storage is space on the licence server where information about activated modules is stored. This storage is secured against unauthorized use.

Trusted storage contains license rights for FLEX enabled applications.

Important: Some computer identities are used as bindings for securing Trusted storage. It is strongly recommended to return licenses to SCIA activation server before any (SW or HW) changes on licence server.

In some cases when security rules are violated the trusted storage become untrusted. It can happen by legal action but also by illegal attempt to unauthorized usage of the licenses. In such case no modules are provided by the licence server and untrusted licence is marked in The Activation manager (see further). Licence administrator should contact SCIA support in this case.

Compatibility with older versions

Older versions of SCIA Engineer (FlexLM protection based on *.DAT file) can read license from new SCIA Licence Server.

New versions of SCIA Engineer (2010 and later) requires SCIA Licence Server with version of FLEXnet 11.8 or newer. It is recommendable to use latest version of SCIA Licence Server provided by SCIA.

In case of using licence server with older version of FLEXnet than 11.10 together with SCIA engineer 2012 (or latest patch of 2011.1) there can be problem with correct interpretation of "@localhost" address. In this case it is recommendable to upgrade SCIA Licence server to latest version or use exact IP adress (127.0.0.1) in the Protection setup.

Installation of SCIA License Server

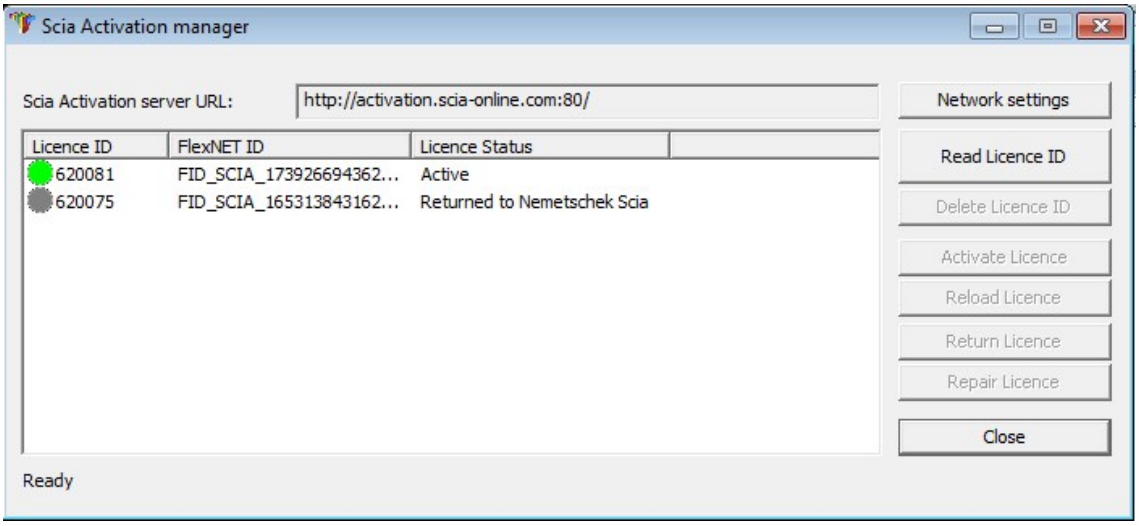
Installation of SCIA Licence server is part of the installation DVD. It is necessary to go through all steps only. No special input is needed.

The default path where the SCIA Licence server is installed is c:\Program Files (x86)\Common Files\SCIA\LicenceServer\ (can be different for different OS).

Activation of modules on SCIA License Server

To activate modules on SCIA License server user needs to receive the LID file from SCIA and go through following steps:

1. Run SCIA Activation Manager



2. Make sure that FlexID dongle is attached and working properly
3. Press **[Read Licence ID]**and select the LID file using the "Open file dialog"
4. Select the licence in the list of licenses by clicking on the licence number
5. Press **[Activate Licence]**
6. The information message is displayed after successful activation
7. SCIA Activation Manager can be closed now

Activated licence is indicated by the green icon.

After the successful activation it is necessary to restart the SCIA License server.

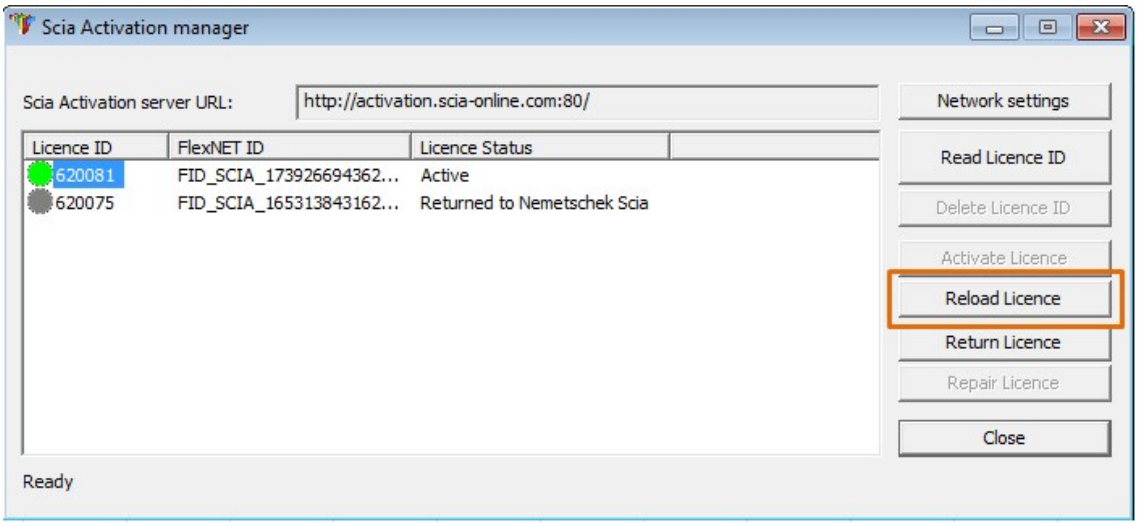
Upgrade of user's version

When user buys new modules, it must be indicated in SCIA database. Then the user (or his IT administrator) needs to update licenses on his SCIA License Server.

The update of modules on the SCIA License server consists of two steps:

1. Return current modules from user's trusted storage to SCIA activation server (return request)
2. Activate modules from SCIA activation server (it will be activated including new modules)

Those two steps can be done manually or at once by pressing **[Reload licence]**.

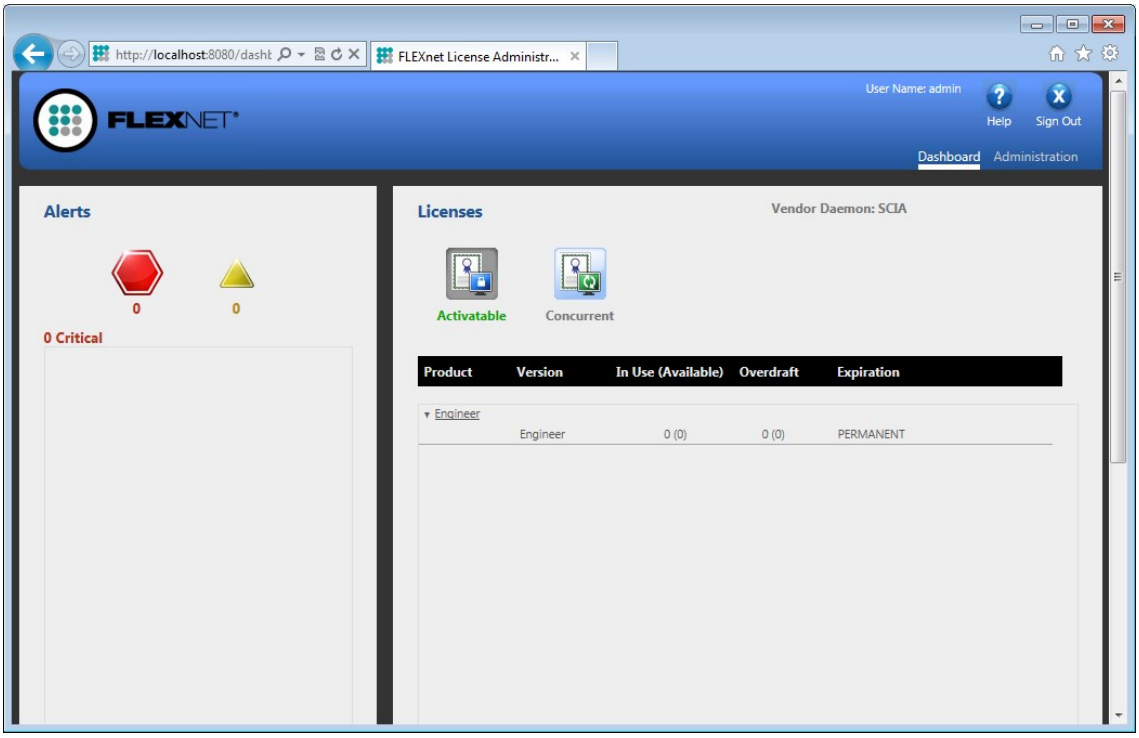


Configuration and start of SCIA Licence server

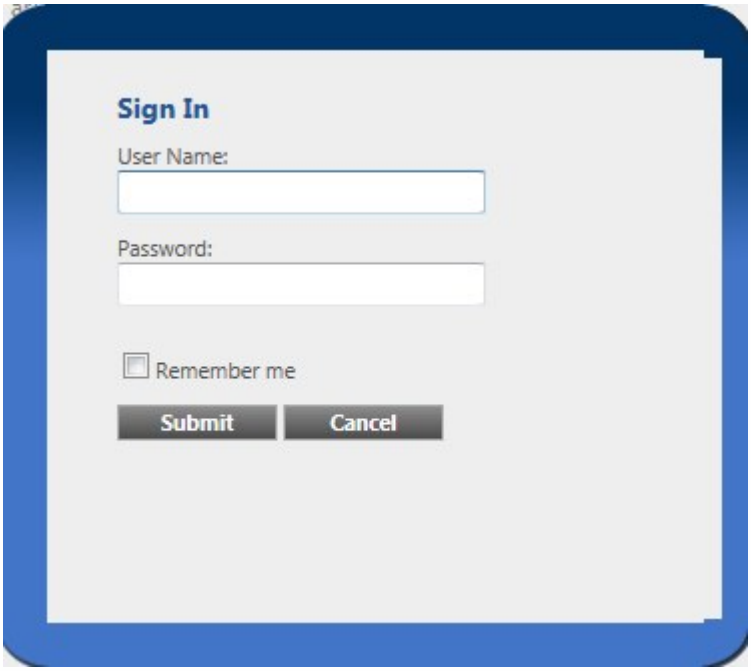
> Note 1: The licence server is configured automatically during the installation of SCIA Licence Server. In usual cases the following steps are done automatically.

> Note 2: More detailed information about general managing of licence server can be found in Licence Administration Guide provided by Flexera Software.

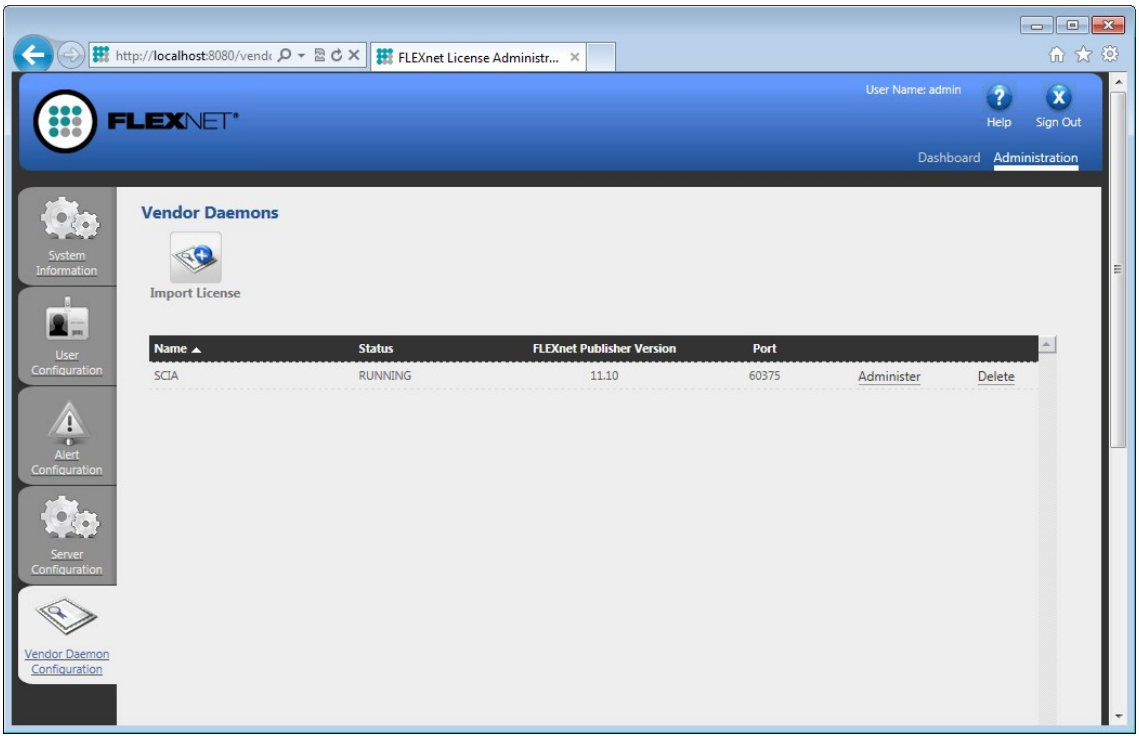
1. make sure that there is running process lmadmin.exe on the license server machine.
2. start Internet browser and navigate to http://serveraddress:8080 (e.g. http://localhost:8080)



3. Click on [Administration] and sign in (for the first login use "admin" for username and password)

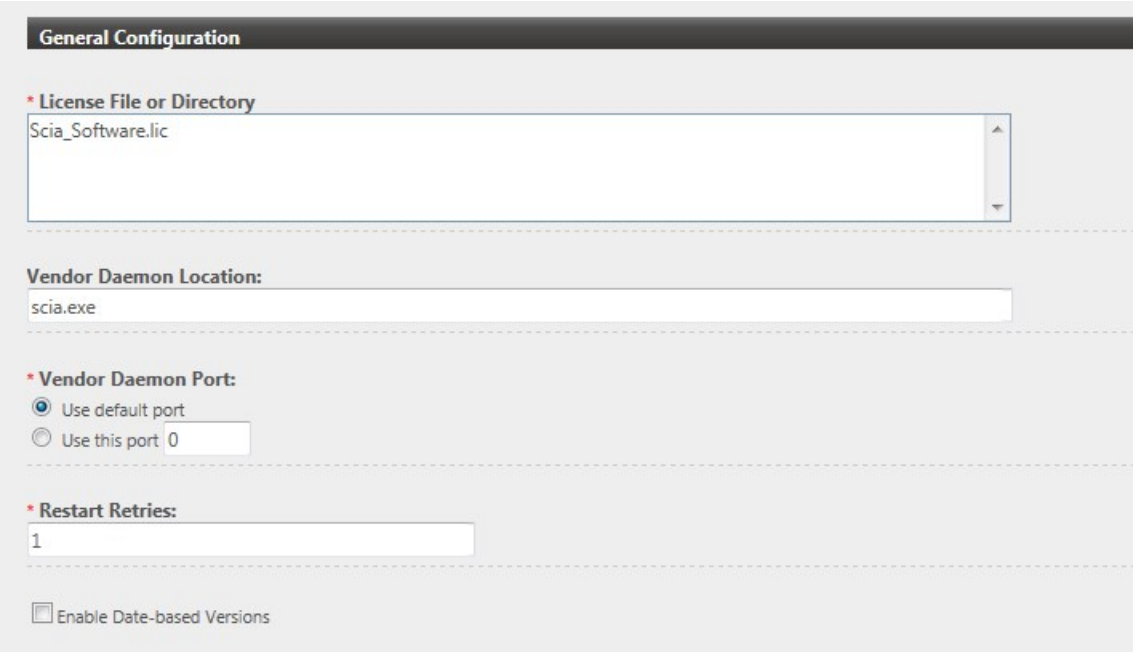


4. Go to "Vendor daemon configuration" and click "Administer"



5. In the "General configuration" section it is necessary to have correct **path to licence file** (SCIA_Software.lic or *.DAT) and correct **path for the vendor daemon** (SCIA.exe).
Vendor daemon port is the port which is used for communication between scia.exe and lmadmin.exe. it is not used for communication between SCIA Engineer and licence server.
Restart retries specifies whether the licence server will be automatically started after restart of licence server machine

(also restart after sleep mode). **Put here the value 1** to enable automatic restart of license server .



6. In the "Vendor daemon log" section it is possible to modify the position of log files or show the log file.

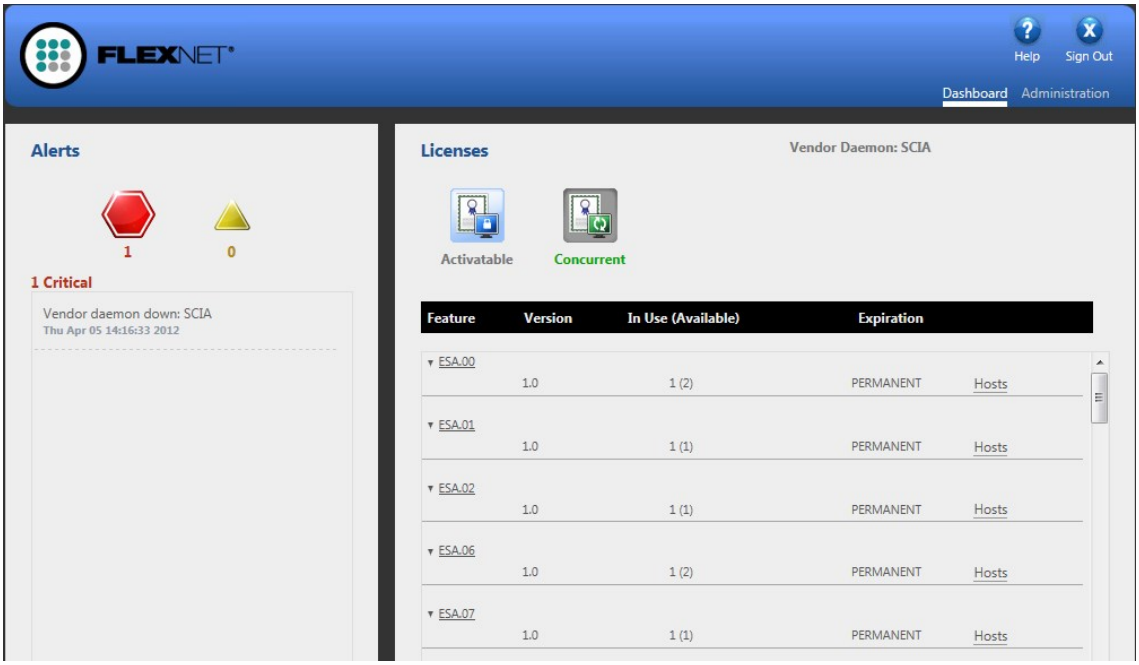
7. Changes in configuration must be **saved**

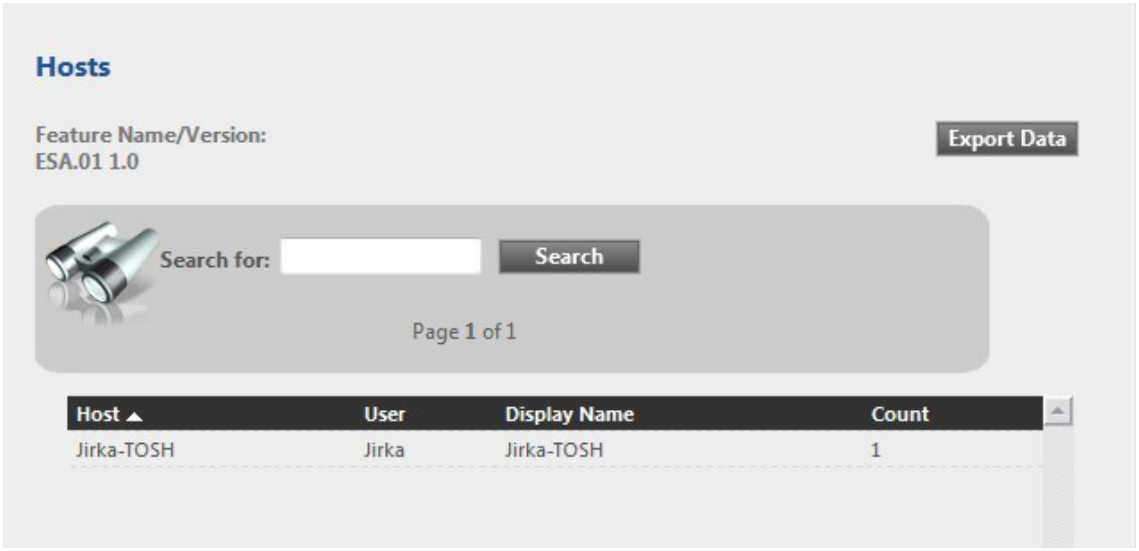
Once the daemon is configured correctly, it can be started using the "Start" button.

Running daemon can be stopped using "Stop" button. Or it is possible to reread the licence file.

Checking the status of licence server

The status of SCIA Licence server can be checked on the "Dashboard" of the web based interface. It is possible to see alerts and warnings here and to see which licenses are available at the server. There are provided information about availability, expiration or current users (Hosts) for each module.

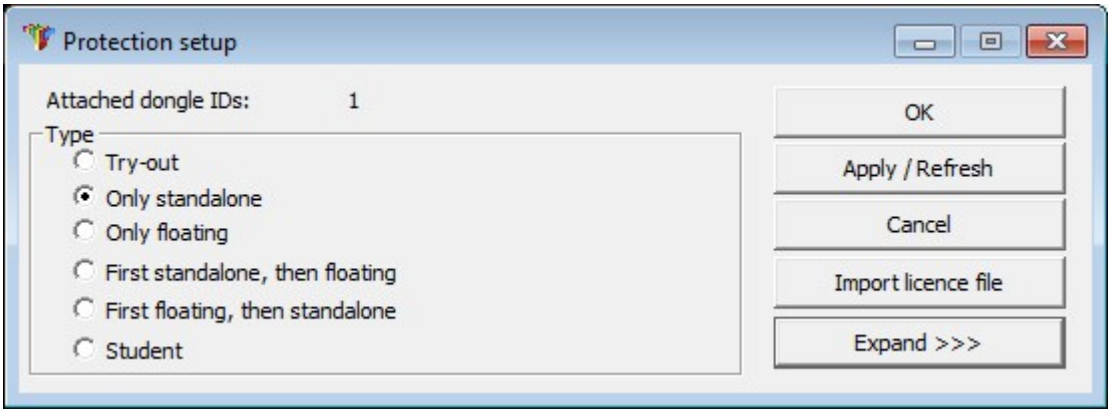




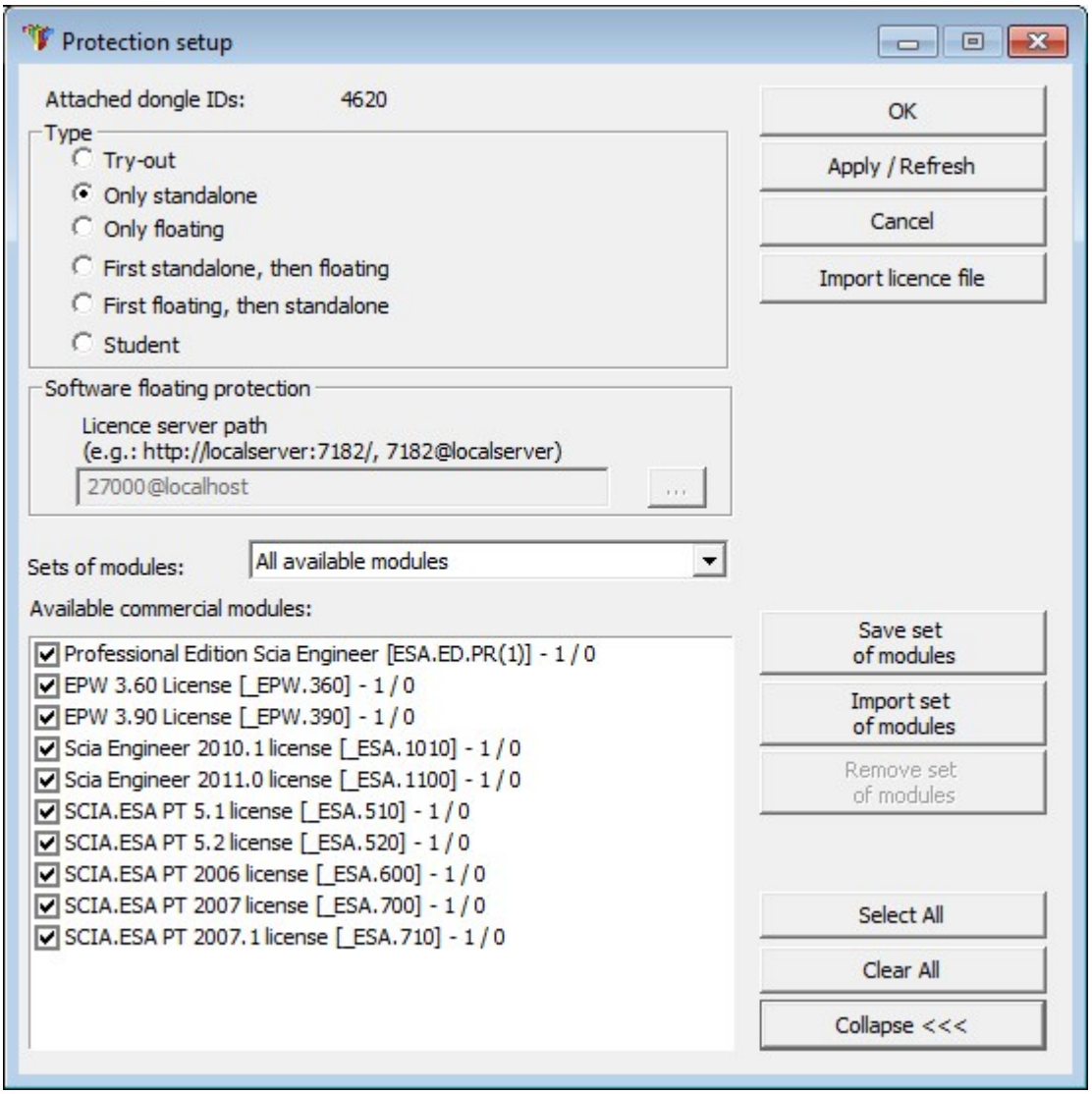
Setting of protection in SCIA applications

Standalone protection

The setting of Protection is done is Protection setup dialogue (lockman.exe). In the most cases user needs just to select "Only standalone" type of protection. This selection ensures that all modules from attached dongle and LIC file are used.



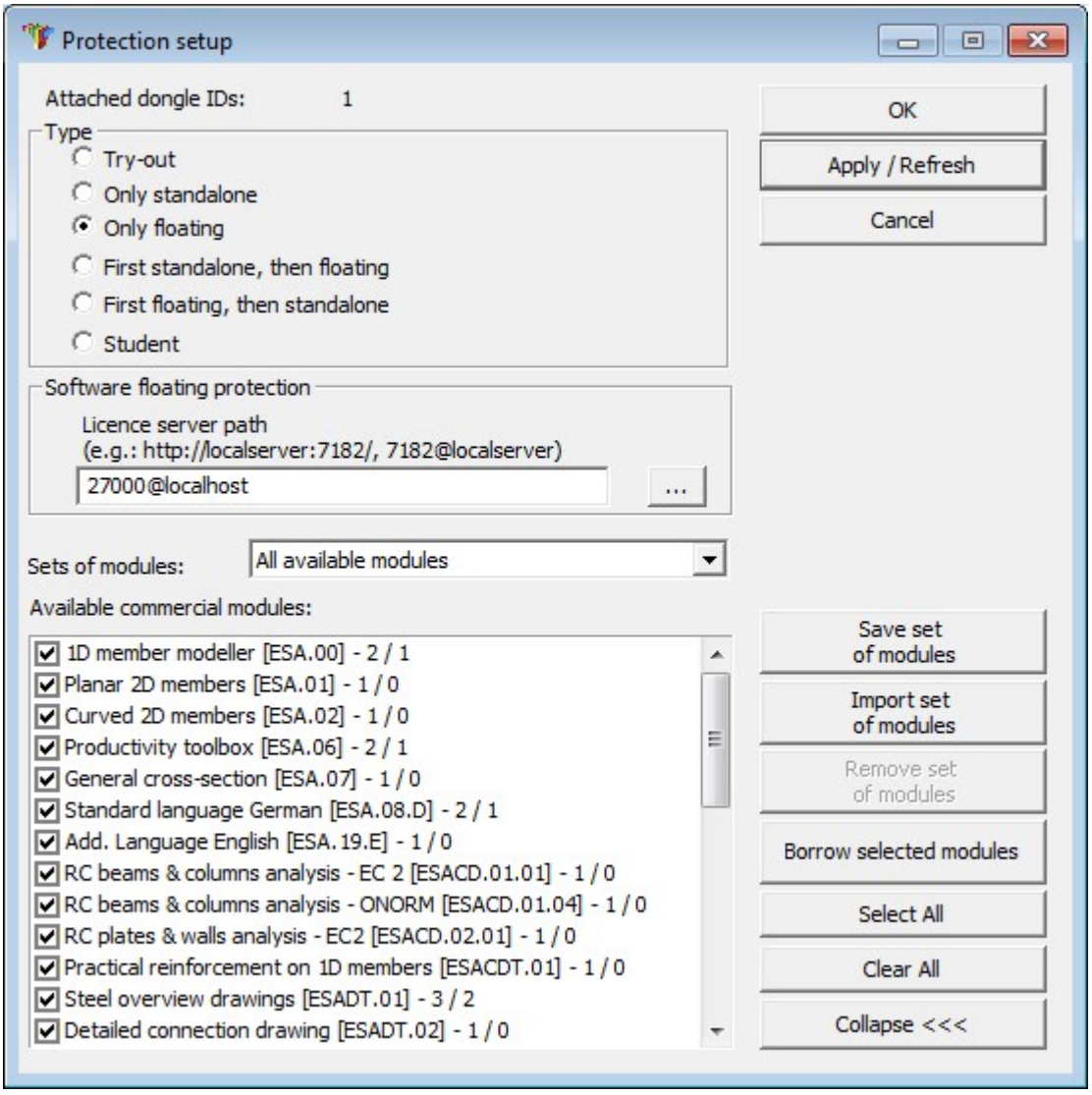
User can check list of his modules in "Expanded" mode of the protection setup.



In case the list of modules is empty or outdated it can be updated using [Import licence file] button. (see chapter related to Upgrade of existing users).

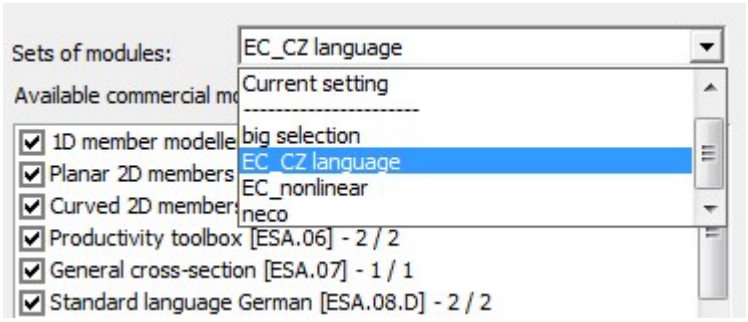
Floating protection

The setting of protection is done in Protection setup dialogue (lockman.exe).



In the Protection setup dialog the user needs to set protection type to “Floating” and fill in the correct port and path to Licence server. (e.g. 27000@localhost or 27000@my_server_name). The list with available commercial modules is displayed after pressing [Apply/Refresh].

User must select modules which he wants to use in SCIA application. Selections of modules can be also saved as "Sets". Those sets can be then quickly accessed in combo box "Sets of modules".

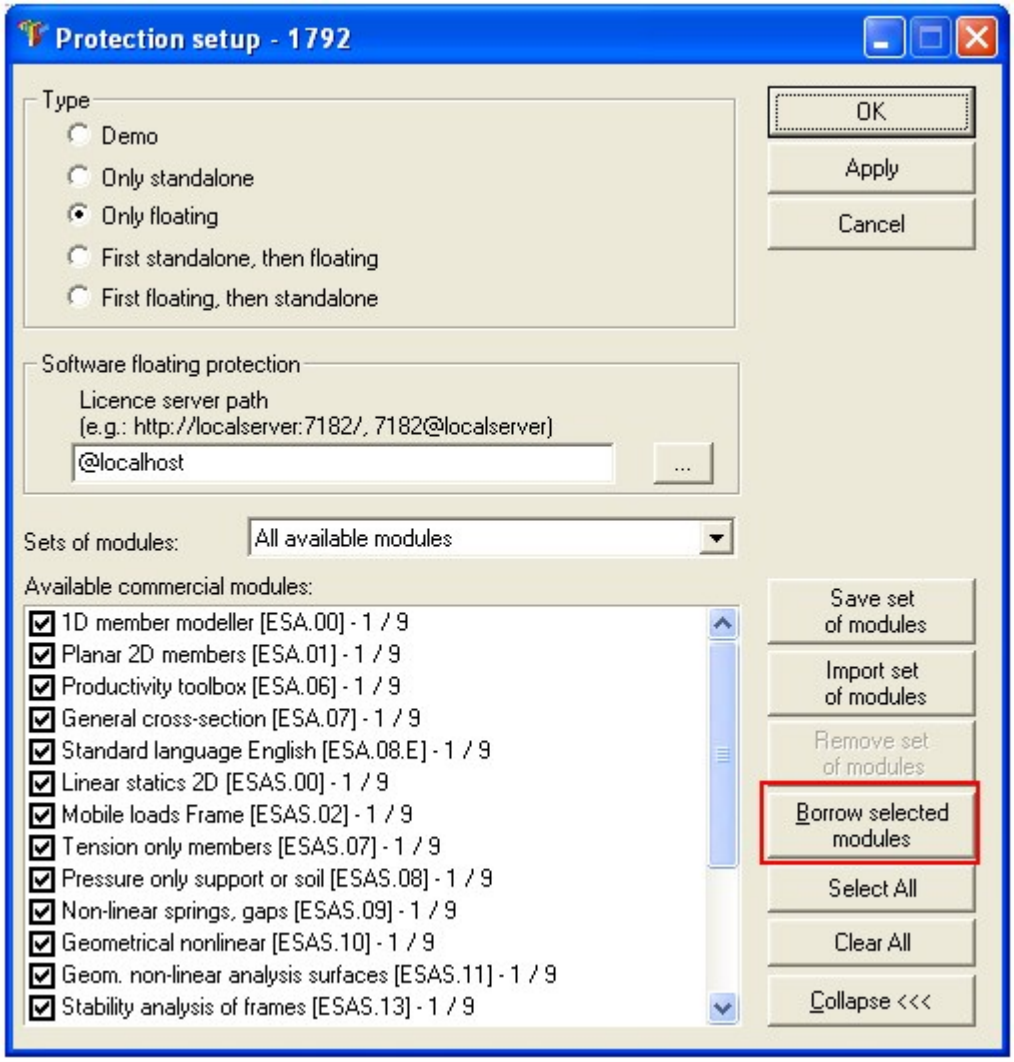


Borrowing

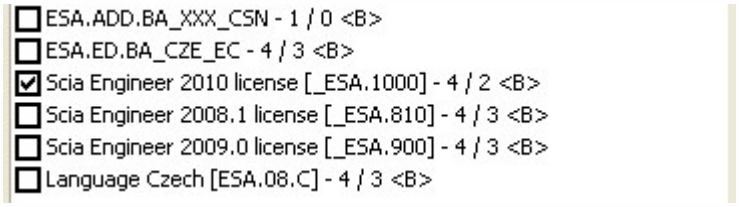
Borrowing enables to the end-user to take some modules from Licence server to his local computer and use them also off-line. The borrowing is done for specified time (maximal one month). After this time borrowed modules are automatically returned back to SCIA Licence server.

> Note 1: If the user wants to use borrowing, it must be explicitly enabled by SCIA. The borrowing can be enabled for each separate commercial module (edition/ package).

Borrowing of modules is controlled from Protection setup utility:

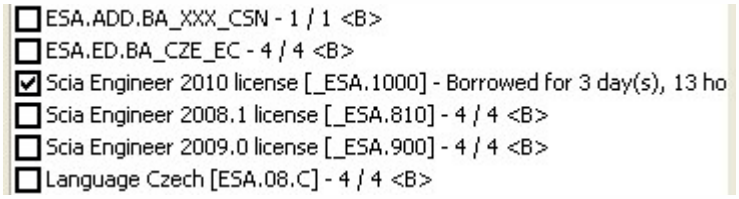


Modules which can be borrowed are indicated by the <B> (since the version 2011).



If the user wants to borrow modules, he needs to select (check) modules he wants to borrow and press **[Borrow selected modules]** button. Then he is asked for number of days (calendar). All modules that are currently selected (checked) are borrowed (if it is allowed for them).

Successfully borrowed modules are indicated by the text with time remaining to automatic returning of modules (since the version 2011).



In case the borrowing is not anabled for selected module the warning message is received.

If the user wants to borrow some more modules (or other selection of modules) then he has to return currently borrowed modules at first. After returning he can make new selection of modules and perform Borrow action again.

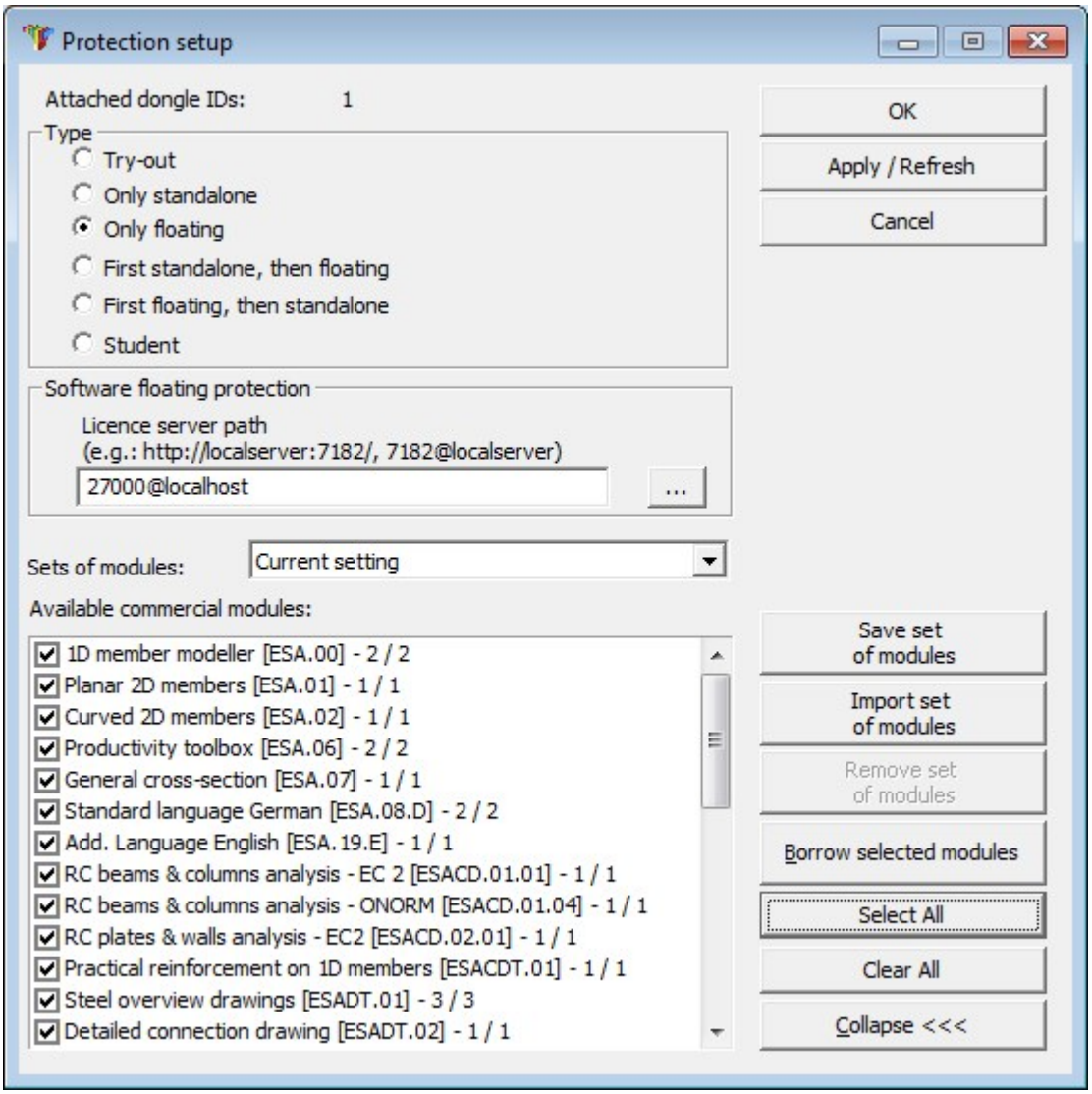
Note: The “early return” is possible only if the end-user is connected to his SCIA Licence server.

It is highly recommended to replace the path to the licence server with one space character (" ") in the protection setup utility. Otherwise the starting and working in SCIA Engineer can become slow due to system searching for the licence server which is not accessible.

List of protection related tools

Protection setup

The protection setup is a separate utility. It can be launched using Lockman.exe. This is basic tool for end-users to configure their licensing.



User can select one of the following protection types:

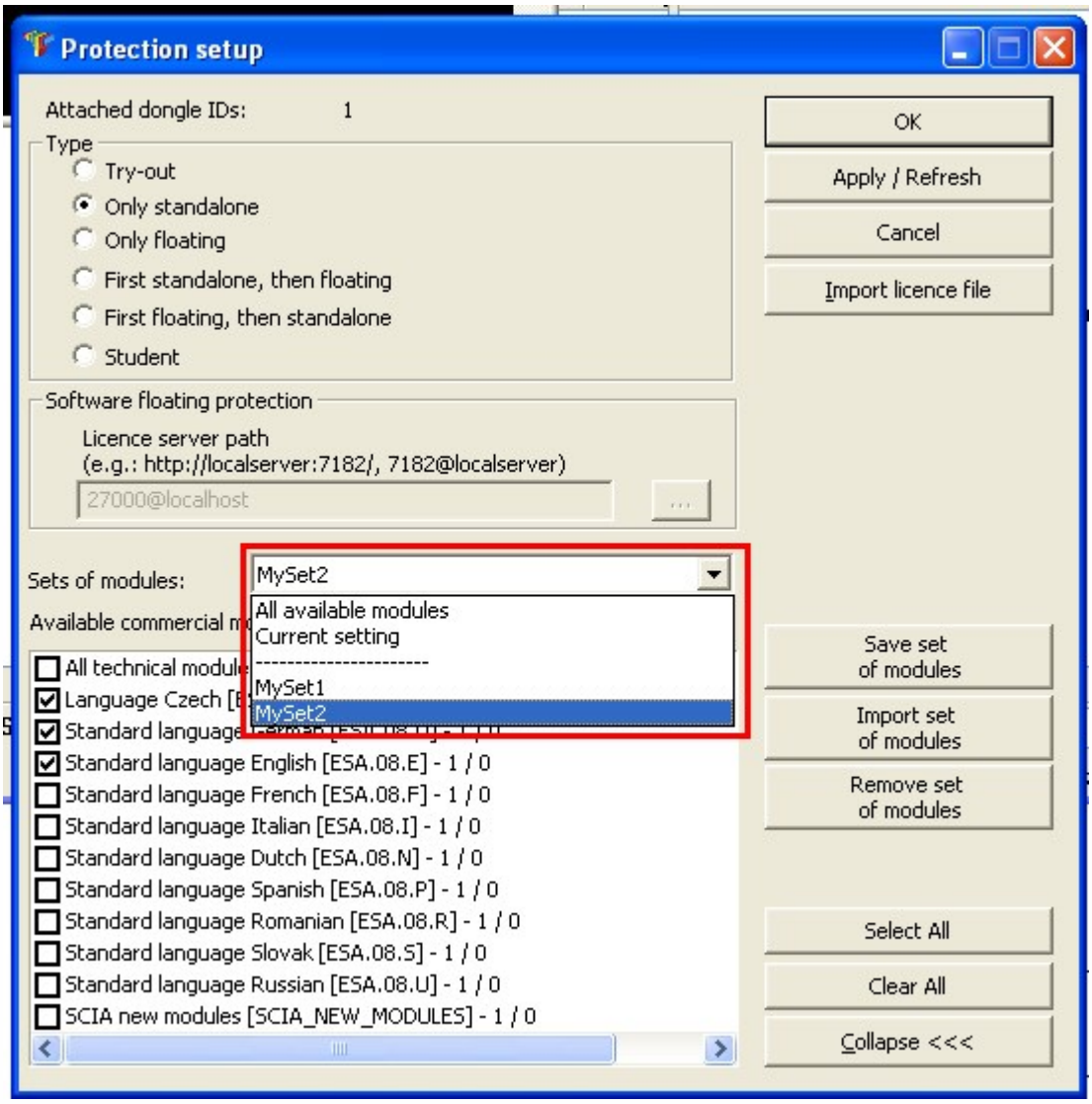
1. **Trial** – it runs SCIA Engineer in Trial mode if there is a "TRIAL" module in the local trusted storage (local SCIA Licence server)
2. **Only standalone** – reads modules only from the dongle and LIC file
3. **Only floating** – reads modules only from the SCIA Licence server
4. **First standalone then floating** – Selected modules are taken primarily from the standalone dongle. If some selected module is not available at the dongle it is taken from SCIA Licence server
5. **First floating, then standalone** – try to read modules from SCIA Licence server, if it is not accessible, then modules are taken from dongle
6. **Student** – it runs SCIA Engineer in Student mode if there is a "STUDENT" module in the local trusted storage (local SCIA Licence server)

If the dongle is attached then its LockID is displayed in the header of the dialog.

When the floating protection is selected, then user must specify the path to his FLEXnet Licence server.

At the bottom of the dialog there is the list with commercial modules which are available in the LIC file or on the user's SCIA License server. Here the user can select modules which he wants to use in SCIA application. The selection of modules also can be saved to Sets.

Sets are predefined settings of commercial modules. They are stored in files in User\protection\Sets directory. In the Protection setup dialog the user can choose between existing sets of modules in the combobox. SETs are intended to be used for floating protection (to make selection of modules easier), however, it can be used also with standalone protection.



[Save set of modules] saves the current selection of modules into defined set (the name of the set is according to file name specified by the user).

SCIA Engineer can be started with specific predefined set of modules (i.e. selection of modules). This feature enables the user to switch between predefined set of modules without starting Protection setup.
The syntax is: esa.exe /Ccustom_filter_file_name where "custom_filter_file_name" represents the name of the predefined set of modules without file extension. If spaces are used in the name of the set the syntax is esa.exe "/Ccustom filter file name".

[Import set of modules] modify the current setting of modules according to selected set, but does not change the name of the current set.

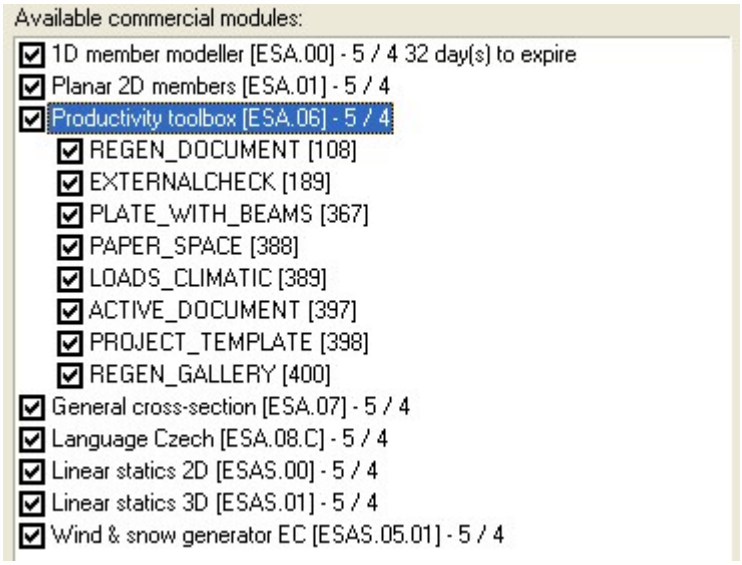
It can be also used for importing SETs from another computer (e.g. if prepared by IT administrator on the server). After the import the SET is copied to USER\Protection\Sets directory and can be used.

[Remove set of modules] deletes current set.

Note 1: Files with Sets can be manipulated (copy, delete, edit) also manually.

Note 2: Numbers next to commercial module are valid only for floating licensing and mean: total number of modules on the server / available number of modules at the moment.

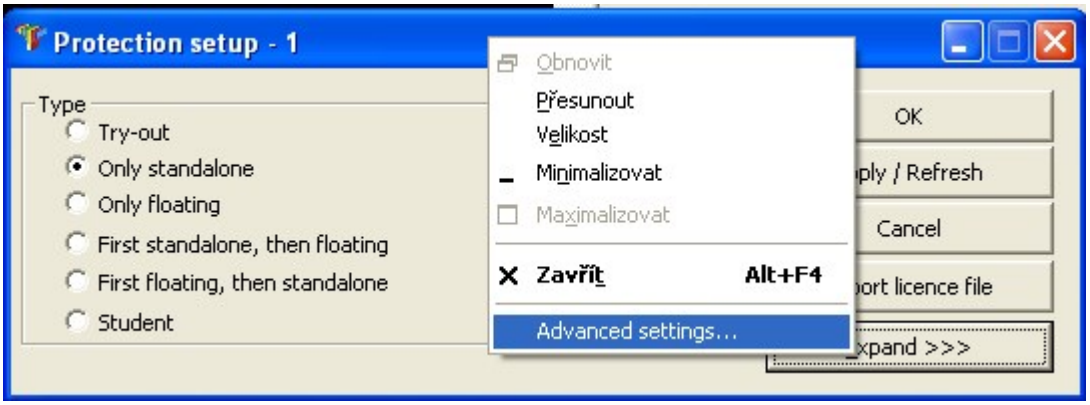
Each commercial module in the list can be expanded (by double-click). Then the technical modules contained in the commercial module are listed.

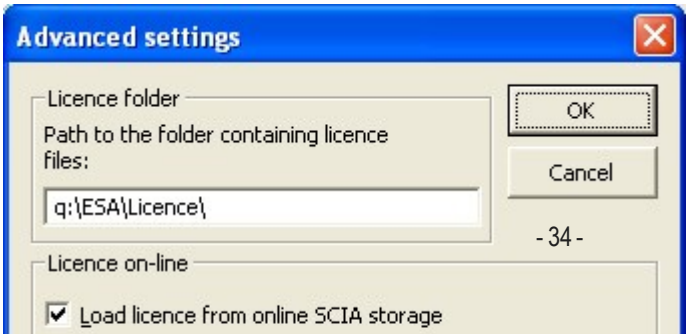


Changing of path to the licence folder from Protection setup

It is possible to change the way to the directory with license file from Protection setup dialog

Use right-mouse-button on the header of the dialog and start Advanced setting. Then you can enter different path to the directory with licence file.

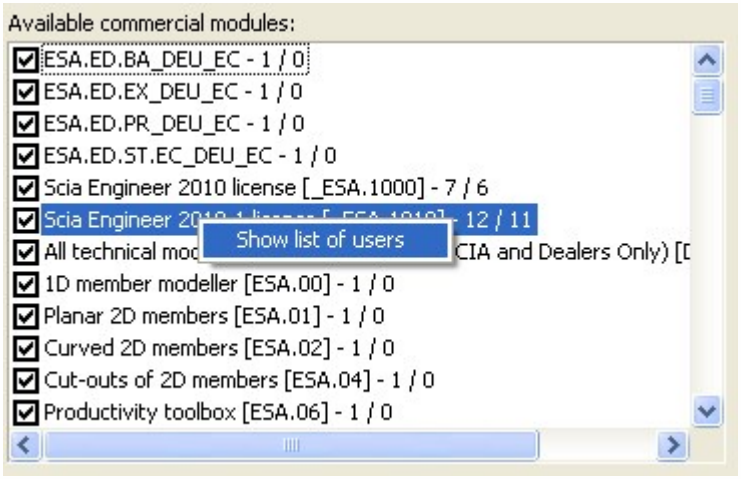




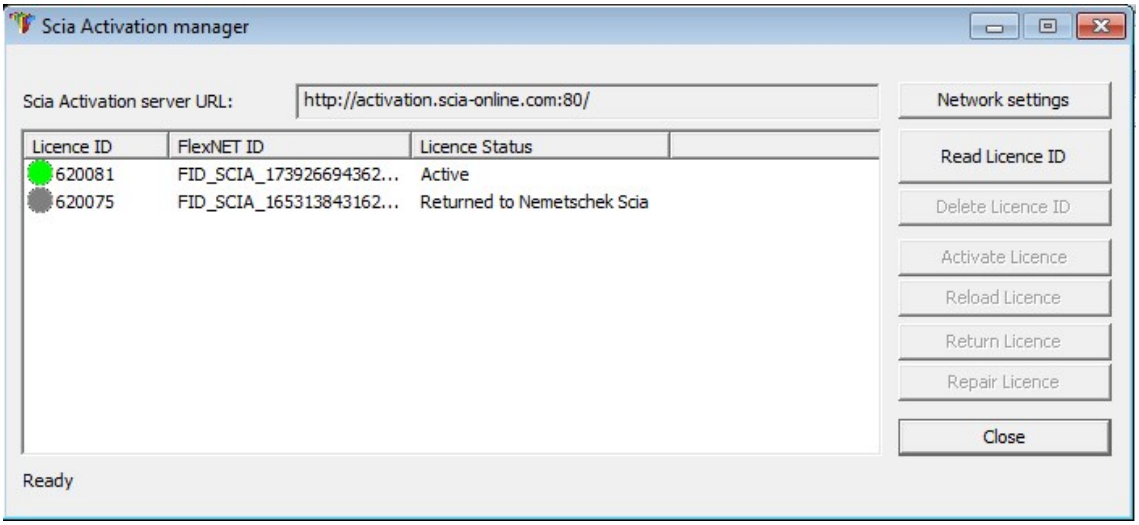
Displaying of usage of some module from Protection setup

This feature is available for the floating licenses only.

To learn who is currently using some module, use right-mouse button above the module and press "Show list of users. Then you can see who is using the module.



SCIA Activation manager



This utility needs to be installed on user's licence server. Its installation is part of SCIA Licence server installation.

The utility is intended to be used by client's licence administrator. It enables to administrate licenses on SCIA License server. Following actions are available:

- **Add new license** (one licence represents one LockID) in the list
- **Delete license** from the list

And also perform transactions with SCIA Activation server

- **Activate** a license
- **Return** a licence to SCIA Activation server (can be lately again freely activated).
- **Repair** damaged trusted storage - it must be explicitly allowed by SCIA.

Note: It is recommended to return licenses to SCIA Activation server before any changes of client's licence server machine. it will prevent damaging of the licence.

Running the program

Starting program

Depending on your personal habits select one of the following ways:

Short-cut on desktop

1. If the short-cut has been placed on the desktop automatically during the installation, proceed to step 3.
2. Place the short-cut on the desktop.
   1. Click the right mouse button on the desktop.
   2. Select New > Short-cut command.
   3. Browse the hard disk to find the folder you have installed SCIA Engineer into.
   4. Select ESA.EXE and finish the New Short-cut command.
3. Double click the short-cut to start the program.

Start menu

1. Click Start button on the left of Windows status bar.
2. Select (All)Programs > SCIA Engineer XXXX.X > SCIA Engineer XXXX.X.

Windows explorer or another file manager

1. Browse the hard disk to find the folder you have installed SCIA Engineer into.
2. Select ESA.EXE file and double click it to start the program.

Program files and folders

The program uses numerous folders and file types to store its data.

Folders

Program folders

main program folder	It contains the program executable and auxiliary files.
set	It contains initialisation files for a new project. (The information stored here may be overridden by the data from files saved in User folders, if available.)
db	It stores system databases (e.g. materials, bolts, etc.)
prof	It contains cross-section databases.
DocumentTemplates	This folder offers a set of default templates for document. Its contents is automatically copied into the appropriate user folder on first program run.
GraphicTemplates	This folder offers a set of default templates for graphical outputs. Its contents is automatically copied into the appropriate user folder on first program run.

Note: All the program folders are ReadOnly.

User folders

set	It contains initialisation files for a new project.
db	It stores files with user-defined databases.
prof	It contains cross-section databases.
DocumentTemplates	This folder holds the templates for document.
GraphicTemplates	This folder holds the templates for graphical outputs.

Note: The destination of this folder may be adjusted in the appropriate program setup dialogue.

Temporary folder

This folder stores all the information that the program needs to store during its run.

Note: The destination of this folder may be adjusted in the appropriate program setup dialogue.

Project folder

This folder stores the user-crated projects.

Note: The destination of this folder may be adjusted in the appropriate program setup dialogue.

Files

ESA	Project file
ESAD	Project file that has been created in a demo or student version of the program. It cannot be read into a standard licensed version of the program.
EPW	Project file created in Esa Prima Win
DB4	Database file
SET	Initialisation file for the adjustment of project and user interface.
OTS	File with table templates for document.
EPD	Template for drawing in Paper space.

Update of the program

You can use menu item Help > Check for Updates to update your installation of the program.

Note: If, for any reason, it is not desirable that the user of SCIA Engineer use this possibility to update the program (e.g. in large companies with IT department controlling the installations of the software), it is possible to disable this item by a simple change of the value of one registry entry. If the value HKEY_ CURRENT_

USER\Software\SCIA\Esa\10.0\Admin\Settings\EnableUpdateMenu = 0, then the Check for Update menu item is disabled.

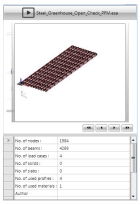
Project manager dialogue

Project manager dialogue is a tool which helps user to open, create or browse projects. It allows to adapt protection settings, use links and other options to manage the SCIA Engineer or find more information about it.

All these functionalities are stored in one dialogue.

Project information preview

The biggest advantage of the dialogue is a project information preview. Each project is complemented by a picture of a structure and basic information such as name, author, number of members, calculated results ...



- The top "play" button opens the selected project;
- Preview displays the slide-show of images from the project, it can be listed by arrows;
- The table contains the standard information about the project data, structure and calculation;

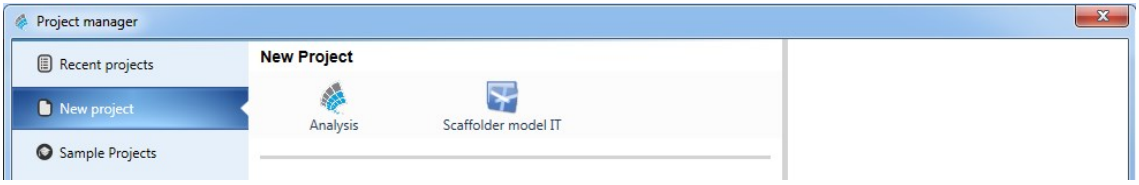
When the Project manager dialogue appears?

The dialogue is displayed when user clicks on icon "Start project manager" on the basic application toolbar.

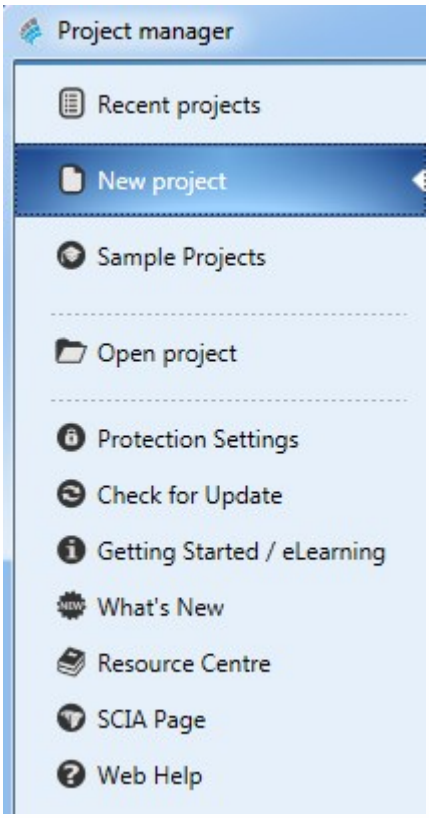
Using Project manager dialogue

New project

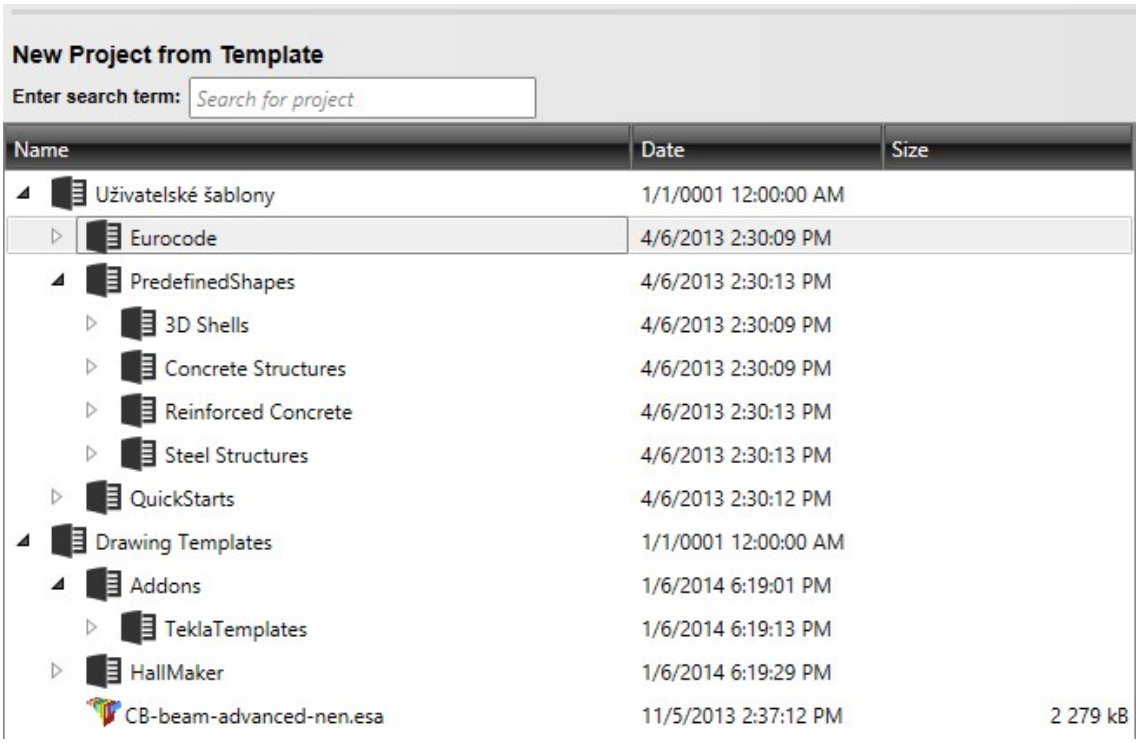
In the New project category user can select the type of a new project.



It is possible to switch to any other possibility on the left side.

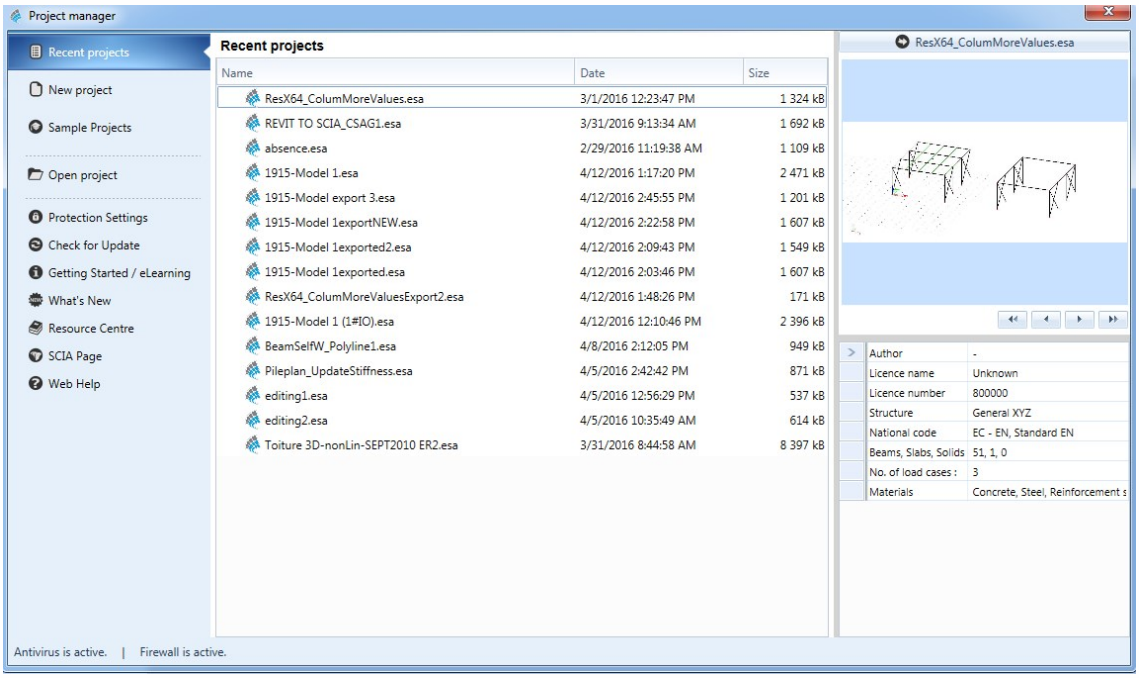


It is also possible to create a new project using predefined project template.



Recent projects

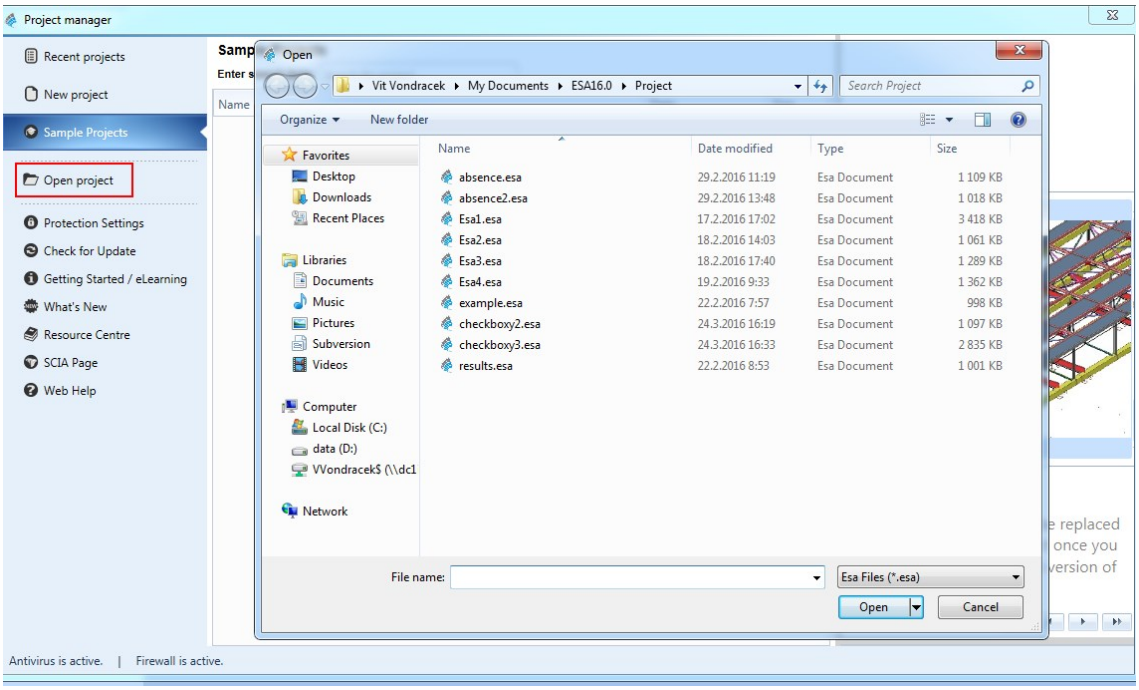
User can select and open one from recent projects he worked on in the past. List of those projects is the same as is available in SCIA Engineer main menu. Opening of the project can be done using double click.



List of recent project is updated after restart of SCIA Engineer.

Open projects

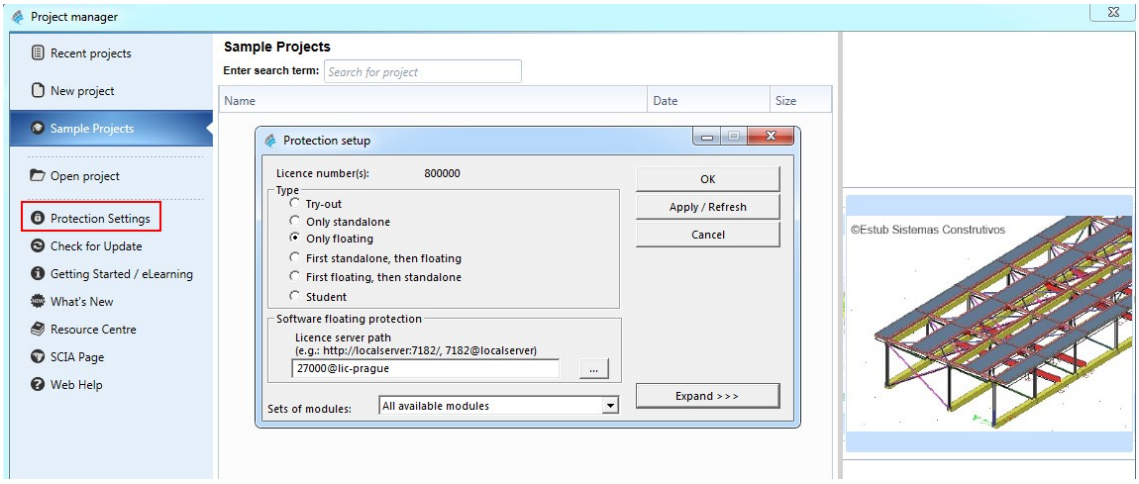
This button launch the system Open file dialogue and enables to select any wanted SCIA Engineer project file.



All standard functions are supported (Copy, Paste, New folder etc.).

Protection settings in Start project dialogue

The Protection setup dialogue can be started using the button Protection setting. Changes done in the protection setting are respected after a small delay. In case the SCIA Engineer is running, any changes done in the protection setting will be applied after restart of the application.



In case of missing license the Protection setting button becomes flashing and red.

Links to web pages

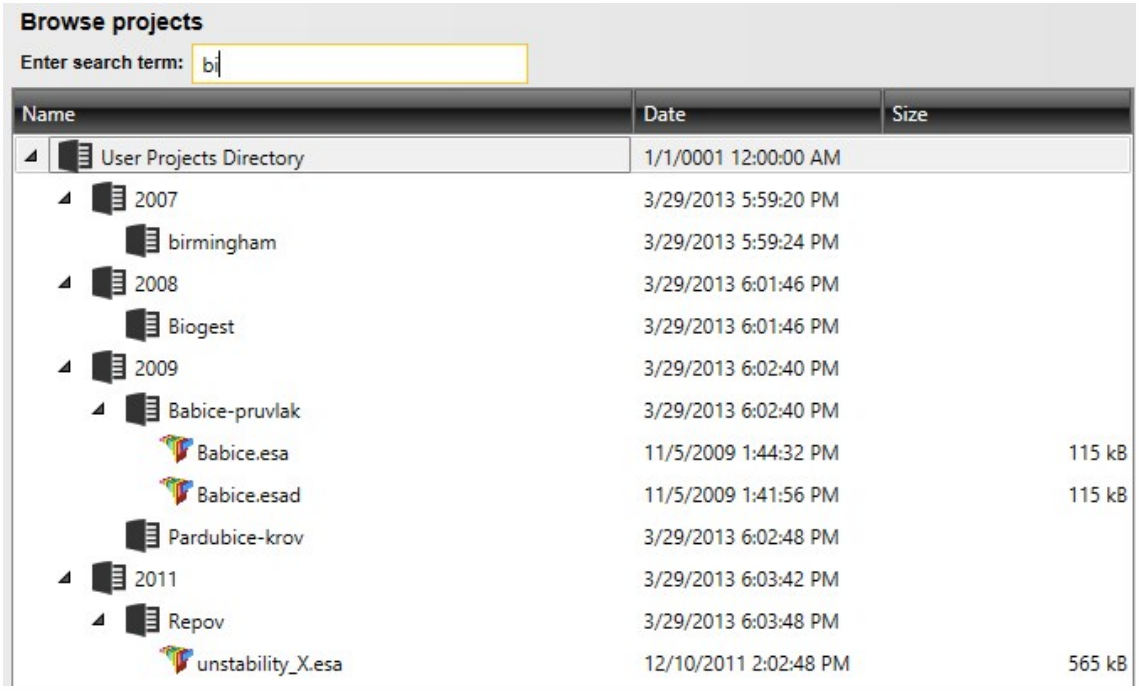
The bottom part of the left panel contains links to important web-pages.

Search and sort the list of projects

The list of projects which is available for above mentioned tabs has common behavior.

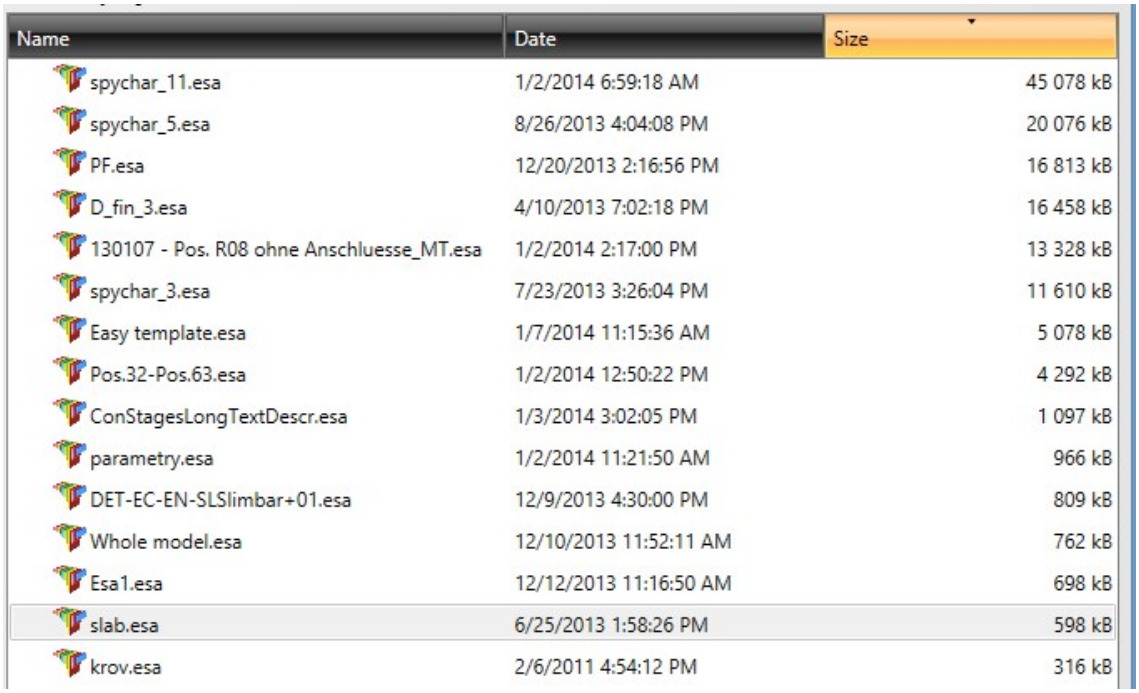
Search

It is possible to filter it using the "Search" field.

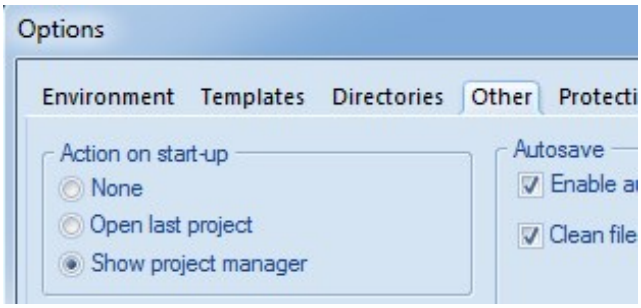


Sort

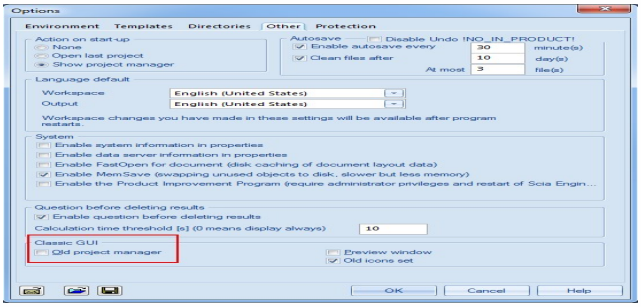
It is possible to sort the list according to one of columns. The sorting is switched by clicking on the column header. It have sequentially following three possible values (none, ascending, descending).



The Project manager dialogue is skipped in case the "Open last project" is selected in the setting of Action on start-up in SCIA Engineer.



User can switch to old project manager in setup.



Terminology and conventions

Terminology

Global terms

additional data entity	An entity that defines properties other than the shape of a structural member, e.g. load, support, hinge, etc.
catalogue block; type structure	A predefined template structure; some of repeatedly used types of structure have been pre-created and can be quickly defined by a simple selection of the appropriate type in the integrated catalogue.
cut-out	A rectangular area created by a mouse when dragged over the screen; the area extends from the point where the drag move started to the point where the left button was released; the sides of the cut-out are always horizontal and vertical.
entity	Either a 1D member, load, support, hinge or any other part of a structure model the properties of which are defined and can be edited.
generator	A part of the program that automatically generates some kind of data, e.g. the finite element mesh, load from a given wind conditions, etc.
geometric entity	An entity that defines the geometry (or shape) of the structure. See member.
intersection line	A polygonal line drawn by a mouse on the screen; the line can intersect as many entities as desired.
member	Any structural member.
mesh	finite element mesh
solver	A part of the program that calculates the structure subject to the defined load using the selected type of calculation. The solver first assemblies the set of equations, then carries out the numerical solution of the problem.

Geometric entities

1D member	A straight or curved member defined by means of its midline and cross-section. The cross-section may be constant or varying along the length of the 1D member.
cross-link	A connection of two intersecting 1D members.
force load	Load in the form of force. It can be either point or continuous.
foundation block	A type of support that represents a pad foundation.
hinge	Connection of two members. It can be either rigid or of defined elasticity.
load	Any kind of load that the structure is subject to.
moment load	Load in the form of bending moment. It can be either point or continuous.
node	Generally a vertex of a member or a point where two or more members intersect.
predefined load	A load defined by means of the composition of e.g. floor. The user defines individual layers of the floor, their height and specific weight.
rigid arm	A 1D member of an infinitely large stiffness.
support	Point or line support of a structure. Several types of supports are available: standard, foundation pad, wall, etc.

Cross-sections

catalogue cross-section	A cross-section that can be defined by selecting from the library of cross-sections. The library is an integral part of SCIA Engineer.
general cross-section	A cross-section the shape of which is completely defined by the user.
reference point	The reference point is defined according to a cross-section type: for catalogue cross-sections it is located in the first point of the cross-section, for general cross-sections and cross-sections defined by a polygon it is identical with point [0,0].

Co-ordinate systems

Introduction to co-ordinate systems

As a user of SCIA Engineer you will come across a set of various co-ordinate systems. Some co-ordinate systems are essential for the work with the program itself, some others may significantly reduce the effort and time necessary to get the required result.

The co-ordinate systems may be divided into several groups according to what they relate to:

global co-ordinate system	the essential co-ordinate system, provides for positioning and orienting of a model and its unambiguous definition
user-defined co-ordinate systems UCS	facilitates the model definition, the user may define it's origin and direction
point definition co-ordinate systems; geometry definition co-ordinate systems	provides for the definition of geometry in the most straightforward way
entity co-ordinate systems local co-ordinate system	defines the orientation of individual entities in a model and provide for the unambiguous interpretation of physical quantities related to the entity

Global co-ordinate system

The global co-ordinate system used in the program is a three-dimensional right-handed Cartesian co-ordinate system.

The axes of the system are marked X, Y, and Z.

Note: It is highly recommended to locate the created model of a structure close to the origin of the global co-ordinate system (i.e. near the point whose global co-ordinates are 0, 0, 0) in order to prevent possible numerical inaccuracy due to numerical operations carried out with excessively great numbers.

It is further recommended to focus on this point especially after the model geometry has been imported from a third-party CAD program.

User-defined co-ordinate system

In order to simplify and speed up work with a model, the user can define its own co-ordinate system or systems and locate their origin, including possible inclination, anywhere in the global co-ordinate system.

The user-defined co-ordinate system is a three-dimensional right-handed Cartesian co-ordinate system.

The axes of the system are marked X, Y, and Z.

The user co-ordinate system may be set arbitrarily and the setting can be changed during work as many times as required. In addition, any number of user co-ordinate systems may be defined simultaneously but just one of them can be active at a time. The user can swap between the previously and also newly defined user co-ordinate systems whenever it seems to be convenient.

For information about setting and using of user co-ordinate systems see chapter Basic Working Tools > User co-ordinate system.

Entity co-ordinate systems

Introduction to entity co-ordinate systems

Each structural entity, that means each member, has got its own local co-ordinate system. This co-ordinate system is a three-dimensional right-handed Cartesian co-ordinate system.

The system provides for:

- unambiguous positioning of the member in space,
- unambiguous definition of load and boundary conditions,
- unambiguous interpretation of results.

This chapter also deals with a group of co-ordinate systems that do not refer to a structural entity in the full meaning of the word, but that is very closely related to it. This group consists of co-ordinate systems used with cross-sections.

Cross-section co-ordinate system

There are several co-ordinate systems used with cross-sections. All the sectional co-ordinate systems are two-dimensional right-handed Cartesian co-ordinate systems.

Principal (or main) axes

The principal axes correspond to the principal moments of inertia of a cross-section. They are marked u and v.

The u axis is called (according to the official Eurocode terminology) a major axis and the v axis is called a minor axis.

The principal axes are used to evaluate important sectional characteristics necessary for design and assessment to technical standards (code check), e.g. moments of inertia, radiuses of gyration, etc.

Centroidal axes

The two centroidal axes pass the centroid of a cross-section and the first moments (the static moments) of the cross-section around these axes are equal to zero.

The centroidal axes are marked y and z.

The centroidal axes are used to evaluate important sectional characteristics necessary for design and assessment to technical standards (code check), e.g. moments of inertia, radiuses of gyration, section modulus, etc.

For symmetrical cross-sections, the centroidal axes are identical to the principal axes.

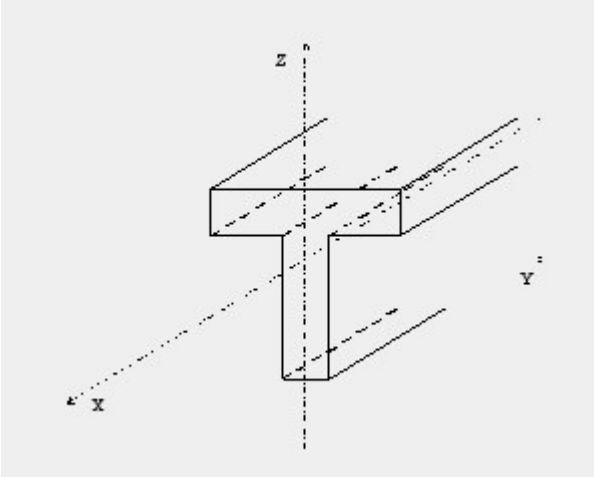
For example, for steel cross-sections the centroidal y axis is parallel to the flanges and the centroidal z axis is perpendicular to the flanges.

Geometric co-ordinate system

The geometric axes are used to define co-ordinates of cross-section vertices. The axes of the system are marked y and z.

Orientation of the cross-section co-ordinate system with reference to the beam local co-ordinate system

A cross-section is oriented so that the centroidal axis y is identical with beam local axis Y and the centroidal axis z is identical with beam local axis Z. If the 1D member is being rotated around its local X axis, also the sectional centroidal axes rotate.

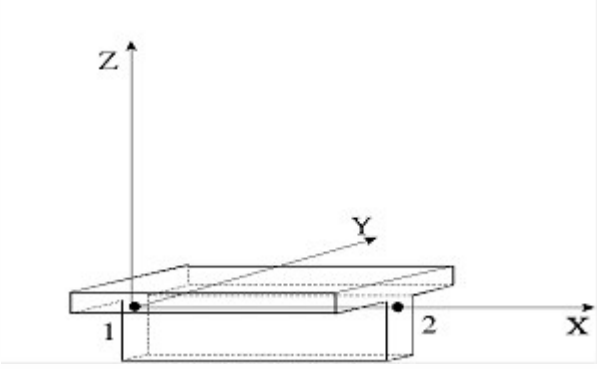


Beam co-ordinate system

The beam co-ordinate system is a three-dimensional right-handed Cartesian co-ordinate system with axes marked x, y, and z.

Each 1D member is defined by means of two end points – by a "starting point" and by an "end point". Each 1D member has got a unique local co-ordinate system, the origin of which is located in the starting point of a 1D member. The x-axis is always identical with the longitudinal beam axis and its direction is from the staring point towards the end point. By default, the y-axis is generally horizontal (unless the beam orientation prevents this) and the z-axis is generally vertical (again, unless the beam orientation in space prevents this configuration).

For example, a horizontal beam has both x and y axis horizontal and the z axis is vertical pointing upward. Similarly, a vertical column has the x axis vertical and both y and z axes are horizontal.



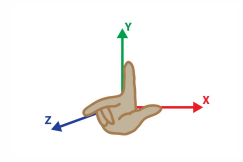
The local co-ordinate system can be rotated around its x-axis if required.

In addition to this local co-ordinate system, also a principal (or main) co-ordinate system can be referred to on a 1D member. The principal co-ordinate system of a 1D member is related to the principal co-ordinate system of the cross-section of a 1D member.

Right-handed coordinate system

It may help you to determine the direction of individual beam axis, if you remember what you (probably) learnt at school. Take your right hand, position your thumb, index finger and middle finger so that they are perpendicular to each other. Then the following applies (see also the image below):

- the thumb points in the direction of the x-axis,
- the index finger points in the direction of the y-axis,
- the middle finger points in the direction of the z-axis.



Geometric block co-ordinate system

Some of geometric blocks use a specific co-ordinate system. The system is used only throughout the phase of block definition. The concrete co-ordinate system, if applied, is always displayed in the dialogue for block definition.

Point definition co-ordinate systems

Introduction to point definition

Any geometric entity is defined by positions of its vertices. The vertices are defined as points inserted into required location. Any inserted point, regardless the entity type it relates to, can be defined in one of the following co-ordinate systems:

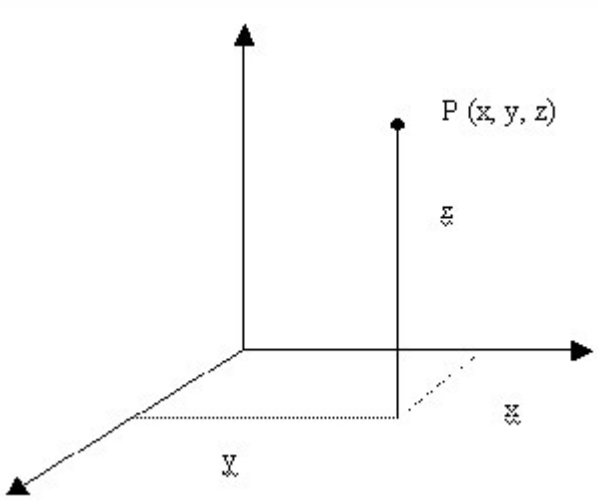
- Cartesian co-ordinate system
- Cylindrical co-ordinate system
- Spherical co-ordinate system

The choice of a particular system depends on several factors:

- how is the point position defined in the model drawings,
- what is the most efficient and most easiest way for the specific situation,
- which particular system is preferred by the user.

Cartesian co-ordinate system

A point in the Cartesian co-ordinate system is uniquely defined by three length co-ordinates x, y, and z. The individual co-ordinates represent the distance of the point from the origin of the co-ordinate system measured along individual axes x, y, and z respectively.



Cylindrical co-ordinate system

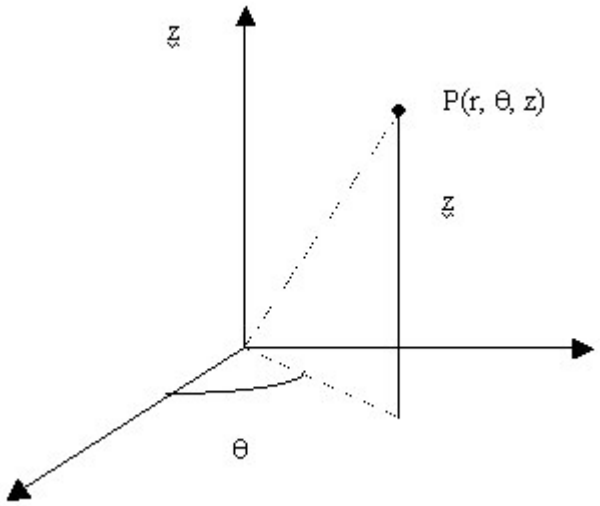
In the cylindrical co-ordinate system the co-ordinate of any point is given by three components r, theta, and z. The co-ordinates r and theta represent polar co-ordinates of a point in xy plane. And the z co-ordinate is a distance of the defined point from xy plane.

Thus the ordinate along x, y, and z axis are respectively:

$$x = r \times \cos(\text{theta}),$$

$$y = r \times \sin(\text{theta}),$$

$$z = z.$$



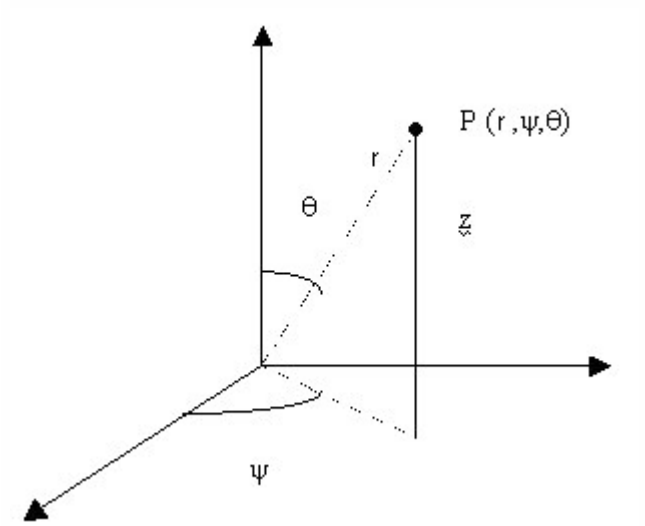
Spherical co-ordinate system

In the spherical co-ordinate system the co-ordinate of any point is given by three components r, Psi, theta. Thus the ordinates along x, y and z axis are:

$$x = r \times \sin(\text{theta}) \cos(\text{Psi}),$$

$$y = r \times \sin(\text{theta}) \sin(\text{Psi}),$$

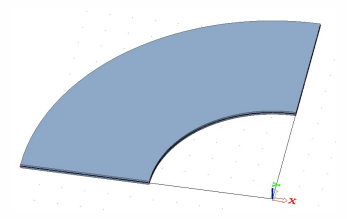
$$z = r \times \cos(\text{theta}).$$



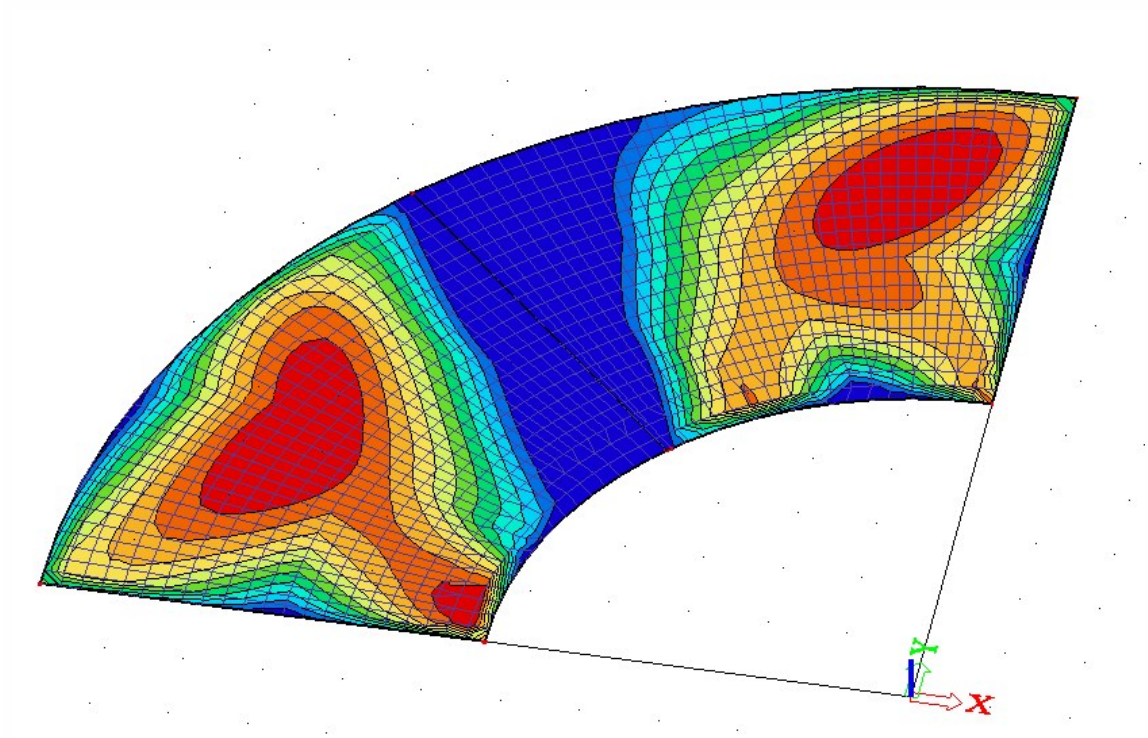
Polar UCS explained

This topic will explain using polar UCS in 2D results dialog (see "Parameters for display of results")

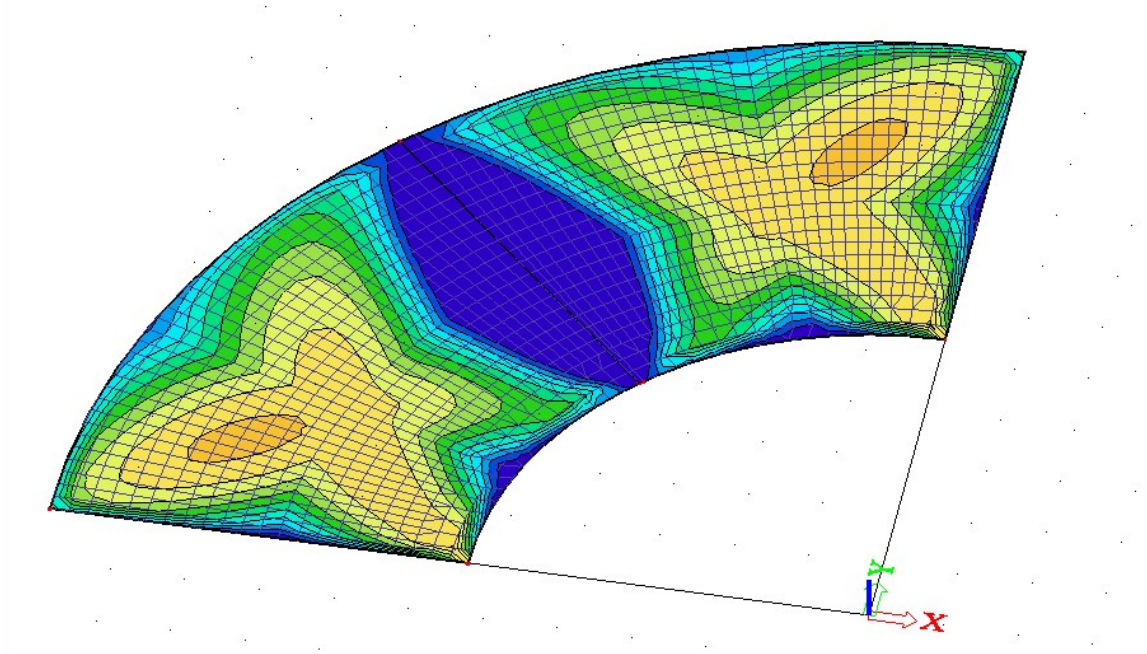
UCS polar option is beneficial for circular, semi-circular or curved slabs. When using this option, all internal forces will be recalculated in tangential and radial direction with centre according to user defined UCS. This values should be used when user will decide to design reinforcement in tangential and radial directions (outside of Scia Engineer). See images below and notice the different internal forces for Local and Polar UCS.



semi-circular slab



UCS local, internal forces mxD-



UCS polar, internal forces, mtD-

User should avoid to use sections with averaging values, in near proximity to centre of polar UCS where big peak values can appear (due the singularity in the centre of polar UCS) and return distorted values.

Conventions for applied physical quantities

Input quantities conventions

The following notation and conventions are user in the program and in the program documentation.

Axes

global	X	Y	Z			
local	x	y	z			

External forces

	Fx	Fy	Fz	Mx	My	Mz

Prescribed displacement and rotation

global	Ux	Uy	Uz	Fix	Fiy	Fiz
local	ux	uy	uz	fix	fiy	fiz

Both external forces and translations are considered as positive when acting in the direction of an appropriate axis. E.g. Force defined in global co-ordinate system and acting in the direction of the positive global X-axis is taken as positive. Force defined in global co-ordinate system and acting in the direction opposite to the direction of the positive global X-axis is taken as negative.

Output quantities conventions

The following notation is user in the program and in the program documentation.

Axes

global	X	Y	Z			
local	x	y	z			

Displacement and rotation

global	Ux	Uy	Uz	Fix	Fiy	Fiz
local	ux	uy	uz	fix	fiy	fiz

Reactions

	Rx	Ry	Rz	Mx	My	Mz

Internal forces

	N	Vy	Vz	Mx	My	Mz

Stress

	sig x	sig y	sig z			
	tau xy	tau yz	tau xz			

Note:Moments Mx and Mz in a beam are considered positive around the positive local x and z axis respectively. Moment My, however, is considered positive around the NEGATIVE local y axis.

Units

Introduction to units

SCIA Engineer supports various unit types.

SI units	International system of units (metric practice)
FPS units	foot-pound-second unit
Imperial, English units, US unit	FPS unit

Length units

Imperial length units

The imperial units for length are:

- inch (in),
- foot (ft).

The official values for conversion are:

quantity	multiply by	to obtain
inch	25.400	millimetre (mm)
foot	0.3048	metre (m)

Display style of length units

Display style of length units is defined by format, precision and unit symbol.

Format

The format can be:

- scientific (1.55E+01)
- engineering (15.50E+00) (the exponent is ..., -09, -06, -03, +00, +03, +06, +09, ...)
- decimal (15.50)
- fractional (15 1/2)

Precision

The precision for scientific and decimal format is defined as follows. Sample value is 3.1415926

Decimal length in Units Setup	Precision	Result
0	0	3
1	0.1	3.1
2	0.01	3.14
3	0.001	3.142
4	0.0001	3.1416
etc.	etc.	etc.

The precision for fractional format is defined as follows.

Fractional precision in Units Setup	Precision
0	1
1	″
2	L'
3	1/8
4	1/16
etc.	etc.

Unit symbol

unit	symbol
millimetre	mm
centimetre	cm
decimetre	dm
metre	m
inch (1st option)	in
inch (2nd option)	"
foot (1st option)	ft
foot (2nd option)	‘
foot-inch (1st option)	ft in
foot-inch (2nd option)	‘ "

Example

The value is 78.24 cm.

Format	Precision	Unit symbol	Result
scientific	0.001	centimetre (cm)	7.824E+01 cm
scientific	0.01	millimetre (mm)	7.82E+02 mm
engineering	0.001	centimetre (cm)	78.240E+00 cm
engineering	0.01	millimetre (mm)	782.40E+00 mm
decimal	0.01	centimetre (cm)	78.24 cm
decimal	0.001	inches (in)	30.803 in
decimal	0.001	inches (")	30.803 "

decimal	0.001	feet (ft)	2.567 ft
decimal	0.001	feet (')	2.567 '
decimal	0.001	feet-inches (ft in)	2 ft 6.803 in
decimal	0.001	feet-inches (' ")	2' 6.803"
fractional	1/16	feet(')	2-9/16'
fractional	1/16	inches (")	30-13/16"
fractional	1/16	inches (in)	30-13/16 in
fractional	1/16	feet-inches (' ")	2' 6-13/16"

Input of length units

For metric units (mm, cm, dm, m), the scientific and decimal formats are supported. Once the value is input, the value is transformed into the defined format, precision and unit.

For the imperial units (in and ft), the scientific, decimal and fractional formats are supported. The use of symbols " and ' is supported. The fractional input (-1/2, -3/4, …) is supported. When entering fractions, the fractions must be separated from the rest by a hyphen. Once the value is input, the value is transformed into the defined format (scientific, decimal, fractional), precision and unit symbol.

It is always possible to enter a number in greater precision than defined by settings. The precise value is stored internally and the displayed value reflects the Units setup.

Examples for imperial units

Input string	Display setting	Result
3.5	decimal, inches (")	3.5"
3-1/2	decimal, inches (")	3.5"
5'	decimal, inches (")	60"
5.3' 6"	decimal, inches (")	69.6"
5.3' 6.6"	decimal, inches (")	70.20"
5.3' 6.6	decimal, inches (")	70.20"
3.5	decimal, feet (')	3.5'
3-1/2	decimal, feet (')	3.5'
5'	decimal, feet (')	5.0'
5.3' 6"	decimal, feet (')	5.80'
5.3' 6.6"	decimal, feet (')	5.85'
5.3' 6.6	decimal, feet (')	5.85'
3.5	fractional, feet (')-inches (")	3' 6"
3-1/2	fractional, feet (')-inches (")	3' 6"
5'	fractional, feet (')-inches (")	5' 0"
5.3' 6"	fractional, feet (')-inches (")	5' 9-5/8"
5.3' 6.6"	fractional, feet (')-inches (")	5' 10-1/4"
5.3' 6.6	fractional, feet (')-inches (")	5' 10-1/4"

Angle units

The display of the angle unit is defined by the format and the precision.

Format

- decimal degrees (45.000)
- degrees/minutes/seconds (45d0'0")
- grads (50.000g)
- radians (0.7854r)

Precision

The precision of angle units is analogous to decimal format of Length units.

Similarly to Length units, the settings for display style of angle units can be made in Units setup.

Units Setup

In SCIA Engineer the user uses and comes into contact with a good number of various physical quantities. In order to allow the user to adjust preferable units and display style of these quantities, the program offers a means for user's adjustment.

The adjustment can be done in Units Setup dialogue.

Unit "parameters"

Unit	It sets the unit in which the value of appropriate quantity is displayed.
Decimal length	It defines number of decimal digits to be displayed when the corresponding quantity is displayed.
Output format	It specifies the format of displayed value for individual the quantity. See below.

Output format

decimal	standard representation of a number	78.24 cm 782.4 mm
scientific	representation of a number by means of a base and an exponent	7.824E+01 cm 7.82E+02 mm
engineering	representation of a number by means of a base and an exponent, where the exponent is always a multiple of three	78.240E+00 cm 782.40E+00 mm
fractional		
fractional	representation of a number by means of a fraction	3/16 in
deg/min/sec	representation of a number used for angles	
ft in	representation of a number used for imperial units	2 ft 6.803 in

The procedure for adjustment of units

1. Open dialogue Units Setup:
   1. either using menu function Setup > Units,
   2. or using button Units () on toolbar Project.
2. Make the required settings.
3. Confirm with [OK].

Note: For more information about units see chapter Terminology and conventions > Units.

Layout and operation

Title bar

The title bar is the heading of the application window. It consists of three parts:

- the program icon (on the left side of the bar)
- text information about the application name
- text information about the name of the opened and active project and the number of the active project window
- three control buttons for (i) minimising the application window, (ii) making the application window full-screen, and (iii) closing the application on the right side of the bar.

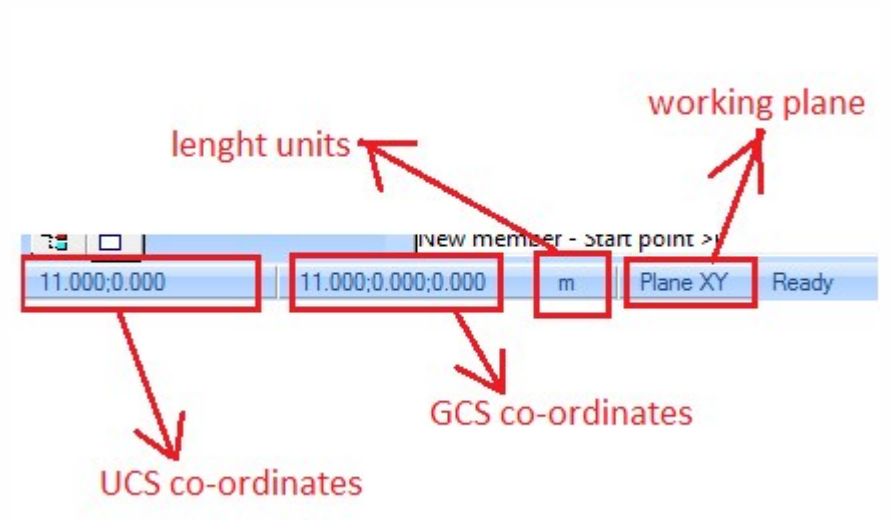
Note: The first and the last feature of the title bar is the common feature of any Microsoft Windows application.

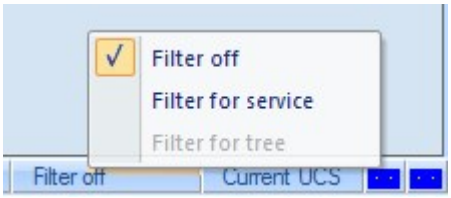
Example of a title bar

Status bar

The status bar is a bar placed at the bottom of the application window. It is used to display information about the program and/or about the functions under process and it contains a few control elements. By default the status bar shows the following information:

co-ordinates of the mouse cursor position in UCS	When a function requiring the definition of a point (e.g. insertion of a 1D member) is running, the status bar shows the cursor position in the current user co-ordinate system.
co-ordinates of the mouse cursor position in GCS	If selected in the application settings the status bar shows the co-ordinates also in the global co-ordinate system.
project length units	The bar displays the current length unit (e.g. meter, inch, etc.). The unit can be easily changed by simple clicking on the unit box on the status bar.
orientation of working plane	The working plane box of the status bar shows the current orientation of the working plane. The orientation can be changed by clicking on the working plane box or by pressing F11.
[SNAP mode]	This button opens SNAP mode dialog. (F6)
[Filter for selections]	Switches selection filter based on main tree or service.
[Current UCS]	This button displays the current UCS for the active window. If pressed, it opens the UCS manager.
[Active code]	A small icon shows the flag of the country whose code is currently set as active.





The status bar also displays a brief help text for program elements like a toolbar button or a menu function if the mouse cursor is just being placed on such an element.

Example of a status bar

Note: The status bar in the picture does not show the global co-ordinates of the mouse position. This option can be switched on or off in the Application settings.

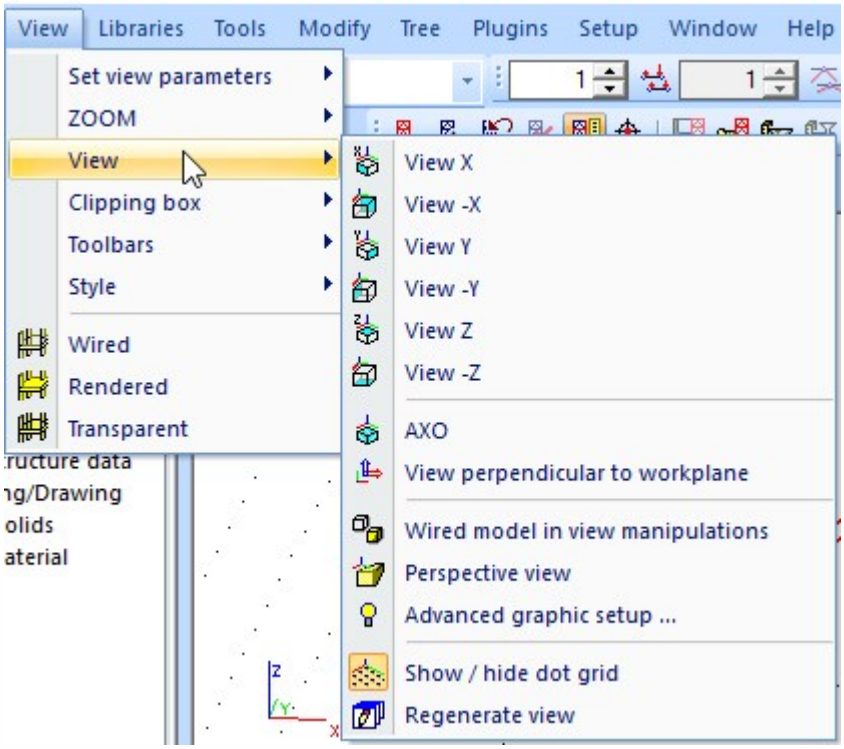
Menu bar

The menu bar is, by default, located just under the Title bar of the application window. It can be, however, moved into another position within the application window. It can be either docked to the left or upper edge of the application window, or it can be let floating anywhere within the work area.

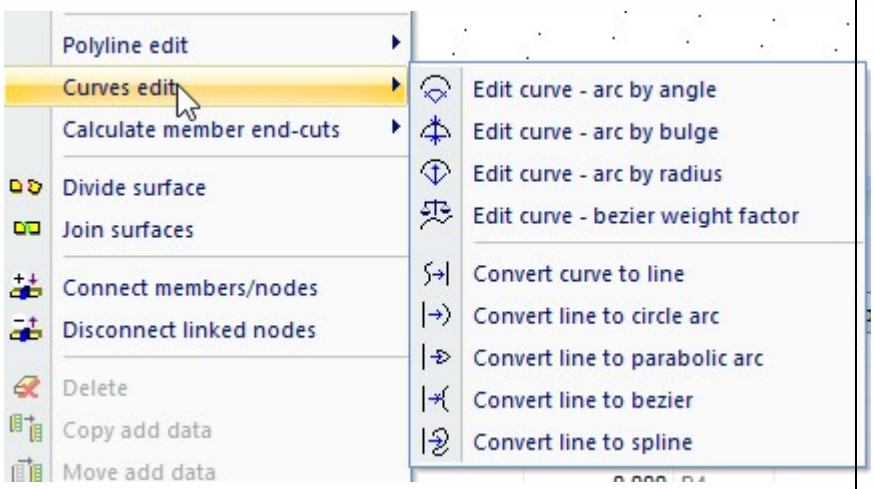
Majority of SCIA Engineer functions is accessible via this menu. There are some functions that can be accessed only from the tree menu of from toolbars.

Example of menus

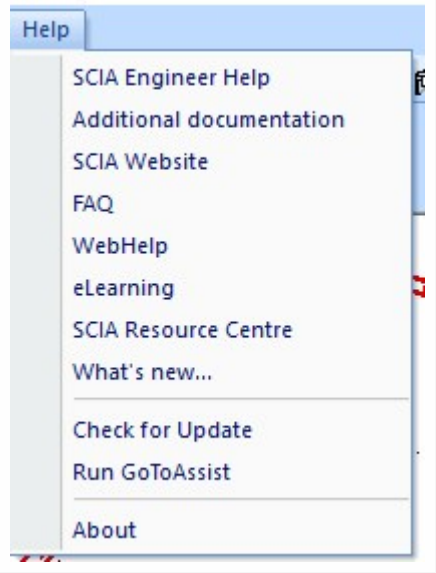
menu View > View



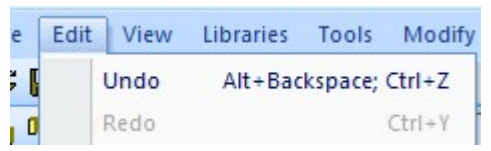
menu Modify > Curve-s edit



Help menu - Local help, WebHelp, eLearning or FAQs are available.



There are shortcuts displayed in menus:



Tree menu window

The tree window is similar in function to the menu bar but it is more readable and user-friendly.

The individual items of the tree may be:

service	It opens another tree menu in the same window. E.g. service Structure, Loads, etc.
function	It opens a specific function, e.g. Point load in node, Cross-link, etc.
branch	It opens a branch of the tree and shows individual functions in it. E.g. branch Point load offers functions Point load in node and Point load on 1D member.

How to operate the tree menu

The procedure to operate the tree menu is very straightforward and closely resembles the operating rules for standard Microsoft Windows tree control.

Opening branches of the tree

The tree consists of a main branch and possible sub-branches. If an item has a sub-branch, it is indicated with a plus sign (+) in front of the item name. The sub-branch can be opened (listed on the screen) by means of either (i) a left mouse button single click on the plus sign or (ii) a left mouse button double-click on the item name. If the same action is made with already opened a branch, the branch is closed.

Activating tree branch items

In order to activate an item of a branch (either a main branch item that opens a service or sub-branch item that opens a particular function), simply double-click on the item name with the left mouse button. Depending on the item type either a corresponding function is activated or a particular service tree menu is displayed.

If the branch item represents a particular function, it can also be activated using a button at the bottom part of the tree menu window.

Closing a service

In order to close the whole service you can do the following:

- press the [Close] button.

Closing a function

In order to close the function, you can use one of the following ways:

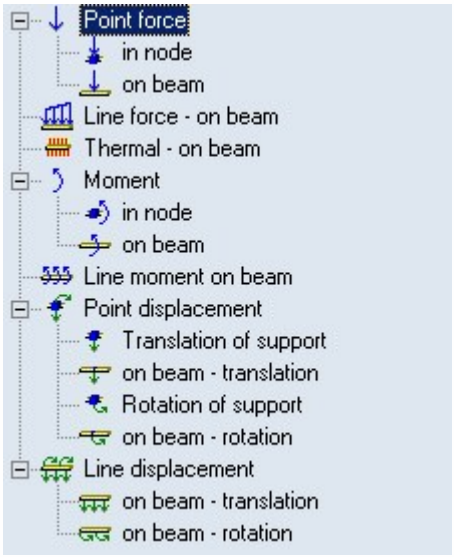
- press the [Esc] key once.
- click the [Arrow] ( ) button on the toolbar at the top of the command line.
- invoke the window pop-up menu and select function End.

Terminating a function

In order to abandon the activated function without accepting the already made changes, press [Ctrl] + Break keys simultaneously.

It is also possible to invoke the window pop-up menu and select function Cancel.

Example of a tree menu



One click in main tree

To enhance speed of work in main tree, there is a possibility to turn ON one click opening in main tree. Setting can be found in Options - Setup - Other.

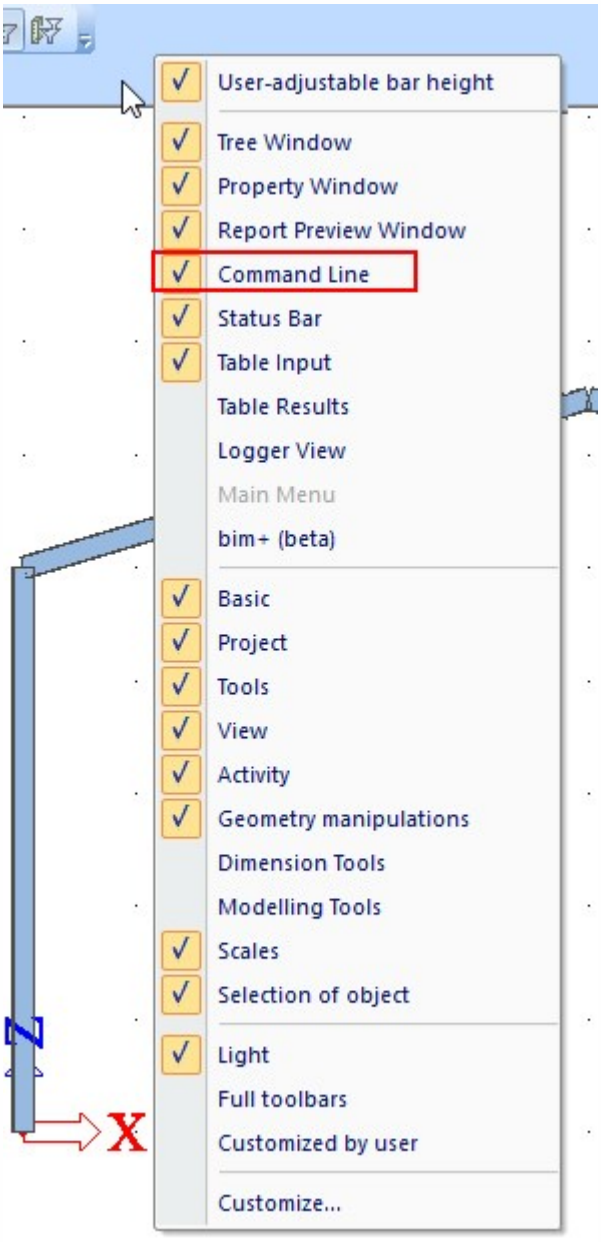
Command line

The command line provides for the following:

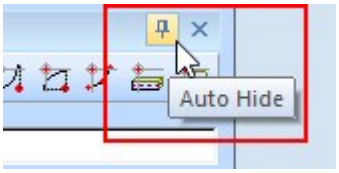
- some functions can be activated via typing the appropriate command,
- List of commands (after typing "?" in command line).
- if any function has been already called (regardless whether via the command line, menu, tree menu, or toolbar button), it displays guiding instructions on the command line,
- if any function requires a numerical input (e.g. co-ordinates of an inserted point), the corresponding value or values may be typed on the command line.

Especially the second feature is very useful particularly for beginning users as they are clearly guided through the function they want to use and can simply follow the presented step-by-step instructions.

You can access command line via right click menu in toolbar area.



Command line can be docked or auto-hidden as other frames in main window.



Syntax of commands

The syntax of a command to the command line is:

command parameter1 [parameter2] [parameter3] [etc.]

Example

SEL BEAM1

This command adds the 1D member named BEAM1 into the current selection.

Syntax for input of co-ordinates

The important thing to be aware of is that if a co-ordinate is typed by means of one or two numbers only, it is considered to be defined in the active working plane of the current user co-ordinate system.

If the point is defined by means of three values, it is considered to be defined in the current user co-ordinate system. In this case, the orientation of the working plane is not taken into account at all.

General syntax for the definition of a point

[prefix] [number] [separator] [number] [separator] [number]

Prefix

none	absolute co-ordinate in UCS
@	relative co-ordinate related to the last input point, defined in UCS
*	co-ordinate in GCS
@*	relative co-ordinate related to the last input point, defined in GCS

Number

[space] [sign] [nnn] [.] [nnn] [exp] [sign] [nnn]

[space]	if any, ignored
[sign]	sign plus or minus ('+' or '-')
[nnn]	row of digits 0,1, ..., 9
[,]	decimal comma or point
[exp]	exponent – sign 'e' or 'E'

Separator

;	length value follows
<	angle value follows

Syntax for the definition of a point in Cartesian co-ordinates

[*,@][X],[Y],[Z]

Examples

12.4;45.8;12.4	absolute point co-ordinate in UCS 12.4, 45.8, 12.4
123.4;345.8	absolute point co-ordinate in the current working plane of the UCS 123.4, 345.8
@123;23;5	relative co-ordinate related to the last inserted point in UCS 123, 23, 5
@123;23	relative co-ordinate related to the last inserted point in the current working plane of the UCS 123, 23
@123	relative co-ordinate related to the last inserted point in the current working plane of the UCS 123, 0
*123;23;5	global co-ordinate in GCS 123, 23, 5
*	the origin of GCS 0, 0, 0

Syntax for the definition of a point in polar co-ordinates

[*,@][length]<[angle]

Examples

123<90	absolute co-ordinate of point in UCS 0, 123, 0
123<180	absolute co-ordinate of point in UCS 0, -123, 0

Syntax for the definition of a point in spherical co-ordinates

[*,@][length]<[angle]<[angle]

Example

123<90<90	absolute co-ordinate of point in UCS 0, 0, 123

Syntax for the definition of a point in cylindrical co-ordinates

[*,@][length]<[angle],[length]

Example

123<90;200	absolute co-ordinate of point in UCS 0, 123, 20

Syntax for imperial units

If the imperial (fractional) units are used, the number must be terminated by a semicolon (;). Without this mark it is not possible to determinate where the number ends.

Examples of valid input in "feet":

8; = 0yd 8ft

12ft 8; = 0yd 12ft 8in

12ft 8in; = 0yd 12ft 8in

3yd 5; = 3yd 5ft 0in

3yd 6ft 8; = 3yd 6ft 8in

3yd 6ft 8in; = 3yd 6ft 8in

format without spaces also may be used :

8; = 0yd 8ft

12ft8; = 0yd 12ft 8in

12ft8in; = 0yd 12ft 8in

3yd5; = 3yd 5ft 0in

3yd6ft8; = 3yd 6ft 8in

3yd6ft8in; = 3yd 6ft 8in

Commands

Abbreviations

Command: By the “command” is meant the string used to run some actions from the command line. Commands usually contain dots, e.g. “Structure.1d.Beam”

Shortcut: By the “shortcut” is meant string used wgicg is shorter substitution to full lenght command. Some commands does not have corresponding shortcuts. Type "?" to command line to see full list of commands.

Using of commands

Installation

Commands and shortcuts are available immediately after the installation of the SCIA Engineer. It is not necessary to install anything special neither to do any changes in the settings.

Each command may be started again without its selection. Use key "Enter" when the command is finished, and the command is started again automatically.

For quick non dialog access you can use "Shift+ Enter " in command line. Useful for command member, when you can immediately input 1d member, without dialog specifying properties.

Protection

Commands and shortcuts belong to general functionality. They are not protected by any module and are available for all users. However user is still able to run actions supported by his licence only. E.g. user which does not have nonlinearity in his licence cannot start inserting of “Beam local nonlinearity”

Customization

Commands

Commands are defined by SCIA and cannot be changed by the user.

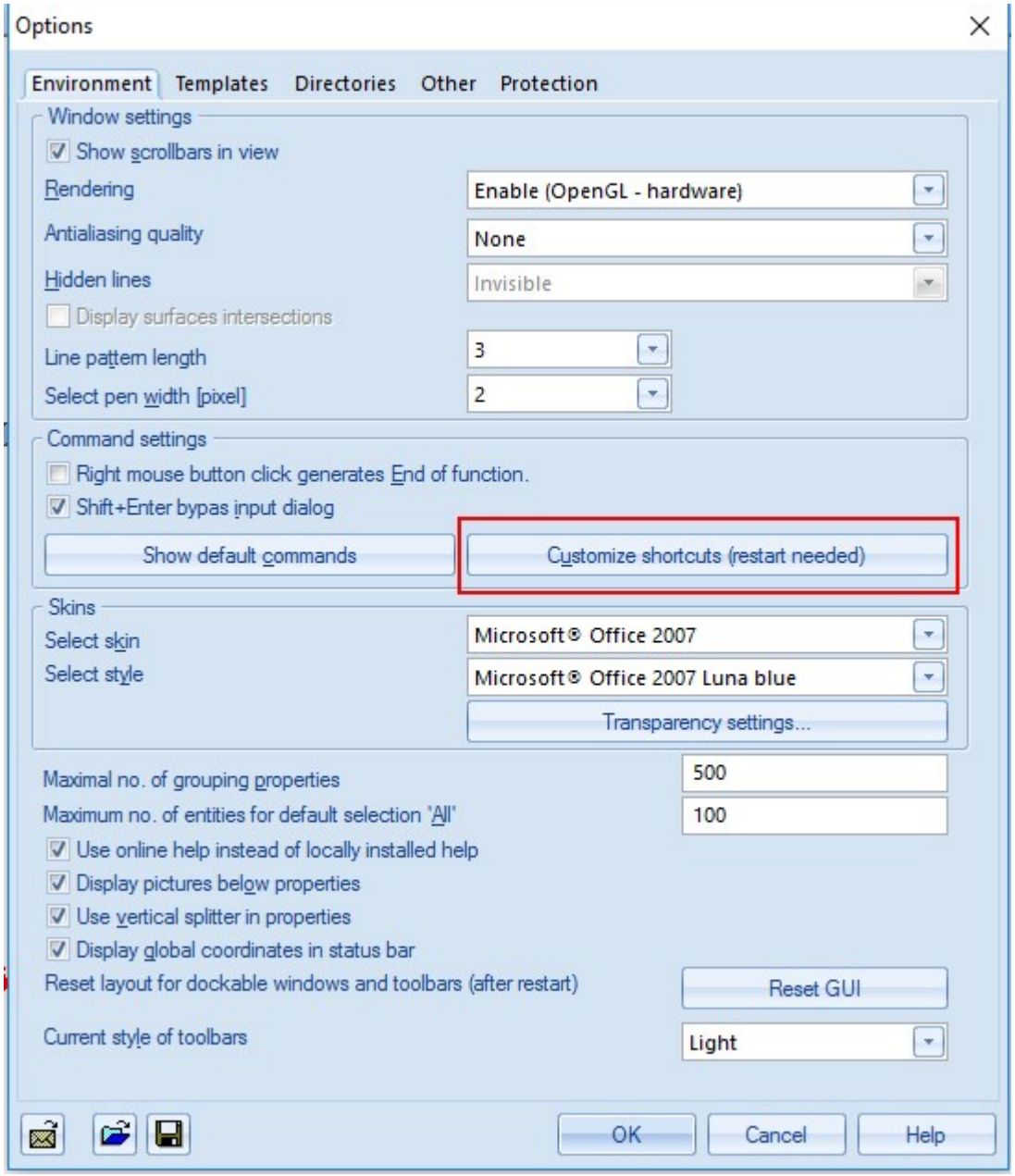
Shortcuts

Shortcuts are also defined by SCIA, but they can be changed (customized) by users.

Steps needed for customization of shortcuts:

1. Go to Setup / Options / Environment

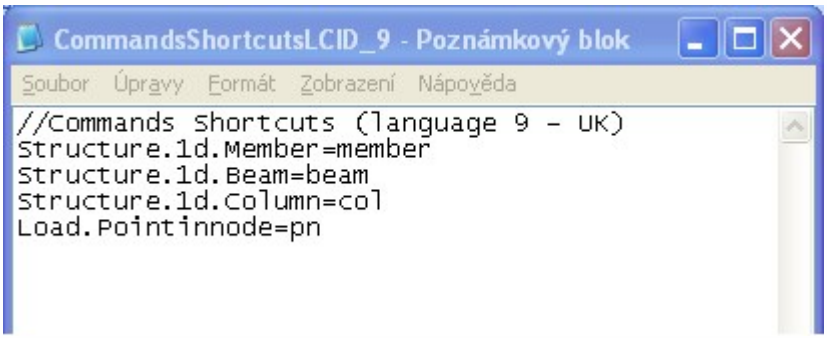
2. Press [Customize shortcuts] button



3. File with customized shortcuts is opened. It is possible to add there new rows with customized shortcuts.Show default commands]

Format of rows in the file is Full.Dot.Command=shortcut

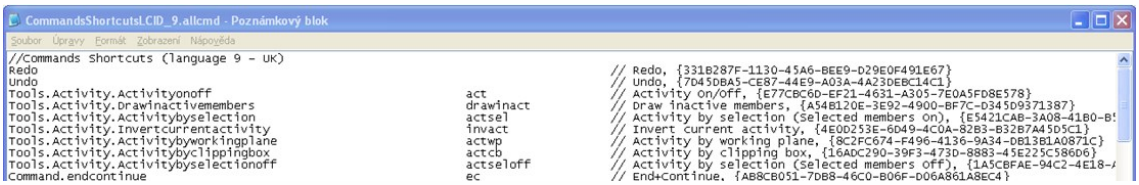
(e.g. Structure.1d.Member = member)



4. **Restart SCIA Engineer**

Original shortcuts are available together with the new ones after the customization. In case of duplicity the customer's shortcuts has higher priority than the original one.

It is possible to open file with default commands to search and copy full names of commands using [Show default commands]



Localization

There can be available different commands and shortcuts for different languages. It depends on level of localization for each country.

In case there are different commands for some country they are used together with English commands. Localized commands have higher priority than the English ones..

The number "9" in the names of files (mentioned in chapter related to customization) indicates the language. In case you are using different language, the number in files names will be different.

It is possible to use different *.usercmd file for each language. In such case the *.usercmd related to current language has higher priority than the English (9) one .

The complete order of searching of the command is following:

- Customized shortcuts for the current language
- Customized shortcuts for English language
- SCIA defined shortcuts for current language
- SCIA defined shortcuts for English language
- Full command names for current language
- Full command names for English version

Basically, if you want to use one user defined shortcuts across versions and languages, use one defined for English (number 9, you can manually overwrite name of the .usercmd file in user folder)

Functionality coverage by commands

General

Commands are available only in open project. If there is not any project opened then it is not possible to use command line for starting actions.

Actions available via commands

Commands and shortcuts cover major part of available actions within SCIA Engineer. Following types of actions are covered by commands:

- actions started from the tree
  - inserting of entities (structural members, supports, loads, libraries ...)
  - running of calculation
  - displaying of results and checks
- actions started from toolbars
  - modification of entities (copy, move, rotate, ...)
  - changing of view (view point, zoom, view parameters, ...)
  - actions duplicate with action started from the tree (calculation, libraries, ...)
- starting of various managers and tools (galleries, document, …)

Actions not available via commands

It is not possible to modify running action using command line. E.g. when the inserting of beam is running it is not possible to change e.g. snapping or geometry of the beam using command line.

Commands are not available in Document, GCS editor, ODA.

Also selections cannot be changed using commands or shortcuts.

List of commands

The following table list the available commands including system-defined abbreviations (where available).

You can access this list by typing "?" into the command line.

Command	Abbreviation	Short explanation
Redo		Redo
Undo		Undo
Tools.Activity.Activityonoff	=act	Activity On/Off
Tools.Activity.Drawinactivemembers	=drawinact	Draw inactive members
Tools.Activity.Activitybyselection	=actsel	Activity by selection (Selected members On)
Tools.Activity.Invertcurrentactivity	=invact	Invert current activity
Tools.Activity.Activitybyworkingplane	=actwp	Activity by working plane
Tools.Activity.Activitybyclippingbox	=actcb	Activity by clipping box

Command	Abbreviation	Short explanation
Tools.Activity.Activitybyselectionoff	=actseloff	Activity by selection (Selected members Off)
Command.endcontinue	=ec	End+Continue
View.Viewx	=x	View X
View.Viewy	=y	View Y
View.Viewz	=z	View Z
View.Axo	=axo	AXO
View.Zoomin	=zoomin	Zoom +
View.Zoomout	=zoomout	Zoom -
View.Zoomall	=zoomall	Zoom all
Tools.Ucs.Ucsby3points	=ucs3p	UCS by 3 points
Tools.Ucs.Xyworkplane	=wpxy	XY workplane
Tools.Ucs.Yzworkplane	=wpyz	YZ workplane
Tools.Ucs.Xzworkplane	=wpxz	XZ workplane
View.Viewparam.Showall	=vpa	Set view parameters
Tools.Ucs.Accordingtoentitylcs		According to entity LCS
Tools.Ucs.Gcs	=gcs	GCS
Tools.Ucs.Gcsparallel	=gcspar	GCS parallel
Tools.Ucs.Verticalorthogonaltox	=wpvx	Vertical orthogonal to X
Tools.Ucs.Verticalorthogonaltoy	=wpvy	Vertical orthogonal to Y
Tools.Ucs.Verticaldefinedbyline	=wpvl	Vertical defined by line
Tools.Ucs.Move	=wpm	Move
Tools.Ucs.Rotate	=wpr	Rotate
Tools.Ucs.Perpendiculartoucssx		Perpendicular to UCS's X
Tools.Ucs.Perpendiculartoucssy		Perpendicular to UCS's Y
Tools.Ucs.Accordingtoview	=wpbyview	According to view
Tools.Ucs.Previous	=wppre	Previous
Command.End	=END	End
View.Cbox.new		Clipping box - new
View.Cbox.Dotgridtracksettings		Dot grid and tracking setting
View.Cbox.Onoff		Clipping box (on/off)
Modify.Move	=move	Move
Modify.Rotate	=rotate	Rotate
Modify.Scale	=scale	Scale
Modify.Mirror	=mirror	Mirror
Modify.Copy	=copy	Copy
Modify.Trim	=trim	Trim
Modify.Extend	=extend	Extend
Modify.Breakinpoints	=bk	Break in defined points
Modify.Connectmembers	=connect	Connect members/nodes
Modify.Polylines.Insertnode		Edit polyline - insert node
Modify.Polylines.Insertnodeatintersections		Edit polyline - insert node at intersections

Command	Abbreviation	Short explanation
Modify.Polylines.Deletenode		Edit polyline - delete node
Modify.Polylines.Breakintosinglecurves		Edit polyline - break into single curves
Modify.Polylines.Fillet		Edit polyline - fillet
Modify.Multicopy	=multicopy	Multicopy
Modify.Curves.Arcbyradius		Edit curve - arc by radius
Modify.Curves.Arcbybulge		Edit curve - arc by bulge
Modify.Curves.Arcbyangle		Edit curve - arc by angle
Modify.Curves.Bezierweightfactor		Edit curve - bezier weight factor
Modify.Curves.Curvetoline		Convert curve to line
View.Zoomallselection	=zoomsel	Zoom all - selection
Modify.Delete	=del	Delete
View.Cbox.Alphanumericaledit		Alphanumerical edit
View.Cbox.Aroundselectedentity		Around selected entity
View.Cbox.Move		Move
View.Cbox.Attachtoworkplane		Attach to workplane
Modify.Breakinintersections	=bix	Break in intersections
View.Cbox.Fromucs		From UCS
Modify.Disconnectlinkednodes	=disconnect	Disconnect linked nodes
Modify.Polylines.Joincurvesintopolyline		Join curves into polyline
Modify.Join	=join	Join
View.Cbox.Aroundallentities		Around all entities
Modify.Stretch	=stretch	Stretch
Modify.Copyadddata	=copyadd	Copy add data
Modify.Moveadddata	=moveadd	Move add data
View.Perspectiveview	=per	Perspective view
Modify.Curves.Linetocirclearc		Convert line to circle arc
Modify.Reversecurve	=reverse	Reverse orientation
Modify.Curves.Loinetoparabolicarc		Convert line to parabolic arc
Modify.Curves.Linetobezier		Convert line to bezier
Modify.Curves.Linetospline		Convert line to spline
View.Generatestructuralmodel	=gensm	Generate structural model
View.Redraw	=re	Redraw
View.Regenerateview	=reg	Regenerate view
Command.Cancel	=CANCEL	Cancel
View.Viewparam.Showselected	=vps	Set view parameters for selected
Modify.Copyadddata	=copyadd	Copy add data
Modify.Moveadddata	=moveadd	Move add data
BIM.Align		Align
Tools.Paperspacegallery	=papg	Paperspace gallery
Tools.Printpicture	=prip	Print Picture
Tools.Printdata	=prid	Print Data

Command	Abbreviation	Short explanation
Tools.Report	=er	Engineering report
View.Viewparam.Showhideloads	=shlo	Show / hide loads
View.Viewparam.Showhidesufraces	=shsur	Show / hide surfaces
View.Viewparam.Rendergeometry	=rengeo	Render geometry
View.Viewparam.Rendermodeldata	=rendata	Render model data (loads, the hinges,...)
View.Viewparam.Showhidesupports	=shsup	Show / hide supports
View.Viewparam.Showhidemasses	=shmas	Show / hide masses
View.Viewparam.Showhidemodels	=shmod	Show / hide models
View.Viewparam.Showhideadditionalmodeldata	=shamod	Show / hide additional model data
View.Viewparam.Showhidefemesh	=shfem	Show / hide FE mesh
View.Viewparam.Showhidedotgrid	=shdg	Show / hide dot grid
View.Viewparam.Showhideresult	=shres	Show / hide result
View.Viewparam.Showhidelabelofnodes	=shlno	Show / hide label of nodes
View.Viewparam.Showhidelabelofmembers	=shlme	Show / hide label of members
View.Viewparam.Showhidelabelofloads	=shllo	Show / hide label of loads
View.Viewparam.Analysismodelvolumes	=amvol	Analysis model - volumes
View.Viewparam.Analysismodelaxes	=ama	Analysis model - axes
View.Viewparam.Structuralmodel	=sm	Structural model
Tools.Activity.Activitybylayers	=actlay	Activity by layers
View.Viewparam.Showhidereinforcement	=shrei	Show / hide reinforcement
View.Viewparam.Showhidelabelreinforcement	=shlrei	Show / hide label reinforcement
View.Viewparam.Showhideothermodeldata	=shmodd	Show / hide other model data
View.Viewparam.Wired	=w	Wired
View.Viewparam.Transparent	=t	Transparent
View.Viewparam.Rendered	=r	Rendered
View.Ortho	=ortho	On/off Ortho
View.Tracking	=trace	On/off Tracking
Tools.Activity.Activitybystorey	=actsto	Activity by storey
Tools.Activity.Activitybystoreyup	=actstou	Move activity by storey up
Tools.Activity.Activitybystoreydown	=actstod	Move activity by storey down
Drawings.Drawingmanager	=drwman	Drawing manager
Results		Results
Results.1d.Internalforcesonbeam	=rifb	Internal forces on beam
Results.Reaction.Reactions	=react	Reactions
Steel.Check	=stec	Check
Results.1d.Deformationsonbeam	=rdb	Deformations on beam
Results.1d.N	=rn	N
Results.1d.Mx	=rmx	Mx
Results.1d.Vy	=rvy	Vy
Results.1d.Vz	=rvz	Vz
Results.1d.My	=rmy	My

Command	Abbreviation	Short explanation
Results.1d.Mz	=rmz	Mz
Results.1d.Ux	=rux	ux
Results.1d.Uy	=ruy	uy
Results.1d.Uz	=ruz	uz
Results.Reaction.Rx	=reax	Rx
Results.Reaction.Ry	=reay	Ry
Results.Reaction.Rz	=reaz	Rz
Results.Reaction.Mx	=remx	Mx
Results.Reaction.My	=remy	My
Results.Reaction.Mz	=remz	Mz
Results.Deformedstructure	=rd	Deformed structure
Results.Displacementofnodes	=rdn	Displacement of nodes
Results.Reaction.Resultantofreactions	=resreact	Resultant of reactions
Results.1d.Relativedeformation	=rrd	Relative deformation
Results.Reaction.Intensity	=ri	Intensity
Structure.Billofmaterial	=bom	Bill of material
Results.Reaction.Reactionline	=rrl	Reaction line
Steel.Checkfireresistance	=sfirecheck	Check - fire resistance
Results.Reaction.Foundationtable	=ftable	Foundation table
Results.1d.Connectionforces	=rcf	Connection Forces
Results.1d.Connectioninput	=rci	Connection input
Results.1d.Memberstress	=rms	Member Stress
Steel.Steelslenderness	=sslend	Steel slenderness
Results.Calculationprotocol	=Calcprot	Calculation protocol
Results.Eigenfrequencies	=rei	Eigen frequencies
Results.Criticalloadcoefficients	=rcc	Critical load coefficients
Concrete.1D.Concreteslenderness	=cslend	Concrete slenderness
Results.2d.Displacementofnodes	=rpldn	Displacement of nodes
Results.2d.Member2dinternalforces	=rplif	Member 2D - Internal Forces
Results.2d.Member2dstresses	=rpls	Member 2D- Stresses
Results.2d.Member2dcontactstresses	=rplcs	Member 2D- Contact stresses
Concrete.2D.Design.Memberdesignuls	=cdesuls2D	Member design ULS
Concrete.2D.Design.Memberdesignulssls	=cdessls2D	Member design ULS+SLS
Results.1d.Relativedeformationnocheck	=rrdnocheck	Relative deformation
Concrete.2D.Design.Deformations	=cdef2D	Deformations
Results.Subsoilcparameters	=rscp	Subsoil - C parameters
Results.Subsoilotherdata	=rsod	Subsoil - Other data
Results.Reaction.Nodalspacesupportresultant	=nssr	Nodal space support resultant
Results.1d.Shearstress	=rss	Shear stress
Results.2d1dupgrade	=r2d1dup	2D/1D upgrade
Results.2d.Sigeplus	=rsigepl	SigE+

Command	Abbreviation	Short explanation
Results.2d.Sigeminus	=rsigemin	SigE-
Results.2d.Mx	=rplmx	mx
Results.2d.My	=rplmy	my
Results.Deformed1d2dstructure	=rd1d2d	Deformed Structure
Concrete.2D.Design.Member2ddeformations		Member 2D deformations
Concrete.2D.Design.Voidedslabshearresistance	=cvoidsres	Voided slab shear resistance
Concrete.Billofprestressreinforcement	=cpbom	Bill of prestress reinforcement
Results.Accelerationofnodes	=rdn	Acceleration of nodes
Results.Displacement	=disp	3D displacement
Results.Stress	=sss	3D stress
Steel		Steel
Steel.Checkdata.Bucklingdata	=sbuckl	Member buckling data
Steel.Checkdata.Ltbrestraints	=ltbres	LTB Restraints
Steel.Checkdata.Webstiffener		Web stiffener
Steel.Checkdata.Diaphragms		Diaphragms
Steel.Checkdata.Fireresistance	=sfire	Fire resistance
Steel.Checkdata.Memberdata		Steel member data
Steel.Setup		Setup
Steel.SetupBS		Setup
Steel.Connections.Check		Check
Steel.Characteristicbeamfactor		Characteristic beam factor
Structure.Modeldata.Sectiononbeam	=bsection	Section on beam
Steel.Checkdata.Webcripplingdata		Web crippling data
Steel.Checkdata.Overlapdata		Overlap data
Steel.SteelsetupEC	=Steelsetup	Steel Setup
Steel.Connections.Setup		Connections Setup
Steel.Checkdata.Additionallateralrestraints		Additional Lateral Restraints
Steel.Checkdata.Localtransverseforcesdata		Local Transverse Forces data
Steel.Slschecks	=sslsc	SLS Checks
OpenConnection		Open connection
Steel.Checkdata.Links	=slink	Links
Steel.Checkdata.Stiffeners	=sstiffen	Stiffeners
Steel.Connections.Framestrongaxis		Frame bolted/welded-strong axis
Steel.Connections.Frameweakaxis		Frame bolted/welded-weak axis
Steel.Connections.Gridpinned		Grid pinned
Steel.Connections.Bolteddiagonal		Bolted diagonal
Drawings.Sectiontolinegrid		Plan view
Drawings.Section	=secv	Section
Drawings.Planviewtodwg		Export section/Plan view to DWG
Drawings.Generalsection		General section
Drawings.Sectiontolinegrid		Storey-to-Plan view

Command	Abbreviation	Short explanation
Drawings.Sectiontolinegrid		Section-to-Line grid
Drawings.dimensions.linear.Aligned	=dal	Aligned
Drawings.dimensions.linear.Horizontal	=dhor	Horizontal
Drawings.dimensions.linear.Vertical	=dvert	Vertical
Drawings.dimensions.Stationing.Horizontal	=dshor	Horizontal
Drawings.dimensions.Stationing.Vertical	=dsvert	Vertical
Drawings.dimensions.Stationing.Aligned	=dsal	Aligned
Drawings.dimensions.baseline.Horizontal	=dbshor	Horizontal
Drawings.dimensions.baseline.Vertical	=dbsvert	Vertical
Drawings.dimensions.baseline.Aligned	=dbsal	Aligned
Drawings.dimensions.baseline.Angular	=dbsan	Angular
Drawings.dimensions.Radius	=dra	Radius
Drawings.dimensions.Diameter	=ddi	Diameter
Drawings.dimensions.Arclength	=darcl	Arc length
Drawings.dimensions.Angular	=dan	Angular
Drawings.dimensions.Arcangular	=darcan	Arc angular
Drawings.dimensions.Label	=dlab	Label
Drawings.dimensions.Labelwithleader	=dlable	Label with leader
Library.Materials	=mat	Materials
Library.Crosssections	=css	Cross-sections
Library.Catalogueblocks	=licb	Catalogue blocks
Library.Namedcsitem		Named item
Library.Analysis.Crosssectionlist	=licsli	Cross-section list
Library.Analysis.Sectionmatrix		Section matrix
Library.Analysis.Productrange		Fabricated Css, Product range, Joists
Library.Analysis.Nonlinearfunctions	=nlf	Nonlinear functions
Library.Analysis.Hingetype	=hit	Hinge type
Library.Analysis.Initialdeformations	=inidef	Initial deformations
Library.Analysis.Buckling	=buckl	Buckling
Library.Analysis.Emodulusfunction	=efun	E modulus function
Library.Prestessing.Fittinghollowcoreslabs	=hcfit	Fitting hollow core slabs
Library.Dampers	=damper	Dampers
Library.Analysis.Orthotropy	=liort	Orthotropy
Library.Steel.Diaphragms	=diaph	Diaphragms
Library.Steel.Bolts	=bolts	Bolts
Library.Steel.Boltdiameterrelation		Bolt diameter relation
Library.Steel.Connectiondatatocss		Connection data to CSS
Library.Steel.Hallconnection		Hall connection
Library.Concrete.Stirrups	=stirrups	Stirrups
Library.Concrete.Longitudinalreinforcement	=longrnf	Longitudinal reinforcement
Library.Concrete.2dreinforcementmesh	=2drnf	2D Reinforcement mesh

Command	Abbreviation	Short explanation
Library.Prestessing.Boreholepatterns	=bhp	Bore hole patterns
Library.Prestessing.Sectionalstrandpatterns		Sectional strand patterns
Library.Prestessing.Stressingbeds		Stressing beds
Library.Prestessing.Tendonsourcegeometry		Tendon source geometry
Library.Prestressing.Typeofstressing		Type of stressing
Library.Composite.Hollowcoreslab		Hollow core slab
Library.Composite.Compositeslabdata		Composite slab data
Library.Composite.Shearconnectors		Shear Connectors
Library.Geotechnics.Subsoils	=subs	Subsoils
Library.Geotechnics.Geologicprofiles	=geop	Geologic profiles
Library.Geotechnics.Geologicareas	=geoa	Geologic areas
Library.Geotechnics.Padfoundations	=pads	Pad foundations
Library.Loads.Predefinedloads	=pdl	Predefined loads
Library.Loads.Windpressures	=winpres	Wind pressures
Library.Loads.Seismicspectrums	=seisp	Seismic spectrums
Library.Loads.Dynamicloadfunctions	=dlfun	Dynamic load functions
Library.Loads.Unitmobileload		Unit mobile load
Library.Loads.Mobileloadsystems		Mobile load systems
Library.Loads.Loadpattern		Load pattern
Library.Fireheat.Thermaldistributioncurves		Thermal distribution curves
Library.Fireheat.Insulations	=insul	Insulations
Library.Fireheat.Temperaturetimecurve		Temperature-time curve
Library.Analysis.Functionoftimedependency		Function of time dependency
Library.Analysis.Functionoftemperaturedependency		Function of temperature dependency
Library.Concrete.Cellularbeamtype	=cellb	Cellular beam type
Library.Loads.Trains		Trains
Library.Drawing.Drawingstyle	=dwst	Drawing style
Library.Layers	=lyr	Layers
Library.Ucsmanager	=liucs	UCS
Library.Loads.Linesettlement	=linesetl	Line settlement
Library.Parameters	=prms	Parameters
Library.Loads.Combinations	=lcomb	Combinations
Analysis.Calculation	=calc	Calculation
Analysis.Hiddencalculation	=hicalc	Hidden calculation
Tools.Picturegallery	=pig	Picture gallery
Tools.Units	=Units	Units
Library.Loads.Resultclasses	=rclass	Result classes
Library.Templatesetting	=templs	Parameters template settings
Project	=proj	Project
Analysis.Checkstructuredata	=checkdata	Check structure data
Library.Loads.Massgroups	=massg	Mass groups

Command	Abbreviation	Short explanation
Analysis.Meshgeneration	=meshgen	Mesh generation
Library.Loads.Nonlinearcombinations	=nlcomb	Nonlinear combinations
Library.Loads.Stabilitycombinations	=stbcomb	Stability combinations
Library.Loads.Combinationofmassgroups	=masscomb	Combination of mass groups
Library.Loads.Concretecombinations	=concomb	Concrete combinations
Absences.Absencegroups		Absence groups
Analysis.Solversetup	=so	

List of keyboard shortcuts

Default shortcuts in SCIA Engineer

Scia Engeneer provides set of basic shortcuts as default. See the list below:

Keyboard shortcut	Definition
F1	Opens help page
F2	Opens structure service
F3	Opens load service
F4	Opens results service
SHIFT+F4	Opens results service (v17)
F5	Refresh of results
CTRL+F5	Recalculation (available for solverlink calculation only)
CTRL+SHIFT+F5	Calculation
F6	Cursor snap setting dialog
CTR+F6	Next view port
CTRL+SHIFT+F6	Previous view port
F7	On/Off tracking
F8	Ortho
F9	Move LCS
F10	LCS according to entitiy
F11	Switches workplane (XY,XZ, YZ)
CTRL+F11	Sets GCS
F12	Sets view perpendicular to workplane
CTRL+F12	Sets view to AXO
CTRL+SHIFT+F12	On/Off perspective
CTRL+Q	Activity On/Off
CTRL+W	Activity by layers
CTRL+E	Activity by selection (Selected members On)
CTRL+R	Rotate
CTRL+T	Create plate
CTRL+Z	Undo
CTRL+O	Opens new project
CTRL+P	Print picture

Keyboard shortcut	Definition
CTRL+A	Selects all
CTRL+S	Save
CTRL+SHIFT+S	Save as
CTRL+D	Activity by selection (Selected members Off)
CTRL+SHIFT+D	Coordinates info dialog
CTRL+SHIFT+F	Match properties
CTRL+G	Opens line grid and storeys service
CTRL+SHIFT+G	Line grid manager
CTRL+SHIFT+H	Create wall
CTRL+J	Cross-sections
CTRL+K	Combinations
CTRL+L	Load cases
CTRL+Y	Redo
CTRL+X	Move entity
CTRL+C	Copy entity
CTRL+N	New project
CTRL+B	Create 1d member
CTRL+SHIFT+B	Create column
CTRL+M	Materials
CTRL+TAB	Next opened SCIA Engineer project
CTRL+SHIFT+TAB	Previous opened SCIA Engineer project

Default shortcuts in Engineering Report

Scia Engeneer provides set of basic shortcuts as default for Engineering report as well. See the list below:

Keyboard shortcut	Definition
CTRL+P	Opens print dialog
F5	Regenerate outdated
F6	Regenerate selected

Property table

A property table is a SCIA Engineer unique control used in the program dialogues and in the Property window. The control looks like a table (basically a two column multi-row table) whose first column contains names of individual items displayed in the table and the second column shows their values.

Generally, the values in the "value cells" of the property table may be modified. There are various means for the change of the value (see bellow). In addition, the individual items of the table may be interlinked either (i) to another part of the program (e.g. another dialogue) or (ii) to a graphical window. Both variants represent a powerful feature increasing significantly the simplicity and speed of editing process.

In order to unify the appearance of the program dialogues, the property table is also used even for passive display of information. In such a case, the "value cells" are disabled to prevent an accidental alteration of the values.

Type of property table cells

name cell	It contains the name of the item whose value is displayed in the coupled value cell.
group cell	This is a special case of the name cell. Sometimes, the name cell is standalone and is not coupled with any value cell. This is used to display e.g. the name of a group of items.
value cell	This cell holds the corresponding data. The data may or may not be edited depending on the particular situation.

The value cell may be of several types. Where possible, the cell terminology is taken from the standard MS Windows terminology for dialogue box components. In parenthesis, a descriptive name is added (if applicable).

edit box (simple value cell)	The basic type of cell provides for manual input of value. Depending on the particular item the value may be either numerical or alphanumerical.
combo box (selection list cell)	This control is used for items where the proper value is defined by selection from a list of available variants.
tick box (yes/no cell)	This type of cell provides for two limit value only – for YES and NO.
button	The button can be used to start a required type of action, e.g. open a dialogue, etc.
colour list	This type is similar to the combo box. The difference is that it offers colours only.

Combination of cell types in one table cell

The individual cell types may be combined within a single cell. That means that, for example, one table cell may consist of a combo box and a button, or of three edit boxes.

This feature is used e.g. in tables where a cross-section should be specified. The table sell then contains:

- a combo box with all cross-section already defined in the current project,
- a button that opens the Cross-section manager and thus provides for the definition of a new cross-section type if none of the existing ones meets requirements on the particular item.

Interconnection between table cells and graphical window

In some dialogues, individual table items may be related to a specific part of the drawing shown in the graphical window. In such a case, it would be useful:

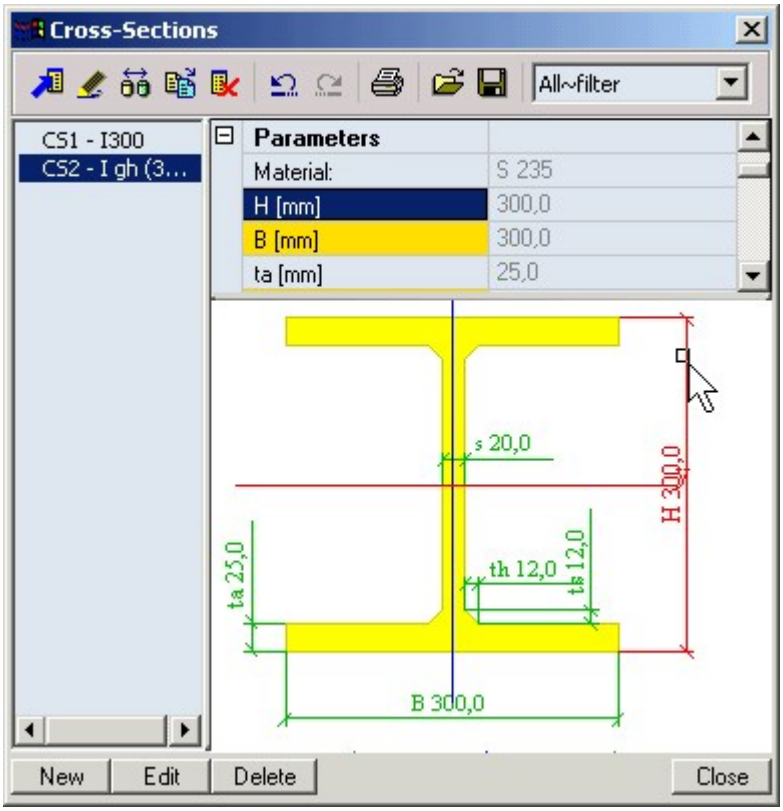
- to highlight the appropriate part of the drawing if the corresponding table cell is selected, or
- to highlight the appropriate table cell if the corresponding part of the drawing has been clicked on.

The SCIA Engineer property table makes this possible. Therefore, where applicable and useful, the appropriate table cells are interlinked with corresponding drawing parts.

As an example we may give the dialogue for editing of a cross-section. Here, the dimensions of a cross-section represent exactly what this feature is ideal for. On clicking any of dimension lines in the drawing, the corresponding table row is highlighted, and vice versa.

Example of a property table

The picture below shows the cross-section editing dialogue. The mouse cursor is positioned in the graphical window of the Cross-section manager over the height dimension line. After the left mouse button was clicked, the corresponding item in the table above the picture got the focus (the blue item).



Progress bar

Especially for large models, some actions performed in SCIA Engineer may be rather time consuming. In order to tell the user what the progress is, a progress bar is shown on the screen.

It simply:

- indicates that the program is working,
- measures what portion of the total work has been already finished.

The progress bar may appear either in a modal dialogue or on a status bar.

It may look like e.g.:

Note: If the application window is not maximized, it may happen that the progress bar cannot fit into the status bar whose length is limited by the adjusted width of the application window. In that case, the progress bar that would normally appear on the status bar is invisible.

User Interface Skins

SCIA Engineer can be run with a standard Graphical User Interface (GUI) or with a simplified user interface. The latter is analogous to skins used in some other programs. In SCIA Engineer, these "skins" do not just alter the look of the program, but they may also reduce the available functionality (they are not capable of extending the functionality). This may be useful

for clients who focus on a particular group of problems. For example, if you use SCIA Engineer as a modeller and do not intend to perform any kind of calculation, it is redundant to have in the menu functions for input of loads, supports, hinges, for start of calculation, review of results, etc. What's more, it is also possible to change the arrangement of toolbars and tree window and change icons on toolbars.

A good example of a practical application of this technology is 3D Free Form Modeller.

3D Free Form Modeller is an application based on the full SCIA Engineer, but leaving aside calculation and code check capabilities. In addition, as it is intended for Allplan users, it replaces some standard SCIA Engineer-style icons on toolbars with icons from Allplan, with the only aim: to make it easier for Allplan users to work efficiently with 3D Free Form Modeller.

Toolbars

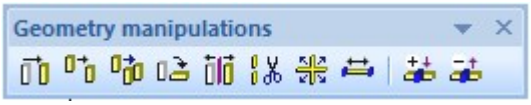
Toolbars

Toolbars are small floating windows-like objects containing sets of buttons. The buttons can be used for opening various functions. The toolbars may be let floating on the screen or may be docked to any side of the screen.

Examples

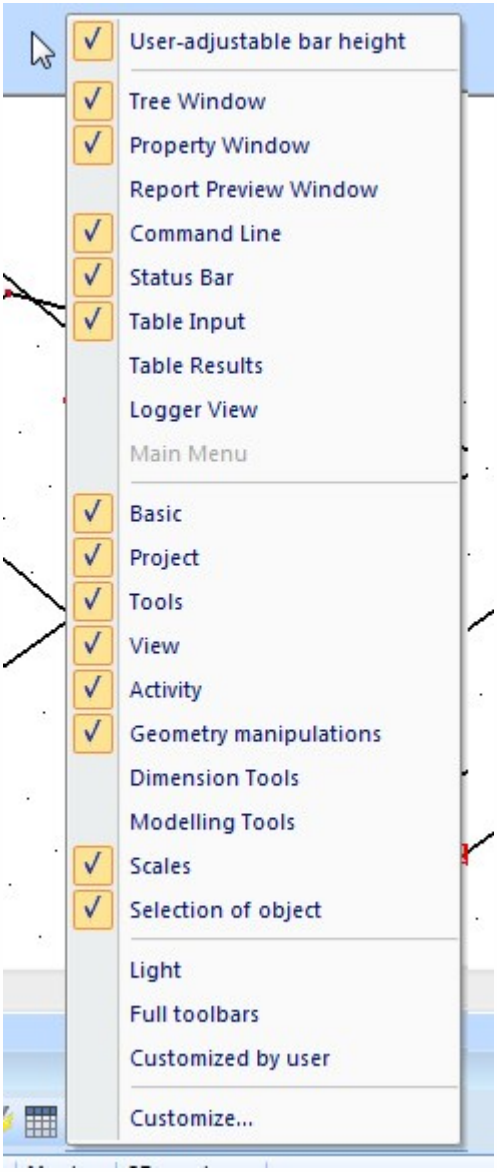
View

Geometricalmanipulations



You may control which toolbars are displayed in menu View > Toolbars. This menu function enables you to switch on or off the required toolbars. Moreover, you may use this function to display or hide other parts of the Graphical User Interface (GUI).

You can also control display of toolbars through right-click drop down menu.



Another parts of GUI that can be displayed or hidden:

- tree window,
- property window,
- report preview window,
- command line,
- status bar,
- table input,
- table result,

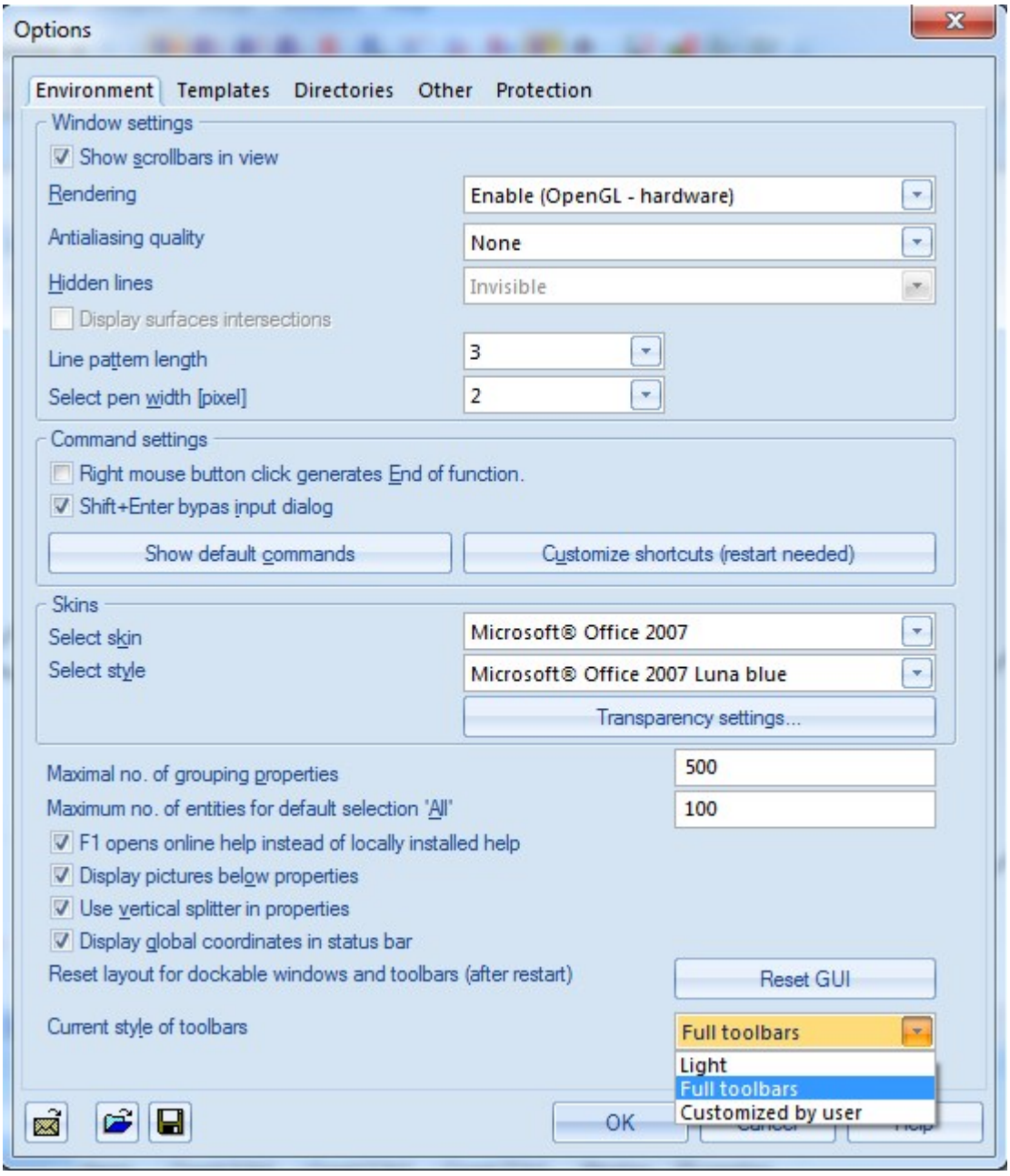
List of available toolbars:

- Basic (New, Open, Project manager),
- Project (e.g. units, cross section, materials etc.),

- Tools (e.g. UCS, clipping box, etc.),
- View,
- Activity,
- Geometry manipulation (e.g. Move, Copy, etc.),
- Dimension Tools,
- Modelling tools (e.g. Boolean operations with general solids, Generation and modification of vertices on general solids, etc.),
- Scales,
- Selection of objects.

Predefined toolbar arrangements

Even though you may freely move the toolbars on your screen and let them "flow" or dock them to any side of application window, you can also select from two predefined configurations of toolbars in right-click drop down menu or in setup.



- Light – Default arrangement ,
- Full – adds Dimension and Modelling tools toolbars and some other buttons to another toolbars
- Customized by user – user defined state of toolbars (position, number or content through Customize dialogue)

Note: Switch of configurations will take effect after restart of SCIA Engineer

Customizing the toolbars

Toolbars can be customised by the user. It is possible to reshape the toolbars, add or remove buttons from individual toolbars and to define new tailor-made toolbars.

Each toolbar has a little-arrow button (the button is located at the right end of the toolbar if the toolbar is docked and at the toolbar header if the toolbar is floating - see the two images below) . When the little-arrow button is clicked a submenu opens with option **Add or remove buttons**. This item then offers several sub-items:

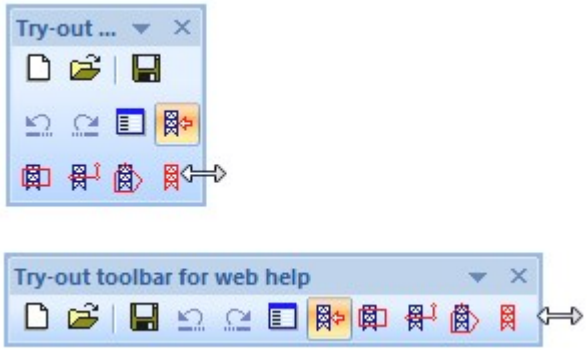
- the names of toolbars that are docked in the same "toolbar-row" (in case of a floating toolbar, it contains only the name of the particular toolbar),
- item **Customise** that opens the **Customize** dialogue (described further in the text).

Picture: little-arrow button (marked with red circle) on a floating and docked toolbar

Reshaping the toolbar

Each floating toolbar can be reshaped. Simply put the mouse cursor over an edge of the toolbar, click the mouse left button and drag.

Example:



Hiding buttons from a toolbar

1. Click the little-arrow button on the required toolbar and open the sub-menu.
2. Select the name of the toolbar you want to modify.
3. Another "sub-menu" with a complete list of available standard buttons for the toolbar is opened.
4. Unmark the buttons you want to hide and select the buttons you want to see.

> If the toolbar if floating, this procedure can modify only the toolbar whose little-arrow button has been clicked. If the toolbar is docked, this procedure can access all the toolbars located in the same "toolbar-row".

Dialogue "Customize"

You can customize toolbars via Customize dialogue (content, order or new buttons).

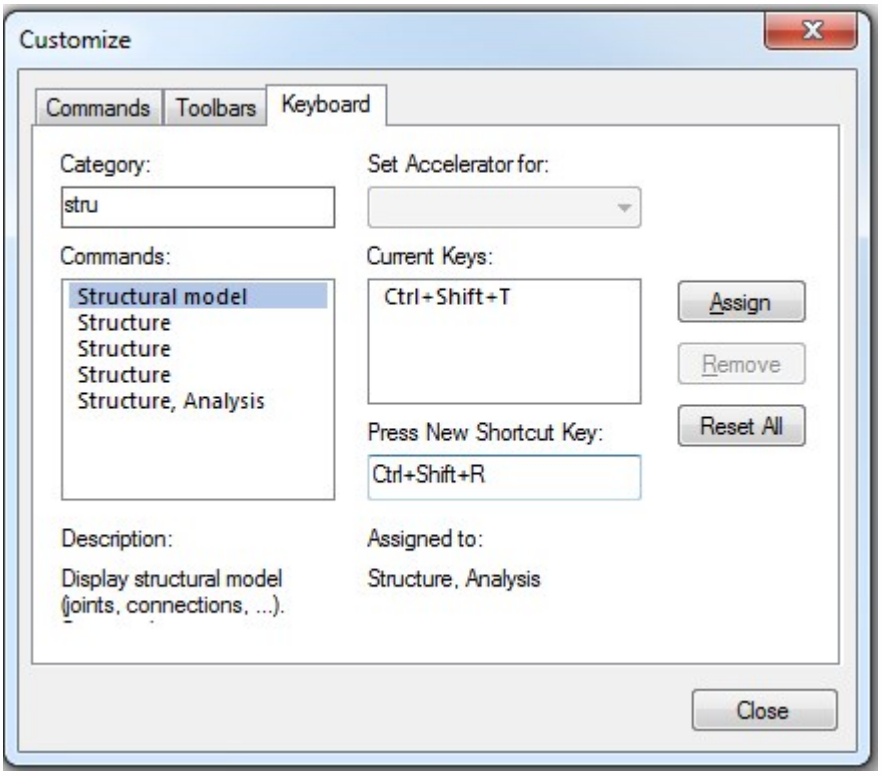
Customizing keyboard shortcuts via customize dialogue

Scia Engineer provides possibility to define user shortcuts in customize menu.

See the default keyboard shortcuts.

Those shortcuts can be replaced and re defined in customize menu as well

Keyboard tab



This tab offers a list of all available commands and possibility to assign keyboard shortcut to them.

You can search for commands through search text line or click into the list and start typing first letters of command you looking for.

Description field can help you to find what you are looking for.

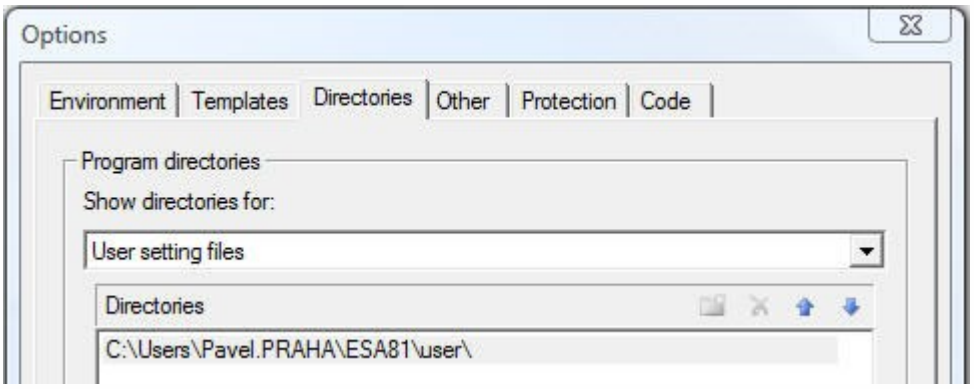
You can see if the command has already assigned an shortcut and you will be notified in case you will try to assign keyboard shortcut which is already in use by other command.

"Reset all" button will discard all user-defined shortcuts and return the setting to factory default.

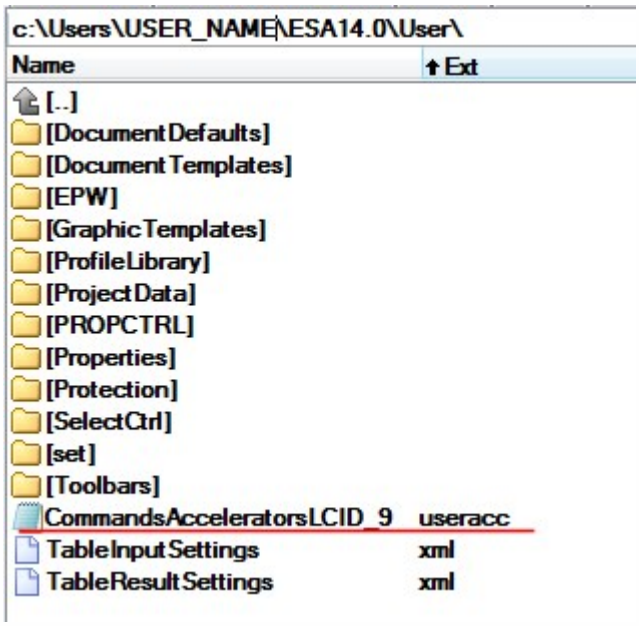
Carry the customised shortcuts between different computers

The settings adjusted on one computer can be easily transferred to another computer. It can be useful, for example, if one engineer works on several different computers or if a team wants to share the same settings.

The settings made by the user are stored in folder for "User settings files" that is defined in the **Setup > Options dialogue.**



Look for .useracc file in user folder. Just simply copy this file and input it under the user folder on another computer

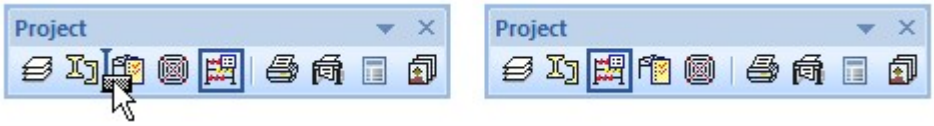


TIP: you can use customize dialogue to assign shortcuts for commands to open services in main tree and make your workflow much faster

Customizing toolbars via customize dialogue

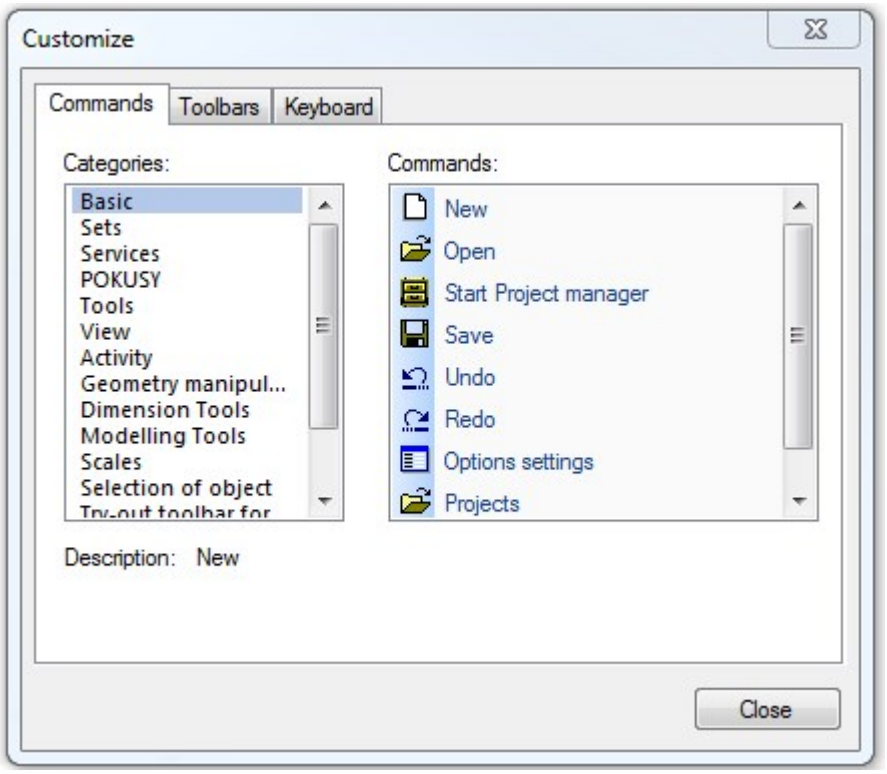
Adjusting of the toolbars when customize dialogue is active

You can change order of buttons within toolbars by easy drag and drop in toolbar itself.



Discard the button from toolbar by drag it and move it outside of toolbar.

Commands tab

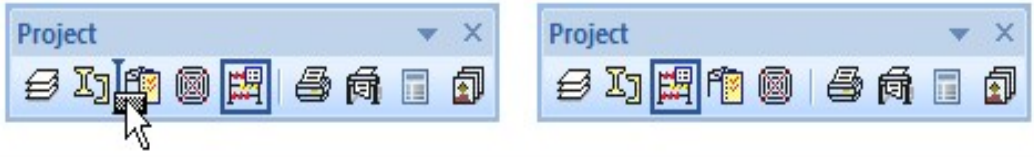


This tab offers a list of all available toolbars and their buttons.

When on this tab, you can drag-and-drop any command from the dialogue to any displayed toolbar.

1. Select the required toolbar in the left list.
2. Select the required button in the right list.
3. Click it and drag to the required toolbar.
4. Release the mouse button - the selected function is added to the target toolbar.

You can change order of buttons within toolbars by easy drag and drop in toolbar itself.



To remove a button from any of the existing toolbars, just "drag" the required function away from the toolbar (the Customize dialogue must be opened).

1. Select the function to be removed from a toolbar.
2. Click it and drag it anywhere away from the toolbar (outside the toolbar area of all toolbars).
3. Release the mouse button and the function is removed from the toolbar.

Discard the button from toolbar by drag it and move it outside of toolbar.

Toolbars tab

On this tab you can:

- display or hide any of the existing toolbars,

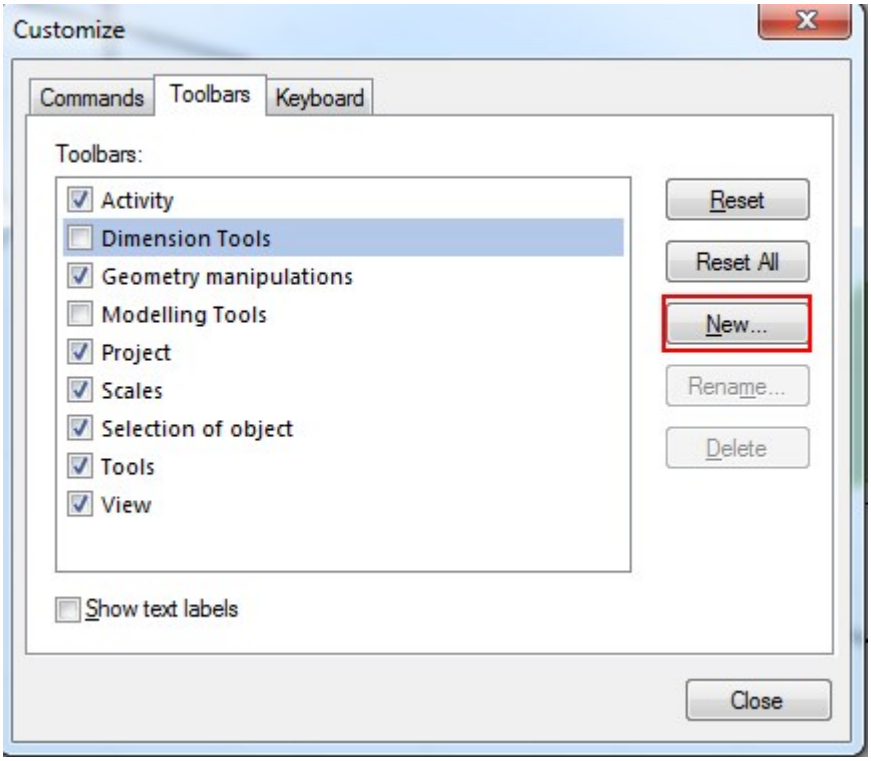
- reset the toolbar to the default configuration,

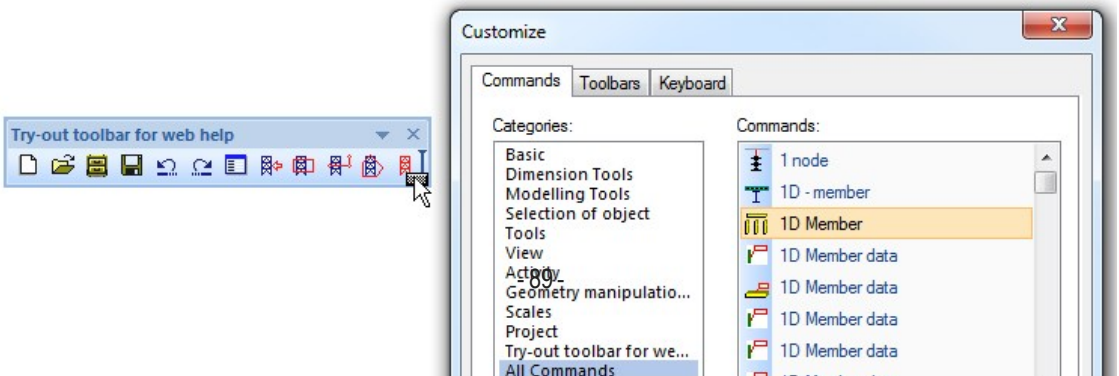
- create a new toolbar(s),

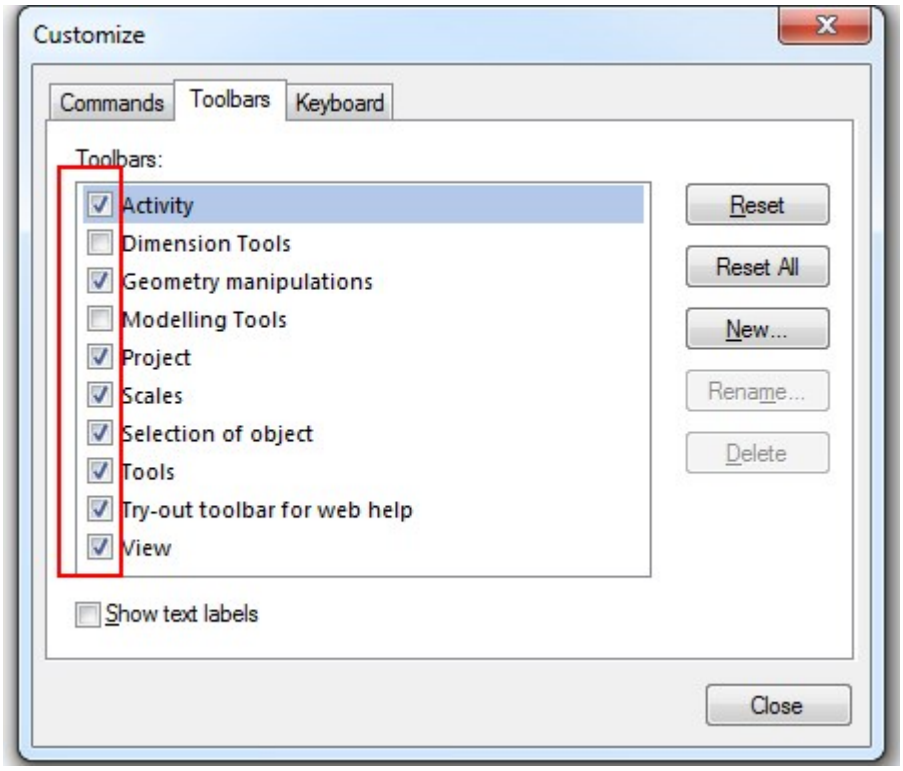
- delete you user-made toolbar(s),

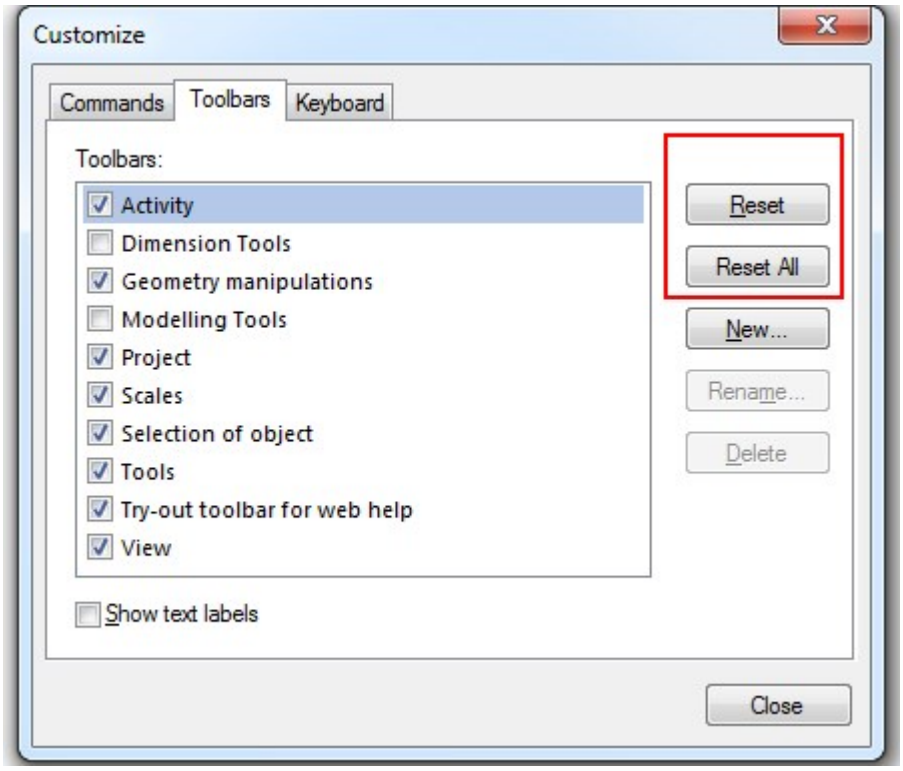
- rename you user-made toolbar(s),

When you create a new toolbar, swap to the Commands tab and drag-and-drop the required functions on it.









TIP: you can use customize dialogue to create new toolbar and add there buttons related to opening services in main tree (structure, results, etc.) and make your work flow much faster.

Reset do not work for user defined toolbars.

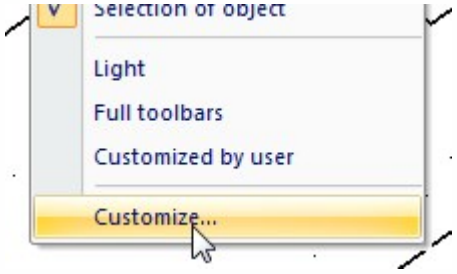
When a new toolbar is created, it may not appear in the list of existing toolbars in the **Customize** dialogue. In that case, close the **Customize** dialogue and reopen it. The new toolbar will be listed there then.

Customize dialogue

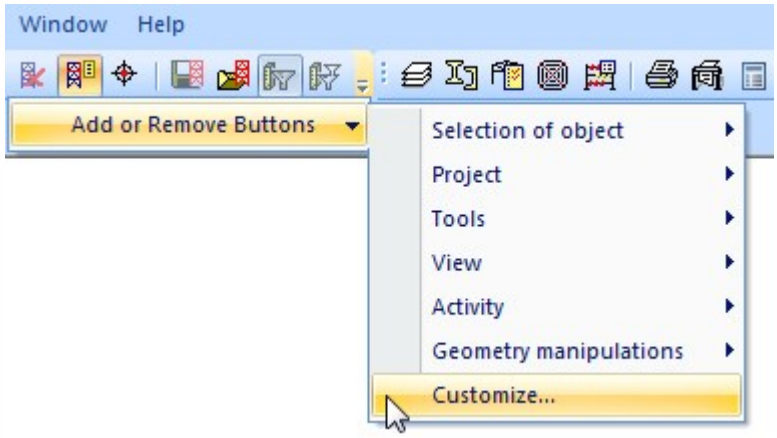
Customize dialogue is a service which allow user to adjust toolbars an d customize keyboard shortcuts.

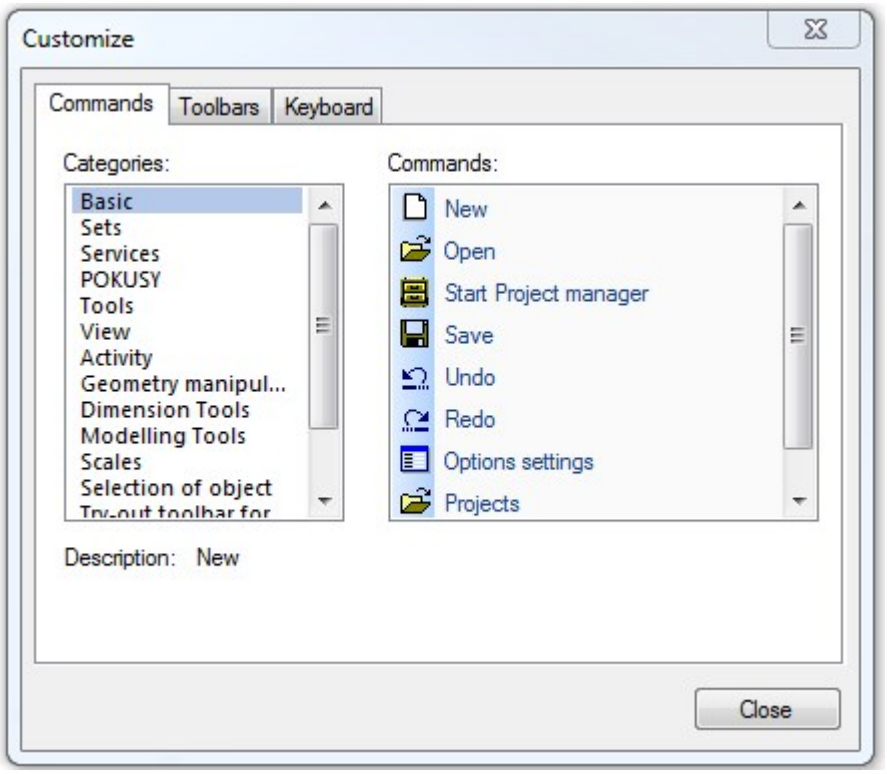
Access to customize dialogue:

Right-click drop down menu



Toolbar drop down menu





Application windows

Introduction to application windows

All the information that the program can give to the user is displayed in an application window. An application window can of the following types:

- graphical window,
- document window,
- preview window.

The user can use all the window types at the same time and swap between them freely, or he may use just one type at a time. It depends completely on his or her will and habits.

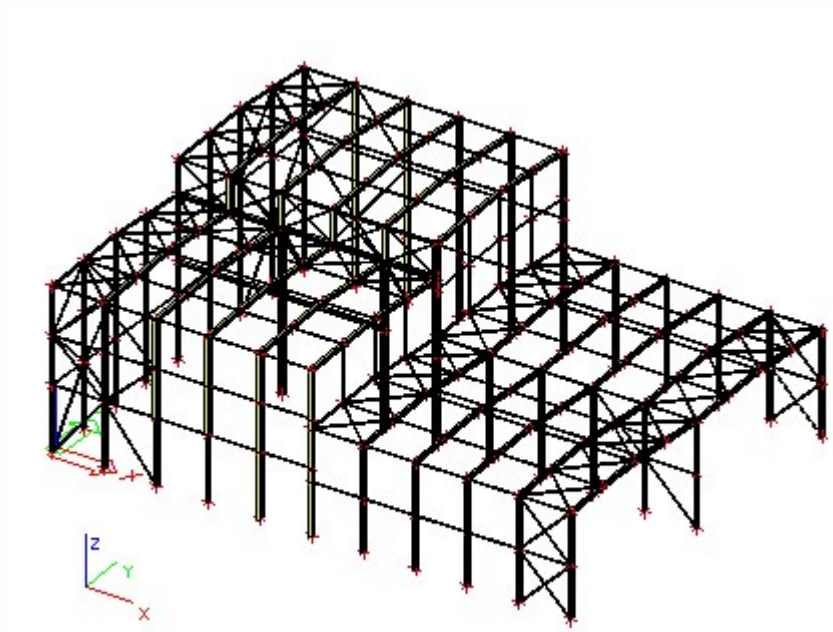
At the same time, as many graphical and document windows can be opened as the user considers convenient to him. On the other hand, there can be opened just one preview window.

Graphical window

This window can be perceived as a drawing board, however with rather advanced functionality. A model defined by the user is displayed in this window. The individual parts of a model can be literally drawn in this window. All selections of any function are made in this window type and any response of the program to the user's action affecting the model is shown in this window. Also the calculated results are shown in this window. The window both displays the project data and receives information from the user provided by means of mouse moves and clicking.

An arbitrary number of graphical windows, regardless of their type, can be opened at the same type for one or several different projects.

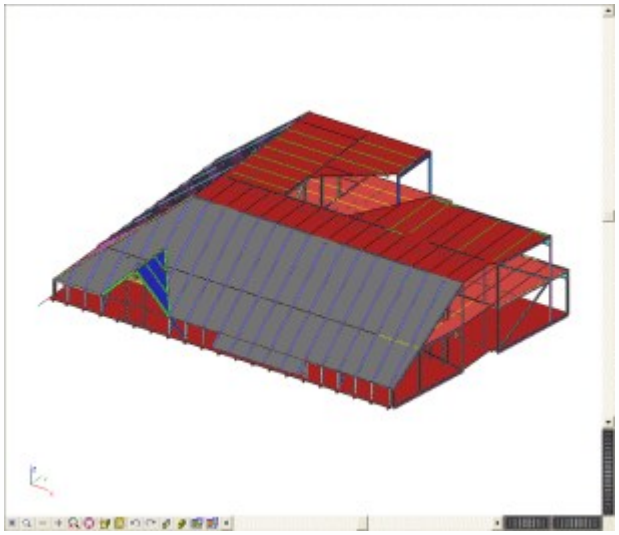
Example of a graphical window

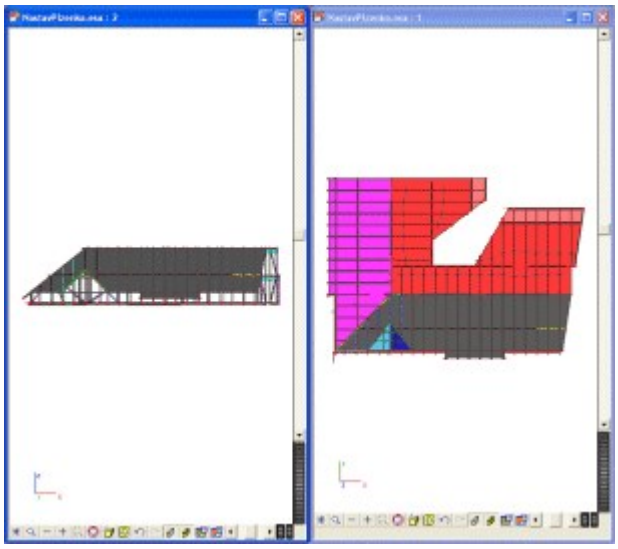
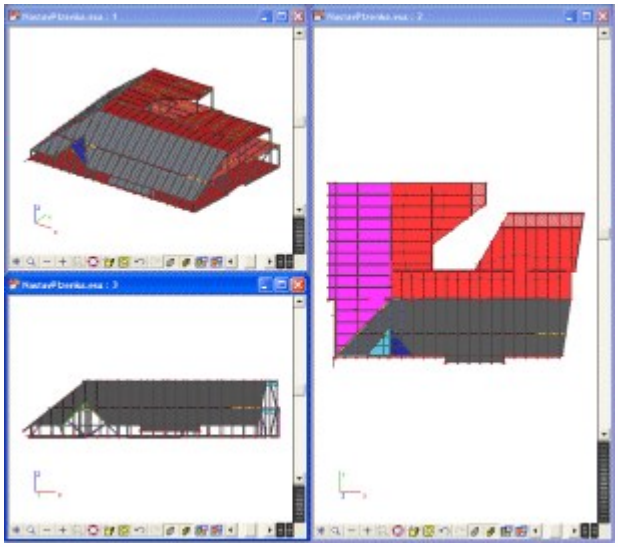
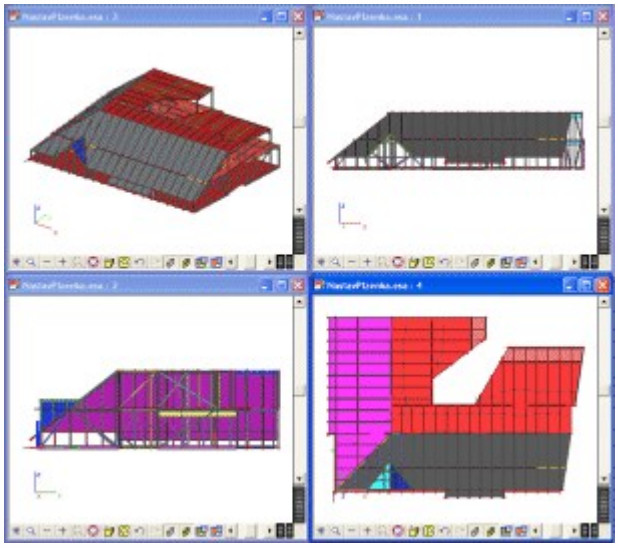


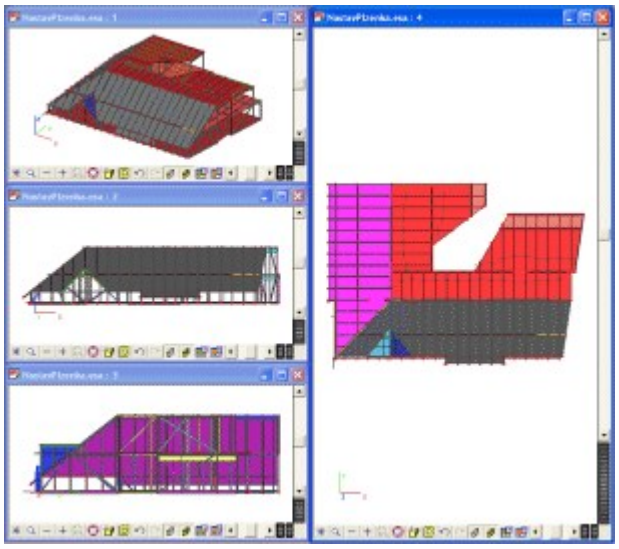
Viewports

The term "viewport" is taken from Allplan and means a graphical window.

The Window menu offers several predefined arrangements of viewports (windows).

1 viewport	

2 viewports	
3 viewports	
4 viewports (1)	

4 viewports (2)	

Besides, it is of course possible to arrange the windows in any other way that suits your needs or habits.

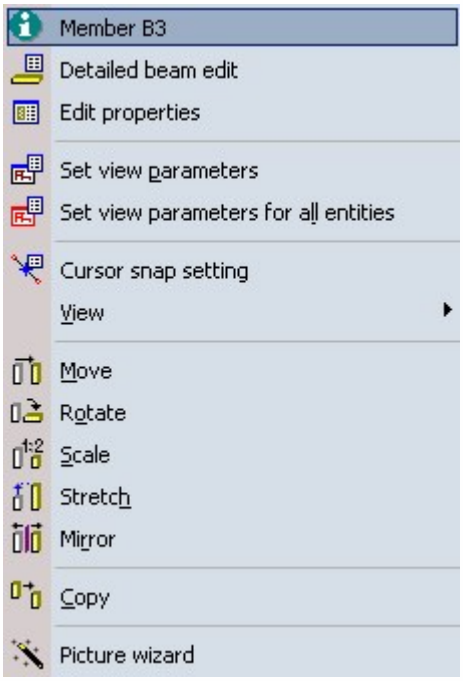
Graphical window pop-up menu

Every graphical window that SCIA Engineer creates has a pop-up menu associated with it. This menu provides for a fast access to some frequently used functions.

To access this menu, move the mouse pointer so that it is within the window - not inside the title bar, nor on the window's borders. Then press the rightmost mouse button to make the menu appear on the screen. Then move the mouse to highlight the required option. Click the leftmost button to start the selected action.

The window pop-up menu is described in detail in a separate chapter Basic working tools > Window pop-up menu.

Example of a pop-up menu

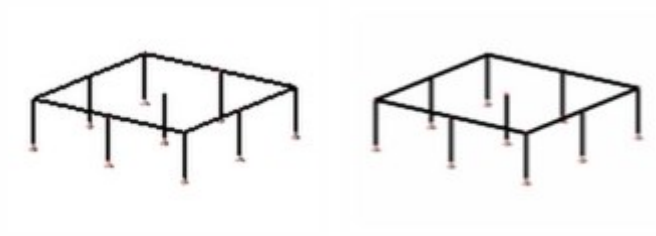


Antialiasing

What antialiasing is

The prefix “anti” can be defined as counteracting or neutralizing. Aliasing is a jagged, stair-step effect on curved or diagonal lines. Therefore anti-aliasing means to counteract and neutralize jagged lines.

Image demonstrating Aliasing effect. On first picture aliasing effect can be noticed on slab edges. Second picture displays same structure with antialiasing (AA) algorithm switched on.



How antialiasing works in SCIA Engineer

There is variety of antialiasing algorithms available. Most of them are directly supported by modern 3D graphic cards hardware as they are part of OpenGL instructions set. Starting SCIA Engineer 2012 we use one from variety of algorithms called Accumulation Buffer Antialiasing. We plan to support more different algorithms in the future to provide user with best possible output, similar as known from other 3D modeling programs or even 3D gaming industry.

Basically all antialiasing algorithms requires to generate 3D Scene more times or on higher resolutions than it is actually displayed on user display. Therefore all antialiasing algorithms slows down response of 3D control. This slowing down effect mainly depends on actual hardware - 3D graphic card. The faster the hardware is, effect of slowing down the 3D Scene is less evident.

SCIA Engineer native antialiasing algorithm comes with 3 predefined settings. Quality of output was carefully balanced to speed reduction.

None
Antialiasing algorithm is completely switched off. Aliasing effect can be seen on curved or diagonal lines.

Medium quality
Antialiasing algorithm runs with very gently and balanced attributes. When using standalone 3D graphic card, user should not notice any significant speed reduction on small to medium sized projects.

High quality
Attributes of antialiasing algorithm are more than double as high as for Medium quality, therefore speed reduction might become evident. This option is comfortable for users using modern 3D graphic card on small to medium sized projects. It is not recommended to use this option on extreme sized projects such as high-rise building or huge apartment blocks. Project size is meant to be number of entities that 3D Control has to display at the same time. Therefore for example even that bridge design project is challenging from engineering perspective, does not mean big challenge for 3D control speed. As opposite to this, a huge steel structure consisting of thousands of beams might become very challenging to handle.

Super quality
This setting is meant to be used by users, who prefer to use latest available hardware. Generally this option makes sense to be used to export high quality output images, such as bitmap or clipboard images. Main difference in algorithm attributes to High quality is level of blur that is used to lower aliasing effect. This setting might become highly subjective, as jagged lines might looks slightly different than original - some lines might appear to be more thick than before while some edges might appear to be smudged comparing the original picture. However at the same time, final output might appear more natural in documentation or print, giving more "photorealistic" feeling. Slow down is noticeable even on fastest hardware, but is fairly usable on small - medium sized projects. Important precondition is to use powerful 3D graphic card.

Typical use

Standard user with typical or older hardware

This typically includes standard notebooks, laptops or desktop computers 3 or more years old. For this kind of users, preferred default option is to switch Antialiasing OFF for 3D Scene manipulations. However users can still benefit from higher antialiasing levels, but only when exporting the final picture - to Document, Gallery, Clipboard or file.

User with latest hardware

This user is using latest desktop hardware with high-resolution display. User use modern 3D graphic card, such as Nvidia Quadro or similar product line. User is keen to experiment with various settings for different projects. It is highly usable for him to use Medium to Super quality levels for his everyday work.

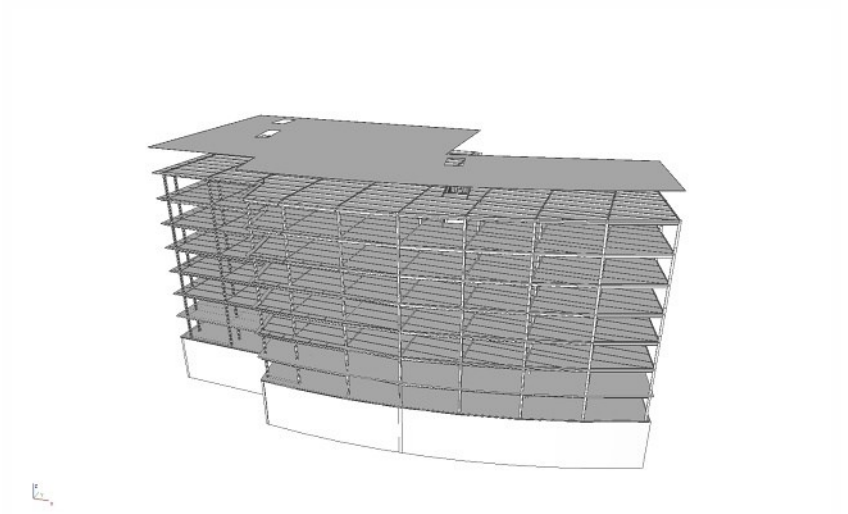
Examples

SCIA Engineer antialiasing algorithm is valid for completely all 3D scene content. As SCIA Engineer is primarily FEM analysis tool, it is worth to explore the option on various types of analysis results, such as deformed mesh or internal forces diagram courses. This will add extra readability level to SCIA Engineer output and it will produce pleasant and professional looking pictures. This output not only represents structural engineer, but extra readability level ads extra benefit to all picture outputs.

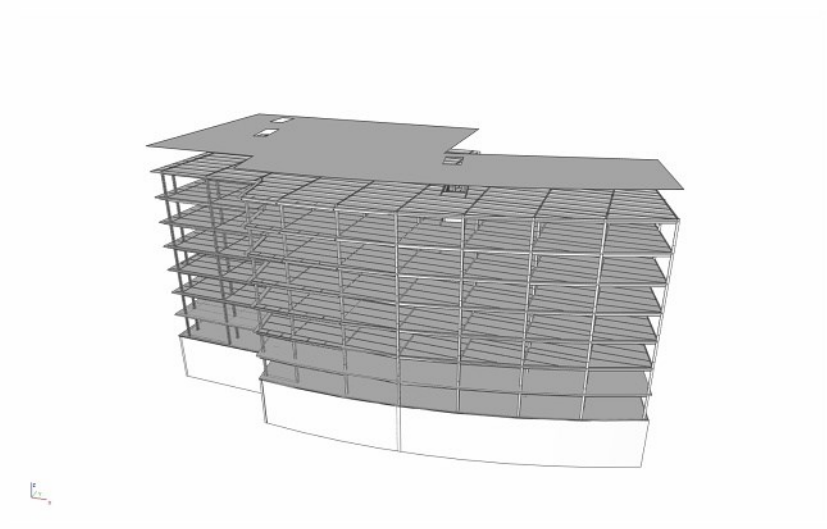
Couple of exported images with None / Super quality switched on.

Any 3D Scene picture with antialiasing switched on can appear different when viewed by individuals with varying levels of color sensitivity. People with various levels of astigmatism might find antialiased output to be uncomfortable to work with. Therefore after any sign of any subjective discomfort it is recommended to keep this setting switched off.

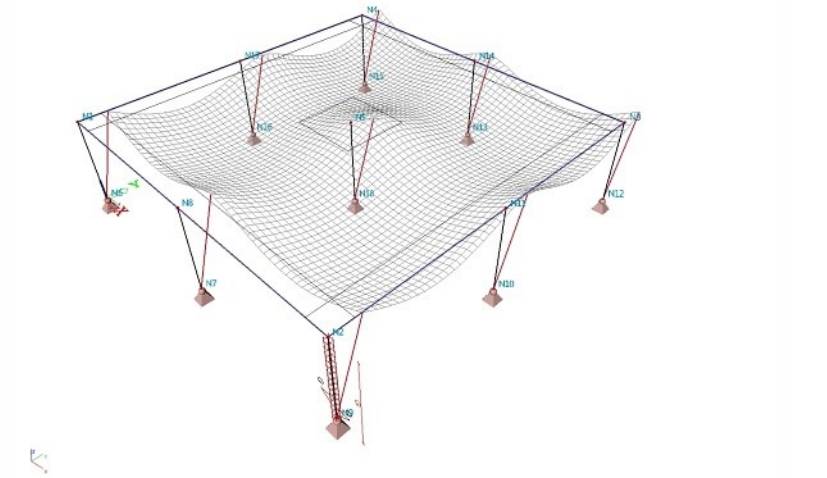
Before:



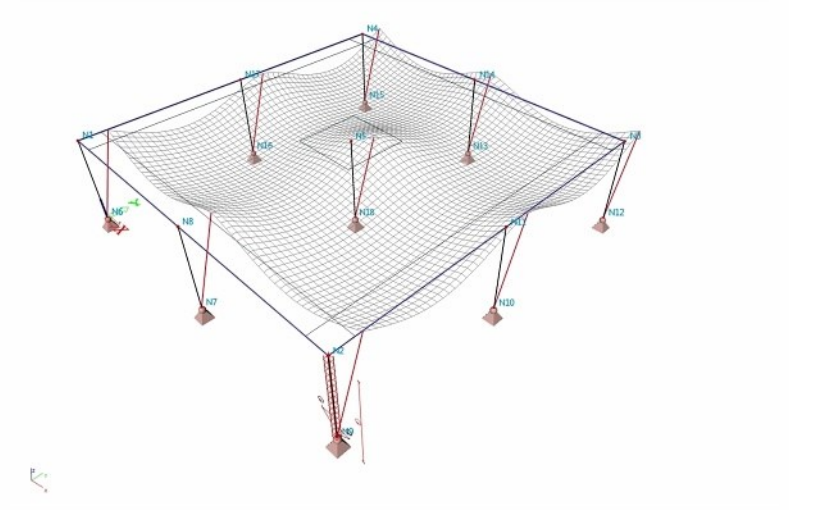
After:



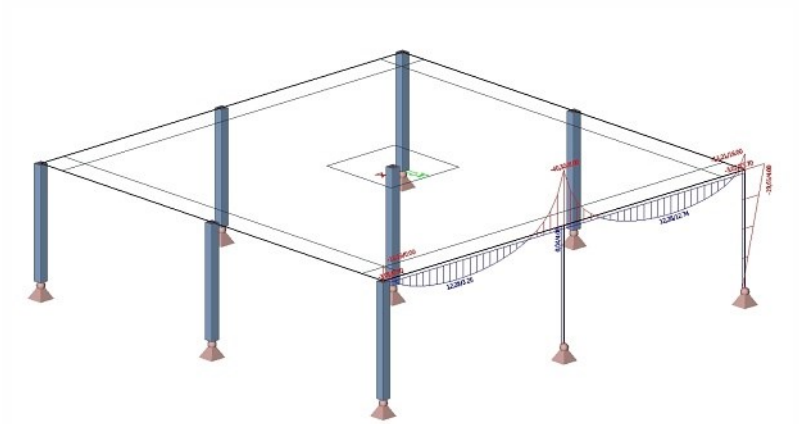
Before:



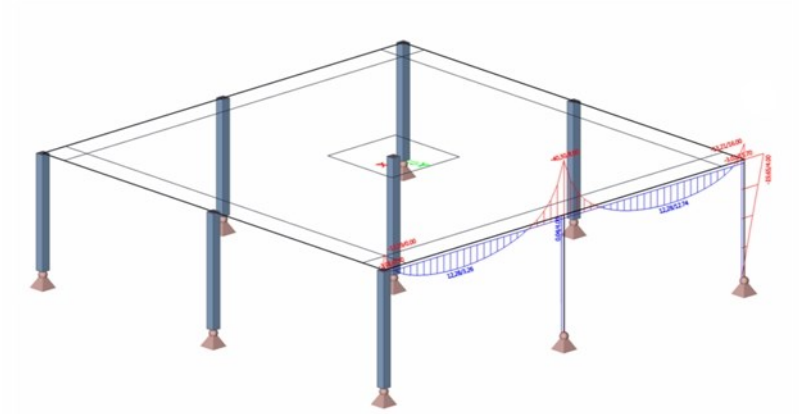
After:



Before:



After:

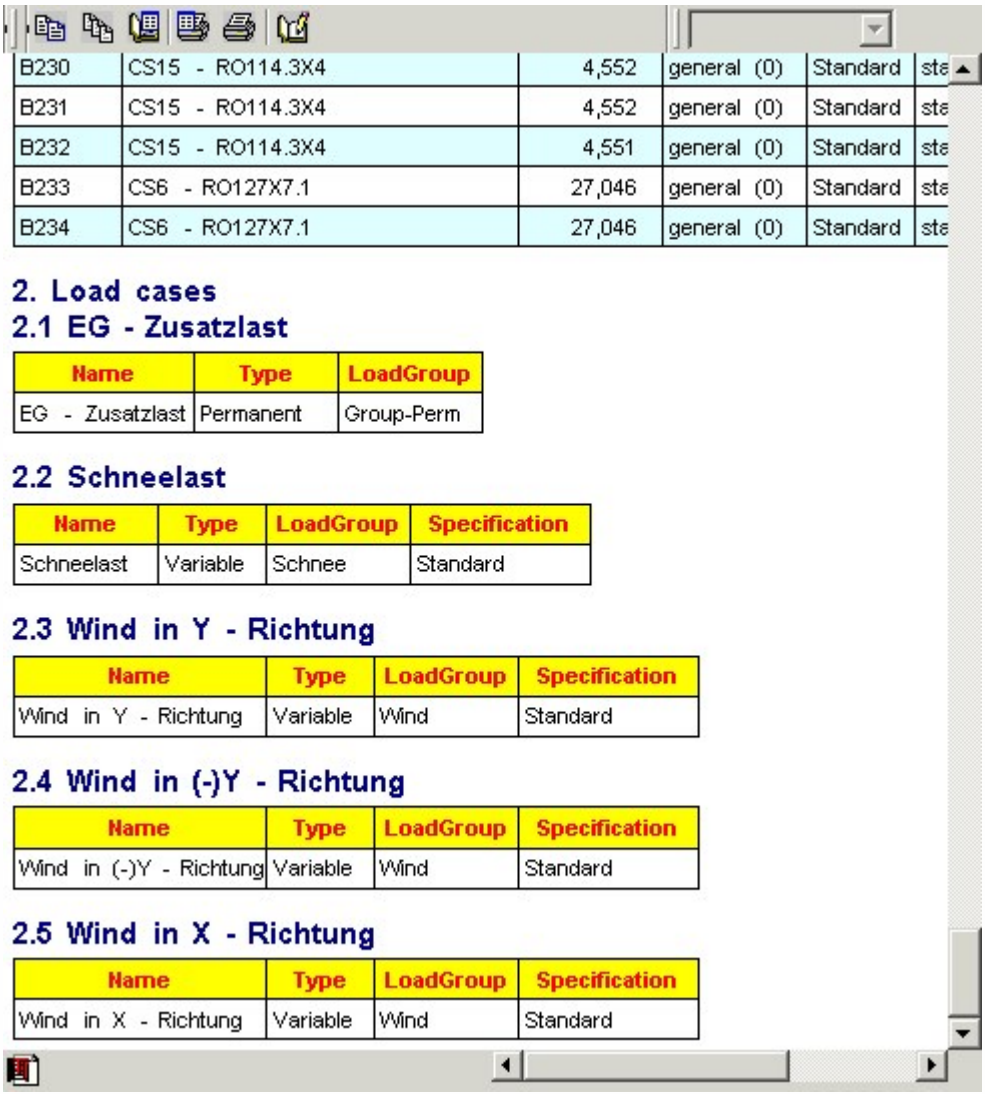


Document window

This window type is used to display a document or report about an analysed model, its input data, results of calculation, and assessment to technical standards (i.e. code check). This window can contain both graphical and text information.

An arbitrary number of document windows, regardless of their type, can be opened at the same type for one or several different projects.

Example of a document window



B230	CS15 - RO114.3X4	4,552	general (0)	Standard	ste
B231	CS15 - RO114.3X4	4,552	general (0)	Standard	ste
B232	CS15 - RO114.3X4	4,551	general (0)	Standard	ste
B233	CS6 - RO127X7.1	27,046	general (0)	Standard	ste
B234	CS6 - RO127X7.1	27,046	general (0)	Standard	ste

2. Load cases

2.1 EG - Zusatzlast

Name	Type	LoadGroup
EG - Zusatzlast	Permanent	Group-Perm

2.2 Schneelast

Name	Type	LoadGroup	Specification
Schneelast	Variable	Schnee	Standard

2.3 Wind in Y - Richtung

Name	Type	LoadGroup	Specification
Wind in Y - Richtung	Variable	Wind	Standard

2.4 Wind in (-)Y - Richtung

Name	Type	LoadGroup	Specification
Wind in (-)Y - Richtung	Variable	Wind	Standard

2.5 Wind in X - Richtung

Name	Type	LoadGroup	Specification
Wind in X - Richtung	Variable	Wind	Standard

Preview window

Introduction

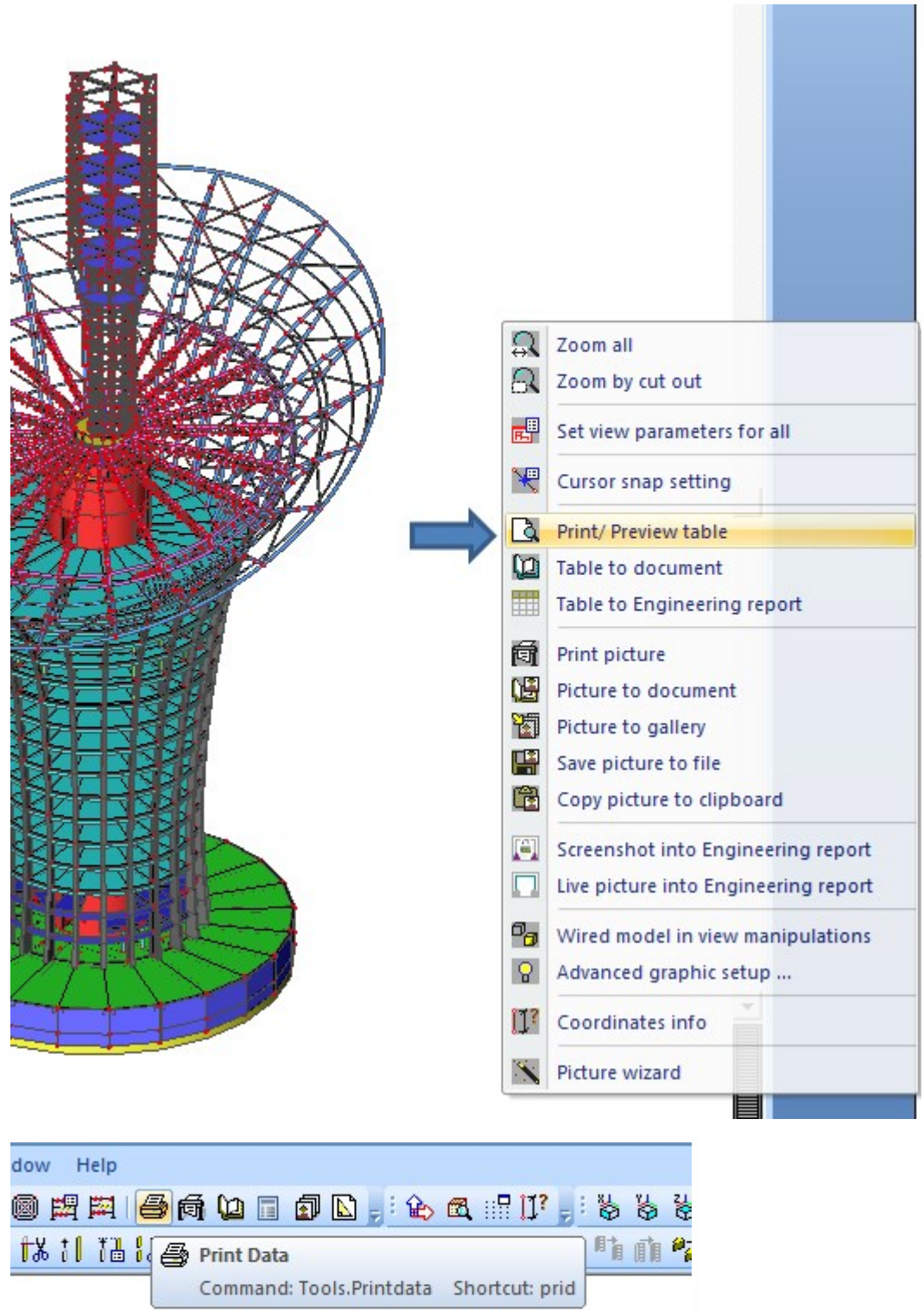
The Preview window is a document-like window that can be used for tabular preview of selected entities or results.

Many principles are shared with Engineering Report, therefore many instructions related to using of Preview window will be identical to instruction related to Engineering report.

Opening the Preview

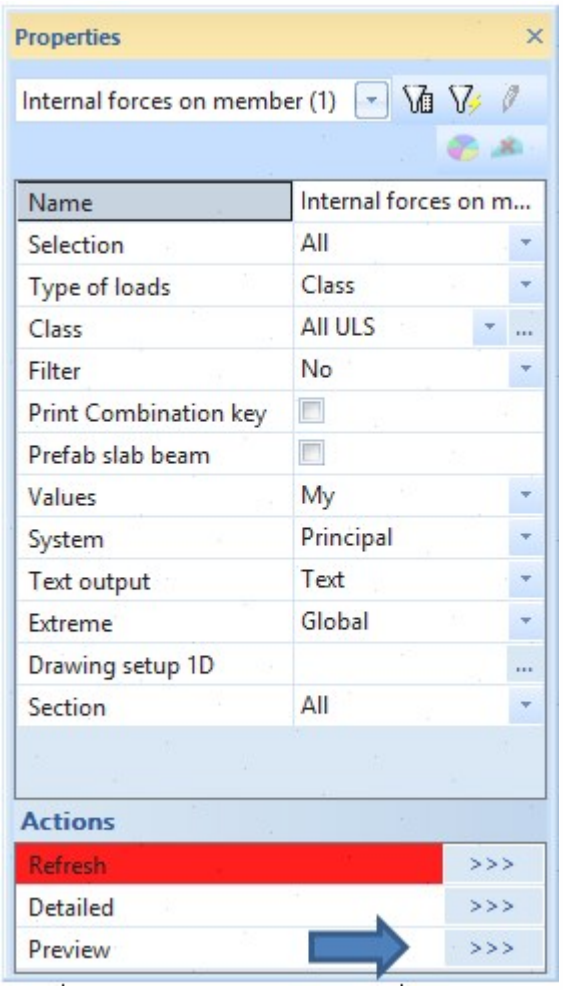
The procedure for the opening of the Preview window for **non-results tables**:

1. In the graphical window of SCIA Engineer select the entities that should be included into the preview.
2. Call function Print / Preview table:
   1. either using menu function File > Print data > Print / Preview table,
   2. or clicking button [Print] on toolbar Project.
3. The preview window is opened and tables are displayed

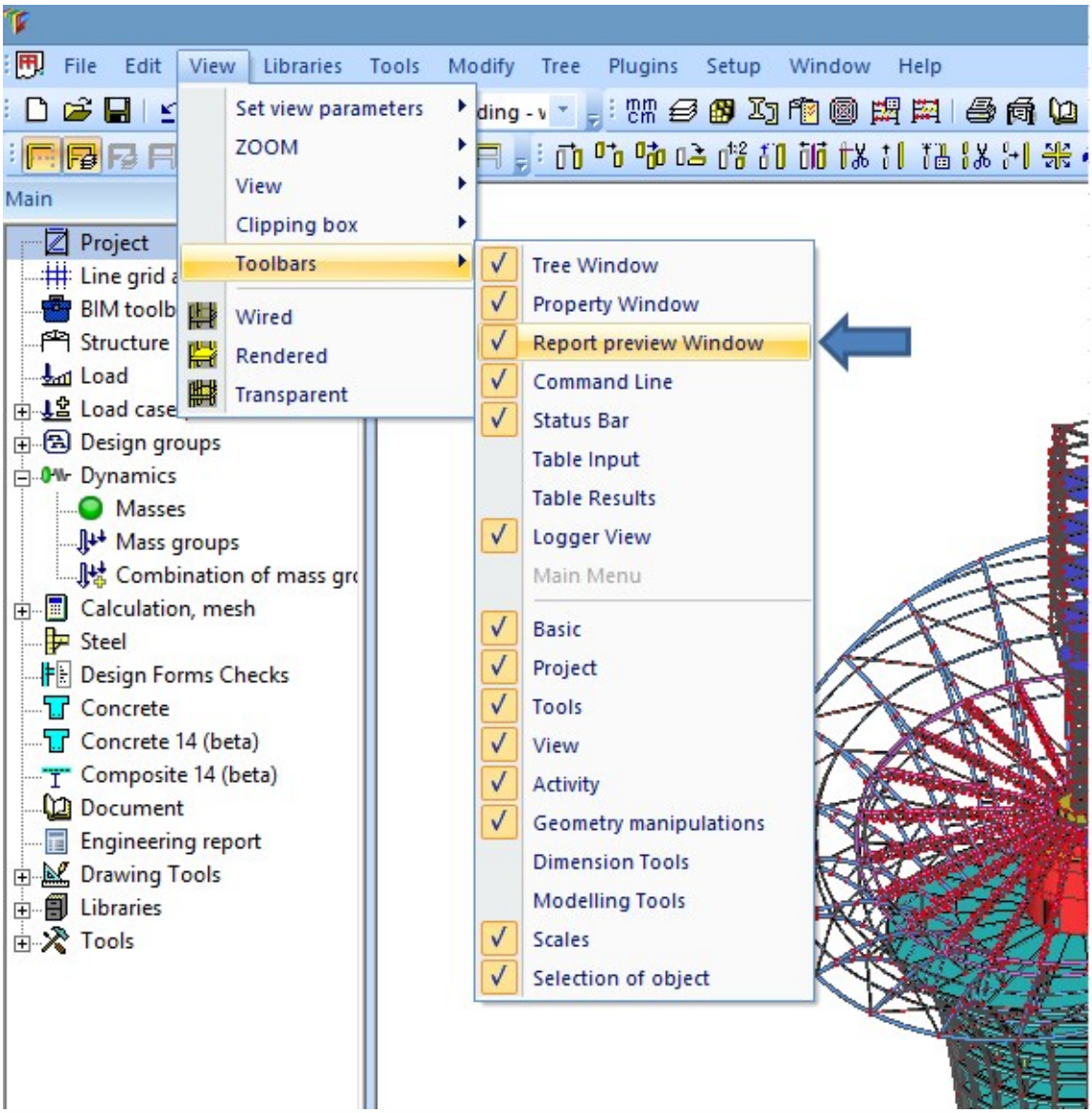


The procedure for opening Preview window for **result tables**

- Press the action button **Preview** in the results properties



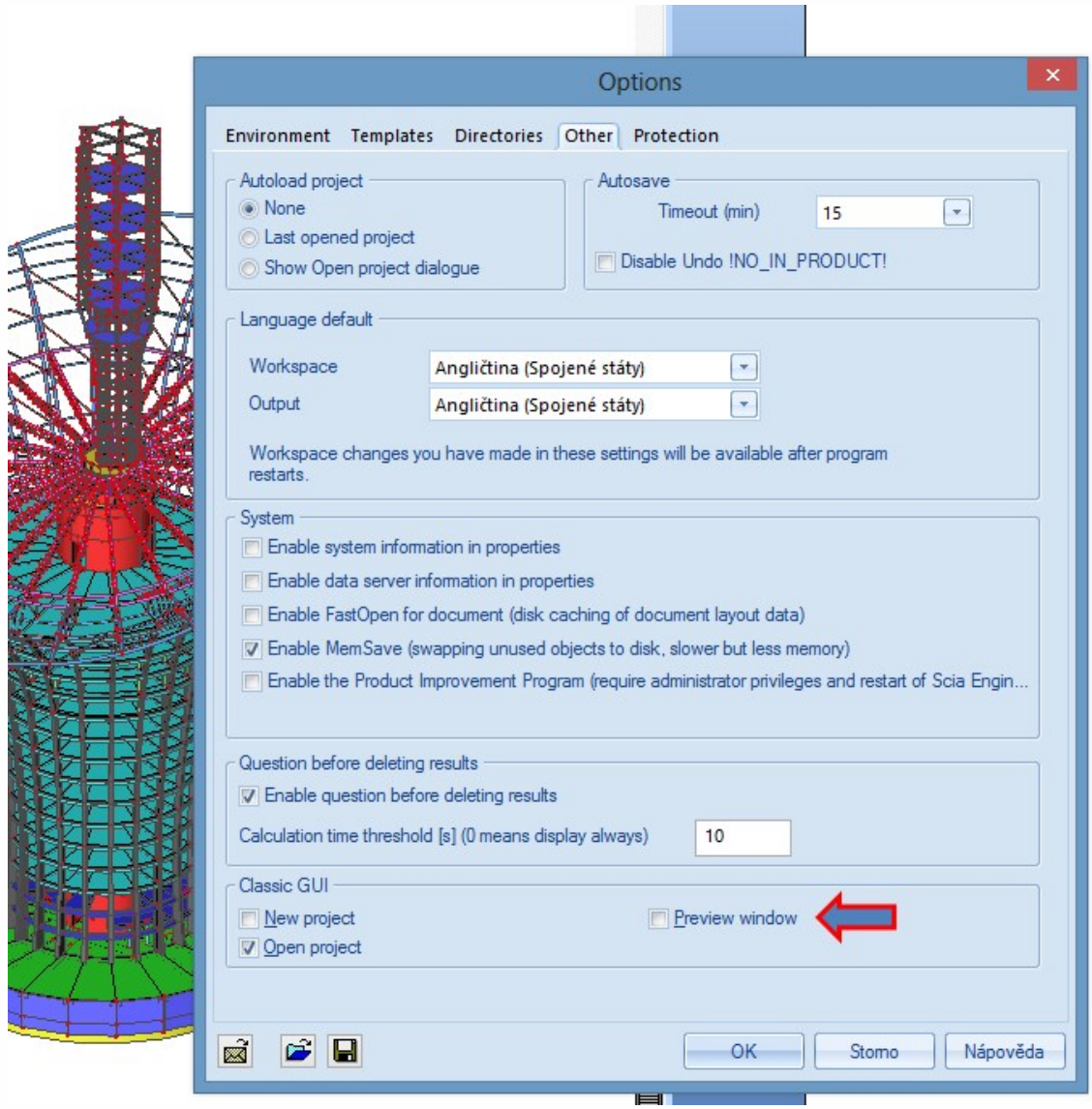
Preview window can be also opened from the menu View / Toolbars



Note: If the Preview window has been already opened, its content is replaced with the appropriate new tables.

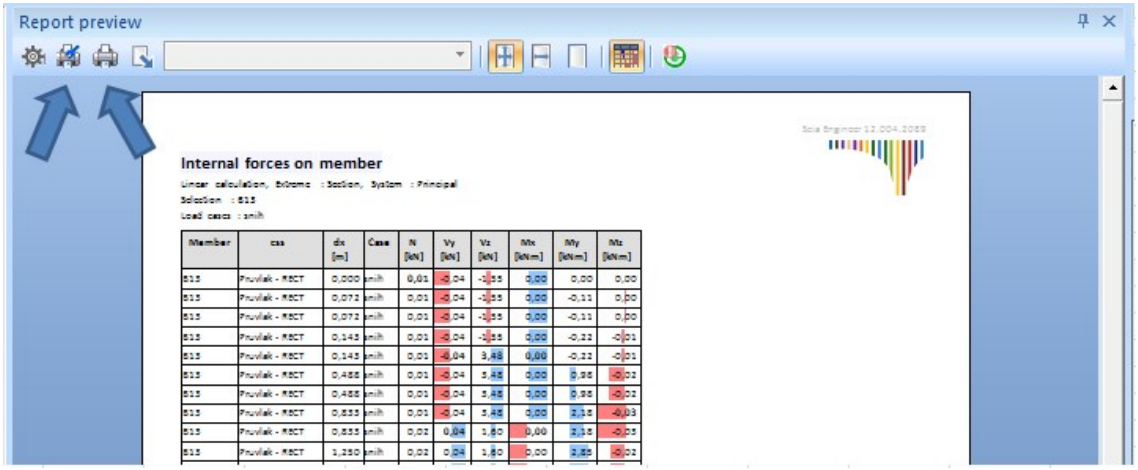
Switching between new on old preview window

Conservative users may still use old preview window. The switching can be done in Setup / Options / Other dialogue on the panel Classic GUI.



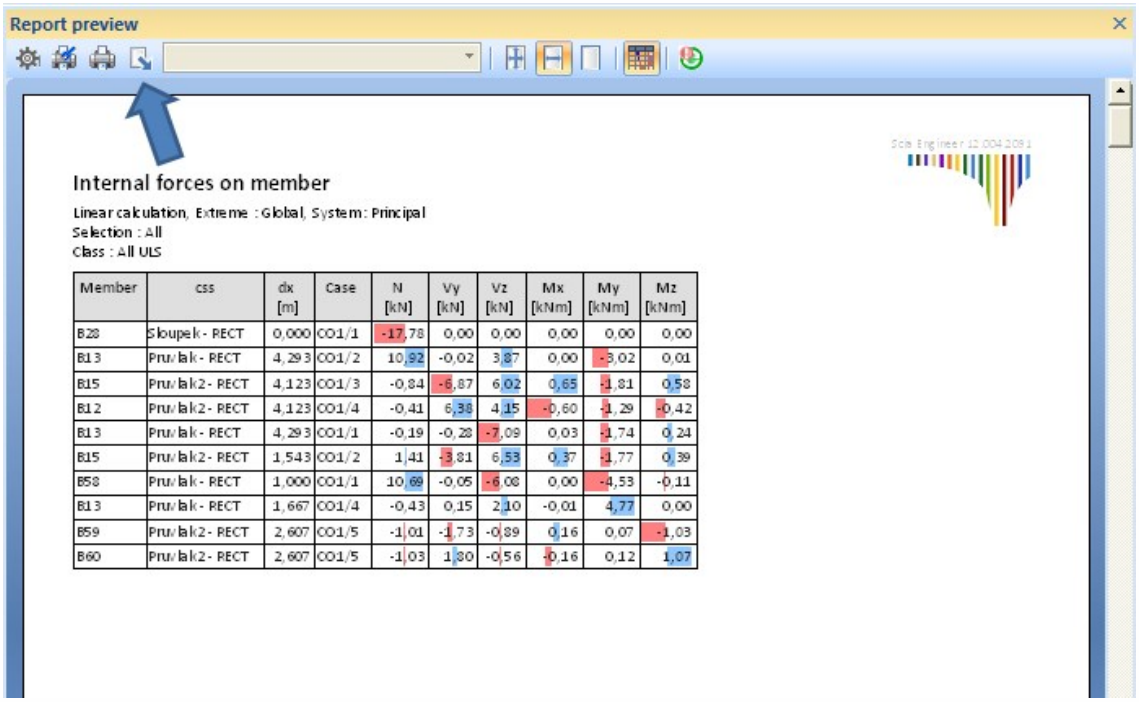
Printing the preview

Content of Preview window can be printed using two buttons. Left one opens print preview backstage view with possibility to modify printing parameters (those parameters are same as in Engineering Report). The right one start printing with the last used setting.



Exporting preview

Content of the Preview window can be exported using the Export button. There are the same possibilities as in exporting of Engineering Report.



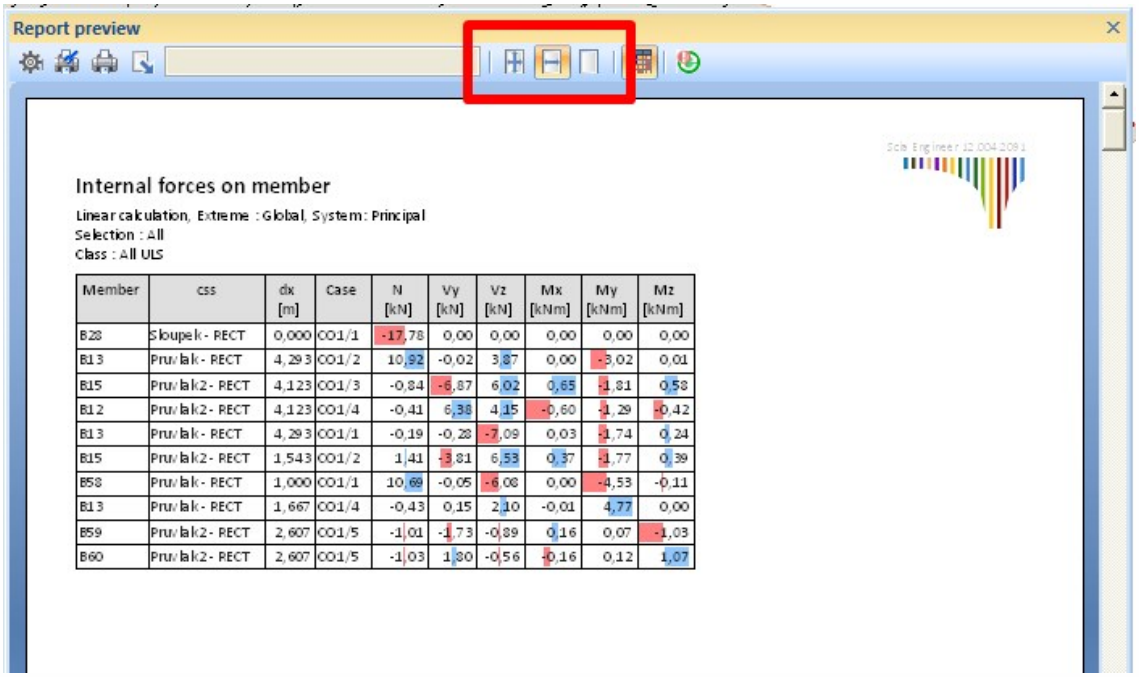
Member	css	dx [m]	Case	N [kN]	Vy [kN]	Vz [kN]	Mx [kNm]	My [kNm]	Mz [kNm]
B28	Sloupek - RECT	0,000	CO1/1	-17,78	0,00	0,00	0,00	0,00	0,00
B13	Pruvlak - RECT	4,293	CO1/2	10,92	-0,02	3,87	0,00	-3,02	0,01
B15	Pruvlak2 - RECT	4,123	CO1/3	-0,84	-6,87	6,02	0,65	-1,81	0,58
B12	Pruvlak2 - RECT	4,123	CO1/4	-0,41	6,38	4,15	-0,60	-1,29	-0,42
B13	Pruvlak - RECT	4,293	CO1/1	-0,19	-0,28	-7,09	0,03	-1,74	0,24
B15	Pruvlak2 - RECT	1,543	CO1/2	1,41	-3,81	6,53	0,37	-1,77	0,39
B58	Pruvlak - RECT	1,000	CO1/1	10,69	-0,05	-6,08	0,00	-4,53	-0,11
B13	Pruvlak - RECT	1,667	CO1/4	-0,43	0,15	2,10	-0,01	4,77	0,00
B59	Pruvlak2 - RECT	2,607	CO1/5	-1,01	-1,73	-0,89	0,16	0,07	-1,03
B60	Pruvlak2 - RECT	2,607	CO1/5	-1,03	1,80	-0,56	-0,16	0,12	1,07

Adjusting Preview window

Adjusting displaying

Zooming

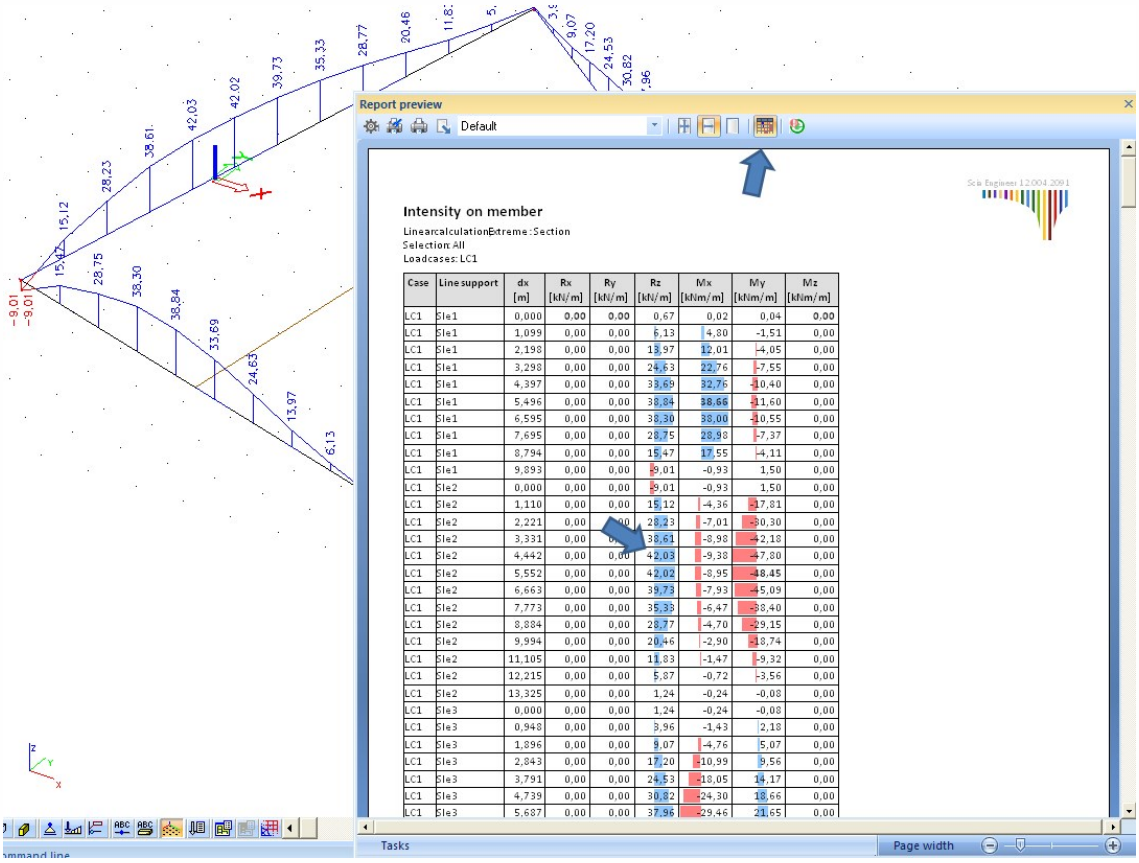
It is possible to select one of major zoom modes (fit to window, fit page width, 100%) using toolbar buttons.



Member	css	dx [m]	Case	N [kN]	Vy [kN]	Vz [kN]	Mx [kNm]	My [kNm]	Mz [kNm]
B28	Sloupek - RECT	0,000	CO1/1	-17,78	0,00	0,00	0,00	0,00	0,00
B13	Pruvlak - RECT	4,293	CO1/2	10,92	-0,02	3,87	0,00	-3,02	0,01
B15	Pruvlak2 - RECT	4,123	CO1/3	-0,84	-6,87	6,02	0,65	-1,81	0,58
B12	Pruvlak2 - RECT	4,123	CO1/4	-0,41	6,38	4,15	-0,60	-1,29	-0,42
B13	Pruvlak - RECT	4,293	CO1/1	-0,19	-0,28	-7,09	0,03	-1,74	0,24
B15	Pruvlak2 - RECT	1,543	CO1/2	1,41	-3,81	6,53	0,37	-1,77	0,39
B58	Pruvlak - RECT	1,000	CO1/1	10,69	-0,05	-6,08	0,00	-4,53	-0,11
B13	Pruvlak - RECT	1,667	CO1/4	-0,43	0,15	2,10	-0,01	4,77	0,00
B59	Pruvlak2 - RECT	2,607	CO1/5	-1,01	-1,73	-0,89	0,16	0,07	-1,03
B60	Pruvlak2 - RECT	2,607	CO1/5	-1,03	1,80	-0,56	-0,16	0,12	1,07

Graphical presentation of numerical values

Graphical presentation of numerical values can be switched ON/OFF using the toolbar button.



Case	Line support	dx [m]	Rx [kN/m]	Ry [kN/m]	Rz [kN/m]	Mx [kNm/m]	My [kNm/m]	Mz [kNm/m]
LC1	Sle1	0,000	0,00	0,00	0,67	0,02	0,04	0,00
LC1	Sle1	1,099	0,00	0,00	6,13	4,80	-1,51	0,00
LC1	Sle1	2,198	0,00	0,00	13,97	12,01	-4,05	0,00
LC1	Sle1	3,298	0,00	0,00	24,63	22,76	-7,55	0,00
LC1	Sle1	4,397	0,00	0,00	33,69	32,76	-10,40	0,00
LC1	Sle1	5,496	0,00	0,00	38,84	38,66	-11,60	0,00
LC1	Sle1	6,595	0,00	0,00	38,30	38,00	-10,55	0,00
LC1	Sle1	7,695	0,00	0,00	28,75	28,98	-7,37	0,00
LC1	Sle1	8,794	0,00	0,00	15,47	17,55	-4,11	0,00
LC1	Sle1	9,893	0,00	0,00	-9,01	-0,93	1,50	0,00
LC1	Sle2	0,000	0,00	0,00	-9,01	-0,93	1,50	0,00
LC1	Sle2	1,110	0,00	0,00	15,12	-4,36	-17,81	0,00
LC1	Sle2	2,221	0,00	0,00	28,23	-7,01	-30,30	0,00
LC1	Sle2	3,331	0,00	0,00	38,61	-8,98	-42,18	0,00
LC1	Sle2	4,442	0,00	0,00	42,03	-9,38	-47,80	0,00
LC1	Sle2	5,552	0,00	0,00	42,02	-8,95	-48,45	0,00
LC1	Sle2	6,663	0,00	0,00	39,73	-7,93	-45,09	0,00
LC1	Sle2	7,773	0,00	0,00	35,33	-6,47	-38,40	0,00
LC1	Sle2	8,884	0,00	0,00	28,77	-4,70	-29,15	0,00
LC1	Sle2	9,994	0,00	0,00	20,46	-2,90	-18,74	0,00
LC1	Sle2	11,105	0,00	0,00	11,83	-1,47	-9,32	0,00
LC1	Sle2	12,215	0,00	0,00	5,87	-0,72	-3,56	0,00
LC1	Sle2	13,325	0,00	0,00	1,24	-0,24	-0,08	0,00
LC1	Sle3	0,000	0,00	0,00	1,24	-0,24	-0,08	0,00
LC1	Sle3	0,948	0,00	0,00	3,96	-1,43	2,18	0,00
LC1	Sle3	1,896	0,00	0,00	9,07	-4,76	5,07	0,00
LC1	Sle3	2,843	0,00	0,00	17,20	-10,99	9,56	0,00
LC1	Sle3	3,791	0,00	0,00	24,53	-18,05	14,17	0,00
LC1	Sle3	4,739	0,00	0,00	30,82	-24,30	18,66	0,00
LC1	Sle3	5,687	0,00	0,00	37,96	-29,46	21,65	0,00

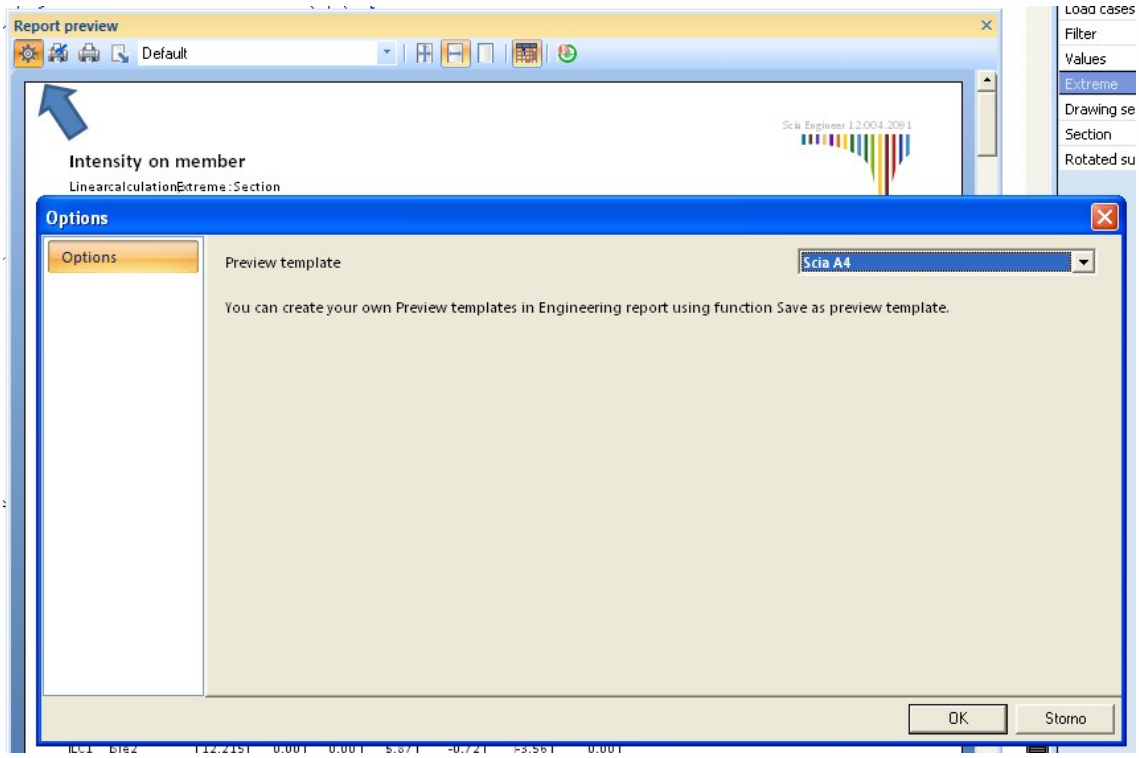
Preview template

Content of the preview window consists of the selected table (see the chapter Opening the Preview) and preview template. Preview template can contain all entities which can be inserted into Engineering Report. The content of Preview template is placed before the selected table.

The typical content of the Preview template is definition of Page layout, Page format and Visual style.

Selection of Preview template

The current preview template can be selected in the Option dialogue:

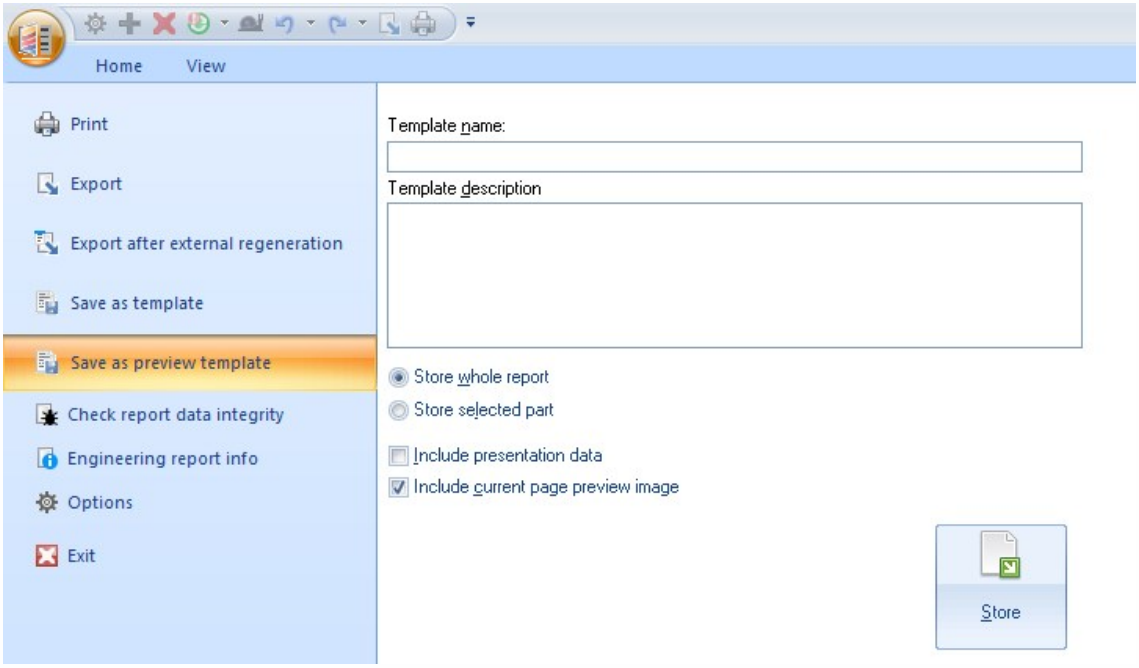


There are two predefined report templates in the installation of SCIA Engineer. The one with Letter page format for United states and the second one with A4 page format for the rest of the world.

Creating of new Preview template

Creating of Preview template is very easy:

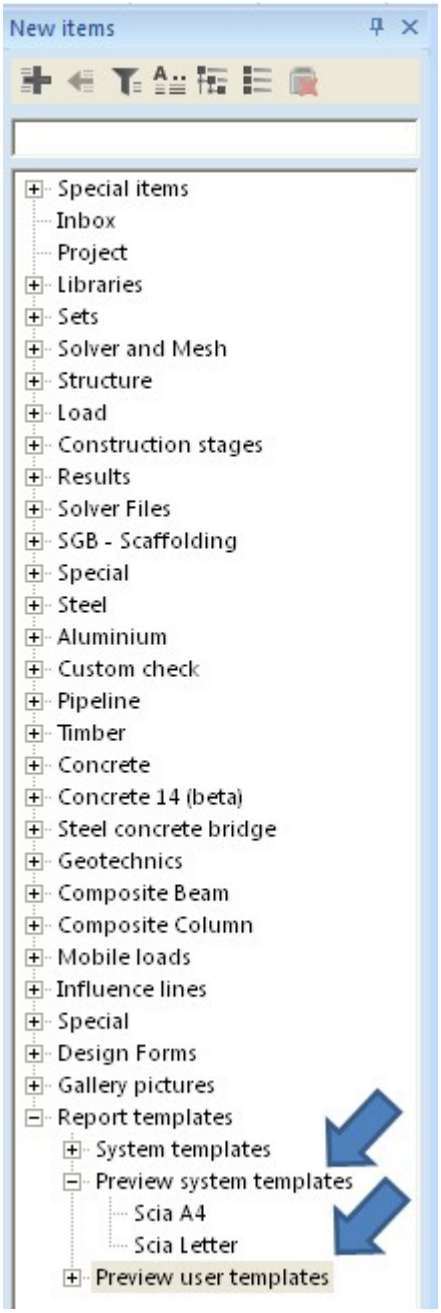
1. Define its content in the Engineering Report.
2. Store it using the **Save as preview template** command.



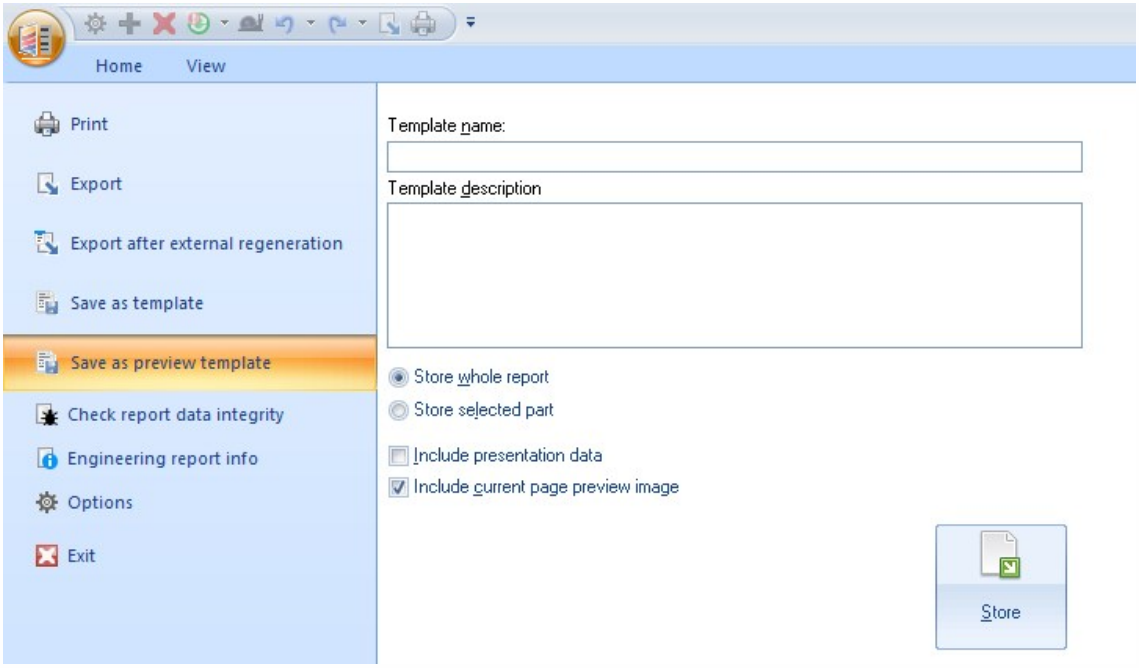
Editing of Preview templates

For editing of existing Preview templates follow those steps:

1. Insert content of existing Preview template in Engineering Report. Preview templates are listed in the list of new items



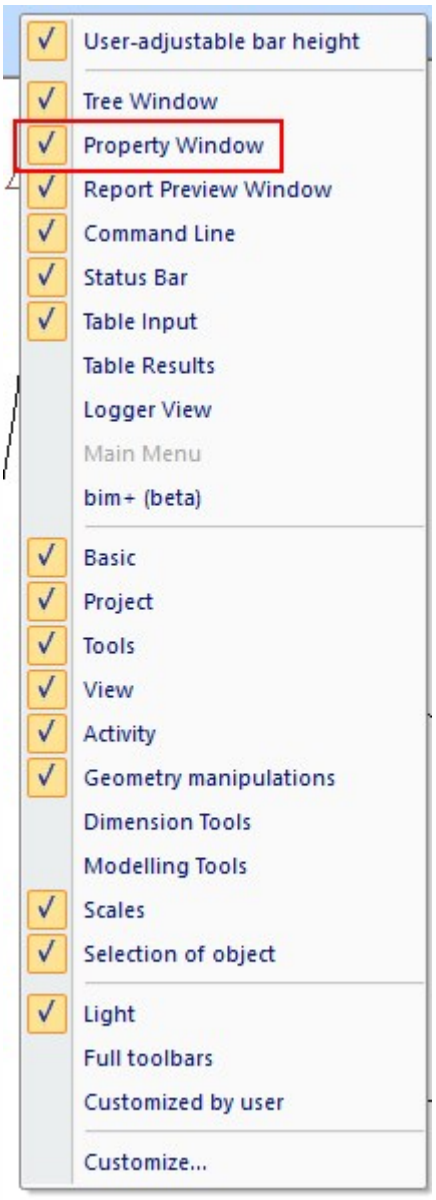
2. Modify the content of the report
3. Store it using the **Save as preview template** command

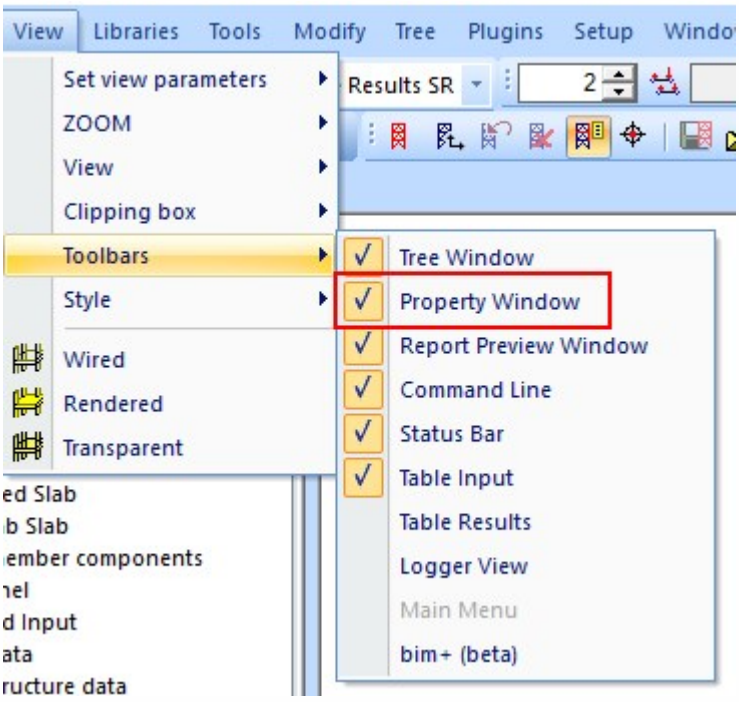


Property window

Property window

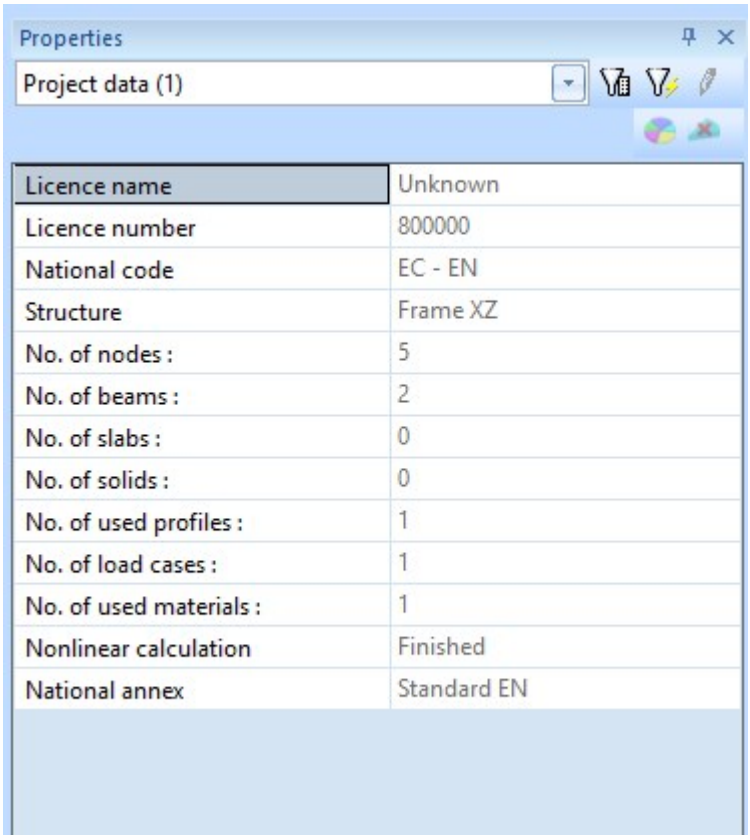
Property window can be easily opened or closed via right-click menu in TB area, or from View menu.


User-adjustable bar height
Tree Window
Property Window
Report Preview Window
Command Line
Status Bar
Table Input
Table Results
Logger View
Main Menu
bim+ (beta)
Basic
Project
Tools
View
Activity
Geometry manipulations
Dimension Tools
Modelling Tools
Scales
Selection of object
Light
Full toolbars
Customized by user
Customize...


View
Libraries
Tools
Modify
Tree
Plugins
Setup
Set view parameters
ZOOM
View
Clipping box
Toolbars
Style
Wired
Rendered
Transparent
Results SR
2
Tree Window
Property Window
Report Preview Window
Command Line
Status Bar
Table Input
Table Results
Logger View
Main Menu
bim+ (beta)

The property window has its name derived from a property table that is displayed in it. The property window summarises parameters, characteristics and selected options of particular entities such as nodes, 1D members, loads, result diagrams, etc.

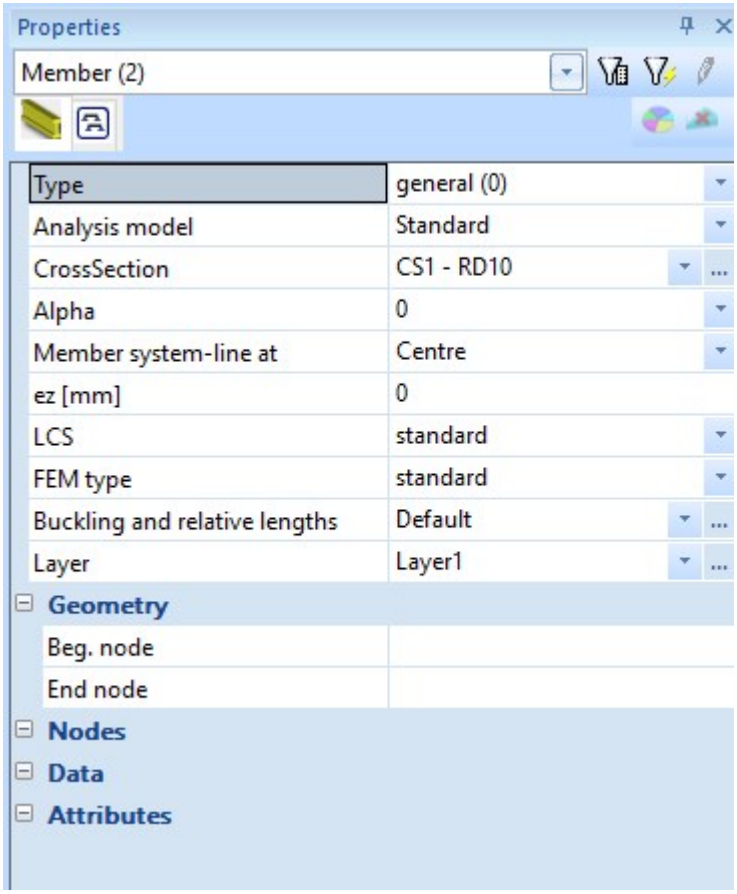
Example of a property window



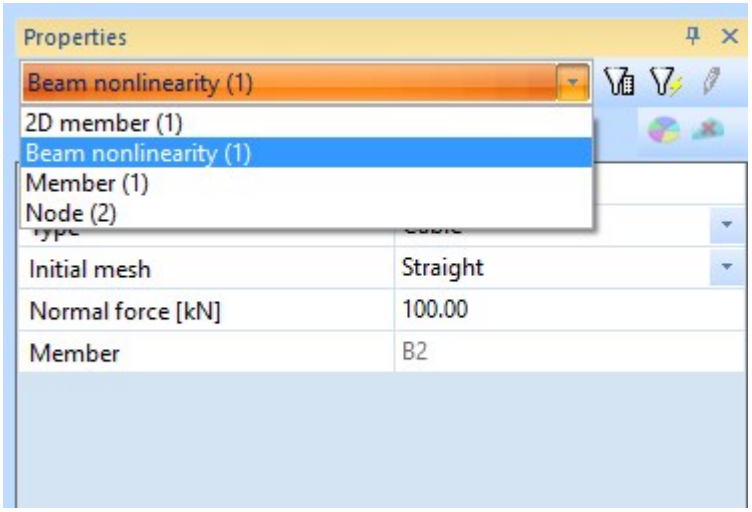
The property window always shows information related to the selected entities or selected function. However, the property window has been designed to not only passively display the properties, but also to provide for fast and easy modification of them.

If the current selection consists of only one entity, generally all the parameters can be modified. If more than one entity has been selected, the property window automatically applies a filter and displays the parameters that the selected entities have in common.

Example of a property with common properties in one selection

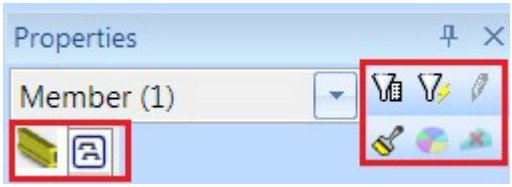


Example of switch between element type in property tree



If a function has been started, the property window may contain some switches that may affect the behaviour of the function. Most of the functions from service Results are good examples as the property window enables the user to select required quantity to-be-displayed, adjust the style of result diagrams, etc.

Property window contains some additional functions, accessed from header



Select elements by more properties	You can select elements based on multiple elements. You will be asked for "master" element first, then dialog with properties will appear. You can select one or more (CTRL+click) propeties. All elements based on selected properties will be added to your selection.
Select elements by property	Will select all elements of same type (beam, slab, line load, etc.) which shares selected property value in property grid. Only one property can be selected.
Update property dialog	Update property dialog
Member	Switch to entity properties tab
Design group	Switch to entity design group
Format painter tool	Start format painter tool
Attributes editor	Opens attribute editors
Delete selected attribute	Delete attribute

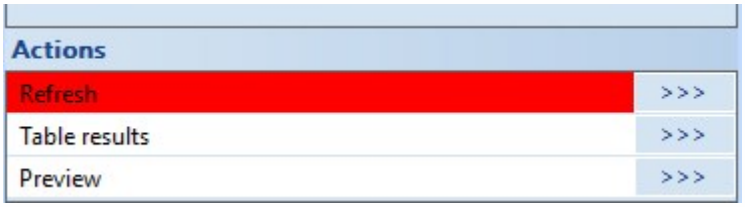
Action buttons

As the name suggests, the Property Table comprises properties of a particular part of a structure model. Sometimes however, the property table contains also a control that starts a particular action related to the element whose properties are displayed in the table.

If such controls (buttons in particular) are put somewhere inside the table, they may be overlooked. Therefore, these buttons were "extracted" from the table and are located in a special section called Actions with Action buttonsin it.

Thus, all the actions that are accessible for the current properties or for the "property-owner" are visibly and clearly separated from the often long list of information and can be easily accessed.

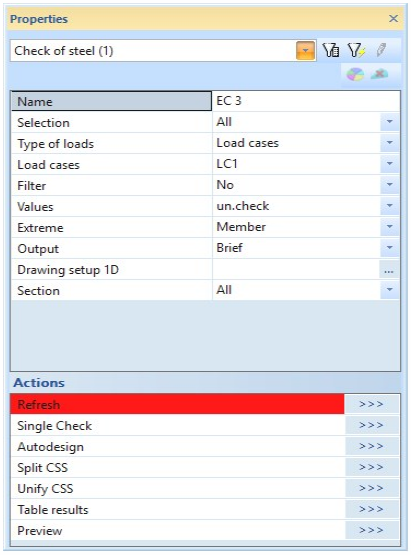
Sometimes action buttons are highlighted in red. This means that some changes which leads to different presentation was done (ex.: changing parameters for Results)

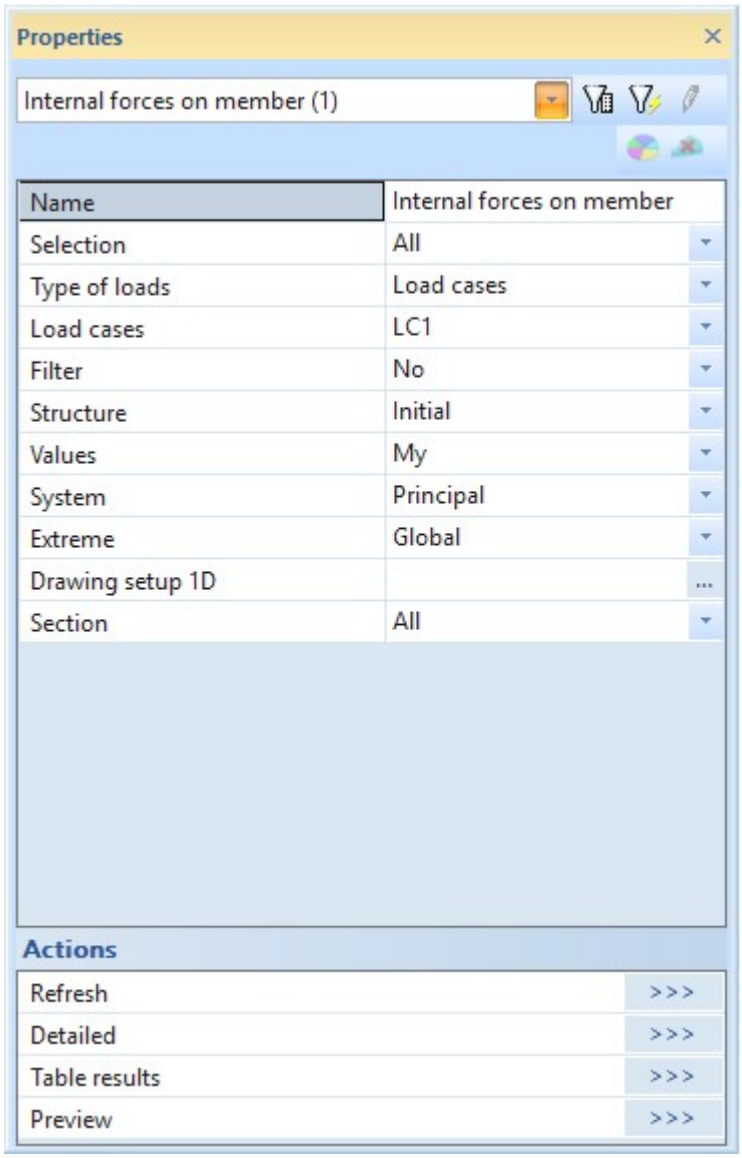


Action buttons are used in various parts of SCIA Engineer.

Action buttons in the Property Window

The table below presents some (not all) applications of Action buttons.

service Steel > function Check	
Refresh	It redraws the screen in order to reflect the changes made in the Property Window (see also Refresh of results). Default keyboard shortcut - F5.
Single check	It opens a dialogue that provides for checking of a single selected 1D member.
AutoDesign	It opens a dialogue for the AutoDesign of selected 1D members.
Split CSS	Splits the cross section
Unify CSS	Unify the cross section
Table results	It opens the Table results and create new tab with results.
Preview	It opens the Preview window and displays the relevant information in it.

service Results > function Internal forces	
Refresh	It redraws the screen in order to reflect the changes made in the Property Window (see also Refresh of results). Default keyboard shortcut - F5.
Detailed	It will draw detailed result of selected member in 3d window.
Table results	It opens the Table results and create new tab with results.
Preview	It opens the Preview window and displays the relevant information in it.

Action buttons in Database managers

Action buttons are used for example in the Combinations dialog.

See more in chapter Exploded combinations.

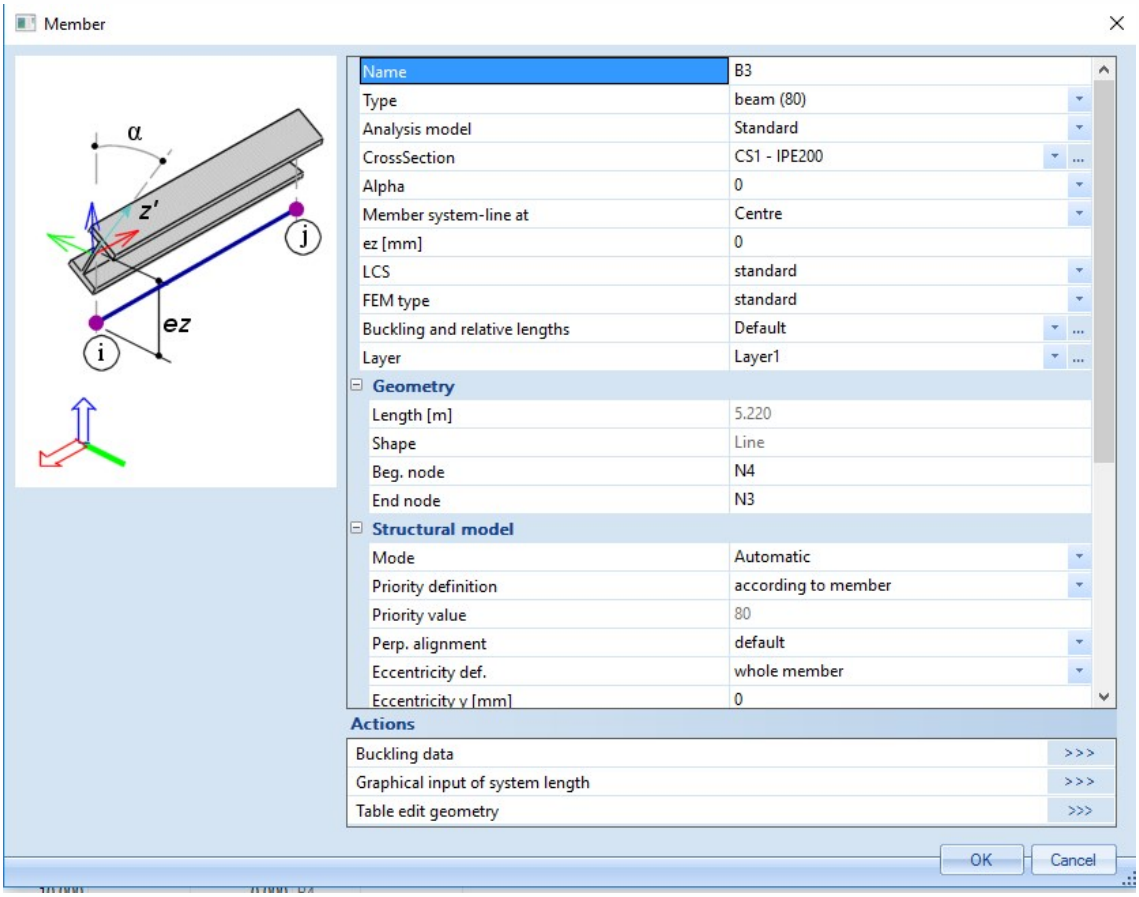
Modal property window

Properties can be displayed (and modified) in two modes.

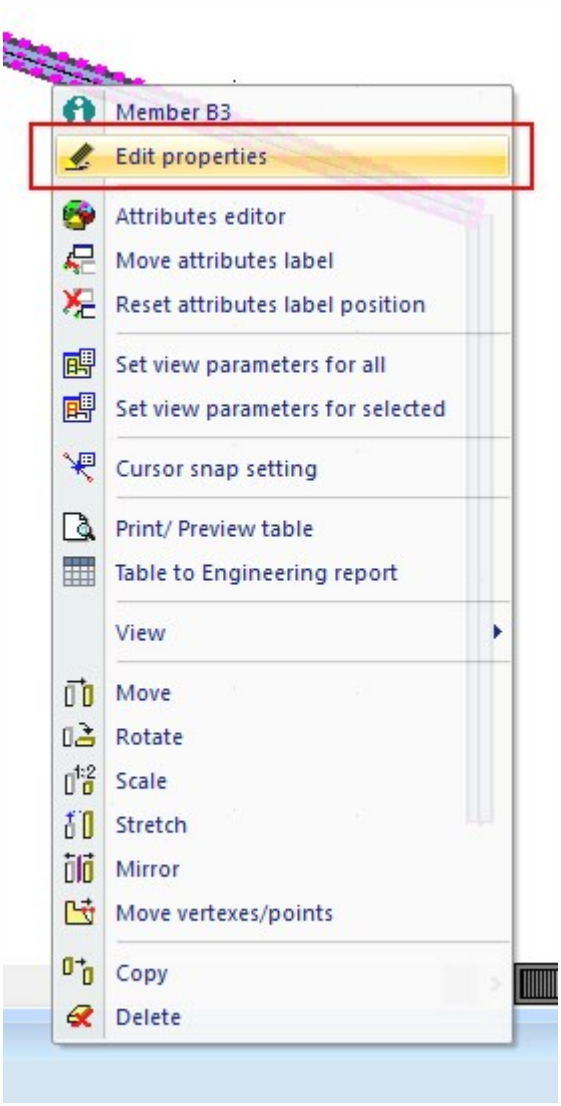
The first mode is standard Property window.

Second possibility is to use Modal property window.

The standard property windows displays properties of selected entities but the modal property window display property of just one entity.



The modal property window can be opened via pop-up menu when cursor is over some entity.



Second possibility (since version 14) how to open the modal property dialogue is using mouse double-click on some entity. One of those two actions opens modal property dialogue with properties of the entity.

Opening of modal property window via double click works only in case of Single selection mode. In case of Multi selection mode is necessary to press combination of keys **Ctrl+Shift together with double click**.

Database managers

Introduction to database manager

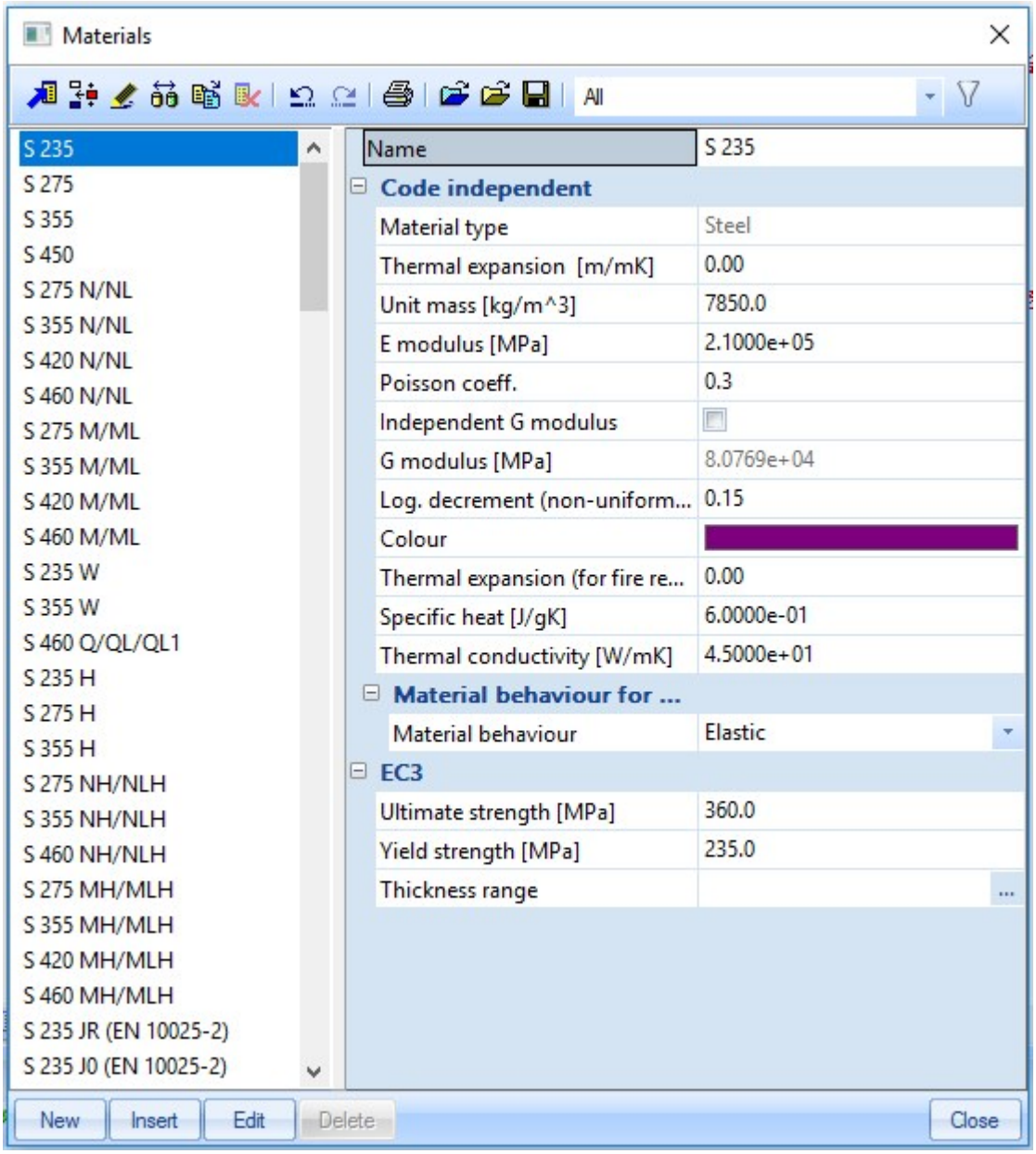
A database manager is a tool that provides for all possible operations related to manipulation with entities stored in some of program databases. The term "program database" stands e.g. for a database of materials, cross-sections, catalogue blocks, etc. defined in a current project.

It is obvious that:

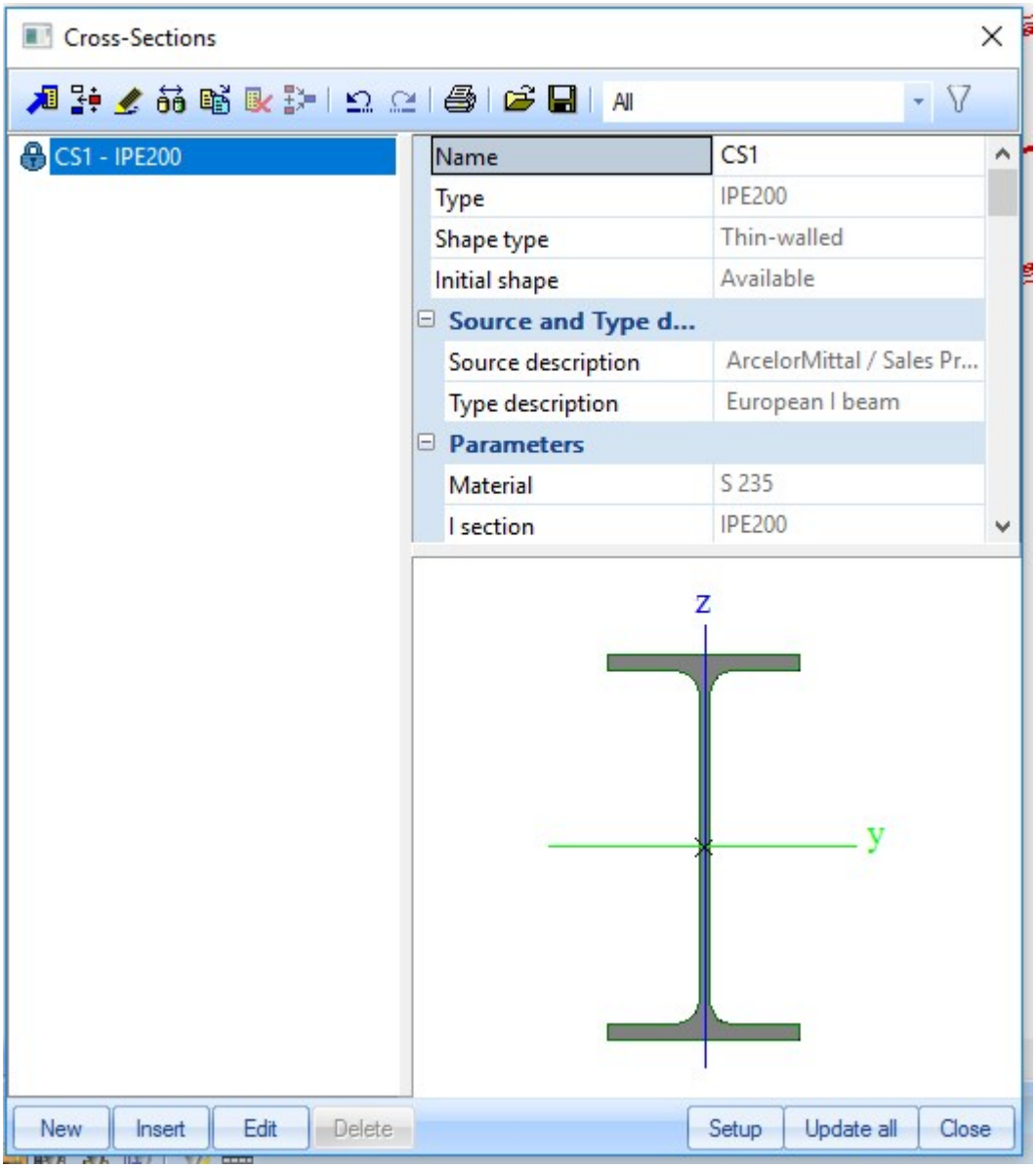
- individual entities of these databases must be somehow defined,
- there must be a way to edit them, copy them, delete them,
- the user must have an opportunity to review parameters of the individual entities,
- there must exist a procedure to select one entity as a "default" for functions requiring an entity of that type as an input parameter,
- the approach to all these points must be unique regardless the type of database.

Examples of database dialogs:

Material



Cross sections



Layout and operation of a database manager

A manager consists basically of the following controls:

List of defined entities of a particular database	The list shows all the entities related to the database of the manager that have been defined in the current project so far.
Property table	This table shows a brief summary of parameters for the database entity that is just selected in the list of already defined entities (see above).
Graphical window	This window displays a drawing of the database entity whose parameters are just listed in the property table.
Control buttons	The buttons provide the access to the functions that are accessible from within the particular manager.
Filter	The filter allows for a readable representation of data in the Manager.

List of defined database entities

The list summarises all the database entities that has been defined in the project. Most often, the list contains names of the entities. However, if useful and practical, some additional information may be added next to the name.

Property table

The property table displays parameters for the entity that is selected in the list of defined entities. It provides for a quick review of the parameter values. Some of the parameters can also be edited here. But normally, the modification of the parameters is performed in the editing dialogue for a particular entity type.

Graphical window

This window contains a schematic drawing of the database entity the parameters of which are presented in the property table. This window is fitted with a pop-up menu. The menu offers the user some important functions related to the displayed entity.

Control buttons

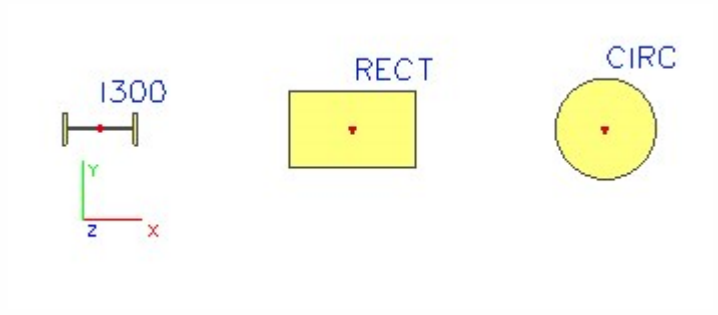
There are several control buttons in the Manager that allow to user to use various actions that may be performed with database entities.

button		meaning
[New]		This button opens the New entity dialogue where a new entity can be defined and inserted into the current project. The newly defined entity is inserted at the end of the list of defined entities.
[Insert]		This button also opens the New entity dialogue where a new entity can be defined and inserted into the current project. But, the newly defined entity is inserted before the currently selected entity in the list of defined entities. This feature can be used to have the entities in user-defined order and not in the order of insertion.
[Edit]		This button opens the Editing dialogue for the entity currently selected in the List of defined entities. The Editing dialogue provides for thorough and detailed review or editing of the entity parameters.
[Delete]		This button allows the user to get rid of those entities of the particular database that are no longer necessary in the project.
[Copy]		The Copy button makes a copy of the entity that is selected in the List of defined entities.
[System database]		It enables the user to read items from a standard system database.
[Read]		It enables the user to read database items from an external file – user's database.
[Save]		It saves selected entities of the database to an external file – user's database.
[Text Output]		This button opens the preview window and displays all the parameters in it for the entity that is selected in the List of defined entities.
[Close]		This button has got two functions. First, it sets the currently highlighted item in the List of defined entities as the active (or current) entity. Second, it closes the database manager.

[Unify]		This button enables the user to select items from the list of defined items that will be united with the currently selected item. Thus it is possible to get rid of excessive number of doubled items, or to establish a single item for entities that originally used several items (e.g. to assign one cross-section to 1D members that originally had different cross-sections). See Example below.

Example - function Unify

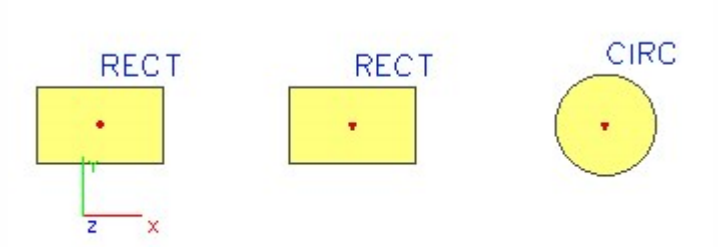
Let us suppose that we have defined three beams, each of them of a different cross-section.



Later you may want to unify the section of the two left beams and have both of them of rectangular cross-section. Of course, you may edit the properties of the beam and change its cross-section. On the other hand, sometimes it may be useful to "unify" the sections (and if required, get rid of the abandoned cross-section type, that can be automatically deleted from the database).

You call the Unify function to merge two cross-sections into one. In our example do the following:

1. select the rectangular cross-section,
2. call function Unify,
3. select the I section,
4. confirm with OK,
5. the I-section is removed from the project database, two beams are assigned the same rectangular cross-section.



Filter

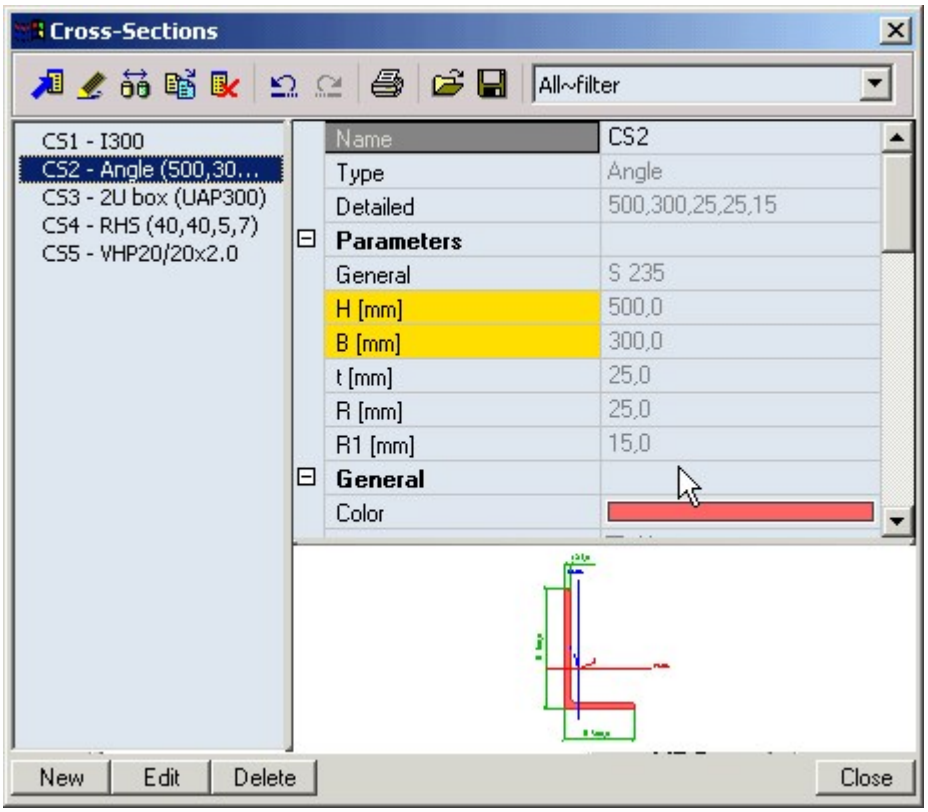
The filter provides for more readable representation of data in the Manager if the current project contains an excessive number of defined entities of the particular type. The filter allows the user to set a limited set of entities that are displayed in the List of defined entities. The entities that do not meet the chosen criterion are "removed" from the list, but still remain normally defined in the project.

Note: Some specific database managers may contain additional functionality. It is added in the form of additional control buttons.

Name

Note: The name of any item in any manager should be up to 8 characters in length. Longer names should not be used and may be truncated by the program.

Example of a database manager



Opening the database manager

A Manager is opened whenever the appropriate function is activated. E.g. function Library > Cross-sections opens the Cross-section manager, etc.

In general, the particular manager is also opened when a general procedure for the definition of a new database entity is invoked. In such a case, the opening of the manager is usually one of the first steps of the procedure.

From the user's point of view a database manager is a standard Windows modal dialogue. That means that:

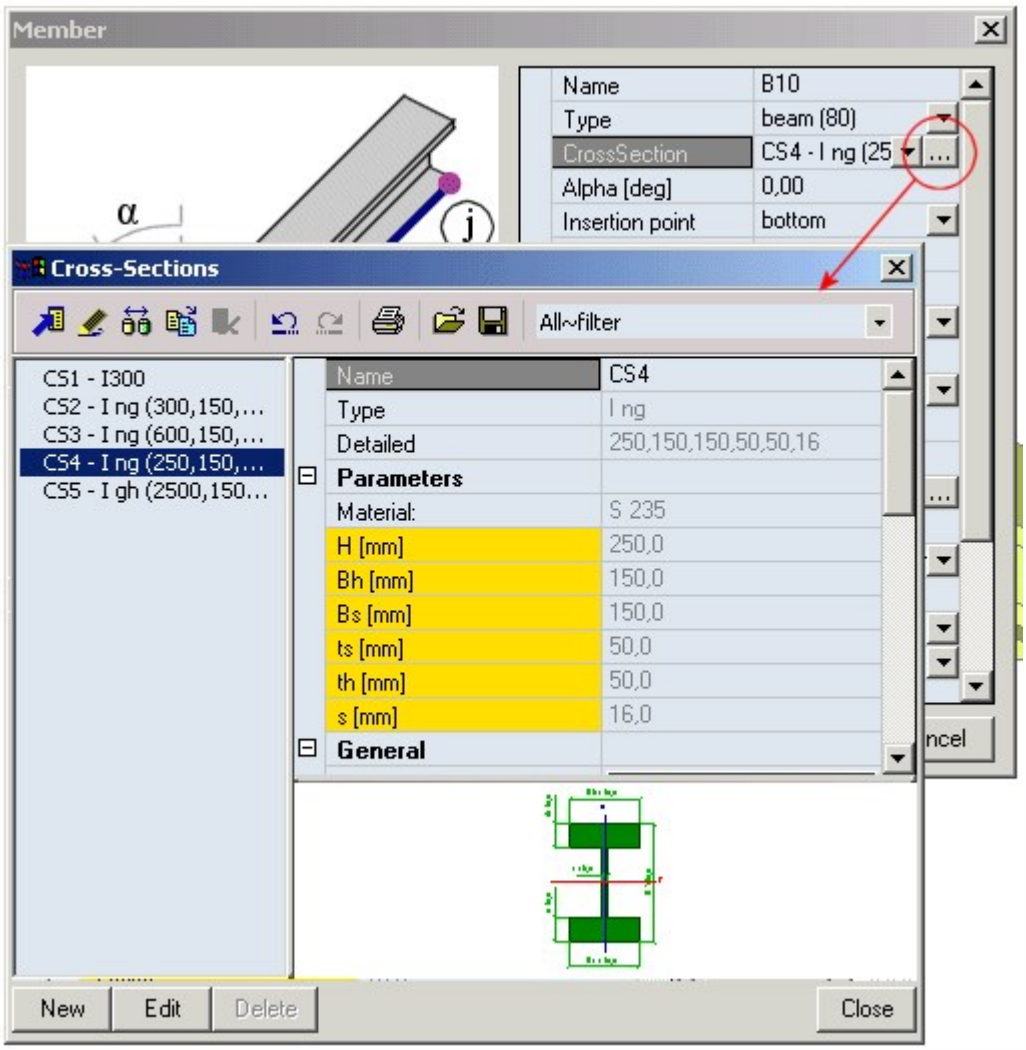
- it is opened via a function associated with it,
- it must be closed before the user can continue with the started multi-step action of before another function can be activated.
- it contains control elements that provide for actions and tasks that are accessible from within the manager.

The operation is simple and straightforward and is clear from the description of layout of a database manager.

Note: The Particular manager can also be opened from various property dialogues that contain an item associated with the particular database manager Such an item contains a button to open the appropriate manager.

Example:

A cross-section manager opened from within the property dialogue of a new 1D member.



Pop-up menu of database manager

The graphical window of a database manager is equipped with a pop-up menu that summarises some important functions.

Zoom rectangle	The user may define the cut-out that should fit into the graphical window.
Zoom all	This option zooms the drawing in or out so that the whole drawing fits the available window area.
Gallery	It copies the drawing into the Picture gallery.
Document	It copies the drawing into the Document.
Print	This function prints the drawing on the connected graphical device.

Copy to clipboard	It copies the drawing into the Windows clipboard.
Copy to BMP file	It saves the drawing into a Windows Bitmap file.
Copy to WMF file	It saves the drawing into a Windows Meta File.

Table input

Introduction

The **Table input** is a tool for the simple editing and creating the model in SCIA Engineer 2011 or higher. The main functionality of the Table input is based on a grid system (e.g. Excel, Open office …). User sees all elements in rows and its properties in columns, so the table is more transparent then the property window.

> Table input shows all properties even for the multi-selection. Each item is displayed on a separate row so it wont hide different properties in one cell like the property window.

The table represents the 3D model in numbers, checkboxes, comboboxes, libraries and strings. It allows editing, creating and deleting items without graphical searching.

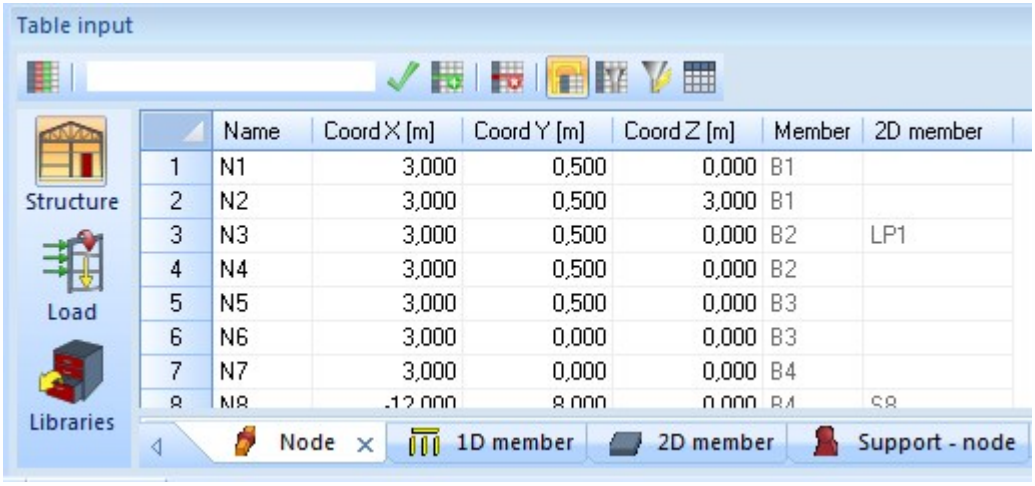
It allows creating new items in structure, libraries, loads and it allows to make modifications.

The filtering row, sort, multiple editbox and activity make the usage very easy and fast.

There are even some use-cases which are difficult of even impossible without Table input - quick renaming more items, multiply all forces by the same number, etc.

Tabs

The table contains tabs for a different types of the structure. All tabs are sorted to 3 groups - Structure, Load and Libraries.



> One tab for 1D members another tab for point support on point and so on.

Each tab shows the icon of the item and the cross button for simple hiding the tab.

Each tab is divided to rows, columns and cells:

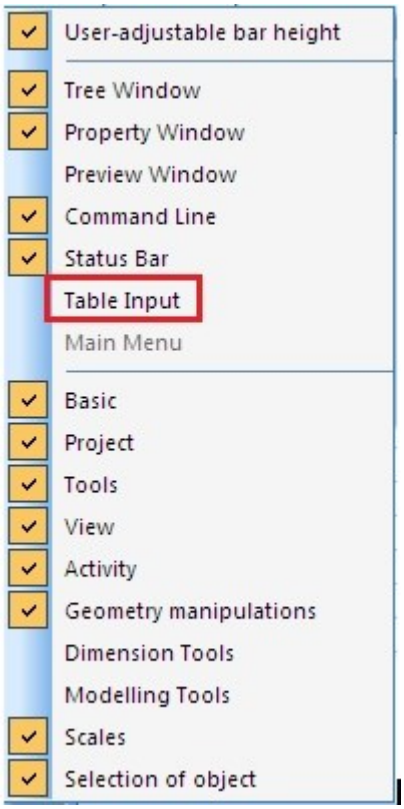
- The row represents one member – node, beam, plate, support, layer …
- The column represents one property of a member – height, FEM mode, colour, name …
- The cell represents one property value – 3m, layer 1, wall (80) …

Cell types

- **Number** – a simple value which can be changed by direct editing or pasting another value
- **String** – a simple value – name – which can be changed by direct editing or pasting another string
- **Combobox** – contains a list of values, change can be done by selecting another value from list or by pasting another value from this list
- **Link button** – this cell is linked to the library, change can be done by selecting another value in the library or pasting another value from the library (There is a Link button  in the cell always displayed with combobox and it has the same function)
- **Checkbox** – two values – yes/no, it can be pasted as 1 or 0

Tabs - default settings, order

The Table input is a dockable window in SCIA Engineer. It is displayed by the context menu on the toolbar (similar to command line, etc.).



Default settings of the Table input

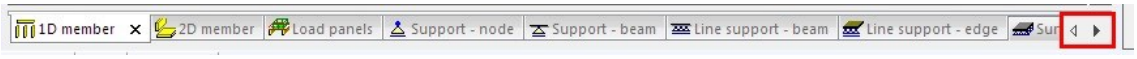
Each tab has a default settings predefined . The order of the visible columns and the width are loaded when the application starts. The last used layout is loaded when SCIA Engineer is reopened.

The last defined layout is saved to the folder User as TableInputSettings.xml.

When it is deleted, the default order is loaded to SCIA Engineer. Another possibility is to use the Reset GUI button. Go to Setup / Options / Environment / **Reset GUI**. The restart of the application is required.

Each tab shows the appropriate icon for the element.

If there is not enough space for all tabs then arrows for moving are displayed.

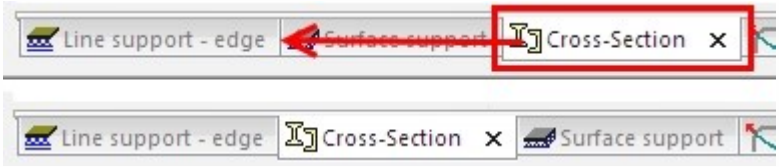


Tabs order and visibility

Each tab can be closed by the cross on it.

If only one tab is displayed, it cannot be closed.

Tabs can be reordered by drag and drop.

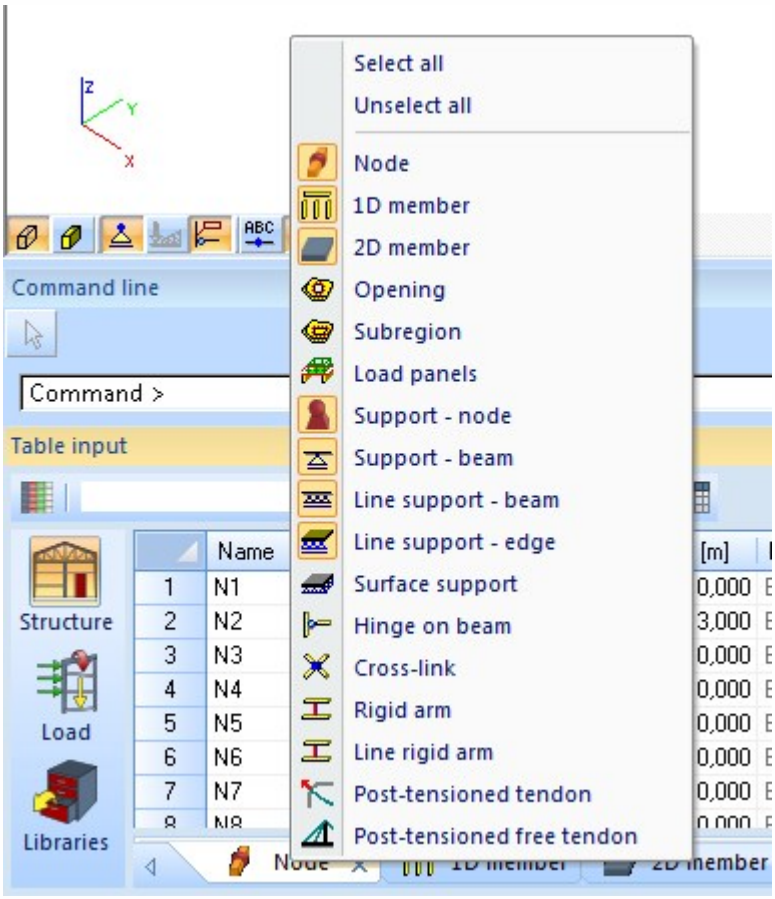


Tab manager

The Table input displays three different groups of items. There are structure, libraries and loads. The context menu on the tab displays all items in one group. Each item may be added as a visible tab or it can be hidden - this is displayed as selected or non-selected icon in the context menu.

Line support - edge is a visible tab;
Surface support is a hidden tab;

The icons in the Tab manager match with icons used in the Tree menu.



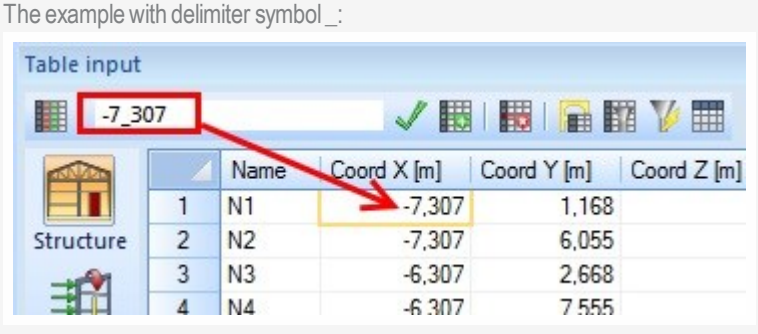
Loading content to the tab

The content of the tab is loaded after switching to it. It can take some time for a larger projects. This loading can be paused by the button on the progress bar.

When the loading is stopped then only a part of the content is displayed. This content can be sorted or filtered. Also pre-defined sort of filter can be used.

Delimiter

The required delimiter for numbers is managed by the operation system. It can be changed to any symbol.



In this case SCIA Engineer accepts 3_555 as a number and convert it to the table with delimiter symbol coma.

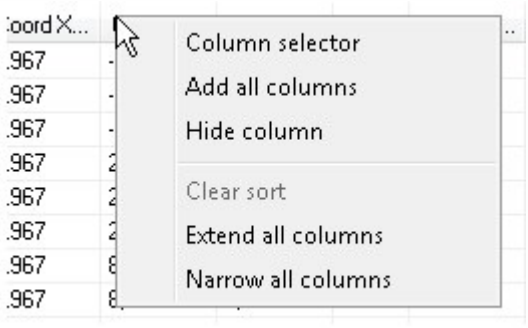
Tools in Table input - work with columns

Context menu

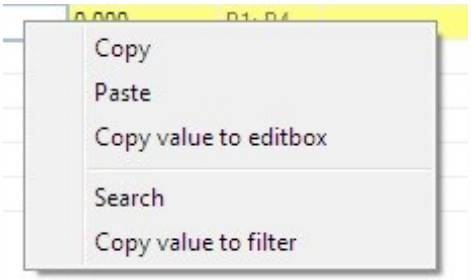
Table input provides more types of the context menu. The first is activated from the column header and the second is activated from the cell.

The special context menu is used on the toolbar edit-box.

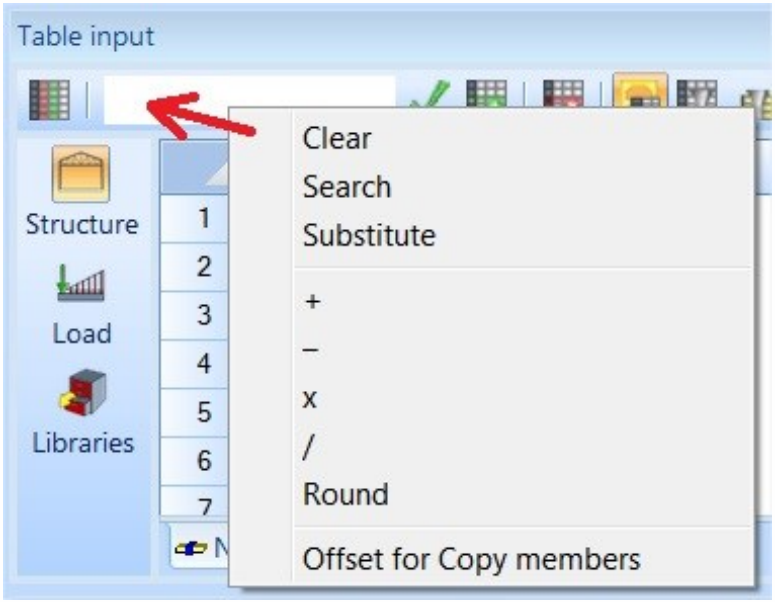
From the column header:



From the cell:



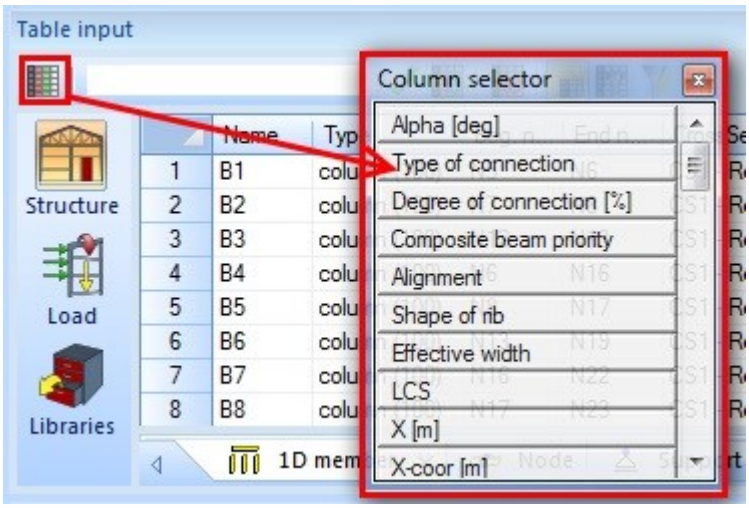
From the edit-box:

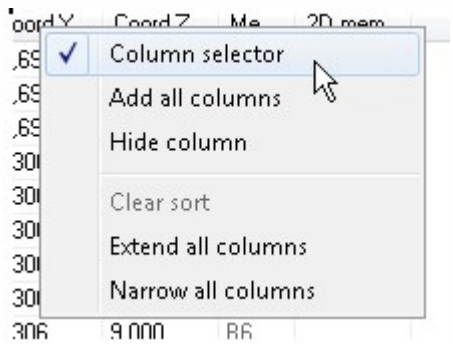


Column selector and work with columns

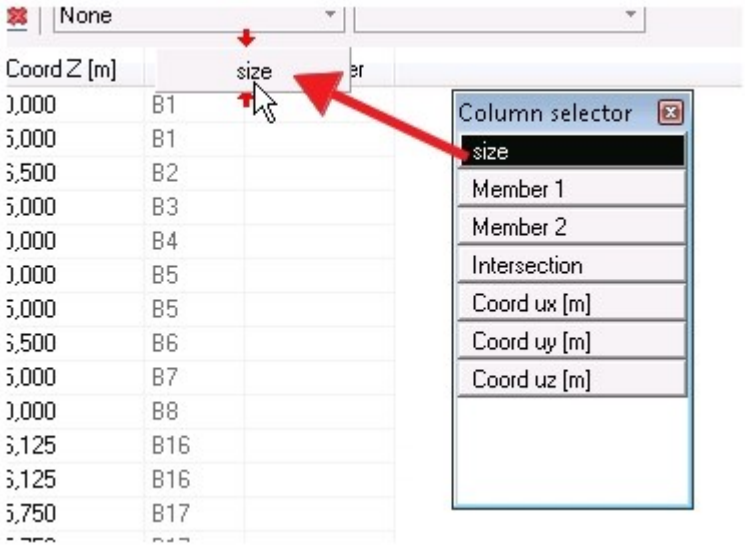
Column selector is a tool for displaying any column with property in the table.

It can be displayed by the button on the toolbar (the first picture) or by the context menu – item Column selector (see the second picture).





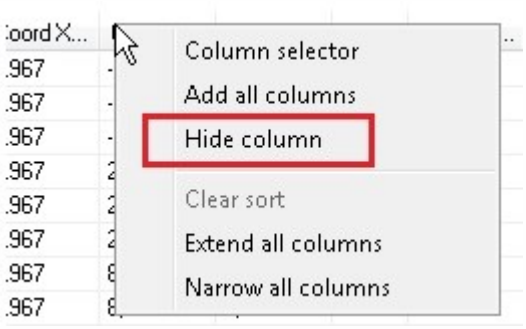
The selected columns with properties can be drag&droped to the Table input and stay here as visible. The property can be dropped between columns when two red arrows are displayed.



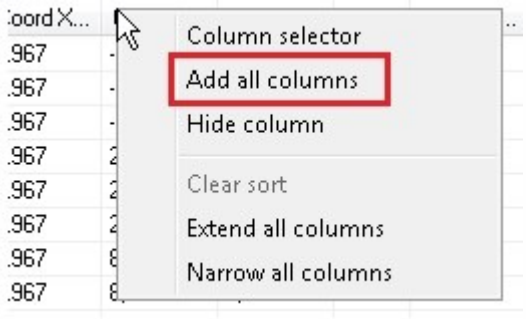
And the result:

The column can be hidden by drag&drop back to the Column selector or by drag&drop to the grid (then it is automatically moved back to the Column selector).

Another way is to use context menu – Hide column.

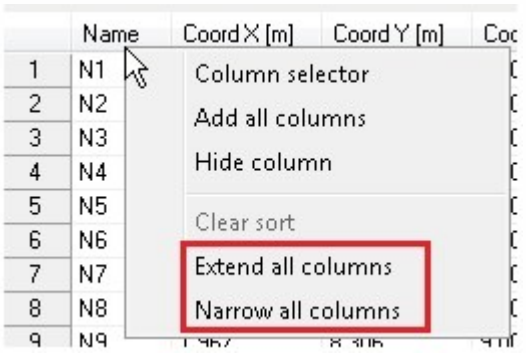


The context menu contains possibility to add all columns at once.



Extend and narrow all visible columns

It is possible to extend and narrow all visible columns at once using context menu on the column header – Extend all columns, Narrow all columns



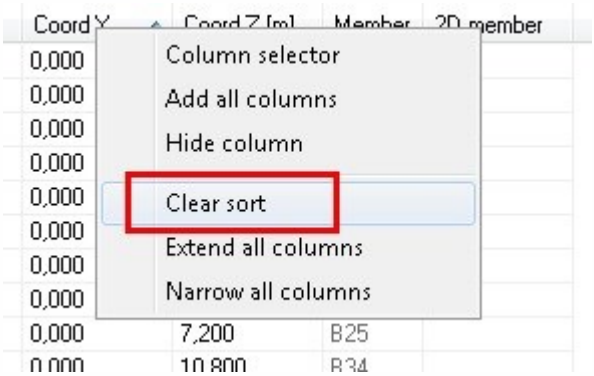
The values are extended or narrowed by approximately 20% of its original size.

> The reason is that the size of a text is managed by the operation system, not SCIA Engineer so this is a fast tool to correct possible problems with partly-visible headers.

Sorting

Each column can be sorted forward or backward. This functionality is activated by the click on the column header. The sorted column is displayed with the little black triangle.

The sort can be cleared by the 3rd click on the column header or by the context menu – Clear sort.

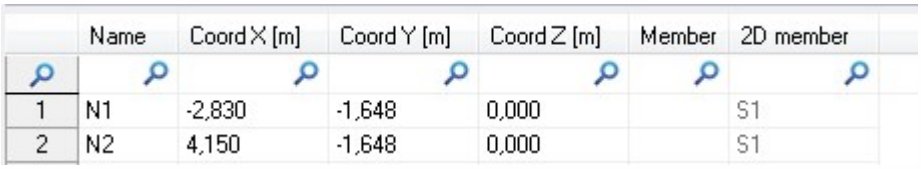


Tools in Table input - filtering row

Property filter

Values can be filtered by any property in the table. The filter is launched by the toolbar button for filter  .

The filter row is display as a first row of the grid. Each cell is marked by a magnifying glass icon  .



The filter cell works with part of the value/string or with the whole value/string.

User may search for all cells which contains letter "b" or whole "beam".

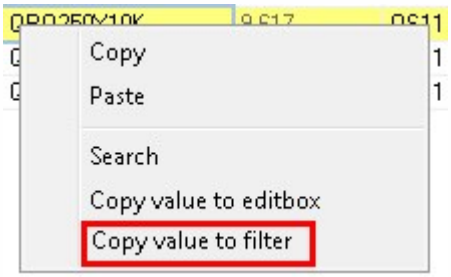
	Name	Type	Beg. n...	End n...
1	B1	column (100)	N5	N6
2	B2	beam (80)	N7	N8
3	B3	gable colum...	N9	N10
4	B4	column (100)	N11	N12
5	B5	column (100)	N13	N14
6	B6	beam (80)	N17	N18
7	B7	beam slab (...	N19	N20
8	B8	beam slab (...	N21	N22
9	B9	column (100)	N23	N24
10	B10	column (100)	N25	N26

	Name	Type	Beg. n...	End n
		b		
2	B2	beam (80)	N7	N8
3	B3	gable colum...	N9	N10
6	B6	beam (80)	N17	N18
7	B7	beam slab (...	N19	N20
8	B8	beam slab (...	N21	N22

	Name	Type	Beg. n...	End n...
		beam		
2	B2	beam (80)	N7	N8
6	B6	beam (80)	N17	N18
7	B7	beam slab (...	N19	N20
8	B8	beam slab (...	N21	N22

Use value from table in the property filter

There is a item in the context menu on the cell:



Invalid property for filter

When the property filter is grey it means that this property cannot be filtered.

..	Independent G ...	G m
	☑	8,50

The filter cannot be used for colours and check-boxes.

When the filter stayed

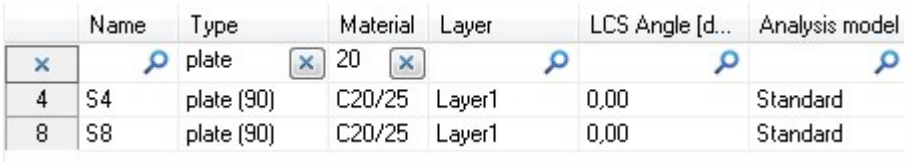
The tab will remember its filters during:

- moving;
- switching to another tab;
- hiding the tab;
- displaying the tab;

The filter will be cancelled when the property is removed from the grid.

More filters together

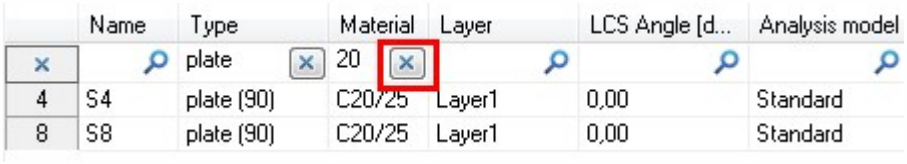
One of more filter cells may be used together. The filter is saved during sorting, removing column or switching to another tab.



	Name	Type	Material	Layer	LCS Angle [d...	Analysis model
		plate	20			
4	S4	plate (90)	C20/25	Layer1	0,00	Standard
8	S8	plate (90)	C20/25	Layer1	0,00	Standard

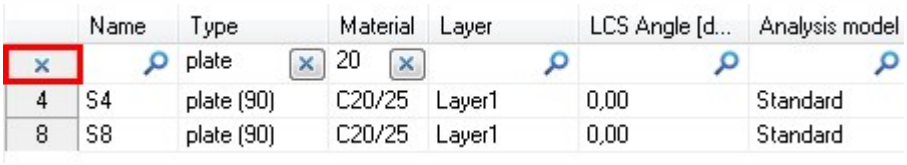
Cancel the filter

1. The filter for one column is cancelled by the x button in that column.



	Name	Type	Material	Layer	LCS Angle [d...	Analysis model
		plate	20			
4	S4	plate (90)	C20/25	Layer1	0,00	Standard
8	S8	plate (90)	C20/25	Layer1	0,00	Standard

2. All filters may be cancelled by the x button in the row header.



	Name	Type	Material	Layer	LCS Angle [d...	Analysis model
		plate	20			
4	S4	plate (90)	C20/25	Layer1	0,00	Standard
8	S8	plate (90)	C20/25	Layer1	0,00	Standard

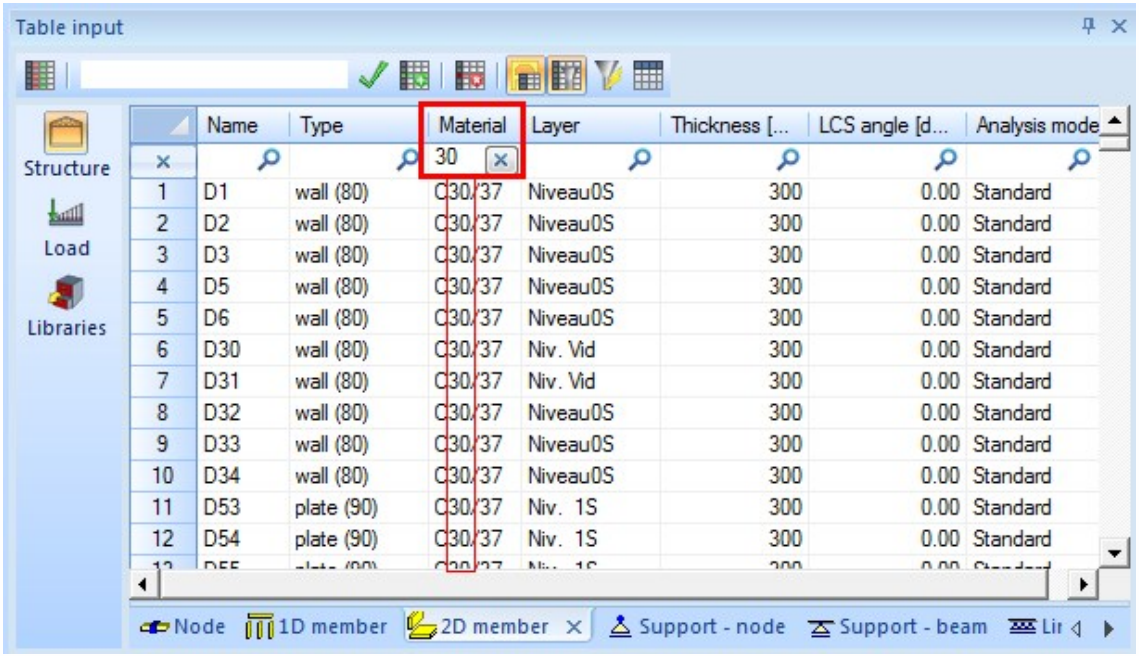
3. The filters in the columns are cancelled by "ESC" key one by one it the reverse order.

Mathematical operators "< bigger than" and "> lower than" and filtering by interval

There are 3 modes of filtering:

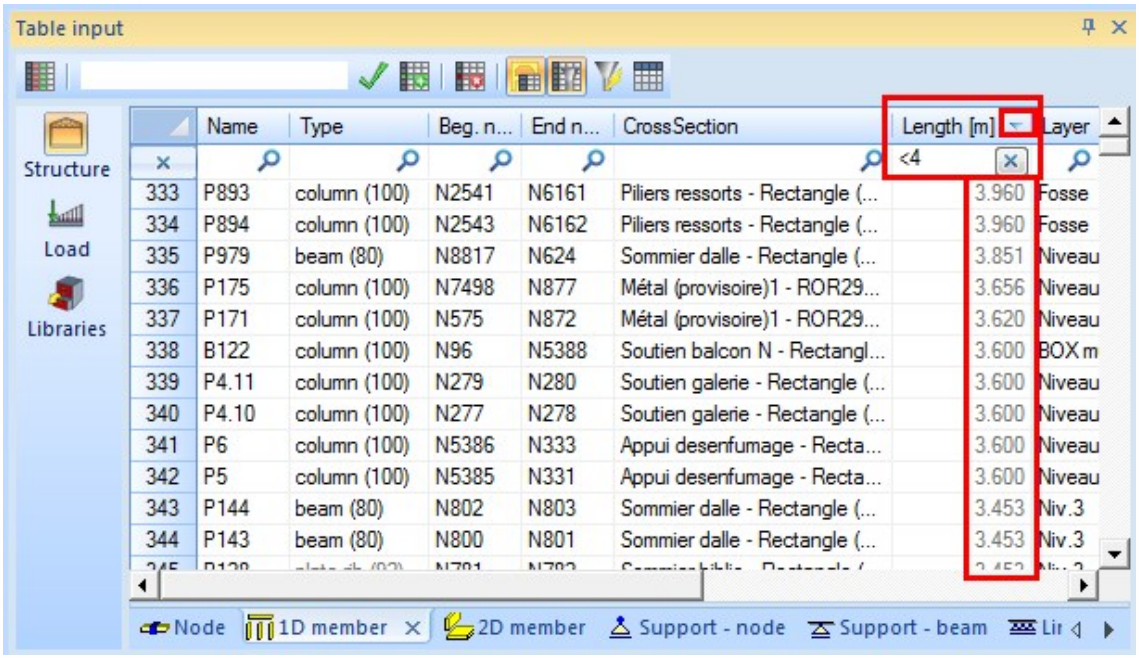
simple comparison

Table displays only rows which have in the selected column value which contain value entered in Filtering row. See following example:



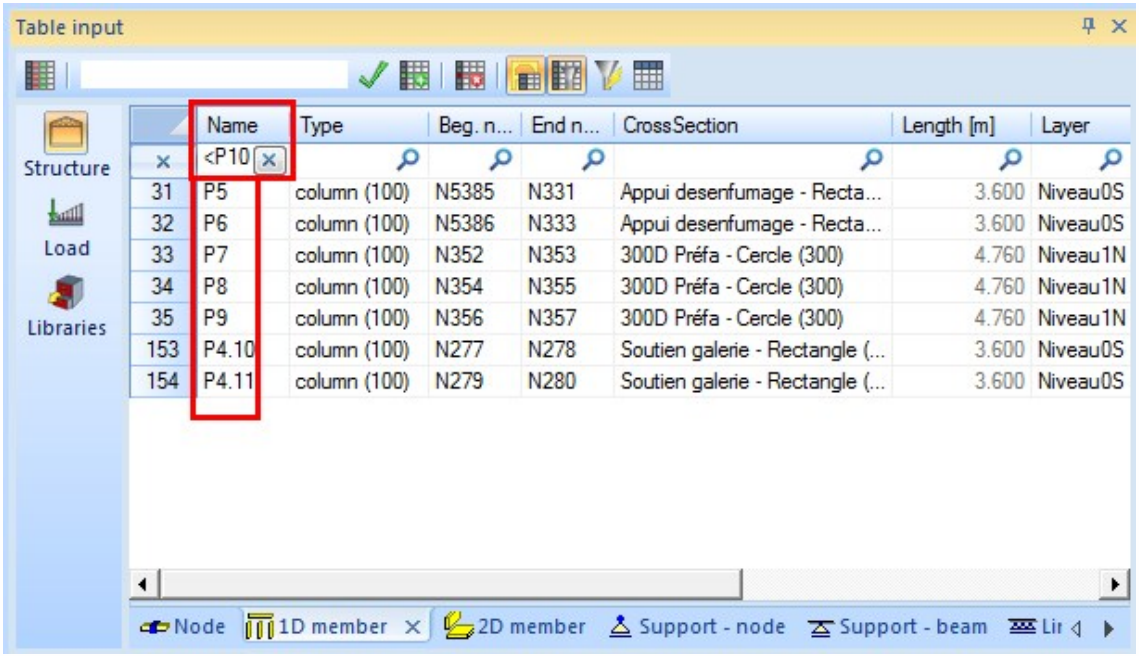
"<" and ">" for numerical values

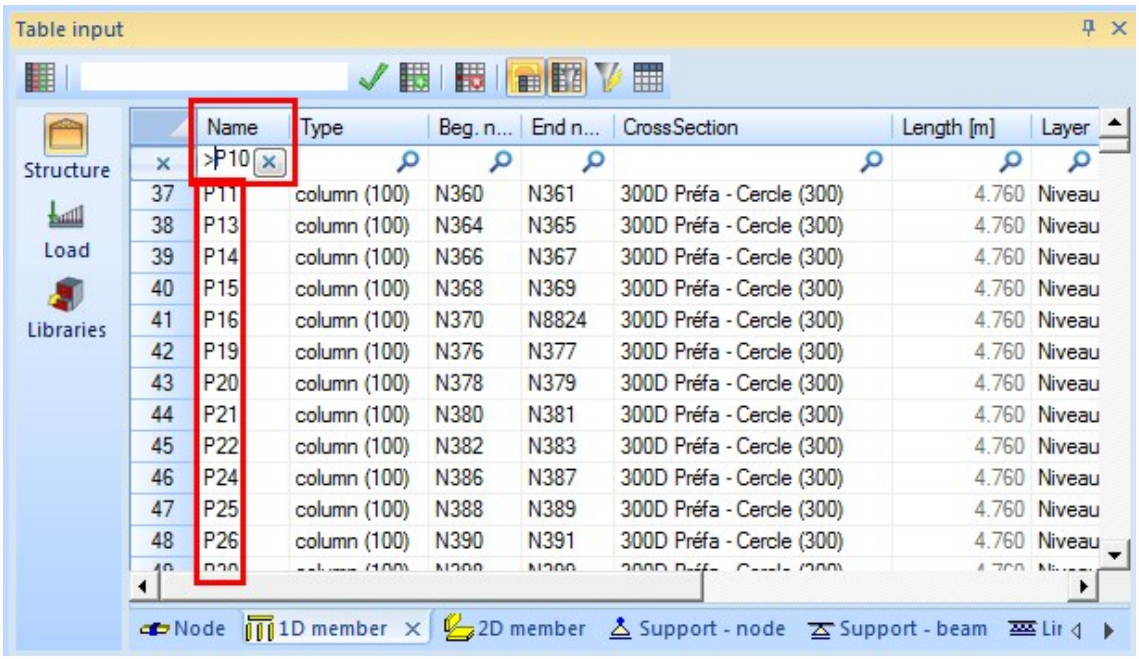
Table displays only rows which have in the selected column value 'bigger than...' or 'lower than...' specified value. See following example:



"<" and ">" for text values

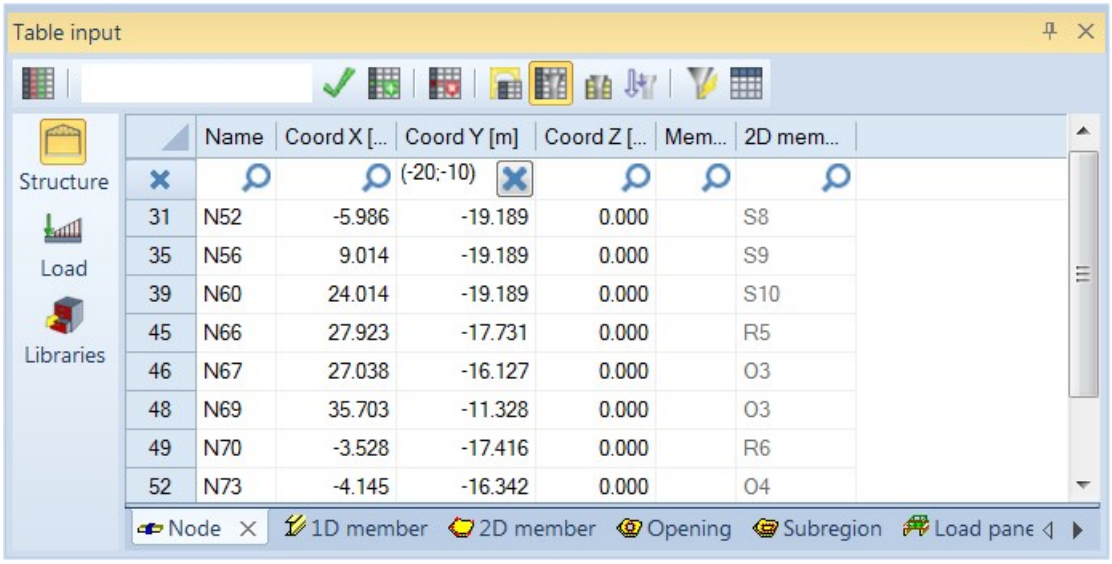
Table displays only rows which have in the selected column value 'bigger than...' or 'lower than...' specified string value. The advanced logic can also handle the naming system often used in civil engineering (e.g. naming of columns - first symbol (letter) according to x line-grid axis, the second symbol (number) according to y line-grid axis). See following examples:





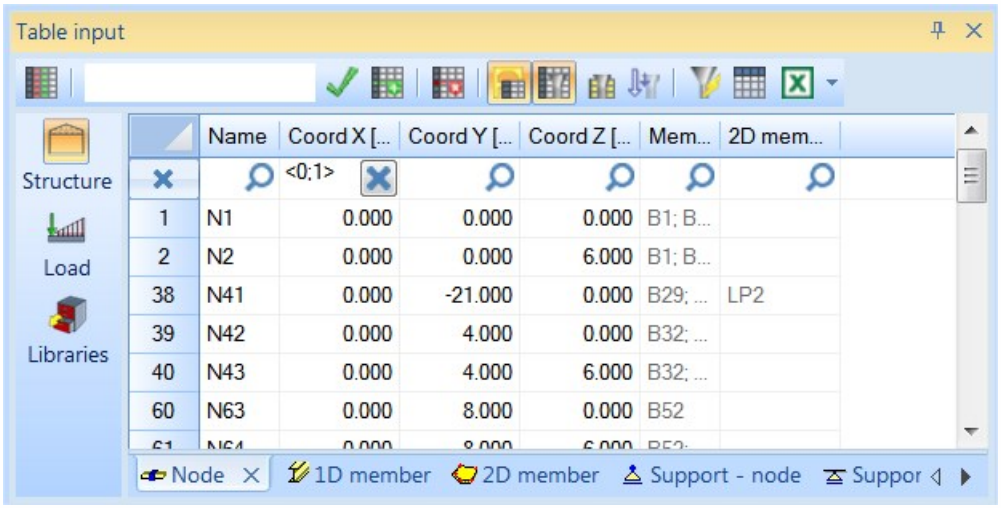
Filter by open interval "(XXX;YYY)"

The table displays only rows that have in the selected column value 'bigger than XXX' and 'lower than YYY', where 'XXX' and 'YYY' can be a numerical or string value.



Filter by closed interval "<XXX;YYY>"

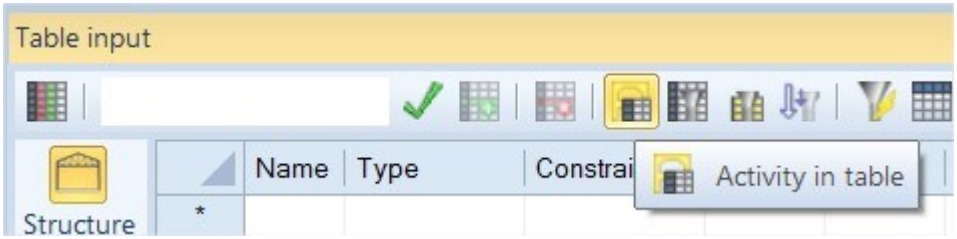
The table displays only rows that have in the selected column value 'bigger or equal to XXX' and 'lower or equal to YYY', where 'XXX' and 'YYY' can be a numerical or string value. The sorting by closed interval is best to be used for discrete and round values, such as names of entities various member properties or coordinates.



Tools in Table input - filtering by activity, selection and active load case

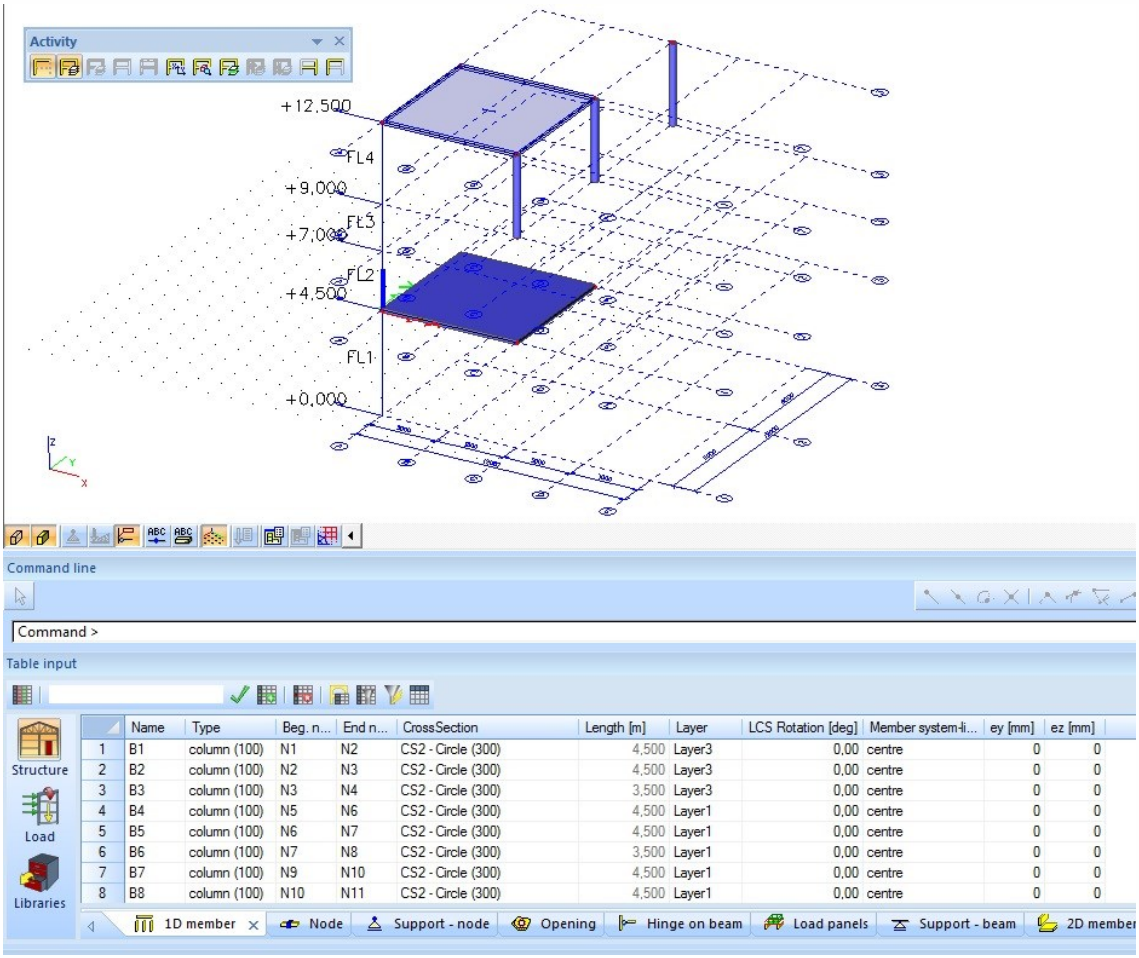
Activity filter in Table input

The activity in the 3D window is controlled by Activity tools. The same activity can be used in the Table input. It is managed by the button next to the filters on the toolbar.



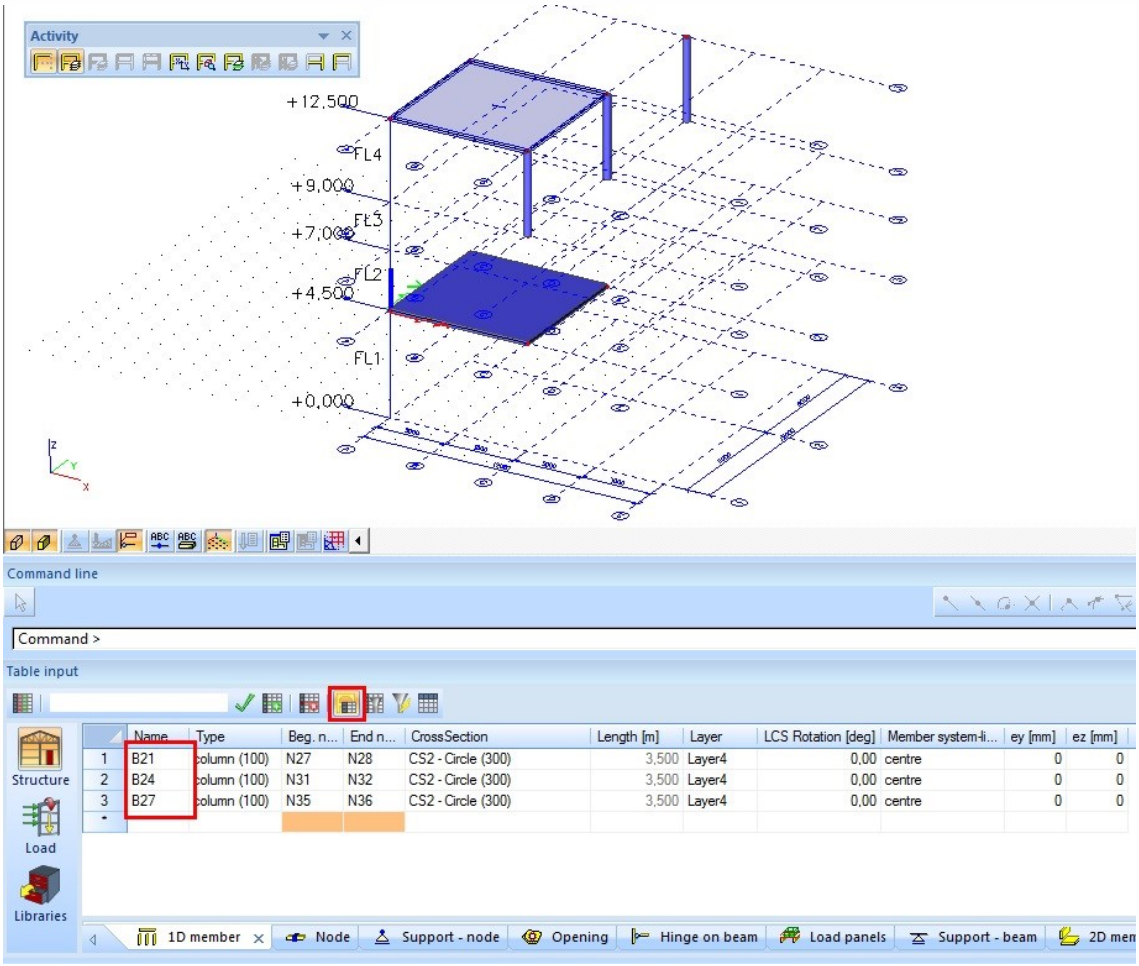
This button allows filtering the content according to current activity in the 3D window. The default status is ON.

When the Activity in Table input is OFF:



	Name	Type	Beg. n...	End n...	CrossSection	Length [m]	Layer	LCS Rotation [deg]	Member system-li...	ey [mm]	ez [mm]
1	B1	column (100)	N1	N2	CS2 - Circle (300)	4,500	Layer3	0,00	centre	0	0
2	B2	column (100)	N2	N3	CS2 - Circle (300)	4,500	Layer3	0,00	centre	0	0
3	B3	column (100)	N3	N4	CS2 - Circle (300)	3,500	Layer3	0,00	centre	0	0
4	B4	column (100)	N5	N6	CS2 - Circle (300)	4,500	Layer1	0,00	centre	0	0
5	B5	column (100)	N6	N7	CS2 - Circle (300)	4,500	Layer1	0,00	centre	0	0
6	B6	column (100)	N7	N8	CS2 - Circle (300)	3,500	Layer1	0,00	centre	0	0
7	B7	column (100)	N9	N10	CS2 - Circle (300)	4,500	Layer1	0,00	centre	0	0
8	B8	column (100)	N10	N11	CS2 - Circle (300)	4,500	Layer1	0,00	centre	0	0

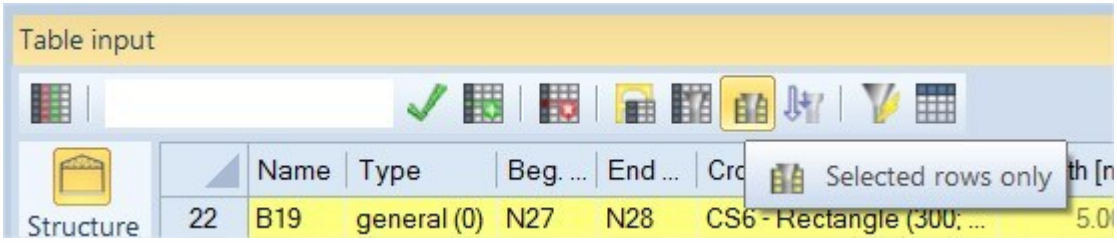
When the Activity in Table input is ON:



The activity in the Table input may be combined with the Property filter tool.

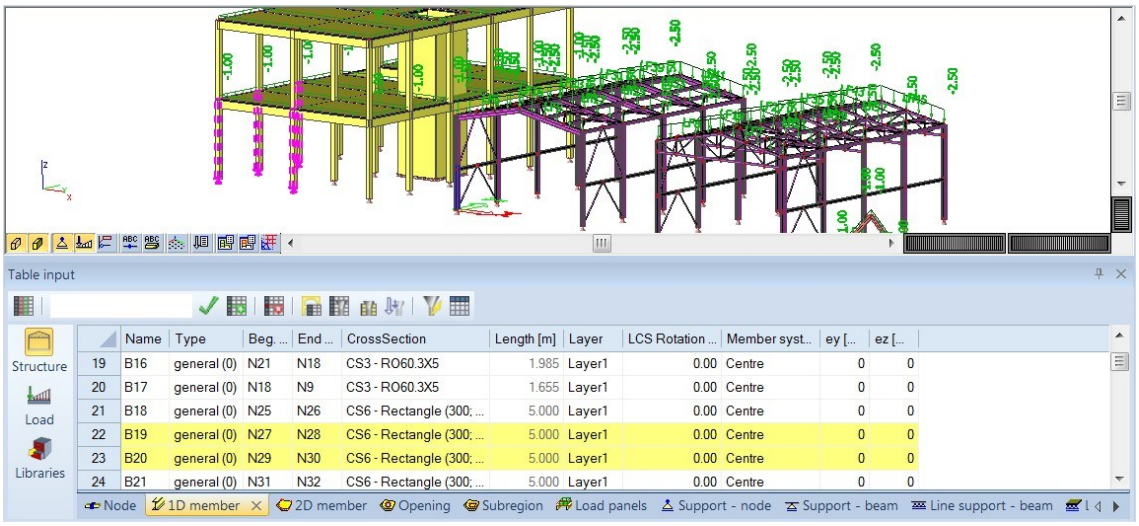
Selection filter in Table input

Any user-selection made in the 3D window is also displayed in the Table Input by yellow highlighted rows. The selection can be used as a filter in the Table input. It is managed by the [Selected Rows Only] button next to the filters on the toolbar.

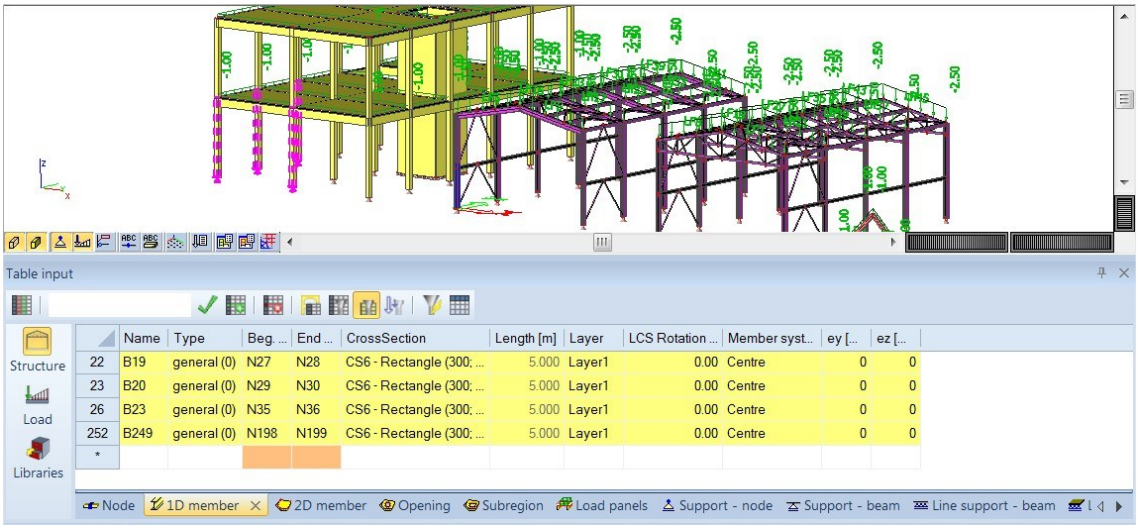


This button allows for filtering of the content according to current selection in the 3D window. The default status is OFF.

When the [Selected Rows Only] is OFF:

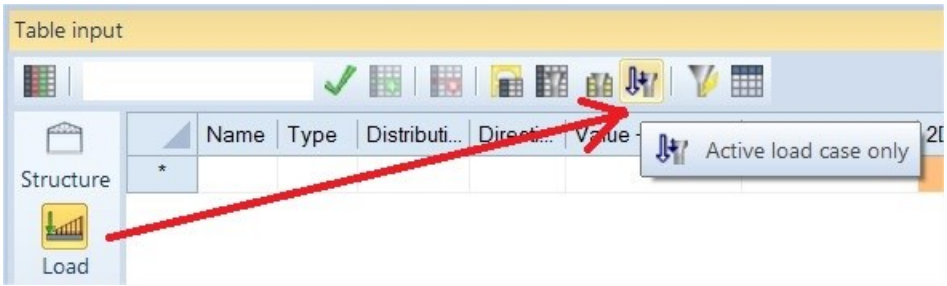


When the [Selected Rows Only] is ON:



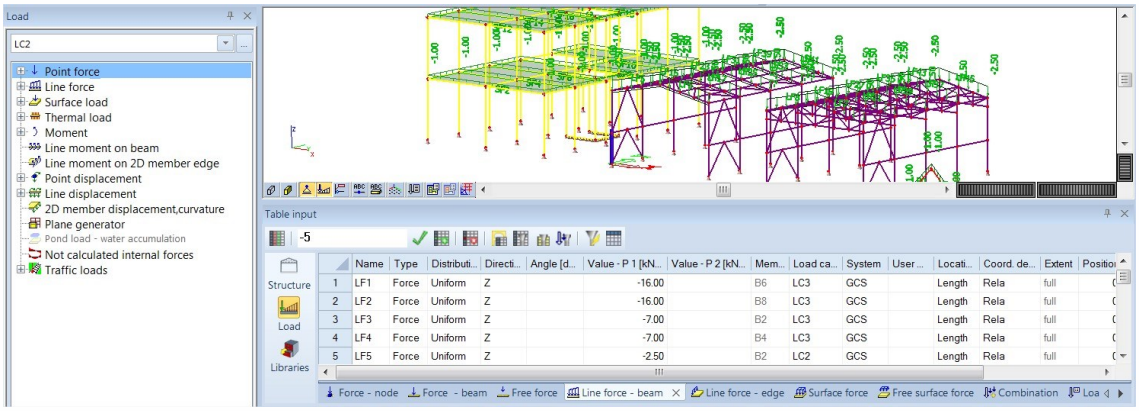
Filter by active load case in Table input

The activity in the 3D window is controlled by Activity tools. The same activity can be used in the Table input. It is managed by the [Active Load Case Only] button next to the filters on the toolbar. Note that button [Active Load Case Only] is enabled only when entities from group: Load are displayed.

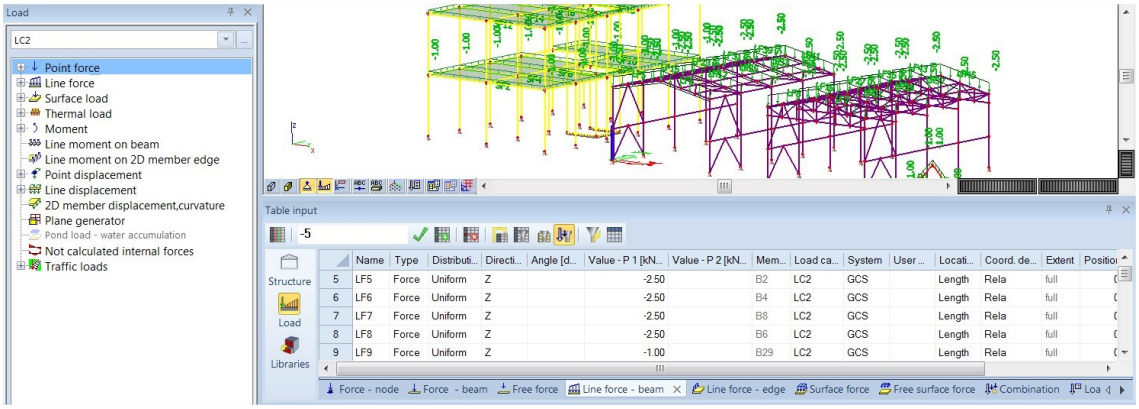


This button allows for filtering of the content according to currently active load case in service Loads. The default status is OFF.

When the [Active Load Case Only] is OFF:



When the [Active Load Case Only] is ON:



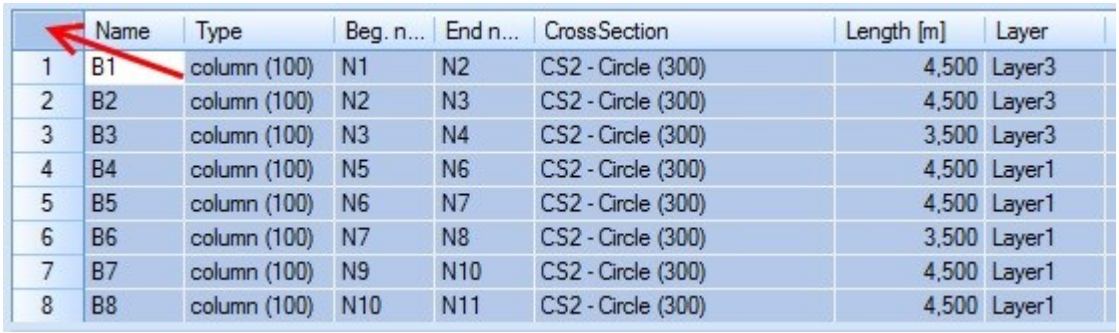
Tools in Table input - selection

Selections

Whole Tab

There are two possibilities to select whole content of the Tab:

1. Click CTRL+A
2. Click to the top left corner:



	Name	Type	Beg. n...	End n...	CrossSection	Length [m]	Layer
1	B1	column (100)	N1	N2	CS2 - Circle (300)	4,500	Layer3
2	B2	column (100)	N2	N3	CS2 - Circle (300)	4,500	Layer3
3	B3	column (100)	N3	N4	CS2 - Circle (300)	3,500	Layer3
4	B4	column (100)	N5	N6	CS2 - Circle (300)	4,500	Layer1
5	B5	column (100)	N6	N7	CS2 - Circle (300)	4,500	Layer1
6	B6	column (100)	N7	N8	CS2 - Circle (300)	3,500	Layer1
7	B7	column (100)	N9	N10	CS2 - Circle (300)	4,500	Layer1
8	B8	column (100)	N10	N11	CS2 - Circle (300)	4,500	Layer1

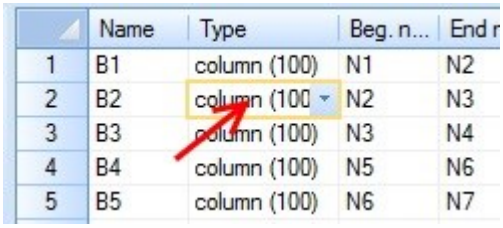
One row (=one member)

The whole row can be selected by drag and drop from the first cell to the last one.

	Name	Type	Beg. n...	End n...	CrossSection	Length [m]	Layer	LCS Rotation [deg]	Member system-li...	ey [mm]	ez [mm]
1	B1	column (100)	N1	N2	CS2 - Circle (300)	4,500	Layer3	0,00	centre	0	0
2	B2	column (100)	N2	N3	CS2 - Circle (300)	4,500	Layer3	0,00	centre	0	0
3	B3	column (100)	N3	N4	CS2 - Circle (300)	3,500	Layer3	0,00	centre	0	0
4	B4	column (100)	N5	N6	CS2 - Circle (300)	4,500	Layer1	0,00	centre	0	0
5	B5	column (100)	N6	N7	CS2 - Circle (300)	4,500	Layer1	0,00	centre	0	0
6	B6	column (100)	N7	N8	CS2 - Circle (300)	3,500	Layer1	0,00	centre	0	0
7	B7	column (100)	N9	N10	CS2 - Circle (300)	4,500	Layer1	0,00	centre	0	0

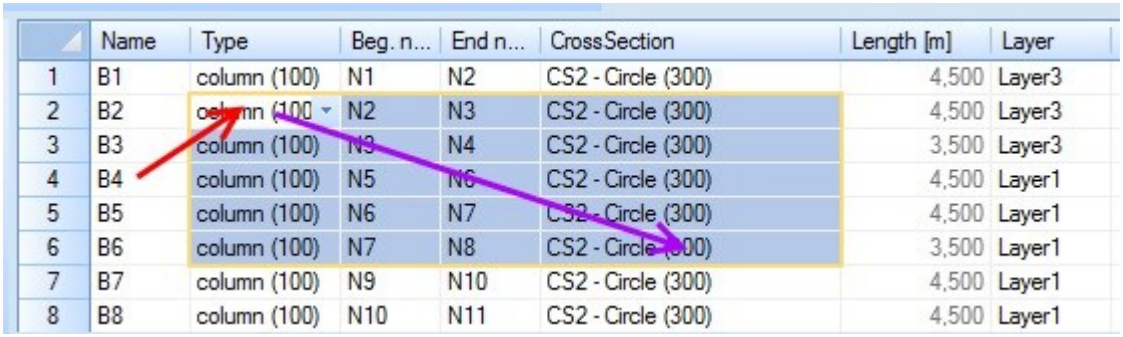
One cell

The cell can be selected by clicking on it.



More cells

More cells can be selected by dragging selection rectangle from one corner to the other.



Tools in Table input - copy and paste

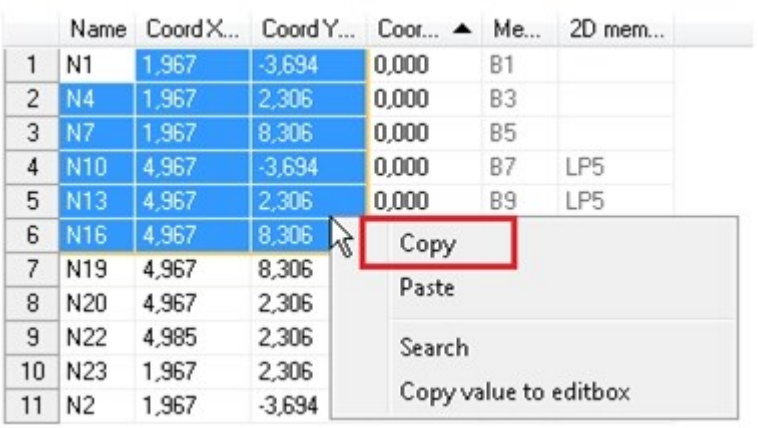
Introduction

One of the most important functionalities is copy and paste content from/to the clipboard. This can be used for different types of applications – most probably MS Excel.

It allows you to create complex geometry (e.g. defined by some function) with many thousands of nodes and insert it into Table input in one step.

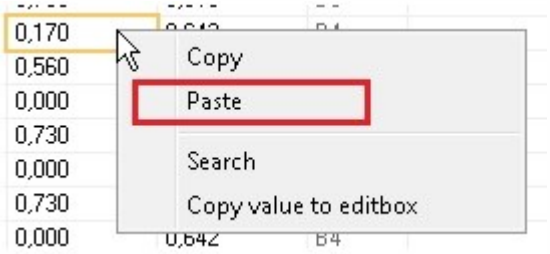
Copy

Selected content is copied to the clipboard by the context menu (item Copy) or by a standard short-cut CTRL+C. Afterwards, the content of clipboard can be inserted e.g. to MS Excel. To allow clear overview of the copied entities, the table header is stored in the clipboard together with the copied data.



Paste

Pasting the selected content back to the table is also possible using the context menu (item Paste) or by a standard short-cut CTRL+V. Data to be pasted can be prepared e.g. in MS Excel. The inserted data may start with the table header as the first line.



Technical details

In this chapter there is brief description how does the pasting of data from clipboard works internally. This knowledge is not important for using the Table input, but it is good for understanding some issues which may arise.

Table input is intended to allow to paste huge amount of data in a batch. We are able to input e.g. 20 000 nodes in less than one minute. There are 2 contradictory requests: to do it quickly and to guarantee that inputted data are consistent. The first requests implies that ideally there is no data check that makes the code execution time demanding. The second requests implies that there should be many checks of input data to guarantee that project won't be damaged by inputting wrong data set. It is good to realize that finding good balance of these two requests is not easy.

We can distinguish two basic operations that can also be combined. The first one is related just to editing the actual data, no new entity is created. The second one regards creating new entities on tabs with the "New row (= new member)" on page 162. The third is combination of them.

1) Pasting data to modify actual entities

In this case the algorithm goes through particular cells one by one (from left to right, from top to down) and if old and new value differ it just try to set new value. Sometimes it happens that new value cannot be set (e.g. due to geometrical check or input value is out of range) at the time. If some value was not successfully changed, at the end of every row the algorithm tries to set those values once again. If the change was not successful even during the second try, algorithm returns warning message (if the entity returns some message) and jump to the next row.

Preceding description implies that if the operation is interrupted by the button beside the progress bar the all already finished changes are accepted.

If editing of some property of some entity causes some warning message, this message is remembered and all messages from the whole editing process are displayed in one summarizing warning message in the end.

2) Pasting data to create new entities using the "New row"

Firstly all new entities are internally created with some default values of properties and than they are written into data-server in one step. If this operation is interrupted the whole operation is cancelled and no entity is created.

The second step is to modify newly created entities and set values defined in clipboard in the way described in "1) Pasting data to modify actual entities".

3) Pasting data both to modify actual entities and creating new entities

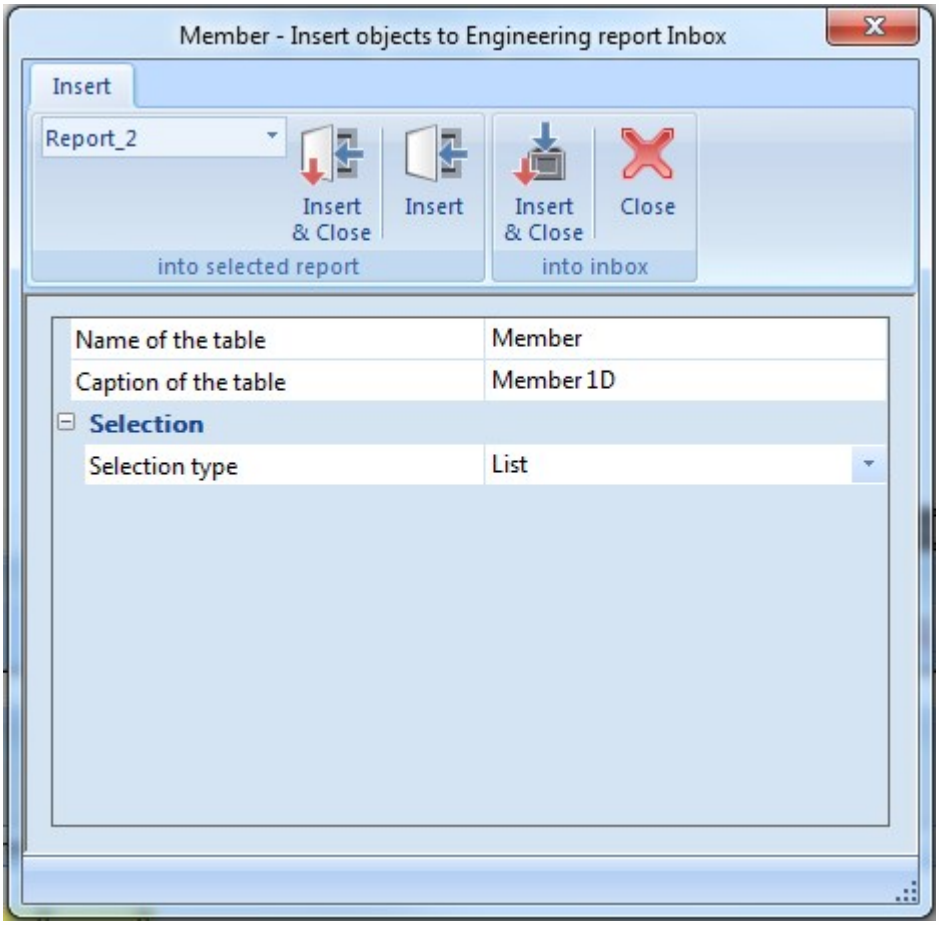
In the first step actual entities are edited [1) Pasting data to modify actual entities].

In the second step new entities are created and edited [2) Pasting data to create new entities using the "New row"].

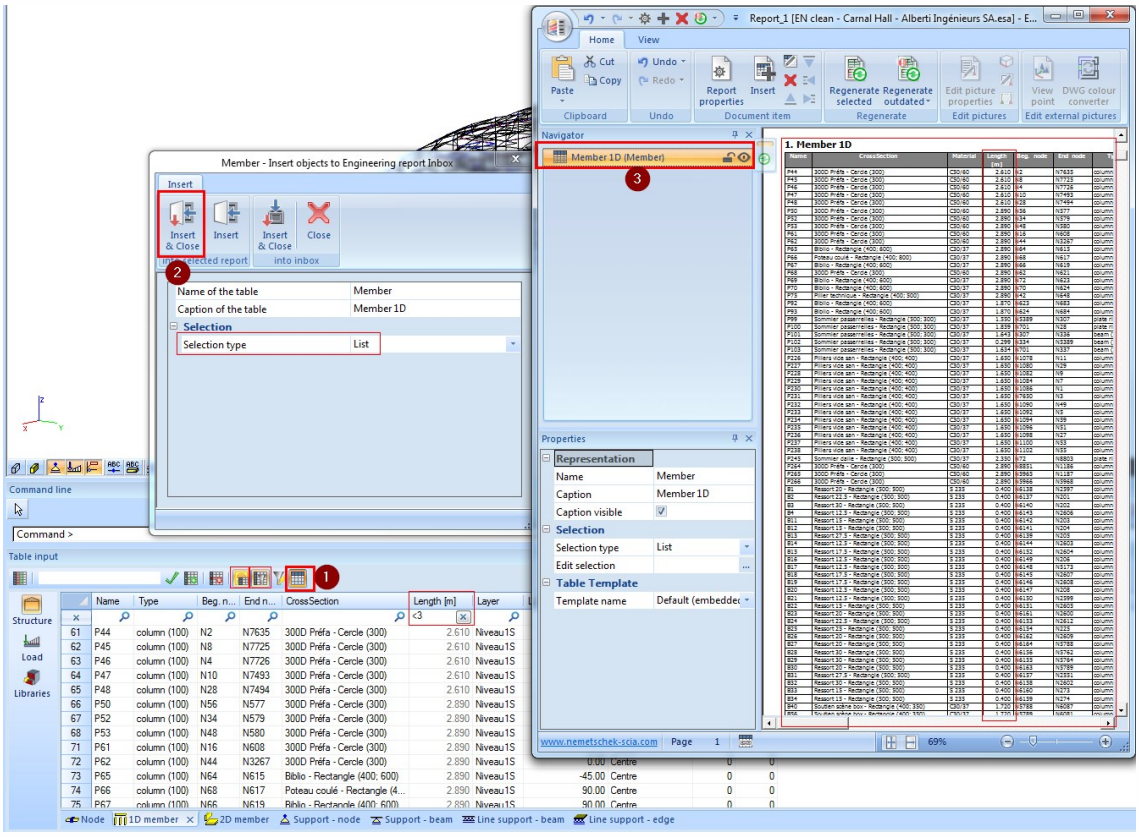
Tools in Table input - send table to Engineering Report

Table can be easily sent to Engineering Report using the related button on the toolbar:

After clicking this button the standard dialogue "Insert objects to Engineering Report" is opened:



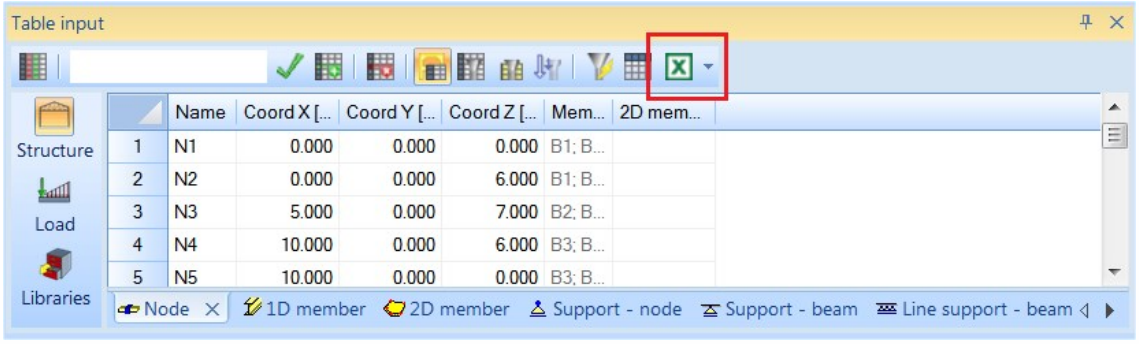
Afterwards the table is sent to Engineering Report with respect to "Filtering row" and "Activity in table":



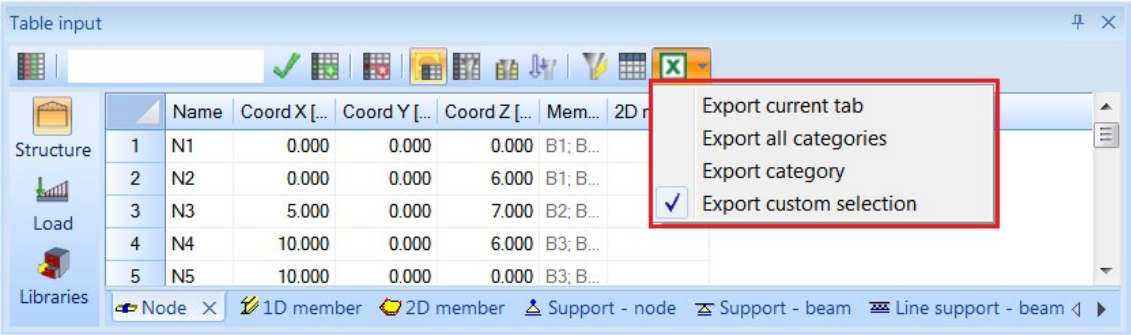
Sorting of the table grid is not respected during sending to Engineering Report. Items are afterwards in Engineering Report sorted in default order. Sorting of tables in Engineering Report can be adjusted in "Table layout editor".

Tools in Table input - send table to MS Excel with one click

Table can be easily sent to MS Excel using the related button on the toolbar:



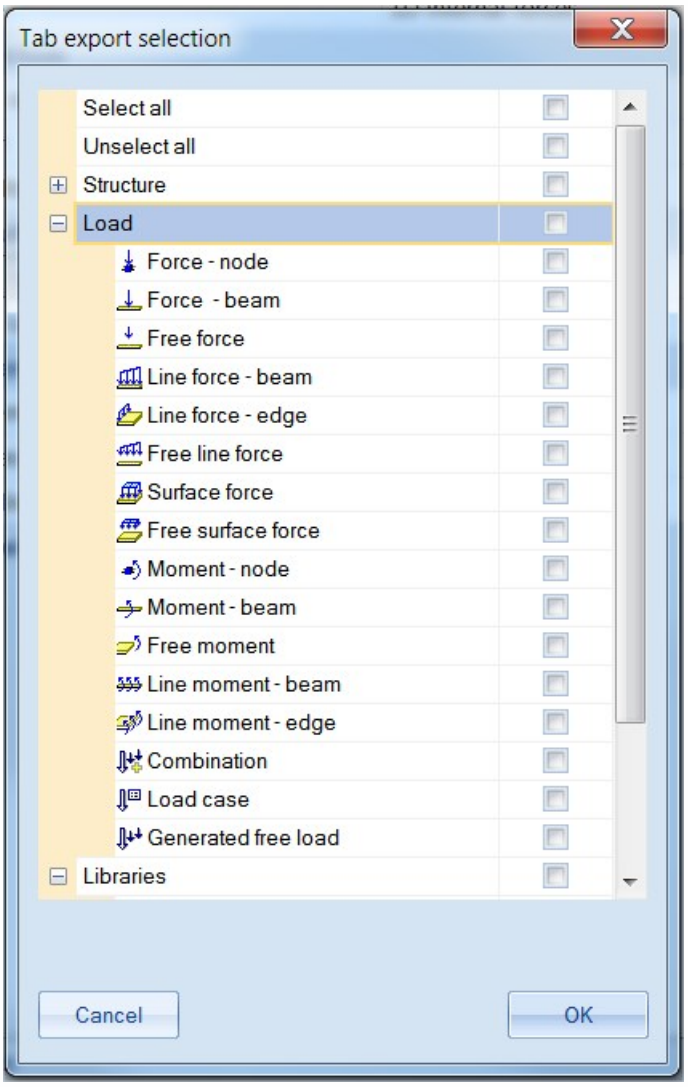
There are two parts of the export button. By simple mouse left click on the main button with icon of Excel, the default or last used settings of tabs are exported and immediately opened in MS Excel. The second part of the button is an arrow with quick settings of what selection of tabs is going to be exported.



There are four possibilities of simple settings:

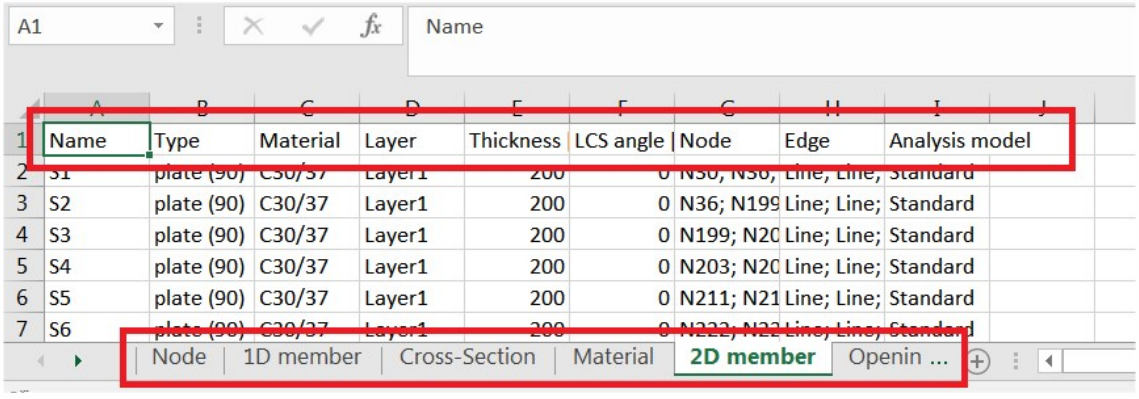
Export current tab	Exports only currently displayed tab of Table input. When you use the Export to Excel button for the first time, its default setting is set to Export current tab.
Export all categories	All tables from all categories (Structure, Load, Libraries) will be exported.
Export category	All tabs from currently open category will be exported.
Export custom selection	Additional dialogue will be presented before the export is performed. In this dialogue custom selection from all possible tabs may be made.

Export custom selection dialogue for table Input is apparent on image below. In the dialogue single entities may be selected and there is also option to select or deselect all or select or deselect whole categories.



For the first use of export button, its default setting is set to Export current tab to prevent possible long and unintentional export procedure. For next use of export button, the last used settings is used. During the export process it is checked whether the tab is empty or not and empty tabs are skipped during the export. When exported, MS Excel is launched and exported file is presented. This file is stored in temp folder and for future use it is advised to save it to a different location.

For better readability of the exported Excel sheet is every tab from table input represented by separate named sheet and together with exported data the table header is visible in the Excel file.

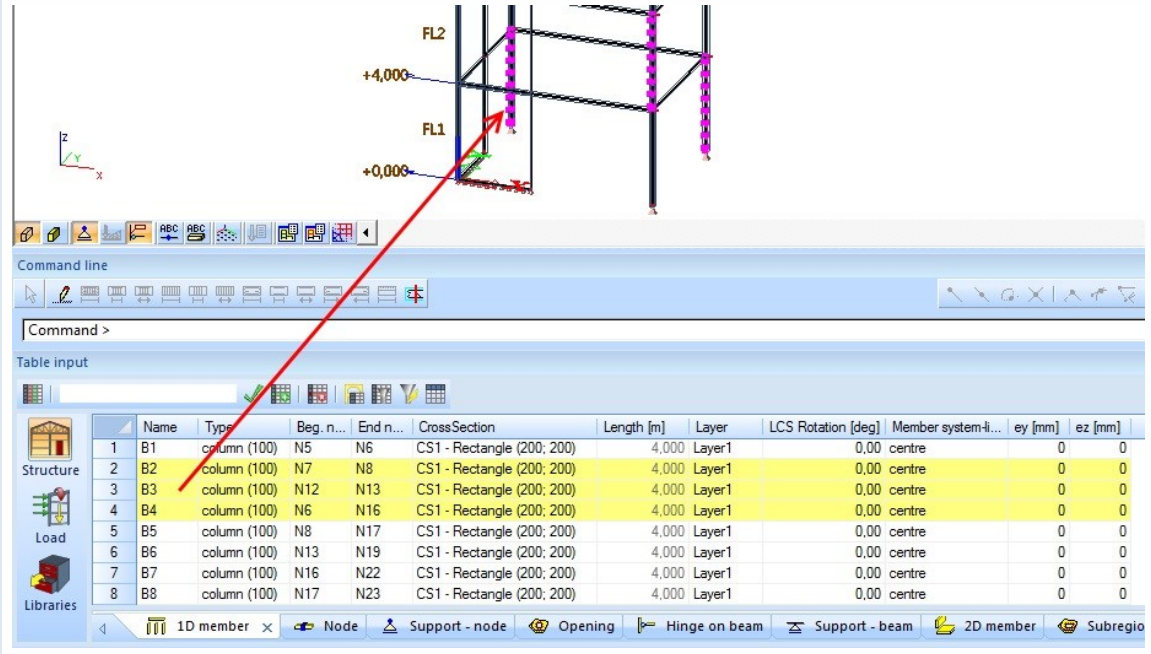


Current sorting or filtering of the table grid is not respected during sending to MS Excel. Items are afterwards in Excel sheet sorted in default order. For copying sorted or filtered parts of Table input please use the Copy and Paste (Ctrl+C) and Paste (Ctrl+V).

Selection of members

Table input allows making the selection of members in the model using the click on the row header. Selected members are highlighted in Table input by yellow color.

Example:
There are three members highlighted in the Table input and it is also selected in the model.



	Name	Type	Beg. n...	End n...	CrossSection	Length [m]	Layer	LCS Rotation [deg]	Member system-li...	ey [mm]	ez [mm]
1	B1	column (100)	N5	N6	CS1 - Rectangle (200; 200)	4,000	Layer1	0,00	centre	0	0
2	B2	column (100)	N7	N8	CS1 - Rectangle (200; 200)	4,000	Layer1	0,00	centre	0	0
3	B3	column (100)	N12	N13	CS1 - Rectangle (200; 200)	4,000	Layer1	0,00	centre	0	0
4	B4	column (100)	N6	N16	CS1 - Rectangle (200; 200)	4,000	Layer1	0,00	centre	0	0
5	B5	column (100)	N8	N17	CS1 - Rectangle (200; 200)	4,000	Layer1	0,00	centre	0	0
6	B6	column (100)	N13	N19	CS1 - Rectangle (200; 200)	4,000	Layer1	0,00	centre	0	0
7	B7	column (100)	N16	N22	CS1 - Rectangle (200; 200)	4,000	Layer1	0,00	centre	0	0
8	B8	column (100)	N17	N23	CS1 - Rectangle (200; 200)	4,000	Layer1	0,00	centre	0	0

The highlight works otherwise also - the selection created in the 3D window is displayed as highlighted row/s in the tab/s.

Example: Member B1 and B2 are highlighted in the tab 1D members, they are also selected in the 3D window and the intersection of their properties is displayed in the Property window.

The selection of members may be created in the 3D model graphically or in Table input by the row/s highlight. Both ways leads to the same result - rows are highlighted, property window displays property intersection and the selection is visible in the 3D window.

The selection could be disabled by View parameters settings, it this case highlight doesn't work either.

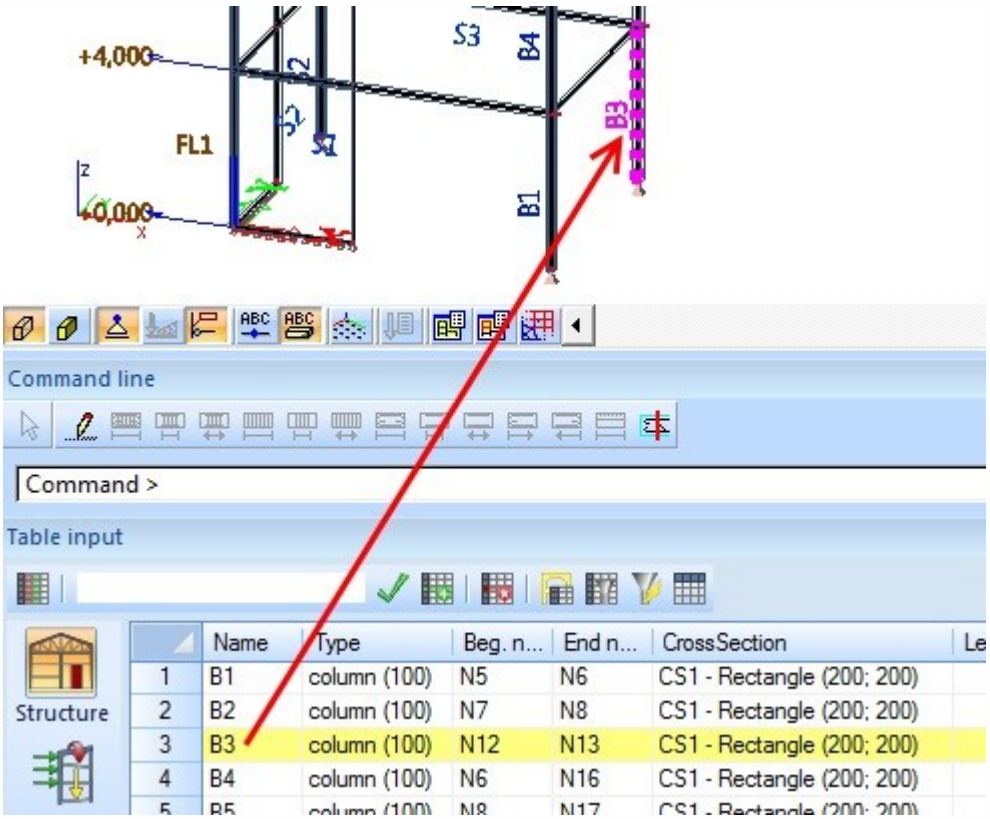
Highlight from the Table input

Single highlight:

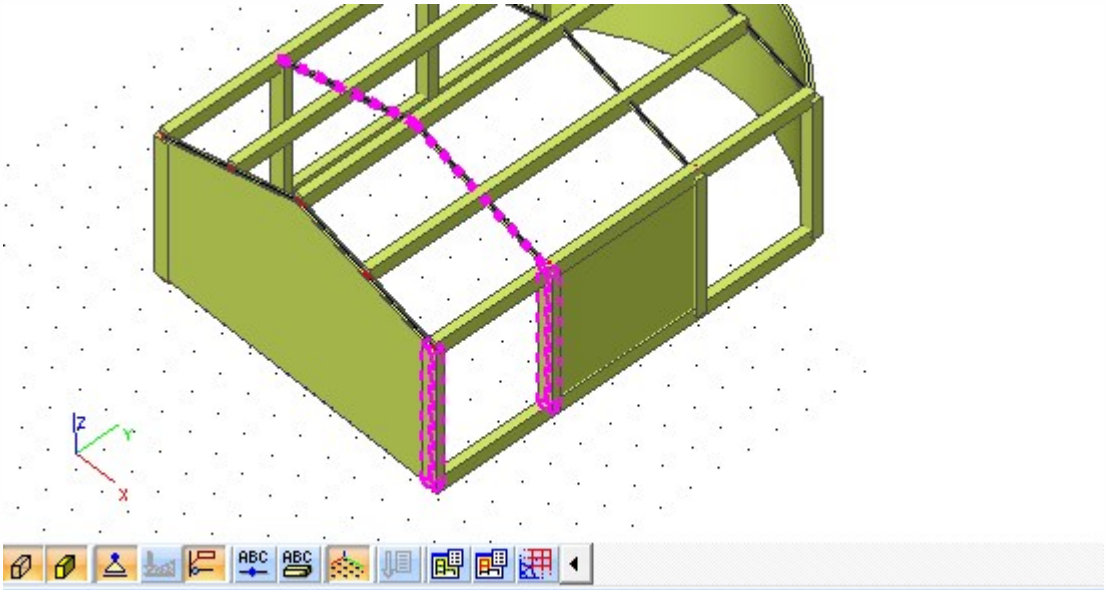
- Click on the row header to select one member.

Multi-highlight:

- Hold SHIFT or CTRL key (standard short-cuts) to create a multi-selection of rows (in one block or several rows together).

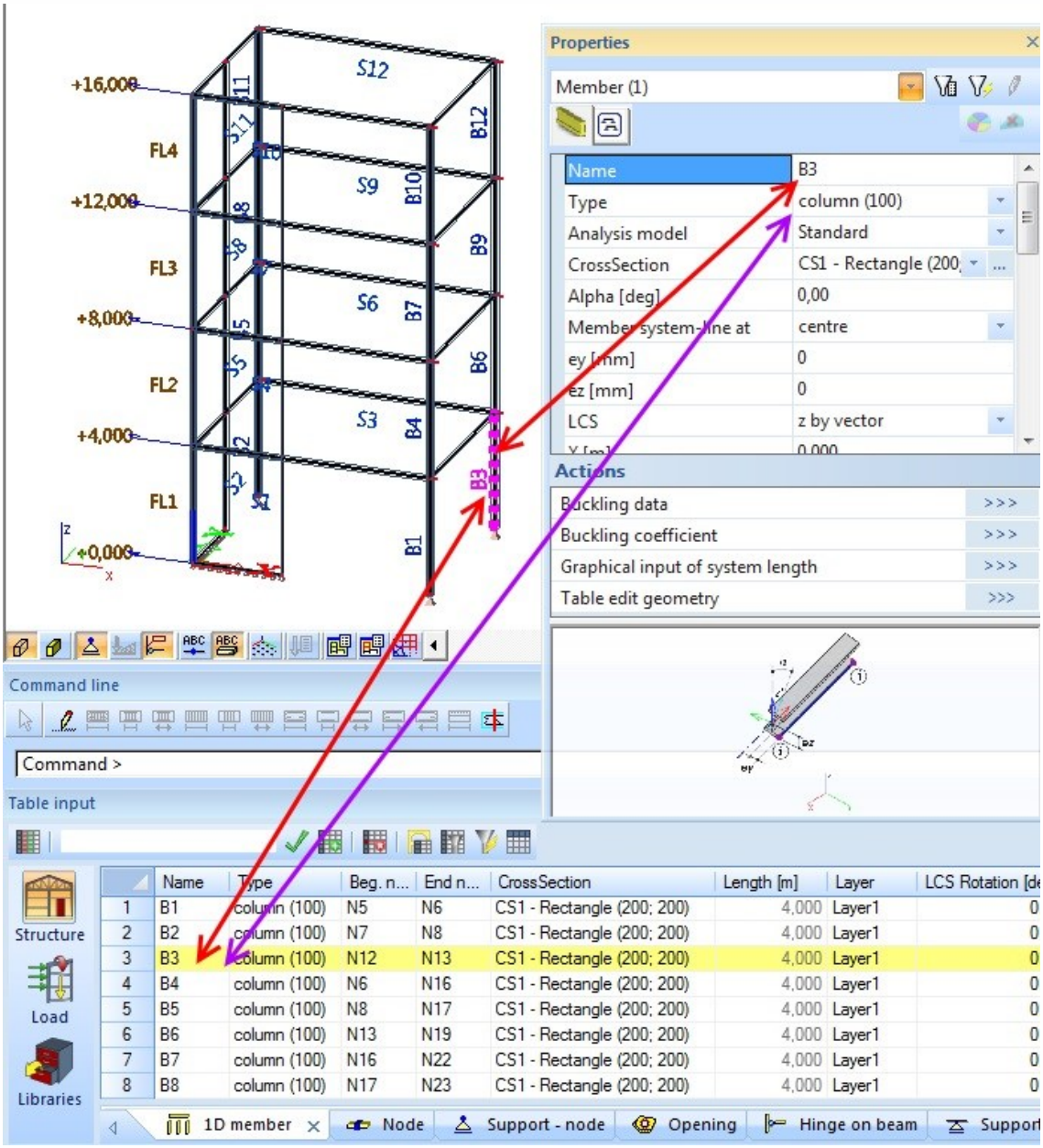


	Name	Type	Beg. n...	End n...	CrossSection	Le
1	B1	column (100)	N5	N6	CS1 - Rectangle (200; 200)	
2	B2	column (100)	N7	N8	CS1 - Rectangle (200; 200)	
3	B3	column (100)	N12	N13	CS1 - Rectangle (200; 200)	
4	B4	column (100)	N6	N16	CS1 - Rectangle (200; 200)	



	Name	Type	Beg. n...	End n...	CrossSection	Length [m]
1	B1	column (100)	N1	N2	CS2 - Rectangle (500; 300)	5,000
2	B2	beam (80)	N2	N3	CS1 - T140	5,220
3	B3	beam (80)	N3	N4	CS1 - T140	5,220
4	B4	column (100)	N5	N4	CS2 - Rectangle (500; 300)	5,000

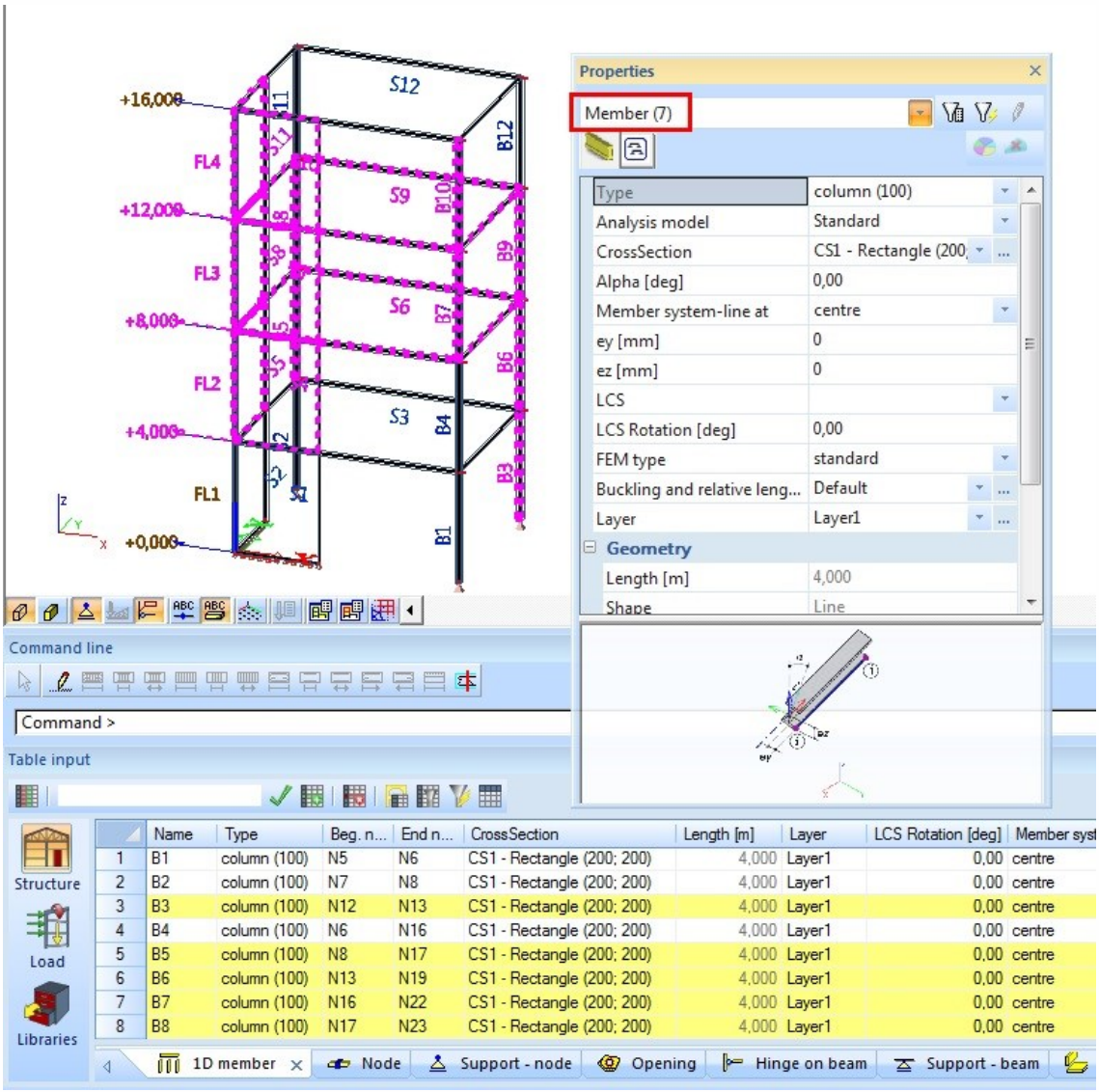
The highlighted = selected member displays its properties in the property window.



	Name	Type	Beg. n...	End n...	CrossSection	Length [m]	Layer	LCS Rotation [de
1	B1	column (100)	N5	N6	CS1 - Rectangle (200; 200)	4,000	Layer1	0
2	B2	column (100)	N7	N8	CS1 - Rectangle (200; 200)	4,000	Layer1	0
3	B3	column (100)	N12	N13	CS1 - Rectangle (200; 200)	4,000	Layer1	0
4	B4	column (100)	N6	N16	CS1 - Rectangle (200; 200)	4,000	Layer1	0
5	B5	column (100)	N8	N17	CS1 - Rectangle (200; 200)	4,000	Layer1	0
6	B6	column (100)	N13	N19	CS1 - Rectangle (200; 200)	4,000	Layer1	0
7	B7	column (100)	N16	N22	CS1 - Rectangle (200; 200)	4,000	Layer1	0
8	B8	column (100)	N17	N23	CS1 - Rectangle (200; 200)	4,000	Layer1	0

Highlight from the 3D window

Selected items in the 3D window are highlighted in the Table input.



Add member to the highlighted / selected items

When user highlights one more row (using CTRL key) or graphically selects additional member it is automatically added to the original selection.

Cancel highlight

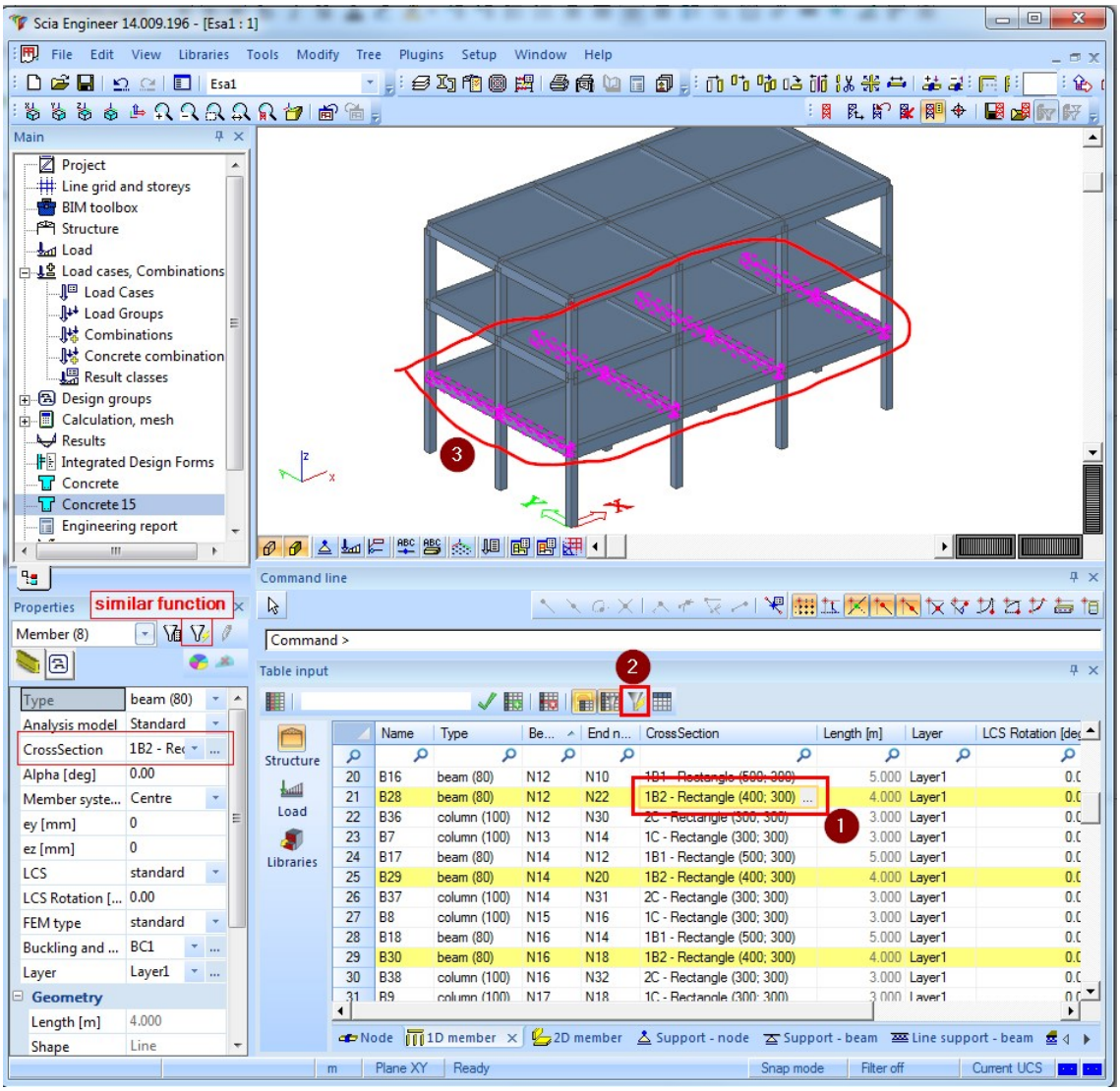
The highlight is cancelled by ESC key. It can be used in 3D window (a standard way) or in the Table input directly.

The ESC key cancel the highlight in the current tab only, other tabs will keep their highlight.

Select by property in cell

In Table input it is possible to select all members with the same property value by one click:

1. Select one cell in the grid of Table input
2. Click the button "Select by property in cell" on the toolbar of Table input
3. All members with the same property value are selected in the model and related rows are highlighted in Table input



This functionality is similar to filtering the 3D model by the selected property in the property dialogue (the icon with funnel).

Multiple editbox

The multiple editbox is a special tool for the Table input. It can be used for editing values in the cells by formulas, to substitute values in the cells, for copy row or copy add data (supports, loads, etc.).

There are two important buttons which confirm the content of the multiple editbox.

"Apply edit" button

The button for substitution, searching and using the mathematical formulas.
It is possible to confirm this button by Enter key.

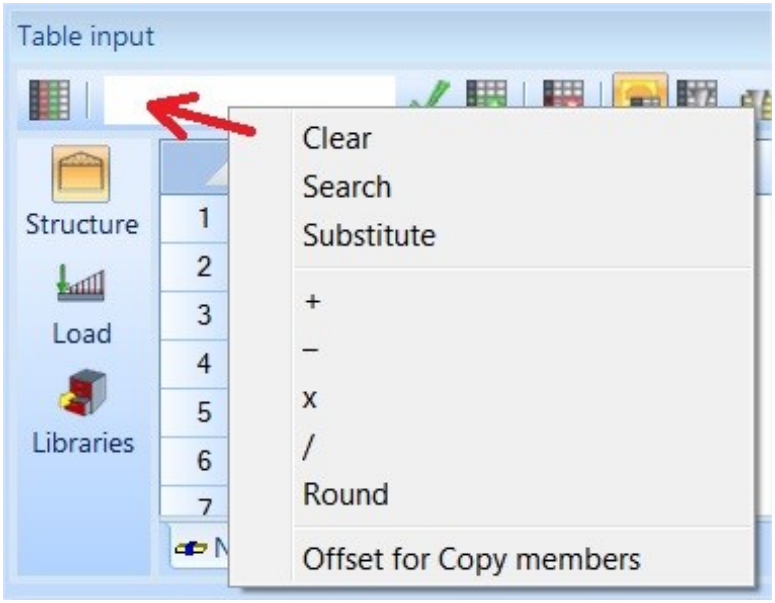
"Copy row" button

The button for copying the row (new member or new add data).
Mouse click is required for this option, Enter key doesn't work.

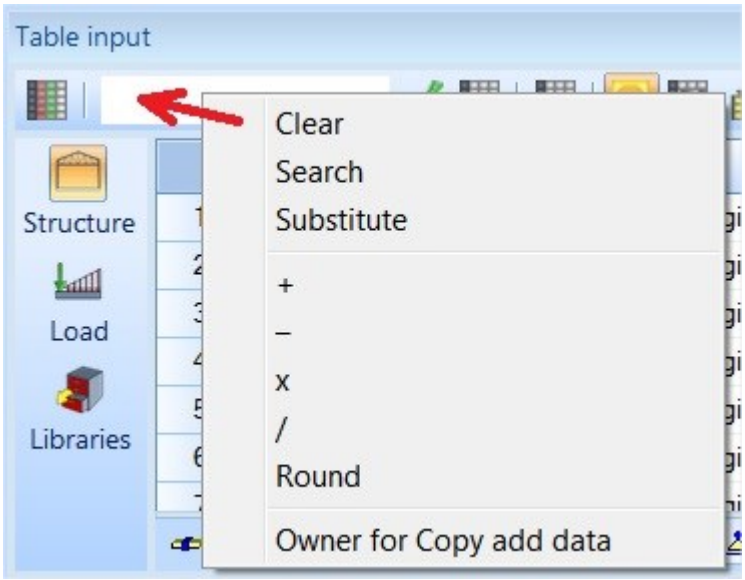
Editbox right-click menu

Right-click menu contains items for mathematical formulas, substitution, search, clear the editbox and copy command (member or add data).

Right click menu for Copy member:



Example for the Copy add data:

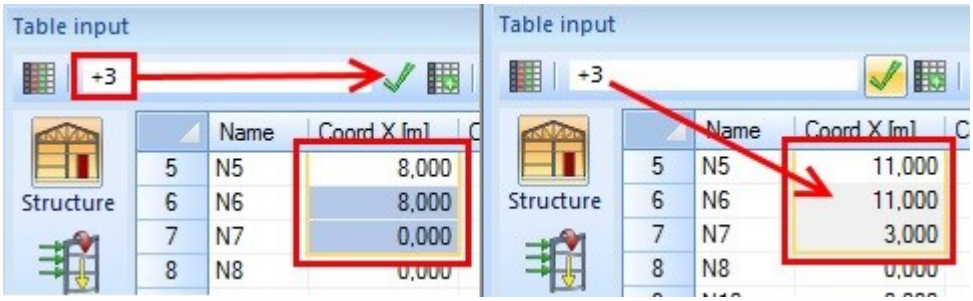


Using simple mathematical formulas in the Table input

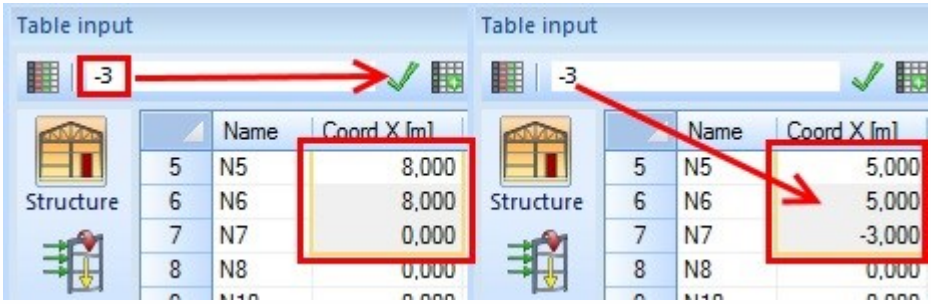
The editbox supports basic mathematical formulas – add, subtract, multiply and divide.

The specific symbol for the formula must be used as the first symbol in the editbox:

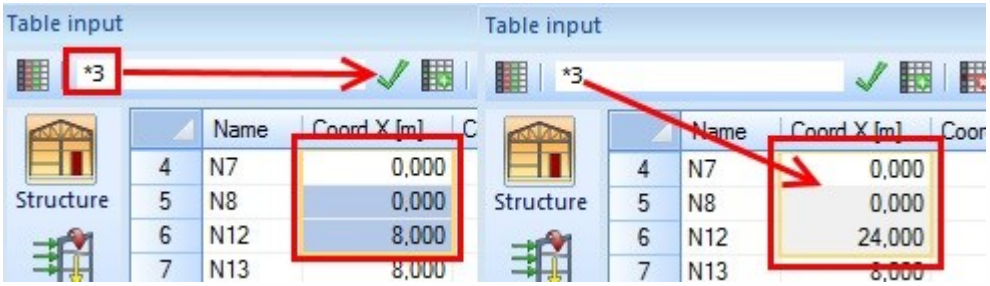
- +3 means that numbers add 3



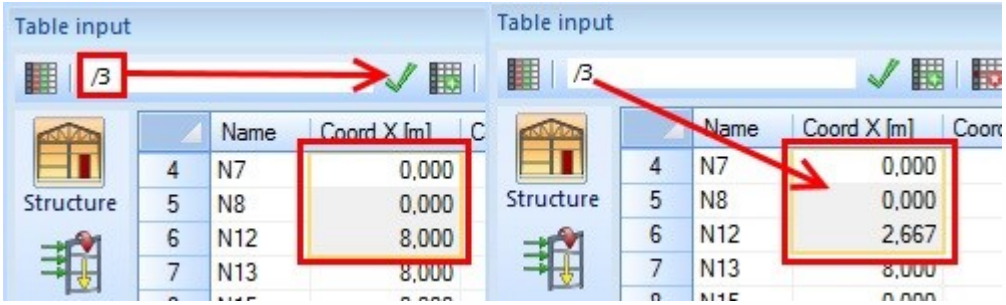
- -3 means that numbers subtract 3



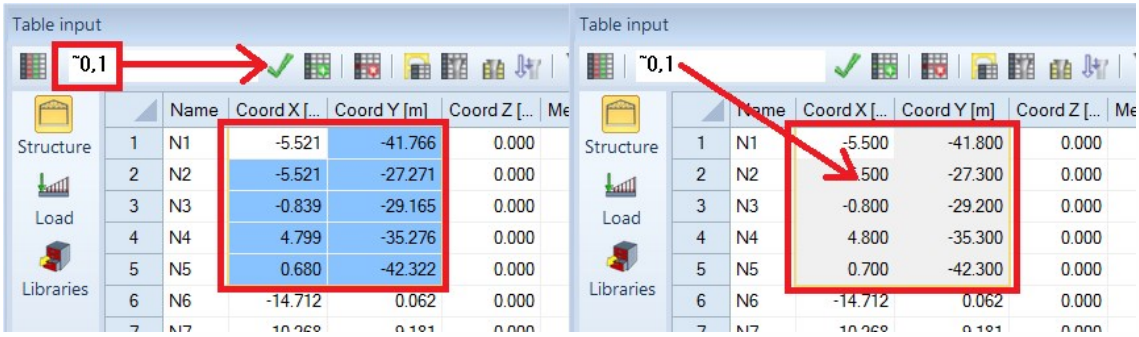
- *3 means that numbers multiply by 3



- /3 means that numbers divide by 3



- ~0,1 means rounding by the 1st decimal



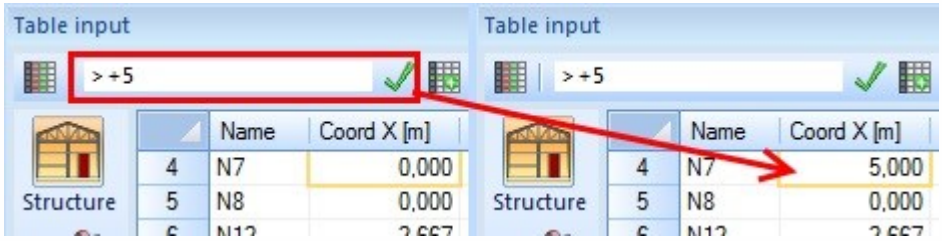
Substitution of the value in the cell

The arrow symbol is used for the substitution (or without any symbol in some cases).

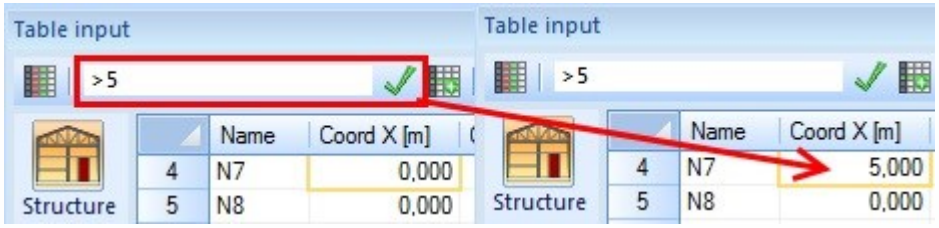
Number substitution

The positive value can be substituted in two ways:

- >+5



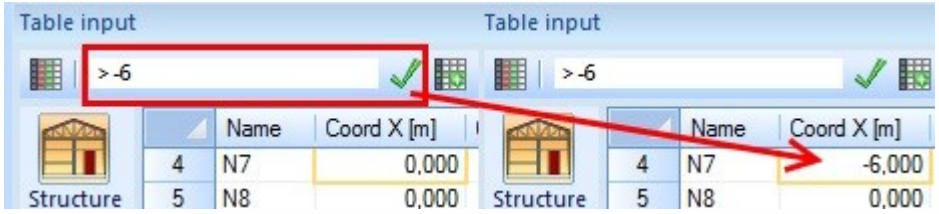
- >5



Both will substitute the number in the cell, the new value will be 5,000.

Negative value:

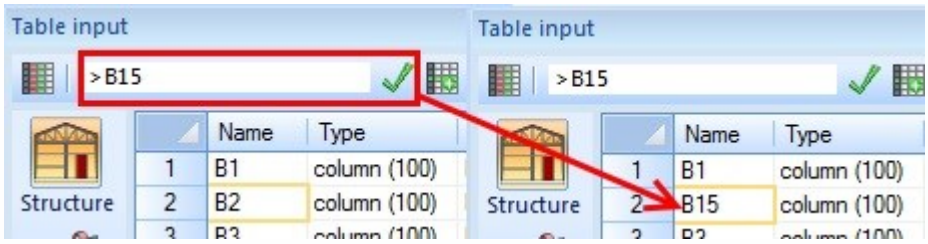
- >-6



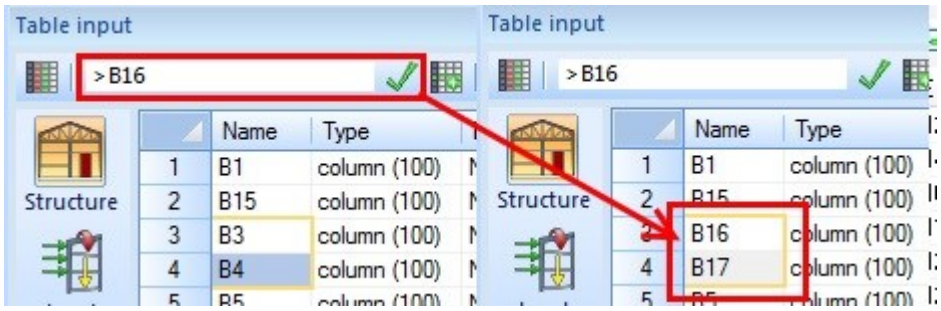
String substitution

String can be substituted in the two ways:

- With symbol: >B15
- Without symbol: B15



- When the substitution is used on more cells together names are automatically renumbered:



B3 is renamed to B16 and B4 is renamed and renumbered automatically to B17

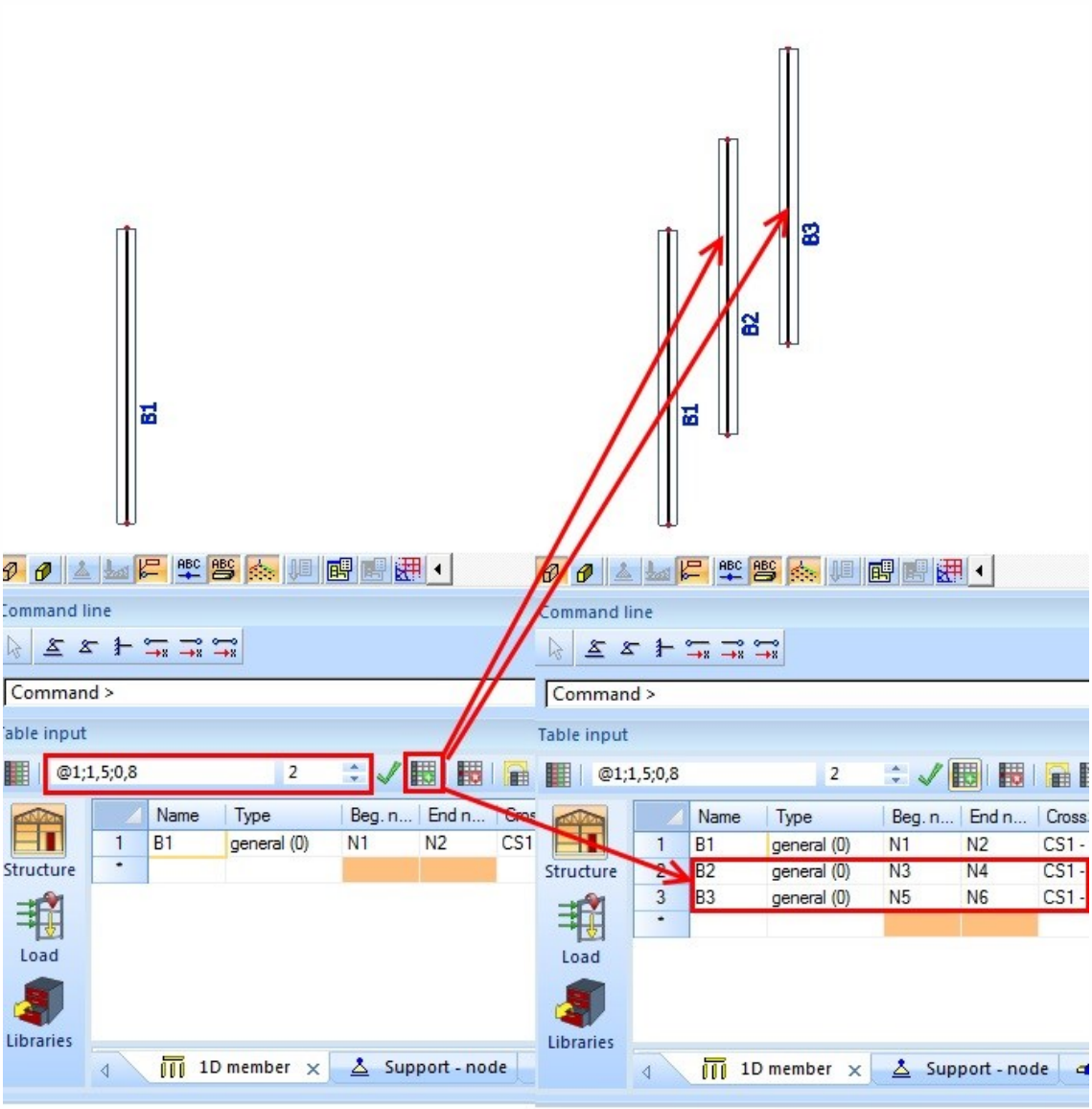
Each entity in the SCIA Engineer has an unique name. B16 can be used when there is no 1D member named this way already. For example new name B2 won't be accepted, its already used.

Offset for copy member

The editbox may be used for copying member. Editbox requires the offset definition. The offset value must be written in a special form - @1 3 5 or @1;3;5. The last settings define the number of copies.

- Symbol @ stands for "copy", 3 numbers define the offset in x, y and z direction.

Example:
@1;1,5;0,8 -> means to copy member with offset 1m in x direction, 1,5m in y direction, and 0,8m in z direction (see the picture).

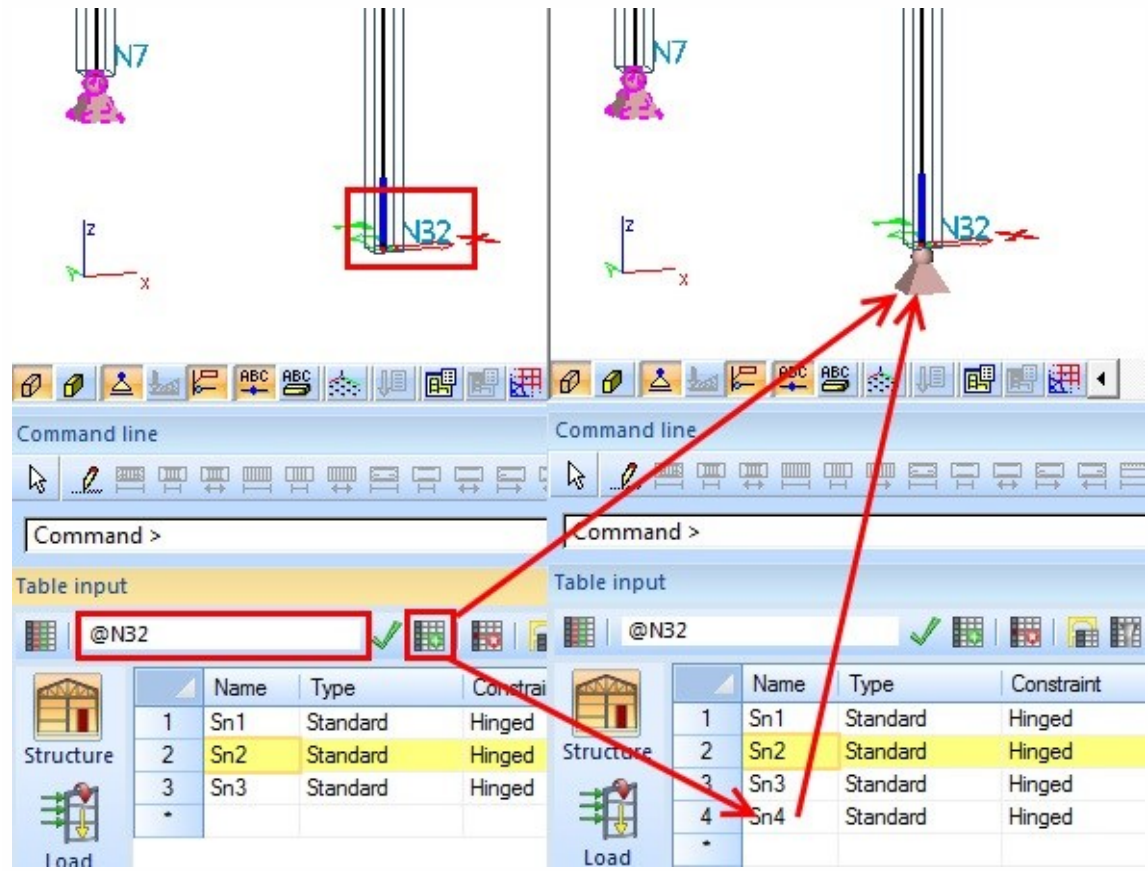


These two notations leads to the same result: @1 2 3 and @1;2;3

Owner for copy add data

If the user want to copy some add data, the definition of the new owner is required. The type of the owner depends on the copied add data (e.g. node name for support in point, member name for line load on beam, etc.).

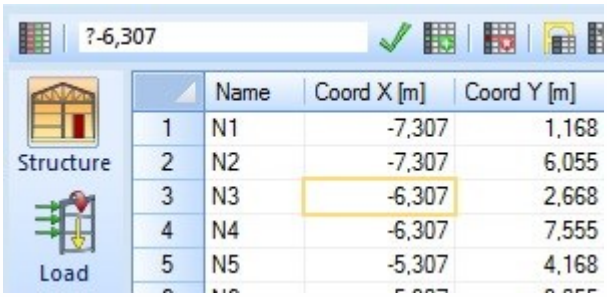
Example:
Copy support in point – owner must be the name of another node;
Copy support on the beam – owner must be the name of another beam;



Search tool

A search tool allows user to find defined value in the table grid. The tool compares the value in the editbox with values in the grid step by step. When the value matches the cell is selected.

A symbol used for the search tool is a question mark – see the picture:



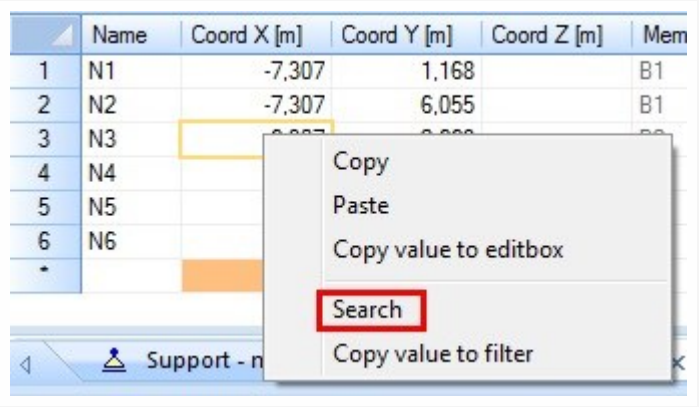
Start searching using short-cuts

- CTRL+F – takes the value from the selected cell and copy it to the multiple editbox. Use key F3 to search the value step by step through the grid.
- CTRL+F3 – takes the value from the selected cell and copy it to the multiple editbox and immediately searches the first exact match. F3 search value to the bottom.

Start searching using manual input

- Put the symbol ? (or use the context menu item Search) to the editbox and write down the required value.
- Press Enter (or F3 key) - the fist cell with exact match is selected in the grid.

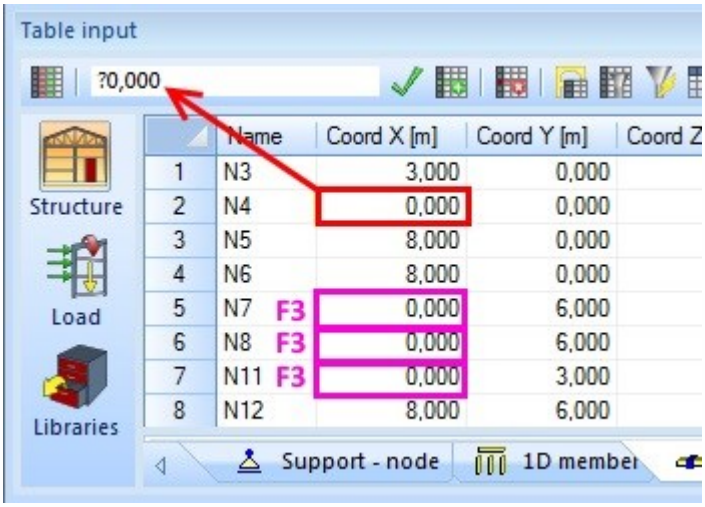
Using the context menu to start searching:



The required value must be exact – e.g. not 0 but 0,000.

Example - how to search value 0,000

1. The first selection is Coord X for N4 - value is 0,000. Use the context menu and select the item Search.
2. The value 0,000 is copied to the editbox..
3. The selection is moved down by F3 key (or Enter key) and it always searches the next cell with 0,000 (it goes through particular columns one by one).

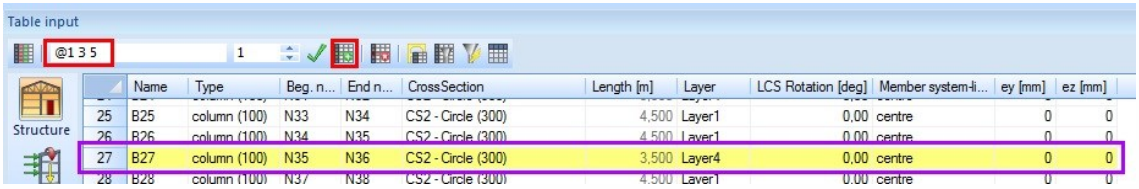


Copy row

The selected row (=member) can be copied by the button on the Table toolbar. The selected row is copied to the bottom of the tab with all properties according to the original row. The offset of the copied member or add data (supports, loads etc.) is defined manually in the editbox.

Copy structural entity

The entities like beams, slabs or load panels can be copied with predefined offset. Selected row is copied to the bottom. Offset must be defined in the multiple editbox with symbol @ (see the next chapter about multiple editbox).



Red colour - offset and "Copy row" button;

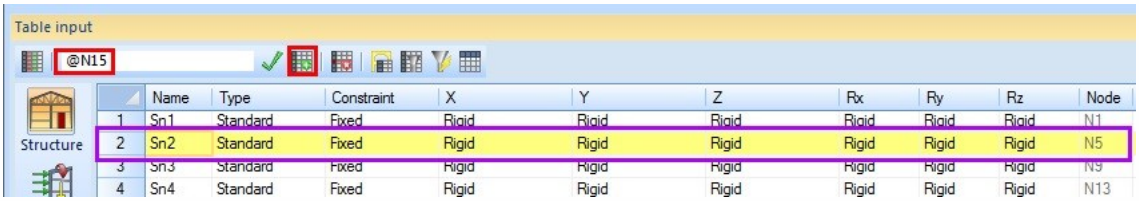
Magenta colour - selected row for a copy process;

How to copy a member:

1. Editbox: write down the symbol @ and the offset for a new member.
2. Select the row which should be copied.
3. Click on the button "Copy row".

Copy add data

Add data is defined by their properties and by the owner. This owner must be predefined for Copy row. New owner for the add data is predefined by the editbox with symbol @.



Red colour - owner for a selected add data and "Copy row" button;

Magenta colour - selected row for a copy process;

The owner is specified by its name:

- B12 – for a beam
- N35 – for a node
- S11 – for a slab

How to copy add data:

1. Editbox: write down the symbol @ and the name of a new owner for add data.
2. Select row with add data which should be copied.
3. Click on the button "Copy row".

See chapter Multiple editbox to find more info about the editbox and how to use it.

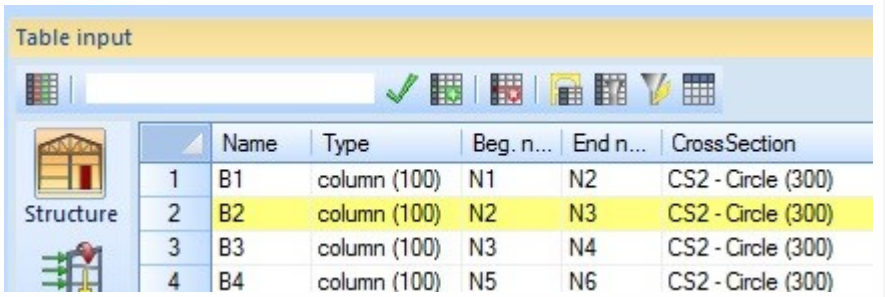
Delete row

The selected rows can be deleted by the button "Delete row" on the toolbar.

How to delete row/s (= member/s):

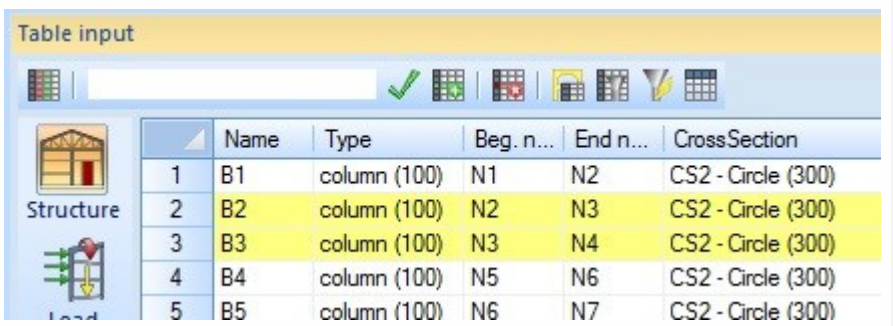
1. Select the row and click the button "Delete row".

Example with B2 for deleting:



2. Or select more rows and click the button "Delete row".

Example with B2 and B3 for deleting:



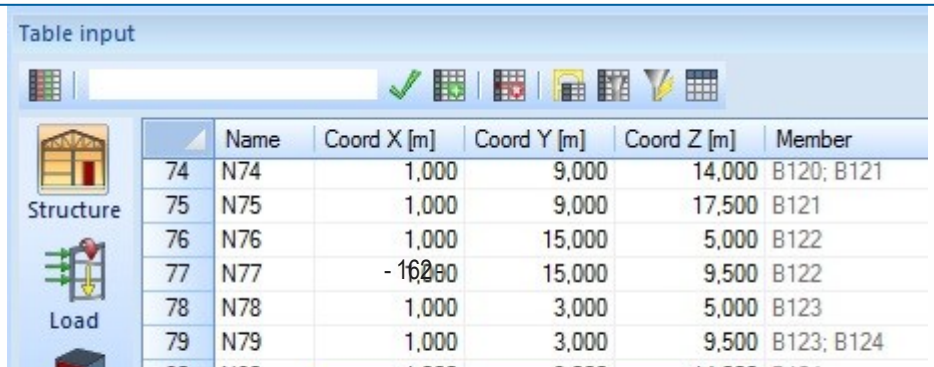
New row (= new member)

A new geometry and entities can be created by the Table input. This functionality is provided by a New row in tabs. It is a row with symbol * in the header.

The new row contains orange cells and white cells.

- Orange cells = Required cells;
  These cells define the geometry – coordinates or an owner for add data.
- White cells = Optional cells;
  These cells enable predefining optional properties (layer, system line, offset, thickness ...).

Example - required cells for the tab Node. The node is defined by X, Y and Z coordinates so it must be defined to be able to create a new one.



The new row is not available in each tab. E.g. it is not in the tab Layer.

Required cells examples

Coordinates for the structural node:

	Name	Coord X [m]	Coord Y [m]	Coord Z [m]	Member
4	N4	2,991	1,421	3,600	B2
5	N5	5,570	1,421	-0,184	B3
6	N6	5,570	1,421	3,416	B3
7	N7	6,730	-3,429	-0,184	B4
8	N8	6,730	-3,429	3,416	B4
*		2	3	5	

Node | 1D member | 2D member | Load panels | Support - node

Begin and End node for 1D member:

	Name	Type	Beg. n...	End n...	CrossSection
1	B1	column (100)	N1	N2	CS2 - Rectangle (500; 300)
2	B2	column (100)	N3	N4	CS1 - Circle (400)
3	B3	column (100)	N5	N6	CS2 - Rectangle (500; 300)
4	B4	column (100)	N7	N8	CS1 - Circle (400)
5	B5	beam (80)	N2	N4	CS2 - Rectangle (500; 300)
*			N5	N1	

Node | 1D member | 2D member | Load panels | Support - node | Support -

The owner for the Force in node:

	Name	Type	Direction	Node	Angle [deg]	Value - F [kN]	Load case	System	Wind pressure c...
1	F1	Wind	Z	N1		0,70	LC2	GCS	WP2
2	F2	Force	Z	N2		-1,00	LC2	GCS	
3	F3	Wind	X	N3		0,70	LC2	GCS	WP2
*				N5					

In the case of creating 2D members and load panels, the only mandatory input is the geometry. The geometry is defined by a sequence of nodes separated by a semicolon ";" or coma ",". When a 2D entity is being created using the Table input, the validity of inputted data is verified and invalid entities such as entities with crossing edges or entities defined by nodes that are not in a plane are not created.

	Name	Type	Material	Layer	Thickness [...	LCS angle [...	Node	Edge	Analysis ...
3	S3	wall (80)	S 235	Layer1	200	0.00	N7; N10; N11; N8	Line; Line; Line; Line	Standard
4	S5	plate (90)	S 235	Layer1	200	0.00	N16; N17; N18	Line; Line; Line	Standard
5	S6	wall (80)	S 235	Layer1	200	0.00	N23; N29; N30; N24	Line; Line; Line; Line	Standard
6	S7	wall (80)	S 235	Layer1	200	0.00	N29; N31; N32; N30	Line; Line; Line; Line	Standard
7	S8	shell (98)	S 235	Layer1	200	0.00	N52; N53; N54; N55	Line; Line; Line; Line	Standard
8	S9	plate (90)	S 235	Layer1	200	0.00	N56; N57; N58; N59	Line; Line; Line; Line	Standard
9	S10	plate (90)	S 235	Layer1	200	0.00	N60; N61; N62; N63	Line; Line; Line; Line	Standard
10	S11	plate (90)	S 235	Layer1	200	0.00	N86; N97; N98; N87; N99	Line; Line; Line; Line; L...	Standard
*									

Node | 1D member | 2D member | Opening | Subregion | Load panels | Support - node | Support - beam | Line support - beam | Line support - ed

Subregions and openings are 2D entities similar to 2D members, however there is additional mandatory input field "2D member" which represents the name of the 2D member in which the opening or subregion is being defined. For the definition of geometry of openings and subregions the same validity rules apply as in the case of 2D members and load panels. Additionally, subregions and openings have to be defined in the same plane as the 2D member in which they are created.

	Name	Material	FEM model	FEM nonline...	Thickness ty...	Thickness [...	Member syst...	Eccentricity z ...	2D me...	Node	Edge
1	R1	S 235	Isotropic	none	constant	600	Centre	0	S2	N19; N20; N21; N...	Line; Line; Line; L...
2	R2	S 235	Isotropic	none	constant	500	Centre	0	S1	N77; N78; N79	Line; Line; Line
3	R3	S 235	Isotropic	none	constant	800	Centre	0	S1	N83; N84; N85	Line; Line; Line
4	R4	S 235	Isotropic	none	constant	800	Centre	0	S5	N46; N47; N48	Line; Line; Line
5	R5	S 235	Isotropic	none	constant	800	Centre	0	S10	N64; N65; N66	Line; Line; Line
6	R6	S 235	Isotropic	none	constant	800	Centre	0	S8	N70; N71; N72	Line; Line; Line
7	R3a	S 235	Isotropic	none	constant	500	Centre	0	S11	N94; N95; N96	Line; Line; Line
8	R2a	S 235	Isotropic	none	constant	800	Centre	0	S11	N91; N92; N93	Line; Line; Line
*											

Node | 1D member | 2D member | Opening | Subregion | Load panels | Support - node | Support - beam | Line support - beam | Line support - ed

	Name	Panel	Cut 1D me...	Convert into 1...	Edge	Wei...	2D me...	Node	Edge
1	O1	☐	☐	☐			S1	N80; N81; N82	Line; Line; Line
2	O2	☐	☐	☐			S5	N49; N50; N51	Line; Line; Line
3	O3	☐	☐	☐			S10	N67; N68; N69	Line; Line; Line
4	O4	☐	☐	☐			S8	N73; N74; N75	Line; Line; Line
5	O5	☐	☐	☐			S3	N7; N10; N76	Line; Line; Line
6	O6	☐	☐	☐			S11	N88; N89; N90	Line; Line; Line
*									

Node | 1D member | 2D member | Opening | Subregion | Load panels | Support - node | Support - beam | Line support - beam | Line support - ed

How to predefine an optional property

The user can predefine optional properties in white cells. Optional property must be filled in before the required cells are defined. Otherwise its not taken into account.

The example with 1D member:

- The user predefines Type, Cross-section, Layer, LCS rotation and the Member system line position;

	Name	Type	Beg. n...	End n...	CrossSection	Length [m]	Layer	LCS Rotation [deg]	Member system-li...	ey [mm]	ez [mm]
1	B1	column (100)	N1	N2	CS2 - Rectangle (500; 300)	3,600	Layer1	0,00	centre	0	0
2	B2	column (100)	N3	N4	CS1 - Circle (400)	3,600	Layer2	0,00	centre	0	0
3	B3	column (100)	N5	N6	CS2 - Rectangle (500; 300)	3,600	Layer1	0,00	centre	0	0
4	B4	column (100)	N7	N8	CS1 - Circle (400)	3,600	Layer2	0,00	centre	0	0
*		beam (80)			CS2 - Rectangle (500; 300)		Layer1	20	top		

- The required cells are filled with N2 and N4 afterwards;
- The result is beam B5:

	Name	Type	Beg. n...	End n...	CrossSection	Length [m]	Layer	LCS Rotation [deg]	Member system-li...	ey [mm]	ez [mm]
1	B1	column (100)	N1	N2	CS2 - Rectangle (500; 300)	3,600	Layer1	0,00	centre	0	0
2	B2	column (100)	N3	N4	CS1 - Circle (400)	3,600	Layer2	0,00	centre	0	0
3	B3	column (100)	N5	N6	CS2 - Rectangle (500; 300)	3,600	Layer1	0,00	centre	0	0
4	B4	column (100)	N7	N8	CS1 - Circle (400)	3,600	Layer2	0,00	centre	0	0
5	B5	beam (80)	N2	N4	CS2 - Rectangle (500; 300)	4,439	Layer1	20,00	top	0	0
*											

- All predefined properties (optional and required) are valid so it is accepted. The rest of the properties are filled by the default settings (the same as we have in a standard inserting dialogue).

Some properties are dependent on others. In this case the master property is required.

The example with Force in node:

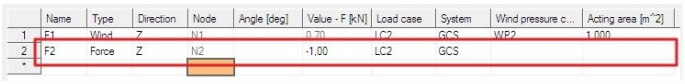
- The Wind pressure curve is dependent on the Type = Wind.

	Name	Type	Direction	Node	Angle [deg]	Value - F [kN]	Load case	System	Wind pressure c...	Acting area [m^2]
1	F1	Wind	Z	N1		0,70	LC2	GCS	WP2	1,000
*										

- What happens when *Type is not predefined* (the default settings with Type Force will be used instead):

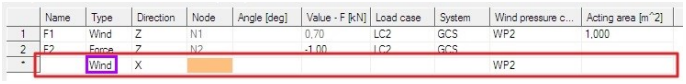
	Name	Type	Direction	Node	Angle [deg]	Value - F [kN]	Load case	System	Wind pressure c...	Acting area [m^2]
1	F1	Wind	Z	N1		0,70	LC2	GCS	WP2	1,000
*			Z						WP2	

- Result: The Wind pressure curve is not accepted. The Type Force is created. Only predefined optional property Z direction is accepted as a valid value.



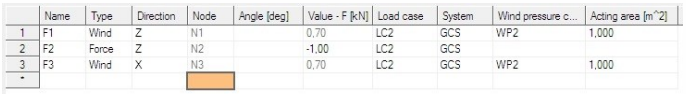
	Name	Type	Direction	Node	Angle [deg]	Value - F [kN]	Load case	System	Wind pressure c...	Acting area [m^2]
1	F1	Wind	Z	N1		0,70	LC2	GCS	WP2	1,000
2	F2	Force	Z	N2		-1,00	LC2	GCS		
*										

- What happens when the *Type Wind is predefined*:



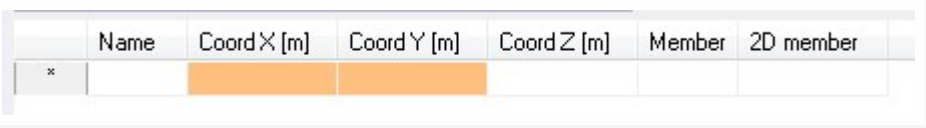
	Name	Type	Direction	Node	Angle [deg]	Value - F [kN]	Load case	System	Wind pressure c...	Acting area [m^2]
1	F1	Wind	Z	N1		0,70	LC2	GCS	WP2	1,000
2	F2	Force	Z	N2		-1,00	LC2	GCS		
*		Wind	X						WP2	

- Result: The Wind pressure curve and Direction X is accepted as valid values. The Type Wind is created as a new force.



	Name	Type	Direction	Node	Angle [deg]	Value - F [kN]	Load case	System	Wind pressure c...	Acting area [m^2]
1	F1	Wind	Z	N1		0,70	LC2	GCS	WP2	1,000
2	F2	Force	Z	N2		-1,00	LC2	GCS		
3	F3	Wind	X	N3		0,70	LC2	GCS	WP2	1,000
*										

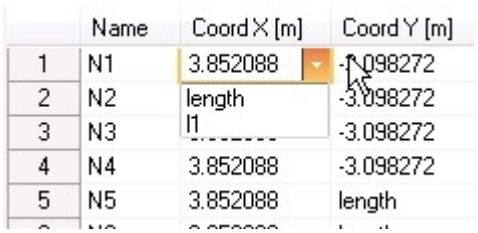
The tow required cells are displayed in 2D type of project (e.g. Wall XY), The third coordinate is displayed with white colour and it is not required.



	Name	Coord X [m]	Coord Y [m]	Coord Z [m]	Member	2D member
*						

Parameters in Table input

It is also possible to work with the parametrized project in the Table input. Parametric values are displayed in the same way as in the property dialogue – comboboxes with all appropriate parameters and possibility to put value in it.



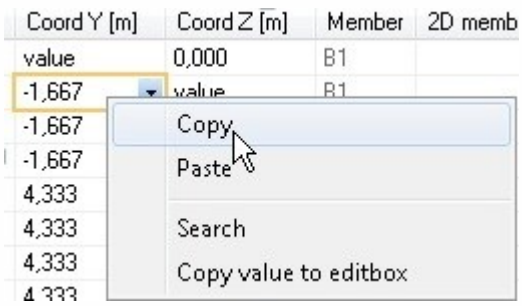
	Name	Coord X [m]	Coord Y [m]
1	N1	3.852088	-3.098272
2	N2	length	-3.098272
3	N3	l1	-3.098272
4	N4	3.852088	-3.098272
5	N5	3.852088	length

Combobox contains only parameters.

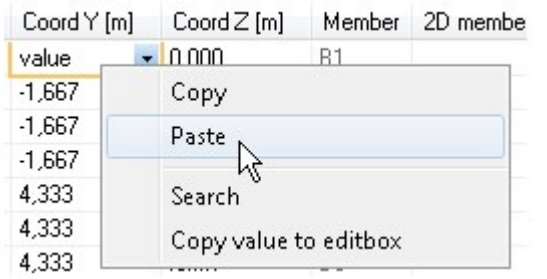
The number can be changed by editing the cell or paste the value from the clipboard.

Example how to paste the number to the cell with the parameter:

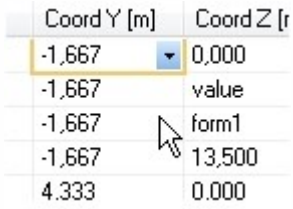
1. Copy the value from another cell or from XLS file.



2. Paste the value to the cell with parameter “length”.



3. The cell is changed to the number.



Tips and Tricks

All changes made in the Table input must be accepted by the model geometry.

Member moving

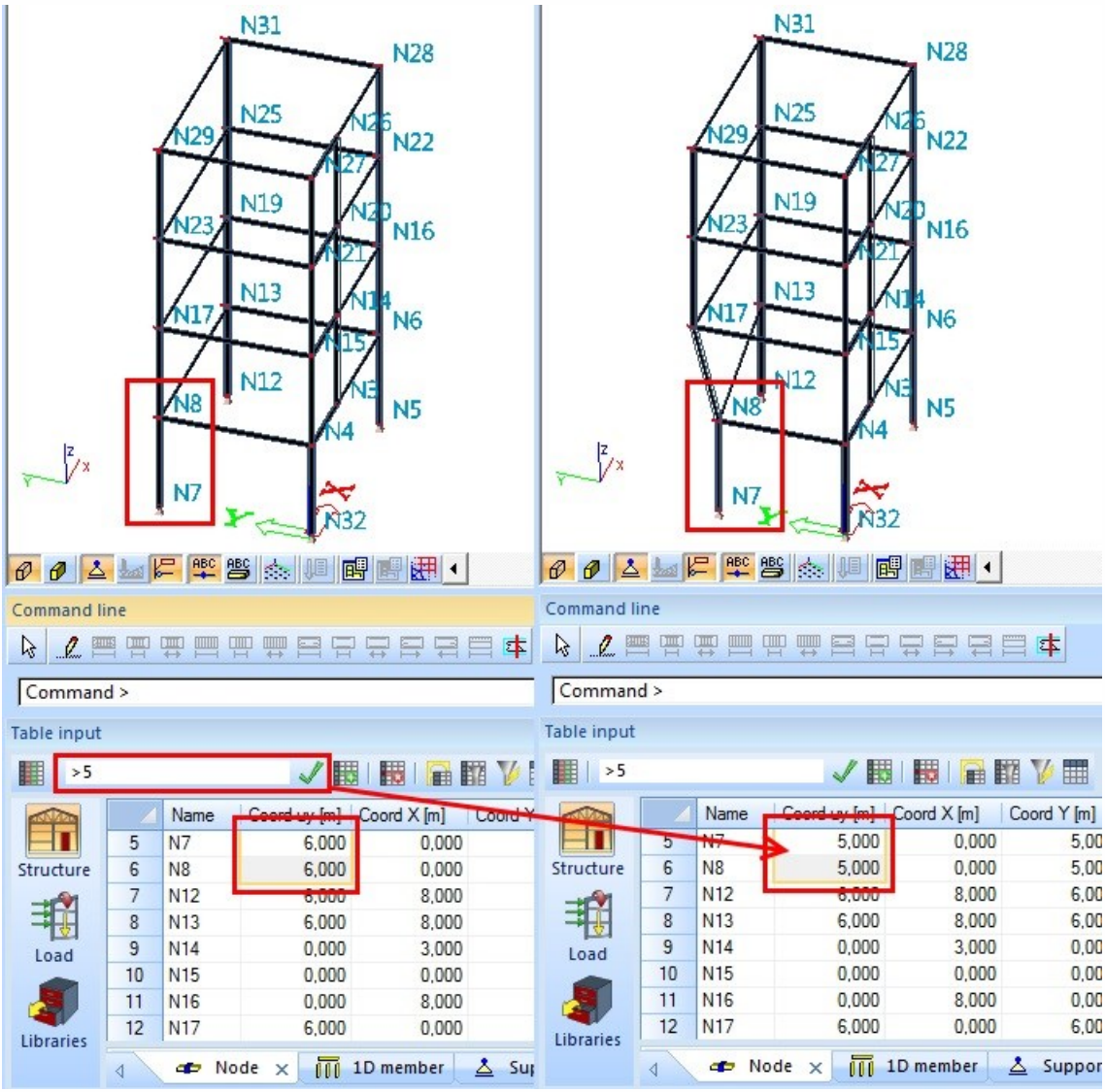
The position of a member depends on its nodes. It is possible to edit the position of nodes and tune the position of the member. It is possible to move more members together.

- One way is to change the value coord x, y, z;
- Or change value coord ux, uy, uz;

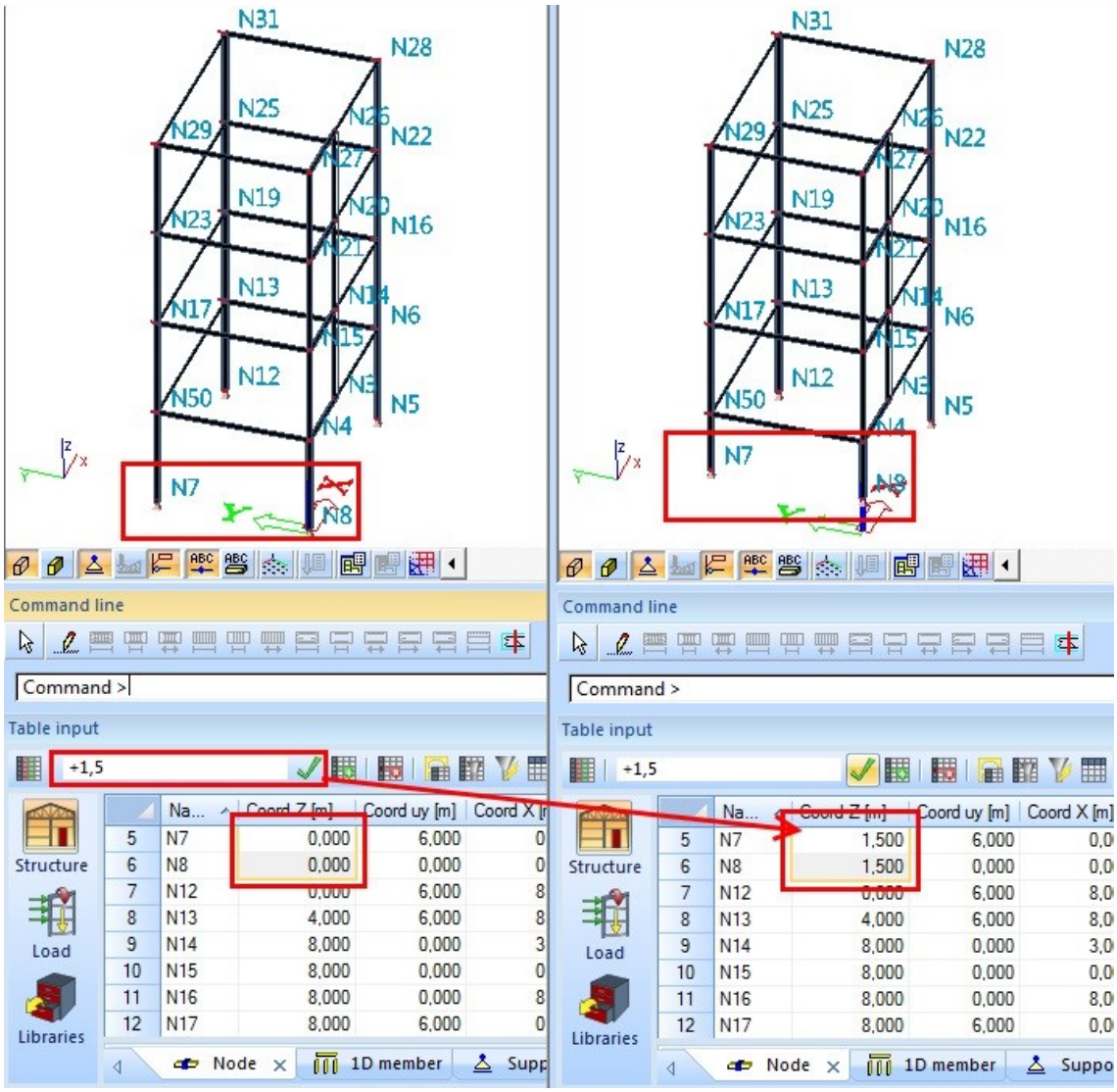
The change can be done by the substitution or using simple formulas.

Example on the picture:

- Using the substitution >5 to the cells for coord uy -> two columns are moved in y direction;



- Add +1,5 to the cells coord Z -> two columns are moved upward;

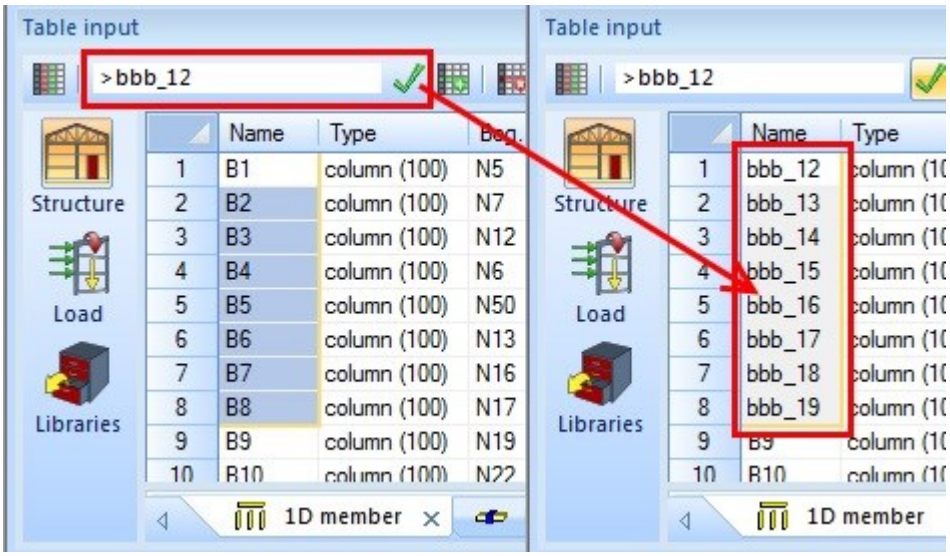


Multi – renaming and renumbering (names)

The editbox is a tool for editing values in the Table input. Renaming and renumbering is important example.

How to rename and renumber more beams together:

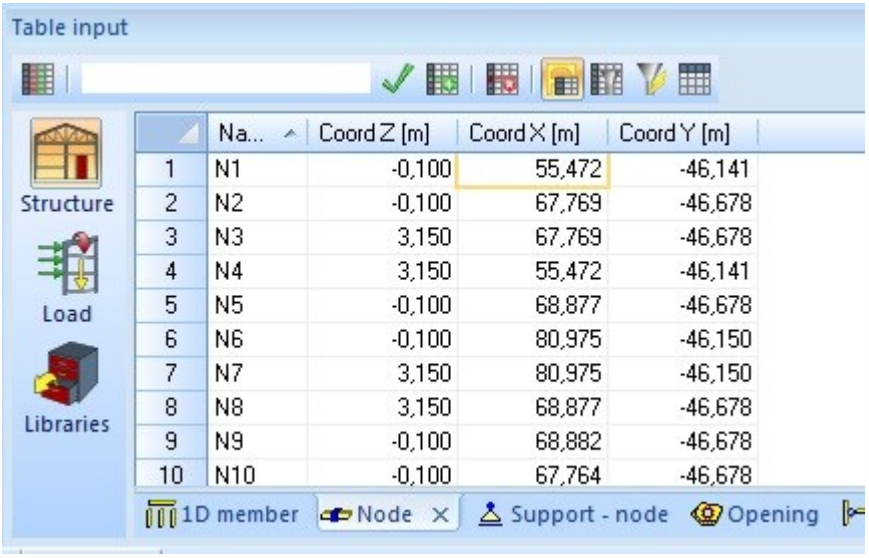
1. The aim is to rename beams to bbb_12, bbb_13, bbb_14, etc. Put the first required string to the editbox (bbb_12).
2. Select cells with beam names which should be changed – click and drag from the first cell to the last one.
3. Click on the Apply button next to the Multiple editbox.
4. Cells are renamed from bbb_12 to the last one bbb_19.



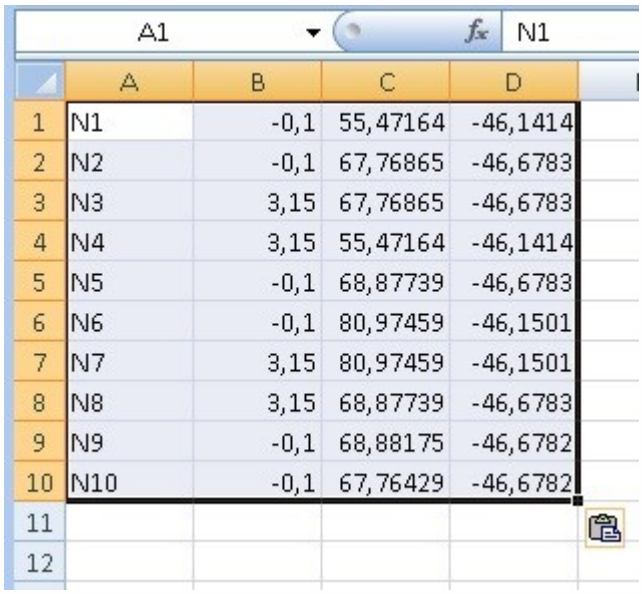
Rounding values

Many imported models are inaccurate. The values need rounding. This can be easily fixed using Table input and MS Excel.

1. Copy the content of the node tab to the clipboard and paste it to MS Excel (select cells and use CTRL+C in Table input):

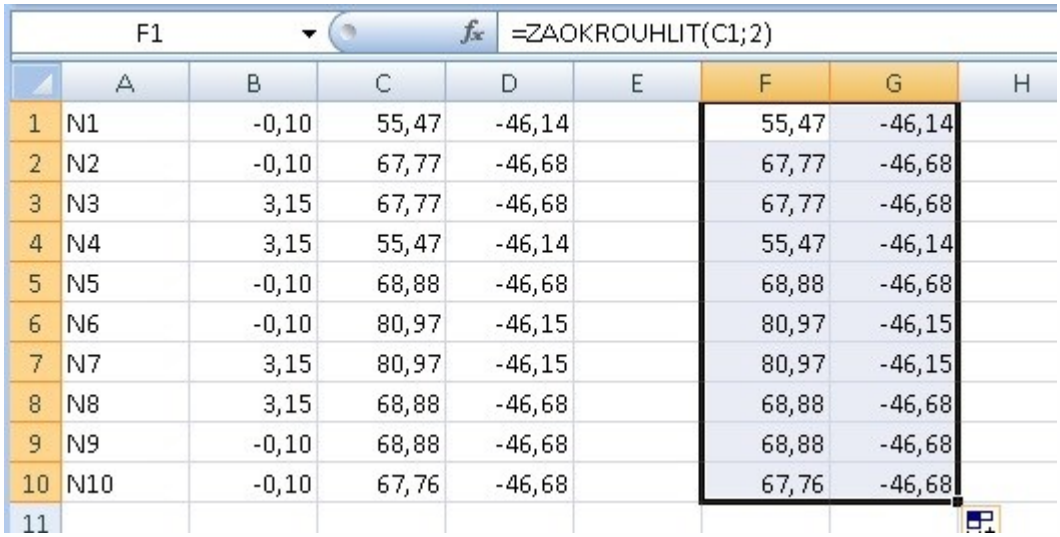


2. Use functionality for rounding:



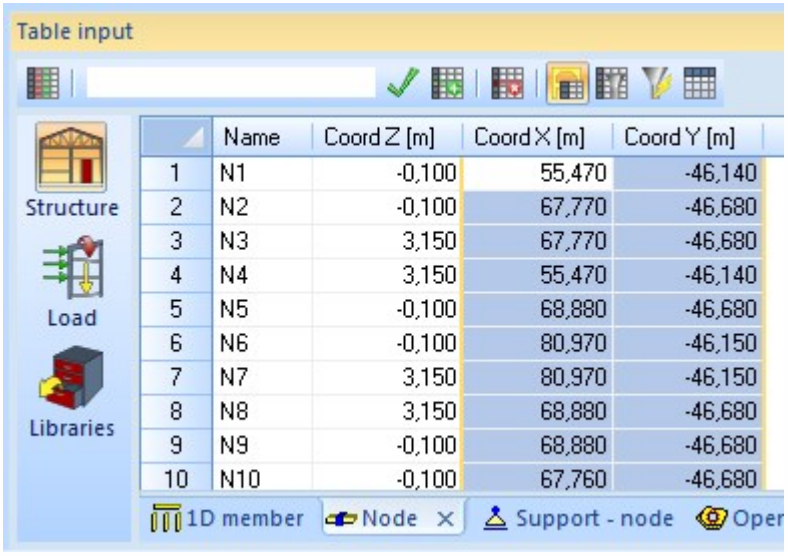
	A	B	C	D
1	N1	-0,1	55,47164	-46,1414
2	N2	-0,1	67,76865	-46,6783
3	N3	3,15	67,76865	-46,6783
4	N4	3,15	55,47164	-46,1414
5	N5	-0,1	68,87739	-46,6783
6	N6	-0,1	80,97459	-46,1501
7	N7	3,15	80,97459	-46,1501
8	N8	3,15	68,87739	-46,6783
9	N9	-0,1	68,88175	-46,6782
10	N10	-0,1	67,76429	-46,6782
11				
12				

The result:



	A	B	C	D	E	F	G	H
1	N1	-0,10	55,47	-46,14		55,47	-46,14	
2	N2	-0,10	67,77	-46,68		67,77	-46,68	
3	N3	3,15	67,77	-46,68		67,77	-46,68	
4	N4	3,15	55,47	-46,14		55,47	-46,14	
5	N5	-0,10	68,88	-46,68		68,88	-46,68	
6	N6	-0,10	80,97	-46,15		80,97	-46,15	
7	N7	3,15	80,97	-46,15		80,97	-46,15	
8	N8	3,15	68,88	-46,68		68,88	-46,68	
9	N9	-0,10	68,88	-46,68		68,88	-46,68	
10	N10	-0,10	67,76	-46,68		67,76	-46,68	
11								

3. Copy the content back to the Table input. All values are rounded.



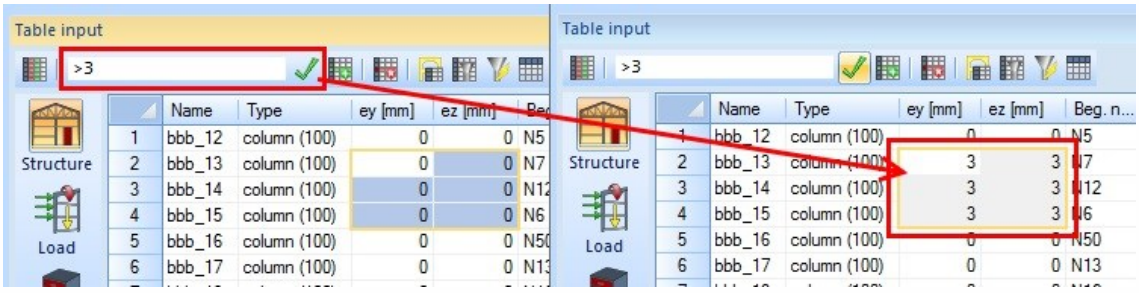
	Name	Coord Z [m]	Coord X [m]	Coord Y [m]
1	N1	-0,100	55,470	-46,140
2	N2	-0,100	67,770	-46,680
3	N3	3,150	67,770	-46,680
4	N4	3,150	55,470	-46,140
5	N5	-0,100	68,880	-46,680
6	N6	-0,100	80,970	-46,150
7	N7	3,150	80,970	-46,150
8	N8	3,150	68,880	-46,680
9	N9	-0,100	68,880	-46,680
10	N10	-0,100	67,760	-46,680

Editing more items using multiple editbox

There is a possibility to apply one change on more selected cells together. Use multiple editbox for this purpose.

How to do it:

1. Put the required value to the editbox alone or with the symbol >.
2. Multiselect cells – click and drag from the first cell to the last one.
3. Click on the Apply button next to the Multiple editbox.
4. All selected cells are changed (e.g. changed to 3, see the picture).



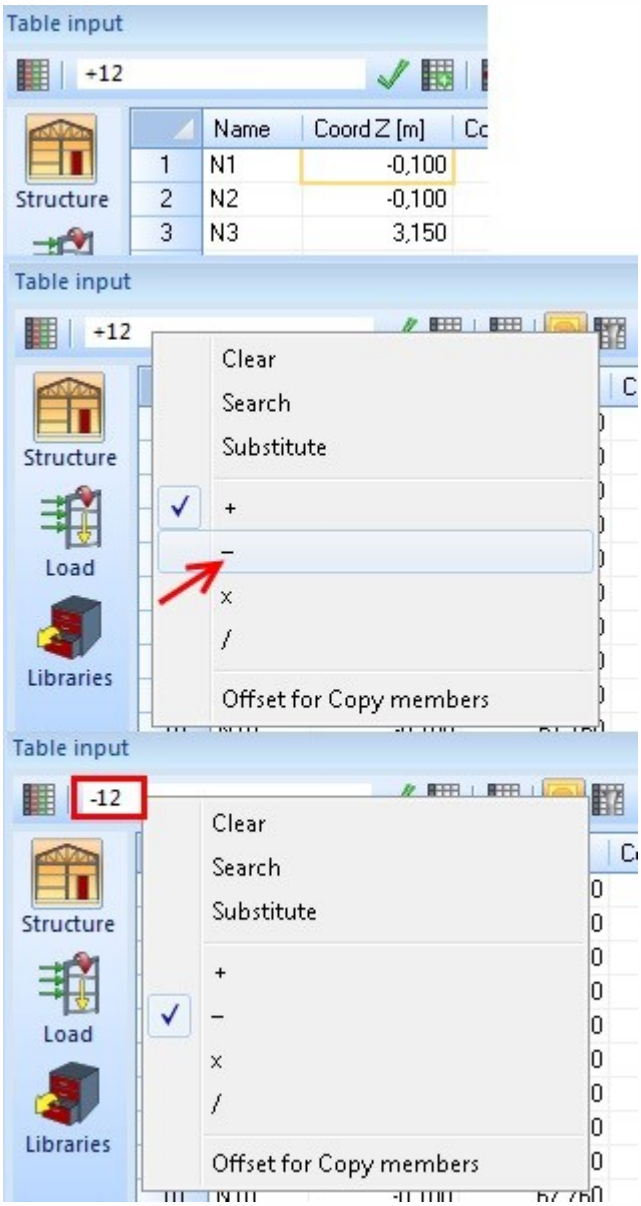
Advantage of the multiple-editbox right-click menu

There are tow ways how to change the symbol for the required operation in the multiple editbox.

- manually - rewrite the symbol (e.g. + to -);
- using right-click menu - select the required operation and the symbol is changed automatically;

Appropriate symbols are displayed in the context menu. It can be used as a help.

See example:



Using filter and activity in Table input

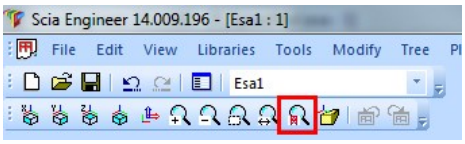
It is possible to use the filter row and switch the activity from the 3D window ON. User has more options to filter elements which he needs.

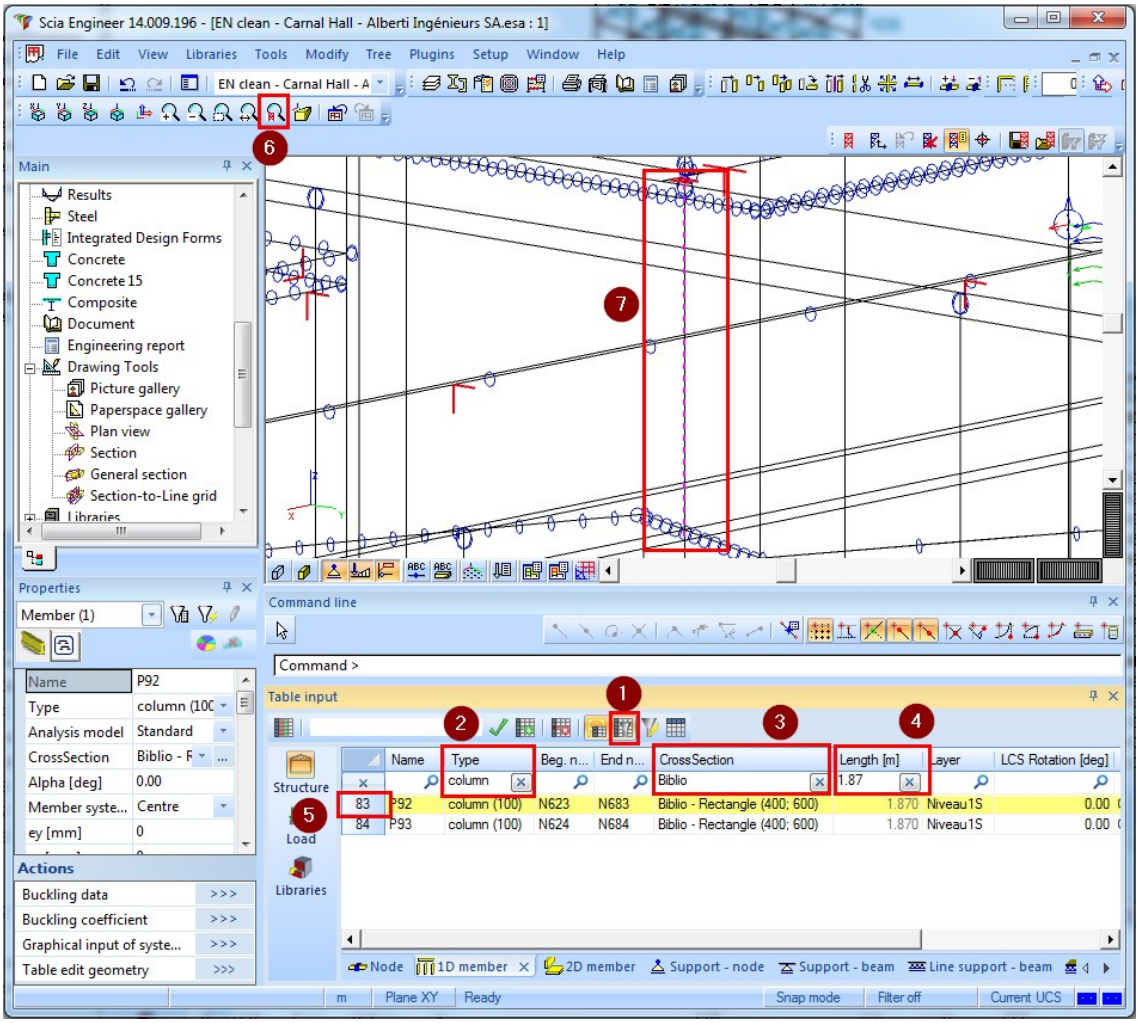
When user highlights a member in the table it is selected automatically in the model (dashed magenta line). The member may be displayed more visible using Activity by selection. This is very advantageous for smaller elements.

Zoom to selection + selection link in Table input

It is very useful tu use the "Zoom to selection" command together with selection in Table input.

If you want to locate some entity in large project, just use the searching or filtering row in Table input, select the entity and push the button "Zoom to selection" (on the "Selection of object" toolbar). The selected entity (which is unique based on some properties filters) is nicely located in the model.

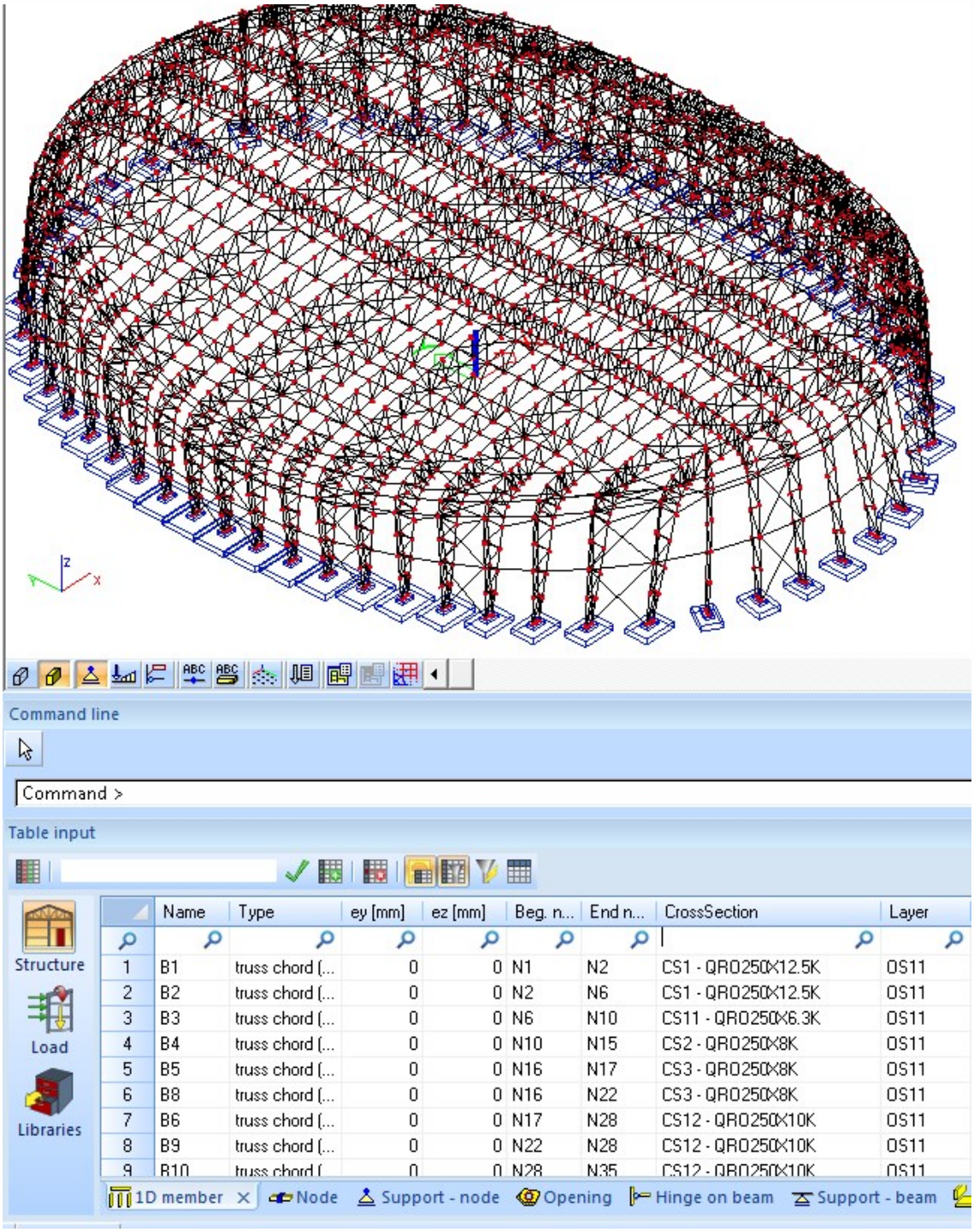




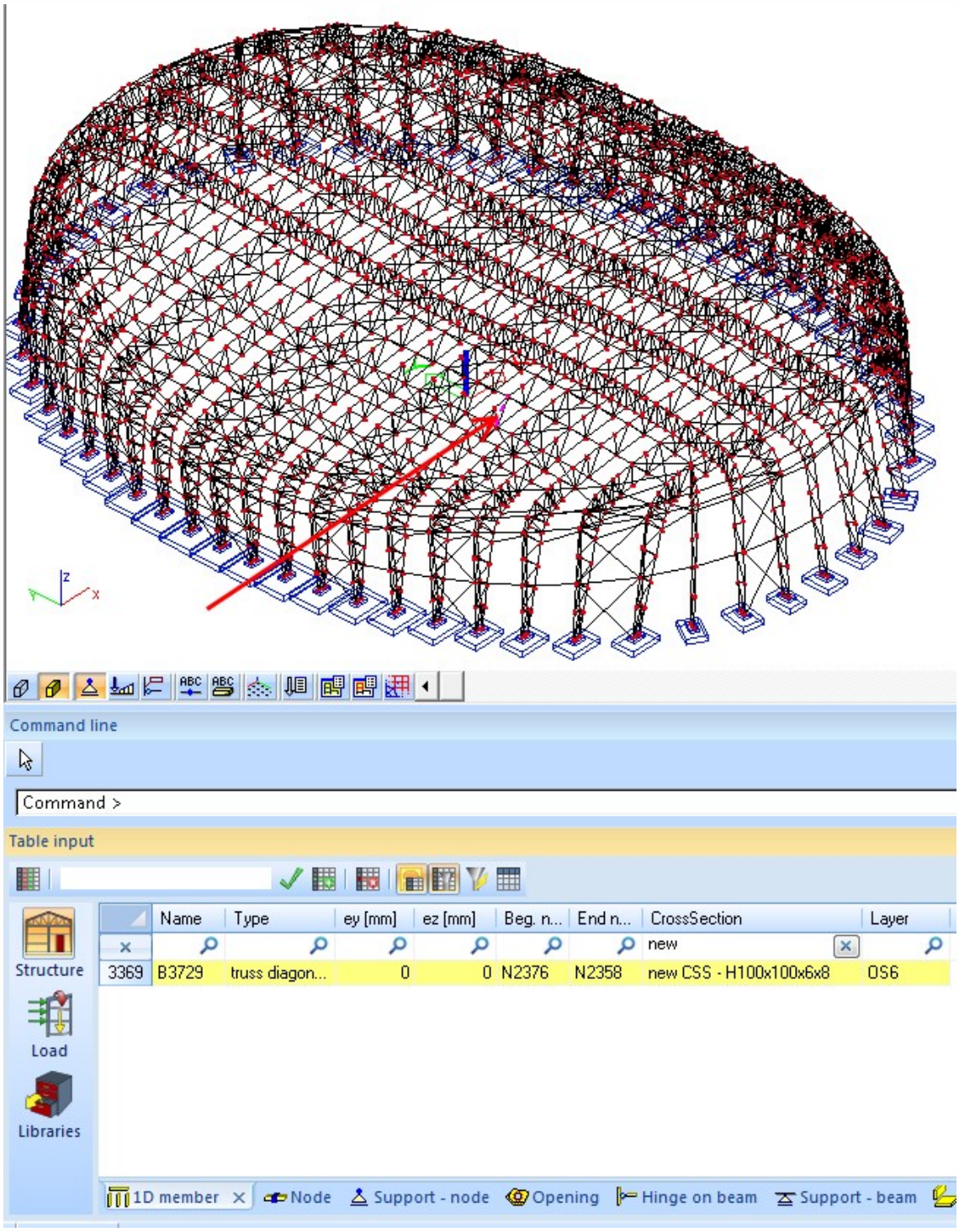
Searching for a cross-section on a big structure

The larger structures generally contain a lot of cross-sections. Sometimes it can be difficult to find one specific cross section. The Table input may help in this kind of searching.

1. Open 1D member tab.
2. Start the filter row, it is displayed with magnificent glasses.
3. Write the name of the cross section to the cell with magnificent glass in the column Cross section.



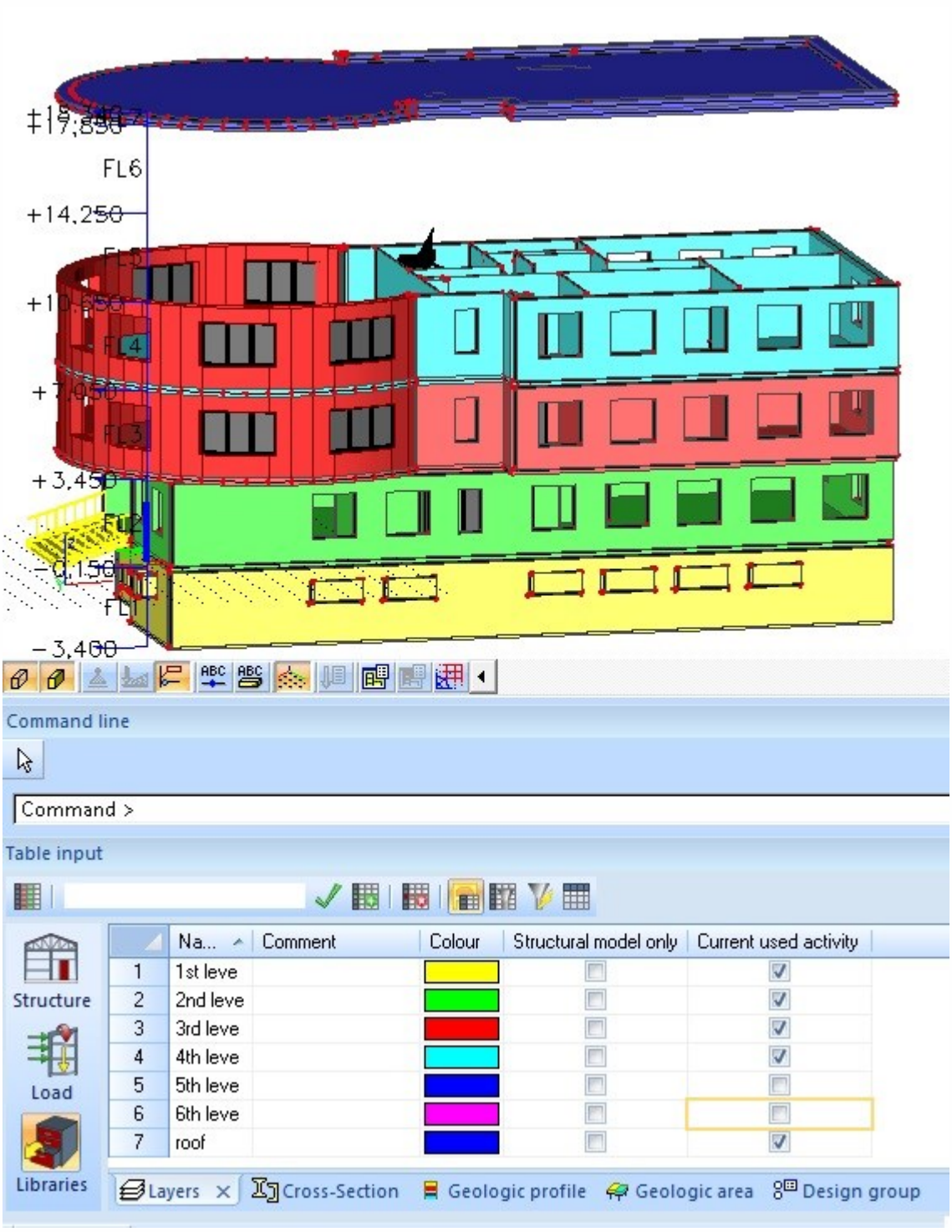
4. Highlight the row with the required element - click on the row header.



5. The element is selected in the 3D window and its properties are displayed in the property dialogue. It can be displayed separately using the activity by selection in the 3D window.

Using tab with layer for activity

If user want to use the activity by layers it is very helpful to display the tab with layers in the Table input (group Libraries). User can change the activity right here. The layer settings are visible here all the time and user is not forced to open the Layer library when he needs to change the activity by layers.

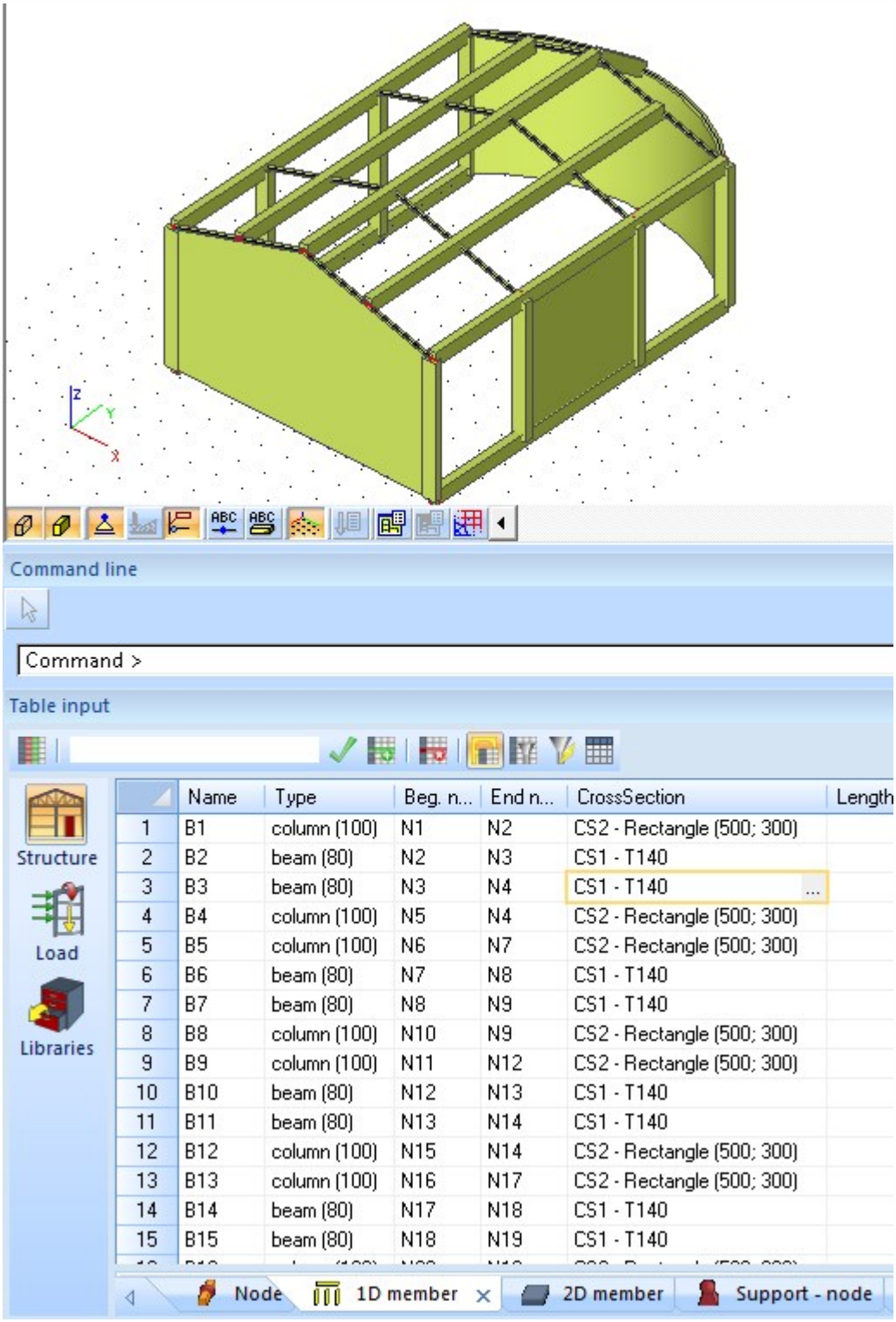


Highlight (=select) members according to the selected property in the cell

It is possible to select all members with the same property value using Table input by one click. This functionality is similar to filtering the 3D model by the selected property in the property dialogue (the icon with funnel).

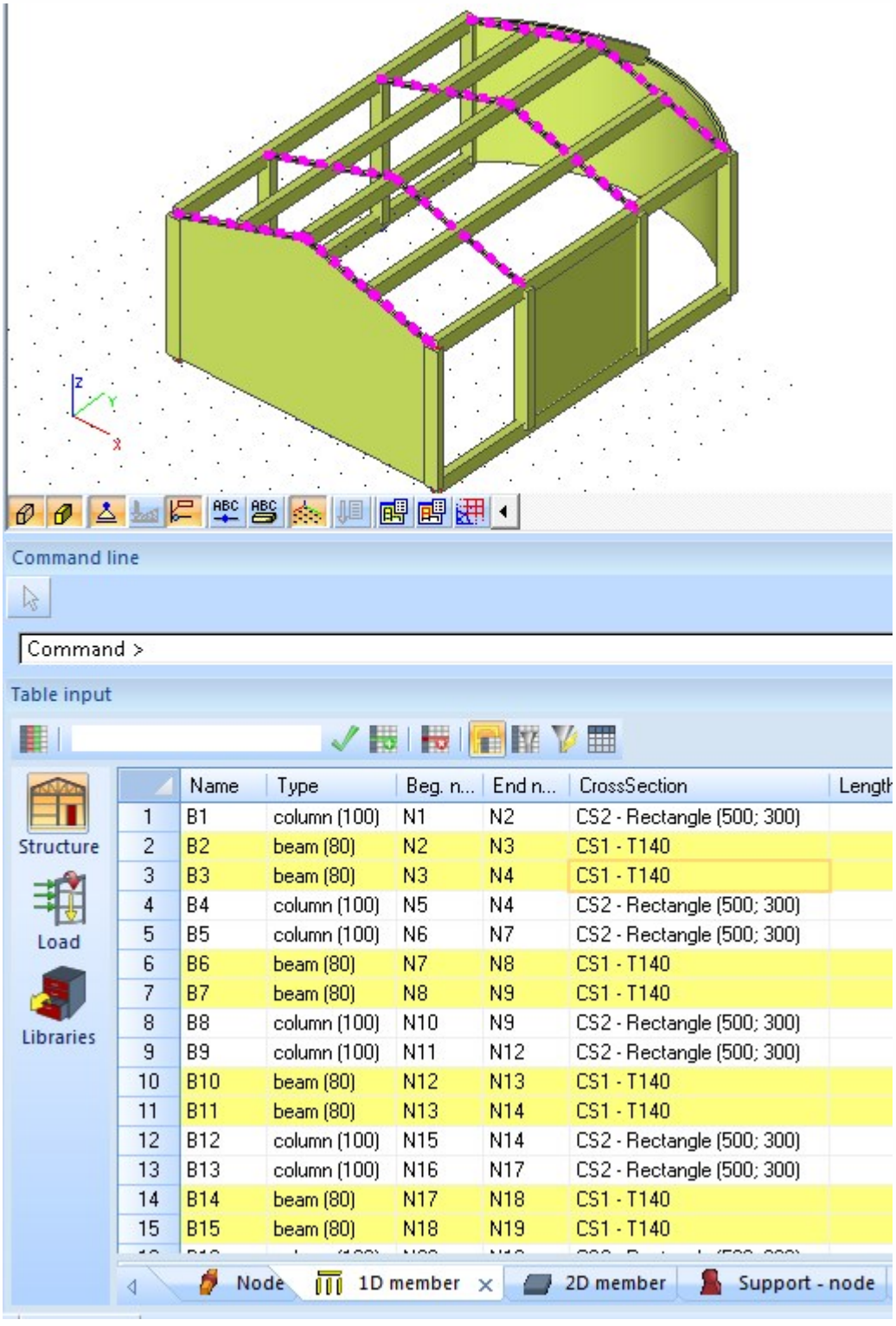
How to select members with the same cross-section:

1. Open the tab with 1D members.
2. Click on the cell with required cross-section.



3. Click on the button "Select by property in cell" on the toolbar.

4. All members with the same cross-section are selected in the model and highlighted in the Table input.



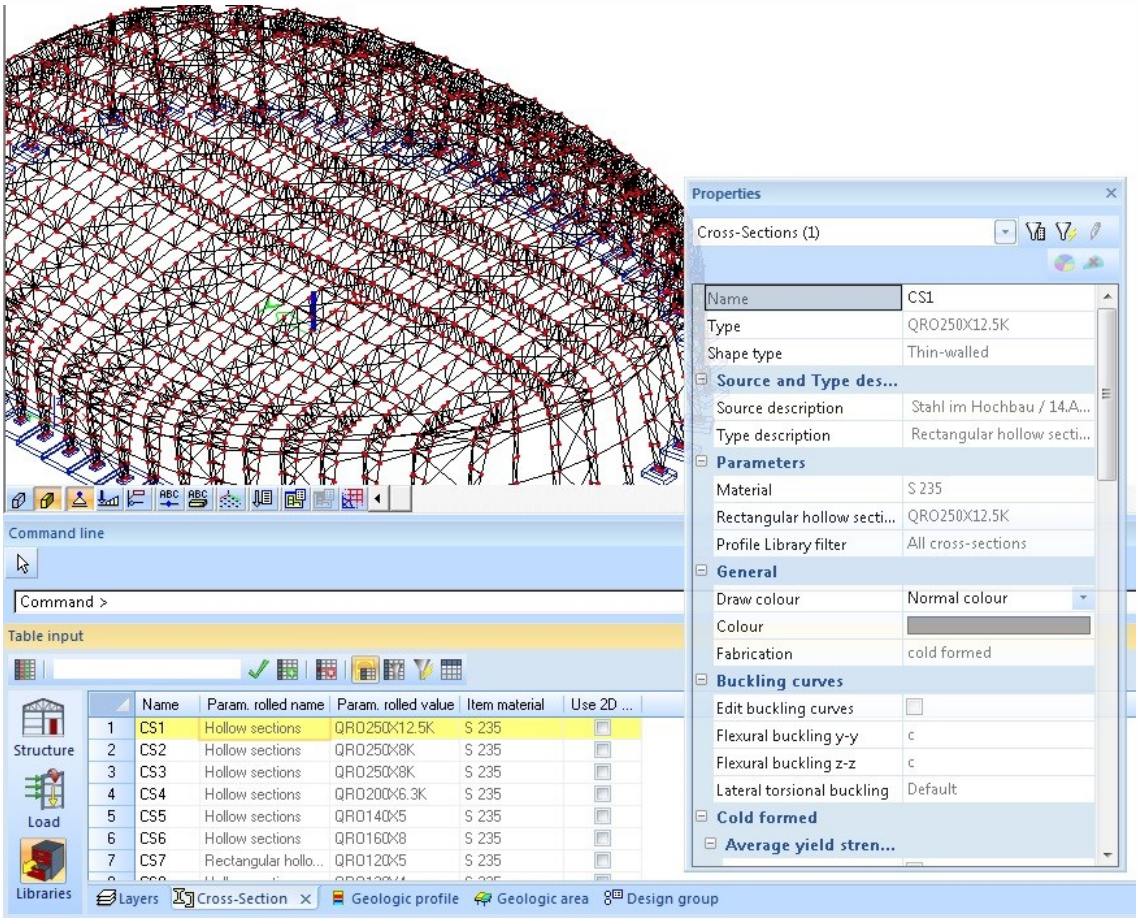
	Name	Type	Beg. n...	End n...	CrossSection	Length
1	B1	column (100)	N1	N2	CS2 - Rectangle (500; 300)	
2	B2	beam (80)	N2	N3	CS1 - T140	
3	B3	beam (80)	N3	N4	CS1 - T140	
4	B4	column (100)	N5	N4	CS2 - Rectangle (500; 300)	
5	B5	column (100)	N6	N7	CS2 - Rectangle (500; 300)	
6	B6	beam (80)	N7	N8	CS1 - T140	
7	B7	beam (80)	N8	N9	CS1 - T140	
8	B8	column (100)	N10	N9	CS2 - Rectangle (500; 300)	
9	B9	column (100)	N11	N12	CS2 - Rectangle (500; 300)	
10	B10	beam (80)	N12	N13	CS1 - T140	
11	B11	beam (80)	N13	N14	CS1 - T140	
12	B12	column (100)	N15	N14	CS2 - Rectangle (500; 300)	
13	B13	column (100)	N16	N17	CS2 - Rectangle (500; 300)	
14	B14	beam (80)	N17	N18	CS1 - T140	
15	B15	beam (80)	N18	N19	CS1 - T140	

Properties of cross-section in the Table input and property dialogue

The tab with cross-sections is in the group Libraries.

Select the required cross section in the tab and check all properties in the row. The properties are displayed in the property dialogue at the same time so there is no need to open the dialogue. Using Cross-section tab in the Table input is much faster.

This is the only way how to display the cross-section properties in the property dialogue.


Properties
Cross-Sections (1)
Name CS1
Type QRO250X12.5K
Shape type Thin-walled
Source and Type des...
Source description Stahl im Hochbau / 14.A...
Type description Rectangular hollow secti...
Parameters
Material S 235
Rectangular hollow secti... QRO250X12.5K
Profile Library filter All cross-sections
General
Draw colour Normal colour
Colour
Fabrication cold formed
Buckling curves
Edit buckling curves
Flexural buckling y-y c
Flexural buckling z-z c
Lateral torsional buckling Default
Cold formed
Average yield stren...
Command line
Command >
Table input
Name Param. rolled name Param. rolled value Item material Use 2D ...
1 CS1 Hollow sections QRO250X12.5K S 235
2 CS2 Hollow sections QRO250X8K S 235
3 CS3 Hollow sections QRO250X8K S 235
4 CS4 Hollow sections QRO200X6.3K S 235
5 CS5 Hollow sections QRO140X5 S 235
6 CS6 Hollow sections QRO160X8 S 235
7 CS7 Rectangular hollo... QRO120X5 S 235
Structure
Load
Libraries
Layers
Cross-Section
Geologic profile
Geologic area
Design group

Properties

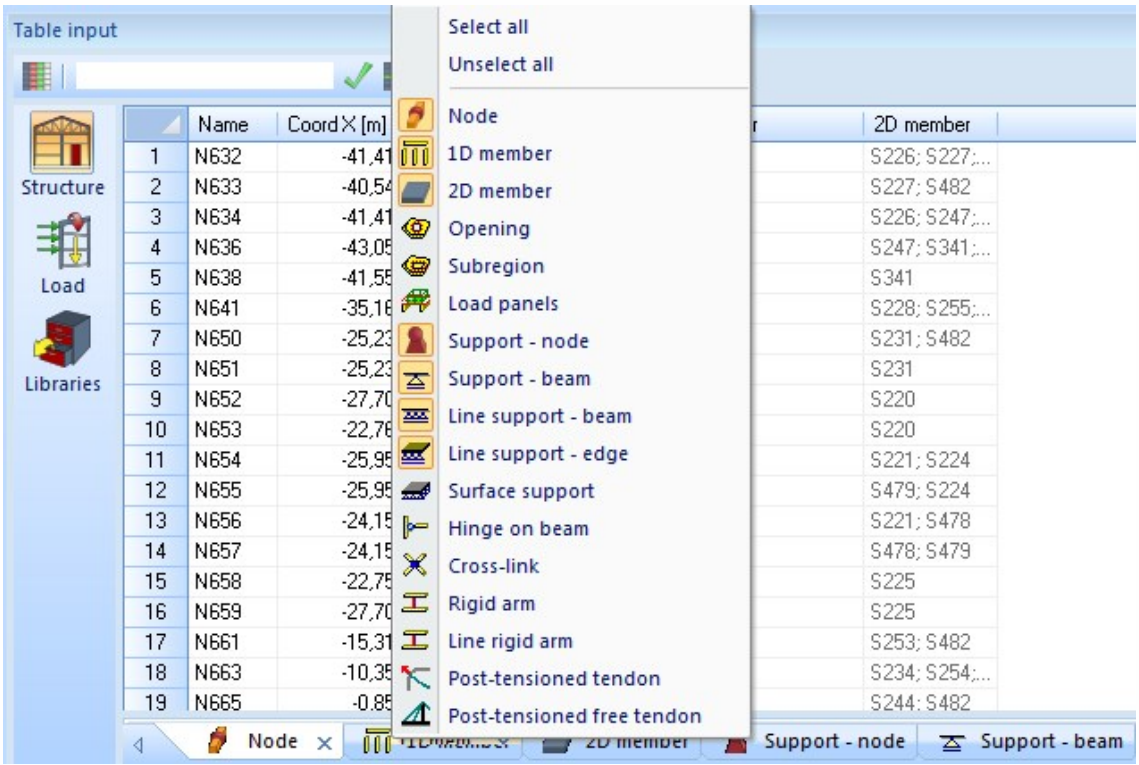
Cross-Sections (1)

Name	CS1
Selector switch	
Type	H100x100x6x8
Shape type	Thin-walled
Source and Type descr...	
Parameters	
General	
Buckling curves	
Fibres and Parts	
2D FEM analysis	
Property Modifications	
Property	
A [m^2]	2,1590e-03
Ay [m^2]	1,3456e-03
Az [m^2]	4,9866e-04
AL [m^2/m]	5,7420e-01
AD [m^2/m]	5,7420e-01
cYUCS [mm]	50
cZUCS [mm]	50
α [deg]	0,00
Iy [m^4]	3,7790e-06
Iz [m^4]	1,3360e-06
iy [mm]	42
iz [mm]	25
Wely [m^3]	7,5590e-05
Welz [m^3]	2,6720e-05
Wply [m^3]	8,6390e-05
Wplz [m^3]	4,1020e-05
Mply+ [Nm]	0,00
Mply- [Nm]	55,00
Mplz+ [Nm]	0,00
Mplz- [Nm]	0,00
dy [mm]	0
dz [mm]	0
It [m^4]	4,9100e-08
Iw [m^6]	2,8200e-09
β y [mm]	0
β z [mm]	0
Geometry	
Formcode	1 - I sections
H [mm]	100
B [mm]	100
t [mm]	8
s [mm]	6
R [mm]	8
W [mm]	52

List of available tabs

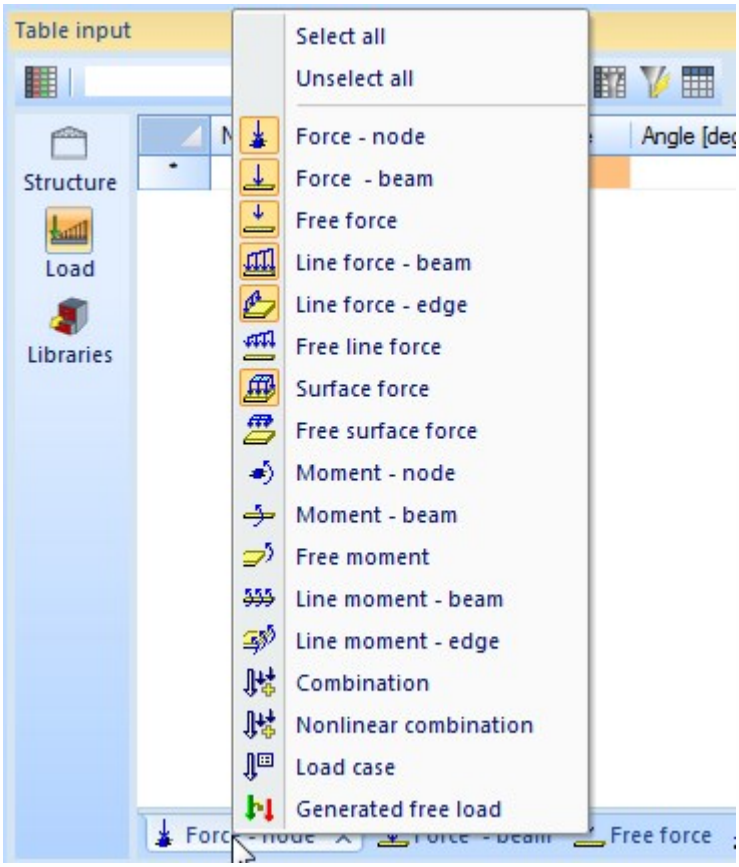
Group Structure

1. **Node** - Structure nodes in 3D model, can be inserted by defining X, Y and Z coordinate;
2. **1D member** - Beams, Columns, Ribs ..., can be inserted by defining begin and end node, it is created as a straight member;
3. **2D member** - Plates, Walls ..., cannot be inserted by Table input;
4. **Opening** - Opening on 2D member, cannot be inserted by Table input;
5. **Subregion** - Subregion on 2D member, cannot be inserted by Table input;
6. **Load panels** - To panel nodes, To panel edges, ..., cannot be inserted by Table input;
7. **Support - node** - Point support in node, can be inserted by defining node (=owner);
8. **Support - beam** - Point support on beam, can be inserted by defining beam (=owner);
9. **Line support - beam** - Line support on beam, can be inserted by defining beam (=owner);
10. **Line support - edge** - Line support on 2D member edge, can be inserted by defining 2D member and edge number (=owner);
11. **Surface support** - Surface support on surface, can be inserted by defining 2D member (=owner);
12. **Hinge on beam** - can be inserted by defining 1D member (=owner);
13. **Cross link** - Fixed, Hinged, Coupler, cannot be inserted by Table input;
14. **Rigid arm** - Rigid arms in nodes, cannot be inserted by Table input;
15. **Line rigid arm** - Rigid arms node-line, cannot be inserted by Table input;
16. **Post-tensioned tendon** - Post-tensioned internal tendon, cannot be inserted by Table input;
17. **Post-tensioned free tendon** - Post-tensioned free tendon, cannot be inserted by Table input;



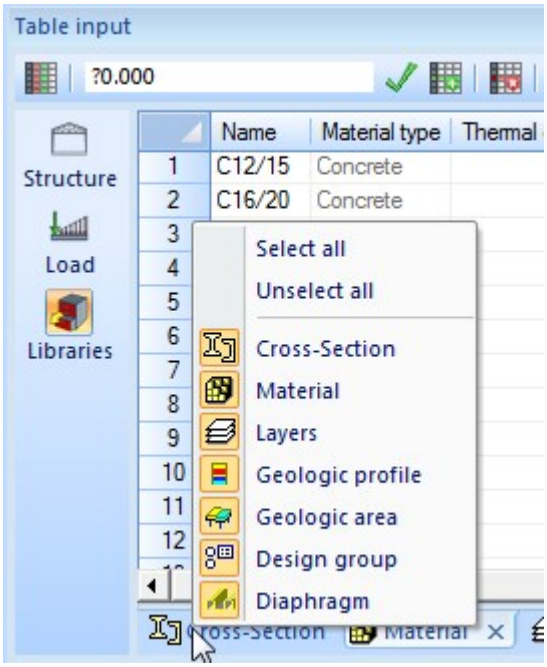
Group Load

1. **Force - node** - Point force in node, can be inserted by defining node (=owner);
2. **Force - beam** - Point force on beam, can be inserted by defining beam (=owner);
3. **Free force** - Free force in point; can be inserted by defining coordinates x,y,z;
4. **Line force - beam** - Line force on beam, can be inserted by defining beam (=owner);
5. **Line force - edge** - Line force on 2D member edge, can be inserted by defining 2D member and edge number (=owner), it is not possible to create a line force on internal edge in this tab;
6. **Free line force** - Free force on line, cannot be inserted by Table input;
7. **Surface force** - Surface force on 2D member, can be inserted by defining 2D member (=owner);
8. **Free surface force** - Free force on surface, cannot be inserted by Table input;
9. **Moment - node** - Point moment in node, can be inserted by defining node (=owner);
10. **Moment - beam** - Point moment on beam, can be inserted by defining beam (=owner);
11. **Free moment** - Free moment in node, can be inserted by defining coordinates x,y,z;
12. **Line moment - beam** - Line moment on beam, can be inserted by defining beam (=owner);
13. **Line moment - edge** - Line moment on 2D member edge, can be inserted by defining 2D member and edge number (=owner), it is not possible to create a line moment on internal edge in this tab;
14. **Combination** - Overview of linear combinations and its coefficients, cannot be inserted by Table input;
15. **Non-linear combination** - Overview of nonlinear combinations and its coefficients, cannot be inserted by Table input;
16. **Load case** - Load case database, cannot be inserted by Table input;
17. **Generated free load** - forces generated from the original free load, cannot be inserted by Table input;



Group Libraries

1. **Cross-section** - Cross section library, tab contains only the basic properties, but the selected row displays all cross section properties in the Property window, cannot be inserted by Table input;
2. **Material** - the library with used materials, cannot be inserted by Table input;
3. **Layer** - Layers library (user can easily switch the activity by layer by this tab), cannot be inserted by Table input;
4. **Geologic profile** - the library with predefined geological profiles, cannot be inserted by Table input;
5. **Geologic area** - the library with predefined geological areas, cannot be inserted by Table input;
6. **Design group** - Design group library, cannot be inserted by Table input;
7. **Sheetings** - Profiled sheeting library, cannot be inserted by Table input;



tutorial: How to use a New row

The Table input is a tool for numerical inputting and editing the structure.

Each tab contains one type of entity. All rows represent members of the same type. The special row on the bottom is displayed if the creating of a new member is supported by the tab. This row has a star symbol as a header. It is called a "New row". It doesn't contain any data just empty cells with two different colours.

A new geometry or add data can be inserted to the project by this row. When the item is created it is also displayed in the 3D model.

New row contains two type of cells:

1. Orange – required cell – the new entity is inserted only if this cell is filled;
2. White – optional cell – the new entity may have a different properties from the default;

E.g. a new node is defined by its 3 coordinates - see the picture:

	Name	Coord X [m]	Coord Y [m]	Coord Z [m]	Member	2D member
48	N68	2,000	4,000	4,285		
49	N69	3,500	4,000	4,285		
50	N70	3,500	4,000	5,785		
51	N71	2,000	4,000	5,785		
52	N72	5,750	6,000	0,985		
53	N73	7,250	6,000	0,985		
54	N74	7,250	6,000	2,485		
55	N75	5,750	6,000	2,485		
56	N76	5,750	6,000	4,285		
57	N77	7,250	6,000	4,285		
58	N78	7,250	6,000	5,785		
59	N79	5,750	6,000	5,785		
*						

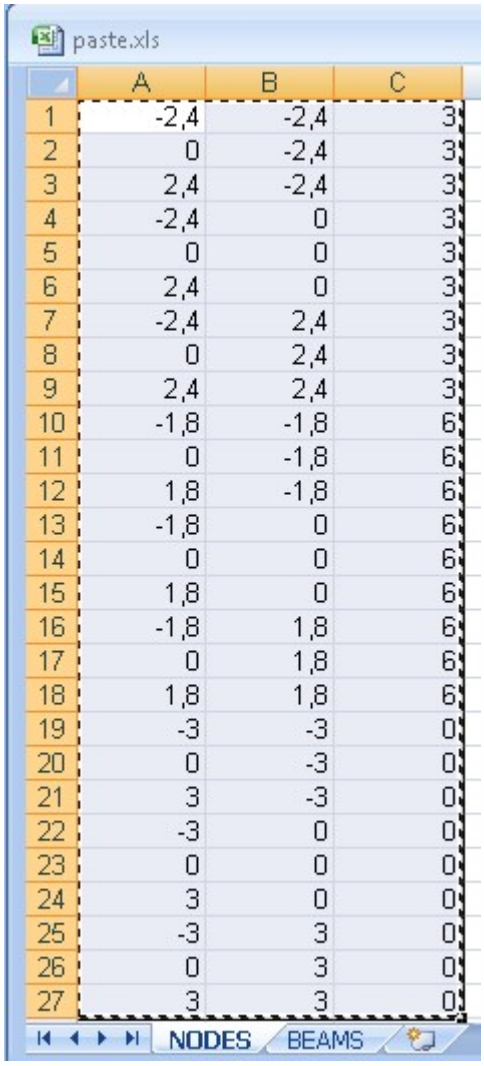
How to insert a new structure by the New row

Create free nodes by copy&paste

1. Open the project "New_row.esa".
   It is an empty project with no libraries or geometry.

2. Open the attached XLS file "paste.xls".
   The file contains three lists named NODES, BEAMS and SUPP.

3. Select all cells with values in the list NODES and copy them to the clipboard. It is just coordinates of nodes with no names.



	A	B	C
1	-2,4	-2,4	3
2	0	-2,4	3
3	2,4	-2,4	3
4	-2,4	0	3
5	0	0	3
6	2,4	0	3
7	-2,4	2,4	3
8	0	2,4	3
9	2,4	2,4	3
10	-1,8	-1,8	6
11	0	-1,8	6
12	1,8	-1,8	6
13	-1,8	0	6
14	0	0	6
15	1,8	0	6
16	-1,8	1,8	6
17	0	1,8	6
18	1,8	1,8	6
19	-3	-3	0
20	0	-3	0
21	3	-3	0
22	-3	0	0
23	0	0	0
24	3	0	0
25	-3	3	0
26	0	3	0
27	3	3	0

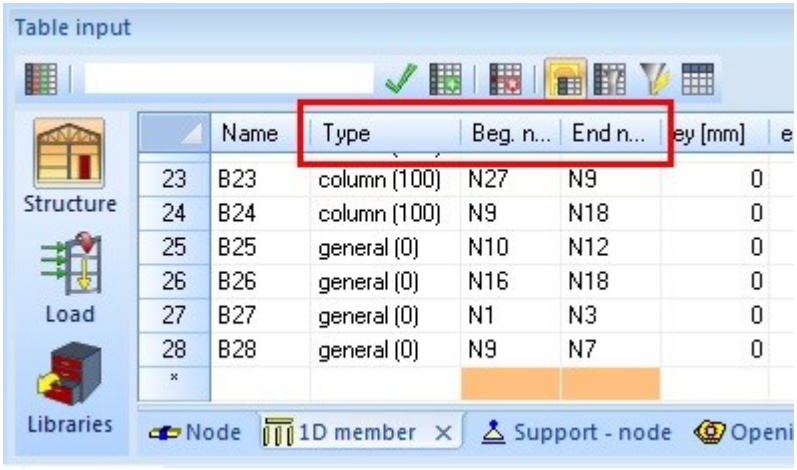
4. Paste coordinates to the New row in the Table input. It must be pasted with the focus on the first orange cell which is “Coord X”. The second must be “Coord Y” and the third must be “Coord Z”. The order is the same as it is in the default settings.
5. The table contains 27 free nodes. When it is pasted, the free nodes are created also in the model window.
   If it is not visible check the activity in the model window.

Create columns and beams by copy&paste

1. Copy all cells with content from the second list in the XLS file – BEAMS – to the clipboard.
   It contains the type, the begin node and the end node of beams.
2. Switch on the tab 1D member in the Table input. Check the order of columns.

> Check that the order of columns is Type, Begin node and End node so the pasted content matches!

3. Paste the content from the clipboard to the New row in this tab. The content must be pasted to the appropriate cells. Begin and end node are required values.

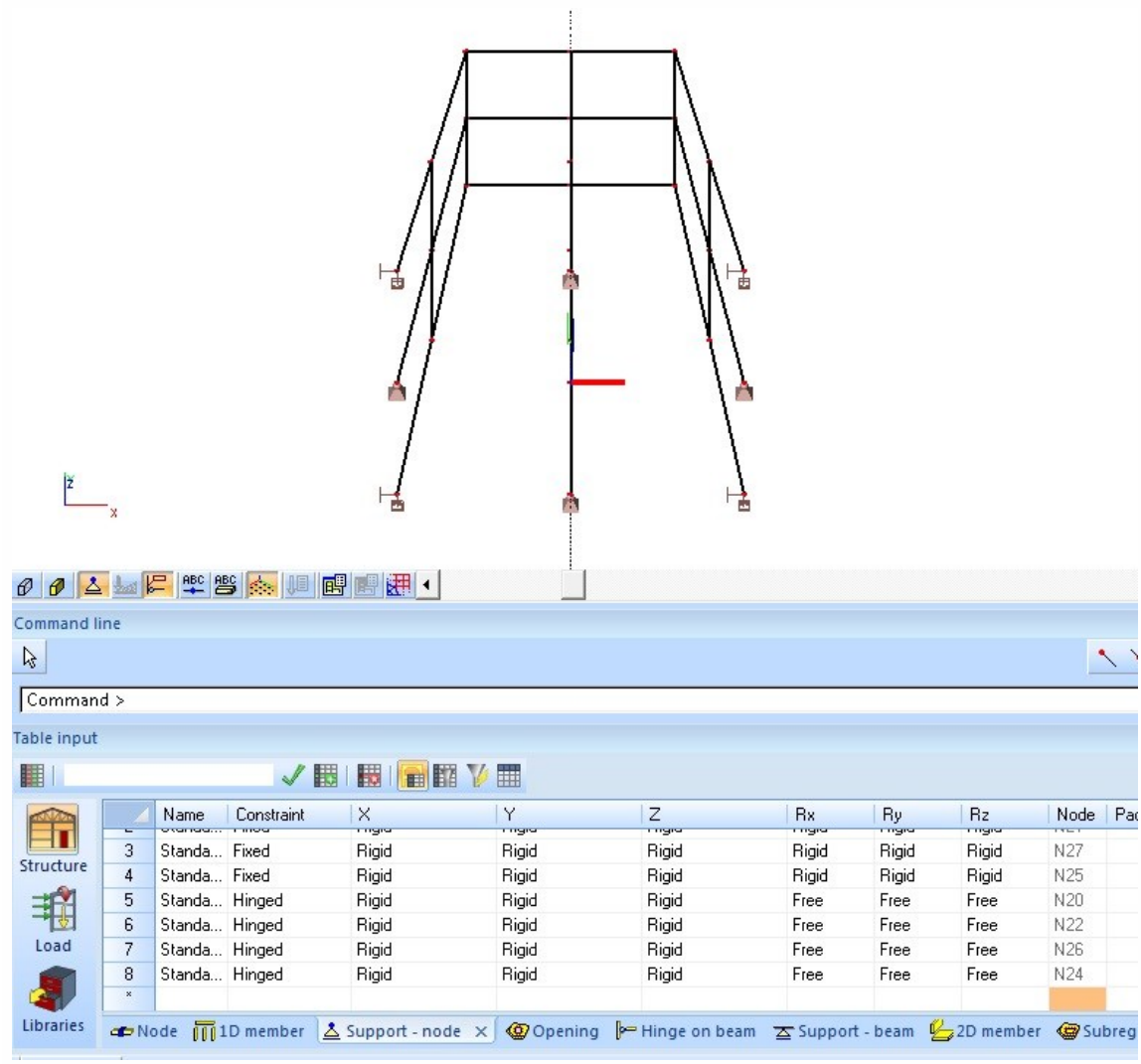


4. The system starts the dialogue for the cross-section definition automatically.Create any new cross section.
5. 1D members are created after the paste.

Create supports by copy&paste

1. Copy all cells from the third list in the XLS file – SUPP – to the clipboard.
2. Open the tab Support - node in the Table input.
3. Paste the content from the clipboard to the New row. The content must be pasted to appropriate cells. Structure node = owner is required cell.

Check the order of columns.



The final project is in attachment "New_row-final.esa".

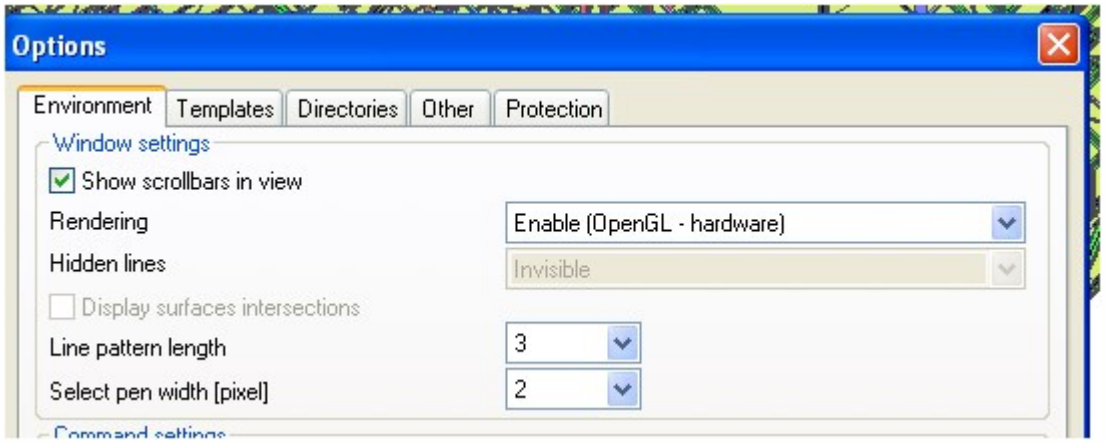
Performance tips

The chapter contains several tips which can help to balance performance and quality of output according to current needs of the user. It contains also some general remarks related to performance of SCIA Engineer.

Rendering

The rendering of model is based on OpenGL technology. There are several settings which influence quality and speed of displaying of the model.

The most significant changes in displaying speed and quality can be done in the setting Setup / Options / Environment - Windows setting.



In the setting **Rendering** user can select between following possibilities:

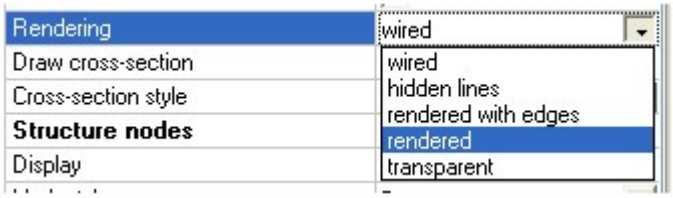
- Enable (OpenGL - hardware) - all calculations are done in graphic card. **It is fastest way of drawing**
- Disabled (Wired) - also very fast way of drawing of the model, however in this the model is displayed only as wired . This selection is useful in case of problems with graphic card.
- Disabled (vector hidden lines algorithm) - very slow way of displaying of the model. In this case visibility of lines is calculated by the processor
- Software emulation (OpenGL) - also very slow way of displaying. It provides the same output as "Enabled (OpenGL - hardware)", but all calculations are done by the processor, This can be useful in case of problems with graphic card.

Antialiasing quality can make lines smooth but time needed for drawing of the picture is multiple times longer.

Antialiasing dos not need to be switched permanently ON. it can be switched ON just for selected pictures (in gallery, in Engineering Report, in document). It is also possible to set level of antialiasing for pictures which are saved into clipboard.

See the manual related to Anitaliasing.

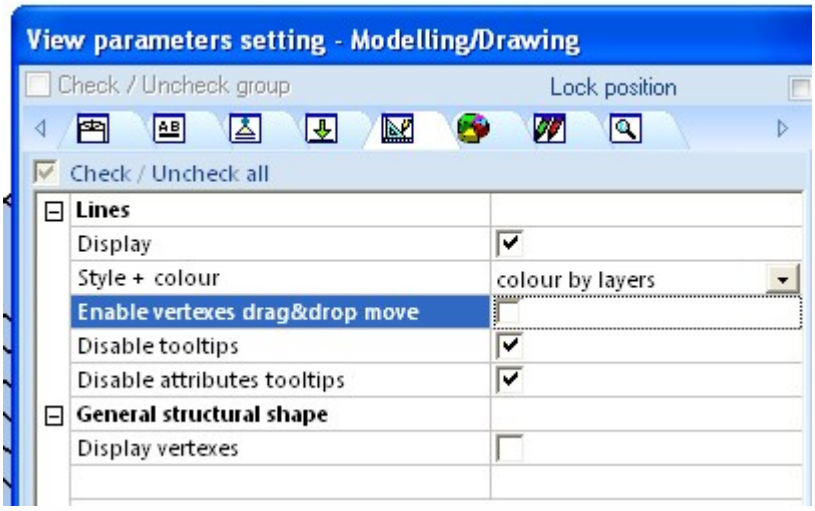
Another setting related to rendering which can influence the speed of drawing is in setting of View parameters. Mode Rendered is faster than Rendered with edges however the speed difference is not so significant.



Drawing of lines

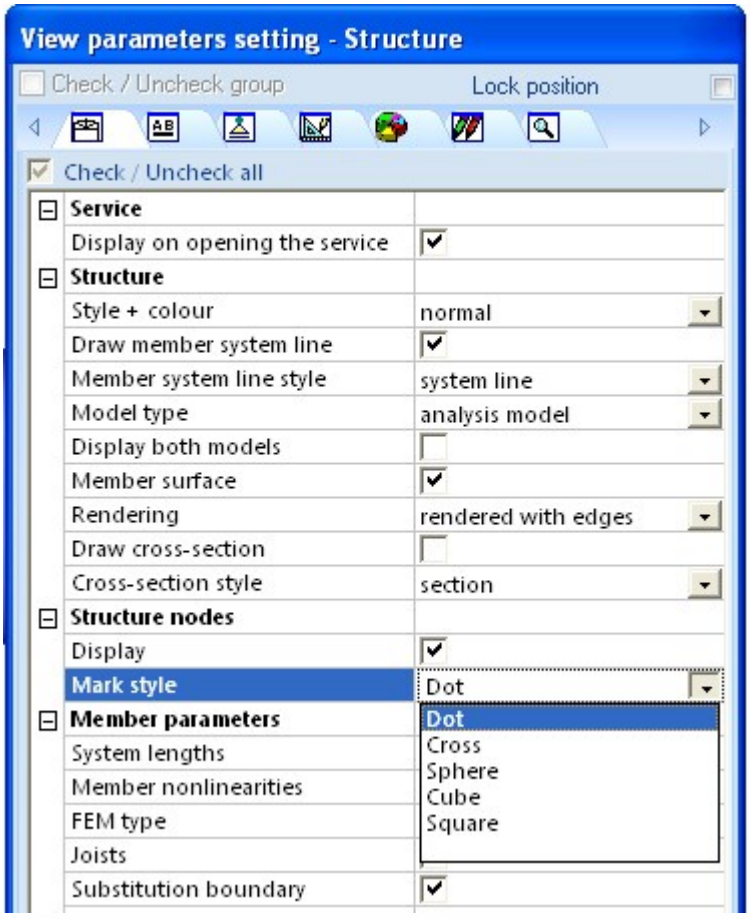
In case of projects with many drawing lines (e.g. imported from DWG), The speed of drawing can be increased (and memory consumed by drawing objects decreased) by switching OFF vertexes on lines (subsequently it is not possible to drag and drop ends of lines) and by disabling tooltips on drawing lines.

Both those settings can be done in View parameters setting.



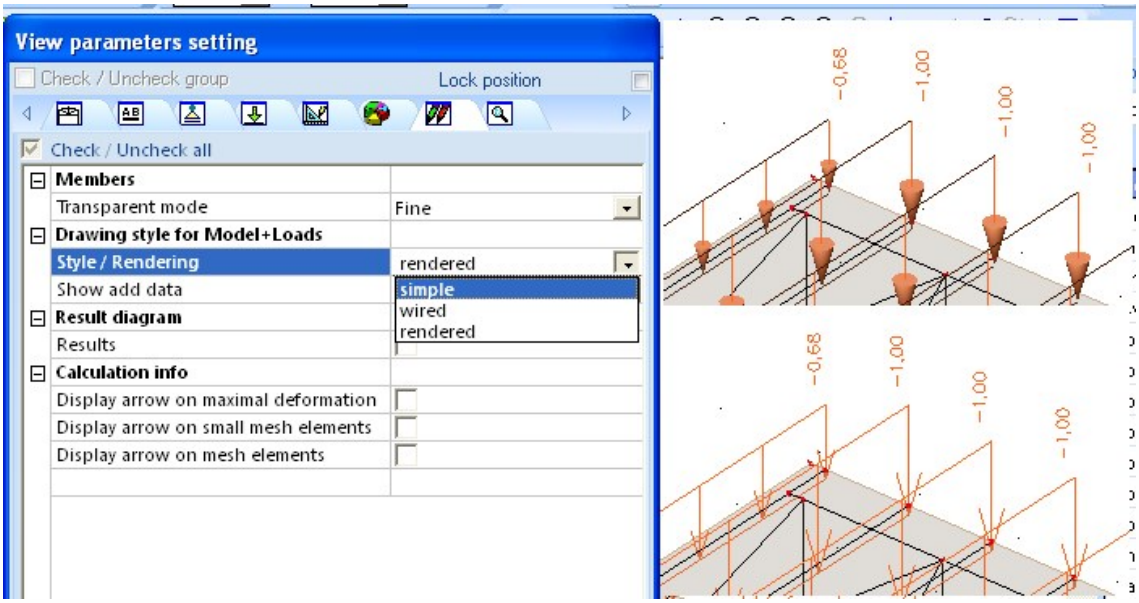
Drawing of structure nodes

In the View parameters setting it is possible to select one of available types of displaying structure nodes (or to switch them completely OFF). Dot is significantly faster than Sphere.



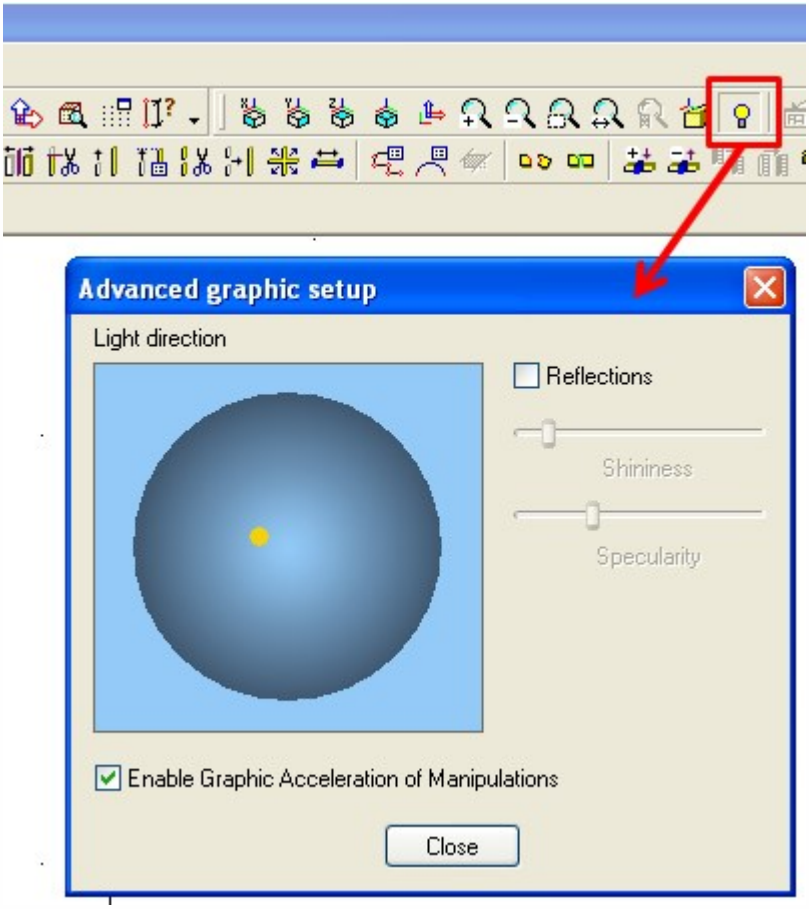
Drawing of loads

In case the project contains many loads the way of their displaying can significantly reduce the speed of drawing. Switching of drawing style to **Simple or Wired is very recommendable** in case of model with many loads (e.g. generated loads)



Changing view point in 3D window

Each change of view point need recalculation of picture displayed and the screen. This recalculation can be accelerated by the checkbox in the Advanced graphic setup.

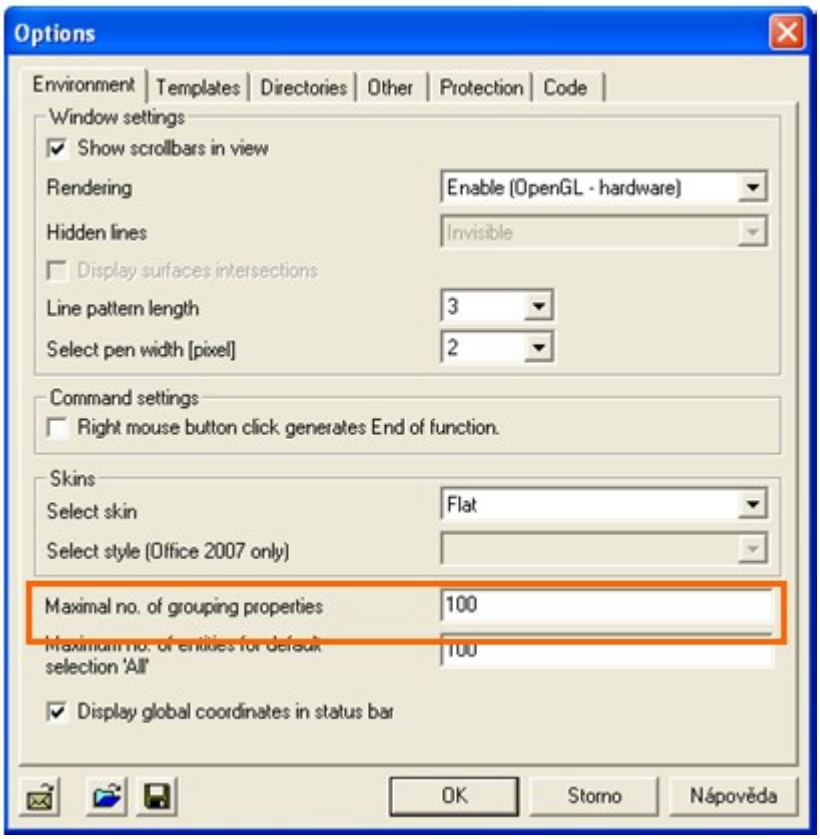


Switching the acceleration ON causes small time gap before the manipulation with view point starts (informations must be loaded into graphic card), then the manipulation is fully calculated by graphic card which is very fast.

Zooming IN and OUT using the mouse wheel is by the operation system presented as a set of small zooming steps. During each this zooming steps the above mentioned time gap occurs. **Therefore it is much faster to use Ctrl+Shift+Right mouse button** while moving the mouse up/down to zoom IN/OUT..

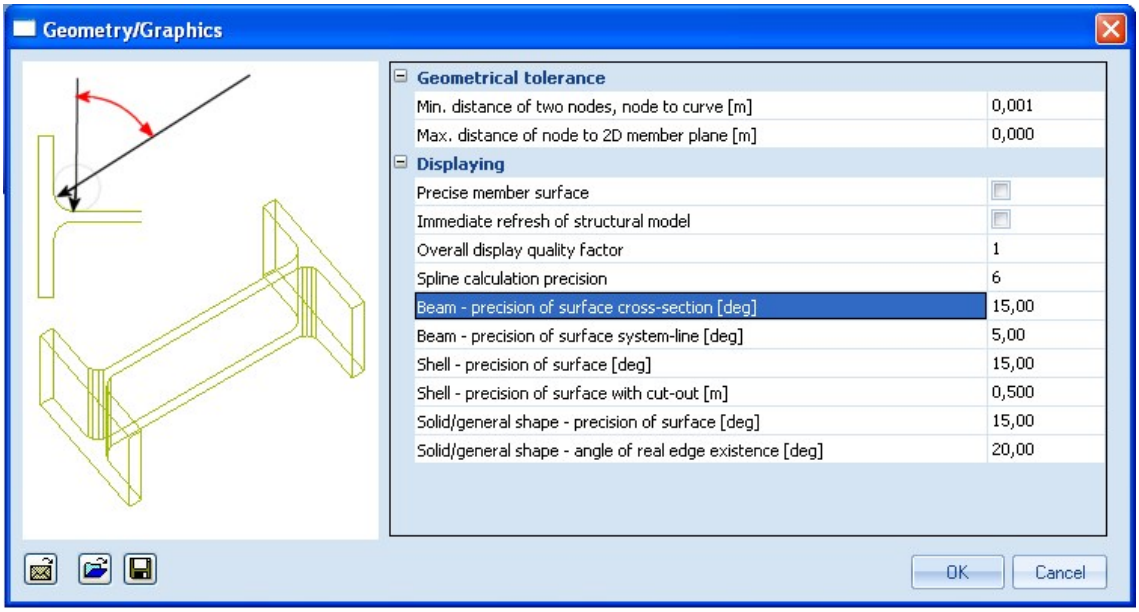
Grouping of properties

After selection of multiple entities in the 3D window the intersection of their property is displayed in the property grid. In case of big model and selection of big number of entities this can take a long time. Therefore there is a limit in the Setup / Options / Environment which sets the maximal number of entities which are used for evaluation of intersection of their property.



In the case displayed on the picture the properties are displayed in case there is selected less than 100 entities.

Geometry / Graphics

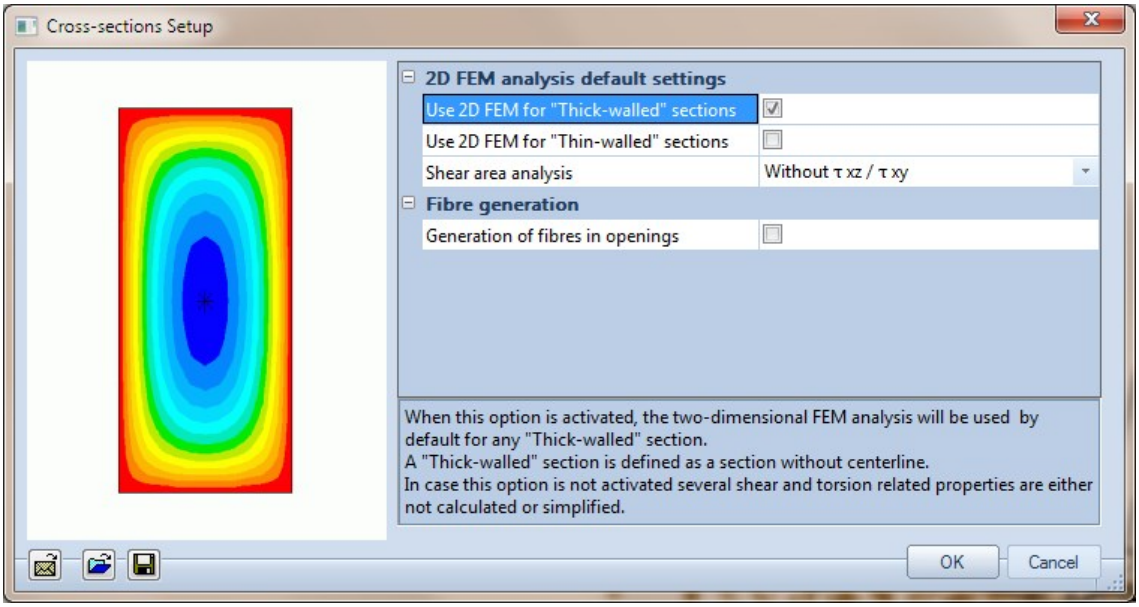


This setting defines the precision of drawing of different types of entities. Higher precision can reduce the speed of displaying and vice versa in case of big number of those entities.

Cross-section analysis

By default, thick-walled cross-sections are analysed using the advanced 2D FEM method which is required for an exact shear and torsional analysis.

In case however shear and torsion are of less importance, the 2D FEM method can be de-activated through Setup / Cross-sections.



This will lead to a significant reduction of the time needed to run the analysis, especially in case arbitrary members are used.

Setting of system

The SCIA Engineer uses very often files on hard drive in the folder with Temporary files. Therefore it is essential to put this folder on fast hard drive. The best is to use the **SSD drive**. The **increase of the working speed and SCIA Engineer response is significant.**

It is also essential to **exclude the folder with Temporary files from antivirus checking**. Files can change very often in this folder and antivirus can block SCIA Engineer to work with them until they are verified. This can lead to significant decrease fo the working speed and sometimes can lead even to crashes of SCIA Engineer.

Program settings

Language of the program

By default, the program starts and works in the language chosen during the installation. For many users, however, another language of the user interface may be more suitable. The language of the application and language for outputs can be set in the Setup > Options dialogue.

The procedure for adjustment of a required language

1. Open function Setup > Options :
   1. using menu function Setup > Options,
   2. using icon Options settings ( ) on toolbar Main.
2. Select tab Other.
3. In the group Language default select the required language for the program.
4. In the group Language default select the required language for outputs.
5. Confirm the settings.

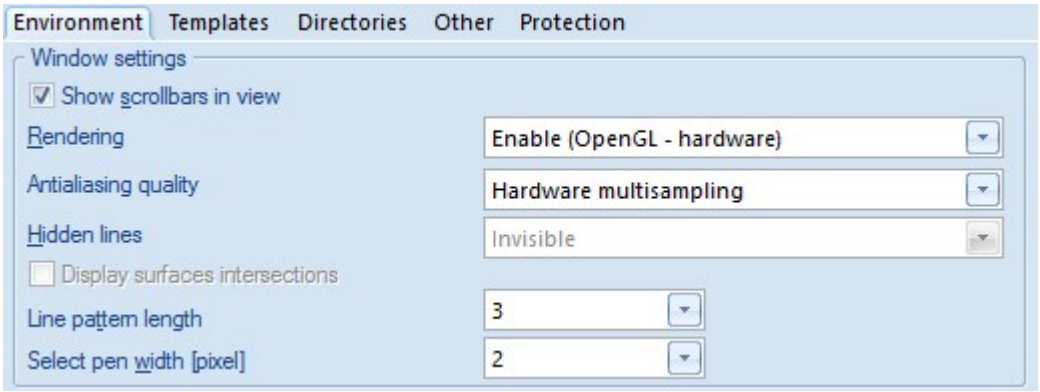
Note: The change takes affect only after the restart of the program.

Application options

Environment settings

Parameters affecting the user interface appearance make up this group of Environment settings.

Window settings



Show scrollbar in view	This item specifies whether the graphical windows are equipped with scroll-bars on their right and bottom edges.
Rendering	The item sets the mode that is used for drawing into application graphical windows.

Antialiasing quality	Allows smoother rendering. Most visible on inclined lines displayed on rectangular pixel matrix of monitors with low resolution.
Hidden lines	This option specifies the mode for hidden lines of individual structural entities.
Display surfaces intersections	This check-box allows user to display the intersection of two surfaces. Option is available only for some rendering modes.
Line pattern length	This item specifies the style of dashed lines.
Select pen width	Allows user to defined the default width of pen for entities not included in Palette settings (Setup - Colours/Lines).

Rendering

Defines the type of rendering.

Disabled (vector hidden lines algorithm)	This mode disables any rendering. The drawing on the screen is fast but reverse surfaces of the structure cannot be hidden and are shown.
Disable – wire	This mode is almost identical to the one above. It is however modified to run even on computers with old types of graphical cards where the mode above may not function properly.
Enable (OpenGL - hardware)	If this option is selected, the hardware rendering capability of the computer is employed. This option may lead to a "distorted" display on some computers, especially those with older models of graphic cards.
Software emulation (OpenGL)	This options tells the computer to simulate the rendering capability by means of software algorithms. This option should work properly on all computers. However, if selected on slower one it may lead to longer response of the computer during regeneration of the screen.

From version 18, SCIA Engineer introduced the Navicube 3D navigation system. That system is available only with hardware OpenGL rendering. Compatibility issues have been noted on some computer systems, when 3D accelerations are not properly supported. For more details, see OpenGL textures troubleshooting.

Hidden lines

The Hidden lines option serves as a substitute for full and proper rendering if the Rendering itself is disabled.

The available options are:

Invisible	The hidden lines (hidden parts of entity surfaces) are not drawn at all.
Dashed	The hidden lines are drawn in dashed style.

In addition to the above-mentioned options, it is possible to select whether the intersections of individual surfaces should be calculated and displayed.

Note: The settings made here determine which mode of rendering and hidden line display is set for the application. This setting does not mean that the rendering of the scene (i.e. of what is displayed on the screen) is really applied. To do so, the rendering must be switched on for the required graphical window. This can be done by means of the appropriate view parameter for the appropriate graphical window.

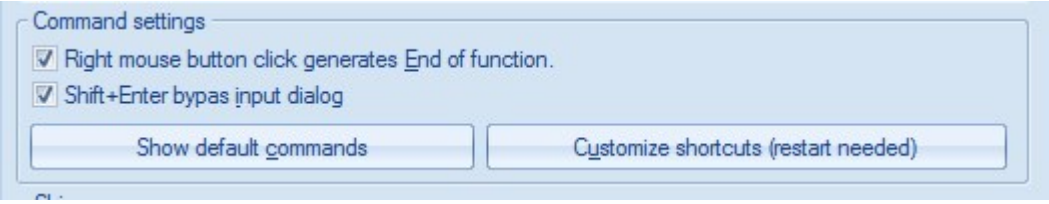
Line pattern length

This item affects the style of dashed lines. The dashed lines may be used whenever within the projects. Any dashed line is controlled by this item.

Small number means short lines used in the dashed line with smaller gaps in between.

Large number means long lines used in the dashed line with longer gaps in between.

Command settings

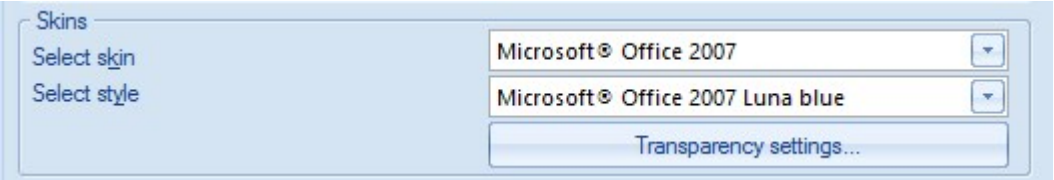


Following setting is related to using commands in Command line.

Right mouse button click generates End of function	If this option is ticked, the right mouse button generates End of command when pressed in any opened function such as definition of a new 1D member, move of 1D member, etc.
Shift+Enter bypass input dialogue	If this option is ticked, the command is immediately performed with last used set of properties (e.g. 'beam' command confirmed by Shift+Enter allows user to input the horizontal beam with last set of properties directly and properties input dialogue is skipped).
Show default commands	This button opens text file with the list of supported commands, default short-cuts and simple explanation.

Customize short-cuts (restart needed)	This button opens text file with the list of user-defined short-cuts. User can define own short-cuts in format <command>=<user_ shortcut> (e.g. Structure.1d.Beam=usershortcut).

Skins



Following settings influence the visual appearance of SCIA Engineer.

Select skin	This option allows the user select from pre-defined screen skins of the application.
Select style	This option allows the user select from pre-defined colour schemes of the particular skin.
Transparency settings	This option allows the user select the value of transparency for entities using transparency (e.g. floating undocked windows).

Other parameters

Maximum number of grouping properties	This value determines the maximum number of entities that can be selected at a time so that the Property window was filled with the parameters of the selected entities. If the number specified here is exceeded, the property window is left blank and can be filled in only on user's explicit request.
Maximum number of entities for default selection 'All'	This value determines if in Properties of result services the DEFAULT (i.e. before the user change) value of Selection is All or Current, related to the number of entities in project. If the number of entities in actual project is bigger than this limit, the default value of Selection will be Current and in default no results will be printed after refresh in order to prevent the long presentation time on large projects.
F1 opens online help instead of locally installed help	By default, the F1 opens the online help. The local help may be used when user works offline.

Display pictures below properties	This check-box controls whether the explaining picture under the Property window is displayed or not.
Use vertical splitter in properties	Enables user to set the horizontal position of vertical splitter in 'Properties' window.
Display global coordinates in status bar	By default, the status bar displays co-ordinates defined in an active user co-ordinate system. In addition, the global co-ordinates may be displayed as well.
Reset layout for dockable windows and toolbars (after restart)	This button loads the program-default settings of toolbars and windows layout. Reset will be taken into effect after the restart of application.
Current style of toolbars	In this combo-box user can select the predefined set of toolbars.

For setting of application options see chapter Adjusting the application options.

Graphic templates settings

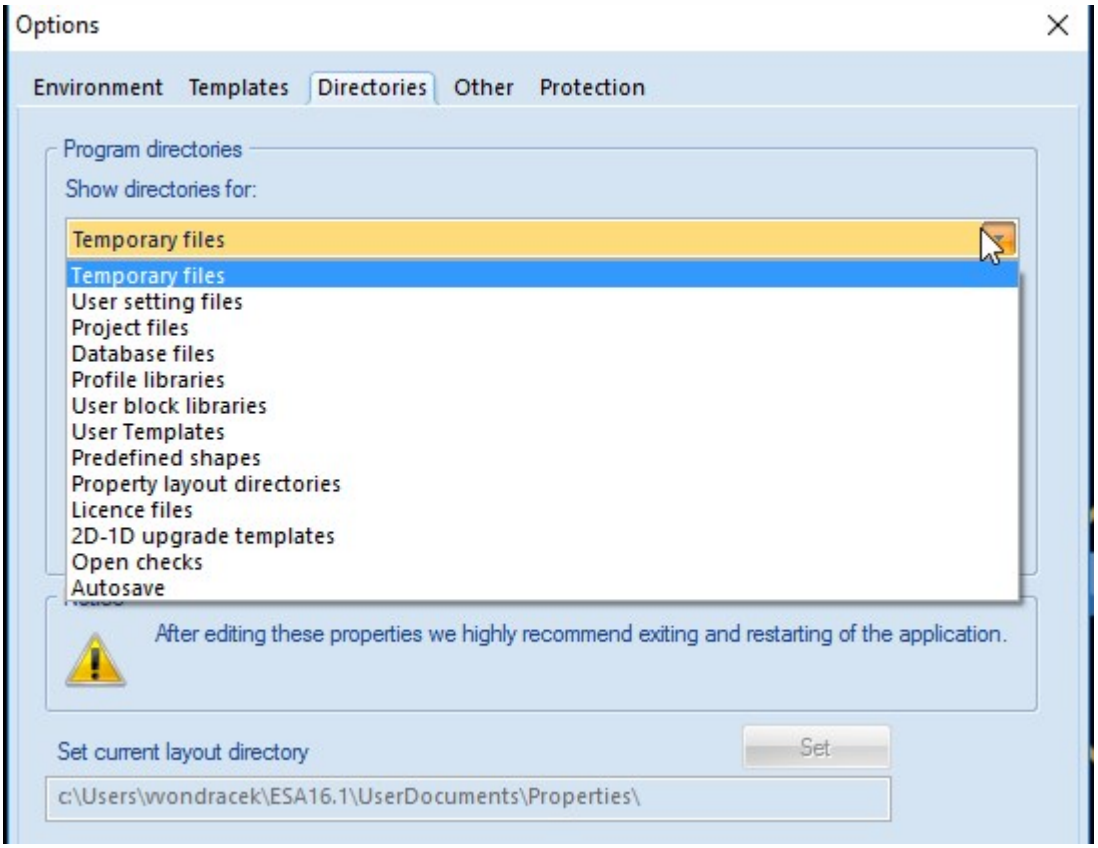
This tab enables the user to define templates which new drawings will be based on. This option may be useful for example if a title block with the company logo should be attached to every drawing.

Print picture	Defines the template for function Print picture.
Overview drawings manager	Defines the template for drawings created in the Paper space gallery.

For setting of application options see chapter Adjusting the application options.

Directories settings

This dialogue allows the user to specify the location of SCIA Engineer files. The adjustment can be made separately for individual file types.



Temporary files	The folder stores any temporary files.
User setting files	The folder stores all files with user-made settings.
Project files	The folder stores projects created and saved by the user.
Database files	The folder stores databases provided with the program.
Profile libraries	The folder stores databases of cross-sections provided with the program.
User block libraries	The folder stores all user blocks that may be arranged in sub-folders of this main library folder.
User Templates	The folder stores user templates (i.e. template projects created by the user).
Predefined shapes	This folder contains predefined shapes such as cylinder, cone, etc.
Property layout directories	This folder contains user defined layouts for Property window.
Licence files	This folder contains licence files.
2D-1D upgrade templates	This folder contains predefined templates for Upgrade 2D->1D export.
Open checks	This folder contains user-defined SCIA Design Forms checks which can be used in 'Design Forms Checks' service.
Autosave	This folder contains projects saved by Autosave functionality.

Note: The changes made in this dialogue will take affect ONLY after the program is closed and restarted. The items on this tab sheet CANNOT be edited if any project is currently opened.

For setting of application options see chapter Adjusting the application options.

Other settings

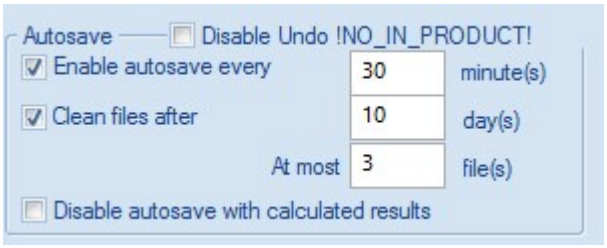
Action on start-up

This part offers a set of settings that relates to projects opened in SCIA Engineer right after start of SCIA Engineer.

None	No action is carried out when the application is started.
Last opened project	The last opened project is automatically loaded into the application on its start.
Show Open project dialogue	When the application is started, the Open project dialogue is automatically displayed to allow for the selection of the project to be processed.

Autosave

This part offers setting that is related to automatic project saving functionality which protects the user against loss of data.

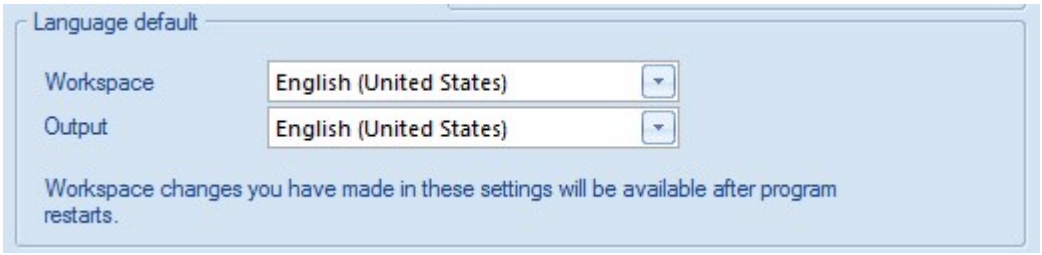


Enable autosave every	If this check-box is checked, the projects are automatically saved in specified interval.
Clean files after	If this check-box is checked, the automatically saved projects are deleted after specified time period.

In Directories tab it is possible to specify the folder used for saving autosave files.

Language default

This part offers the selection of language of the environment and output.



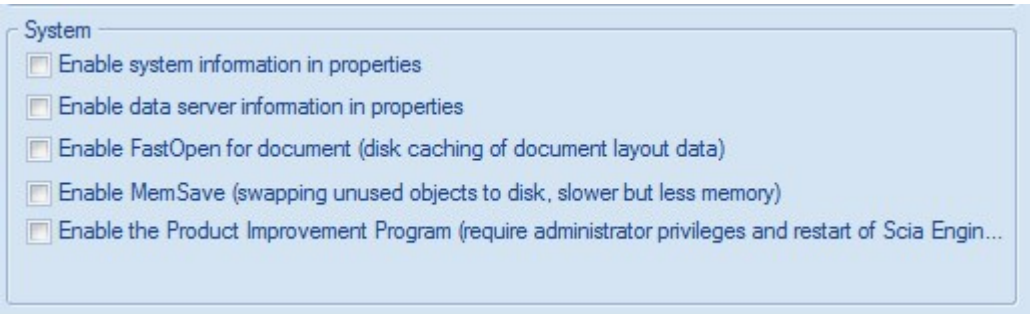
Workspace	User can select the language of the application environment.
Output	User can select the default language of the outputs from the application (e.g. print preview).

It is necessary to restart SCIA Engineer to take effect of language change of both Workspace and Output in all parts of software.

Language of output from Document or Engineering Report is controlled in particular document/report properties.

System

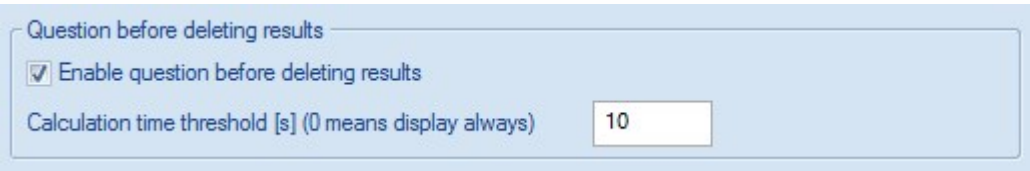
This part offers settings related to the system behaviour.



Enable system information in properties	Display system informations in Property window just for debugging purposes.
Enable data server information in properties	Display data server informations in Property window just for debugging purposes.
Enable FastOpen for document (disk caching of document layout data)	The cache file is created to speed up the loading of document data.
Enable MemSave (swapping unused objects to disk, slower but less memory)	Unused objects are saved to disk which allows to free up the memory. Efficient on computers with small memory.
Enable the Product Improvement Program (require administrator privileges and restart of SCIA Engineer)	This functionality allows to send statistics helping developers to improve the software.

Question before deleting results

This part offers setting regarding displaying the dialogue just before erase of results induced by the change of model.



Enable question before deleting results	If check-box checked, SCIA Engineer asks the user after the change of model has been done whether to accept the change and delete results or not.
Calculation time threshold [s] (0 means display always)	Defines how long last calculation must have taken to display the question. If last calculation lasted less than specified time, the question will not be displayed.

Classic GUI

This part offers user to activate the old style GUI in some parts of SCIA Engineer.

Old project manager	If this check-box is checked, the old-style 'New project' dialogue is displayed instead of the new Start project dialogue.
Preview window	If this check-box is checked, the 'Preview' of results using old-style document is used instead of new 'Report preview' based on Engineering Report technology.
One click on main tree	If this check-box is checked, one click opening is active for main tree.
Old icon set	If this check-box is checked, old icon set is used.

For setting of application options see chapter Adjusting the application options.

Protection settings

This tab summarises the Protection settings.

For more information read chapter about Protection.

Adjusting the application options

The procedure for the adjustment of application options

1. Open Options dialogue
   1. either: using menu function Setup > Options,
   2. or: using button [Options] ( ) on Main toolbar.
2. Make required settings on individual tabs.
3. Confirm with [OK].

The dialogue also contains three save/read buttons.

Read application default		Reads settings as they were pre-defined by the developer of the program.
Read user default		Reads settings that have been previously saved as user's own default.
Save as user default		Saves the current settings as the user's default settings. These settings may be later read by the above mentioned function.

Project settings

Basic project data

The basic data of a project describe the project and define some of its main parameters.

Project filename

It shows the name of the project.

Project data

This group of items allows the user to enter some statistical data about the project

Name	name of the project E.g. Eddy Merckx's Airport – Brussel
Part	name of the project part, if the project is complex and consists of several partial sub-projects E.g. Western hall + connection footbridge
Description	E.g. variant A (underground parking, restaurants on first floor, check-in desks on second floor)
Author	name of the project author E.g. Sven Nijs
Date	date of the last project modification, or date of the program creation, etc. E.g. 02/02/02

Structure

Here, you can choose the type (or we can say "dimension") of the structure you want to model. Depending on the type selected, some of the functions and options of the program may be disabled or hidden (e.g. in the case of 2D frame oriented in plane XZ, the button for setting the sight of the model from the direction of X and Z axes respectively won't be present on the View toolbar). This feature leads to a significant simplification in the operation of the program for simpler types of structures. The functions and options that are not appropriate (are not possible practically) for the particular type are hidden and do not add to the complexity of the program. The idea behind this feature is: A complex task requires a complex tool, but a simple task can get by a simple tool.

Truss XZ	The 1D members of a model are capable of carrying axial forces only. That means that pin ends (hinges) are meaningless, supports do not have rotation degrees of freedom defined and results consists of axial forces only. Only a 2D model can be created.
Frame XZ	The 1D members can represent a planar frame structure. Only a 2D model can be created.
Truss XYZ	This mode is similar to Truss XZ, but a real 3D structure can be created.
Frame XYZ	This option is similar to Frame XZ, but a real 3D structure can be created.
Grid XY	A horizontal grate can be modelled in this mode.
Plate XY	This mode provides for analysis of combined 1D member and slab structure. All the members must be located in a horizontal plane. Only a 2D model can be created.
Wall XY	This mode is similar to Frame XZ mode, but vertical walls can be inserted as well. Only a 2D model can be created.
General XYZ	This option allows the user to model and analyse a 3D structure consisting of any structural members: 1D members as well slabs (plates, walls, shells).

Note: Item Structure is compulsory and the user has to make a choice from the available variants.

Material

This option tells the program which materials will be used for members of the structure. The advantage of this in advance selection is that the program functions working with materials will know, which material the user is interested in. Therefore, the functions will not offer other material types and, consequently, the dialogs, lists and similar items will be lucid and readable as much as possible.

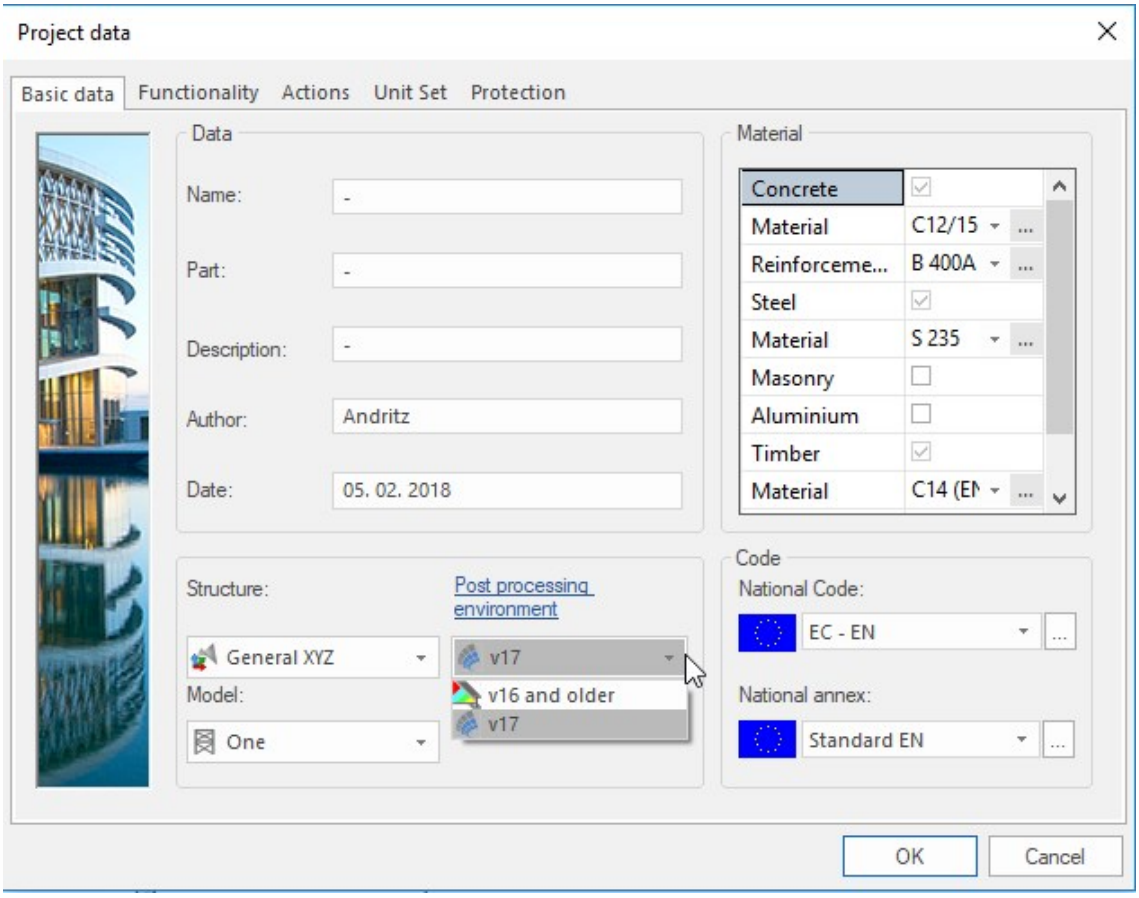
If the user realises later that some other material type is necessary, it is of course possible to call the setting dialogue any time in the future and widen the selection of used material types.

Note: At the beginning, i.e. at the time when a new project is being created, it is necessary to select at least one material type.

Post-processing environment

Starting from version 17.0, it is possible to use a new post-processing environment for displaying of results. The new post-processing environment can be enabled in the Project dialogue.

v16 and earlier	functions for the evaluation of results are consistent with the appearance in previous versions (i.e. v16 and older)
v17	enhanced methods and possibilities of drawing style and other options for results are available



Note: Post-processing environment v17 covers linear calculation (except prestressing), linear stability, nonlinear calculation (beam local nonlinearity, 2nd order calculation, Initial deformation and curvature) and modal analysis. Eurocode building standards for concrete

and steel are supported. Please, check content of your Engineering Report after calculation. If you want to use advance analysis types change Post-processing environment to v16 and older.

Detailed description can be found in chapter "v17 Post-processing environment " below

Model

One	the project will contain a single model of a structure
Absence	the project can contain some members that may be missing in some stages of the analysis
Construction stages	the project will represent modelling of construction stages appearing during the execution of the structure

Code

The selection of the active code determines how the program deals with data related to a specific technical standard. In practice it means that the code selection affects:

- the materials offered as code-related materials, e.g. steel or concrete grades, etc.
- the procedures, algorithms and possible parameters performing and necessary to perform code checks.

Note: The choice of a particular national standard may have an effect on the layout and even functionality of numerous functions. E.g. functions like Load case and Load group have got parameters that depend on the current code of the project. That means that these function offer the user different parameters for e.g. Czech standard than for let's say Eurocode. Also the functionality of some functions or services is different for different codes.

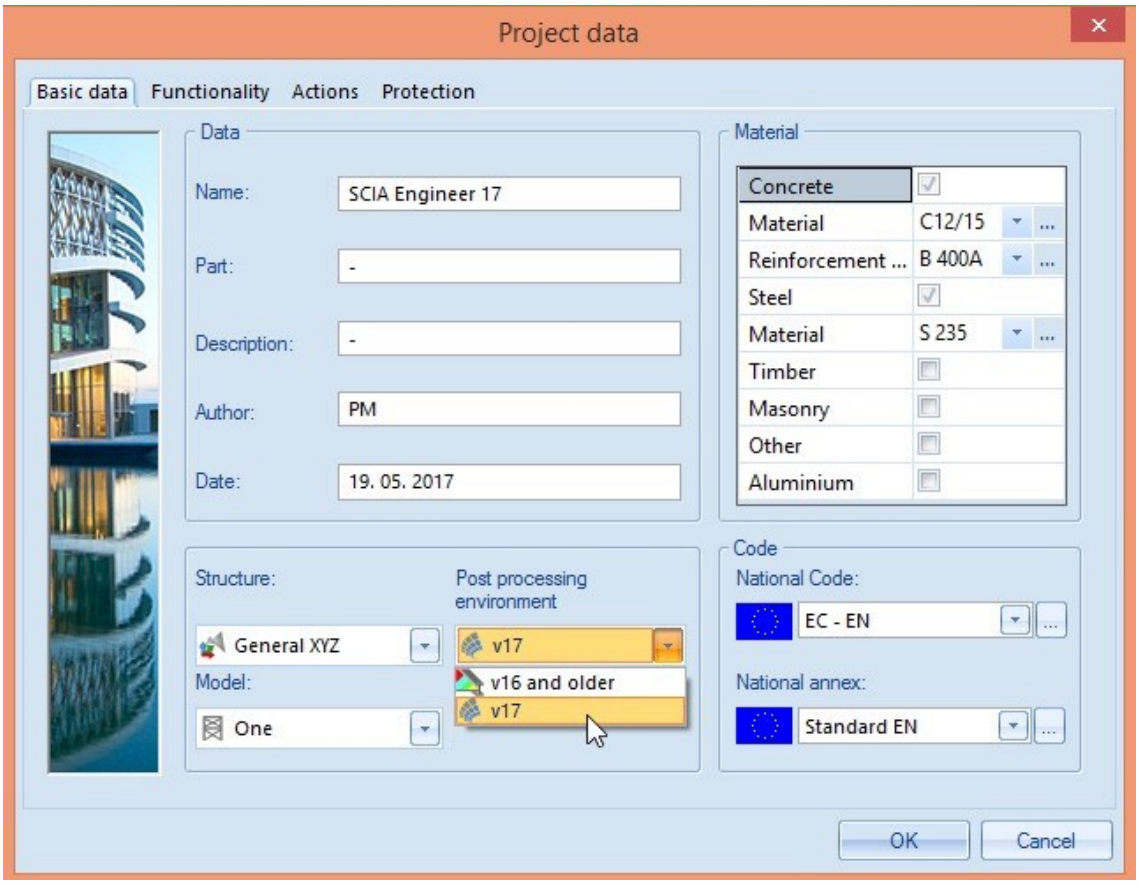
The procedure for setting the parameters is the same as for other project parameters.

v17 Post-processing environment

SCIA Engineer (from version v17) provides the possibility to choose between two post-processing environments for displaying results and performing the design.

Available options:

v16 and previous	This option contains the functionalities of SCIA Engineer 16 and previous versions (all analysis types, results and building standards). Calculation dialog v16
v17	This option contains upgraded visualization styles and results handling enhancements, as well as long term deflections analysis to the Eurocodes. *(Not all functionalities are supported yet)* Calculation dialog v17



More in detail:

What does "post-processing environment" mean?

The post-processing environment provides access to steps occurring after the calculation: visualization of results, code checks design.

The development of the new post-processing environment v17 included redesigning the interface to the solver, to provide a variety of enhancements to the user in terms of speed and accessibility of results.

Some insights into the v17 post-processing environment

The v17 environment covers most of the functionalities which enable you to perform linear calculation (except pre-stressing), nonlinear calculation and modal analysis.

When within this environment you get several benefits in working with results, such as: a faster response while displaying results, new visualization options, a legend with displayed load case / combination, value, units, etc. next to every picture and enhancements in reactions (nodal reactions and intensity in one view).

Some functionalities have not been implemented yet; below you can find an overview of functionalities which are supported or not in the v17 environment.

All options which are supported within v16 and previous, but not yet in the v17 environment, are accessible by switching between the two. There is full compatibility of the model between the two environments.

Supported functionalities the in v17 post-processing environment

Linear calculation	All standard functionalities
Subsoil	Pad foundation
	Soil interaction
	Soil loads

Dynamics	Modal analysis
Nonlinearity	2nd order – geometrical nonlinearity
	Beam local nonlinearity
	Nonlinear soil spring
	Initial deformation and curvature
	General plasticity
Others	Initial stress
	Property modifier
	Linear stability

Functionalities not supported in the v17 environment -> please use the “v16 and previous” environment

Pre-stressing	Pre-stressing
Dynamics	Harmonic band analysis
	General dynamics
	Non-proportional damping
	Seismic
Nonlinearity	Membrane element
	Pressure only 2D members
	Cables
	Non-linear springs, gaps
Concrete	Concrete Advanced
	Fire resistance
	Hollow core slab
	Voided Slab
Others	Climatic loads (3D wind generator)
	Parameters
	Mobile loads
	Absences and construction stages
	Results on integration strip
	Results on averaging strip
	Results on averaging strip
	2D/1D upgrade
	Old document

Notes

Note1: Interaction between the two post-processing environments

The two post-processing environments are fully compatible. Existing projects from previous SCIA Engineer versions or using the v16 post-processing can be run using the v17 post-processing and vice-versa. Every time the post-processing environment is switched, a recalculation of the model is required.

It may happen that you start your project in "v17" and you realize that an advanced functionality you need is missing. You can switch back to v16 post-processing by the "project data" dialogue.

If you change the post-processing environment, as mentioned above you will use another export of the model to the solver, but also another method of showing results and new algorithms which might lead to slightly different result values.

Note 2: Code design

In the Eurocode concrete design (except for the CDD calculation which is available only in "v17"), and steel checks, new results options are available from the property window, independent from the chosen environment

Note 3: Engineering report

The Engineering Report is fully compatible with the two post-processing environments and can also contain images from both environments. Checking the Engineering Report might be required if content from the other post-processing environment is used, to avoid missing results from unsupported features.

Note 4: New developments and the v17 environment

Usage of the v17 environment is recommended to get the benefits from some of the latest developments of SCIA Engineer 17: ability to run a selection of load cases or non-linear combinations, CDD (Code Dependent deflection) for concrete according to the Eurocode, upgraded result service. For more info: visit the page what's new SCIA Engineer 17

Functionality settings

SCIA Engineer offers a wide range of capabilities. In order to make the operation of the program as clear and simple as possible, the project settings allow for selection of those features that are needed and required.

The **Functionality settings** dialogue comprises options that control both the appearance and function of the program. That means that until some advanced feature is selected in this dialogue, the program neither performs the specific task nor even offers it in the menu.

The functionality options are divided into several groups.

Property modifiers

Certain properties of both 1D and 2D members can be modified for the calculation using special attribute called Property m. Properties that can be altered include: stiffness, self weight, mass.

Parametric input

Advanced users of SCIA Engineer may find it very useful to define some of the program input values as parameters. Parameters, if applied, provide for fast, easy and simple change of e.g. structure dimensions, load values, etc. One single modification of the appropriate parameter leads to automatic regeneration of the model with the new defined value.

Climatic loads

If wind or snow loads are supposed to act on the structure, this functionality option must be set ON.

Mobile load

If ON, the model of the structure can be subjected to different types of mobile load.

Subsoil

The Subsoil functionality represents an important and powerful feature of the program especially if the interaction of the analysed structure with its subsoil must be taken into account. The Subsoil functionality is always ON. The following subfeatures can be enabled:

Soil interaction
The interaction between the structure and subsoil can be analysed using the structure-soil interaction module.

Pile design
Design of piles can be performed using the NEN method.

Pad foundation check
Pad foundations can be checkes using the relevant code regulations.

Dynamics

When ticked the option makes the dynamic analysis features available to the user. The appropriate dynamics-related functions and parameters become available in menus and solver adjustment dialogues.

Modal & harmonic analysis
If this option is ON, modal analysis and dynamic calculation with harmonic loading can be performed.

Seismic spectral analysis
If this option is ON, seismic calculations can be performed.

Dynamic time-history analysis
One or more time-dependant dynamic load cases can be defined with name, mass combination, damping, total time and integration steps. Multiple time-dependant dynamic load functions can be used as modal and basic functions. It is possible to use the results in combination with other (static) load cases for further post-processing.

Non-proportional damping
This option enables the user to define separate damping for selected parts of structure.

Dynamic wind
Available only for CSN.

Harmonic band analysis
New way of dealing with the calculations in harmonic analysis by doing multiple analysis on a range of frequencies. Harmonic analysis is possible for a range of frequencies controlled by the user. The frequency of the harmonic force varies over a range and a harmonic analysis is done for many values in that range.

Stability

This option allows the user to calculate stability problems.

Non-linearity

This option controls whether the non-linear analysis is available in solver options and, therefore, whether the user can perform a non-linear calculation of his/her problem. The Non-linearity functionality comprises several sub-items. These sub-items are independent on each other and only some of them may be selected for a particular project.

Beam local non-linearity
If this option is ON, functions for non-linear analysis of 1D members are available (e.g. 1D member acting only under compression, etc. may be analysed).

Support non-linearity/basic soil spring
If this option is ON, functions for non-linear analysis of supports are available.

Initial imperfections
If this option is ON, functions for introduction of initial deformations before calculation are available.

Geometrical nonlinearity
If this option is ON, functions for geometrically non-linear calculation are available.

General plasticity
General plastic analysis of 2D members; several material models are available, such as Tresca, Von Mises, Mohr-Coulomb, Drucker-Prager.

Compression-only 2D members
Masonry or plain concrete walls and similar structural elements can be analysed using this option.

Cables
Specific 1D member nonlinearity for the analysis of cables; combines the behaviours *tension only, no bending stiffness, initial stress* and *initial curved geometry*.

Friction support/Soil spring
If ON, friction supports may be defined in the model, as well as advanced line soil springs

Membrane elements
Thin plates can be calculated as membranes.

Sequential analysis
Two types of calculation can be combined in one analysis.

Dynamic relaxation
Dynamic relaxation can be used as an alternative solver method for nonlinear problems.

Structural model

This option enables the user to use two different "shapes" in his/her model. Normally, the calculation model is created and used for calculations, evaluation of results and design and checking to a particular technical standard.

In addition, the user may also define a structural shape that is derived from the calculation shape and can be used for impressive drawings and is also useful during the design of connections.

IFC properties

This option is useful when the project is imported through the IFC format. The architect may have provided his/her model with some requirements such as required fire resistance, fact that the member transfers no load, etc.

Advanced concrete checks

If ON, old concrete checks (see "Concrete Code Check") are still visible in tree as Concrete Advanced. Otherwise Concrete is there only.

Some settings are available also for the "material" used in the project.

Prestressing

This option provides for calculation of prestressing.

Bridge design

Load combinations
Handling of specific EN code combinations for bridges

Concrete checks extension
Extension of standard concrete code checks for bridges

Slabs with void formers

This option is necessary when slabs with void formers are to be analysed.

Excel checks

If ON, checks in external applications can be performed and the results displayed directly in SCIA Engineer.

Document

This setting enables using of "old" document. Please note this "old" document is not maintained and fixed any more. it is recommended to use Engineering Report instead.

Steel

Plastic hinge analysis
This option enables nonlinear analysis of steel frame structures with plastic hinges.

Fire resistance checks
The type of fire resistance for steel members may be defined after this option has been selected.

Steel connections
This option opens possibility for the definition and check of steel connections.

Scaffolding
Enables the user to design and check scaffoldings.

7DoF 2nd order analysis for LTB
Enables the user to either calculate the elastic critical moment for lateral torsional buckling numerically or to perform a 2nd order analysis which includes lateral torsional buckling.

ArcelorMittal
Enables the user to check cellular beams using the ArcelorMittal ACB software.

Girders with sinusoidal webs
Enables the user to check welded sections with sinusoidal webs.

Aluminium

Scaffolding
Enables the user to design and check aluminium scaffoldings.

7DoF 2nd order analysis for LTB
Enables the user to either calculate the elastic critical moment for lateral torsional buckling numerically or to perform a 2nd order analysis which includes lateral torsional buckling.

Concrete

Fire resistance
The type of fire resistance for steel members may be defined after this option has been selected.

Available only when Old concrete checks are active.

Hollow core slab checks
This option is necessary when hollow core slabs are to be designed.

Available only when Advanced concrete checks are active.

1D physical nonlinearity
The material nonlinearity can be taken into account in the nonlinear analysis of reinforced concrete frame structures.

2D physical nonlinearity
Plate or shell nonlinearity of SFRC 2D members can be accounted for.

CADS Composite add-on

A Composite beam or column can be designed using the CADS composite functionality when this option is set ON. Also fire resistance of the composite elements can be checked if required.

Composite beam
Design of composite beams.

Composite column checks
Design of composite columns.

Fire resistance checks
Fire resistance check of composite members.

Important: the Composite Analysis Model uses a different method and is not compatible with this setting. See the chapter about the Composite Analysis Model for more information.

Pipelines

Pipeline systems can be calculated.

LTA Load cases

When this option is ON specific load cases for Lattice Tower Applications can be defined.

Dynamics

When ticked the option makes the dynamic analysis features available to the user. The appropriate dynamics-related functions and parameters become available in menus and solver adjustment dialogues.

Seismic
If this option is ON, seismic calculations can be performed.

Harmonic band analysis
New way of dealing with the calculations in harmonic analysis by doing multiple analysis on a range of frequencies. Harmonic analysis is possible for a range of frequencies controlled by the user. The frequency of the harmonic force varies over a range and a harmonic analysis is done for many values in that range.

General dynamics
One or more time-dependant dynamic load cases can be defined with name, mass combination, damping, total time and integration steps. Multiple time-dependant dynamic load functions can be used as modal and basic functions. It is possible to use the results in combination with other (static) load cases for further post-processing.

Non-proportional damping
This option enables the user to define separate damping for selected parts of structure.

Dynamic wind
Available only for CSN.

Initial stress

The option, when selected, opens possibility for the introduction of initial stress state in members of a structure being modelled in SCIA Engineer.

Subsoil

The Subsoil functionality represents an important and powerful feature of the program especially if the interaction of the analysed structure with its subsoil must be taken into account.

Soil interaction
The interaction between the structure and subsoil can be analysed using the structure-soil interaction module.

Soil loads
Soil loads can be applied to selected structural members.

Pile design
Design of piles can be performed using the NEN method.

Pad foundation check
Pad foundations can be checkes using the relevant code regulations.

Non-linearity

This option controls whether the non-linear analysis is available in solver options and, therefore, whether the user can perform a non-linear calculation of his/her problem. The Non-linearity functionality comprises several sub-items. These sub-items are independent on each other and only some of them may be selected for a particular project.

Initial deformations and curvature
If this option is ON, functions for introduction of initial deformations before calculation are available.

2nd order – geometrical non-linearity
If this option is ON, functions for geometrically non-linear calculation are available.

Physical nonlinearity for reinforced concrete
The material nonlinearity can be taken into account in the design of concrete structures in Concrete Advanced module.

Plate/shell nonlinearity
Plate or shell nonlinearity can be accounted for.

Beam local non-linearity
If this option is ON, functions for non-linear analysis of 1D members are available (e.g. 1D member acting only under compression, etc. may be analysed).

Support non-linearity/Soil spring
If this option is ON, functions for non-linear analysis of supports are available.

Friction support/Soil spring
If ON, friction supports may be defined in the model.

Membrane elements
Thin plates can be calculated as membranes.

Press only 2D members
Masonry or plain concrete walls and similar structural elements can be analysed using this option.

General plasticity
General plastic analysis of 2D members; several material models are available, such as Tresca, Von Mises, Mohr-Coulomb, Drucker-Prager.

Cable
Specific 1D member nonlinearity for the analysis of cables; combines the behaviours *tension only, no bending stiffness, initial stress* and *initial curved geometry*.

Sequential analysis
Two types of calculation can be combined in one analysis.

Dynamic relaxation
Dynamic relaxation can be taken into account.

Nonlinear line support
If ON, nonlinear line supports may be used in the model.

Stability

This option allows the user to calculate stability problems.

Climatic loads

If wind or snow loads are supposed to act on the structure, this functionality option must be set ON.

Prestressing

This option provides for calculation of prestressing.

Pipelines

Pipeline systems can be calculated.

Structural model

This option enables the user to use two different "shapes" in his/her model. Normally, the calculation model is created and used for calculations, evaluation of results and design and checking to a particular technical standard.

In addition, the user may also define a structural shape that is derived from the calculation shape and can be used for impressive drawings and is also useful during the design of connections.

BIM properties

This option is useful when the project is imported through the IFC format. The architect may have provided his/her model with some requirements such as required fire resistance, fact that the member transfers no load, etc.

Parameters

Advanced users of SCIA Engineer may find it very useful to define some of the program input values as parameters. Parameters, if applied, provide for fast, easy and simple change of e.g. structure dimensions, load values, etc. One single modification of the appropriate parameter leads to automatic regeneration of the model with the new defined value.

Mobile load

If ON, the model of the structure can be subjected to different types of mobile load.

Automated GA drawings

Automated General Arrangement Drawings can be created for the model. This option activates a whole service with sections, hatching and label styles, etc.

LTA Load cases

When this option is ON specific load cases for Lattice Tower Applications can be defined.

CADS Composite Checks

A Composite beam or column can be designed using the CADS composite functionality when this option is set ON. Also fire resistance of the composite elements can be checked if required.

Composite beam
Design of composite beams.

Composite column
Design of composite columns.

Fire resistance
Fire resistance check of composite members.

Important: the Composite Analysis Model uses a different method and is not compatible with this setting. See the chapter about the Composite Analysis Model for more information.

External application checks

If ON, checks in external applications can be performed and the results displayed directly in SCIA Engineer.

Slabs with void formers

This option is necessary when slabs with void formers are to be analysed.

Property modifiers

Certain properties of both 1D and 2D members can be modified for the calculation using special attribute called Property m. Properties that can be altered include: stiffness, self weight, mass.

Document

This setting enables using of "old" document. Please note this "old" document is not maintained and fixed any more. it is recommended to use Engineering Report instead.

Old concrete checks

If ON, old concrete checks (see "Concrete Code Check") are still visible in tree as Concrete Advanced. Otherwise Concrete is there only.

Some settings are available also for the "material" used in the project.

Steel

Fire resistance
The type of fire resistance for steel members may be defined after this option has been selected.

Connection modeller
This option allows the user to model (not check) the connections.

Frame rigid connections
This option opens possibility for the definition of frame connections of steel members.

Frame pinned connections
This option opens possibility for the definition of pinned connections of steel members.

Grid pinned connections
This option opens possibility for the definition of grid pinned connections of steel members.

Bolted diagonal connections
This option opens possibility for the definition of bolted diagonal connections of steel members.

Expert system
If this option is ON, the user may use the expert system for the design of connections. User defined connections may be saved into this system and the saved connections may be applied later to other joints.

Connection monodrawings
This option activates a wizard that helps the user create drawings of defined connections.

Scaffolding
Enables the user to design and check scaffoldings.

LTB 2nd order
Enables the user to either calculate the elastic critical moment for lateral torsional buckling numerically or to perform a 2nd order analysis which includes lateral torsional buckling.

Plastic hinges
This option enables nonlinear analysis of steel frame structures with plastic hinges.

ArcelorMittal
Enables the user to check cellular beams using the ArcelorMittal ACB software.

Girders with sinusoidal webs
Enables the user to check welded sections with sinusoidal webs.

Aluminium

Scaffolding
Enables the user to design and check aluminium scaffoldings.

LTB 2nd order
Enables the user to either calculate the elastic critical moment for lateral torsional buckling numerically or to perform a 2nd order analysis which includes lateral torsional buckling.

Concrete

Fire resistance
The type of fire resistance for steel members may be defined after this option has been selected.

Available only when Old concrete checks are active.

Hollow core slabs
This option is necessary when hollow core slabs are to be designed.

Available only when Old concrete checks are active.

Code depend deflections
Calculation of long-term deflection and deflection from creep.

Bridge design

Load combinations
Handling of specific EN code combinations for bridges

Concrete checks extension
Extension of standard concrete code checks for bridges

The procedure for setting the parameters is the same as for other project parameters.

Loads settings

The procedure for setting the parameters is the same as for other project parameters.

Acceleration of gravity

This parameter defines the acceleration of gravity g. By default $g = 9.81\ m/s^2$.

It is used in the following contexts (non exhaustive list):

- calculation of self-weight of materials based on their mass density (e.g. concrete density = 2'500 kg/m^3 > self-weight = 24.525 kN/m^3)
- calculation of seismic acceleration in some response spectrum generators (e.g. IBC)

Wind region

This parameter defines the region where the modelled structure will be located. The region may influence wind loads that the building will be exposed to. The user may choose from three options for this item:

None	There is no wind load applied.
According to code	The wind region is defined according to appropriate national standard.
User defined	The user specifies the height-wind pressure curve. The real load is then defined as a force load but its type must be set to Wind. The load value input by the user then represents the load width.
User	The user specifies more height-wind pressure curves. The real load is then defined as a force load but its type must be set to Wind.

defined - more curves	The load value input by the user then represents the load width. Each input load has defined a wind pressure curve in its properties.

Wind setup for IBC

If the option According to code is set for IBC code a Wind setup dialog is shown.

Provisions	Defines if formulas and values are used from provisions ASCE7-05 or provisions ASCE7-10.
Importance factor/Risk category	Defines Importance factor category, value and hurricane regions for ASCE7-05 or Risk category and hurricane regions for ASCE7-10.
Directionality factor	Defines Directionality factor according to code for selected structure type.
Gust effect factor	Defines Gust effect factor according to code for selected structure type.
Basic wind	Defines a basic wind value.
Exposure and Topographic factor	Defines Exposure and Topographic factor according to code.
Calculation method	Defines a calculation method - All-heights/Low-rise for ASCE7-05 or Directional/Envelope for ASCE7-10.
Enclosure class	Defines if building is enclosed, partly enclosed or open.

Library of wind pressures

When the option User defined is selected, it is possible to open [using the three-dot button] the Wind pressure database manager. In this manager the user may input the required wind curves.

Editing dialogue for the input of wind pressure

The Editing dialogue for the input of wind pressure can be opened from the Wind pressure database manager.

This dialogue contains the following controls.

Name	Specifies the name of the wind curve.
Graphical window	The diagram of the defined wind curve is displayed in this small graphical window. The graphical window offers standard functions such as zoom-in and zoom-out, pan, copy to clipboard, save to file. The functions can be accessed through a pop-up menu or using mouse move with the right button held pressed with simultaneously held Ctrl and/or Shift key(s).
Table with curve values	The table contains the values that define the shape of the curve. Depending on the option in item Input, this table can be "read-only".
Input	User The curve is defined manually by the user in the table with curve values. EC-EN The EC-EN-defined wind curve is used. For this option the table with curve values is "read only". DIN The DIN-defined wind curve is used. For this option the table with curve values is "read only". NEN The NEN-defined wind curve is used. For this option the table with curve values is "read only". BS-EN The BS-EN-defined wind curve is used. For this option the table with curve values is "read only".

Height range	This parameter defines the height range of the curve. This item is accessible only for code-based wind curves. It is disabled for user-input curve.
Edit curve	This button opens a special where the parameters of the code-based curve can be edited. This item is accessible only for code-based wind curves. It is disabled for user-input curve.
OK	Confirms the changes and closes the dialogue.
Cancel	Discards the changes and closes the dialogue.

Note: The defined wind curves can be reviewed and edited also through tree menu function Library >Loads > Wind pressures. This function becomes accessible only if parameter Wind region in the project setup has been set to Library.

Note: For more information about the generation of wind load see chapter Loads > Load generators > Wind generator.

Snow region

This parameter defines the region where the modelled structure will be located. The region may influence snow loads that the building will be subject to. The user may choose from three options for this item:

None	There is be no snow load applied.
According to code	The snow region is defined according to appropriate national standard.
Snow weight	The user specifies the snow weight per square meter. The real load is then defined as a force load but its type must be set to Snow. The load value input by the user then represents the load width.

Note: For more information about the generation of wind load see chapter Loads > Load generators > Snow generator.

Pond load

The Model factor concerns a reduction factor which is used during the calculation of Pond loads.

More specifically, during the deflection calculation of the water accumulation, the stiffness of the structure is divided by this factor.

Note: More information about Pond loads is given in the chapter Loads > Load generators > Pond water.

Seismic combinations

For codes that support seismic load case combinations with concomitant direction components (e.g. Eurocode), the parameter *Factor for concomitant components* defines the coefficient applied to concomitant direction seismic load cases in a seismic envelope. Typically, with 2 seismic load cases *EQX* and *EQY* and *Factor for concomitant components = 0.30* (default), the seismic envelope will contain the following combinations:

- +/- 1.00 * EQX +/- 0.30 * EQY
- +/- 0.30 * EQX +/- 1.00 * EQY

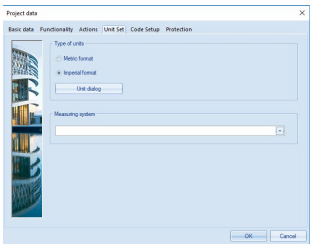
Note: More information about seismic combinations is given in the chapter Seismic combinations - Eurocode.

Combinations settings

This tab provides for the adjustment of load case parameters for automatic generation of load case combinations based on a particular national standard.

The procedure for setting the parameters is the same as for other project parameters.

Units Set tab



In SCIA Engineer the user uses can setup "unit set" which will be used for project.

This set affect two aspects of units Visualisation and default values.

Visualisation and default values

Visualisation	Affect unit in which the value is shown. Example for Imperial unit set: Length - ft L = 10 ft
Default values	When user creating any object in workspace, it uses this logic of default values: 1. Values from previously created object 2. Default value according to unit set, if is defined 3. Program default = values in SI (even if visualisation is set to any imperial based unit) Example: (1) I have already inputted column with my defined length of 12 feats. Now I am inserting column again - it will use previously inputted value - 12 feats. (2) I am inserting column for the first time - it will use default value for metric system - 10 feats. (3) I am creating rectangle cross-section for the first time, and it is not supported by unit set - I will get default values for SI (H = 19.685 inch ; B = 11.811 inch)

Unit "sets"

Metric	Metric units visualisation and SI defaults will be used for newly created elements.
Imperial	Imperial units visualisation and their default values will be used for newly created elements.

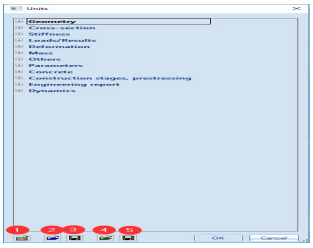
Unit set is automatically applied according to code in "basic data" tab.

EC and others = Metric unit set

IBC = Imperial unit set

Unit dialog

User is able to open and double-check / modify units for chosen set. **This settings affects visualisation of units only**, not default values for new elements - these are set by unit set radio button (Metric / Imperial).



(1) Load program-default settings	Load program default, based on chosen unit set
(2) Load user-default settings	Load quick user default, which was saved by user
(3) Save user default settings	Save and create quick user default
(4) Load settings from file	Load settings from file which was saved by user, convenient when user uses more than one set, or when He wants to load settings saved in previous version of SEN
(5) Save settings to file	Save current settings to file. This file can be used afterwards in next version or different project.

More information about Units here.

Measuring system

Measuring system is available when IBC code and Metric format unit is applied together. More info here.

Procedure for setting project data

The procedure for the adjustment of project parameters is similar for all groups of parameters.

1. Procedure for setting Basic project data
2. Open dialogue Project data in one of the following ways:
    1. Use Menu function Tree > Project.
    2. Click Tree menu item Project.
3. Select the required tab (Basic data / Functionality / Loads / Combinations / Protection / National annexes).
4. Adjust the required parameters or select options that should be applied in the project.
5. Click [OK] to confirm the settings.

Display Setup palettes

SCIA Engineer uses a set of palettes to display project data (i) in the graphical window of the program, (ii) in the document window, and (iii) on an external graphical device. The palette comprises settings of:

- colours,
- line styles,
- fonts,
- dimension lines,
- beam types,
- isolines.

It is possible to adjust separate palettes for individual output "directions". What's more, it is possible to use settings from one palette for another one, i.e. load settings of one palette into the other one.

The available palettes are:

white background	Used for the screen, the structure is drawn in colours on white background.
black background	Used for the screen, the structure is drawn in colours on black background.
document – colour	Used for the document, the structure is painted in colours.
document – monochrome	Used for the document, the structure is painted in black-and-white style. Important note: In order to have the pictures with result-isolines in the document black-and-white, it is necessary to have all the pens / brushes in the Setup dialogue set to black / white / grey colour. Otherwise, if even a single pen is set to any other colour, the whole output is made as coloured.
graphic output – colour	Used for the graphical output (paper space gallery), the structure is painted in colours.
graphic output – monochrome	Used for the graphical output (paper space gallery), the structure is painted in black-and-white style. See the note in document – monochrome above.

The procedure for selection of palettes for individual output "directions"

1. Open any of the following Setup dialogues (all of them can be found under function Setup of the main menu):
   1. Colours / lines,
   2. Fonts,
   3. Beam types.
   4. Dimension lines.
2. At the top of the dialogue, select the tab corresponding to the "device" you want to adjust.
3. In the combo box named Current palette select the required palette.
4. If required, make any changes to the settings (see chapters Colours setup, Font setup, Beam type setup, Dimension line setup for more information).
5. Confirm with [OK].

Loading and saving defined settings

Loading and saving settings for all the palettes at the same time

If required, you may use one the three buttons at the bottom edge of the dialogue to reload or save the settings for all the palettes used in the program.

Load program default settings	This option loads default settings as they were adjusted by the developer of the program.
Store user default setting	This option saves the current settings for all palettes as your personal settings.
Store user default setting	This option loads the settings that have been previously saved by means of the button described one line above.

Loading and saving settings for a separate palette

If required, you may use one the three buttons in the top part of tabs Screen, Document and Graphic output to reload or save the settings for the selected group of parameters. Each of the following buttons works just with one sub-tab of the main tabs, i.e. for example with tab Screen > Fonts, Document >Structural types, etc.

Load program default settings	This option loads settings for the current tab as were define by the manufacturer.
Store user default setting	This option saves the current settings as a user-defined default.
Load user default setting	This option reads the settings that have been previously saved with button Store user default setting.
Load settings from other palette	This option enables the user to load into the current tab settings from the corresponding tab of any other palette.
Convert colours to grey scale	This option converts the colours on the current tab into grey scale. This option is not available for dimension lines.
Convert colours to black	This option converts all the colours on the current tab into black colour. This option is not available for dimension lines.

Colours Setup

Adjustment of colours is a part of settings made for graphical palettes.

The adjustment of colour and line style can be made separately for each entity type and drawing part. The following parameters can be adjusted for each available entity or symbol:

colour	The user may select from a set of basic pre-defined colours or may mix his/her own shade.
line style	The user may select from a set of available line styles.
width	This parameter defines the thickness of the line. If the width type is set to pixels, the user may select the thickness in pixels of the screen. If the width type is set to metric, the user may adjust the thickness in metric units.
width type	This options tell in which units the line thickness is specified. Pixels are useful if the drawing is "tuned" for screen display. Metric option is usually the right choice if the final drawing is made on a graphical device such as printer, plotter, etc.

The procedure for adjustment of colours

1. Open dialogue Colours Setup using menu function Setup > Colours / lines,
2. Make the required settings.
3. Confirm with [OK].

Note: The settings are made separately for individual palettes. For more information about the use of palettes see chapter Display Setup palettes.

Font Setup

Adjustment of fonts is a part of settings made for graphical palettes.

The following parameters can be adjusted:

Character set
Several character sets are available. Proper choice of a character set is important especially when you want to use characters that are specific for a certain language (e.g. diacritics).

Font type
Standard line fond and two variants of true type fonts are available.

TT fonts smooth
If a true type font is selected, its smoothness can be adjusted here.

For each of the texts the following parameters can be adjusted:

size	Specifies the size of labels.
size definition	Specifies how the size is measured. It may be measured in units of graphical device or in absolute units (i.e. the units in which the structure is defined).
colour	This item specifies the colour of the text.
placement	The labels may be put into: the plane of the screen plane XZ plane XY
line font	This option is meaningful ONLY if a true type font is selected in the Font type item. This item selects the font for the labels.
bold	Labels are in bold letters.
italic	Labels are in italic letters.
underline	Labels are in underlined letters.
strikeout	Labels are in stroked out letters.

The procedure for adjustment of fonts

1. Open dialogue Fonts Setup using menu function Setup > Fonts,
2. Make the required settings.
3. Confirm with [OK].

Note: The settings are made separately for individual palettes. For more information about the use of palettes see chapter Display Setup palettes.

Beam type Setup

Adjustment of beam types is a part of settings made for graphical palettes.

For each of the types the following parameters can be adjusted:

colour	The user may select from a set of basic pre-defined colours or may mix his/her own shade.
style	The user may select from a set of available line styles.
width	This parameter defines the thickness of the line. If the width type is set to pixels, the user may select the thickness in pixels of the screen. If the width type is set to metric, the user may adjust the thickness in metric units.
width type	This options tell in which units the line thickness is specified. Pixels are useful if the drawing is "tuned" for screen display. Metric option is usually the right choice if the final drawing is made on a graphical device such as printer, plotter, etc.
middle line	This option specifies the style that is used to display 1D member middle line.
surface	This option specifies the style that is used to display 1D member surface.
labels	This option specifies the style that is used to display 1D member labels.
cross-section	This option specifies the style that is used to display 1D member cross-section.

The procedure for adjustment of beam types

1. Open dialogue Beam types Setup using menu function Setup > Beam types,
2. Make the required settings.
3. Confirm with [OK].

Note 1: The settings are made separately for individual palettes. For more information about the use of palettes see chapter Display Setup palettes.

Note 2: The Setup dialogue supports the standard Windows feature – multiple selection. Therefore, if the same property should be set for several beam types, the types can be selected at the same time and the property adjusted in one step. The multiple selection is accessible via [Shift] + click and [Ctrl] + click combination.

Note 3: For more information about structural types see chapter Geometry > Structural model.

Dimension line Setup

Adjustment of dimension lines is a part of settings made for graphical palettes.

The dialogue enables the user to set the following parameters of dimension lines:

end mark style	This option defines the shape of end mark (slash or arrow).
size definition	This option specifies how the size is measured. It may be measured in units of graphical device or in absolute units (i.e. the units in which the structure is defined).
end mark size	This parameters specifies the size of end mark.
font size	This parameters specifies the size of dimension line font.

plot line style	This parameters specifies the style of plot line.
plot line offset	This parameters specifies the offset of plot line.
1stdimension line offset	This parameters specifies the offset of the dimension line closest to the dimensioned object.
next dimension line offset	This parameters specifies the offset of other dimension lines.

The procedure for adjustment of dimension line style

1. Open dialogue Dimension lines Setup using menu function Setup > Dimension lines,
2. Make the required settings.
3. Confirm with [OK].

Note: The settings are made separately for individual palettes. For more information about the use of palettes see chapter Display Setup palettes.

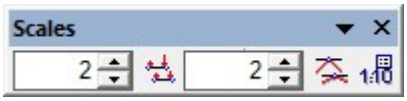
Adjusting the scales

The entities displayed on the screen are displayed in a specific scale. The user can control the scales through settings made in the Scales manager.

The scales are stored and can be adjusted in the Scales manager. This allows the user to have several "sets" of defined scales for different purposes and to simply swap between them.

The Scales manager always contains the set named "Current". The "Current" set is always associated to graphical windows. All graphical windows use the same set of scales. In addition, you can define as many user-defined sets of scales as required. When you create a new set of scales and assign it to the graphical window, this set is stored in the Scales manager under the name you define and it is also copied to the "Current" set. This way, the graphical windows are always associated with the "Current" set of scales.

Toolbar Scales



Scales toolbar contains the following controls:

Spin control for fast adjustment of scales for additional data
This control enables you to quickly multiply all the scales for additional data by the factor in the spin control.

Button "Autofit scales for data" to recalculate the scales for additional data
This button recalculates the adjusted scales for additional data so that the drawing fits the screen. The ratio between scales for individual entities is maintained, but the absolute value of the scales is changed. The button performs no action if the Scale type in the "Current" set of scales is set to Symbol size.

Spin control for fast adjustment of scales for results
This control enables you to quickly multiply all the scales for results by the factor in the spin control

Button "Autofit scales for results" to recalculate the scales for results
This button recalculates the adjusted scales for results so that the drawing fits the screen. The ratio between scales for individual entities is maintained, but the absolute value of the scales is changed. The button performs no action if the Scale type in the "Current" set of scales is set to Symbol size.

Button "Scales" for access to Scales manager

This button opens the Scales manager.

Scales manager

The Scales manager is a standard database manager. It can be used to:

a) create a new set of scales,

b) edit the existing set of scales,

c) activate one of the defined sets of scales,

d) copy, delete, export or import the sets of scales.

Note: The "Current" set of scales cannot be deleted.

Scales parameters

General parameters

Name
Specifies the name of the set of scales. (The "Current" set of scales cannot be renamed).

Group data

Scale type
Symbol size

You define the absolute size of the symbol that is used for each type of entity. The multiplier is taken into account during drawing.

Real ratio
You define the scales for individual types of entities. This scale is used directly (taking into account the multiplier) to display the data.

Automatic ratio

You define the scales for individual types of entities. These scales, however, are used only to determine the ratios between the size of individual entities. The absolute size is determined using the following algorithm: the largest entity (e.g. the largest force) is so scaled, so that its size in the graphical window is 1 metre. All other entities are scaled using the calculated ratios. The multiplier is taken into account, which means that if the multiplier is set to 2, the size of the largest entity is 2 metres..

Multiplier
This multiplier is used to increase (or decrease) the real size of the displayed entities.

Point data
This value specifies the scale for "point" data such as point load, concentrated moment, etc.

Line data
This value specifies the scale for "line" data such as line load, line moment, etc.

Surface data
This value specifies the scale for "surface" data such as surface load, etc.

Group Result

Scale type
See Group Data (above).

If Symbol size option is selected for the results, the behaviour is similar to Automatic ratio, but each group of entities (beams, slabs, etc.) is treated separately. It means that the largest value is determined for every group of entities.

Multiplier
See Group Data (above).

Reaction, Deformation, Internal forces, Stress, Contact stress, Unity check, Other results

The value specifies the scale for individual type of result value.

Group Symbols

Scale type
See Group Data (above).

Multiplier
See Group Data (above).

Point symbols, Line symbols, Surface symbols, Structure node symbol, Local axis symbols, Other symbols
The value specifies the scale for individual type of symbols.

The procedure to open the Scales manager

a) Use menu function Setup > Scales, or

b) click button Scale () on toolbar Scales.

The procedure to create a new set of scales

a) Open the Scales manager,

b) Click button [New].

c) Define the name of the Scales set and, if necessary, adjust the individual parameters.

d) Close the Scales manager.

The procedure to assign a set of scales to the graphical window

a) Open the Scales manager,

b) Select the required Scales set.

c) Close the Scales manager.

Document Setup

The Document Setup dialogue enables the user to adjust default values for the style of document. The parameters are described in chapter Document > Adjusting the document default settings.

Picture gallery Setup

The Gallery Setup dialogue enables the user to adjust default values for style of pictures inserted into or created in the Picture gallery. The parameters are described in chapter Graphic output >Picture gallery > Picture gallery manager > Inserting a new picture into the Picture gallery > Adjusting the default values for new pictures.

Note: The settings adjusted in this dialogue are taken into account whenever a new drawing is inserted into the picture gallery by means of Picture to gallery function (). For example, if the default picture style is set to "wire", the drawing from the graphical window is inserted as "wired" even though it was e.g. rendered in the graphical window. The style may be later edited in the Picture gallery manager.

FE mesh Setup

Finite element mesh is generated automatically by the program. The user, however, may specify parameters that control the generation process.

These parameters may be defined in the calculation dialogue or in the program setup.

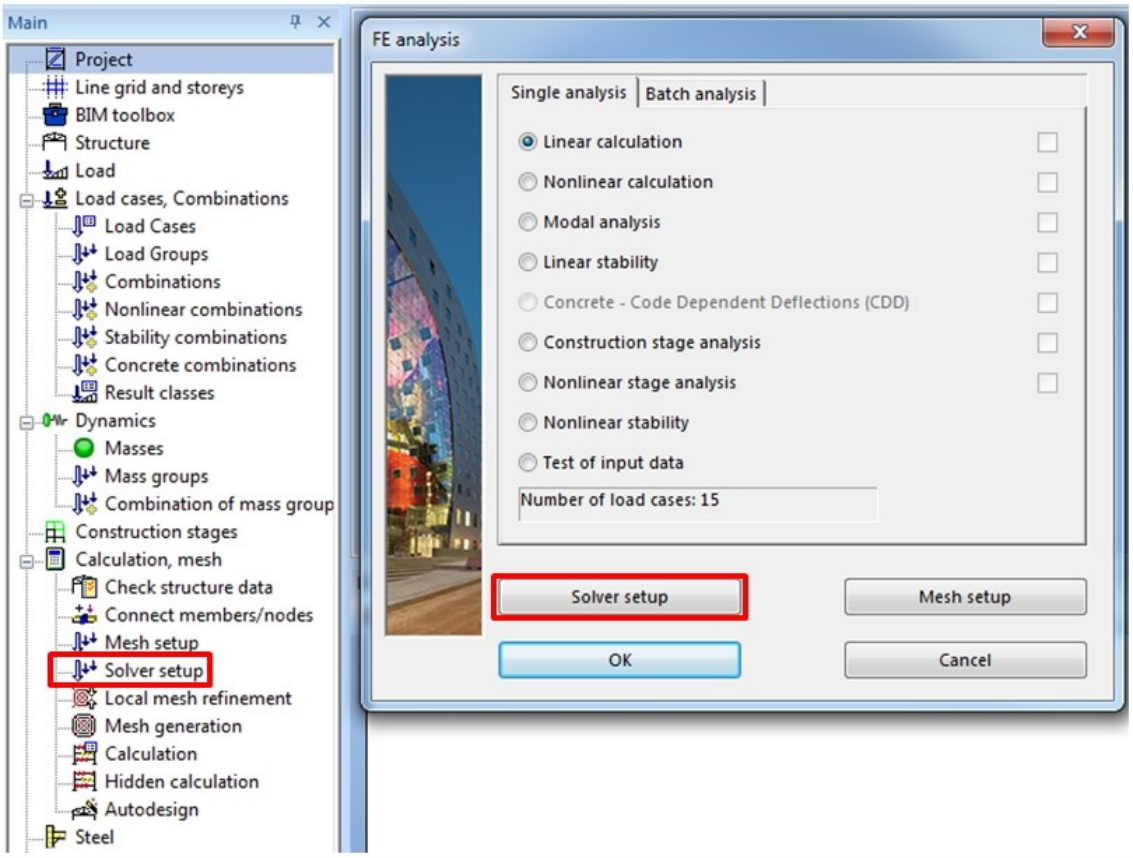
The setup dialogue can be opened using menu function Setup > Mesh.

The meaning of individual parameters is given in chapter Calculation > Generating the FE mesh > Parameters of FE mesh.

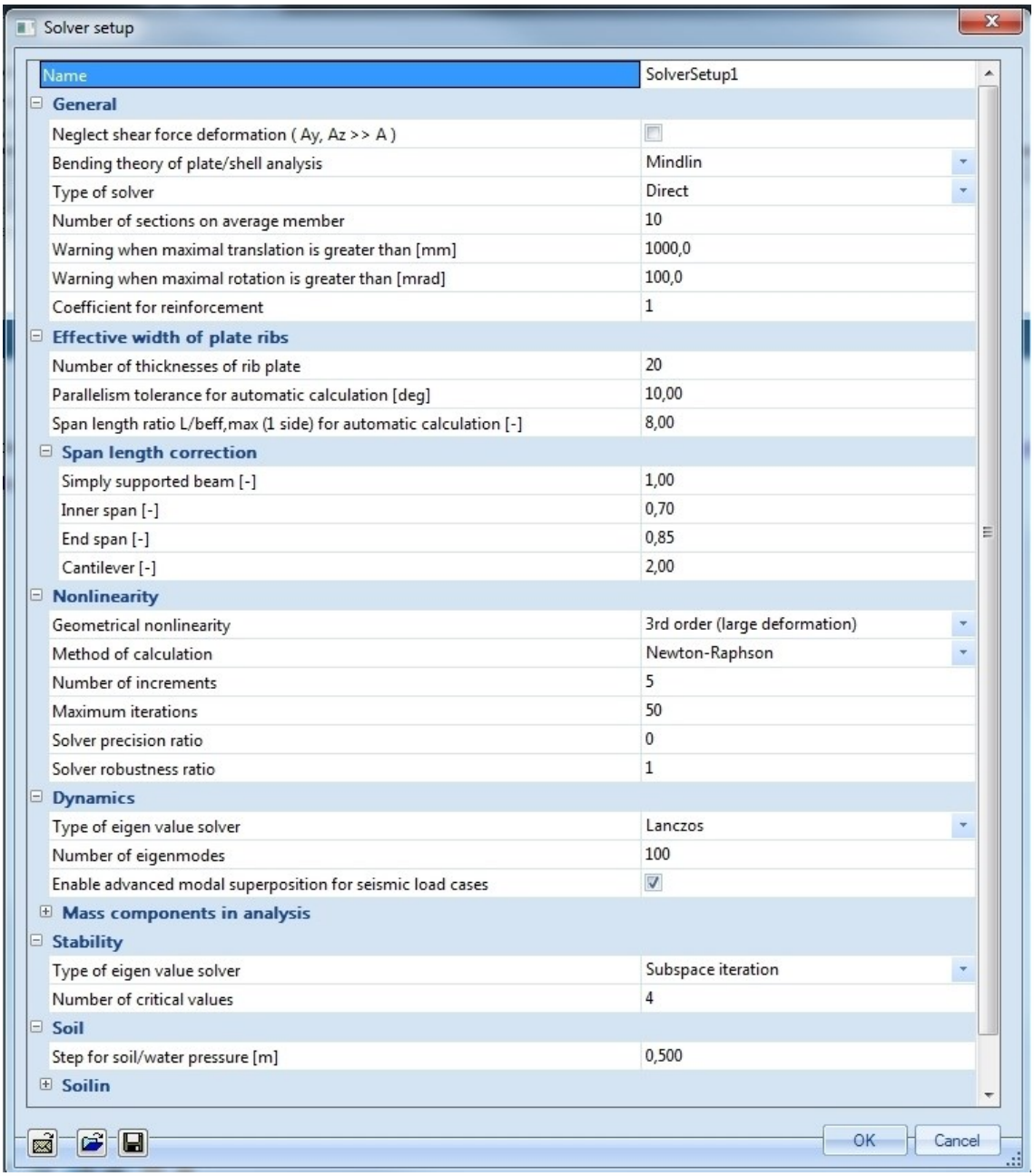
Solver Setup

This setup dialogue provides for adjustment of basic parameters controlling the calculation.

Solver setup is possible to open in two ways. Either in standard project tree in group "Calculation, mesh", or in the dialog "FE analysis" just before the calculation is executed..



Parameters in Solver setup are divided into several groups according analysis type (Nonlinearity, Dynamics...), or as parameters for special functionality (Soil, Mass...).



Solver setup groups of parameters

- General

This group of parameters is available for all type of calculations in the project.

There is only limitation in "type of structure" in "project data" dialog where there is no settings for slabs in for example project type "frame".

Parameters for this group

- Neglect shear force deformation

- Bending theory of plate/shell analysis

- Type of solver

- Number of sections on average member

- Warning when maximal translation is greater than

- Warning when maximal rotation is greater than

More information about these parameters is here : " General calculation parameters"

- Effective width of plate ribs

This group of parameters is available for all type of calculations in the project.

There is only limitation in "type of structure" in "project data" dialog where there is no settings for slabs in for example project type "frame".

Parameters for this group

- Number of thickness of rib plate

- Parallelism tolerance for automatic calculation [deg]

- Span length ratio L/b_{eff},max (1 side) for automatic calculation [-]

- Span length correction

Parameters in subgroup "Span length correction"

- Simply supported beam

- Inner span

- End span

- Cantilever

More information about these parameters is here : "Automatic Calculation of the Effective Width of Plate Ribs"

- Nonlinearity

This group of parameters is available only for nonlinear calculation, in the project with some nonlinear functionality and non-linear combination.

Parameters for this group

- Geometrical nonlinearity

- Method of calculation

- Number of increments

- Maximum iterations

- Solver precision ratio

- Solver robustness ratio

- Plastic hinge code

- Geometrical nonlinearity

- Allow compression in membrane members

More information about these parameters is here : " Static non-linear calculation"

- Dynamics

This group of parameters is available only for dynamic calculation, in the project with some dynamic LC and dynamic load.

Parameters for this group

- Type of eigen value solver
- Number of eigenmodes
- Use IRS (Improved Reduced System) method
- Produce wall eigenmode results (needed for ECTools)
- Enable advanced modal superposition for seismic load cases
- Method for time history analysis
- Mass components in analysis

Parameters in subgroup "Mass components in analysis"

- Translation along global X axis
- Translation along global Y axis
- Translation along global Z axis
- Rotation around local X axis of 1D members (torsion)

More information about these parameters is here : "Dynamic natural vibration calculation"

- Stability

This group of parameters is available only for stability calculation, in the project with stability functionality and Stability combination.

Parameters for this group

- Type of eigen value solver
- Number of critical values

- Soil

This group of parameters is available only if functionality "Subsoil" is available in the project.

Parameters for this group

- Soil combination
- C_{1x}
- C_{1y}
- C_{1z}
- C_{2x}
- C_{2y}
- Thickness of loose layer at contact level [m]

More information about these parameters is here : "Required parameters for Soil-in calculation"

Advanced geometry setup

The parameters in the first group are identical with some items from the Parameters controlling the alignment of the structure. These values are used for all geometrical operations and for your convenience they are added into this dialogue as well

Geometric tolerances

Min. distance of two nodes, node to curve	Specifies the min. distance of two nodes for which the two nodes are considered separate nodes. If the real distance of two nodes is lower than this parameter, the two nodes are merged together. This parameter is used by the function for connection of entities and by the function for check of data.
Max. distance of node to 2D member plane	Specifies the maximal allowable distance of a node from the plane of a 2D member. If the actual distance is larger than this limit value, the geometry is considered invalid and a corresponding warning is issued. Note: The algorithm tries to find a new plane with normal vector, for the given definition points, which fulfils the condition for maximum distance of the node to plane. It means if there is input a node outside the plane then a new basic plane is found between these points. If all points are closer than the defined maximum value then it is considered as a valid state.

Displaying

Precise member surface	This parameter comes into account only if surfaces are switched on. If ON, the shape is displayed as precisely as possible. If OFF, only the schematic shape of the cross-section is displayed. The parameter has meaning in particular for steel rolled sections.
Immediate refresh of structural model	If ON, the structural model is automatically refreshed after all changes. If OFF, the structural model is refreshed manually on user's request.
Precision of displayed curves	This parameter control smoothness of curves and curved surfaces. The higher the number the smoother the curve. On the other hand, the higher the number the slower the response of your computer may be. The parameter must be from interval <1, 10>. The parameter does not affect the precision of the calculation.
Precision of cut-out mesh	This parameter controls smoothness of the displayed shape of intersecting surfaces. The parameter does not affect the precision of the calculation.

Load panels

Disable automatic update of load panels	If ON, during manipulation, adding and removing members selection of beams and edges supporting load panels is not updated. The update can be forced manually by pressing the action button **Update all load panels** which is in **Properties** of each panel. If OFF, beams and edges supporting load panels are automatically updated during manipulation actions.

Code-check parameters and National Application Documents

National Application Documents

Scope of the chapter

This chapter describes the way the parameters for code checks of concrete, steel and aluminium structures according to EC-EN code can be adjusted in SCIA Engineer 2010 and later versions. The manual presents the environment in which the parameters can be set and explains the procedures how to do it.

This chapter covers ONLY the EN-EC code. The dialogues and procedures for other codes may be different. For those differences we refer to the other parts of the documentation where proper explanation is given.

In particular, Steel Code Checks manual, Concrete Code Checks manual and Aluminium Code Checks manual are useful sources of information.

Code Check Parameters

Code checks for any material require that numerous parameters appearing in the check formulas be properly defined. Some of these parameters form a part of National Application Document, some result from the procedures implemented.

The program enables the user to adjust all these parameters in what is called Concrete Setup, Steel Setup, Aluminium Setup, etc.

The layout and behaviour of these Setups will be explained for the Concrete Setup. For other materials, the principles are the same.

National Application Document versus other code-check parameters

The Concrete Setup can be opened from several places of the program, i.e. using different functions. The behaviour of the Concrete Setup may differ depending on the way it was opened. This is important mainly for one reason. In order to make the Setup dialogues clearer, you can never see in it together the (i) parameters of the National Application Document and (ii) other parameters.

National Application Document parameters

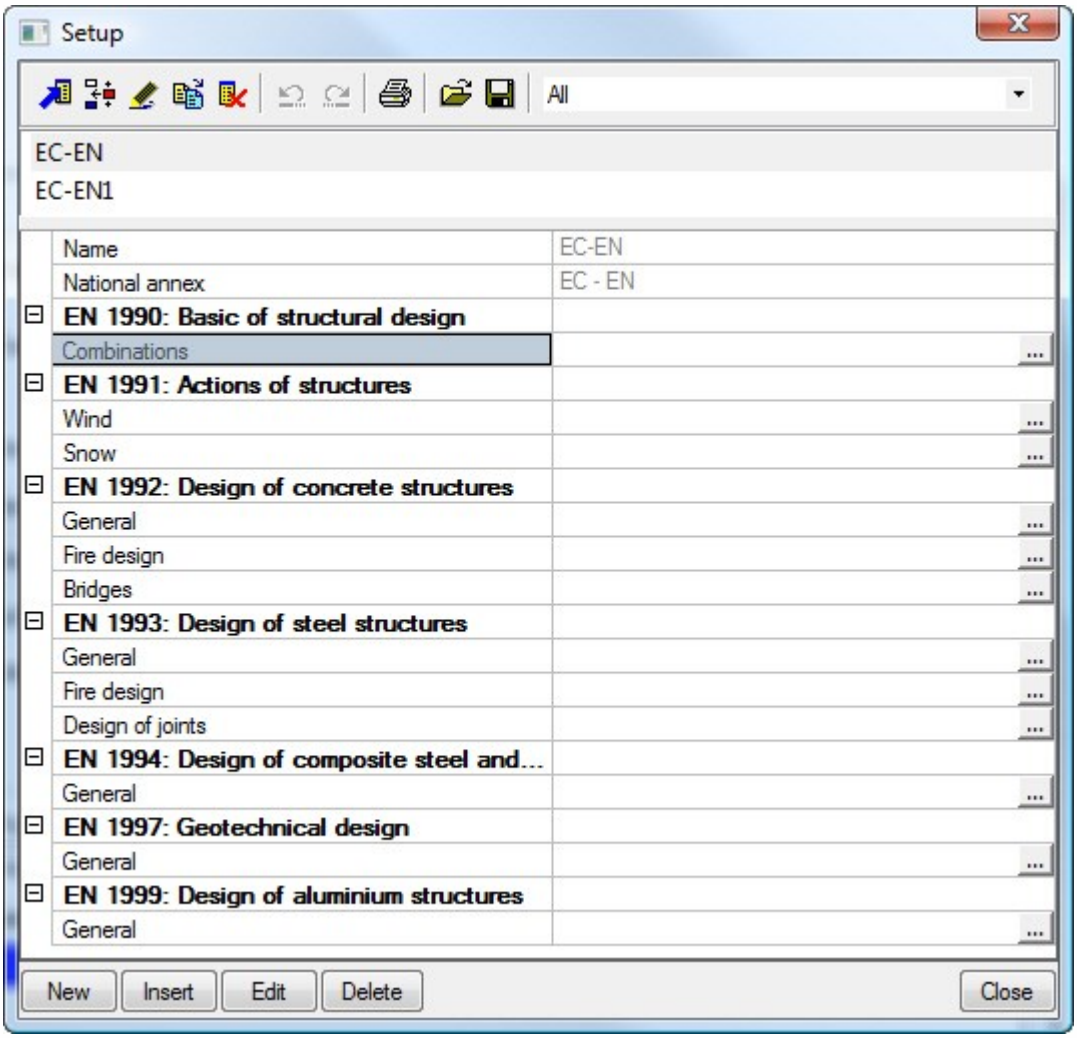
To see, or modify, National Application Document parameters (hereinafter NAD parameters), you must open the Concrete Setup Dialogue in the following way:

1. In the main tree, double click item Project.

2. This opens the Project data dialogue.

3. In it, click the three-dot button next to item National annex.

4. This opens the Setup Manager dialogue.



This dialogue s a standard database manager as e.g. Material Manager, Load Case Manager, etc. It means that you can define as many sets of NAD parameters as required and you can simply select the set that is the most appropriate one for your current work. The sets can also be stored on the disk and read to another project.

By default, one set (EC-EN in the picture) is defined automatically by the program. This set ensures that even if you do not open the Setup Manager, the parameters are defined and the program can use them.

What is important about this default set is that it is READ-ONLY. It means that you cannot alter the values of the parameters. If you need different values for some parameters, you must define a new set (e.g. as a copy of the default one) and then you can adjust the required parameters.

List of defined sets of parameters
The top part of the dialogue lists all the sets that have been defined in the project.

Individual National Application Documents
The bottom part of the dialogue contains a list of individual codes and their parts. Each item is accompanied with a three-dot button that opens a separate dialogue where individual parameters can be reviewed or modified (with the exception of the default set that is read-only).

Other code-check parameters

To see, or modify, other code-check parameters, you must open the Concrete Setup Dialogue from different place than from the Project Data dialogue. There are several ways to do so, each of them showing certain specifics.

Libraries > Setup

Function Libraries > Setup (opened either from the tree menu or from the main menu) opens the Setup Dialogue with all available parameters (except the NAD parameters).

Concrete > Design Defaults

Function Concrete Advanced > Design Defaults opens the Setup Dialogue with just the basic parameters (again except the NAD parameters).

Concrete > Member Data

Button [Concrete Setup] in the input dialogue of function Concrete Advanced > Member Data opens the Setup Dialogue with parameters that are relevant for the selected type of member (again except the NAD parameters).

Setup > Concrete Solver

Menu function Setup > Concrete Solver opens the Setup Dialogue with just the parameters relating to the solver for concrete structures (again except the NAD parameters).

Setup Dialogue

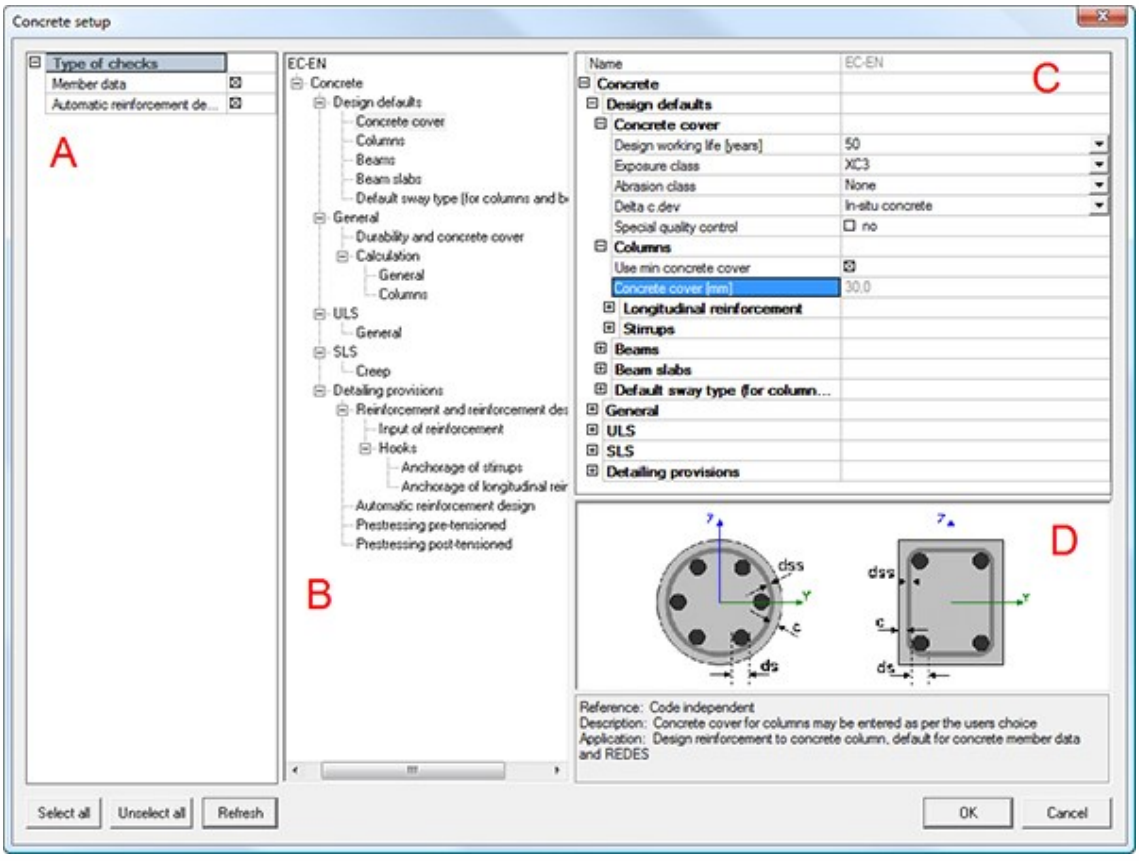
In general, the Setup Dialogue consists of four parts (see the picture below):

• filter window (A),

• tree window (B),

• property window (C),

• explanation window (D).



Filter Window (A)
This window contains a list of "filters" that can be used to reduce the number of parameters displayed in other parts of the dialogue.

Note: This window may be hidden for some configurations of the dialogue.

Tree Window (B)
This window contains a list of parameter groups arranged in a tree structure.

Clicking on a branch in the tree expanses the corresponding part of the property Window (with other branches automatically collapsed).

Property Window (C)
This window lists all the parameters that are available for the current configuration of the dialogue. The parameters are sorted in groups (branches). You may expand or collapse the required groups.

Explanation Window (D)
Once an item is selected in the Property Window, an explanation is displayed in this window. If possible, the explanation is accompanied with an illustrative picture.

National Annexes

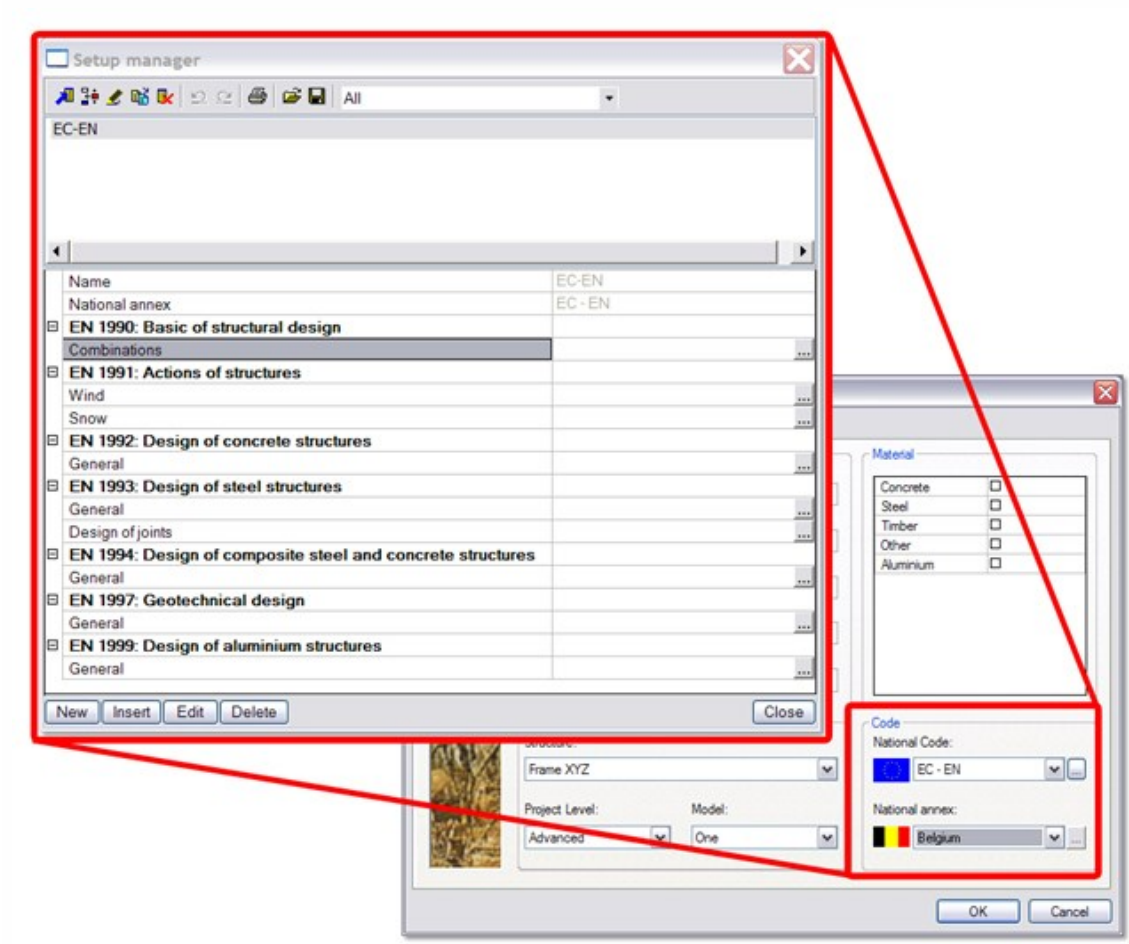
Supported National Annexes

The current coverage of national annexes in SCIA Engineer can be found in the SCIA Resource Centre or on our Eurocode microsite.

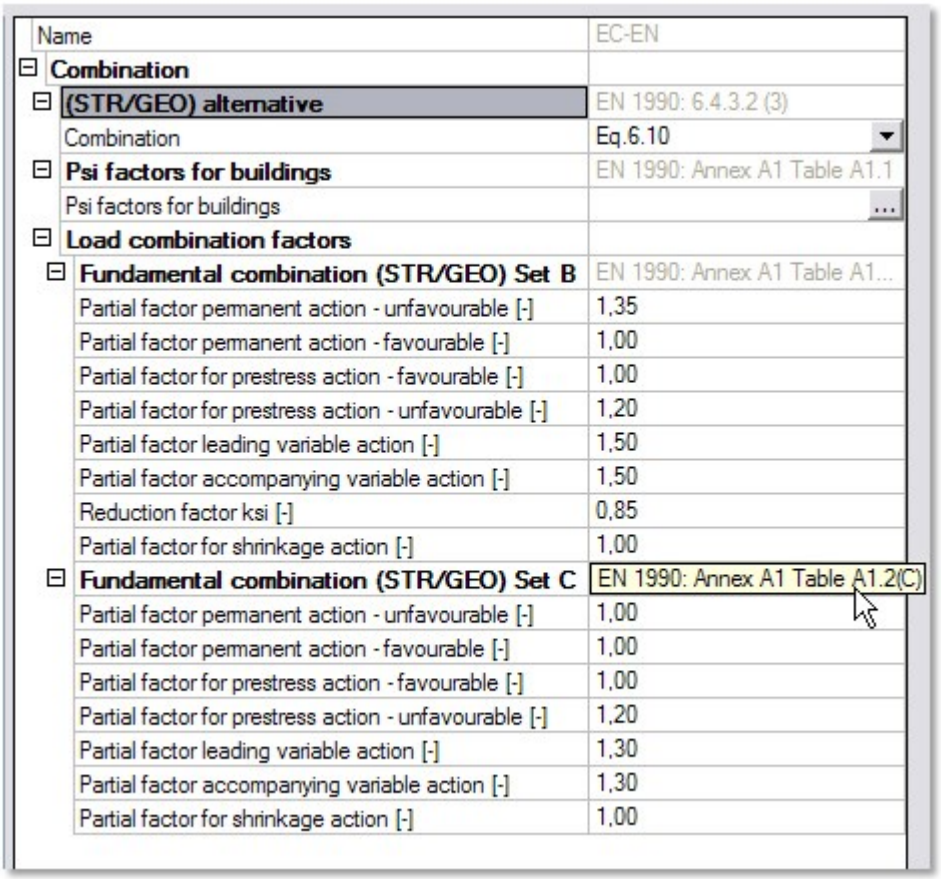
The parameters described in the National Determined Parameters (NDP) in the foreword to each EN Eurocode part have been extracted and collected in one National Annex dialogue. On condition that the code of the project is set to Eurocode, the user can select the required National Annex and the National Annex dialogue becomes available, where the National Determined Parameters can be reviewed. All parameters are open, which means that the user can edit the values if desired. In this way, the parameters can be adapted according to the appropriate National Annex. This gives the possibility to use the National Annex of each European country.

Furthermore, it is possible to save the settings and use them in future projects.

Project Data dialogue with opened National Annex Manager



National Determined Parameters for EC0



Name	EC-EN
Combination	
(STR/GEO) alternative	EN 1990: 6.4.3.2 (3)
Combination	Eq.6.10
Psi factors for buildings	EN 1990: Annex A1 Table A1.1
Psi factors for buildings	...
Load combination factors	
Fundamental combination (STR/GEO) Set B	EN 1990: Annex A1 Table A1...
Partial factor permanent action - unfavourable [-]	1,35
Partial factor permanent action - favourable [-]	1,00
Partial factor for prestress action - favourable [-]	1,00
Partial factor for prestress action - unfavourable [-]	1,20
Partial factor leading variable action [-]	1,50
Partial factor accompanying variable action [-]	1,50
Reduction factor ksi [-]	0,85
Partial factor for shrinkage action [-]	1,00
Fundamental combination (STR/GEO) Set C	EN 1990: Annex A1 Table A1.2(C)
Partial factor permanent action - unfavourable [-]	1,00
Partial factor permanent action - favourable [-]	1,00
Partial factor for prestress action - favourable [-]	1,00
Partial factor for prestress action - unfavourable [-]	1,20
Partial factor leading variable action [-]	1,30
Partial factor accompanying variable action [-]	1,30
Partial factor for shrinkage action [-]	1,00

Basic working tools

Selections

Introduction to selections

Whenever the user needs to do anything with any part of his/her model, s/he must, first of all, determine which part of the model should be treated. In other words, the user has to make a "selection" of members that will be processed.

Once the selection is defined, the required operation may be started. The selection may be formed by a single entity or it may hold as many entities as required. Generally, the selection may contain entities of the same type, or it may contain several entity types. Which of the two cases is applied depends on the intended operation. Some operations require specific entity types, other operations may be carried out with any entity types.

In general, there are two approaches to start an operation:

- the user first makes the selection and then starts the appropriate function (the function then deals with the prior made selection),
- the user first starts the required function and then (i.e. from within the function) makes the selection.

Which approach is actually applied depends only on work habits of a particular user.

To sum up, the selection can be not only made and utilised in a function, but it can be also modified (reduced or extended), cleared, saved into a file for later use or loaded from a previously created file.

Selections are controlled by:

- Menu View > Selections,
- Selections toolbar.

Making a selection

In order to make a selection, the program must be in the selection-enabled mode. This mode is the default mode of the program and only a limited number of functions changes this mode into a selection-disabled mode. The selection-enabled mode is identified by the mouse cursor that looks like a diagonally oriented arrow with a small square attached to the tip of the arrow. Once this cursor is on the screen, it is possible to make selections freely.

There are two basic ways to make a new selection: using the mouse or typing a command on the command line. In both ways it is a piece of cake.

In addition, a selection can also be made via filters. That means, that the user specifies a condition that should be fulfilled by all selected entities. For example, the user may specify the condition that the cross-section must be a rolled IPE 300. The filter-controlled selection then looks for and selects all 1D members with such a cross-section.

Making a selection by the mouse cursor

When using the mouse cursor, there are several selection modes:

single selection	One entity is selected each time the user clicks the mouse button.
intersection	The user draws a line (or a polygon) on the screen. The program selects all entities that have an inter-

line	section with the drawn line.
rectangular cut-out	The user draws a rectangle on the screen. The program selects all entities located inside the rectangle or overlapping it (see the paragraph below for details about this selection mode).
polygonal cut-out	The user draws a closed polygon on the screen. The program selects all entities located inside the polygon.
working plane	The program selects all entities located in the current working plane.
select-all	All currently displayed entities are selected
previous	Activates the last made selection.

How to activate the required selection mode:

selection mode		via toolbar Selections	via menu Tools > Selections
Single selection		click button [Selection by mouse]	call function Selection by mouse
Intersection line		click button [Selection by cut-out]	call function Selection by intersection line
Rectangular cut-out		click button [Selection by intersection line]	call function Selection by cut-out
Polygonal cut-out		click button [Selection by polygonal cut-out]	call function Selection by polygonal cut-out
Working plane		click button [Select by working plane]	call function Select by working plane
Select all		click button [Select all]	call function Select all
Previous		click button [Previous selection]	call function Previous selection
Single selection mode toggle		swaps between "First found" and "All found" mode	see paragraph Selection of entities with overlapping edges
Selection mode toggle		swaps between "Select" and "Deselect" mode	see topic Removing the entities from selection
Visibility selection mode		swaps between "normal" selection mode and a special mode for OPGL	see paragraph Visibility selection mode see also paragraph Visibility selection acceleration

Single selection

In order to make a selection, the user has to:

1. place the mouse cursor on the entity he/she wants to select,
2. click the left mouse button.

That is all that is necessary to make a selection by mouse. To add another entity, the user just puts the cursor on another entity and clicks the left mouse button.

Intersection line

When this mode is invoked, all entities that are intersected by a defined line are added into the selection. The line may be either a single straight line or a polygon consisting of straight lines.

The procedure to define a polygon

1. Position the mouse cursor to the place where the polygon should start.
2. Click the left mouse button.
3. Position the mouse cursor where the end point of the polygon line segment should be located.
4. Click the left mouse button.
5. Repeat the previous two steps as many times as required.
6. Close the polygon, ie. either
    1. press [ESC] key, or
    2. invoke the pop-up menu, select End polyline command and run it, or
    3. define the last point with a double-click on the left mouse button.

Cut-out

This mode enables the user to select all entities located inside a mouse defined cut-out. There are two different kinds of the cut-out. The first one serves for selection of entities located fully inside it. The other one can be used to select entities that are both fully inside and overlap the cut-out.

The procedure to define a cut-out that selects inside-located entities only

1. Place the mouse cursor to the TOP LEFT corner of the rectangular cut-out.
2. Press the left mouse button and hold it down.
3. Drag the mouse to the BOTTOM RIGHT corner of the rectangular cut-out.
4. Release the button.

The procedure to define a cut-out selecting both inside-located and overlapping entities

1. Place the mouse cursor to the TOP RIGHT corner of the rectangular cut-out.
2. Press the left mouse button and hold it down.
3. Drag the mouse to the BOTTOM LEFT corner of the rectangular cut-out.
4. Release the button.

Polygon

This mode is similar to the previous one. The difference is that the user draws an arbitrarily shaped closed polygon instead of a simple rectangle.

The procedure to define a polygonal cut-out

1. Position the mouse cursor to the place where the polygon should start.
2. Click the left mouse button.
3. Position the mouse cursor where the next vertex of the polygon should be located.
4. Click the left mouse button.

5. Repeat the previous two steps as many times as required.
6. Close the polygon:
   1. either press [ESC] key, or
   2. invoke the pop-up menu, select Close polygon command and run it.

Working plane

In this mode, the program automatically selects all entities located in the current working plane.

Select-all

All displayed entities are automatically selected.

Selection of entities with overlapping edges

In a real-life model it is frequent situation that several entities (e.g. beams, walls) meet in one place (joint, corner). In that case it may be difficult to select the proper entity, because when you place the mouse cursor over the intersection of these entities, the program does not know, which one to select. To solve such situations, the program offers a special toggle: Single selection. This switch enables you to work in two modes:

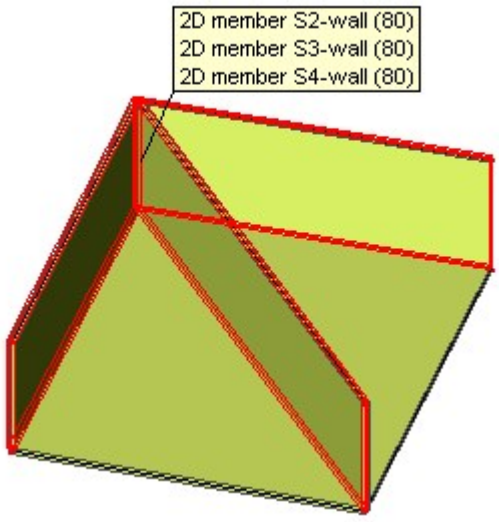
First found

In this mode, the first entity found by the selection algorithm is selected (usually, it is the entity that was input first).

All found

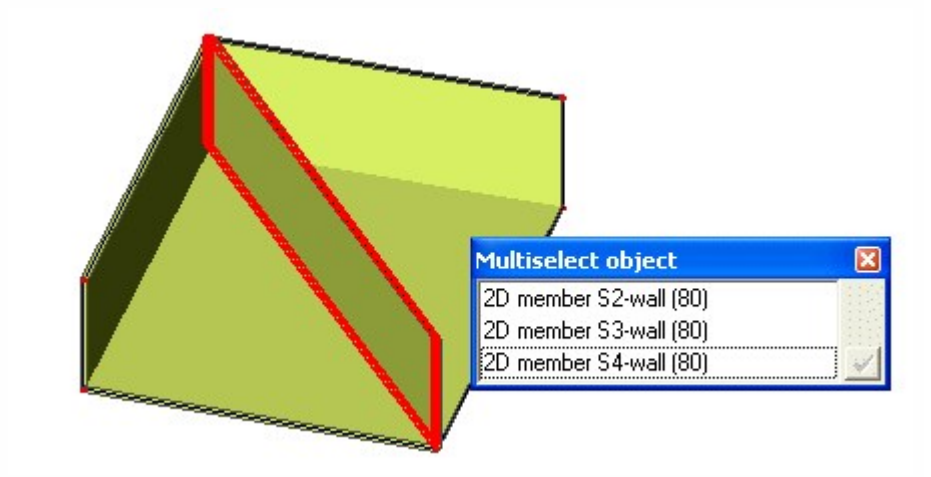
In this mode, the program finds all entities under cursor and offers you a list of them so that you may decide yourself which one(s) should be selected.

Imagine a simple model of three walls.

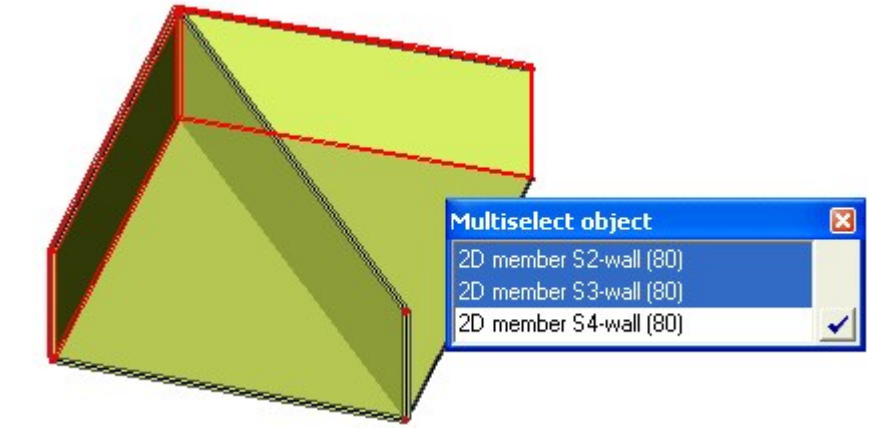


If you place the cursor over the corner in which the three walls meet and click the left mouse button, the program opens a small dialogue with a list of found entities.

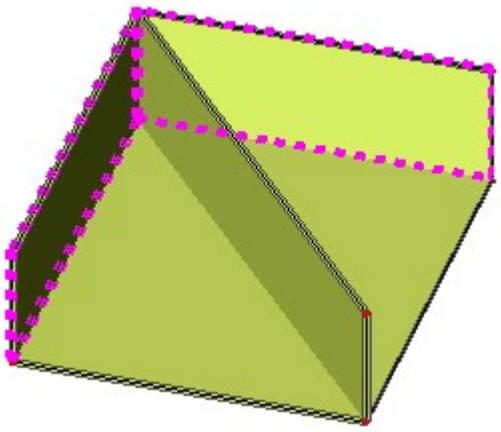
You may roll the mouse cursor over the list. The entity over which the cursor is just placed is highlighted in the graphical screen, so it is easy to find out which entity is which.



If you want to select a particular entity, just click on its name in the list. You may select as many entities as you want.



When you press the green-mark button, the selection is confirmed.



Visibility selection mode

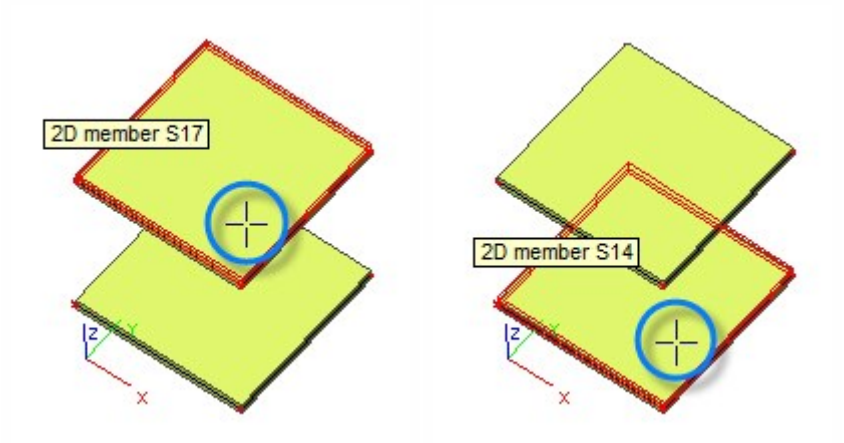
In the "normal" selection mode, you must select an edge of an entity in order to select it.

However, if the Visibility selection mode is activated, you may just put the mouse cursor anywhere on the displayed member and it can be selected. The only precondition is that Rendering display style is active.

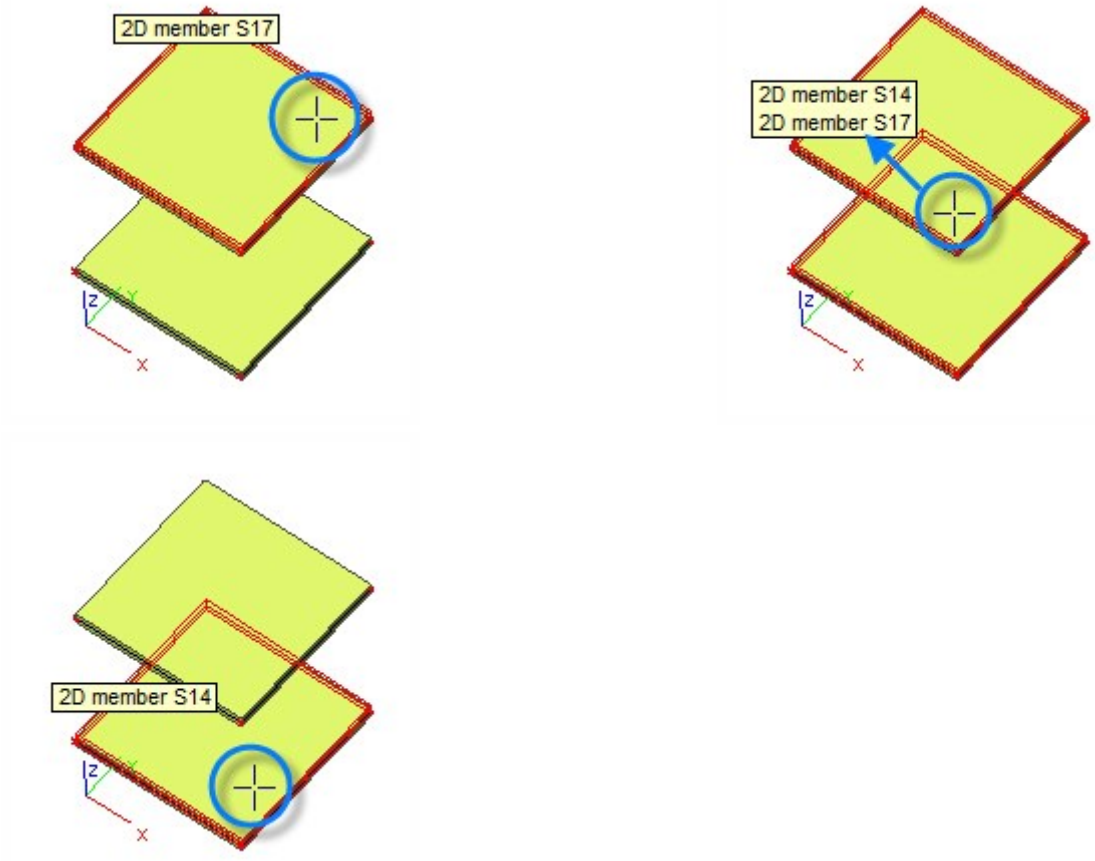
The Single selection mode toggle is taken into account in the Visibility selection mode.

Examples (the little cross in the blue circle indicates the position of the mouse cursor):

A) Single selection mode toggle set to FIRST FOUND



B) Single selection mode toggle set to ALL FOUND



Please note that the cursor changes its shape when the program is in the visibility selection mode.

Visibility selection acceleration

When you use the Visibility selection mode, the acceleration can be used to speed up the manipulation with large models. However, it is important to know that the final effect of the acceleration depends on the model of the structure and that the acceleration may, under certain circumstances, even slow down the program response.

The principle of the acceleration is that the model that is displayed in the graphical window is converted to an OPGL scene (a special graphical scene optimised for the graphical card) that is usually processed faster by the graphical card. However, in order to allow for selections, the OPGL scene must handle also a list of all members that are displayed. And this may be the core of the problem. If the "workload" related to the management of this list exceeds the "workload" related to displaying of the graphical scene, the effect of the acceleration may be negative. This can be better understood on the following example.

Let us have a structure model composed of 1000 members (beams and columns).

A) All the members have a rectangular cross-section. It is quite a simple task for the graphical card to display such a simple shape (even if it is repeated 1000 times). On the other hand, the maintenance of the list of 1000 items is a rather complex matter. As a result, if the acceleration is ON, the response of the program will be most likely slower.

B) All the members have a circular (pipe) cross-section. It is more complex task for the graphical card to display such a shape. As a result, the acceleration will probably have no effect. In other words, the time saved during the display operation equals the time needed for the management of the list of the members.

C) All the members have a cross-section of a complex shape (e.g. a complex aluminium profile). In this case, the time savings reached due to faster display operation are much greater than the time-losses due to the management of the list of the members. Which means that the acceleration has a positive effect.

The acceleration can be switched on/off in menu Tools > Selection.

Filter-controlled selection

The filter-controlled selection is useful if the user wants to select all entities that meet a specific condition. This type of selection is described in the following chapter.

Making a selection from the command line

A selection can be also made (sometimes very effectively) from the program's command line.

The procedure is similarly simple as the "mouse procedure". The user types a command on the command line and the selection is made.

Command syntax

SEL [switch] parameter [parameter2] [parameter3] [etc.]

or

SELM [Enter]

name1 [Enter]

name2 [Enter]

...

END [Enter]

The latter alternative provides fo multiple selections. SELM + [Enter] starts the multi-selection mode. Then you can type the names of required entities one by one – each one followed by [Enter] key. The selection can be completed with command END (followed by [Enter] key).

Switch

switch	meaning
+	adds into selection
-	subtracts from the current selection
|	inverts the current selection

Parameter

parameter	example	description
entity name	SEL BEAM23	selects entity named BEAM23
entity name with a wild-card	SEL BEAM2?	selects all entities whose name starts with BEAM2 and is followed with a single character
	SEL B?	if 1D members are named B and numbered, this command selects all "one-digit" 1D members
	SEL B??	if 1D members are named B and numbered, this command selects all "two-digit" 1D members
	SEL B*	selects all entities whose name starts with letter B
	SEL B1 B2	selects entities named B1 and B2
NONE	SEL NONE	clears the selection

Examples

sel none	clears the selection
sel *	selects all entities
sel N1	selects entity N1
sel + N*	adds into the current selection entities whose name starts with N
sel – B*	removes from the current selection entities whose name starts with B
sel | B1	inverts entity B1 in the selection (i.e. if the entity WAS in the selection, it is removed; if the entity WAS NOT in the selection, it is added)

Making a selection of a mesh node from the command line

To select a mesh node by its number, command SELMN can be used.

Procedure to select required mesh nodes

1. Type command SELMN in the SCIA Engineer command line and press Enter.

2. The selection function starts.

3. Type the number of the node to be selected and press Enter. The node is marked with an arrow.

4 . If required, repeat step 3.

5. Press Esc to end the command.

IMPORTANT: In order to see the arrow denoting the selected node(s) it is necessary to switch ON the appropriate flag in the View parameters settings Dialogue - Miscellaneous > Calculation info > Display arrow on mesh elements.

Procedure to remove the highlighting arrow from selected mesh nodes

1. Select the arrow or the accompanying text with the mouse cursor.

2. Press key Delete on your keyboard OR use the Delete function from the pop-up menu OR use the Delete function from the Modify menu.

Removing the entities from selection

When using complex and extensive selections, it may be necessary or at least useful to remove a particular entity or entities from the already made selection.

In general, there are two ways to remove an entity from an existing selection: "[Ctrl] key" method and "Inverted selection mode" method.

"[Ctrl] key" method ("[Shift] key" method)

All the selection modes for making a selection can be used as well for removal of specific entities from the current selection. In order to activate the "subtract from selection" mode, the user must press down and hold [Ctrl] key on the keyboard.

Note: The [Ctrl] key method in fact INVERTS the selection (i.e. it is not REMOVAL from the selection, but an unselected item is selected and a selected item is deselected).

Note: The [Shift] key can be used instead of the [Ctrl] key - these two keys are inter-changeable (in this function).

Example 1:

Let's assume that a selection of some entities has been already made. Now, the user needs to remove one particular entity.

The procedure will be:

1. Position the mouse cursor over the entity that should be extracted from the selection.
2. Press down and hold [Ctrl] key.
3. Click the left mouse button.
4. The entity is removed from the selection.
5. Release [Ctrl] key.

Example 2:

Let's assume that a selection of some entities has been already made. Now, the user needs to remove a few entities that are parallel to each other and located close to each other.

The procedure will be:

1. Select Intersection line selection mode.
2. Position the mouse cursor next to the first entity that should be removed from the selection and outwards from the others.
3. Press down and hold [Ctrl] key.
4. Define the intersection line, i.e. the line or polygon intersecting all the required entities.
5. Close the intersection line.
6. Release [Ctrl] key.

"Inverted selection mode" method

It is also possible to press button [Selection mode toggle] () on toolbar Selections. All the selection modes described in chapter Making a selection then remove entities from the previously made selection.

> Note: It is also possible to remove entities from selection using command "SEL" typed on the command line with the appropriate switch and parameter. For more information see chapter Making a selection.

Making a selection based on a specific property

Very often the user needs to select all entities that meet some specific condition. For example, to select all 1D members made of one material type or all supports allowing for free movement in X-direction, the filter-controlled selection is the right choice.

The procedure to apply a filter-controlled selection

1. Select one entity that meets the required condition.
2. In the property table click the left column cell of the row that contains the required condition.
3. Click icon [Quick select] () at the top of the property dialogue frame.

> Note: This type of selection may be used to select e.g.

> - all the 1D members of the same cross-section,

> - all the slabs of the same thickness,

> - all the entities located in the same layer, etc.

Adjusting the filter for selections

Sometimes it may be very useful to limit the selection on some entity types only. SCIA Engineer enables the user to specify the filter for selections.

There are three filter options:

OFF	The filter is OFF and any entity of any type may be selected.
For service	Function for making a selection recognises only those entities which the active service can deal with.
For tree	The type of entities that can be selected is defined by the position of cursor in the tree.

Remark: In case of active one click main tree option is turned ON, selection in main tree is set on right-click

Filter for service

If this filter option is selected, the set of entities for selections is defined by the currently opened service. The user can select only those entities that the service can deal with.

For example, if service Loads is open, and this filter option is ON, only 1D members, nodes, and loads of all types can be selected.

Filter for tree

If this filter option is selected, the set of entities for selections is defined by the currently opened service. and by the position of cursor (by the focus) in the tree menu. The user can select only those entities that are specified by the function "under focus".

For example, if service Loads is open, and this filter option is ON, and the focus in on function Line force on beam, only line loads can be selected.

The procedure to adjust the required filter

1. Click button [Filter] on the Status bar.
2. A short menu is opened.
3. Select the required filter.

The alternative procedure for the adjustment of the required filter

1. Press in button [Filter for selection on/off] () on Selection of objects toolbar in order to select Filter for service.
2. This action makes another filter button available – [Filter by service tree on/off] ().
3. Press in button [Filter by service tree on/off] on Selection of objects toolbar in order to select Filter for tree.

Modifying a selection

Any existing and active selection may be modified, i.e. some of the selected entities may be removed from it and some other entities may be added to it.

Removal of entities from the selection

In order to remove an entity from the current selection, follow the procedure given in chapter Removing the entities from selection.

Adding another entity into the selection

In order to add another entity into the current selection, simply follow the procedure for making the selection. Until you clear the selection, any new selected entities are added to the current selection.

Applying a selection

A selection is usually made to carry out an action (i.e. call one or more of SCIA Engineer functions). In fact, vast majority of SCIA Engineer functions works with a selection and modifies the entities in the selection according to defined functionality. Therefore, it must be clear how to associate the selection with the required action. Fortunately, this crucial step is completely automatic and absolutely straightforward in SCIA Engineer despite the fact that there exist two opposing approaches.

Applying a pre-created selection

This approach leads to the following steps:

1. Select the required entities.
2. Start the function.
3. The function "works" with the previously made selection.

Applying a post-created selection

On the other hand, this approach means:

1. Call the required functions.
2. Select the entities that should be treated with the function.
3. The function then processes the in-function-defined selection.

Both approaches have their advantages. The latter is useful mainly if the user wants to apply the same function on several different selections. It is possible to change the function parameters for each particular selection, but the main function itself must be called just once.

Clearing a selection

If a selection is no longer useful, or if it was made improperly (e.g. wrong entities have been selected), or if any other reason occurs, the selection may be cleared. It means that the selected entities are removed from the selection but NOT from the project. Just the selection is emptied.

There are several ways to clear a current selection:

- Press [ESC] key,
- Click [Cancel selection] () button on the Selections toolbar,
- Call function Cancel selection from menu Tools > Selections.

All the possibilities are equivalent to each other.

Saving and reading a selection

Any selection made for any purpose can be saved to a disk for later re-use.

Any selection can be saved or loaded through appropriate functions from the Selections toolbar or via menu Tools > Selections.

The selections are stored with the project. It is however possible to export the selections out of the project and use them in another project. This export can be made in the Selection manager (see below).

There are, in principle, three actions that can be done with selections:

- a new selection can be saved (done through Save selection dialogue),
- an existing selection can be loaded (done through Selection manager dialogue),
- an existing selection can be updated (done through Selection manager dialogue).

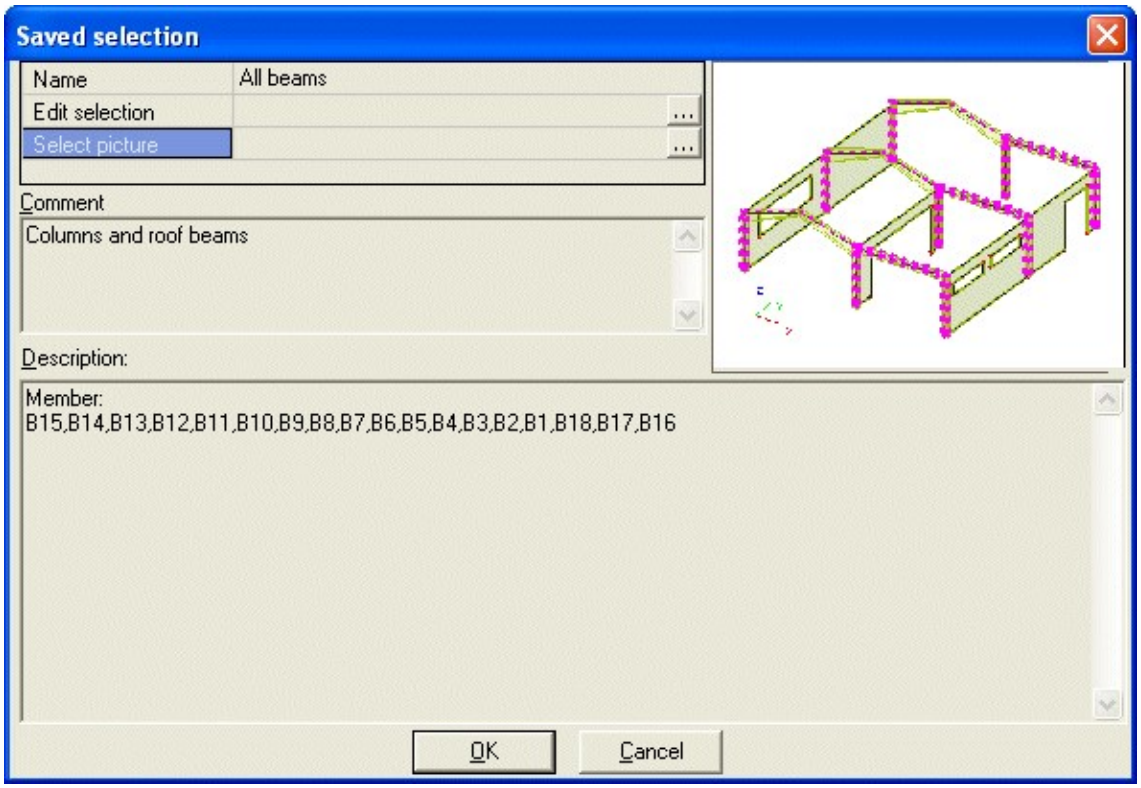
In addition (as already stated), any selection can be exported from the project to an external file that can be later imported into another project.

> Note: Be careful when using one selection (EPS) file with multiple projects. The program makes no special checks and mechanically reads the selection from the file. However, the entities stored in the selection that do not exist in the project are, naturally, ignored.

Saving a new selection

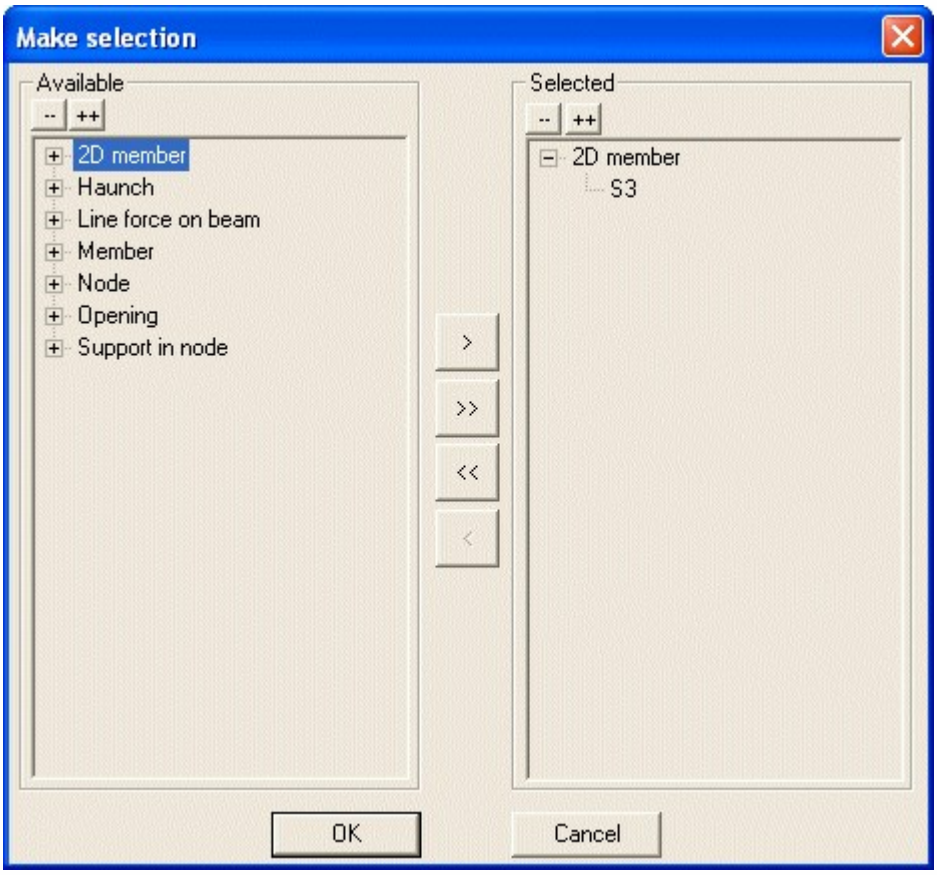
When a selection is being saved, the Saved selection dialogue is opened on the screen.

Saved selection dialogue



Name	You can assign an arbitrary name to each saved selection.
Edit selection	The selection that was made in graphical window before invoking this dialogue can be further modified or reviewed in the Make selection dialogue – see below. This option can be also useful when a new selection is made directly in the Selection manager – see below.
Select picture	Each saved selection can be accompanied with an illustrative picture – e.g. screen copy – that may be worth thousand words in explaining which entities are in the selection in question.
Comment	You may add e few lines of comment.
Description	This field contains an automatically generated list of all entities included into the selection.

Make selection dialogue



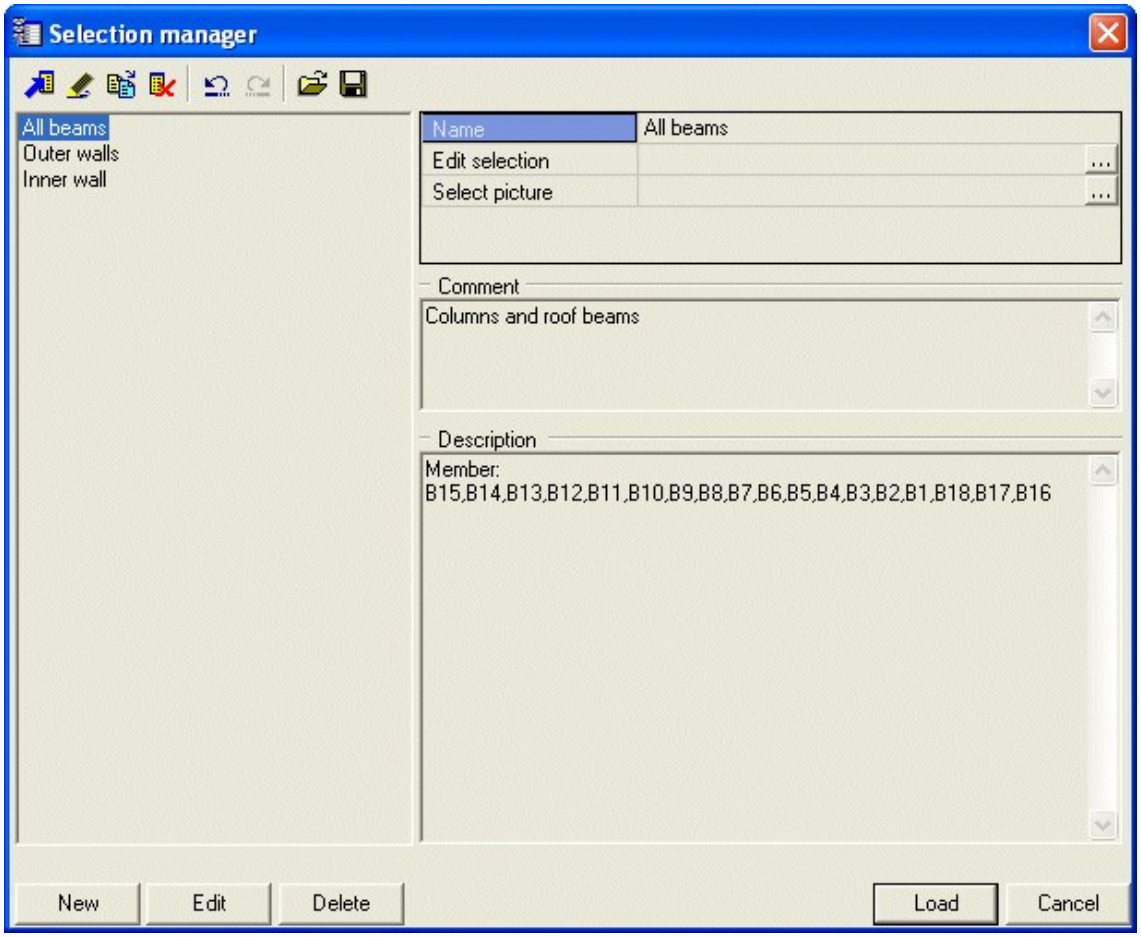
On condition that you know the names of individual entities in your project, you can manually add or remove the required entities to or from the selection.

Available (entities)	This list contains a list of all available entities that can be included into the selection. This list does not contain the entities that have been already inserted into the current selection.
Selected (entities)	This list names the entities that have been inserted into the currently edited selection.
Button [--]	This button collapses all the branches of the tree with the list of (available/selected) entities. Each list in the dialogue has its own button.
Button [++]	This button expands all the branches of the tree with the list of (available/selected) entities. Each list in the dialogue has its own button.
Button [>]	Use this button to move the highlighted item from the "available" to the "selected" list, i.e. add it to the selec-

	tion.
Button [>>]	Use this button to move all the items from the "available" to the "selected" list, i.e. add them to the selection.
Button [<]	Use this button to move the highlighted item from the "selected" to the "available" list, i.e. remove it to the selection.
Button [<<]	Use this button to move all the items from the "selected" to the "available" list, i.e. remove them to the selection.

Loading a saved selection

Once a selection was saved, it is possible to load it back for use with any function that works with selections. Any saved selection can be loaded through the Selection manager. The Selection manager is a standard SCIA Engineer database manager.



Selection manager

New	Creates a new selection.
Edit	Edits the existing selection.
Copy	Creates a copy of an existing selection.
Delete	Deletes the existing selection.
Undo, Redo	Takes back the last action done in the manager.
Read from	Reads a selection that was saved to an external EPS file.

disk	
Save to disk	Saves the selection into an external file with extension EPS.
Selection properties	The right-hand side of the Selection manager dialogue contains the information about the selection. The content is identical to the Saved selection dialogue described above.

Updating a saved selection

Any saved selection can be updated any time later, if required. The update is made through the Selection manager. That is however modified a bit. The button load is replaced with button [Update].

The fact that the update uses the Selection manager has one major advantage. Let us assume that you loaded a saved selection, modified it in the graphical window, started function Update named selection and only then you realise that you do not want to lose the original selection – that it is still useful and necessary. You are in the Selection manager, so instead of selecting (and updating, i.e. changing) one of the existing named selections, you can create a new selection and use it for the update function. In other words, you can swap from "Update" to "Save as new" even if the update function is already in the progress.

The procedure to save a new selection to a disk

1. Make a selection.
2. Start function Save selection (either on toolbar Selections or in menu Tools > Selections).
3. Select Save as new from the submenu.
4. Fill in the parameters in the Saved selection dialogue.
5. Confirm with [OK].

The procedure to update the existing selection

1. Load the required selection (see below for the procedure).
2. Modify the selection as required.
3. Start function Save selection (either on toolbar Selections or in menu Tools > Selections).
4. Select Update existing from the submenu.
5. The Selection manager is opened on the screen.
6. Select the selection to be updated.
7. (If you change you mind, you can create a new empty selection to be "replaced" by the updated selection).
8. Confirm with [Update].

The procedure to read a selection from a disk

1. Start function Load selection (either on toolbar Selections or in menu Tools > Selections).
2. Select the selection you want to read.
3. Confirm with [Load].

Selections versus editing of properties

Selections are very advanced feature of SCIA Engineer. They do not provide just for the passive selection of entities that will be further treated in some way. The selections represent a powerful tool for editing of the project.

The principle is that whenever whichever entity is inserted into a current selection, its properties are automatically and immediately displayed in the property window of the application.

If multiple entities of the same type are selected then the intersection of their properties is displayed in the window. If multiple entities of different types are selected, the user may choose the type whose properties should be displayed. It is of course possible to simply swap between the types once the parameters for one type have been reviewed.

What's more, any data displayed in the property window can be edited and the change is immediately recorder.

Editing in the property window for one selected entity

If just one entity is selected, the property window shows it's properties and, if possible, co-ordinates of its endpoints. Once the user changes any of the values in the property window, the change is recorder and the entity is re-displayed to reflect the changes.

Editing in the property window for multiple selected entities of the same type

If several entities of the same type are selected, the property window displays the intersection of their properties. That means that the dialogue contains values of those parameters which are identical. If any parameters are of different value for different 1D members in the selection, the value cell in the property window is left blank.

The user may once again edit any item in the property window. This relates even to the blank cells. If a value is input into any of the cells, that value is assigned to all the entities in the selection.

Editing in the property window for multiple selected entities of various types

Here the same can be said as in the paragraph above. What's more, the combo box at the top part of the property window contains a list of all types whose entities are in the current selection. When the used selects any item from this list, the properties of this entity type are shown in the property window.

The user may then review or edit them as described above.

Note: See also chapter Controlling the selection-versus-editing process.

Controlling the selection-versus-editing process

The principle of editing in the Property window as described in chapter Selections versus editing of properties can be controlled by means of settings made in the Environment settings.

The Environment settings dialogue contains item Maximum number of grouping properties. This items tells the program what is the maximum number of entities for which the "selection-versus-editing" process should be started.

In other words, if the user selects fewer entities than specified in the parameter Maximum number of grouping properties, the Property window is filled in with the parameters of selected entities. Consequently, the parameters can be easily edited.

On the other hand, if the number of selected entities is greater than the number specified in parameter Maximum number of grouping properties, the Property window is left blank. If required, the Property window may be filled in manually by pressing button [Update property dialogue] () located at the top right corner of the Property window..

This feature may be useful particularly for large projects with a great number of entities. The time that is necessary to collect and sort all the parameters of all selected entities is growing with the number of selected entities. In addition, it is assumed

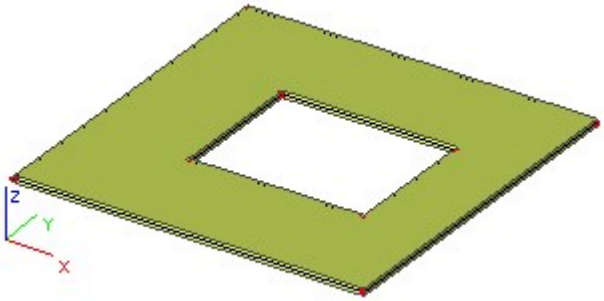
that usually the user will select only a limited number of entities for direct editing in the Property window. And, if the user selects a really vast number of entities, it is assumed that the selection was made for some of manipulation functions and not for direct editing.

Therefore, it is possible to make as large selection as necessary and apply any of manipulation function to it, but the Property window is not filled in for excessive selections. If, however, the user does want to edit directly even the enormous number of entities, he/she may fill in the Property window manually by means of the above mentioned button.

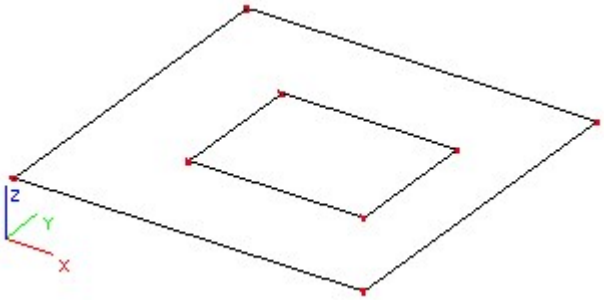
Selections of slabs with openings

If a slab has an opening or a subregion, there are a few rules concerning the selection of such a slab.

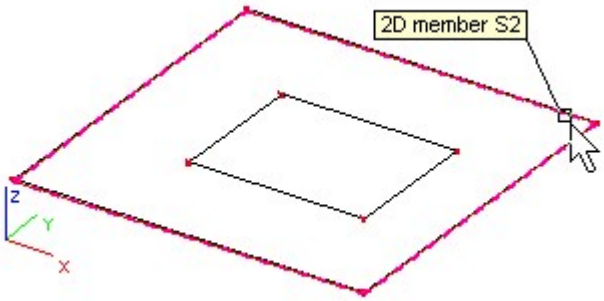
Let's assume a simple rectangular slab with an opening.



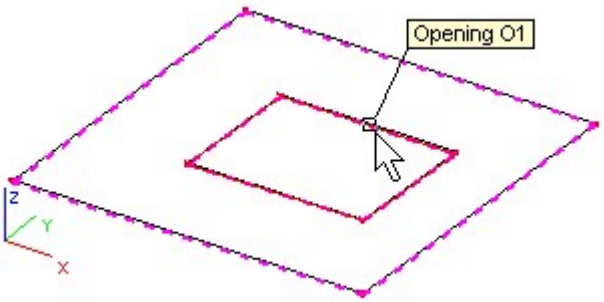
Adjust the view parameters so that only the middle line of a slab is displayed on the screen



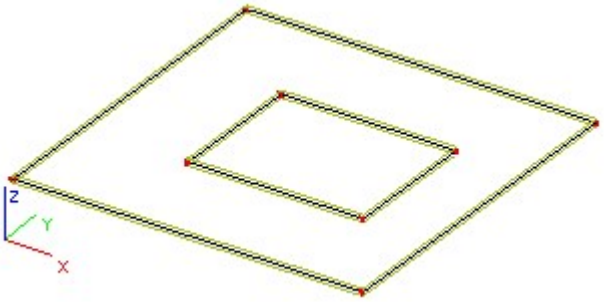
If you select the outline of the main slab, the main slab is highlighted and also selected.



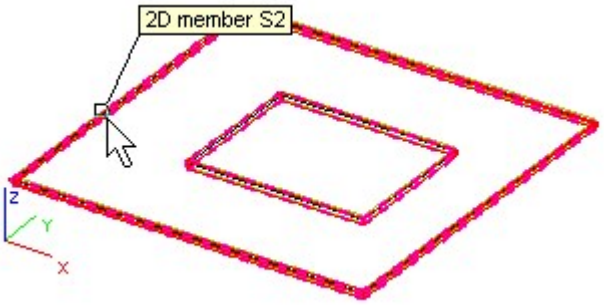
In order to select the opening, you must select the opening itself.



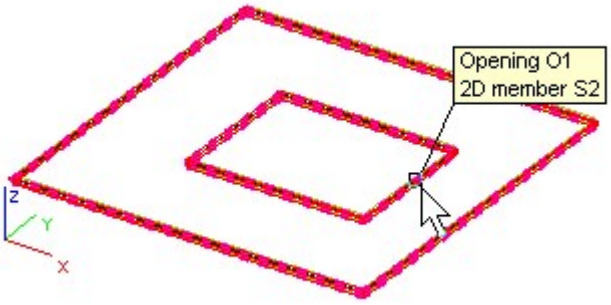
And now, let's change the view parameters and let also the surfaces of the slab displayed.



If you select the surface outline of the main slab, the main slab is highlighted and also selected. In addition, the surface of the opening is highlighted as well, BUT be careful, it is NOT selected (see the middle line of the opening – it is NOT highlighted).



In order to select the opening, you must select the opening middle line itself.



Activity

Introduction to activity

The concept of activity is based on the assumption that it is convenient to hide a part of the modelled structure and work only the remaining part. This is useful mainly for larger projects where a great number of 1D members and other entities may reduce the lucidity and comfort of performed operations.

The activity feature provides for selection of only those members that are essential for a certain manipulation or operation. The rest of the structure is temporarily hidden from the user's view.

In SCIA Engineer the activity can be realised by means of two approaches:

- Layers - see chapter Layers for more details
- Activity functions – see individual activity functions.

Activity types

There are several approaches the user may choose to determine which part of the structure should be active (i.e. visible and available for manipulations).

Layers	The activity is completely controlled by layers. See chapter Basic working tools > Layers > Displaying and hiding a layer.
Working plane	Only members located in the current working plane become active.
Selection	Only members being currently selected become active. Optionally, the selected members may become inactive and all the others remain active.
Clipping box	Only members located inside the current clipping box become active.

Switching the activity On or Off

Despite the currently selected type of activity, the user may decide whether the activity as a whole should be switched on or off. In other words, whether only the "active" part of the modelled structure should be visible or whether the whole structure should be displayed and available for manipulations.

The procedure to switch the activity (i.e. to switch it ON if the activity is OFF and vice versa)

- Either call function Tools > Activity > Activity On (or Activity Off).
- Or click button Activity On (or Activity Off) on Activity toolbar ().

Note: Both the menu item and the tooltip of the function mentioned above contain the information about the current Activity type.

Activity according to layers

When this activity type is selected, the information specified in the Layers manager controls the activity of structure members.

For more details about layers and their use see chapter Basic working tools > Layers > Displaying and hiding a layer.

The procedure to adjust the activity according to layers

- Either call function Tools > Activity > Activity by layers.
- Or click button Activity by layers on Activity toolbar ().

Activity according to current selection

The user may simply select (using standard SCIA Engineer selections) members that he/she wants to make either active or inactive. In general, there are two approaches:

- selected members are let active; all the others become inactive,
- selected members become inactive; all the others are let active.

Making the selected members active

The procedure to adjust the activity according to selection – selected members become active

- Either call function Tools > Activity > Activity by selection (Selected members On).
- Or click button Activity by selection (Selected members On) on Activity toolbar ().

Making the selected members inactive

The procedure to adjust the activity according to selection – selected members become inactive

- Either call function Tools > Activity > Activity by selection (Selected members Off).
- Or click button Activity by selection (Selected members Off) on Activity toolbar ().

Adjusting the activity for ribbed plates

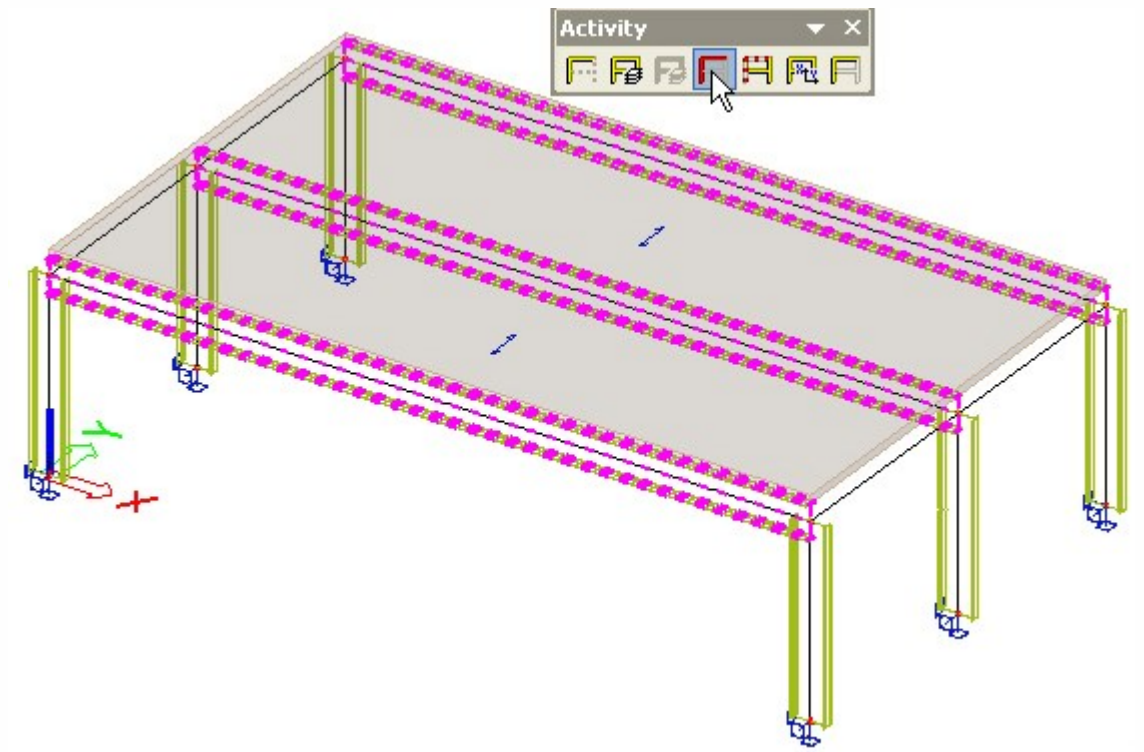
When the 'Activity according to selection' function is used for any type of ribbed plates or for plates composed of beams, it is important to know that the parent members (plates) control the activity of the ribs (beams).

The following picture with a composite slab is an example of such situation. Even thought the beams are selected, the result would be that the whole structure would be set inactive:

- the columns are not selected, so they will become inactive,

- the slab itself is not selected, so it will become inactive,

- the beams belong to the slab, so they will become inactive as well (even thought they are selected - but they are sub-ordinate to the plate).



Activity according to working plane

When this activity type is selected, the members located in the currently adjusted working plane become active. All other members become inactive.

For more details about working plane see chapter Basic working tools > working plane > Adjusting a working plane.

The procedure to adjust the activity according to working plane

- Either call function Tools > Activity > Activity by working plane.
- Or click button Activity by working plane on Activity toolbar ( ).

Activity according to clipping box

When this activity type is selected, the members located inside the currently adjusted clipping box become active. All other members become inactive.

For more details about clipping box see chapter Advanced tools > Clipping box > Introduction to clipping box.

The procedure to adjust the activity according to clipping box

1. Activate the clipping box and adjust it in required way.
2. Adjust the activity type to "by clipping box":
   1. Either call function Tools > Activity > Activity by clipping box.
   2. Or click button Activity by clipping box on Activity toolbar ( ).

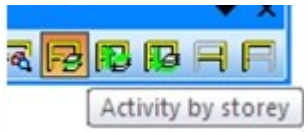
Activity by storyes

'Storey activity' is a tool that allows for visualising of the model entities based on storey allocation. Allocated members on a (or multiple) storey(s) can be displayed and edited in the Scia modelling environment while all other entities are hidden.

This activity option is located on the Activity Toolbar and it contains three buttons – 'Activity by storey,' 'Move activity by storey up,' 'Move activity by storey down.'

There are three ways to use 'Storey activity.'

The options linked to 'Activity by storey' define which type of activity is used – 'Single storey activity,' 'Unallocated members' or user-defined settings.

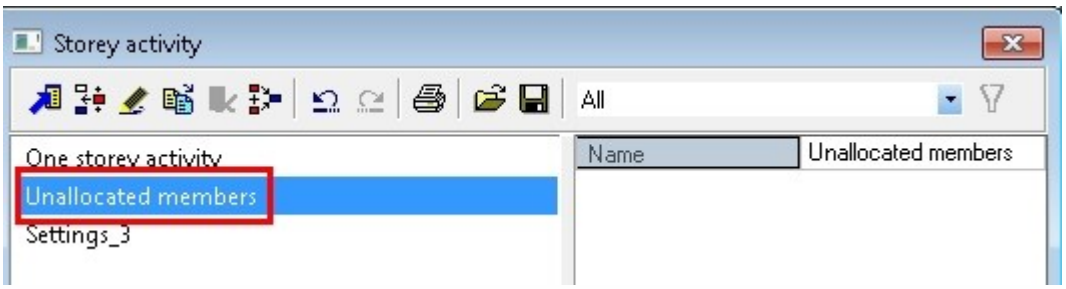


One storey activity

The user can only activate a single storey. All storey levels are available for selection via the combobox.

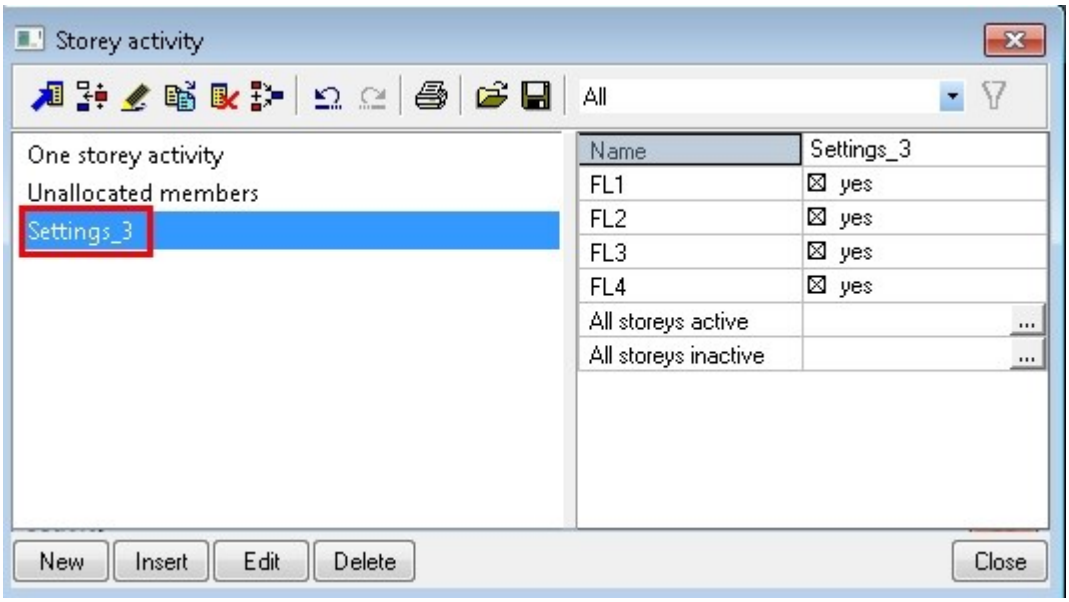
Unallocated members

A model may contain members that are not allocated to any storey. There are also object types which cannot be allocated to any level at all (linegrids, a storey itself, etc.). Those items can be displayed by the option 'Unallocated members.'



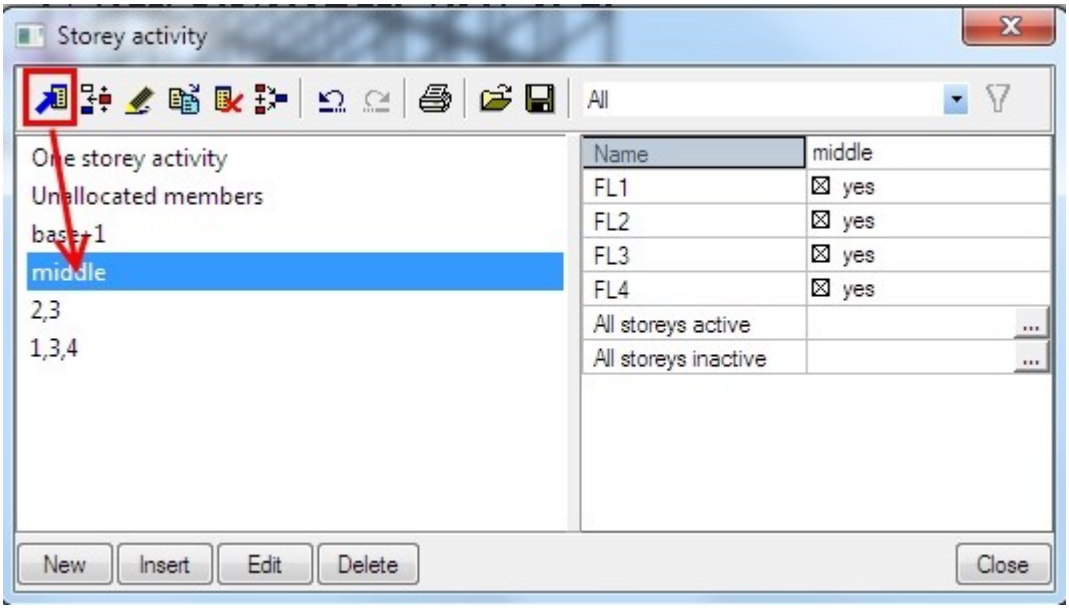
Multiple levels together – Settings

The third option displays selected levels together. This type can be created by user.



The user selects the active levels by checkboxes. All storeys can be activated and deactivated at once by the two buttons at the bottom right corner of the dialogue.

The 'Storey activity' dialogue may contain more user-defined settings to address different needs. New settings can be added by the button 'New' on the toolbar. The names of settings are user-defined.



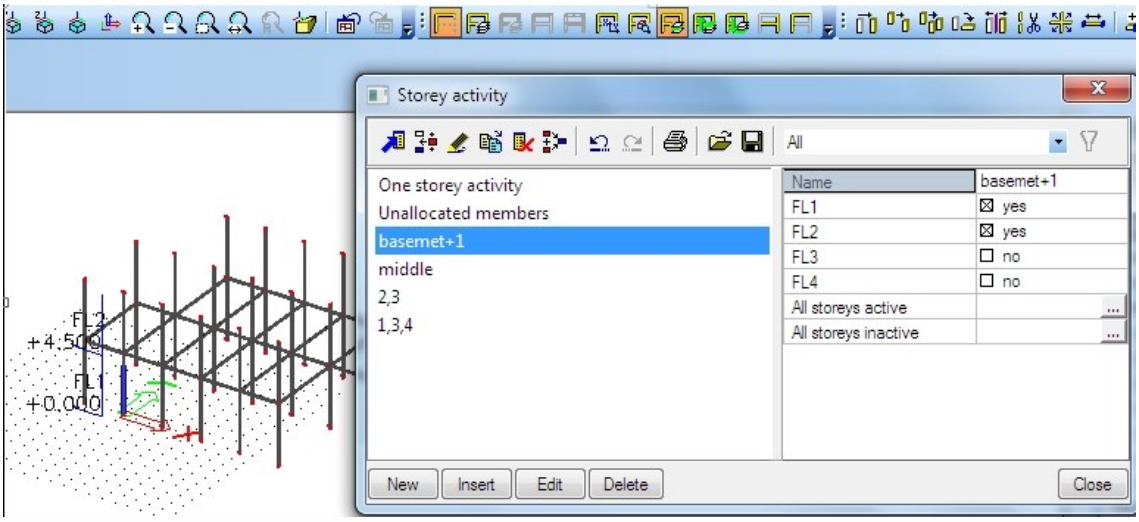
Move activity by storey up/down

The two additional buttons can be used only when the storey activity is selected.

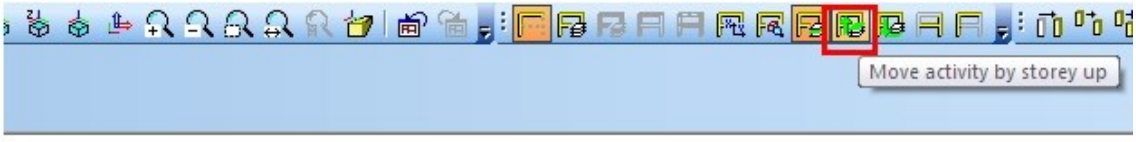
The first one moves the storey activity in the dialogue to the next one, the second to the previous.

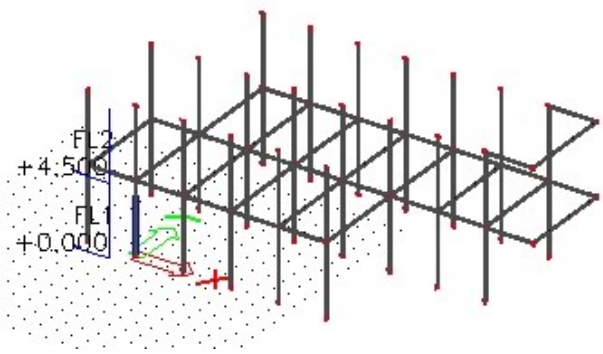
Example:

1. The initial setting – “basement+1” is active, thus, only FL1 and FL2 are visible in the 3D window.

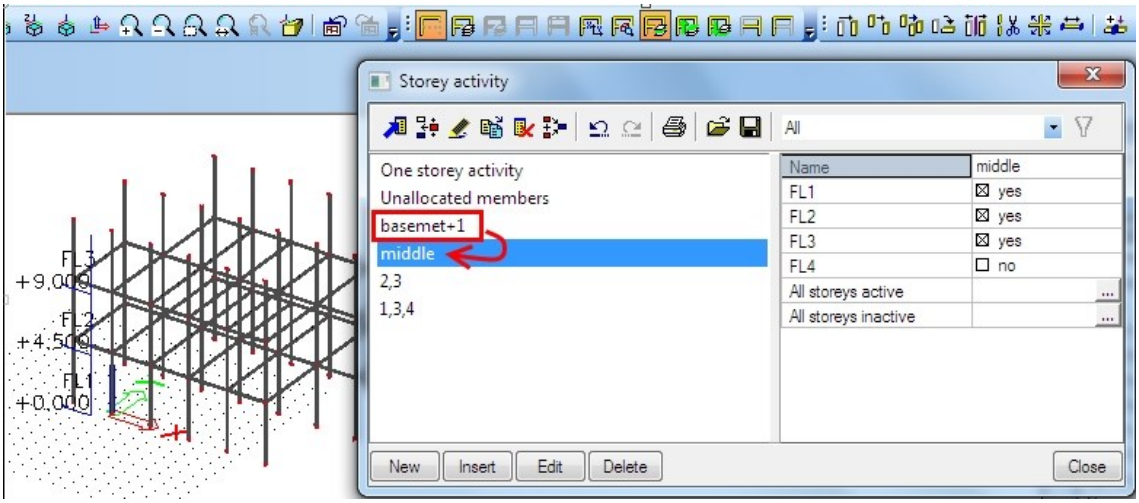


2. User presses the “Move activity by storey up”.





3. The model is changed. Three levels are now visible – FL1, FL2, and FL3. The active setting in the dialog is changed from “basement+1” to “middle.”



Inverting the activity

If required, the currently adjusted activity may be inverted so that:

- the currently active members become inactive,
- the currently inactive members become active.

The procedure to invert the activity

- Either call function Tools > Activity > Invert current activity.
- Or click button Invert current activity on Activity toolbar ( ).

Controlling the display style of inactive members

The user may decide whether the members that are currently inactive should be partly visible or completely hidden.

The procedure to display inactive members

- Either call function Tools > Activity > Draw inactive members.
- Or click button Draw inactive members on Activity toolbar ( ).

Note 1: When visible, the inactive members are drawn in a style defined for Inactive members in Colours setup (see chapter Program settings > Project settings > Display style settings > Colour setup).

Note 2: The function works like an ON / OFF switch. That means that if the inactive members ARE NOT drawn, the function makes them appear. If the inactive members ARE drawn, the function hides them.

Clipping box

Introduction to clipping box

Clipping box is a very powerful tool that facilitates manipulation mainly with excessive structures. The Clipping box defines an area (3D-area) that is visible on the screen. The rest of the structure located behind the given area is temporarily hidden from the user's view.

Defining a new clipping box

The definition of a new clipping box is similar to the adjusting of the clipping box in the setting table.

The procedure for the definition of a new clipping box

1. On toolbar View click button [Clipping box for active view] () and select function Clipping box - new.
2. Define the origin (i.e. the centre) of the clipping box.
3. The setup dialogue appears on the screen.
4. Fill in the table.
5. Confirm with button [OK].

Defining the clipping box around the working plane

Sometimes it may be very useful to define the clipping box in such a way so that only entities located in the working plane are visible.

The procedure for attaching the clipping box to the working plane

1. On toolbar View click button [Clipping box for active view] () and select function Attach to workplane.
2. The clipping box is adjusted accordingly.

Defining the clipping box around an entity

Sometimes it may be very useful to define the clipping box in such a way so that only selected entities are visible.

The procedure for adjusting the clipping box around selected entities

1. On toolbar View click button [Clipping box for active view] () and select function Around selected entity.
2. The clipping box is adjusted accordingly.

Defining the clipping box around the model

Sometimes it may be very useful to define the clipping box in such a way so that it "outscribes" the whole model.

The procedure for adjusting the clipping box around the whole model

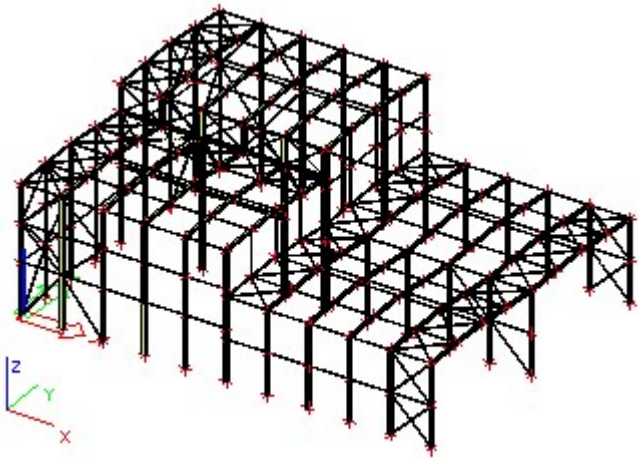
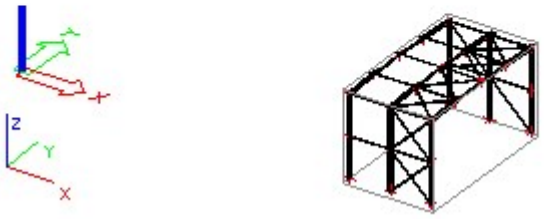
1. On toolbar View click button [Clipping box for active view] ( ) and select function Around all entities.
2. The clipping box is adjusted accordingly.

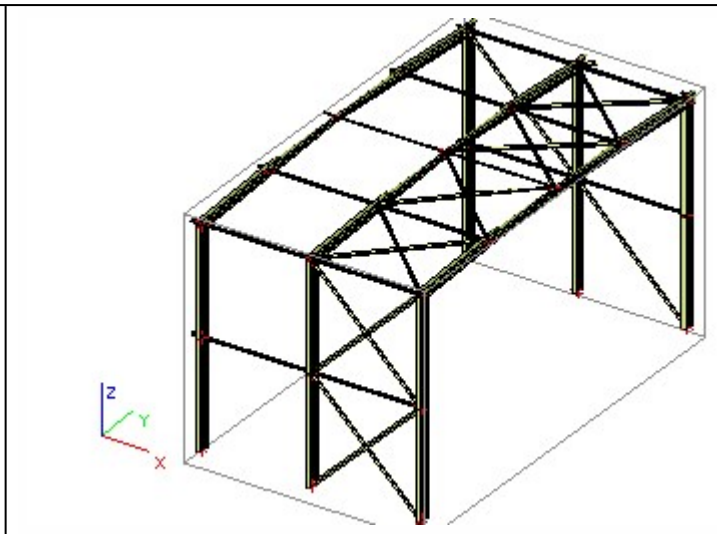
Using the clipping box

The procedure to switch the clipping box ON or OFF

1. On toolbar View click button [Clipping box for active view] ( ) and select function Clipping box On/Off.
2. The clipping box is activated or switched off accordingly.

Example of clipping box application

view without clipping box	
view with clipping box ON	

view with clipping box ON after being zoomed in	

Adjusting the clipping box in the setting table

The procedure for tabular adjustment of the clipping box

1. On toolbar View click button [Clipping box for active view] ( ) and select function Alphanumerical edit.
2. The setup dialogue appears on the screen.
3. Fill in the table.
4. Confirm with button [OK].

> Note 1: If the clipping box was not displayed before the setup dialogue was invoked, the clipping box is switched ON on confirming the settings with [OK] button.

> Note 2 : If the clipping box is ON and has been defined around the current working plane, the setting dialogue looks different and allows the user to specify the depth around the working plane.

Adjusting the clipping box using the mouse

The procedure to adjust the clipping by mouse

1. Turn the clipping box ON.
2. Position the mouse cursor over one of the clipping box borders.
3. Click the left mouse button to select the clipping box.
4. Special box-editing symbols are displayed in the centre of all clipping box surfaces. The ball symbol provides for resizing of the box, the cylinder symbol enables the user to rotate the box.
5. Select corresponding symbol for required manipulation.
6. Position the mouse cursor over the symbol.

7. Press the left mouse button and hold it down.
8. Drag the mouse to adjust the clipping box as required.
9. Release the mouse button.
10. Repeat steps 5 to 9 as many times as required to tune the adjustment of the box.
11. Press [Esc] key to close the adjustment function.

The alternative procedure for mouse controlled adjustment of the clipping box

1. On toolbar View click button [Clipping box for active view] () and select function Graphical edit.
2. The clipping box is turned ON and swapped into the editing mode.
3. Follow the procedure described above.
4. Confirm with button [OK].

> Note: If the clipping box was not displayed before the graphical dialogue was invoked, the clipping box is switched ON before enabling the adjusting.

Moving the clipping box

If required, the current clipping box can be moved to a new location. The size of the clipping box remains unchanged, only its position in space is altered.

The procedure for moving of the clipping box

1. If it is not the case, activate the clipping box (i.e. switch it on).
2. On toolbar View click button [Clipping box for active view] () and select function Move.
3. Define the new origin (i.e. the centre) of the clipping box.
4. The clipping box is moved accordingly.

Layers

Introduction to layers

One of the important entity properties that should be understood well is the layer property. Experienced users definitely use layers all the time and that is why their work is so effective. Good use of layers is one of important aspects of a good model-making-and-evaluating practice.

Basically, layers are the computer equivalent of tracing overlays on a drawing board. However, layers are much more powerful because you can have many layers in a single project and you can control the visibility and colour of layers independently. This makes working with very complicated projects much more efficient.

When you start a new project, it has only one layer. The first thing you should do, therefore, when you start a new SCIA Engineer project is to create some new layers.

Layers manager

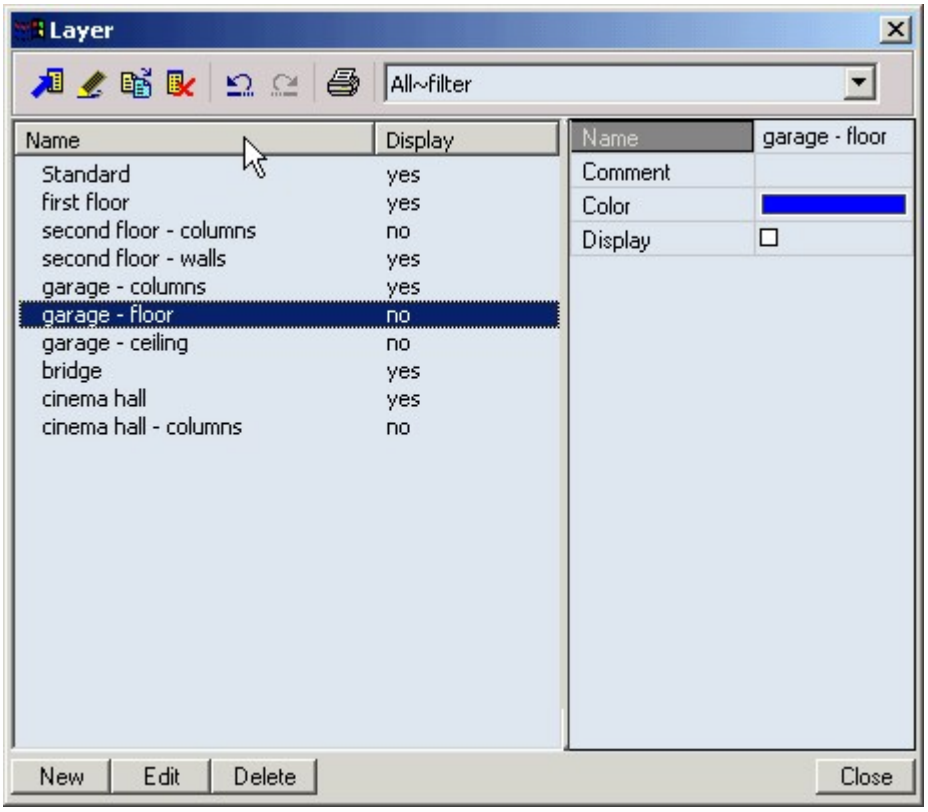
The Layers manager is a tool to control the layers defined in a project. The Layers manager provides for creating, editing and deleting of layers.

The manager itself uses the same "manager philosophy" as other SCIA Engineer managers do. It contains control buttons for standard manager operations:

New	It creates a new layer. The new layer is created with default properties that may be later edited.
Edit	It opens an editing dialogue where the layer's properties may be changed.
Copy	This function creates a copy of the selected layer.
Delete	It removes the selected layer from the project database.
Undo / Redo	It performs an Undo or Redo operation.
Text Output	It opens a small document window with a table that summarises properties of selected layers.

In order to open the Layers manager use either menu function Tools > Layers or tree menu function Tools > Layers.

The Layers manager can also be opened from various property dialogues that contain item Layer. Such an item contains a button to open the Layers manager.



Displaying and hiding a layer

The Layers manager also enables the user to specify which layers should be visible and which ones should be hidden.

Defining a new layer

A new layer can be defined in the Layers manager.

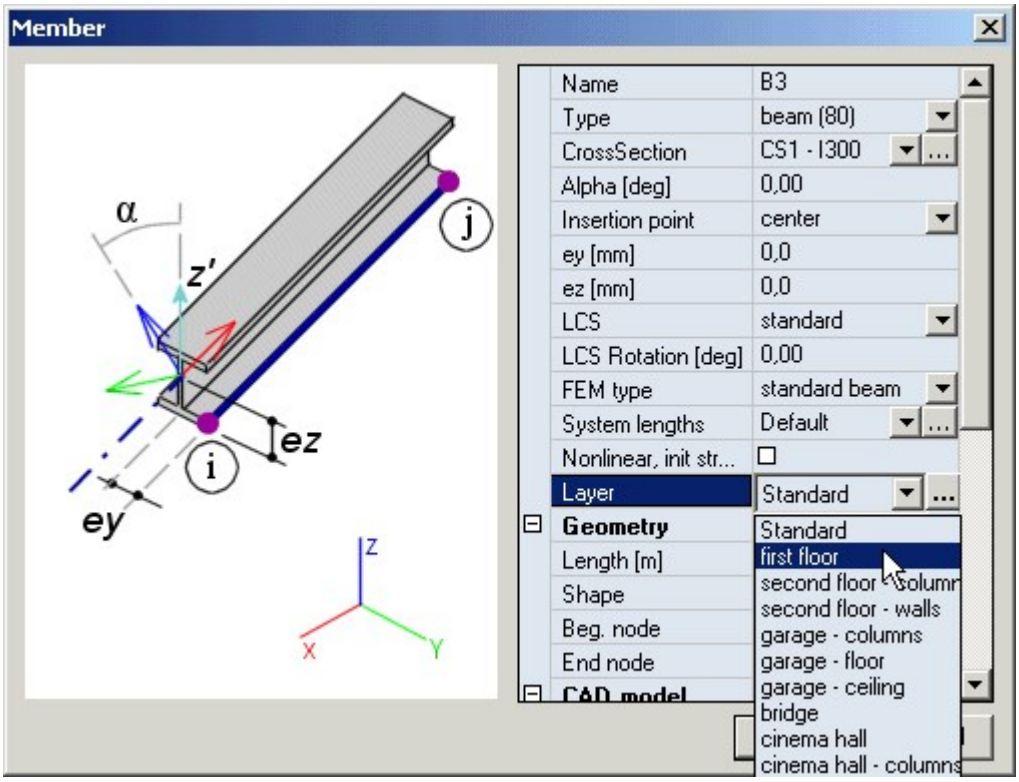
The procedure to define a new layer

1. Open the Layers manager.
2. Click button [New] to create a new layer.
3. If required, click button [Edit] to change the default layer parameters (name, colour, visibility).
4. If required, repeat steps 2 and 3 as many times as you need.
5. Close the Layers manager.

Applying defined layers

A defined layer may be applied in the property dialogue of each particular entity. One of the table items contains the layer name. This item defines the layer that the entity is put into.

Once the layer is specified in the property dialogue of an entity, the entity may be displayed or hidden according to the settings made in the Layers manager.



The picture above shows the selection of the appropriate layer for a 1D member.

Displaying and hiding a layer

One of the important features of a layer is that it can be hidden. Let's assume that the user have finished modelling of one part of the structure. Let's suppose that now s/he needs to work on another part of the same structure and that the new part is independent on the first part. The best s/he can do is hide the whole first part or at least its major part. This can be done by switching the appropriate layers off. The new part of the structure then can be modelled in new layers.

The layers can be switched OFF or ON (i.e. displayed or hidden) in the Layers manager.

The procedure for hiding (or displaying) a layer

1. Open the Layers manager.
2. In the right hand side part of the dialogue is located a layer property table containing option Display.
3. Select the layer or layers you want to display.
4. Tick the option Display.
5. Select the layer or layers you want to hide.
6. Remove the tick from the option Display.
7. If necessary, repeat the steps 3 to 6 as many times as required.
8. Close the Layers manager.

Ignoring selected layers in calculation

It may happen that a calculation model of a structure may be quite simple. Simultaneously, the structure may contain a lot of additional parts that have no load-bearing function but that are important for production of nice-looking and accurate drawings.

Such a situation calls for using of special type of layers – layers used in structural model only and ignored in the calculation. This feature may be adjusted in the Layers manager.

The procedure for extracting the layer from calculation

1. Open the Layers manager.
2. In the right hand side part of the dialogue is located a layer property table containing option structural model only.
3. Select the layer or layers you want to ignore in calculation.
4. Tick the option Structuralmodel only.
5. Select the layer or layers you want to consider in calculation.
6. Remove the tick from the option Structural model only.
7. If necessary, repeat the steps 3 to 6 as many times as required.
8. Close the Layers manager.

User co-ordinate system (UCS)

Introduction to a user co-ordinate system

The definition of points may be facilitated by the application of a user co-ordinate system. This system can be so defined (i.e. positioned and oriented) that it reflects the geometry of the model (or its part) that is being defined.

The user can define as many user co-ordinate systems as necessary. However, only one of them can be active in one graphical window at a time. Nevertheless, the user may swap between individual user co-ordinate systems at any time. What's more, even a new user co-ordinate system may be defined any time it is necessary or efficient to do so.

The active user co-ordinate system is indicated on the program status bar.

Adjusting a user co-ordinate system

UCS defined by three points

A new UCS can be defined by means of three points that do not lie on the same line. Each of the points has a precisely specified meaning:

1st point	It defines the origin of the new co-ordinate system.
2nd point	It defines the direction of X-axis of the new co-ordinate system.
3rd point	It defines the side to which the Y-axis of the new co-ordinate system will point.

Note: Please read Rules for using a UCS.

Horizontal UCS defined by one point

A new co-ordinate system is defined by a single point. The point defines the origin of the new user co-ordinate system (UCS). The axes of the user co-ordinate system are parallel with corresponding global co-ordinate axes.

That means that:

- the X-axis of the UCS is parallel with the X-axis of the global co-ordinate system (GCS),
- the Y-axis of the UCS is parallel with the Y-axis of the GCS,
- the Z-axis of the UCS is parallel with the Z-axis of the GCS.

The XY plane of this user co-ordinate system is always horizontal.

Note: Please read Rules for using a UCS.

Vertical UCS defined by two points

A new co-ordinate system is defined by two points (or one line).

The first inserted point defines the origin of the new co-ordinate system. The second point defines the direction of the X-axis of the new system. However, the X-axis is not defined precisely by the second point. The X-axis is always horizontal, and therefore, the second inserted point specifies the direction of the X-axis of the new user co-ordinate system. The Y-axis of the new user co-ordinate system is always vertical.

The XY plane of this user co-ordinate system is always vertical with the Y-axis pointing upwards.

Note 1: The two inserted points defining the new system MUST NOT lie on a vertical line.

Note 2: Please read Rules for using a UCS.

Vertical UCS perpendicular to global X-axis

A new user co-ordinate system is defined by a single point. The point defines the origin of the new user co-ordinate system.

The axes of the user co-ordinate system are oriented in such a way so that:

- the X-axis of the user-coordinate system is always horizontal,
- the Y-axis of the user-coordinate system is always vertical,
- the XY plane of the user-coordinate system is perpendicular to the global X-axis.

The XY plane of this user co-ordinate system is always vertical with the Y-axis pointing upwards.

Note: Please read Rules for using a UCS.

Vertical UCS perpendicular to global Y-axis

A new user co-ordinate system is defined by a single point. The point defines the origin of the new user co-ordinate system.

The axes of the user co-ordinate system are oriented in such a way so that:

- the X-axis of the user-coordinate system is always horizontal,
- the Y-axis of the user-coordinate system is always vertical,
- the XY plane of the user-coordinate system is perpendicular to the global Y-axis.

The XY plane of this user co-ordinate system is always vertical with the Y-axis pointing upwards.

Note: Please read Rules for using a UCS.

UCS identical with the global co-ordinate system

A new user co-ordinate system is identical with the global co-ordinate system.

Note: Please read Rules for using a UCS.

UCS perpendicular to the current UCS's X-axis

A new user co-ordinate system (UCS) is defined according the following rules:

- the X-axis of the new UCS is put into the Y-axis of the current UCS,
- the Y-axis of the new UCS is put into the Z-axis of the current UCS.

Note: Please read Rules for using a UCS.

UCS perpendicular to the current UCS's Y-axis

A new user co-ordinate system (UCS) is defined according the following rules:

- the X-axis of the new UCS remains unchanged,
- the Y-axis of the new UCS is put into the Z-axis of the current UCS.

Note: Please read Rules for using a UCS.

UCS defined according to an entity's LCS

A new user co-ordinate system is defined by means of an existing entity (e.g. beam).

The new user co-ordinate system has got its origin in the starting point of the selected entity. The axes of the user co-ordinate system are identical with the local co-ordinate axes of the selected entity.

Note: Please read Rules for using a UCS.

UCS defined from a view direction

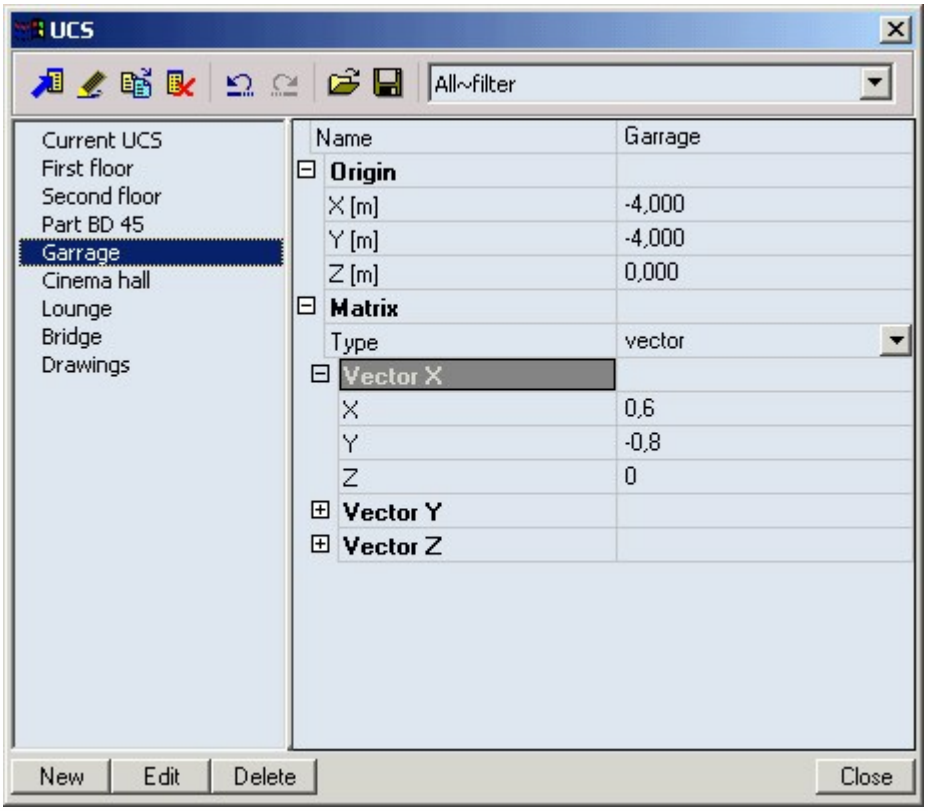
A new user co-ordinate system is calculated from the current view direction. In other words, the X-axis of the new co-ordinate system appears horizontal on the screen, the Y-axis of the new co-ordinate system appears vertical on the screen, and the Z-axis of the new co-ordinate system points towards the user's eyes.

Note: Please read Rules for using a UCS.

Editing a user co-ordinate system

UCS Manager

The UCS manager gives the user full control over the existing user co-ordinate systems. Similarly to other database managers, it provides for the definition of a new UCS, for the modification or copying of existing systems, and for removal of no-longer-used co-ordinate systems.



Association of the active graphical window with a particular UCS

The UCS manager is also used to select a particular UCS and associate it with (assign it to) the active graphical window. The UCS that is selected (highlighted) in the list of defined UCSs becomes the one associated with the graphical window.

The procedure for the selection of UCS for the active graphical window

1. Open the UCS manager:
   1. in tree menu call function Tools > UCS,
   2. on status bar click button showing the name of UCS associated with the active graphical window.
2. Select the UCS that should be assigned to the active graphical window.
3. Close the UCS manager.

Modifying an existing UCS

An existing user co-ordinate system may be edited and thus its origin or direction of axes or both may be altered. In general, there are two ways to modify an existing UCS:

- type values of UCS parameters into the editing dialogue of the UCS,
- apply one of many modifying functions collected in submenu UCS (opened either from menu Tools > UCS, or under button [Setting of UCS for active view] ( ) on toolbar View) (see chapter Adjusting a user co-ordinate system).

The procedure for direct editing of UCS parameters

1. Open the UCS manager.
2. Select the UCS you want to modify.
3. Click button [Edit] to adjust parameters of the new UCS.
4. Type in the required values for the origin of the UCS and for direction of its axes.
5. Close the editing dialogue.
6. Close the UCS manager.

The procedure for the modification of a UCS by means of UCS submenu functions

1. If it is not the case that the UCS you want to modify is the current (active) one, make it current first.
2. Open submenu UCS (either in menu Tools > UCS, or under button [Setting of UCS for active view] () on toolbar View).
3. Select the required way of modification.
4. If necessary, input required parameters (i.e. required point or points).
5. The UCS has been modified and is now kept as the current UCS.
6. Open submenu UCS (either in menu Tools > UCS, or under button [Setting of UCS for active view] () on toolbar View).
7. Select function Store the current UCS ().
8. Select the name of the UCS that has been modified and rewrite it with the new adjustment.

Defining a new UCS

UCS manager

The UCS manager can be used to define completely a new user co-ordinate system if the user knows numerically the parameters of the system. That means, if the user knows the exact global co-ordinates of the UCS's origin and the exact direction vectors of individual UCS's axes. Otherwise, the UCS manager is used to create a new UCS instance, and one of numerous UCS-modifying functions is later applied to specify the origin and orientation of the UCS exes.

The procedure for the definition of a new UCS from within the UCS manager

1. Open the UCS manager.
2. Click button [New]. This creates a copy of the current UCS.
3. Click button [Edit] to adjust parameters of the new UCS.
4. Type in the required values for the origin of the UCS and for direction of its axes.
5. Close the editing dialogue.
6. Close the UCS manager.

The procedure for the definition of a new UCS parameters using a menu/toolbar function

1. Open submenu UCS (either in menu Tools > UCS, or under button [Setting of UCS for active view] ( ) on toolbar View).
2. Select the required way of definition.
3. If necessary, input required parameters (i.e. required point or points).
4. Once more open submenu UCS (either in menu Tools > UCS, or under button [Setting of UCS for active view] ( ) on toolbar View).
5. Select function Store the current UCS ( ).
6. Type the name of the UCS and confirm with [OK].
7. That's it. A new UCS is defined and will appear in the UCS manager.

Copying an existing UCS

Any of already defined UCSs may be copied. The copy may be further modified to define a new unique user co-ordinate system.

The procedure to make a copy of an existing UCS

1. Open the UCS manager.
2. Select the UCS you want to copy.
3. Click button [Copy] to create a new UCS that is identical in its parameters with the selected one.
4. If required, click button [Edit] to adjust parameters of the new UCS and type the required values for the origin of the UCS and for direction of its axes. Then close the editing dialogue.
5. If required, repeat steps 2 to 4 as many times as necessary.
6. Close the UCS manager.

Moving an existing UCS

An existing UCS can be moved to a new origin. The orientation of the system remains unchanged, only the UCS's origin moves to a new position.

The procedure to move a UCS to a new origin

1. If it is not the case that the UCS you want to move is the active one, make it active first.
2. Call menu function Tools > UCS > Move (You may as well activate toolbar function Setting of UCS for active view > Move from toolbar View).
3. Define the new origin of the UCS.

Note: Please read Rules for using a UCS.

Rotating an existing UCS

An existing UCS can be rotated by a specified angle. The origin of the system remains unchanged, only the direction of UCS's axes changes accordingly. The rotation is performed in the adjusted working plane, i.e. the axis of rotation is normal to the current working plane.

The procedure to rotate a UCS

1. If it is not the case that the UCS you want to move is the active one, make it active first.
2. Make sure that the working plane is adjusted properly, i.e. that it is oriented in such a way that a normal to the working plane is parallel with the axis of intended rotation.
3. Call menu function Tools > UCS > Rotate (You may as well activate toolbar function Setting of UCS for active view> Rotate from toolbar View).
4. Type the angle by which the UCS should be rotated.
5. Close the dialogue.

Note: Please read Rules for using a UCS.

Deleting an existing UCS

It may happen that some of the previously user co-ordinate systems are no longer necessary, or even that some of the user co-ordinate systems have been defined by mistake. Such user co-ordinate systems may be removed from the project.

The procedure to delete an existing UCS

1. Open the UCS manager.
2. Select the UCS you want to delete.
3. Click button [Delete].
4. If required, repeat steps 2 and 3 as many times as necessary.
5. Close the UCS manager.

Storing the user co-ordinate system

Any UCS created by the user may be stored as a named UCS. The user can specify the name and once stored, the UCS is listed in the UCS manager.

The procedure to store the UCS as named UCS

1. Adjust the UCS as required.
2. Call menu function Tools > UCS > Store the current UCS (You may as well activate toolbar function Setting of UCS for active view > Store the current UCS from toolbar View).
3. Input the required name.

Using a user co-ordinate system

Rules for using a UCS

There are some rules concerning the use of user co-ordinate system that should be clearly stated here in order to prevent a possible confusion.

UCS in windows

Each graphical window can have a different UCS. The UCS can be assigned to a particular window from the UCS manager.

The procedure for association of a particular graphical window with a particular UCS

1. Select the graphical window you need to associate with the required UCS.
2. Open the UCS manager.
3. Select the required UCS.
4. Close the UCS manager.

Modification of an existing UCS in the UCS manager

If a UCS is edited in the UCS manager (i.e. edited numerically), the changes are made to the UCS that is being edited.

Modification of an existing UCS by means of modification functions

If a current UCS assigned to a particular window is edited by means of a function for modification of UCS, IT IS IMPORTANT TO KNOW that:

- Before the modification itself, the window is associated with the default (called current) UCS.
- The modification is made with the current UCS.
- The current UCS is let associated with the window.

If a named user-created UCS was associated with the window before the modification has been performed, that UCS remains unchanged.

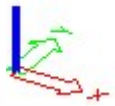
If a named user-created UCS should be modified using modification functions, the following procedure must be executed.

The procedure for modification of a named user-created UCS

1. Use modification function or functions to define the UCS as required.
2. Call function for storing of the current UCS.
3. Rewrite the original named user-created UCS with the newly defined one.

Using a UCS in the graphical window

The origin of the current user co-ordinate system is always displayed in the graphical window. Also directions of individual co-ordinate system axes are shown.



If the program is in point definition mode or point selection mode, the co-ordinates of the mouse cursor are displayed on the program status bas. The co-ordinates are given in user co-ordinate system.

Note: If required, the co-ordinates of position of the mouse cursor may also be displayed in the global co-ordinates.

Using a UCS from the command line

If co-ordinates of an inserted point are typed on the command line without any prefix, the co-ordinates are considered to be in the current UCS. For more information about the syntax of the command line see chapter Command line in book Layout and operation > User interface.

Working plane

Introduction to a working plane

A working plane is a plane in which the mouse cursor moves in the three-dimensional modelling space. The working plane can be adjusted arbitrarily to reflect the current needs of the user. The working plane is always placed into one of the basic planes of a user co-ordinate system (UCS). It means that the working plane is very closely bound to UCS.

Adjusting a working plane

A working plane can be adjusted in any direction. There is only one limitation. A working plane is always bound to the currently set user co-ordinate system. The working plane may be oriented in one of the main planes of this co-ordinate system, i.e. into XY, XZ or YZ plane.

Therefore, in order to adjust the working plane into the required direction, the user may need to adjust the user co-ordinate system first.

The procedure to adjust the working plane into the required UCS main plane

1. Verify that the current UCS is defined as required.
2. Adjust the working plane into XY or YZ or XZ plane of the UCS:
    1. Either using toolbar View and its button [Setting of UCS for active view] (),
    2. Or calling function Tools > UCS,
3. In both cases, select one of the following items: XY workplane, YZ workplane, or XZ workplane.

Cursor SNAP modes

Introduction to SNAP modes

Whenever the user needs to define a new point (e.g. an end-point of a new 1D member), it is possible to do so by typing the point co-ordinates on the command line. It is clear that this approach will not be always the most efficient one. Very often, a new point is identical with one of the already defined points (e.g. individual 1D members are connected to each other). What's more, the geometry of the structure is usually regular in some way, and therefore, end-points of individual entities fit into a regular scheme. Both of these facts have been taken into account during the design of SCIA Engineer's SNAP modes.

A SNAP mode is a mode for locking a mouse cursor into alignment with an invisible rectangular grid or with characteristic points of already defined entities (such as their end-points, middle points, centres of circles, etc.).

When the SNAP mode is on, the screen crosshairs and all input coordinates are snapped to the nearest point on the grid or to the nearest characteristic point.

Grid SNAP modes

The grid SNAP mode is a SNAP mode where the mouse cursor is locked into alignment with a grid. SCIA Engineer offers two types of grid:

- a dot grid (that may be either orthogonal or radial),
- a line grid (that may be both two- and three-dimensional).

When this SNAP mode is on, the screen crosshairs and all input coordinates are snapped to the nearest point of the grid.

The grid SNAP mode can be combined with the object SNAP mode if required.

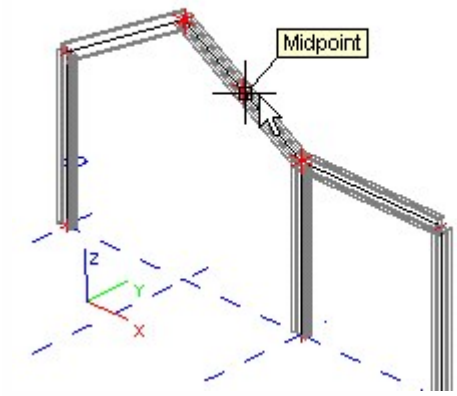
The activation of the grid SNAP mode can be done in the Cursor snap setting dialogue.

Object SNAP modes

The object SNAP mode is a SNAP mode where the mouse cursor is locked to commonly needed points, or we can say characteristic points, on entities (such as their end-points, middle points, centres of circles, etc.).

If required, the object SNAP mode can be combined with the grid SNAP mode.

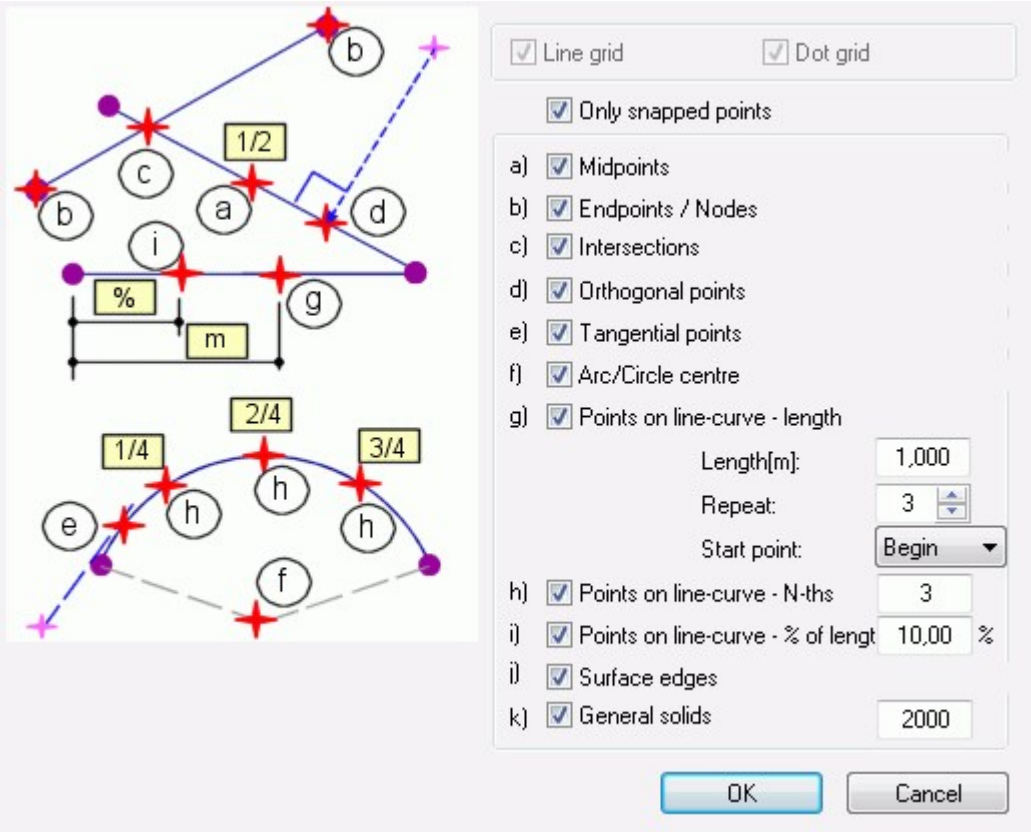
A required kind of the object SNAP mode can be selected (activated) in the Cursor snap setting dialogue.



The picture above shows "in action" the SNAP mode set to Midpoints.

Adjusting a SNAP mode

Adjustment of the required SNAP mode or modes can be done in the Cursor snap setting dialogue Default keyboard shortcut is set to F6.



The dialogue offers a vide range of SNAP variants:

Line grid	The cursor is locked to the vertices of a defined line grid.
Dot grid	The cursor is locked to the points of a defined dot grid.
Only snapped points	If this option is ON, the first two variants are automatically turned OFF and only characteristic points of already defined entities may be used to snap to. In other words, only the object SNAP mode is enabled.
Midpoints	Middle points of entities are used as snap points.
Endpoints / Nodes	End points of entities are used as snap points.
Intersections	Intersections of entities are used as snap points.
Orthogonal points	This option snaps to a point which forms a perpendicular with the selected object.
Tangential points	The Tangential point SNAP mode snaps to a tangent point on a circle.
Arc / circle centre	This option snaps to the centre of a circle, arc or polyline arc segment. The cursor must pass over the cir-cumference of the circle or the arc so that the centre can be found.
Points on line / curve - length	The program automatically "creates" SNAP points in the defined distance from the beginning and/or end of the line/curve.
Points on line / curve N-th	The program automatically divides a selected entity into N segments and thus generates (N+1) points on an entity under cursor. The points may be used to snap to.
Points in line / curve	This option is similar to the one above. But the division of a 1D member is defined by percents and not by the number of segments.

% of length	
Surface edges	This option is available only if at least one of the above listed object SNAP modes is ON. If this option is ON, the mouse cursor snaps also to the surface lines of entities.
General solids	This option snaps to characteristics points of general solids. The number defines the number of characteristic points that are generated for the SNAP purposes. The number-parameter is useful especially if there are complex solids in the model. Such solids can have a huge number of characteristic points. If all of them were offered for snapping, the scene in the graphical window would be unclear. Therefore, the parameter can be used to limit the number of offered snapping points.

The procedure for the adjustment of the required SNAP mode:

1. Open the Dot grid setting dialogue. The dialogue can be opened in two ways:
   1. via [Snap mode] button on the Status bar,
   2. via [Cursor snap setting] button () on the toolbar at the command line.
   3. using menu function Tools > Cursor snap setting.
2. Select the required SNAP option or options.
3. Press button [OK] to close the dialogue.

Adjusting the temporary one-step SNAP mode

Sometimes it may be useful to let the current SNAP mode AS IS, and change the SNAP mode just and only for a single step (single action). For example, all new end-points of a set of 1D members are defined as end-points of existing entities, but suddenly it may happen that one particular point would be easily defined as a midpoint.

In SCIA Engineer the user may change the SNAP mode temporarily for a single step only.

The procedure for the adjustment of a temporary SNAP mode

1. Once a function requiring the definition of points is started a toolbar is displayed at the top of the command line.

2. Proceed with the opened function up to the moment you need to change temporarily the SNAP mode.
3. Click the required icon on the mentioned toolbar.
4. The SNAP mode is temporarily re-adjusted for the following single step.
5. Once you define the point, the SNAP modes returns to the original setting.

Tracking

Introduction to tracking

The tracking is a tool for easier modeling in the 3D window. It helps user to define directions and distances by tracking rays and tracking points.

Tracking uses already defined nodes and its coordinates in the project. The new node overtakes coordinate from the original one. It is possible to overtake more coordinates, e.g. X coordinate from node A and Y coordinate from node B.

The distance from one point to another can be defined by value in the command line.

Tracking options allows adding user defined angles for tracking rays. This can be convenient for structures which are not perpendicular.

The organizing of the project is much more clearer, easier and proper.

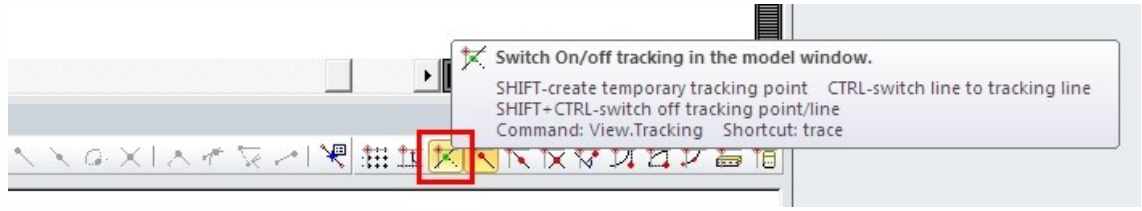
See the videotutorials on web:

1) Tracking ray: http://campaigns.nemetschek-scia.com/scia-movies/movies/model/tracking_insert.htm

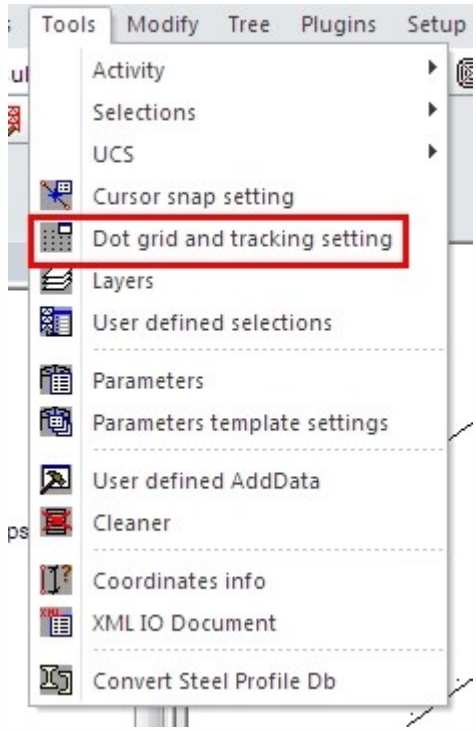
2) Tracking line: http://campaigns.nemetschek-scia.com/scia-movies/movies/model/tracking_point+line.htm

Tracking options

The tracking is switched ON and OFF by a button on the bottom window toolbar.

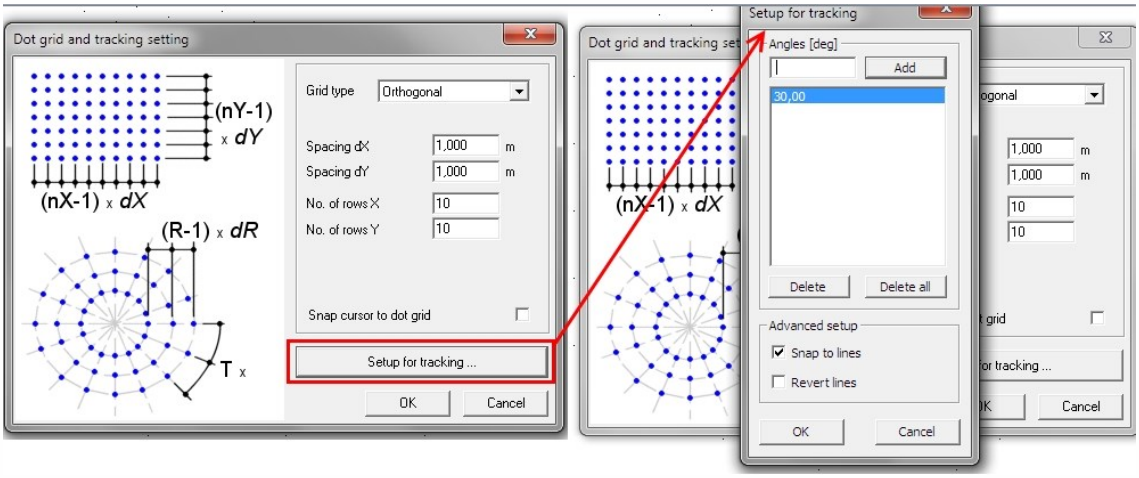


The settings are in the dialogue Dot grid and tracking settings.



The dialogue consist from two parts.

1. The list of user defined angels for tracking rays
2. Settings for tracking lines



User defined angles

The default angles are according to the global coordinate system. When user needs more, the additional angles can be inputted to the tracking settings.

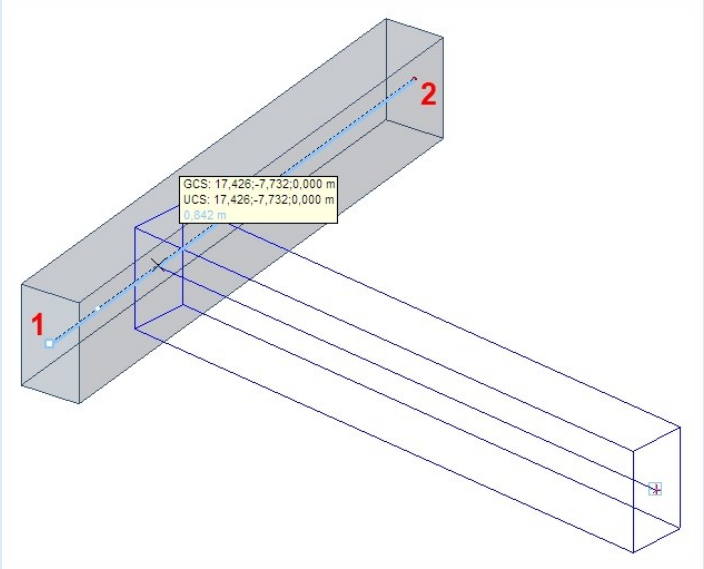
Settings for tracking lines

The second part of the dialogue contains settings for the tracking lines. The first checkbox defines if the tracking lines are used - Snap to lines. The second checkbox defines the position of the start point for tracking on the line.

See the separate chapter to learn how to switch the common geometry to the tracking line.

Unchecked Revert lines

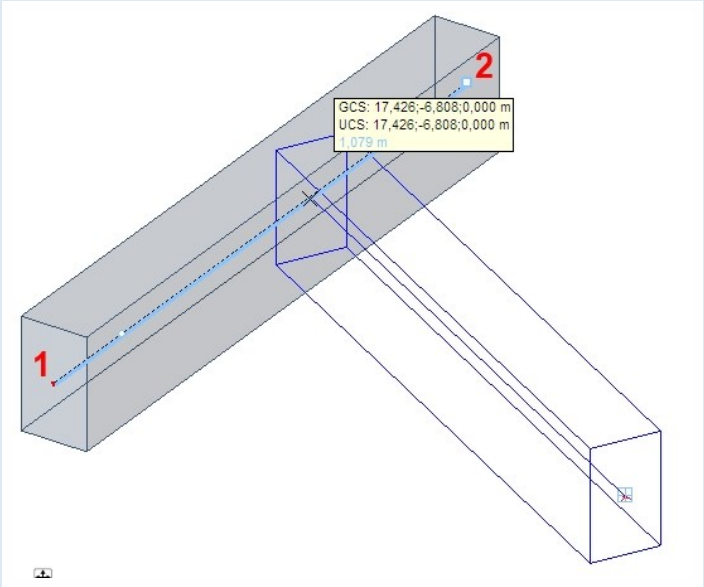
The start point of the tracking line is defined during the switching the tracking line on. The node closer to the cursor movement is used as a start point. When the second checkbox in the dialogue is unchecked, it works this way .



1 - begin node of the line
2 - end node of the line
The begin node is set as a start point and distances are calculated from it.

Checked Revert lines

The start point is defined during the switching again, but this time the farther node is set as the start point.



1 - begin node of the line
2 - end node of the line
The end node is set as a start point and distances are calculated from it.

Tracking tools

Tracking point

The point is displayed by a light blue rectangle. The last inserted point is automatically switched to the tracking point. Other tracking points are user defined. It can be switched snapping point or tracking point placed anywhere by user.

There is no geometry (or add data) created when user creates the tracking point, it is only an assistant point. It helps to track another position.

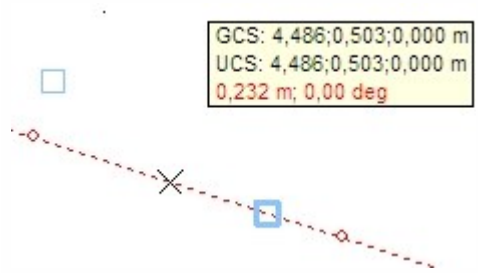
There are two types of tracking points:

1. The first with the crossed blue rectangle is the last inserted point in the project and it cannot be disabled. It is the default point for creating the next member.

2. The second point is user defined. It can be disabled. User creates it from the snapping point or it is inserted as user defined tracking point anywhere in the project.

The tracking points are automatically deactivated when the actual command is finished or canceled.

The active tracking point is displayed by thicker rectangle. It is the tracking point from which the tracking ray is displayed.



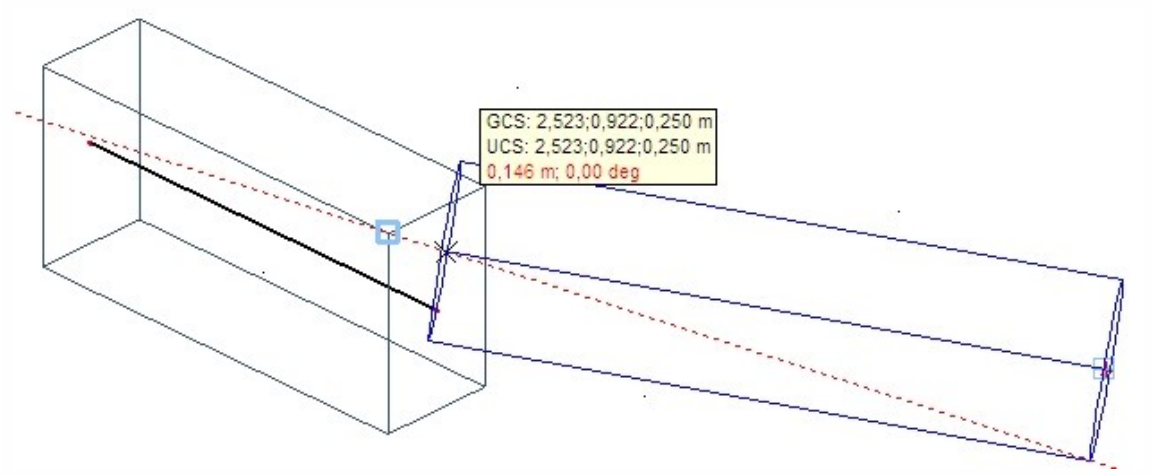
The tracking points can be inputted more times to the project.

How to disable the tracking point - see the separate chapter.

Switched tracking point

The snapping point is displayed by moving the cursor over the point, it is displayed by the red rectangle. This point is switched to the tracking point by moving the cursor over the snapping point and holding the SHIFT key.

No clicking is needed.

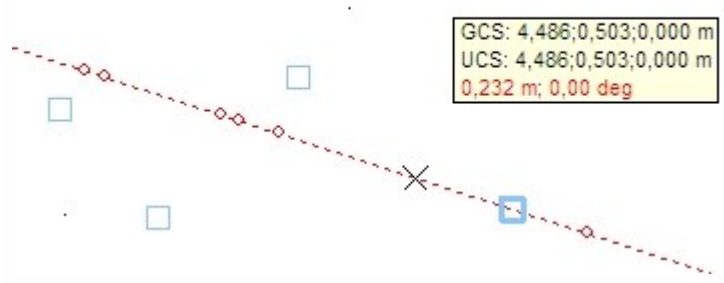


The tracking points can be created on all types of snapping points.

The displaying of the snapping points depends on its settings.

User defined tracking point

The user defined tracking point can be inserted anywhere in the project . This point is created by holding the SHIFT key and clicking to its position.

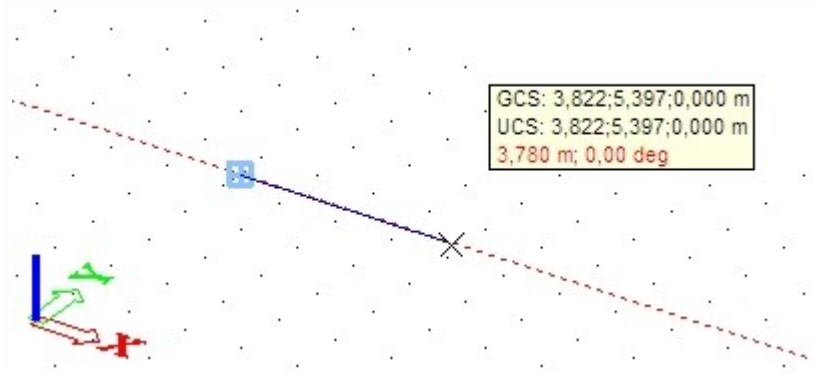


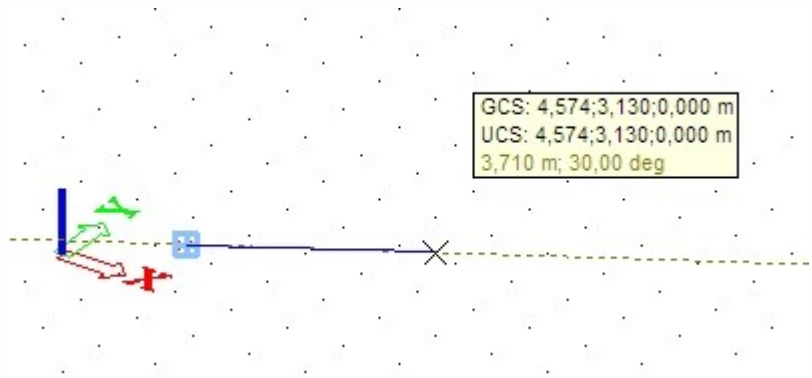
Tracking ray

This is the dot line which is displayed from the active tracking point.

The directions are according to the global coordinate and according to the user defined angles. When the tracking line corresponds to the coordinate direction, it has the same colour. If user defined angle is used, the tracking ray is displayed by the yellow colour.

The cursor is automatically snapped to the ray so it is easy to move on it.

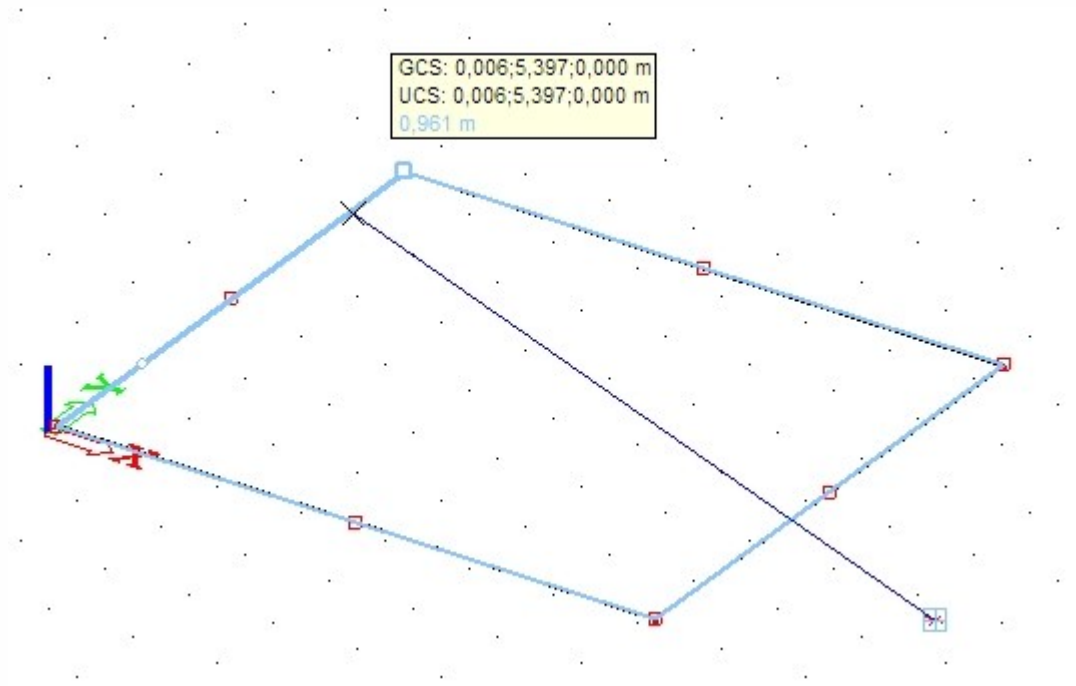




Tracking line

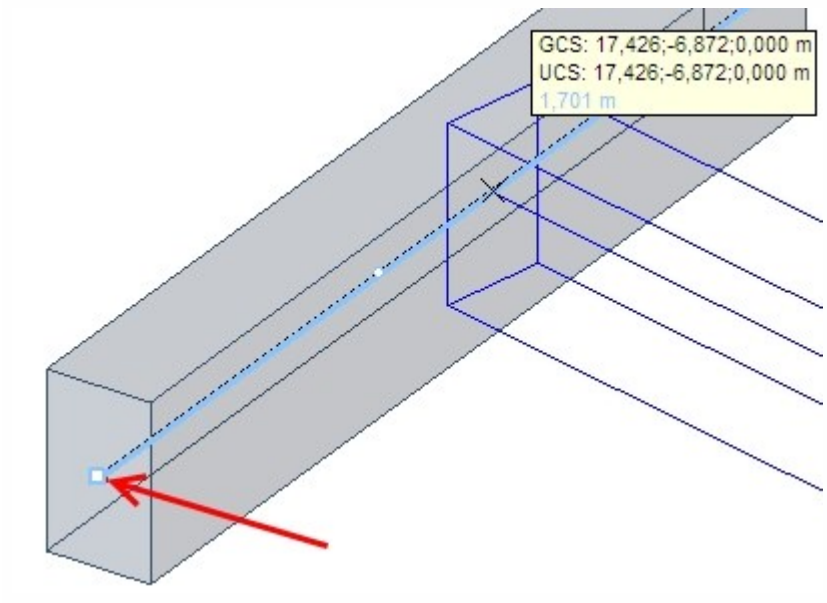
This tool works with already inserted lines and polygons in the project (center line, edge, 3D line).

It is similar to the tracking ray. It doesn't display the dot line, but it marks geometry as a tracking line. The colour of the tracking line is light blue (the same colour as we have for the tracking point). The major difference is, that tracking ray doesn't have the begin and end node but the tracking line has it because it is limited by the used geometry.



Start point on tracking line

The start point on the tracking line is displayed as a blue circle. It is begin or end node of the line - it is controlled by the tracking settings.



Disable switched tracking point or tracking line

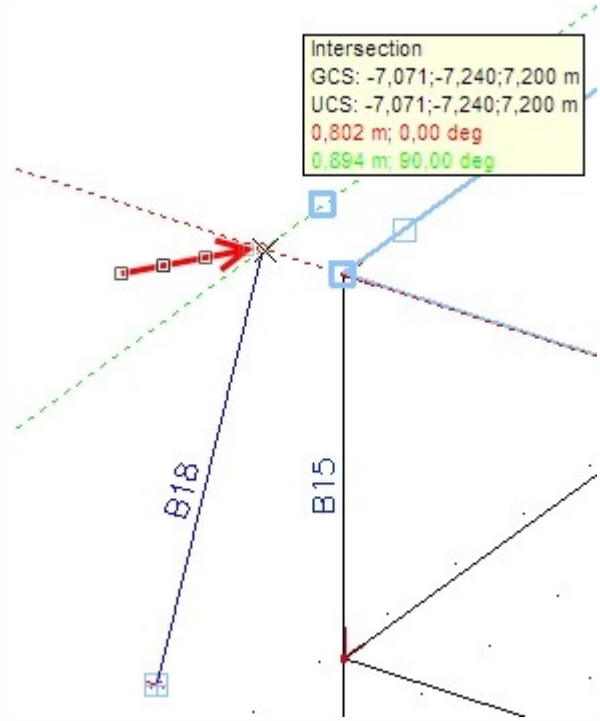
The tracking point (line) can be disabled by moving the cursor over the tracking point with holding SHIFT + CTRL key.

The original snapping point or line is displayed again.

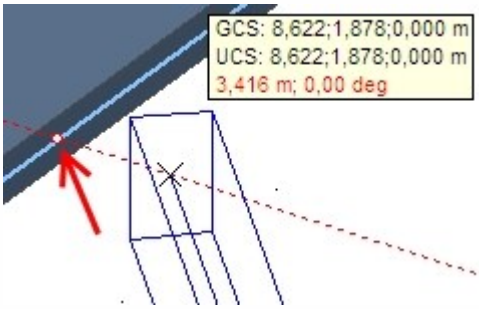
Intersection point

The intersection of two tracking rays (or the tracking line and ray) is displayed by a small circle. This point is exact intersection and user can snap to it.

Between two tracking rays:



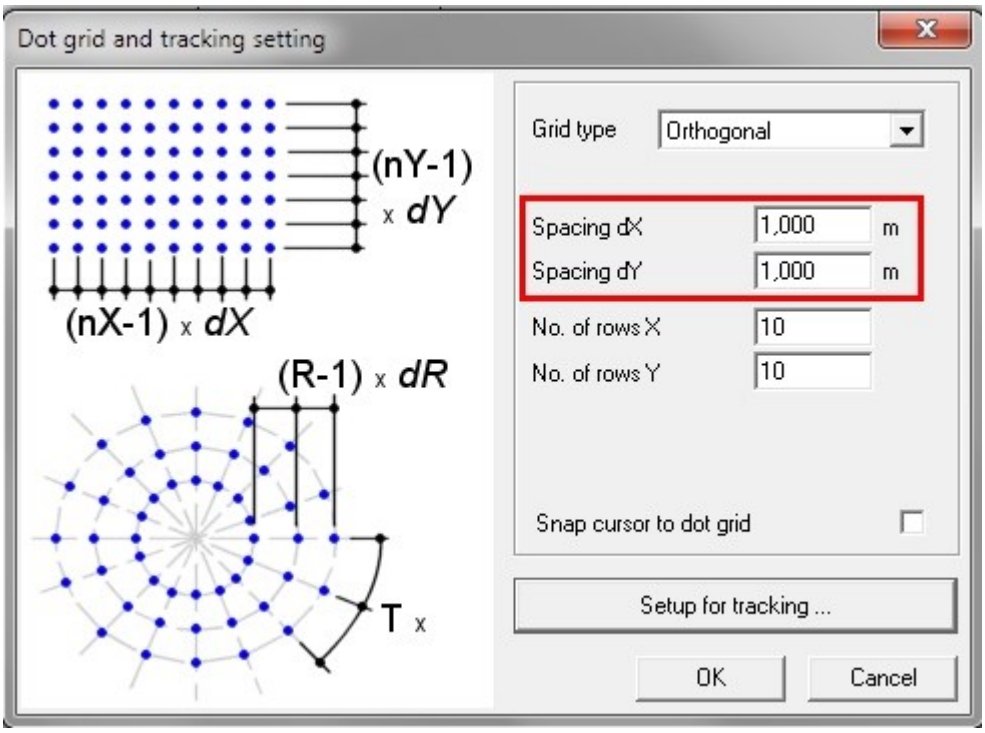
Between the tracking ray and tracking line:



Using grid step on tracking ray and line

The settings for dot grid can be used on tracking rays and lines.

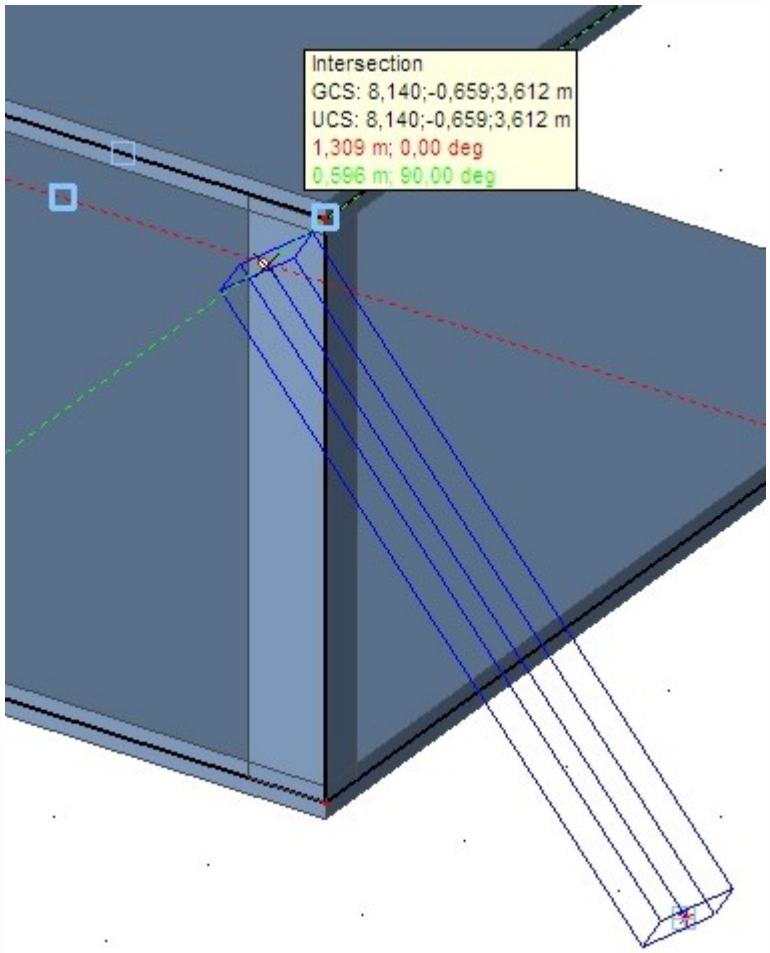
The snap to the dot grid must be switched ON.



The step is taken from the Dot grid and tracking setting. The cursor is snapped to steps on the tracking ray and tracking line. The start tracking point is always taken as zero and its position has no influence on the steps.

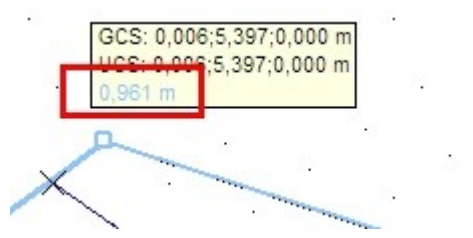
Tooltip

The tooltip contains all information about the cursor position and distance from the start point. It also displays information about intersection point. Colours correspond to the global axis directions.

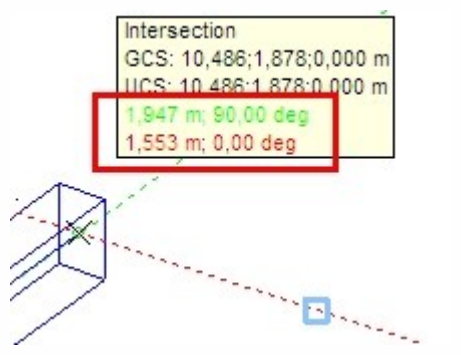

Intersection
GCS: 8,140;-0,659;3,612 m
UCS: 8,140;-0,659;3,612 m
1,309 m; 0,00 deg
0,596 m; 90,00 deg

Tooltip on the tracking ray - there is a green part which displays the distance from the start point in the Y direction. The start point is the first structural node in this case.

Tooltip on the tracking line - there is a light blue part which displays the distance from the start point. The start point is a begin node on the line in this case.



Tooltip for the tracking intersection point - there are two values, one green and one red. It displays distances of the cursor from the start points on both rays.



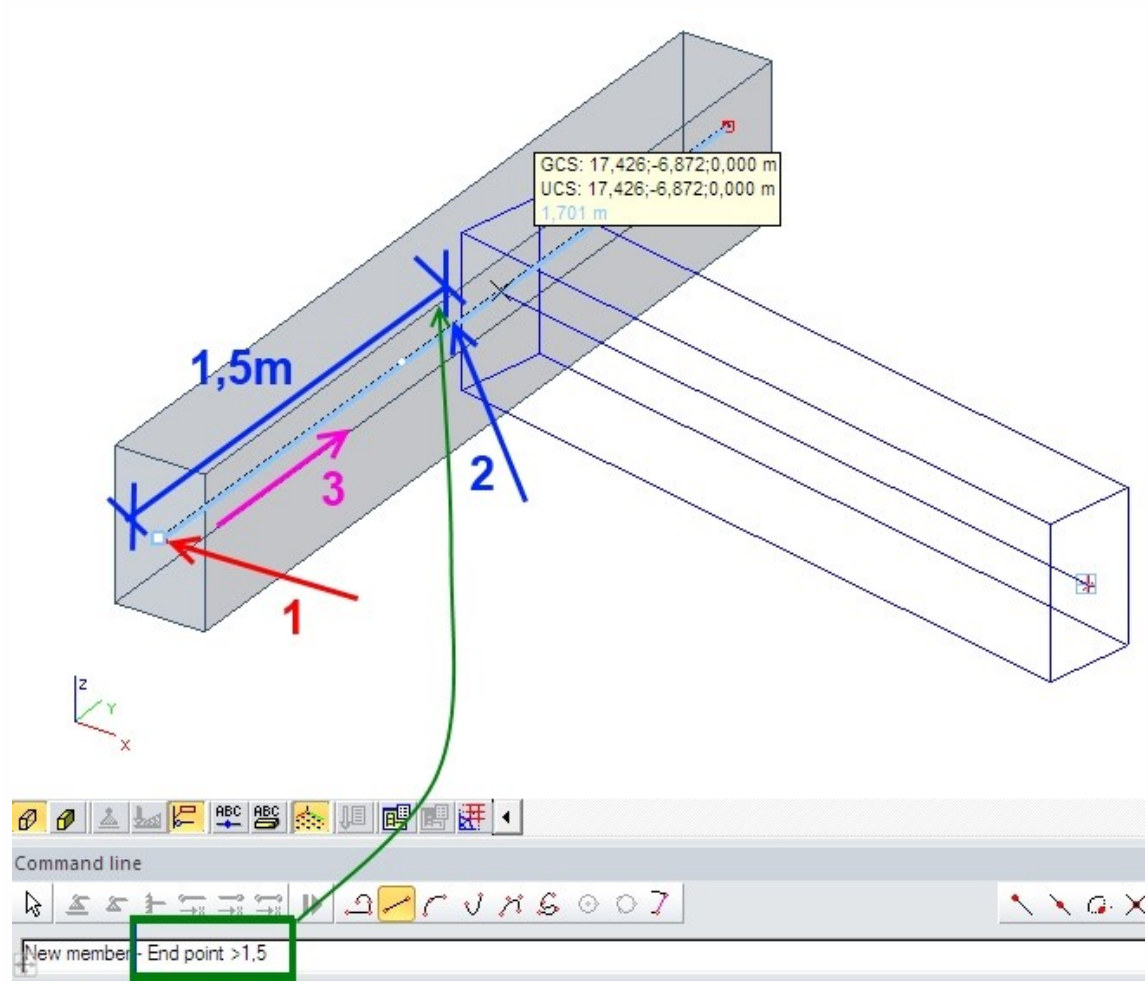
User defined distance

The value typed to the command line is taken as a distance from the start point. This option can be used on tracking rays and tracking lines.The structural node is created after the confirmation the value by ENTER key.

There are two condition which must be fulfilled:

1. The tracking ray or tracking line must be displayed
2. The cursor shows the correct direction for the typed distance

The picture shows the typical situation:



1 - the starting point for the typed distance
2 - the position of a new structure node (after the confirmation of the distance)
3 - the user defined direction

Process:

a) The tracking line is displayed and the start point is set to the begin node of the member.

b) User wants to place a new structural node 1,5 m from the start point.

c) The cursor is placed on the line to show the direction and user types 1,5 to the command line. The value is confirmed by ENTER .

d) The new node is inserted 1,5 m from the begin node.

Tracking shortcuts summary

SHIFT - inserting the user defined tracking point

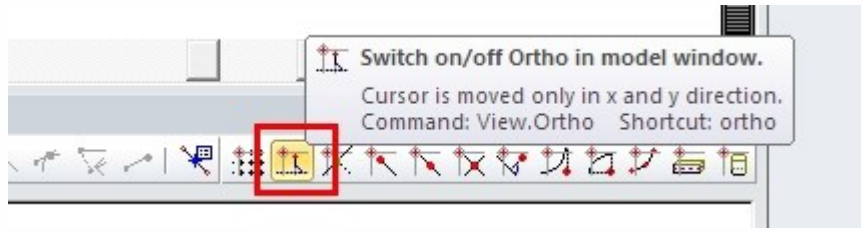
CTRL - switch the geometrical line to the tracking line

SHIFT+CTRL - disable tracking point or line

Ortho functionality

The ortho is a simple functionality which allows user to mover the cursor only in an orthogonal system.

The ortho is switched ON and OFF by the button on the bottom window toolbar.

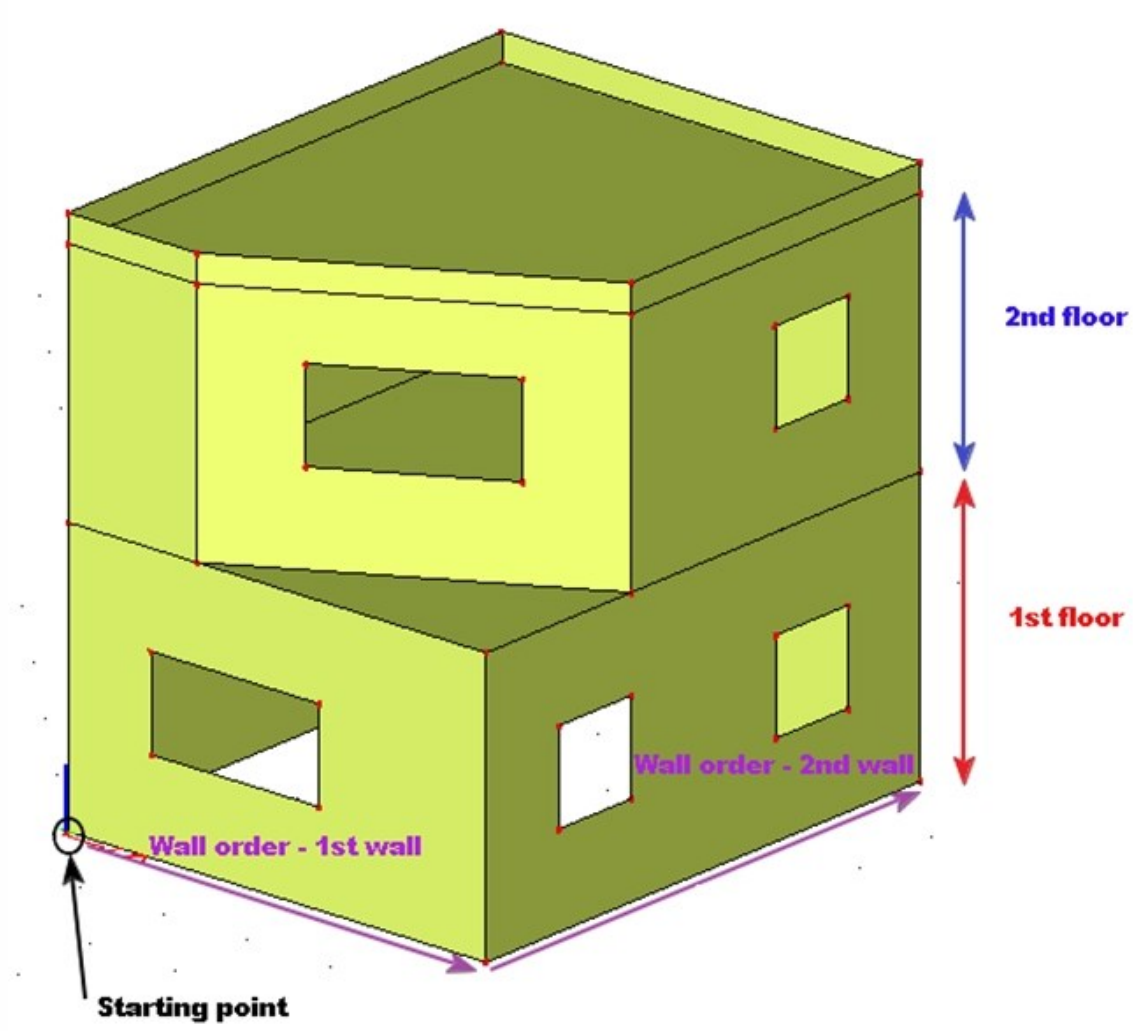


This functionality works only in the working plane and directions are always according to the global coordinate system. User can't use points, rays and lines as in the tracking. User defined distances aren't supported.

How to use tracking mode to insert a new structure

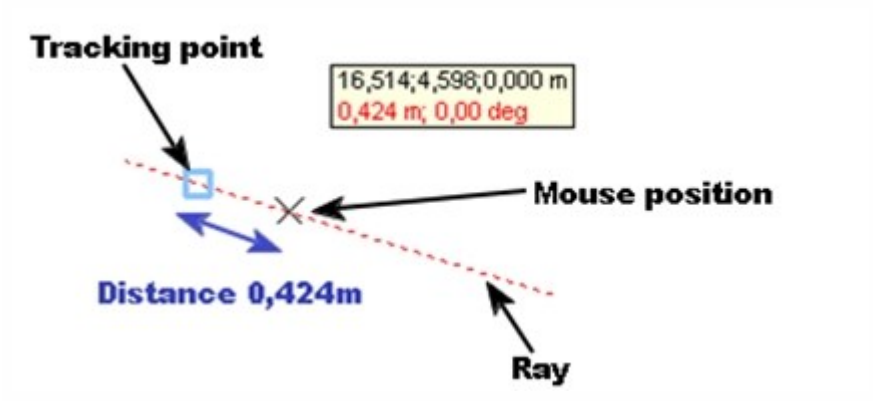
Create a simple structure using tracking mode:

Building with 2 floors and one inclined wall on the second floor.

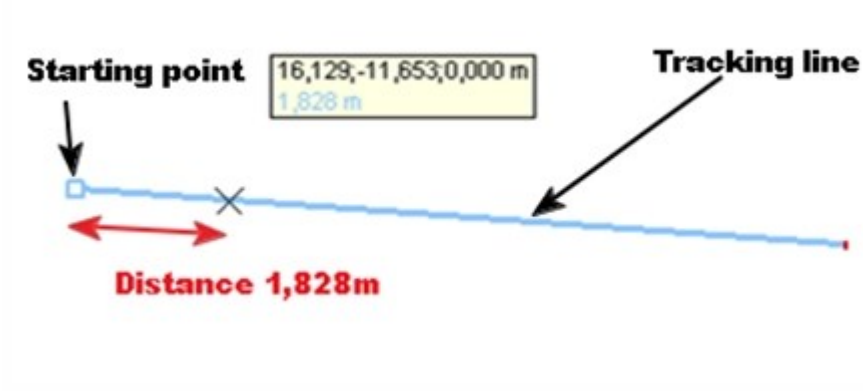


Introduction to basic tracking tools - point, line, intersection:

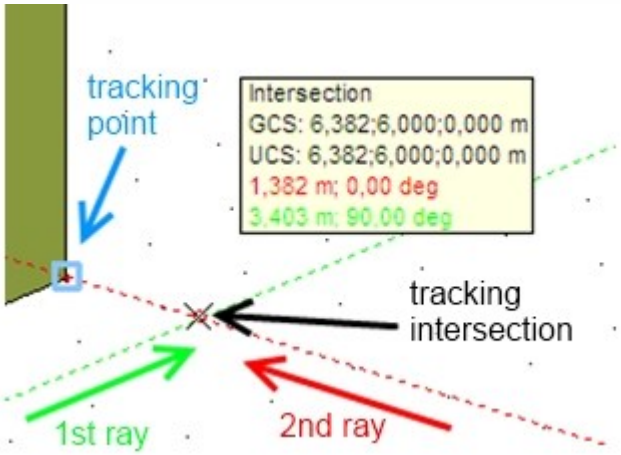
Tracking point - point which is a starting point for a tracking rays



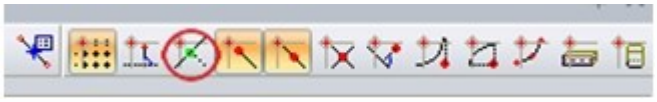
Tracking line - geometry line or polyline which can be used in the same way as tracking ray



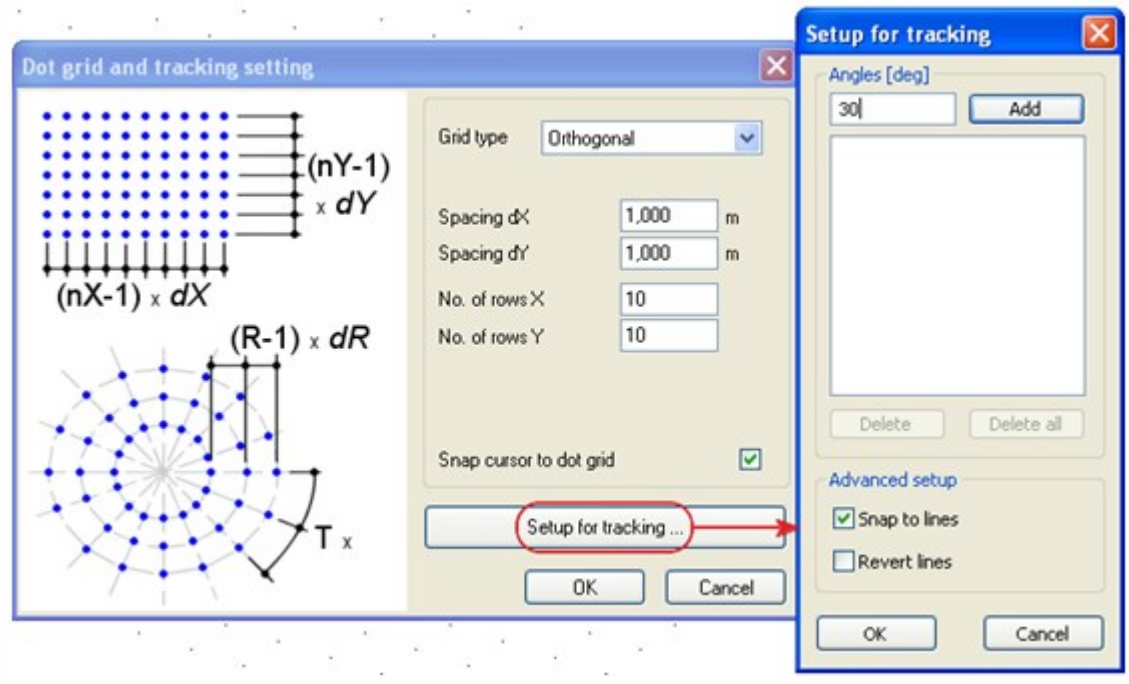
Tracking intersection - is a point where one ray cross the other ray



Start with a new empty project:

1. Open a new empty project with concrete. Tracking is controlled by the button on the bottom of the 3D window. Switch it on. 
2. The tracking settings for is under “Dot grid and tracking settings".

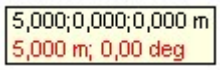
3. We need to set one angle of rays. Open dialogue “Dot grid and tracking settings” and use button – “Setup for tracking …”. We use the tracking ray with this angle for inserting inclined wall on the second floor.



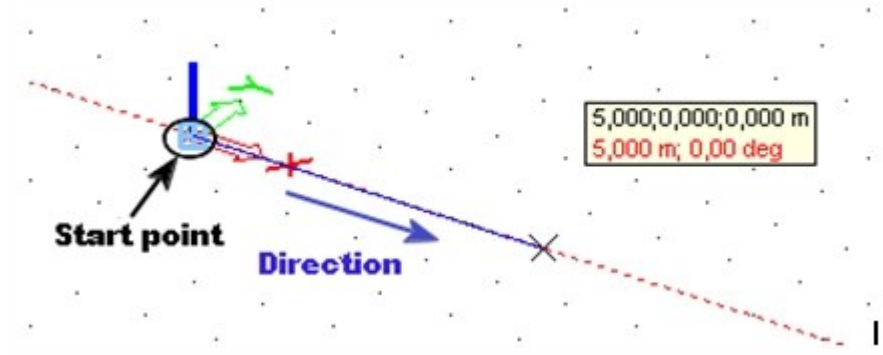
4. Add a new angle 30°. Write the value to the row and use the button "Add". Check if the checkbox for “Snap to lines” is active.

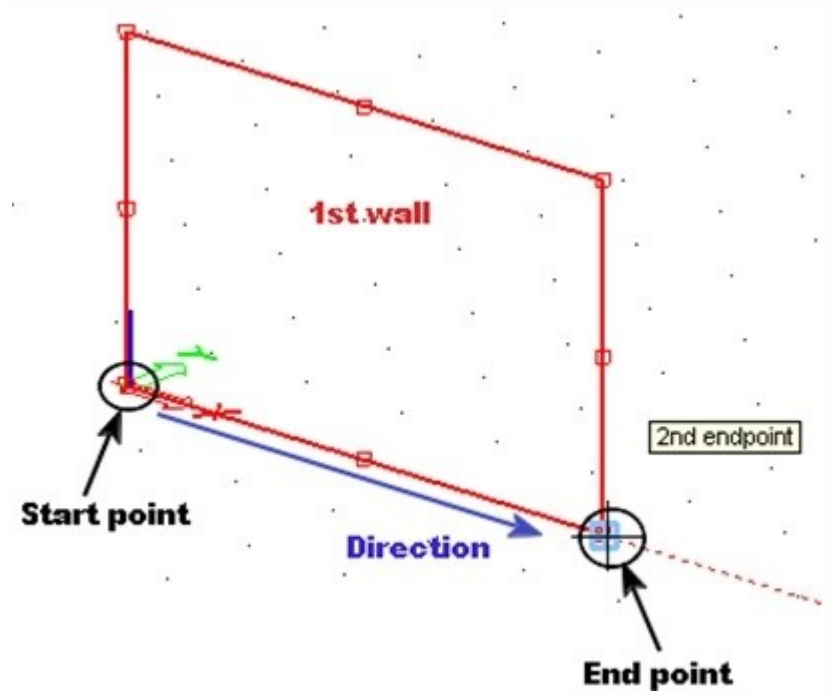
Modeling

1. Create the first level of our building on working plane. We will create 4 walls, 1 plate and some openings. Activate the dot-grid snapping to define the size by the simplest way.
2. It is possible to snap to the dotgrid and also see the distance in the tooltip.

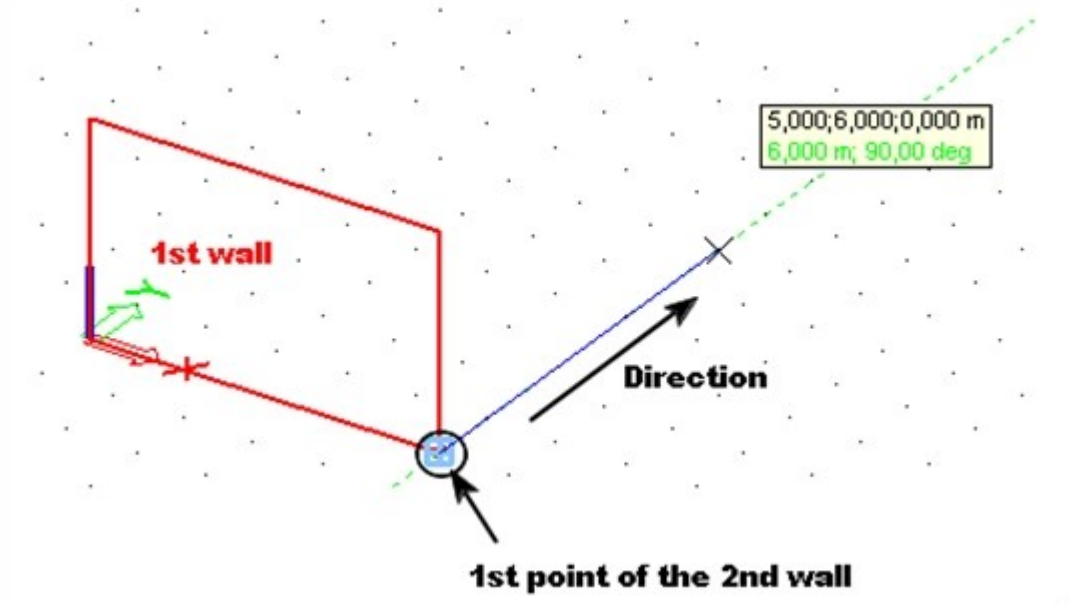


3. The 1st wall starts in point 0,0,0. The length is 5m. The tracking tooltip shows the distance





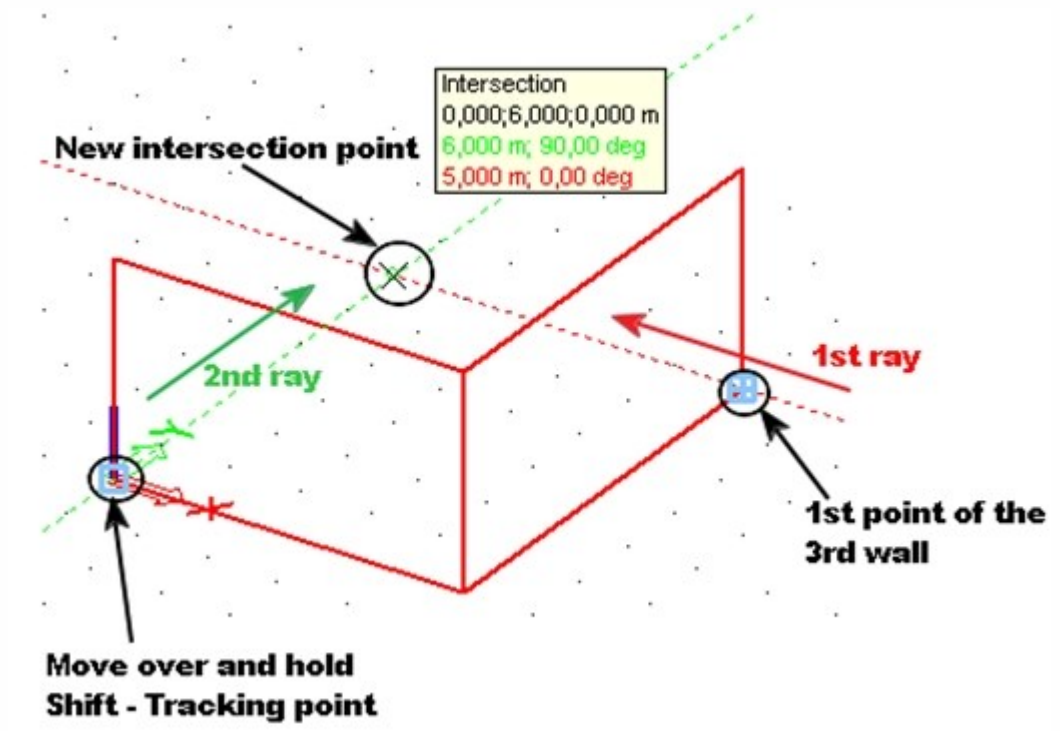
4. Create the 2nd wall from end point of the 1st wall. Length of the 2nd wall is 6m. See the next picture.



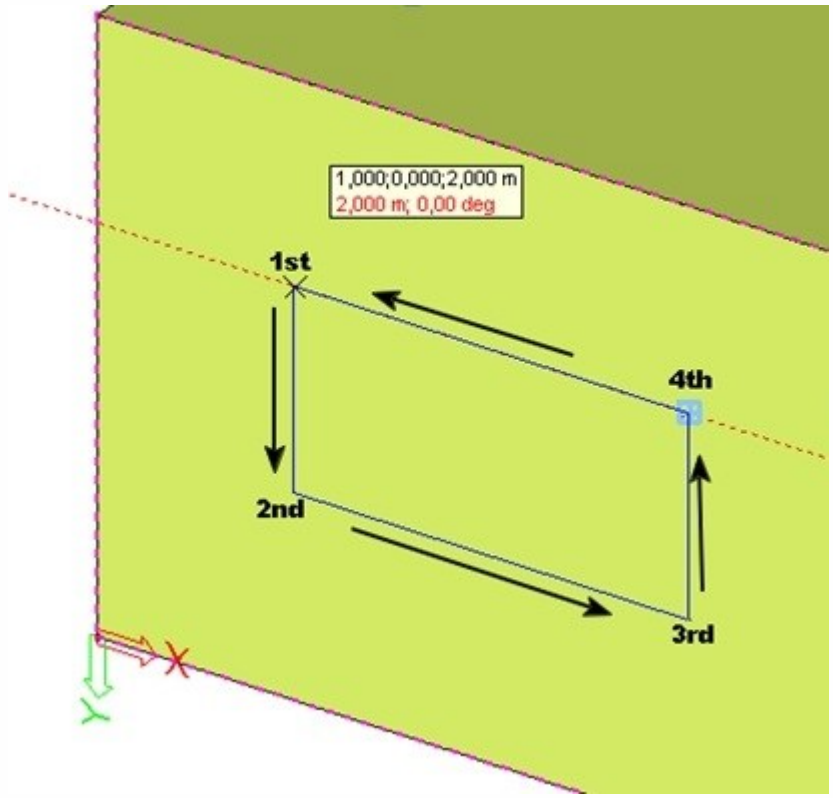
5. We use a new type of intersection tracking point to create the 3rd wall. The length of the 3rd wall is 5m.

How to create the tracking intersection point:

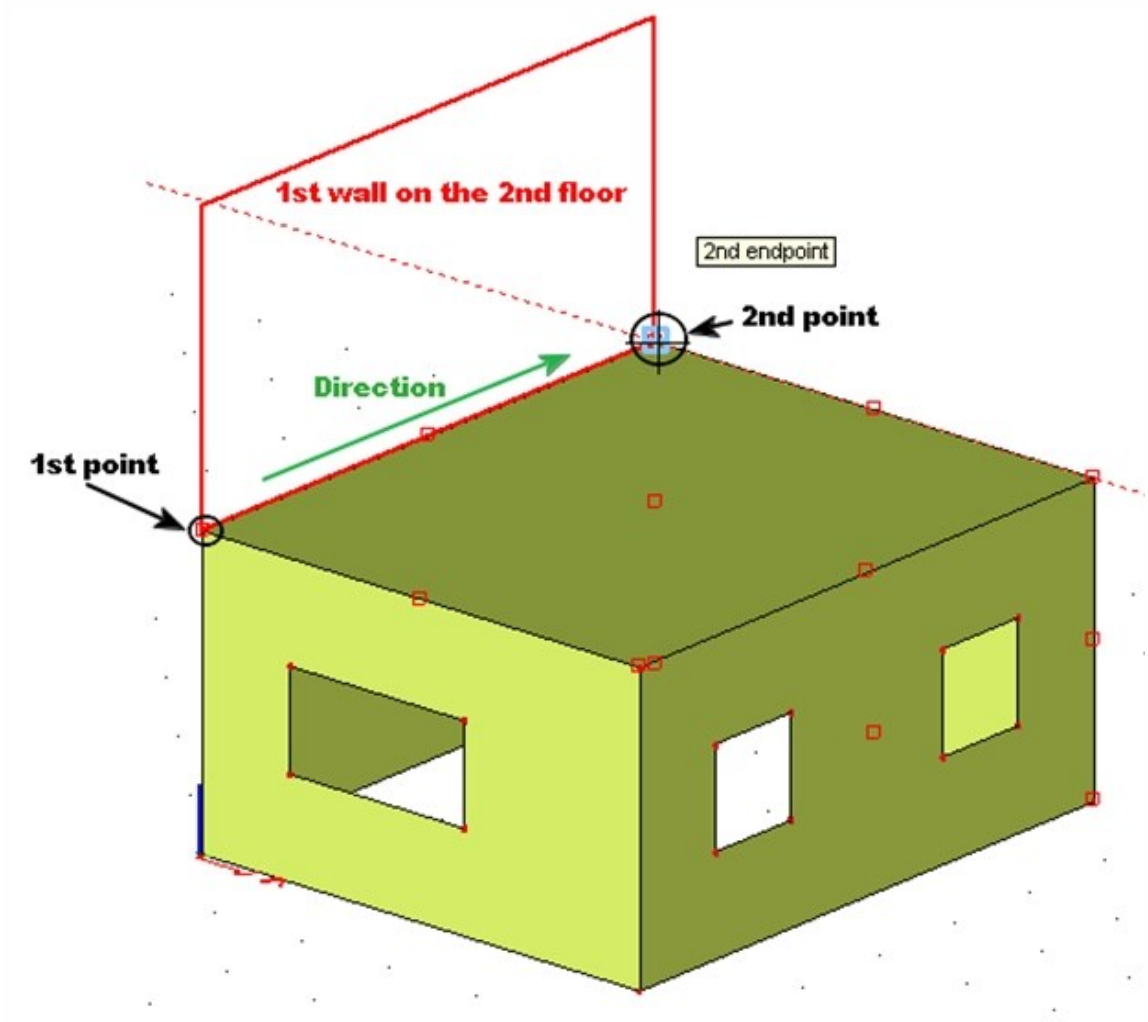
a. Insert 1st point of the 3rd wall.
b. Move over the node in 0,0,0 and hold SHIFT. This node is automatically marked with theblue rectangle.
c. This point now works as tracking point – it means that we can use rays from this point. Now we have two points with rays.
d. Intersection point is on the intersection of those two rays. New intersection point is automatically marked with little circle.



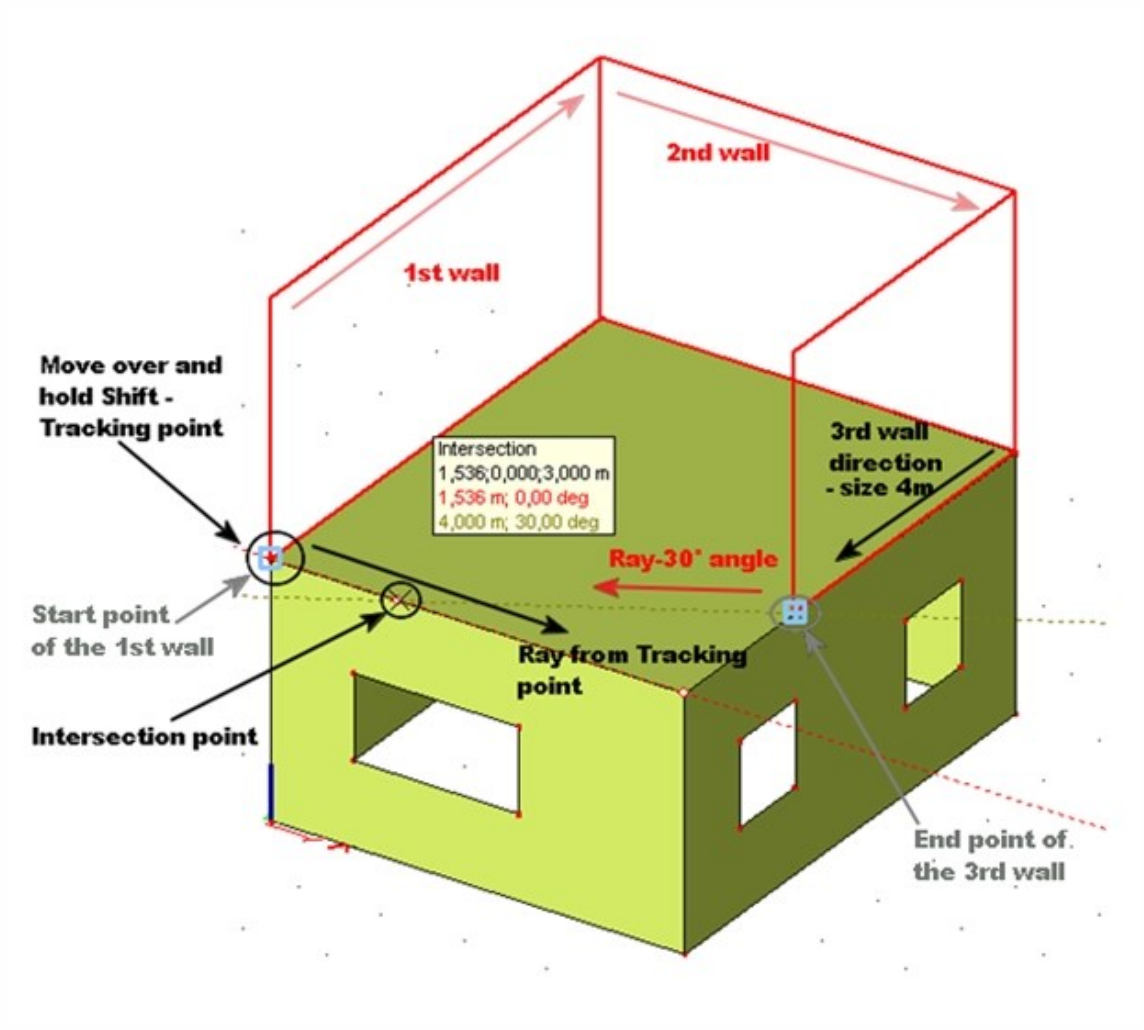
5. Insert a plate on walls. Create some openings in the walls.
6. Use tracking rays during the creating openings. Sizes are defined by the dotgrid.



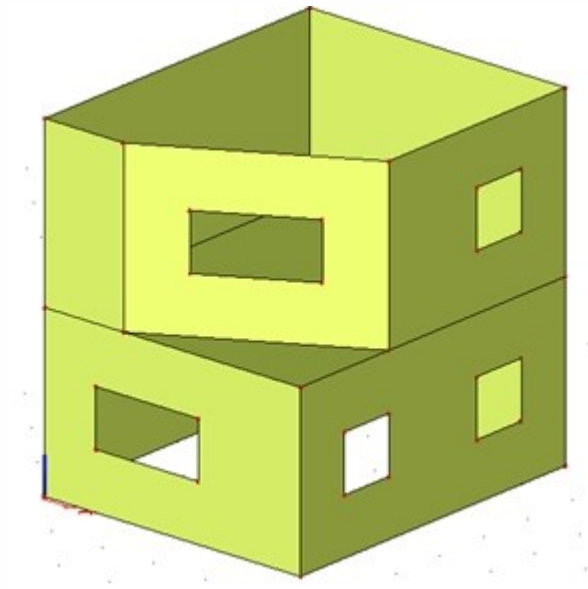
7. The second floor is defined by 5 walls and we use ray under 30 ° angle to create the 3th wall as inclined wall.
8. The 1st wall has the same size as wall under it.



9. The 2nd wall has thethe same size as the wall under it. The 3rd wall is shorter – the length is only 4m.
10. Create the 4th wall using the tracking ray under angle 30°. Start from the end point of the 3rd wall and use the user-defined tracking point on the starting point of the 1st wall. The intersection point is created by ray under 30° and the ray from the new tracking point on the left corner. See the picture.

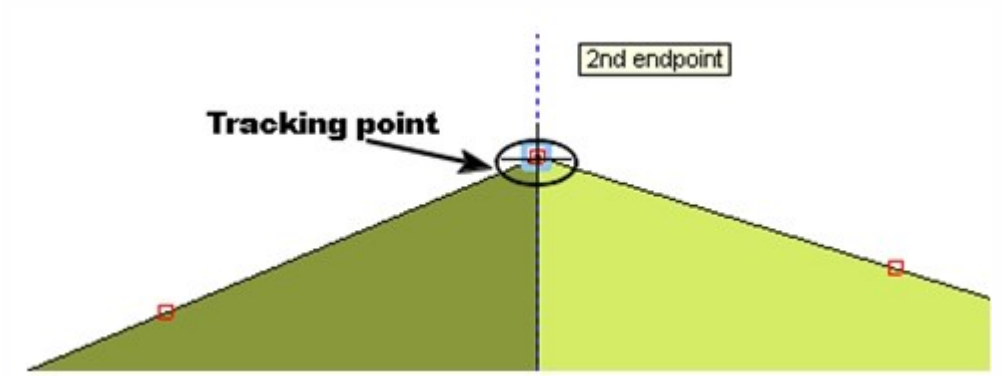


11. Create some opening on the second floor.

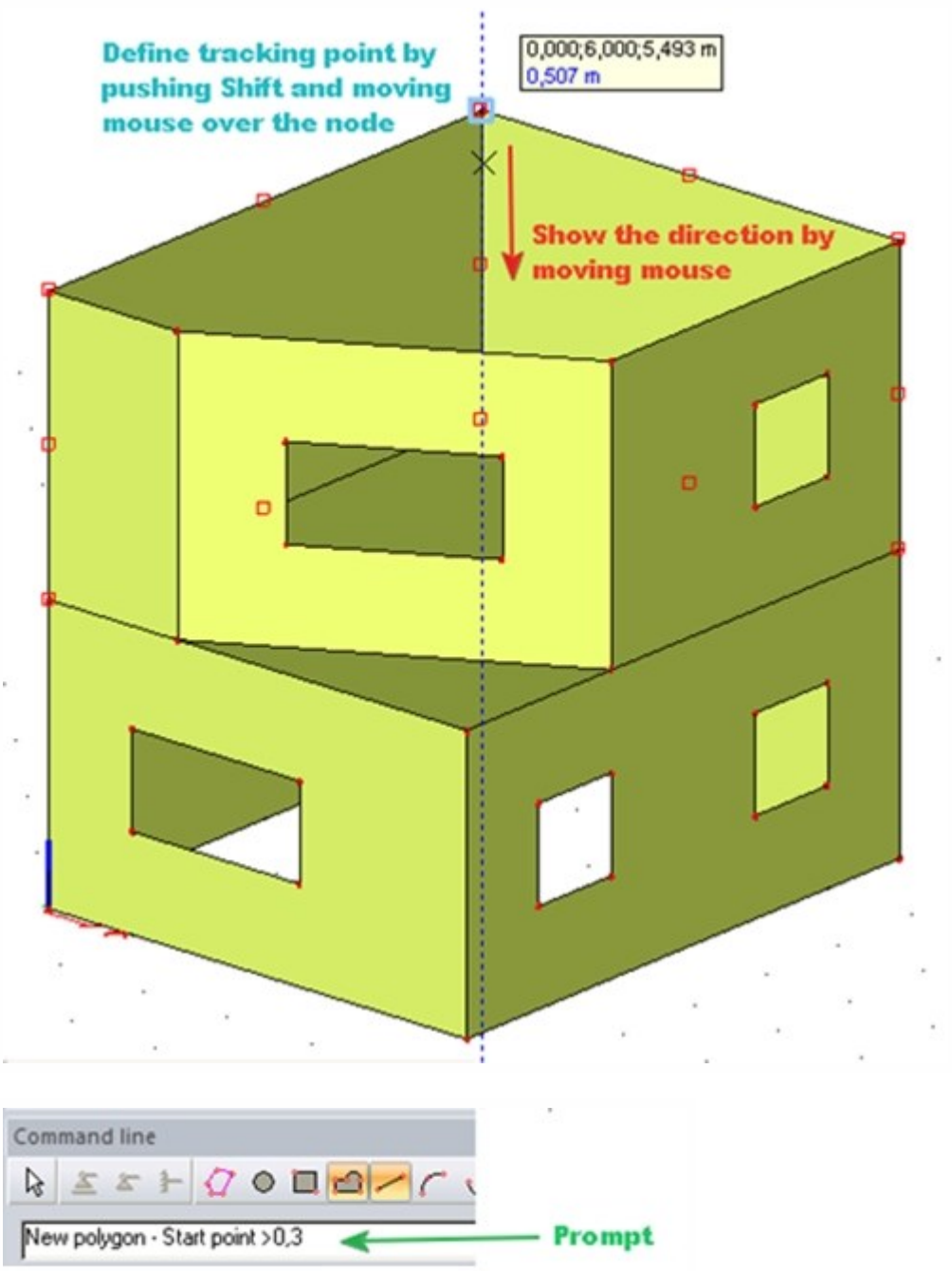


Create a roof using user-defined length in the command line.

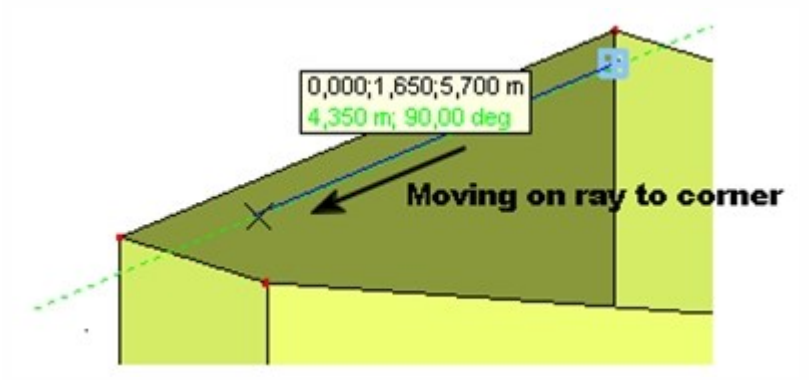
12. We use command line (prompt) to define attic height.
13. Start a command for inserting a new plate but don't create any point.
14. Set a new user-defined tracking point on the top node.



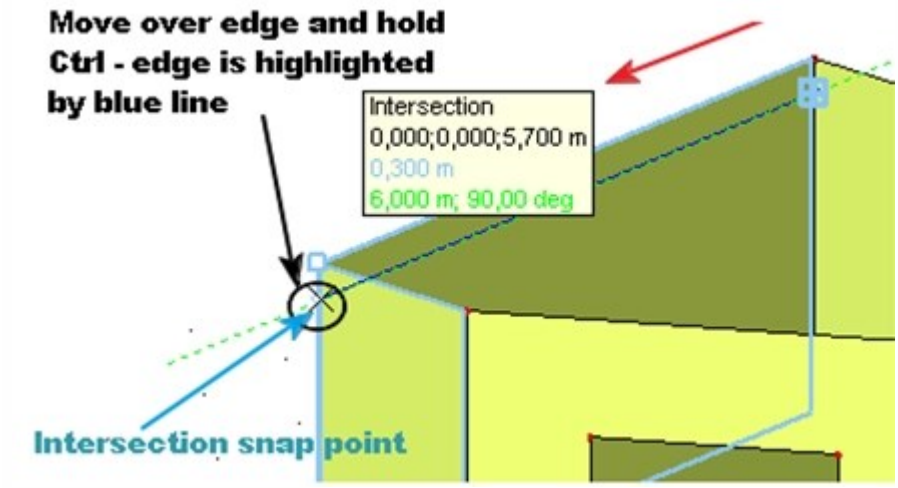
15. Move the cursor on the ray downwards to define the correct direction. Write 0,3 to the command line (just write 0,3 on keyboard, the value is automatically inputted to the command line) and press Enter. The first inserting point of the plate is created.



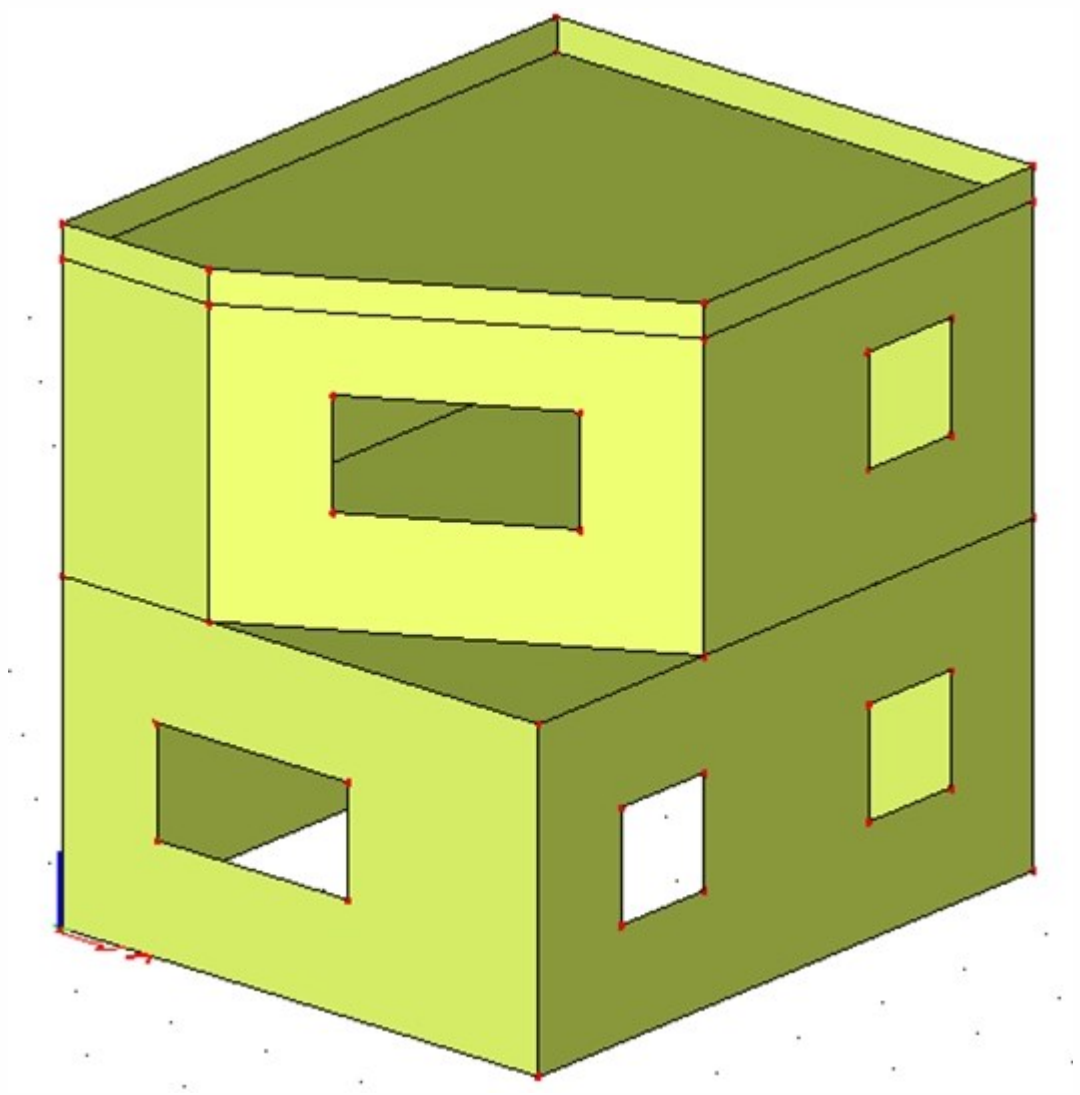
16. Move the cursor on the ray from last point next corner – see the picture.



17. Define the edge of the wall as a tracking curve. Move cursor over the edge and hold CRTL. Curve is automatically highlighted by the blue color.



18. There is an tracking intersection point on the tracking curve and the tracking ray.
19. Use the same approach for creating the whole roof – using tracking lines on edges and tracking rays from the last inserted points.



The final structure is in the project "final_track.esa".

Dot grid

Introduction to a dot grid

A dot grid is an area in the graphical window covered with regularly spaced dots to aid drawing. The spacing between grid dots is adjustable. The grid dots are not plotted.

The dot grid is always put into the current working plane, so that it can be used for the definition of points (e.g. end points of individual members) by means of mouse.

Properly adjusted dot grid may significantly speed up the process of geometry definition.

SCIA Engineer offers two types of the grid: orthogonal and radial.

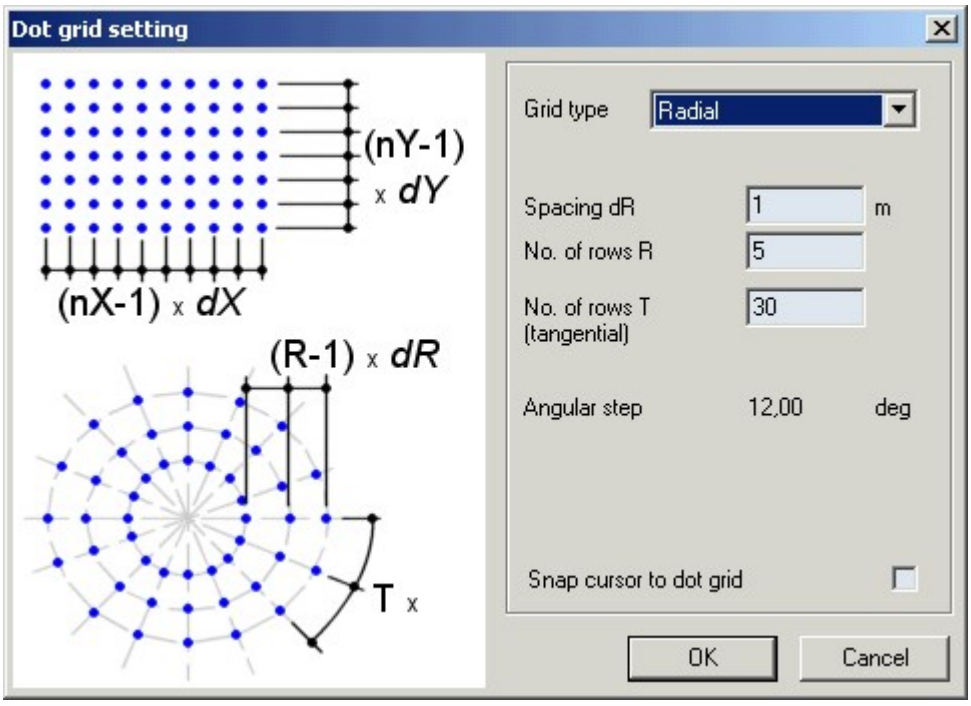
Adjusting dot grid parameters

The dot grid can be adjusted to meet the needs of a particular project. Sometimes, it may be good idea to re-adjust the grid settings from time to time, especially if the geometry of the whole structure is not regular and varies from one part to

another.

The procedure to adjust dot grid parameters

1. Open the Dot grid setting dialogue:
   1. Either using toolbar View and its button [Setting of the dot grid] ( ),
   2. Or via menu function Tools > Dot grid settings
2. Select the required type of the grid: orthogonal or radial
3. Type in the parameters of the grid (the individual parameters are self-explicable).
4. Close the dialogue.



7. The adjusted grid will be displayed on the screen unless it is switched off.

Using the dot grid

The dot grid may be used to insert points if the following conditions are met:

- the dot grid is switched on (i.e. it is displayed),
- the snap mode is adjusted to stick to the grid points,
- the program is in the point definition mode.

To be precise, the first condition does not have to be fulfilled and the dot grid may still be used. But as the dots of the grid are not visible, it is not recommended to use this configuration (unless you are a really advanced and skilful user of SCIA Engineer).

Displaying the dot grid

The dot grid may be switched on and off using menu function View > View > Show / hide dot grid.

Setting the snap mode to use the dot grid

The capability of the snap mode to stick to the dot grid can be set in two different dialogues. The result is the same regardless of which dialogue is used.

Setting in Snap mode dialogue

1. Open the Cursor snap setting dialogue.
2. Tick the option Dot grid on or off (as required).
3. Close the dialogue.

Setting in Dot grid settings dialogue

1. Open Dot grid setting dialogue.
2. Tick option Snap cursor to dot grid on or off (as required)
3. Close the dialogue

Line grid

Introduction to a line grid

A line grid is a kind of a three dimensional grid. Individual vertices of the grid can be used to define points of the modelled structure.

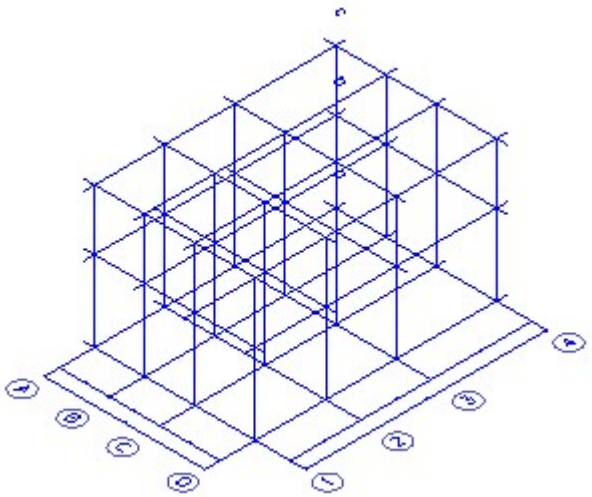
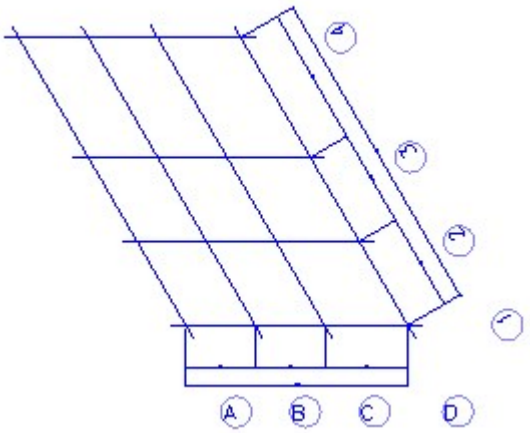
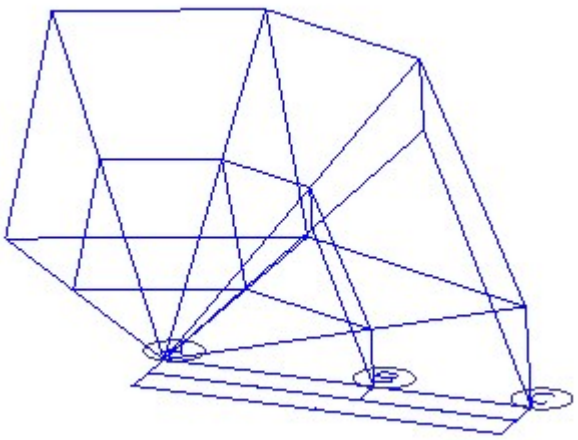
One can imagine the line grid as a set of wire cubes placed one next to another to create a larger wire cube. The vertices of individual small wire cubes are the vertices of the line grid. What's more, the cubes may be not only regular cubes, but also other solids like a tetrahedron, irregular hexahedron, etc. The grid may be of either regular or irregular (variable) dimensions in any direction.

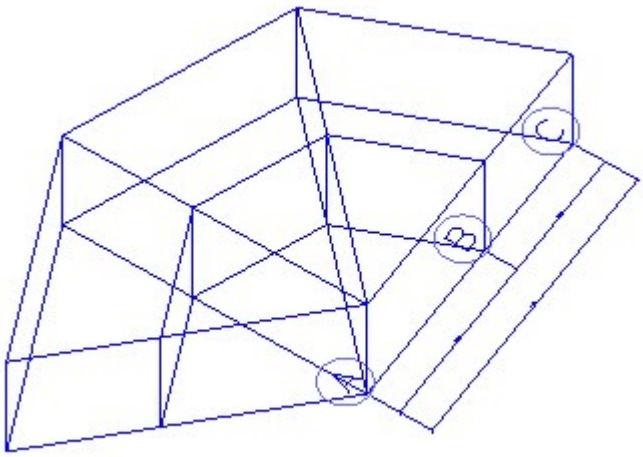
The tool is extremely useful for the definition of complex 3D structures on condition that at least some parts of the structure are regular (i.e. of the same spans or of the same height).

Types of line grid

A line grid may be of several types. Each type may be useful for different "configuration" of the geometry of a modelled structure.

Cartesian	This line grid represents the basic type. The vertices of the grid are defined in Cartesian co-ordinates and the grid as a whole resembles a regular rectangular prism.

	
Oblique	This type is based on the previous one. In addition, the user may define two angles that make the grid oblique. 
Spherical	Vertices of this grid type are defined by means of spherical co-ordinates. 

Cylindrical	Vertices of this grid of this type are defined by means of cylindrical co-ordinates. 

Line grid manager

The Line grid manager provides for operations related to line grids. It can be used to create a new line grid, to switch the existing line grids on or off, to modify an existing grid, to copy it or delete it.

The manager is operated the same way as any other SCIA Engineer database manager.

To open the Line grid manager click button [Line grid manager] on View toolbar.

Creating a new line grid

Similarly to a great number of other "objects" in SCIA Engineer, a new line grid can be created in the appropriate database manager. The Line grid manager has been designed to create and edit line grids

The procedure to create a new line grid

1. Open service Line grid and storeys.
2. Select function 3D line grid..
3. The editing dialogue is opened.
4. Specify grid dimensions.
5. Adjust its display parameters.
6. Close the editing dialogue.

Note: If no line grid has been defined in the current project so far, step 1 leads directly to opening of the editing dialogue. As a result, step 2 is automatically skipped.

Adjusting line grid parameters

Each line grid is defined by means of:

- dimensions in individual directions,
- location of its origin (i.e. the insertion point),
- possible rotation,
- angles of obliqueness (for oblique line grid only),
- name,
- parameters of its display style.

Line grid type

The combo box at the bottom part of the dialogue selects the required grid type.

Line grid dimensions

Depending on the grid type, the dimensions are defined in Cartesian, spherical, or cylindrical co-ordinates.

There are two ways to define the individual "spans" and "storey heights":

- the user inputs the dimensions of individual "spans" and "storey heights",
- the user inputs the co-ordinates of individual line grid vertices (i.e. co-ordinates of end-points for individual "spans" and "storeys").

The approaches are independent for each direction. In other words, the user can specify the dimension of the grid in X and Y direction by means of "span" lengths and then use grid absolute co-ordinates for the definition of individual "storeys" (in the case of Cartesian type) or vice versa. Which approach will be used can be set in the combo box located above the table for each particular direction.

Another general rule is that:

- either each "span" and each " storey" of the line grid is defined explicitly,
- or a "span" or "storey" dimension is input only once and the number of repetition of this dimension is added (if "spans" or "storeys" of the same dimension are adjacent to each other).

The latter can be user for grid with repetitious "spans" and may significantly speed up the definition of the grid.

Insertion point and rotation

This point defines the location of the grid in the global co-ordinate system.

If required, the whole line grid may be rotated around the global Z-axis.

Name

The name serves for easy identification of individual line grids if more than one grid are defined.

Parameters of display style

The user can control the way the line grid is displayed on the screen.

Adjusting the display style of line grid

The user can easily control the appearance of the grid on the screen by means of a few parameters. The parameters are grouped on the Drawing setup tab of the line grid editing dialogue.

Base plane	This parameter specifies which plane is the base plane for the labelling system of the grid.
Lines between planes	Connecting lines may be or may be not drawn between individual grid layers (i.e. "floors" or "spans" depending on the base plane).
Label format	The user can control the format of the labels.
Visibility of grid layers	Individual grid layers (i.e. "floors" or "spans" depending on the base plane of the grid) may be visible or hidden.
Labelling of grid layers	Individual grid layers may be labelled.
Dimensioning of grid layers	Dimension lines may be added to individual grid layers.

Base plane

The base plane defines the plane where the main grid labels will be located. The user can select from the three base planes oriented in the three main planes of the global co-ordinate system (XY plane, YZ, plane, XZ plane).

Lines between planes

The individual grid layers (e.g. "floors" in case of XY base plane) may be graphically connected to each other or may be drawn as separate layers. If the lines are drawn, the final line grid looks like a three dimensional solid. If the lines are not drawn, the final grid resembles of a set of sheets put one above the other.

Label format

The user may adjust the format of the labels. The following parameters can be specified:

- position of labels,
- offset of labels,
- text size,
- a circle drawn around labels.

Visibility of grid layers

Each layer can be separately set as visible or hidden. This may be very useful especially for large and complex line grids.

Labelling of grid layers

Labels are added to individual layers according to the user's settings. There are two types of labels:

- labels for individual "spans" in a grid layer,
- labels for the whole grid layer.

Each of the types is controlled by a separate parameter.

Dimensioning of grid layers

The individual grid layers may be equipped with dimension lines. The dimension lines may dimension:

- either individual spans in individual directions,
- or the total dimension in individual directions.

Displaying and hiding a line grid

A line grid can be switched on / off (in other words displayed / hidden or activated /deactivated) in the Line grid manager. It is possible to switch on as many different line grids as required.

The procedure for switching a line grid on / off

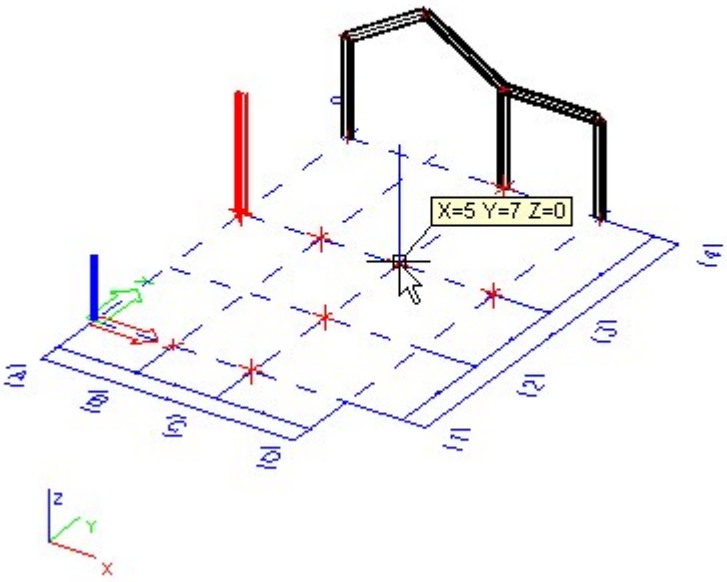
1. Open the Line grid manager.
2. In the list of defined line grids select the line grid you want to switch on or off.
3. In the grid property table tick option Visible in order to switch the grid on, or remove the tick from this option to hide the grid.
4. Repeat points 2 and 3 as many times as required.
5. Close the Line grid manager.

Using a line grid

In order to use a previously defined line grid, two conditions must be met:

- at least one line grid must be switched on,
- the SNAP mode must be set to pick points of line grid.

Once the two conditions are met, the vertices of the displayed line grids may be used to define points. When the mouse cursor is positioned on a line grid point (vertex), the program automatically detects it, snaps to it and shows its co-ordinates. If the user wants to use the highlighted point, the only thing he/she have to do is click the left mouse button.



The picture above shows the use of line grid for the insertion of columns during creation of a model of a hall.

Editing an existing line grid

The way to edit parameters of an already defined line grid is very straightforward and simple.

The procedure to edit an existing line grid

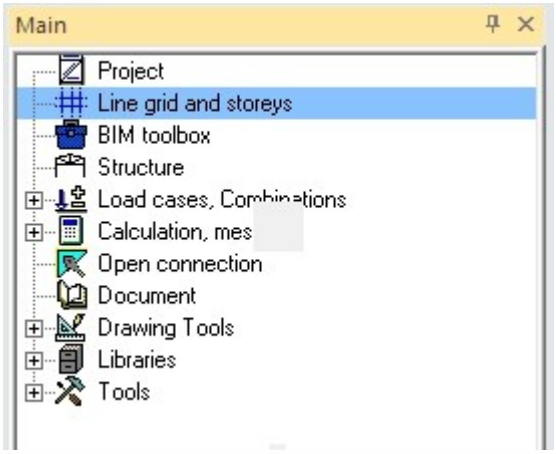
1. Open the Line grid manager.
2. Select the grid you want to modify.
3. Click button [Edit] to open the editing dialogue.
4. Change the required parameters on the Input data tab.
5. Change the required parameters on the Drawing setup tab.
6. Close the editing dialogue.
7. Close the Line grid manager.

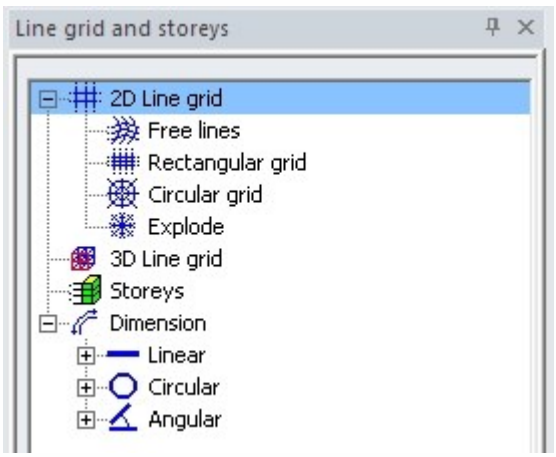
If a defined line grid is no longer needed it may be deleted. The Line grid manager's button [Delete] can be used for this operation.

2D Linegrid

2D linegrid

2D linegrid is a tool for simple orientation in the project. It is located in the service Line grid and storey.





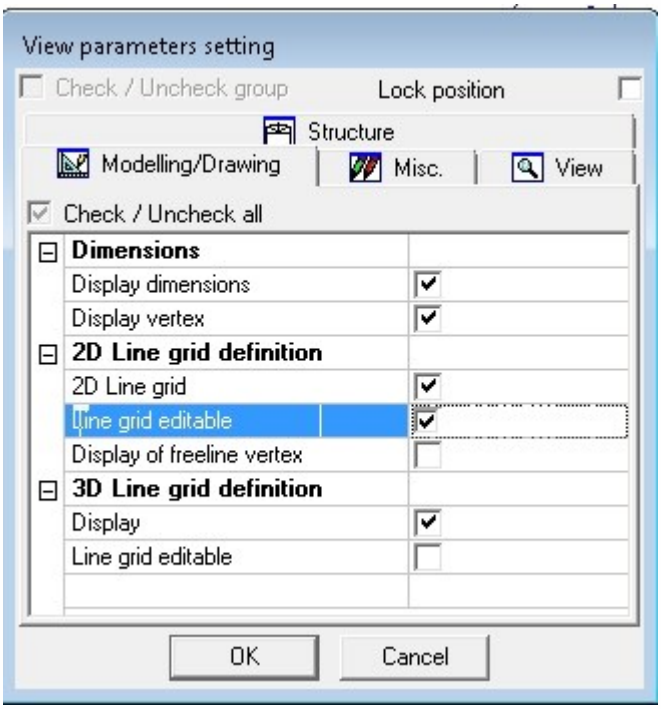
There are three different types of 2D linegrid: Free line, Rectangular and Circular.

2D linegrid is a object similar to the 3D linegrid.

2D linegrid is always inserted to GCS of project. It can be displayed higher only by storeys view parameters settings.

Overall settings

Linegrids are non-editable in the 3D window when the service Linegrid and Storeys is not active. This is controlled by view parameters.



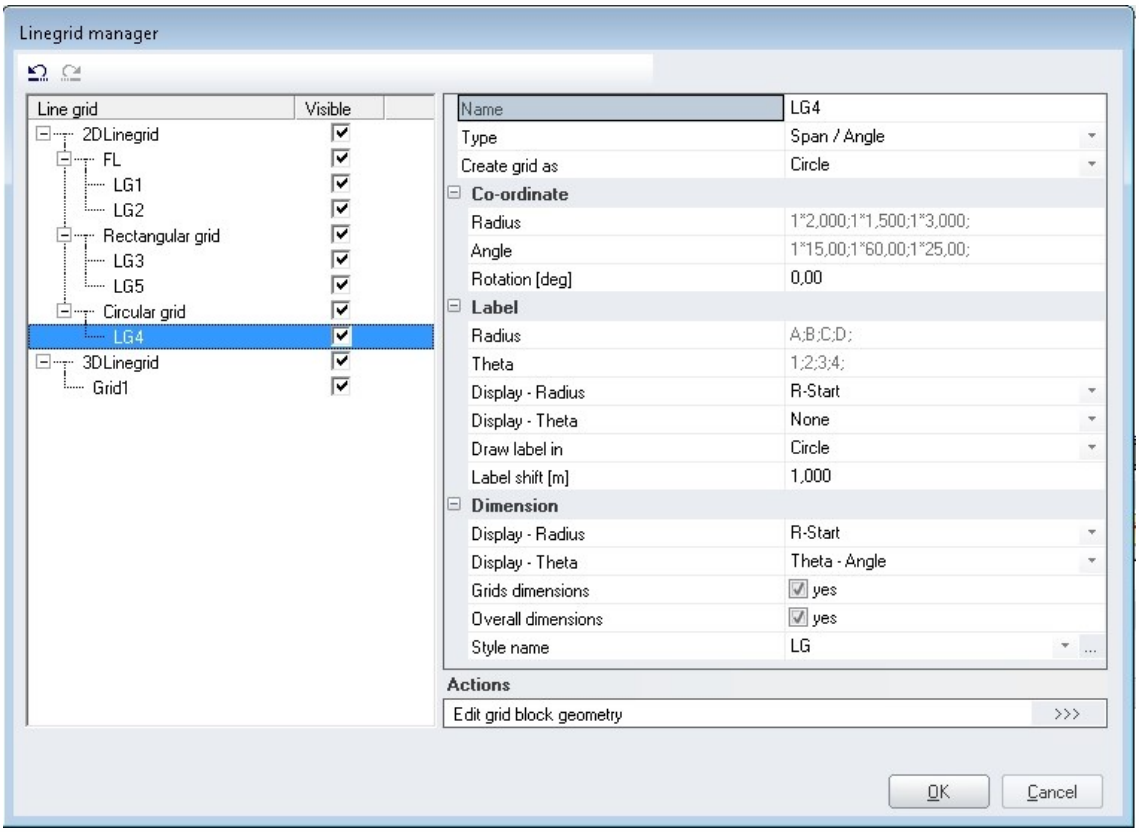
Linegrid manager

All linegrids in the project are displayed according the settings in the Linegrid manager.

It can be found on the bottom of the 3D window.

The dialogue contains all linegrids in the project and checkboxes set if the linegrid is visible or hidden. The right part shows properties of the selected linegrid.

Linegrids are grouped according to is types, so all linegrids from one type can be hidden by one checkbox.



Linegrid and Storeys service

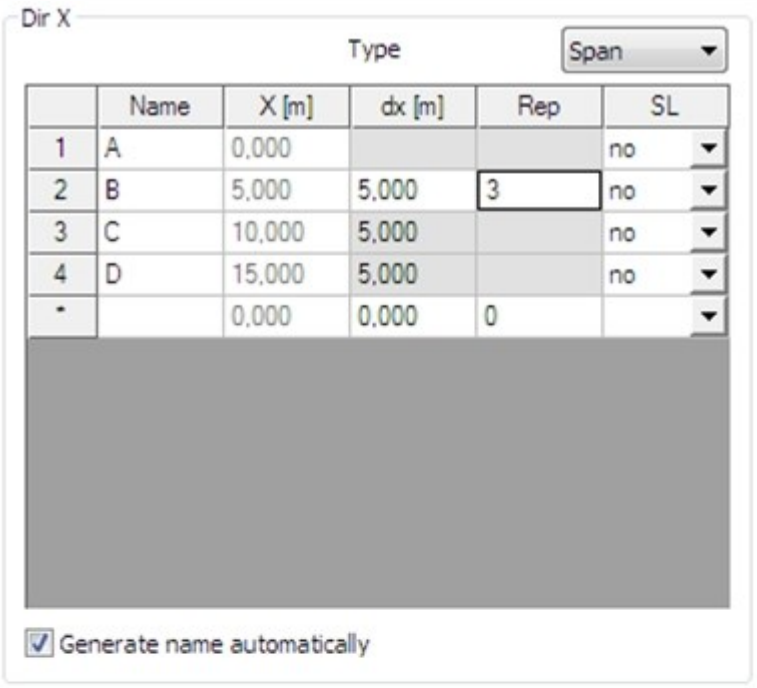
Chapter about Free lines

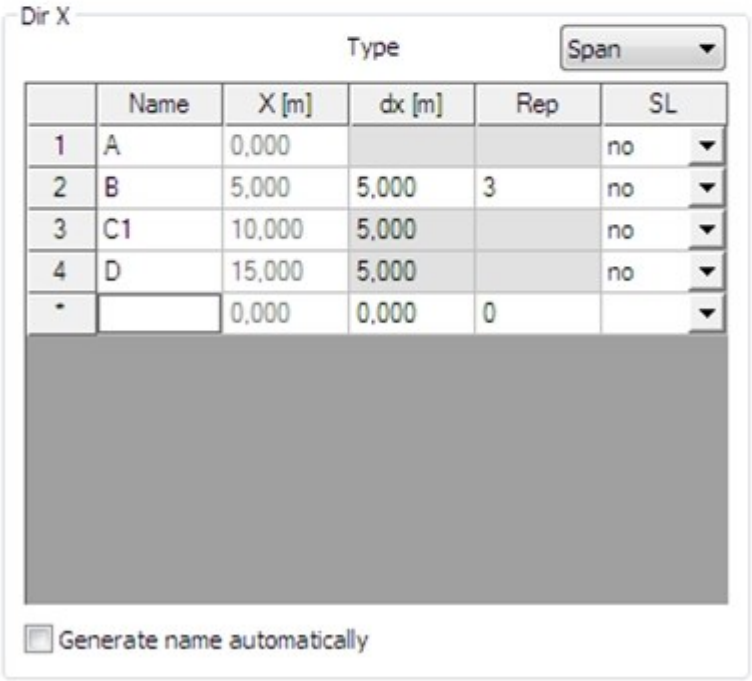
Chapter about Rectangular linegrid

Chapter about Circular linegrid

Generate name automatically

The checkbox for automatic generation is available for 2D rectangular and circular linegrid (it is also available in the dialogue for 3D linegrid).

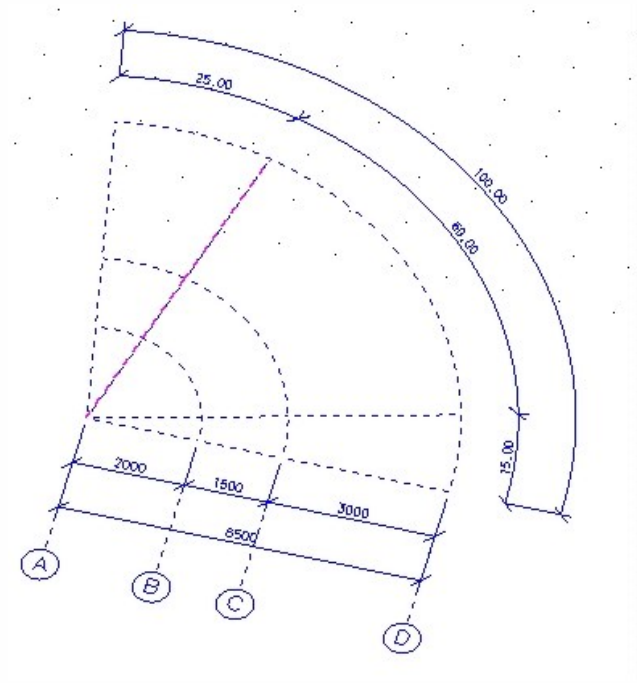


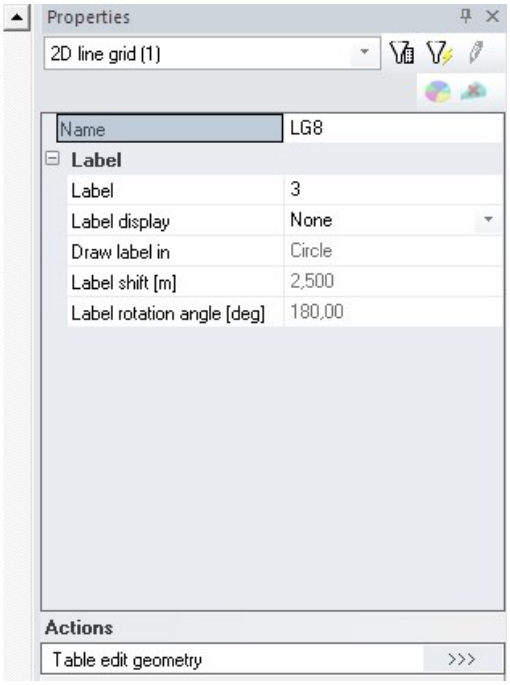


Automatic naming is defined by the first name and numbers are generated automatically. When the checkbox is unchecked then user can define any name without automatic function. Re-check the functionality starts automatic generation according to the first name.

Checkbox is unchecked automatically when any name is changed by user except the first one.

Explode



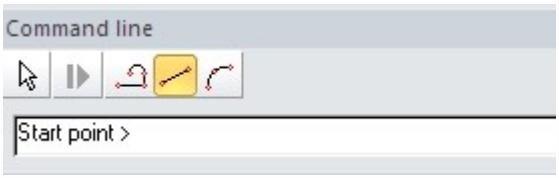


The explode functionality split up the whole Linegrid to the separate Free lines. There are properties of a free line displayed in Property window.

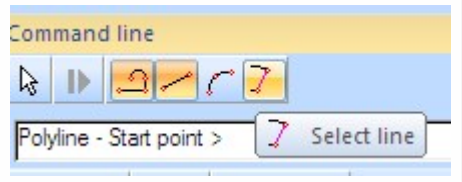
Explode works only for 2D Linegrid.

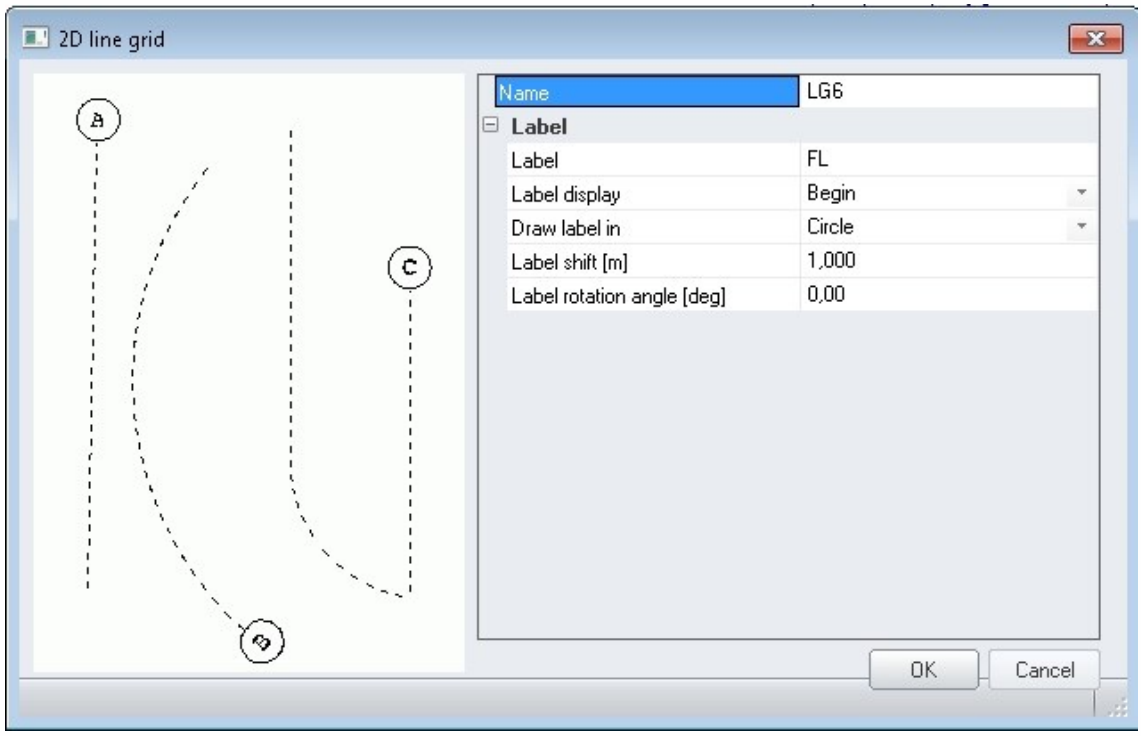
Free lines

It is a simple line which works like linegrid line. It is displayed by the dashed line and label. The curve is defined in the same way as geometrical members.



There is a possibility to convert the general 3D line to a Free line. This is done by a standard toolbar button "Select line".





Name
Standard property for all entities in SCIA engineer

Label
Description in label

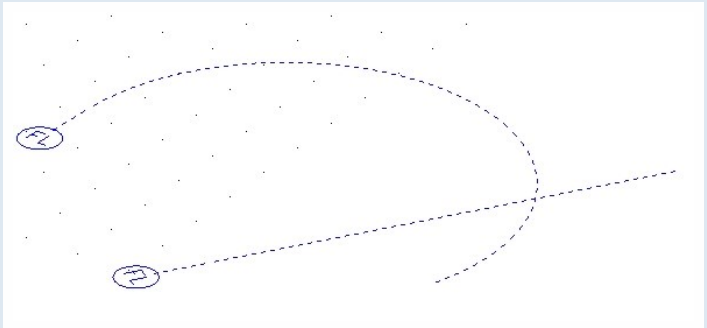
Label display
Position of label according to the linegrid orientation

Draw label in
Shape of label

Label shift
The distance of the label from linegrid

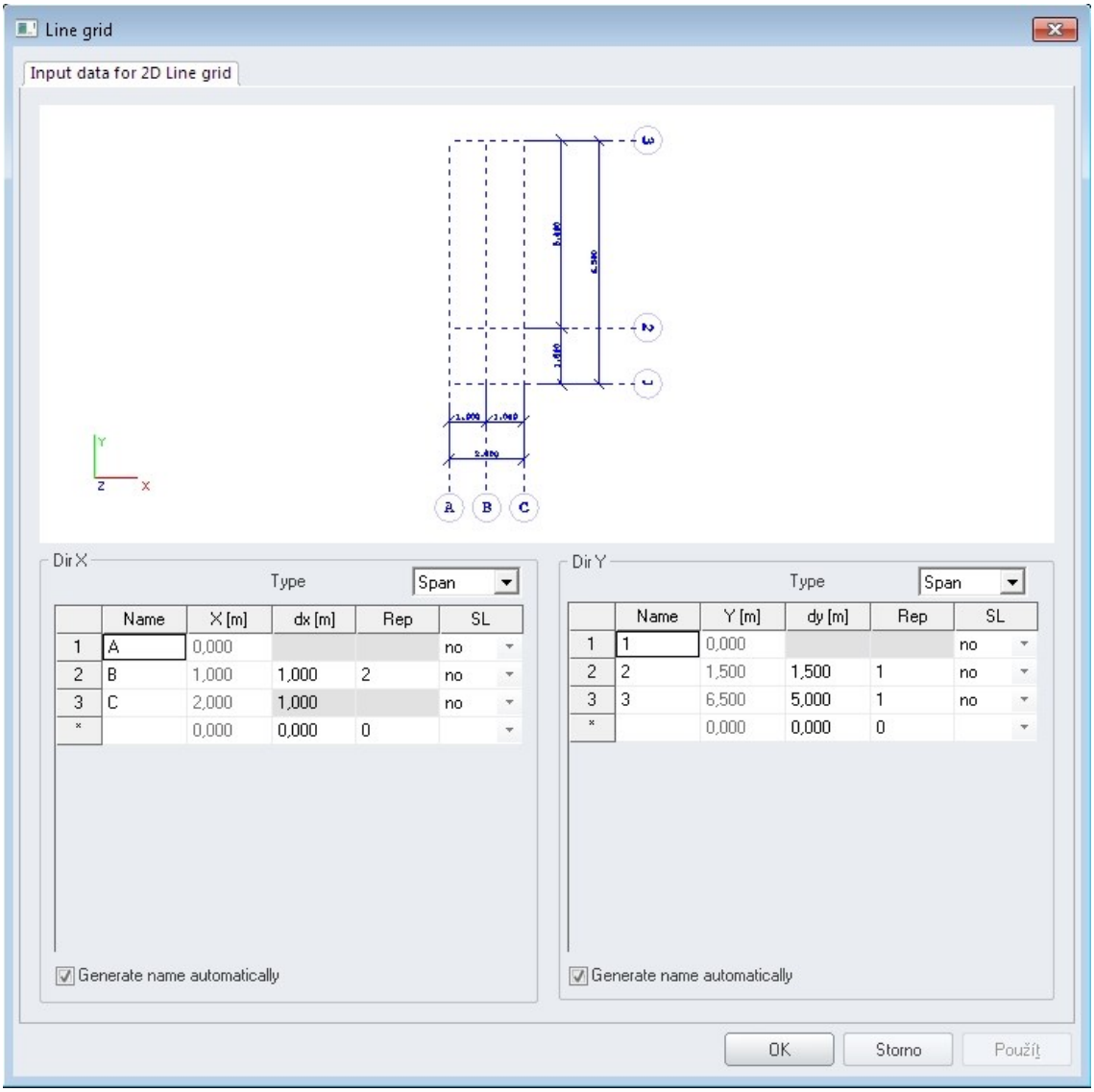
Label rotation angle
Rotation of label with respect to the linegrid

Example of free line linegrid:



Rectangular linegrid

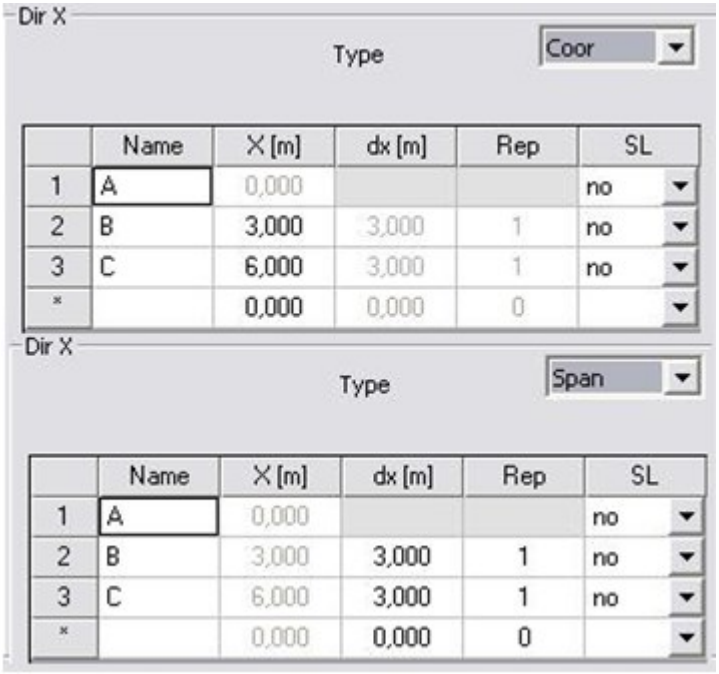
This type is created by the grid. There are two grids, one for x direction and one for y direction. User fills in the coordinates and number of repetition. The dialogue shows also a preview of the linegrid.

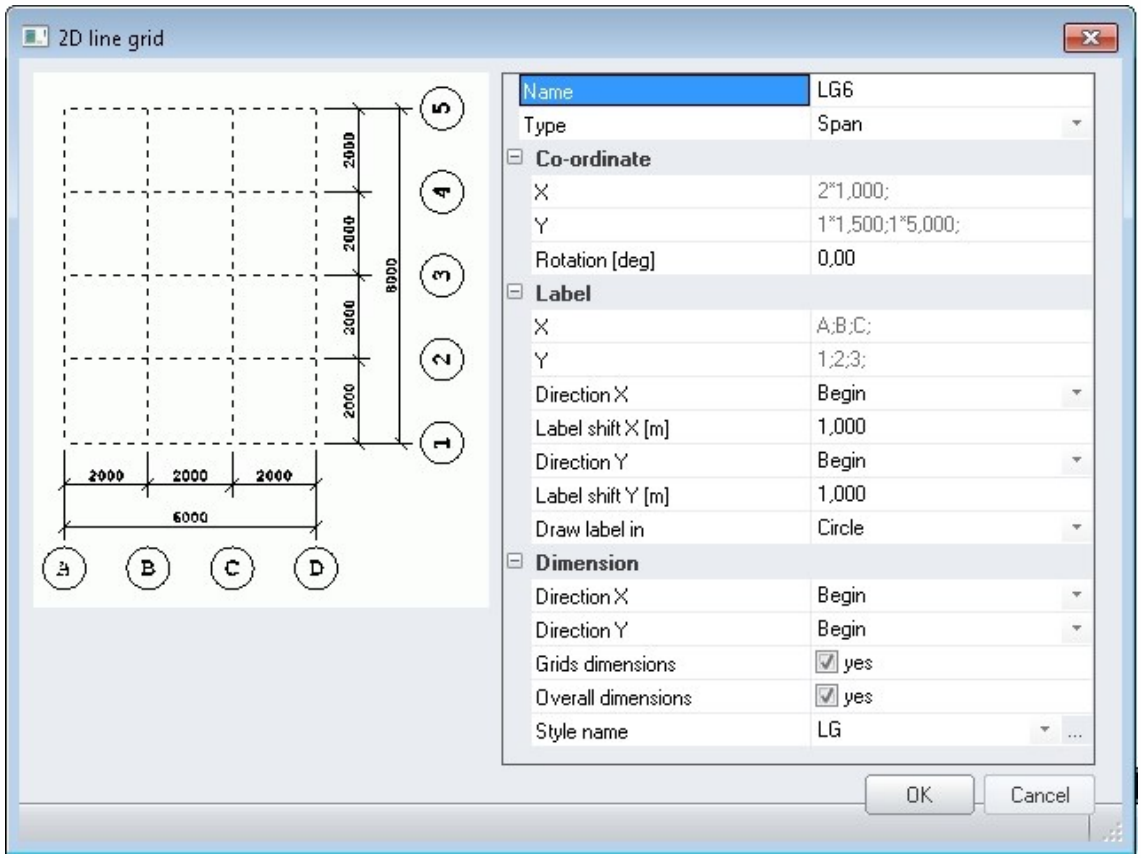


The values for distances can be set by two types:

1. Coor – user inserts coordinates to the second columns, the third and the fourth columns are inactive.
2. Span – user inserts distances between linegrid lines to the third column and number of repetition in the fourth one. The second column is inactive.

The setting SL marks the line by a special label.





Name
Standard property for all entities in SCIA engineer

Type
Type of the grid

Co-ordinates X, Y
Values from the grid

Rotation

Rotation of linegrid

Label X, Y
Values from the grid

Direction X (Direction Y)
Position of the labels in X direction according to the linegrid orientation – None, Begin, End, Both

Label shift X (Label shift Y)
Distance of the label from linegrid in X direction

Draw label in
Shape of label

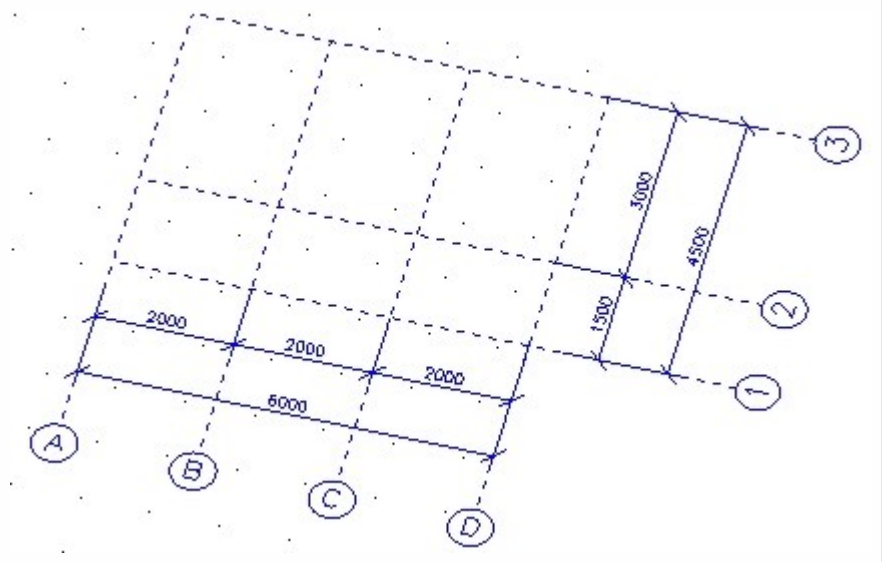
Dimension – direction X (Dimension – direction Y)
Position of dimensions in X direction according to the linegrid orientation

Grids dimensions
Visible/hidden dimensions on grids – distances between linegrid lines

Overall dimensions
Visible/hidden overall dimensions – distance from the first to the last line

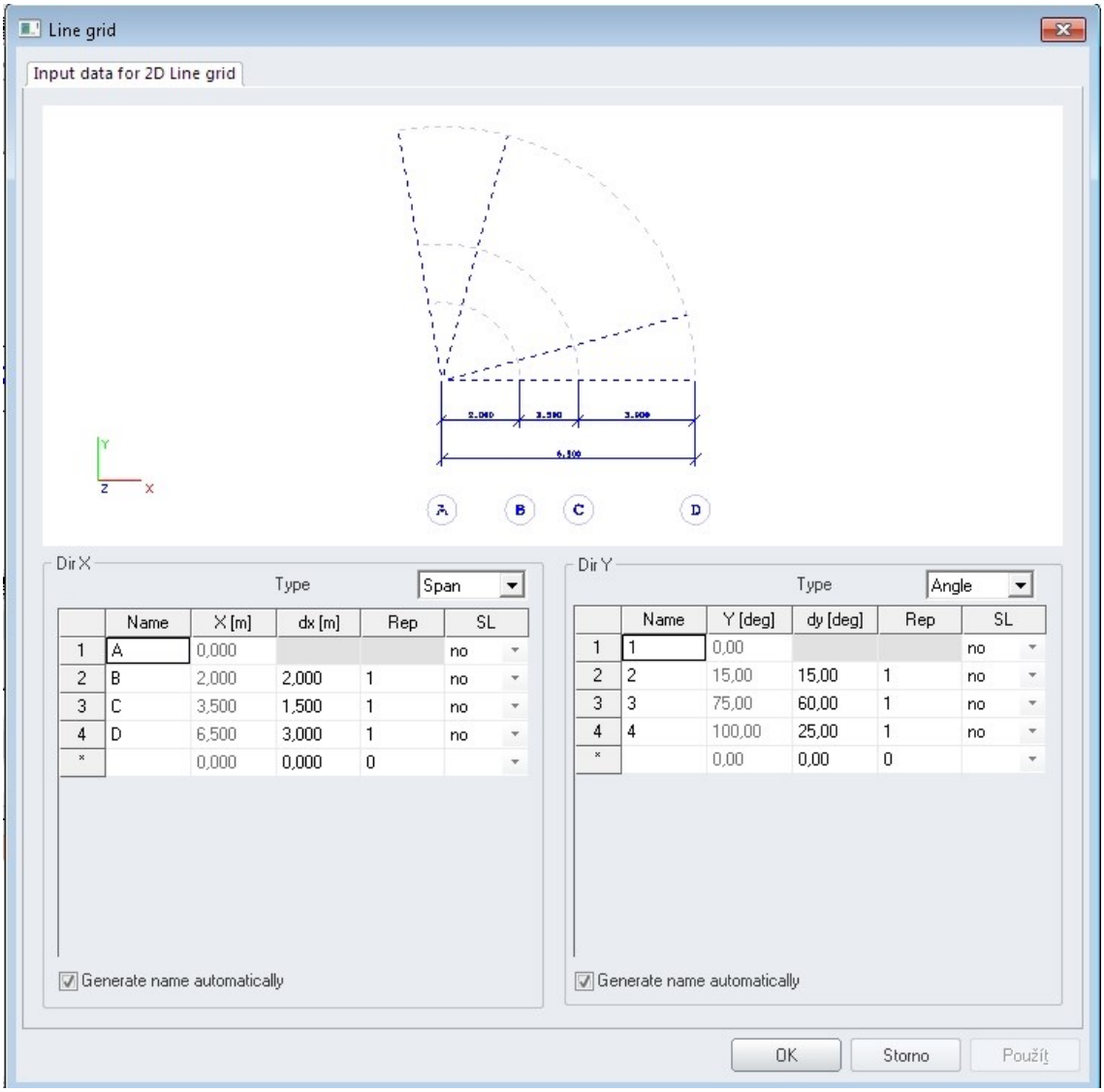
Style name
Dimension style

Example of Rectangular linegrid:



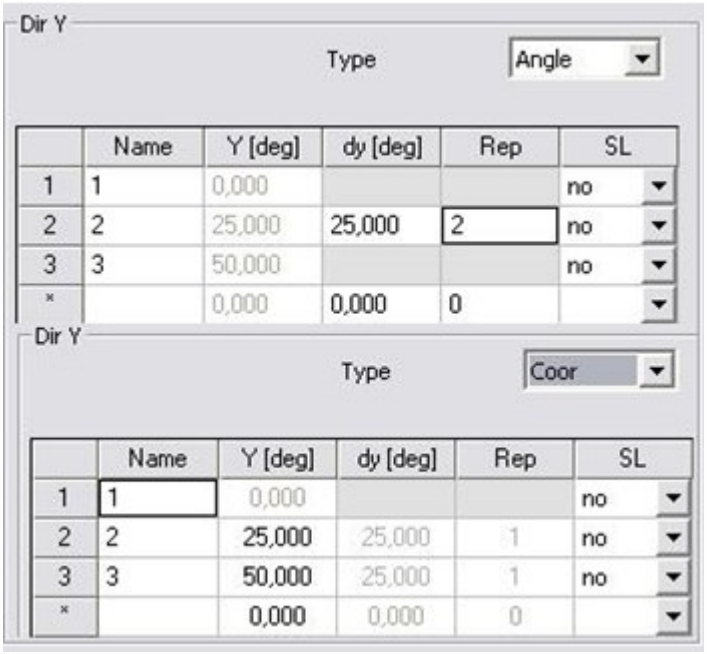
Circular linegrid

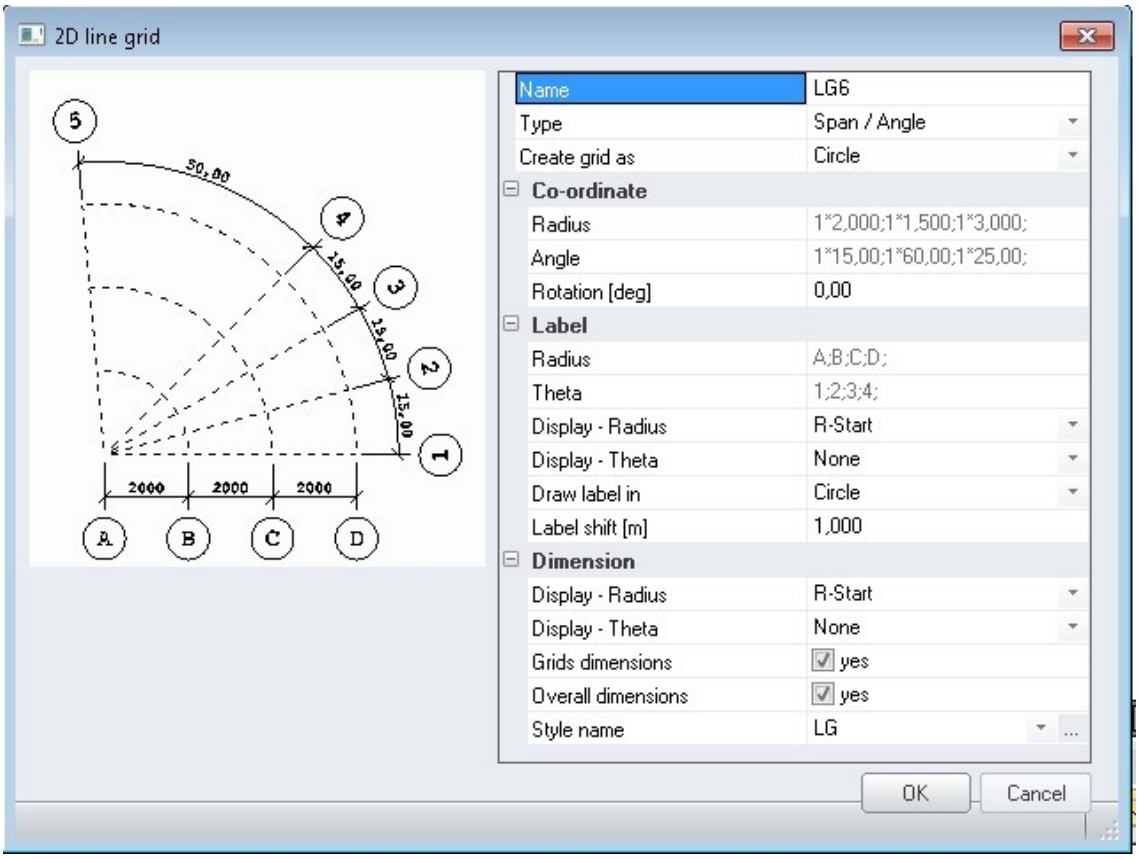
This type of linegrid is similar to the rectangular one. There is one grid for coordinates and one for angles.



The Y direction can be defined by angles or by coordinates:

1. Angle – user sets the values for angles to the third column and sets the number or repetition.
2. Coors – user sets distances from one linegrid line to another.





Name
Standard property for all entities in SCIA engineer

Type
Type of inserting values in table

Create grid as
Linegrid can be displayed as circle or polygon

Co-ordinates Radius, Angle

Values from the grid

Rotation
Rotation of linegrid

Label Radius, Theta
Names for labels from the grid

Label Display - Radius
Position of labels in radius – None, R-Start, R-End, Both

Label Display - Theta
Position of labels in theta – None, Theta-End

Draw label in
The shape of the label

Label shift
The distance of the label from linegrid

Dimension Display - Radius
Position of dimensions – None, R-Start, R-End, Both

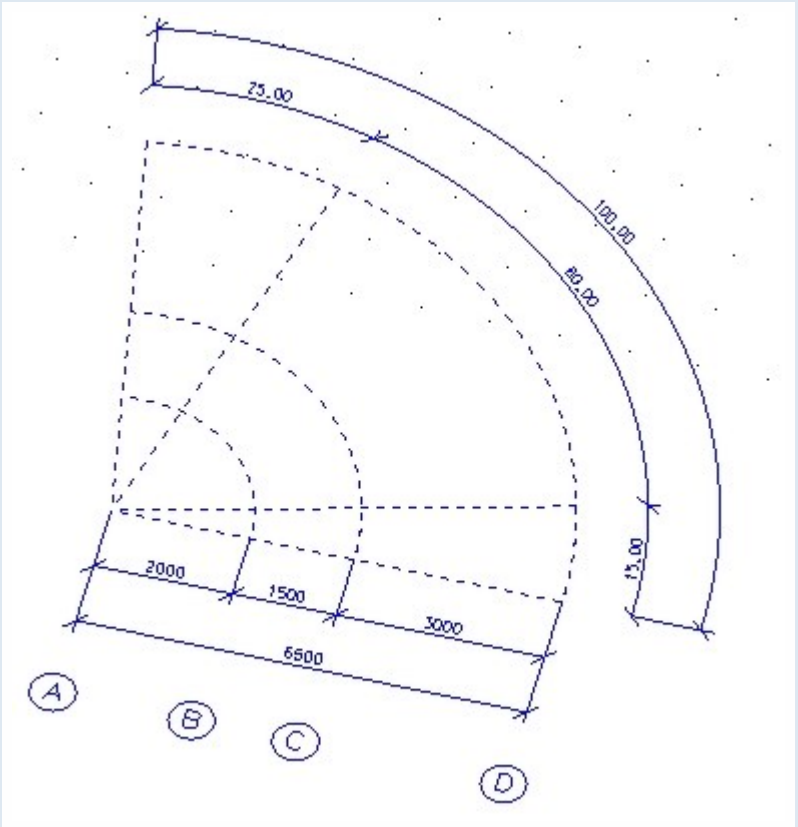
Dimension Display - Theta
Position of dimensions – None, R-Start, R-End, Both

Grids dimensions
Visible/hidden dimensions on grids – distances between linegrid lines

Overall dimensions
Visible/hidden overall dimensions – distance from the first to the last line

Style name
Dimension style

Example of circular linegrid:

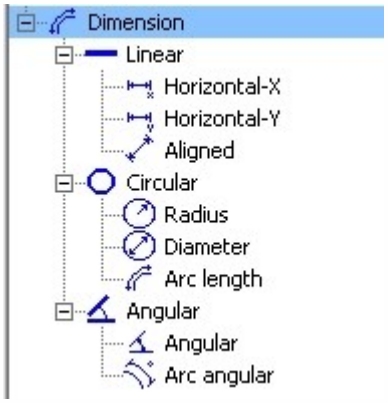


2D linegrid dimensions

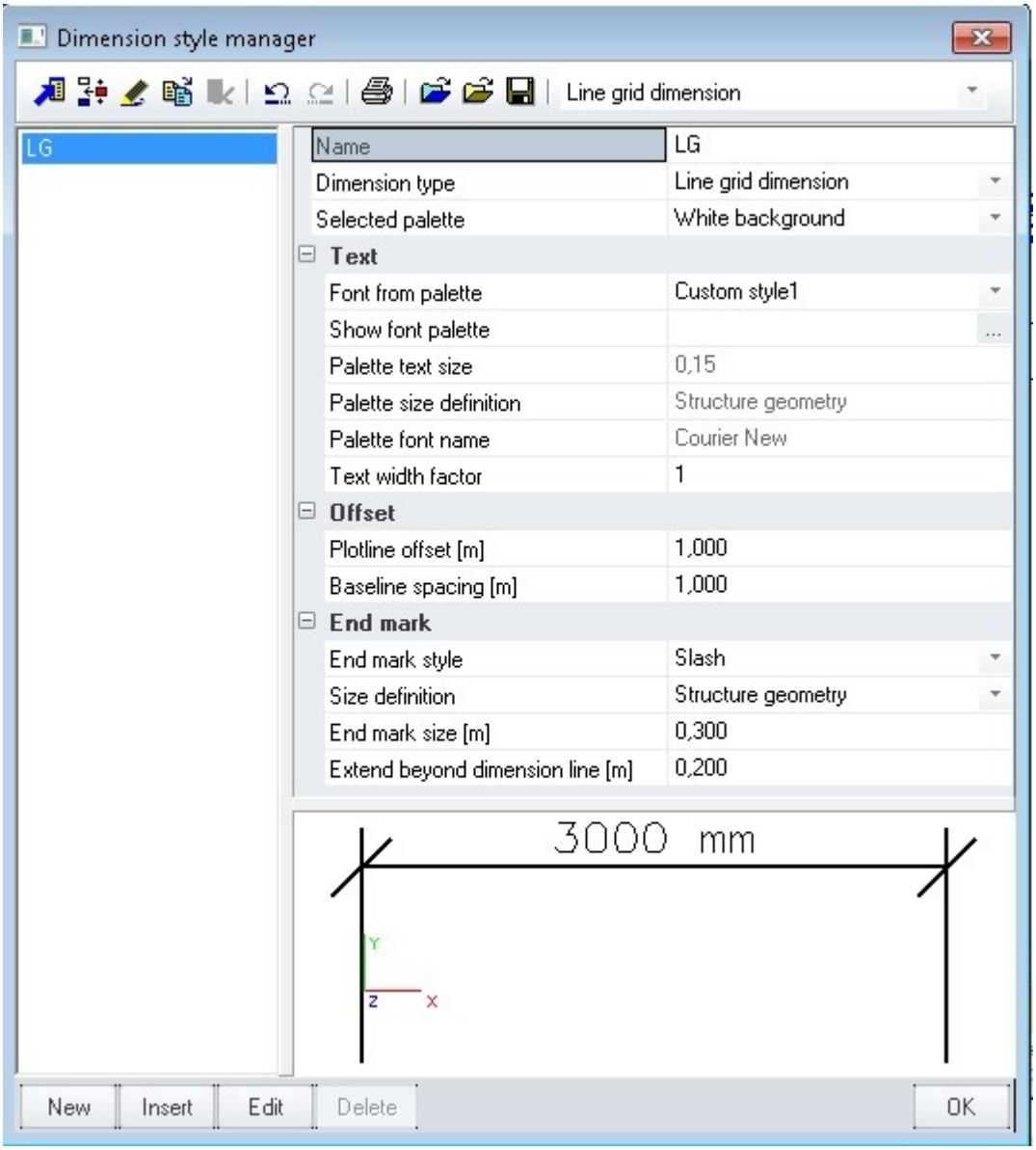
Linegrids can be adapted in the 3D window by 2D linegrid dimensions. Those dimensions are developed specially only for linegrids. The 2D linegrid dimensions are inserted to the working plane, it cannot be used in the same way as 3D dimensions.

Dimensions types – Linear, Circular and Angular

Placing of 2D dimension is similar to the placing 2D Linegrid.

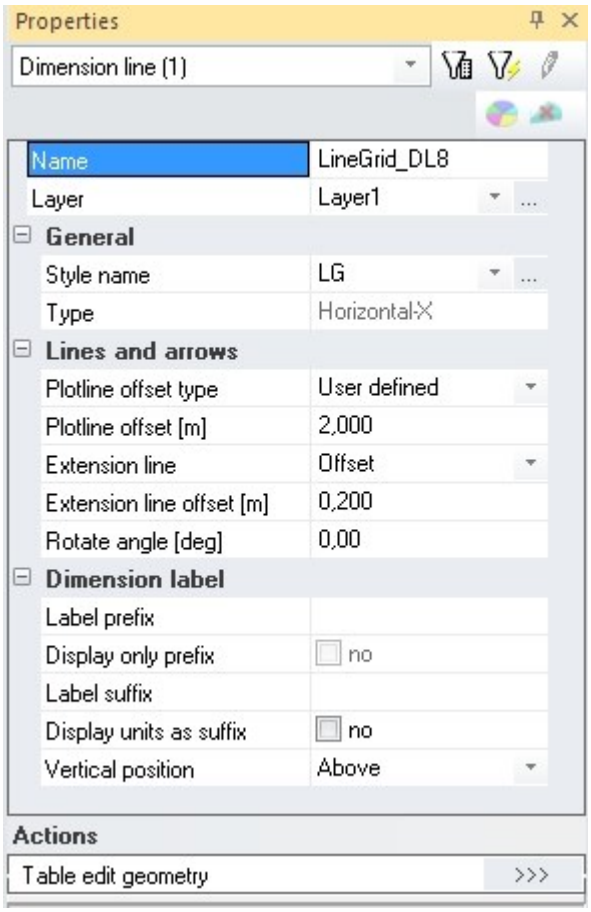


The linegrid dimension has its own type of dimensions style – Linegrid.



Dimension line properties

There is a possibility to adapt the dimension line geometry in the properties or display the library of Dimension styles and adapt the style.



Name
Standard property for all entities in SCIA Engineer.

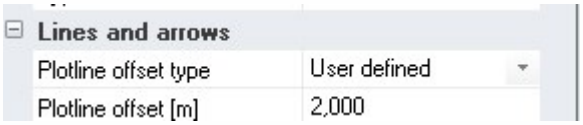
Layer
Standard property for all entities in SCIA Engineer.

Style name
Tells which predefined dimension style is used for creating dimension line.

Type
Inactive, type of used dimension.

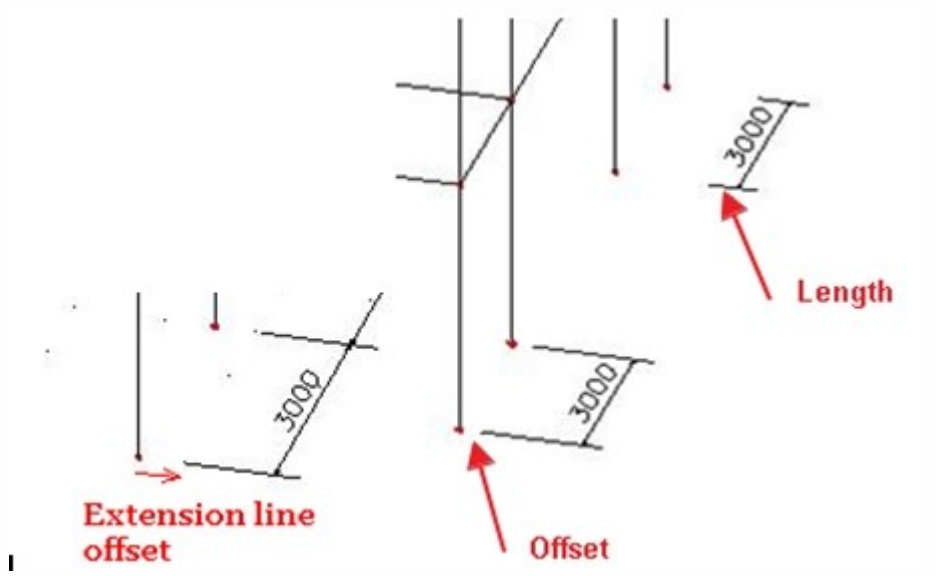
Plotline offset type
1) Style – the plotline offset is loaded from the Dimension style

2) User defined – the user can set the value for offset manually in the properties



Extension line
1) Length – user set the size of extension line beyond end mark

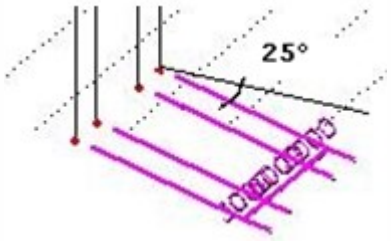
2) Offset – user set the size from inserting point of the dimension line to the end of the extension line



Extension line offset
The value for offset from the dimensioned point to the starting point of extension line.

Extension line	Offset
Extension line offset [m]	0,200

Rotate angle
Value specifying the rotation of the dimension line from the working plane, example with 25°.



Label prefix
This value can be used as prefix for text label of dimension line.

Display only prefix
Here the user can check if he wants to use only the prefix on selected dimension lines, then he will have all dimensions with same text label on them.

Label suffix
This value can be used as suffix for the text label of dimension line.

Display units as suffix
This check box automatically adds units as suffix to the text label of dimension line.

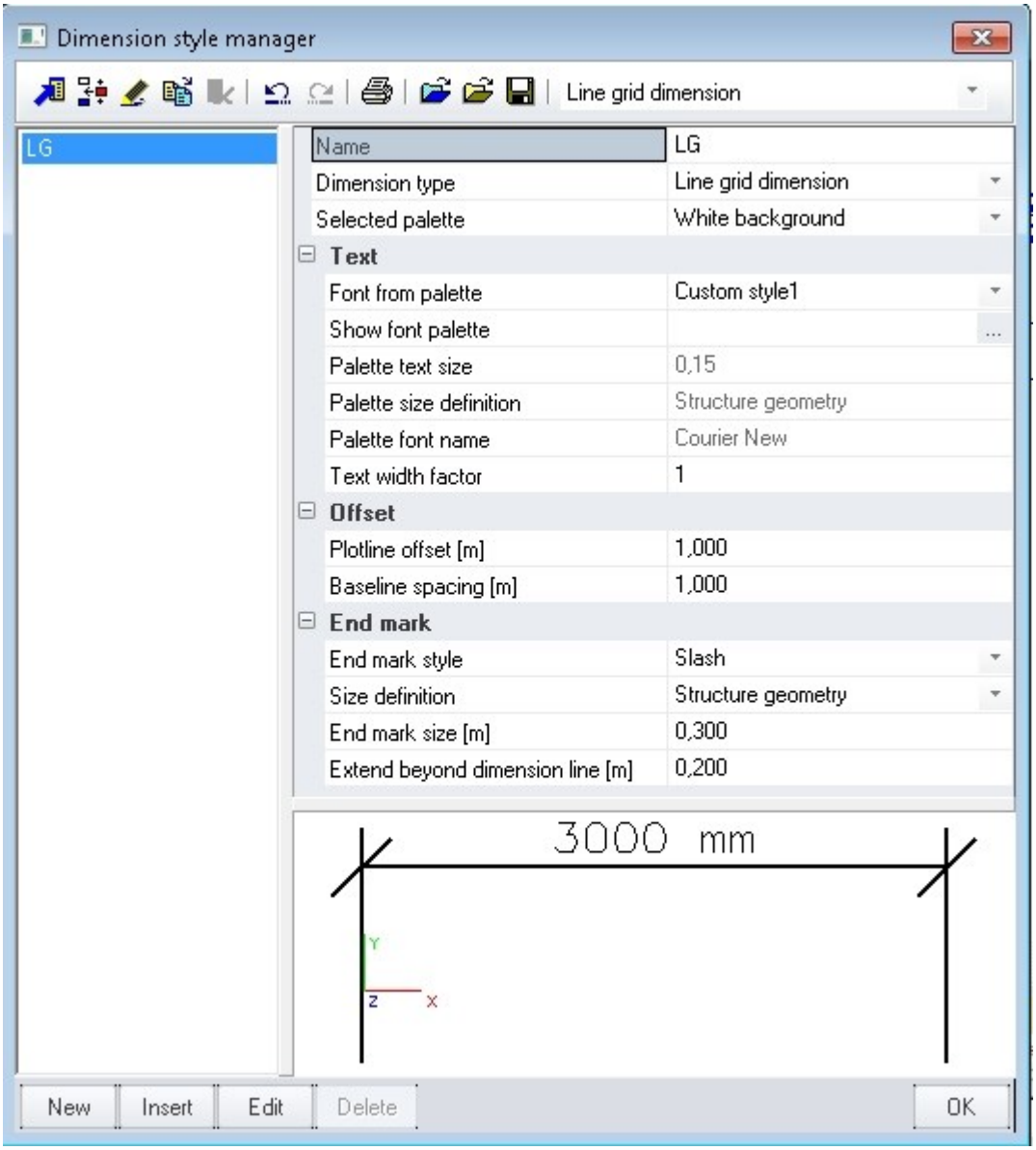
Dimension label
Above, Inline, Below – moves position of the text label

Vertical position	Above

Dimension style manager

Definition of Dimension style for dimension lines in 2D and 3D window is set in Dimension style manager. It's a standard LIB manager with possibility to save (load) styles to the format DB4.

When the plotline offset type is set on Style in the properties, the offset is taken from the library. When it is switched to the User defined, values can be modified in properties of the selected dimension.



Name
Standard property for all entities in SCIA engineer

Dimension type
Filter for 2D, 3D and Linegrid dimension style

Selected palette
Defines palette: White background – 3D model, Graphic output – 2D window..., for this palette are the settings under this combobox predefined

Font from palette
Text font used for dimension, adjustment is possible through Palette settings – link Show font palette

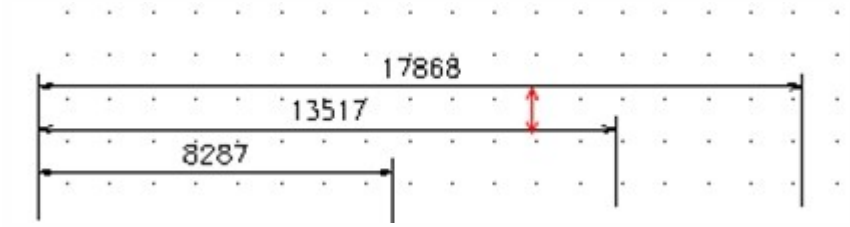
Palette text size, Palette size definition, Palette font name
Values are taken from the Palette settings – it is inactive

Text width factor

Width of text

Plotline offset
The dimensions line offset from its inserting nodes

Baseline spacing
Offset between dimension lines and its baseline dimension line



Endmark style
Shape of end mark

Size definition
Settings for end mark – if it will be displayed according to the Graphic device or Structure geometry

Endmark size
Size definition

Extend beyond dimension
Length of the line beyond the end mark, it is also dependent on the Size definition

Window pop-up menu

Introduction to window pop-up menu

Every graphical window that is created in SCIA Engineer has a pop-up menu associated with it. This pop-up menu provides for fast access to frequently used functions. The menu can be invokes by clicking the right mouse button if the mouse cursor is positioned inside the window.

The list of functions offered in the pop-up menu depends on several factors:

- whether any function is opened (has been activated),
- whether some entities are selected,
- whether the mouse cursor is positioned on some entity (at the moment when the right mouse button is being pressed),
- if the mouse cursor is positioned on some entity ,what kind of entity it is.

In addition to the pop-up menu in graphical window, SCIA Engineer offers also a similar menu in a document window. This particular type of pop-up menu is described in chapter covering the document.

Functions of the pop-up menu

The pop-up menu of the graphical is created dynamically. That means that the functions offered in the menu vary according to the current state of the program.

Standard pop-up menu

Zoom all	Displays the whole model.
Zoom – cut-out	Displays the selected cut-out so that it fits the whole area of the graphical window.

Set view parameters	Opens the dialogue for adjustment of view parameters, i.e. the parameters that control the way the modelled structure is displayed on the screen.
Cursor snap setting	Opens the dialogue for adjustment of required SNAP mode.
Copy picture to clipboard	Copies the contents of the graphical window into clipboard of Windows system.
Export picture to file	Saves the contents of the graphical window into an external file. The user may choose from a list of supported file formats.
Picture to document	Inserts the contents of the graphical window into the document as a new picture.
Picture to gallery	Inserts the contents of the graphical window into the Picture gallery as a new picture.
Print picture	Opens the graphic output dialogue and allows the user to carry out the print set-up before the print itself.
Wire model in manipulations	If the option is ON and the view direction or zoom is being adjusted by means of mouse (i.e. appropriate control keys and right mouse button held down during mouse movement), only a simplified wire representation of the structure is displayed during the operation of adjustment. If the option is OFF, the normal (or full) display is used during the operation. It is clear that the latter may lead to slower response of the program.
Picture wizard	Starts the wizard for generation of pictures. See appropriate chapters in the Picture gallery.

Pop-up menu if a function is opened

If a function (e.g. Insert a new beam, Define load, etc.) is opened, SCIA Engineer adds an additional function to the pop-up menu.

End of command	This command may be used to close the currently opened function. The command closes just the function and lets the current service opened.

Pop-up menu if some entities are selected

If at least one entity is selected, the contents of the pop-up menu is rearranged in order to provide for common manipulations with the selected entities. The pop-up menu consists of the following functions:

Set view parameters (for the selected entities only)	Opens the dialogue for adjustment of view parameters, i.e. the parameters that control the way the modelled structure is displayed on the screen. The settings made here are applied to the selected entities only. As this function deals with a specified set of entities, the range of the view parameters in the setting dialogue is limited to parameters related to the selected entities.
Set view parameters for all entities	Opens the dialogue for adjustment of view parameters, i.e. the parameters that control the way the modelled structure is displayed on the screen. The settings made here are applied to all entities in the model.
Cursor snap setting	Opens the dialogue for adjustment of required SNAP mode.
View	This sub-menu comprises majority of the standard pop-up menu functions.
Move	Start function for move of 1D members.
Rotate	Start function for rotation of 1D members.
Scale	Starts function for change of the scale of 1D members.

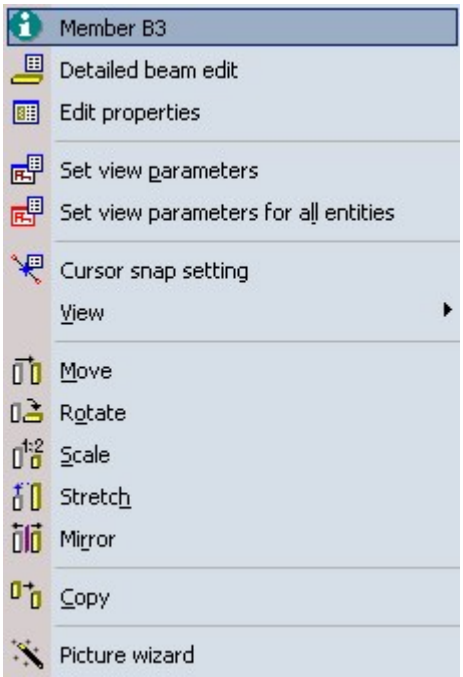
Stretch	Opens function for stretching of 1D members.
Mirror	Opens function for mirroring of 1D members.
Copy	Starts function for copying of 1D members.
Copy Add data	Starts function for copying of additional data. This item is only available if at least one entity of additional data is in the current selection.
Move Add data	Starts function for moving of additional data. This item is only available if at least one entity of additional data is in the current selection.
Delete	Opens function for deletion of selected entities.
Picture wizard	Opens wizard (i.e. a set of dialogues) that helps the user generate pictures of the modelled structure.

Pop-up menu if the cursor is positioned over any entity

If the mouse cursor is located over an entity at the moment the mouse button is clicked, the program adds a few special items that are related to the very entity under the cursor.

Brief information about the entity under cursor	This menu item contains type and name of the entity under cursor. This item performs no action, it just says the user which entity the mouse cursor is positioned over.
Edit properties	Opens the property dialogue for the entity under cursor. In this property dialogue the parameters of the entity may be changed as required.

The picture below shows a sample pop-up menu that was invoked with the mouse cursor positioned over an entity called B3.



Using the window pop-up menu

The pop-up menu of the graphical window can be invoked any time the graphical window is displayed and holds the focus.

The procedure for opening and using the pop-up menu

1. Place the mouse cursor into the drawing area of the required graphical window (please notice that several graphical windows may be opened at a time and therefore the cursor must be put into the required one).
2. If required, position the cursor over particular entity.
3. Click the right mouse button.
4. The pop-up menu appears on the screen.
5. Select the function that should be invoked and click the left mouse button.
6. The function starts or is performed (if the function does not require any parameters or response of the user, it is carried out immediately).
7. Finish the opened function.

Note: If the pop-up menu is invoked accidentally, just place the mouse cursor anywhere into the empty area of the graphical window and click the left mouse button. The pop-up menu disappears.

Adjusting the viewpoint (view direction + zoom)

Introduction to view adjustment

If a simple two-dimensional structure is being modelled and analysed, it may be sufficient enough to have just one side view of the structure during the whole design and evaluation process. However, if a complex three-dimensional structure is handled, the user needs to:

- view the structure from different sides,
- zoom in important details,
- zoom out to get the overall view,
- possibly limit the view to only a part of the structure.

All the points mentioned above can be covered by one term – the user needs to adjust the view.

This task may be carried out by means of numerous view adjusting functions that SCIA Engineer offers in its menus and toolbars.

Adjusting the view

The adjustment of the view may consist of two separate operations:

- definition of the view direction (i.e. from which side the structure is looked at),
- specification of the distance of the view point from the structure (i.e. how big the structure appears to be on the screen).

SCIA Engineer offers a wide range of functions to adjust the required view. Some functions perform just one of the two mentioned operations, others merge both of them into one action.

Menu functions for adjustment of the view

View > ZOOM > Zoom +	Zooms in.
View > ZOOM > Zoom -	Zoom out.
View > ZOOM > Zoom Cut-out	Requires to define a cut-out for the zoom. The cut-out is then magnified in order to fit into the whole area of the graphical window. Once the function is started the mouse cursor changes. Position it to the upper left corner of the cut-out. Press the left mouse button and hold it down. Drag the mouse to place the cursor to the bottom right corner of the cut-out. Release the button.
View > ZOOM > Zoom All	Zoom in or out in order to fit the whole structure into the whole area of the graphical window.
View > ZOOM > Zoom All – Selection	Zoom in or out in order to fit the selected entities into the whole area of the graphical window.
View > View > View X	Adjusts the view in such a way so that the structure is viewed from the positive X-axis direction. Simultaneously zooms in or out to fit the whole structure into the whole area of the graphical window.
View > View > View Y	Adjusts the view in such a way so that the structure is viewed from the positive Y-axis direction. Simultaneously zooms in or out to fit the whole structure into the whole area of the graphical window.
View > View > View Z	Adjusts the view in such a way so that the structure is viewed from the positive Y-axis direction. Simultaneously zooms in or out to fit the whole structure into the whole area of the graphical window.
View > View > View AXO	Sets the view point vector to (1, -1, 1). Simultaneously zooms in or out to fit the whole structure into the whole area of the graphical window.

Toolbar functions for adjustment of the view

Functions for the adjustment of the view are arranged on toolbar View.

View in direction X	Adjusts the view in such a way so that the structure is viewed from the positive X-axis direction. Simultaneously zooms in or out to fit the whole structure into the whole area of the graphical window.
View in direction Y	Adjusts the view in such a way so that the structure is viewed from the positive Y-axis direction. Simultaneously zooms in or out to fit the whole structure into the whole area of the graphical window.
View in direction Z	Adjusts the view in such a way so that the structure is viewed from the positive Y-axis direction. Simultaneously zooms in or out to fit the whole structure into the whole area of the graphical window.
View in direction AXO	Sets the view point vector to (1, -1, 1). Simultaneously zooms in or out to fit the whole structure into the whole area of the graphical window.
Zoom in	Zooms in.
Zoom out	Zooms out.
Zoom by cut-out	Requires to define a cut-out for the zoom. The cut-out is then magnified in order to fit into the whole area of the graphical window.

	Once the function is started the mouse cursor changes. Position it to the upper left corner of the cut-out. Press the left mouse button and hold it down. Drag the mouse to place the cursor to the bottom right corner of the cut-out. Release the button.
Zoom all	Zoom in or out in order to fit the whole structure into the whole area of the graphical window.
Zoom all – selection	Zoom in or out in order to fit the selected entities into the whole area of the graphical window.

Window scroll-bar wheel-like buttons for adjustment of the view

Each graphical window has got three wheel-like buttons on the scroll-bar. If the scroll-bar is visible the "wheels" may be used to adjust the required view. The function of the three wheel-like buttons is:

Zoom (located on the bottom scroll-bar)	Zooms in or out.
Rotate horizontally (located on the bottom scroll-bar)	Rotates the structure around the vertical axes (i.e. vertical axis of the screen).
Rotate vertically (located on the right hand side scroll-bar)	Rotates the structure around the horizontal axes (i.e. horizontal axis of the screen).

The operation of the wheel-like buttons is simple. Just place the mouse cursor over the "wheel", press the left mouse button, hold it down and "turn the wheel" with left-right, or up-down, movement of the mouse over the pad.

Mouse controlled adjustment of the view

In addition to the standard menu and toolbar functions SCIA Engineer offers also a set of fast-access functions for the view adjustment.

Zoom in	Press [Ctrl] and [Shift] keys simultaneously and hold them down. Then press the right mouse button and hold it down as well. Move the mouse up (away from you) over the pad.
Zoom out	Press [Ctrl] and [Shift] keys simultaneously and hold them down. Then press the right mouse button and hold it down as well. Move the mouse down (towards you) over the pad.
Rotate	Press [Ctrl] key and hold it down. Then press the right mouse button and hold it down as well. Move the mouse over the pad in order to get the required view direction.
Shift	Press [Shift] key and hold it down. Then press the right mouse button and hold it down as well. Move the mouse over the pad in order to get the required position of the structure on the screen.
Zoom All	Double-click the middle-button of your mouse to invoke function Zoom All.

Rotation of view

The centre of rotation depends on initial conditions.

No entity is selected	The centre of rotation is put into the point that forms a centroid of an imaginary rectangular prism outscribed around the existing model.
Some entities are selected	The centre of rotation is put into the point that forms a centroid of an imaginary rectangular prism outscribed around the selected entities.
One node is selected	The selected node is the centre of rotation.
Clipping box is ON	The centre of rotation is put into the point that forms a centroid of the current clipping box.

Limiting the view

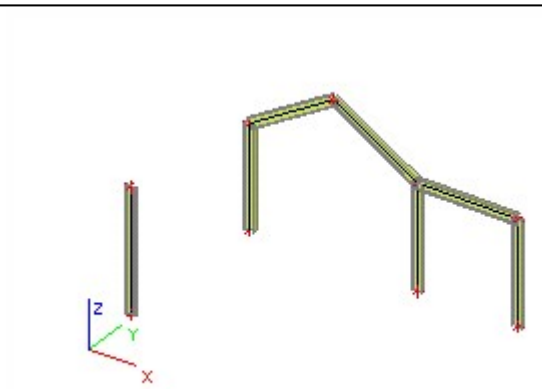
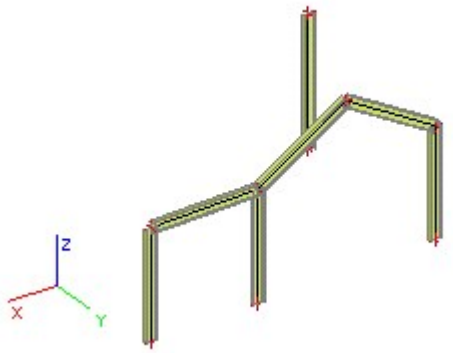
If a modelled structure is larger and complex, it may be convenient to display only a limited part of it. This "limitation" can be achieved in two different ways:

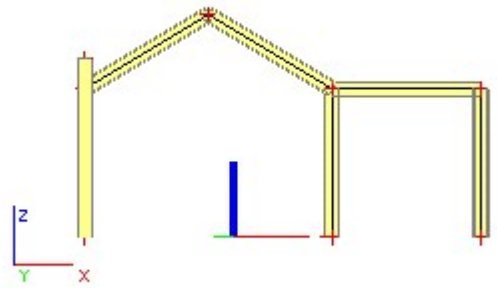
Activity or layers	The parts of the structure that are not necessary for the current operations may be hidden, in other words removed from the view. This approach is described in chapter Basic working tools > Layers or Basic working tools > Activity.
Clipping box	The view can restricted to a three-dimensional area (defined as a rectangular prism) called clipping box. If the clipping box is defined, only entities located inside it are displayed. Features of the clipping box are described in chapter Advanced working tools > Clipping box.

Adjusting the view numerically

The view direction may be specified also numerically by means of view direction vector. The vector can be defined in the View parameters dialogue on tab View. The three numbers in the table represent the X, Y, and Z components of the view direction vector.

Examples:

View direction vector	View
-1.0 1.4 -1.0	
-1.0 -1.4 -1.0	

0 0 1	

Adjusting perspective projection

By default, an orthogonal projection is used to display three-dimensional models on the screen. As an alternative, also a perspective projection can be activated.

The perspective projection can be set using:

- Either: Menu function View > View > Perspective view,
- Or: Button [Switch view to perspective mode] ( ) on toolbar View.

Special view settings

In addition to the adjustment of the viewpoint (view direction and zoom), some other properties of the view can be controlled by the user.

Wire model in manipulations

This option can be set in:

Either: Menu View > View > Wire model in manipulations,

Or: Right mouse button pop-up menu of the graphical window.

Option is ON	Only a simplified representation of the structure is displayed during the mouse controlled adjustment of the view. This option increases significantly the response of the computer during the above mentioned operation. It is therefore more than recommended for standard speed computers and other than very simple models.
Option is OFF	This option results in "fully displayed structure" during the mouse controlled adjustment of the view. This option may lead to slow response of the computer and is recommended only for very state-of-the-art and fast computers and simple models.

View parameters

Introduction to view parameters

Each entity that is defined in SCIA Engineer is not "just a geometrical shape". There is a good number of various attributes attached to each entity. The attributes may be for example material, cross-section, layer, name, construction type, etc. Each

of the attributes that is defined for a particular entity can be displayed on the screen.

What's more, some of the attributes such as for example cross-section or surface can be drawn in several ways. SCIA Engineer enables the user to control the way individual entities are displayed by means of view parameters.

These view parameters tell the program which particular attribute of each entity should be shown and which graphical representation should be used.

View parameters can be defined en block for the whole structure as unique, or they may be defined separately for individual entities. Each entity can be displayed with different view parameters.

Overview of view parameters

Available view parameters

Tab Structure | Tab Labels | Tab Model
Tab Loads | Tab View | Tab Miscellaneous

Note: In addition to these general view parameters, there are a few specialised tabs with view parameters for a particular advanced module, e.g. Steel code check, etc. These tabs are not shown until such a module is initialised.

Note : The following list contains the available view parameters. It should be noted that not all of them are always offered in the Setup dialogue. The Setup dialogue offers only those parameters for which the appropriate entity type has been already defined. E.g. until you define at least one support in your model, view parameters for supports are not shown in the dialogue.

Tab Structure

Tab Structure > Group Service

Display on opening the service

If ON, entities appropriate for the service are automatically displayed as soon as the service is opened (in the tree menu). If OFF, no change of display takes place when a service is opened.

Tab Structure > Group Structure

Style + colour

It controls the style and colour of members of the model (beams, plates, shells, etc.)

normal: settings made in Setup > Colour/Lines dialogue are used,

by layers: each member is displayed in the colour of the appropriate layer, all members assigned to the same layer are of the same colour,

by material: each member is displayed in the colour of the appropriate material, all members made of the same material are of the same colour,

by cross-section: each member is displayed in the colour of the appropriate cross-section, all members of the same cross-section are of the same colour,

according to structuraltype: each member is displayed in the colour corresponding to its structural type.

Note : If e.g. two materials, two layers, two cross-sections have assigned the same colour, than the same colour is used for members of different controlling property.

Draw member system line

The system line (midline) is drawn if this option is ON.

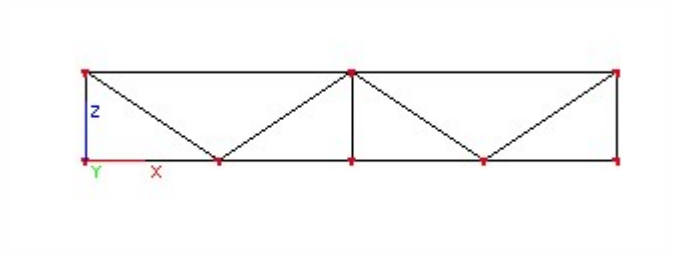
Note: If this option is OFF and also Member surface is OFF, then the whole structure disappears from your view.

Member system line style

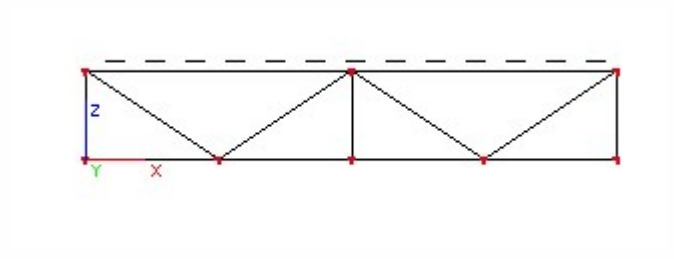
It controls the style of the member's system line (midline)

Definitions: System line is a line connecting the nodes of a member. This line is what you define when you input a new member. Fine elements are also generated on this system line. Reference line coincides with the system line if no eccentricity of a member is defined. If eccentricity is defined, the reference line is the centroidal axis of the member. Even if eccentricity is defined, the finite elements are generated on the system line (and the defined eccentricity is used in the relevant formulas of the finite elements). Bar is a highlighted system line. However, the bar is not drawn from the node to the node. It just indicates the member and leaves some space around the node for further information to be displayed.

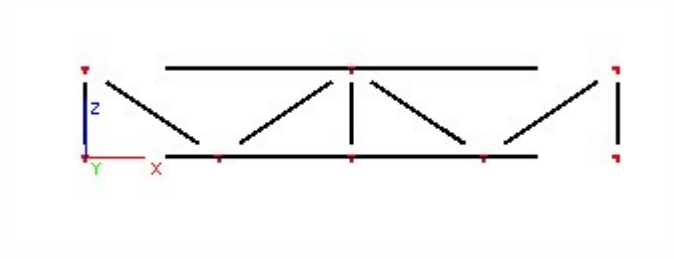
system line: the system line of members is drawn.



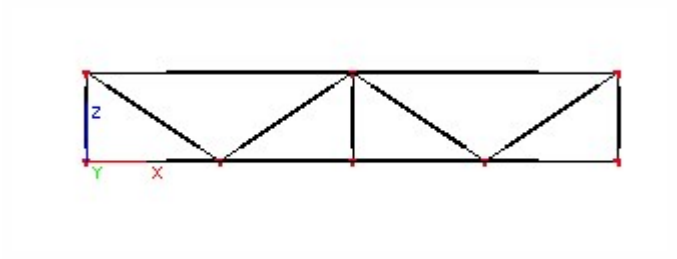
system line + reference line: system line (solid) and possibly reference line (dashed) is displayed



bar: highlighted "system line" is drawn



system line + bar: the system line is displayed and it is highlighted with the bar



Model type

You can define different geometry parameters for the "calculation model" of your structure and for the " structural model" of your structure. The calculation model is used for the numerical analysis, the structural model can be used for drawings, detailing, attractive presentations of your project, etc. For example, you can define different eccentricities in the two models, you can define cut-offs at ends of 1D members in the structural model, etc. This parameter tells the program which model you want to see on the screen.

analysismodel: the parameters relating to the calculation model are used to display the structure

structural model: the parameters relating to the structural model are used to display the structure

Example: When you open the property table of a member, the calculation-model-parameters are in the top part of the table. The structural -model-parameters are grouped lower in the table under heading structuralmodel.

	Name	B8
	Type	truss chord (95)
	CrossSection	
	Alpha [deg]	0,00
	Member system-line at	center
	ey [mm]	0
	ez [mm]	0
	LCS	standard
	LCS Rotation [deg]	0,00
	FEM type	standard
	Buckling and relative ...	Default
	Layer	Standard
⊞	**Geometry**	
⊟	CAD model	
	Priority definition	according to member
	Priority value	95
	Perp. alignment	default
	Eccentricity def.	whole member
	Eccentricity y [mm]	0
	Eccentricity z [mm]	500
	End-cuts	Manual
	begin x-offset [mm]	500,0
	begin Rz [deg]	0,00
	begin Ry [deg]	0,00
	end x-offset [mm]	500,0
	end Rz [deg]	0,00
	end Ry [deg]	0,00
⊞	**Nodes**	

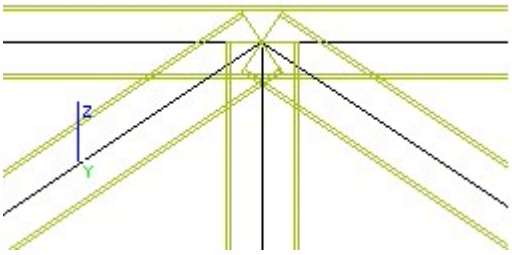
Member surface

It defines whether the surface of members should be displayed.

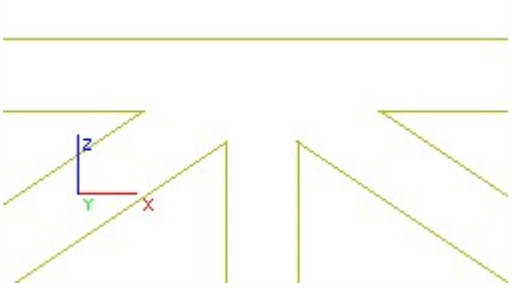
Rendering

specifies the style the surface of members is displayed

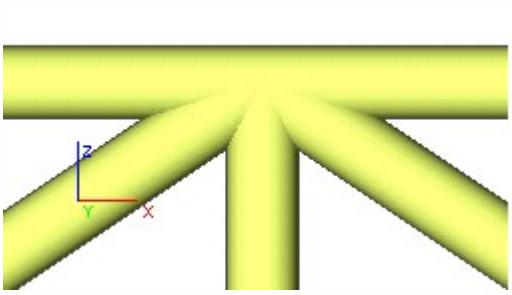
wired: only the wired scheme of the surface is displayed



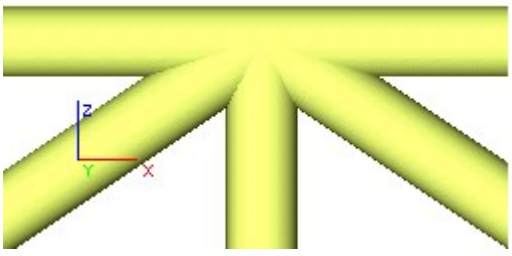
hidden lines: the real surface is calculated and those surface lines that are hidden from the view are not drawn



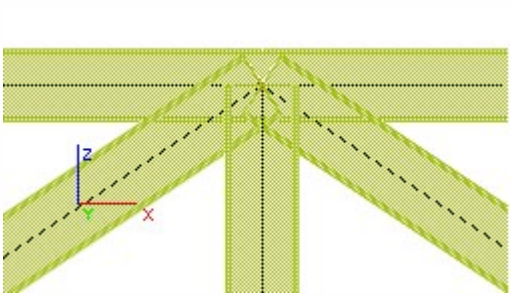
rendered with edges: the real rendered view with outlined edges is displayed



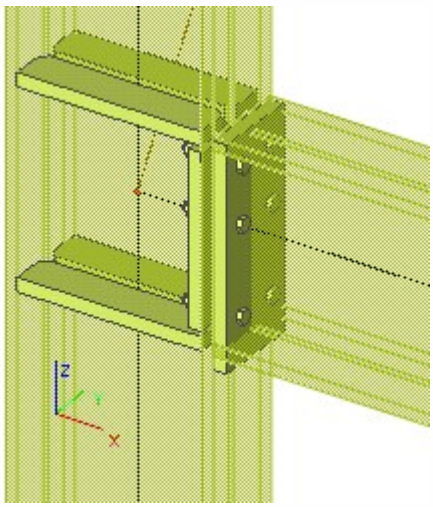
rendered: the real rendered view is displayed



transparent: the surface is filled but it is transparent (this rendering style may be e.g. useful when you want to present designed steel frame connections - the structure may be transparent, the connection may be fully rendered)



Example: The picture shows the combination of transparent rendering style for 1D member and full-rendering for connection.



Draw cross-section

This option tells if the cross-section of a 1D member should be displayed.

Cross-section style

If the previous option is ON, this item defines the style of the displayed cross-section.

section: one section is drawn about in the middle of each 1D member. The section is 3D oriented, i.e. it is displayed AS IS in the structure and in some views may not be clearly recognisable.

in screen plane: one section is drawn about in the middle of each 1D member. The section is transformed into the screen plane so that it is clearly recognisable in all view of the structure.

longitudinal XZ: a short part of XZ projection of the 1D member surface is drawn. In other views than side view, the section may be hardly recognisable.

longitudinal XY: a short part of XZ projection of the 1D member surface is drawn. In other views than plan view, the section may be hardly recognisable.

Tab Structure > Group Structural node

Display

The FE nodes of the structure can be displayed or hidden. Especially in very large models, it may be convenient to hide the nodes when a picture of the whole model is to be drawn.

Mark style

Specifies the style)shape) of the node mark.

Mark size

Specifies the size of the node mark.

Tab Structure > Group Member parameters

Buckling lengths

Buckling lengths (in all directions) for individual 1D members are displayed.

Member non-linearities

If a non-linearity is assigned to a member, a symbol is displayed indicating the type of the assigned non-linearity.

FEM Type

Various FEM types can be assigned to individual members (tension only, normal 1D member) and a description indicating the selected type is displayed if this option is ON.

Tab Structure > Group Mesh

Draw mesh

The generated mesh is displayed (the mesh can be displayed only if it has been already generated).

Draw refinement

The FE mesh can be refined in manually defined area and the defined refinements are displayed if this option is ON

Note : The finite element mesh can ONLY be displayed if at one calculation has been already performed and its results are still available.

Tab Structure > Group Local axes

Nodes

Axes of local coordinate systems of individual nodes are displayed.

Members 1D

Axes of local coordinate systems of individual 1D members are displayed.

Members 2D

Axes of local coordinate systems of individual plate and shell members are displayed.

Tab Structure > Group Sections

Members 1D

Sections (i.e. sections for the evaluation of results) on 1D members are displayed.

Members 2D

Sections (i.e. sections for the evaluation of results) through plate/shell members are displayed.

Tab Structure > Group Calculation info

Display singularity

If a calculation fails, the problematic place is shown.

Tab Labels

Tab Labels > Group Beam labels

Display label

It controls the group as a whole - if ON, the selected labels are displayed, if OFF, no labels of the group are shown.

Name, Cross-section name, Cross-section type, Length, Layer, Type and priority

Individual labels correspond to individual items in the property table of a member.

Tab Labels > Group Node labels

The meaning is more or less self-explanatory.

Tab Labels > Group Slab labels

The meaning of most of the view flags is more or less self-explanatory.

Edges

Each edge of a slab has a unique number (unique within the single slab) and these edge numbers are displayed if this option is ON.

Tab Labels > Group Mesh

Display label

see above

Nodes

FE-node numbers

Elements 1D

Numbers of 1D finite elements.

Elements 2D

Numbers of 2D finite elements.

Note: The finite element mesh can ONLY be displayed if the calculation has been already performed and its results are still available or if the mesh has been generated by means of function Calculation > Generate mesh.

Tab Labels > Group Buckling lengths

Display label, Name

The meaning is more or less self-explanatory

Label

Description of the buckling length including the dimension.

Tab Labels > Group Sections

Display label, Name

The meaning is more or less self-explanatory.

Tab Labels > Group Non-linearities

Display label

The label of the defined type of non-linearity.

Tab Model

Tab Model > Group Service

Display on opening the service

If ON, entities appropriate for the service are automatically displayed as soon as the service is opened (in the tree menu). If OFF, no change of display takes place when a service is opened.

Tab Model > Group Supports

The meaning of most of the view flags is more or less self-explanatory.

Tab Model > Group Other model data

The meaning of the view flags is more or less self-explanatory.

Tab Model > Group Support labels

Displays the label of supports.

Tab Model > Group Labels of other model data

Displays the label of other model data such as hinges, cross-links, etc. This view parameter displays or hides the label for all the types of other model data at the same time. It is not possible to attach the label to e.g. one type of other model data only.

Tab Loads

Tab Loads > Group Service

Display on opening the service

If ON, entities appropriate for the service are automatically displayed as soon as the service is opened (in the tree menu). If OFF, no change of display takes place when a service is opened.

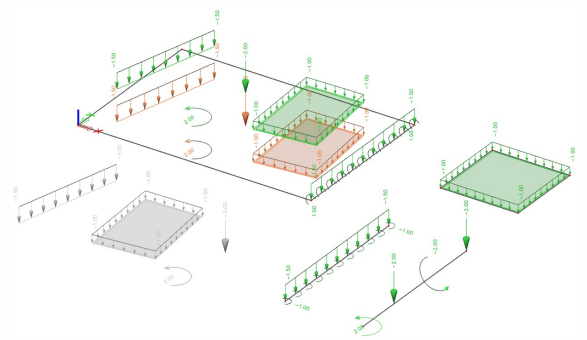
Tab Loads > Group Display loads

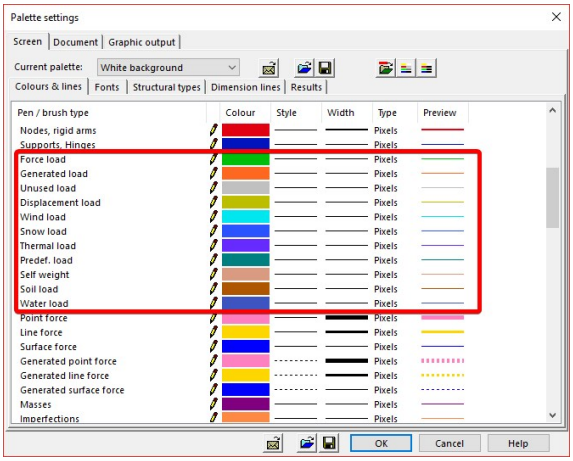
Display

If OFF, no load is displayed at all. This item controls the whole tab.

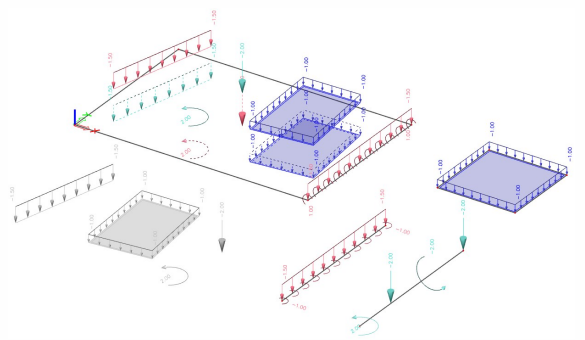
Style

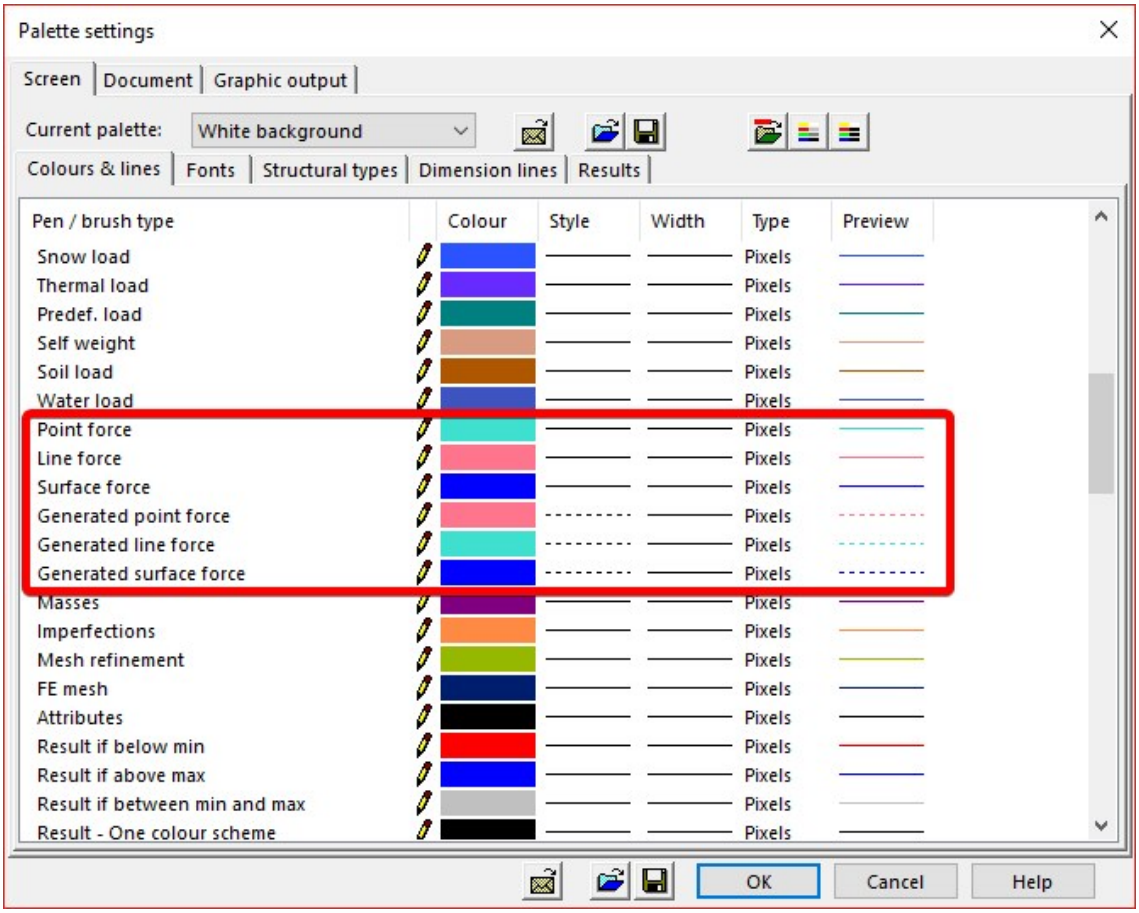
If the style *Colour by action type* is selected, the loads are displayed depending on the type of action (forces, wind, snow, imposed displacement, temperature, etc...). Line colour and style can be configured in the palette for each type of action.





If the style *Colour by load type* is selected, the loads displayed depending on the type of force (point force, line force, surface force). Line colour and style can be configured in the palette for each type of load. Please note, that generated loads have their own, separate line styles.





Please note, that the style of some types of loads/actions do not depend on the selected display style. Those exceptions are:

- Unused loads
- Displacement loads
- Thermal loads

Load case

You can select here the load case to be displayed.

Plane load generator

Displays the loading polygon of the plane load generator.

Absences

Displays the absences.

Absence

You can select here the absence group to be displayed.

Tab Loads > Groups for individual type of load

The meaning of the view flags is more or less self-explanatory.

Tab Loads > Groups Labels of loads

Display label

This item controls the display of load labels.

Name

If ON, the name of the load is attached to every loading impulse (force, moment, temperature load, etc.)

Value

Shows the "input" value of the load.

See the note below.

Total value

Shows the "real" value of the load.

See the note below.

Note: Items Value and Total value are significant for loads that are not defined directly by its force or moment impulse, but that were defined by means of a wind generator, load generator, or as a predefined load. For such loads, SCIA Engineer can display two different types of data. First, the input value (e.g. width load) can be shown, i.e. the value. Second, the calculated load per meter of length can be displayed (i.e. the total value).

Tab Loads > Groups Masses

Displays the masses.

Tab Loads > Groups Labels of loads

Display label

This item controls the display of mass labels.

Name

If ON, the name of the mass is attached to the mass symbol.

Value

Shows the size of the mass.

Tab View

Tab View > Groups Display tools

Disable tooltips

If ON, no tooltips in the graphical window are shown. I.e. no information concerning the entity under cursor is displayed. This option may reduce the response time in large projects. It also reduces the size of images in the Picture gallery.

Before this option takes effect, the screen must be regenerated.

Disable layers

If ON, no information on layers is stored in the data for the graphical window. This option may reduce the response time in large projects. It also reduces the size of images in the Picture gallery.

However, if this option is ON, it is not possible to e.g. make export of the drawing into DXF file including layers – only one "universal" layer is exported.

Before this option takes effect, the screen must be regenerated.

On the other hand, this option does not prevent you from using e.g. activity by layers. This feature is fully working regardless of this parameter.

View vector X, Y, Z

Enables the user to numerically adjust the view direction.

Clipping box

Switches the Clipping box ON/OFF.

Tab Miscellaneous

Tab Miscellaneous > Group Results diagram

Results

Displays the result diagrams on members.

Tab Miscellaneous > Group Construction stages

Display

Displays data relating to construction stages.

It controls the group as a whole - if ON, the selected labels are displayed, if OFF, no labels of the group are shown.

Already installed

Already installed members are displayed.

Currently installed

Currently installed members are displayed.

Not yet available

Members that are not yet available are displayed.

Already removed

Members that have been already removed are displayed.

Tab Miscellaneous > Group Construction stages data labels

Label local beam history

Attaches labels to the local 1D member history.

Tab Miscellaneous > Group Connection force

Display

Displays the forces in connections (in joints of several 1D members)

Tab Miscellaneous > Group Connection force labels

Display label

Displays the labels of connection forces.

It controls the group as a whole - if ON, the selected labels are displayed, if OFF, no labels of the group are shown.

Name

The name is attached to connection forces.

Tab Miscellaneous > Group Connection force

Display arrow on maximal deformation

Shows the view flags "Maximal translation" and "Maximal rotation" which are available after the calculation

Display arrow on small mesh elements

Displays the mark (arrow) on elements whose angle is smaller than 5° after the mesh generation, if any

Display arrow on mesh node

Displays the mark on the mesh nodes, if any was defined by the user using command SELMN

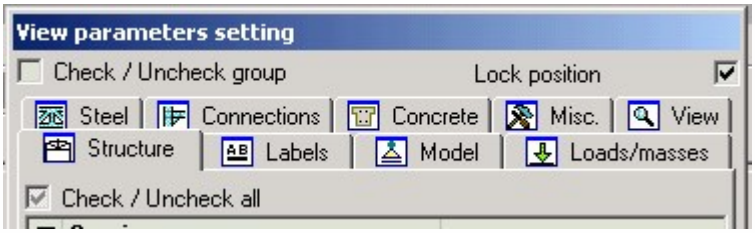
Adjusting the view parameters

In general there are three ways to adjust the view parameters:

- in the Setup dialogue,
- using the fast-access group-commands,
- using the fast-access window-buttons for certain types of entities.

Adjusting the view parameters using the Setup dialogue

The Setup dialogue provides for the adjustment of all available view parameter. In addition to the parameters themselves, the dialogue contains also other controls. They are grouped at the bottom of the dialogue.



Check / uncheck group	If the cursor is placed on the name of a group of view parameters (in any of the tabs), it is possible to use this check box to select or deselect the whole group.
Lock position	You can move the dialogue to any position on your screen and check this option. When you closed the dialogue and open it again, it is not displayed in the centre of the screen (which is the default position), but in the place you "locked" it.
Check / uncheck all	This check box can be used to select or deselect all the view parameters on the active tab.

The procedure to open the Setup dialogue

The Setup dialogue can be opened using:

- the button Fast adjustment of viewflags on whole model (or if required Fast adjustment of viewflags on selection) on the button-bar of the graphical window and selecting command Setup dialogue,
- the pop-up menu (opened by a click of the right mouse button on the area of the graphical window) and selecting the function Set view parameters for all (or if required Set view parameters for selected).

Adjusting the view parameters using the fast-access group-commands

For selected groups of entities (the groups in terms of the overview of available parameters) fast-access group-commands are available in the menu opened through the button Fast adjustment of viewflags on whole model (or if required Fast adjustment of viewflags on selection) on the button-bar of the graphical window.

Most groups from the Setup dialogue can be quickly controlled (switched ON/OFF) through these commands. Each command in the menu can be used to display or hide the entities (labels) covered by the corresponding group. The commands work like a toggle menu item: one click on them selects the group, next click deselects the group, etc.

Detailed "toggling"

The fast-access group-commands can work in two modes. The required mode can be set in the menu that opens when you click on the button Fast adjustment of viewflags on whole model (or Fast adjustment of viewflags on selection) on the button-bar of the graphical window.

Default (i.e. Detailed Off)	In this mode, whenever you turn the corresponding group OFF, the whole group becomes hidden. Whenever you toggle the group ON, the whole group is displayed.
Detailed (i.e. Detailed On)	In this mode, whenever you turn the corresponding group OFF, the whole group becomes hidden (so far it is the same as in the pervious mode). But, whenever you toggle the group ON, the only those entities are displayed that are "ticked" (selected) in the Setup dialogue. See the example below.

Note: The Detailed mode is not available until you at least once open the Setup dialogue for View parameters, make your settings there and confirm them with [OK] button.

Example

Let us take group Other model data. It can offer the following entities:

- hinges on 1D members,
- hinges on slabs,
- cross-link,
- rigid arm,
- relative node,
- internal node,
- internal edge.

Let us suppose that you use Fast adjustment of viewflags on whole model.

First, let us talk about the Default mode. If you toggle the group OFF, all the above listed entities become invisible. If you then toggle the group ON, all the above listed entities are displayed on the screen.

Now, let us move to the Detailed mode. Let us suppose that in the Setup dialogue, the following settings were made when the dialogue was edited last time:

hinges on beams	☑
hinges on slabs	☐
cross-link	☑
rigid arm	☑
relative node	☑
internal node	☐
internal edge	☐

If you toggle the group OFF, all the above listed entities become invisible. There is no difference in hiding the group. However, when you toggle the group ON, only the selected entities are shown on the screen (i.e. hinges on 1D members, cross-link, rigid arm, relative node) while the entities that are not marked in the Setup dialogue remain hidden (i.e. hinges on slabs, internal node, internal edge).

This mode is intended for such a style or phase of work when you need to check your model repeatedly and you want to see and hide in turns some part of your model.

Adjusting the view parameters using fast-access window-buttons for certain types of entities.

The button bar of the graphical window offers a set of buttons for fast displaying or hiding of certain types of entities or their labels.

Show / hide surfaces	Displays / hides the surface outline of members (1D members, slabs, shells).
Render geometry	Switches ON/OFF rendering of members.
Fast adjustment of view-flags on whole model	Offers a menu with fast-access group-commands (see above) or opens the Setup dialogue (see above).The adjustment is valid for all entities in the model.
Fast adjustment of view-flags on selection	Offers a menu with fast-access group-commands (see above) or opens the Setup dialogue (see above). The adjustment is valid for currently selected entities.
Show / hide label of nodes	Displays / hides numbers of nodes. It effects the whole model.
Show / hide label of members	Displays / hides numbers of members (1D members, slabs, shells). It effects the whole model.
Show/hide dot grid	Displays / hides the dot grid.
Select load case for display	Selects the load case that will be displayed if the view parameter for load is switched on.

Note: Please note that some view parameters always relate to the whole structure. For example, it is not possible to display reinforcement in selected 1D members only, it is either shown in the whole structure, or hidden everywhere. In order to see e.g. the mentioned

reinforcement in selected 1D members only, function Activity must be used to hide (or display in grey colour) the "unwanted" members.

Note: Not all view parameters are always offered in the Setup dialogue or in the menu with fast-access group-commands. The Setup dialogue and the menu with fast-access group-commands offer only those parameters for which the appropriate entity type has been already defined. E.g. until you define at least one support in your model, view parameters for supports are not shown.

Predefined view parameters settings

Full and complete setting of all the view parameters may be awkward and tiresome task. Especially if the user needs to repeatedly swap between two types of display.

Consequently, SCIA Engineer offers several sets of predefined settings. The predefined sets should cover most of commonly needed cases. The predefined sets can be found in menu View > Set view parameters and they are:

Model of structure		This variant displays the structure itself as is. Any supports, loads, etc. are not shown to provide for clear view of the structure.
Analysis model		This option displays the model with the focus laid on the numerical calculation. Therefore, only axes of individual 1D members are displayed and they are accompanied with supports, loads, local co-ordinate systems and other data that are important from the calculation point of view.
Structural model		This variant shows the structural model of the structure.

Other predefined views may be found in the main View menu.

Wired		This option displays the wired representation of the model. Surfaces of members are switched on.
Rendered		This option switches on the rendering of entities. Surfaces of members are switched on.
Transparent		This option displays members using transparent rendering. Surfaces of members are switched on.

Note: The number and types of predefined views may vary depending on the "skin" and mode you select for SCIA Engineer.

Drawing of input data with eccentricity

Terminology

system line

The line inputted by the user, it has nodes at its ends.

eccentricity of beam

The offset of a 1D member defined in the local coordinates of the 1D member. We have the eccentricity in y- and z-direction.

reference line

The reference line of a 1D member if obtained when the eccentricity is added to the system line. The reference line corresponds to the centroidal axis of the 1D member.

eccentricity of loads

The offset of the load (or we may say add-data in general) related to the reference line.

	Current status
	Recent versions of SCIA Engineer drew loads relatively to the system line of the corresponding 1D member. Consequently, users could not check their real position on the 1D members, which could result in the wrong interpretation of input data and also results as we have to realise that results are related to the reference line and not to the system lines.
	A related topic is the drawing of surfaces (and reference lines) of 1D members with regard to Construction Stages (CS). Cross-sections could change their shapes over time (in general the shape may differ for every CS). This influences the position of the reference lines of 1D members in individual CS and, of course, it also influences the drawing of loads and results on 1D members.

Drawing of input data with the eccentricity taken into account

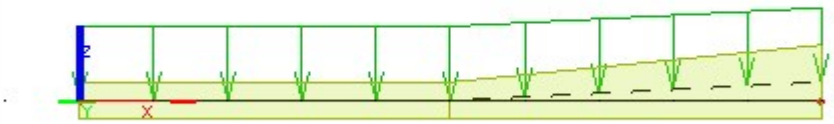
Loads

So far, the load was displayed on the system line of the 1D member. This was a correct solution only if the load was defined without any eccentricity and if the reference line (centroidal line) of the 1D member coincided with the system line (i.e. in the case of a straight 1D member the line connecting the end nodes of the 1D member). However, as soon as any eccentricity was introduced either to the 1D member or to the load, this display style became misleading.

The new solution is based on the principle that all the loads (and other displayed quantities such as hinges, and even results) are always displayed in their real position.

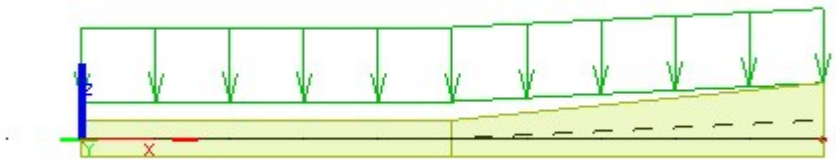
A few examples dealing with input data follows.

A 1D member with a one-side haunch subjected to a distributed load.

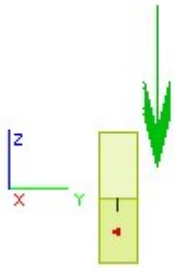


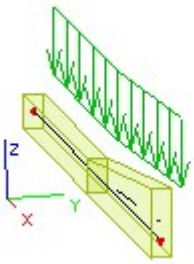
As you can see, the load follows the reference line (centroidal axis) of the 1D member.

A 1D member with a one-side haunch subjected to an eccentric distributed load.



Here, the load acts on eccentricity defined in the z-direction. In the next picture, also the eccentricity in y-direction was introduced to the load.

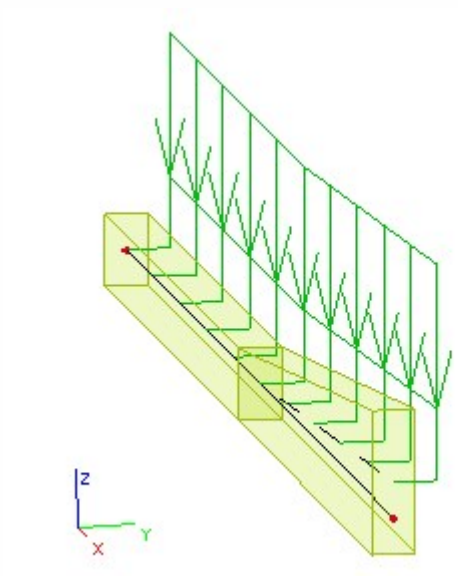




When required, also a line showing the defined eccentricity of the load can be drawn. Thus, you can more easily see what the real action of the load is. In addition, in the case of several eccentrically loaded 1D members located close to each other, it will be unambiguous which load belongs to which 1D member.

The procedure to display the "eccentricity lines"

1. Open View parameters settings dialogue.
2. Select Tab Loads/Masses.
3. Tick option Display eccentricity.
4. Confirm with [OK].

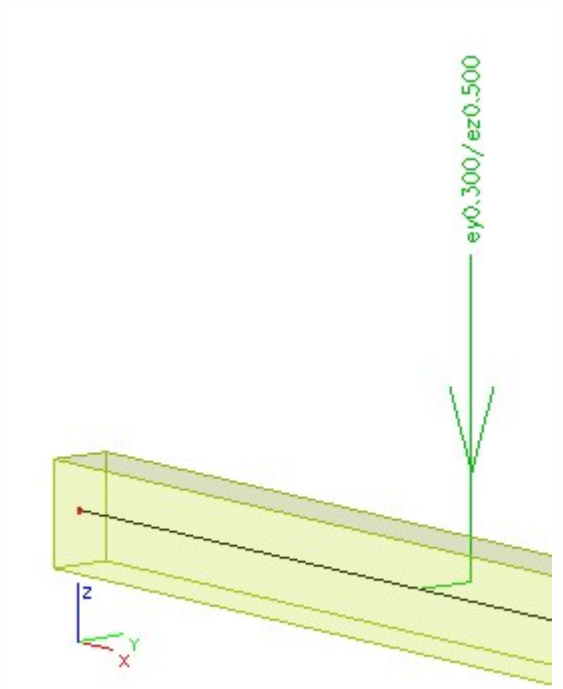


Display loads	
Display	☑
Load case	LC5
Display eccentricity	☑

In addition to the "eccentricity lines", you can also display the magnitude of the specified eccentricity.

The procedure to display the eccentricity label

1. Open View parameters settings dialogue.
2. Select Tab Loads/Masses.
3. Tick option Labels of loads > Display label and Labels of loads > Eccentricity label.
4. Confirm with [OK].

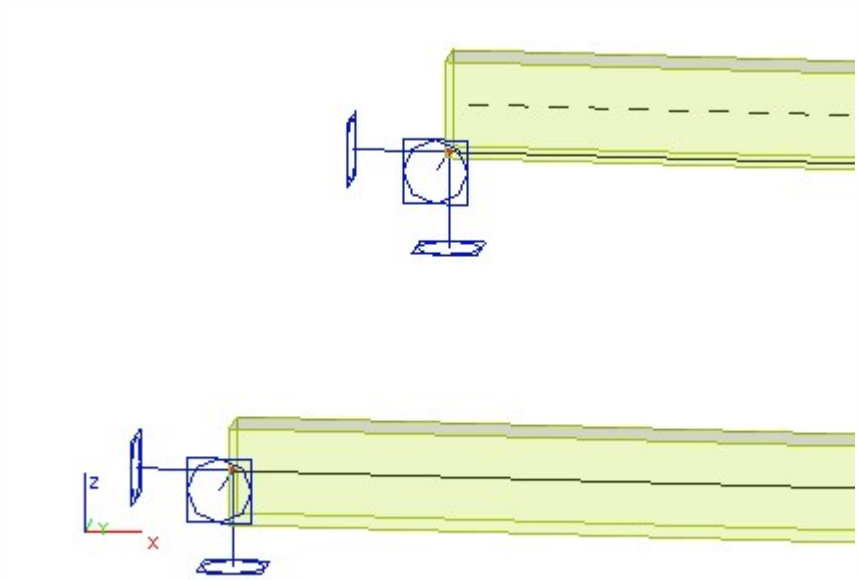


Labels of loads	
Display label	☑
Name	☐
Value	☐
Tot. value	☐
Eccentricity label	☑

Note: Loads are always drawn at their real location. View parameter Miscellaneous > Drawing style for Model+Loads > Show add data, results at has no effect on the loads.

Supports

Let us have two beams supported at the end. One of the beams is defined with the system-line in the centre line of the beam. The second beam has the system line at the bottom surface.



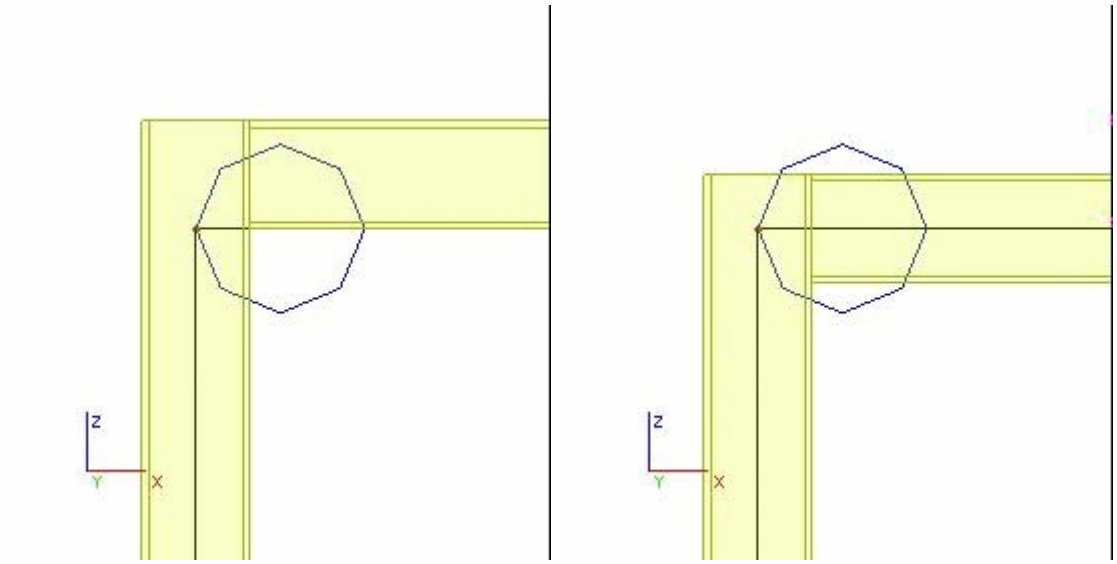
The support is displayed where its real location in the calculation model is: (i) in the first case at the centre line of the beam, (ii) in the second case at the bottom edge of the beam.

Note: Supports are allways drawn at the system line of the beam. View parameter Miscellaneous > Drawing style for Model+Loads > Show add data, results at has no effect on the supports.

Hinges

Hinges, which also belong to additional data of the SCIA Engineer model, can also take into account possible eccentricity of the 1D member at which they are defined.

Unlike loads and supports however, hinges allow the user to decide on the drawing style.



The procedure to select the display mode

1. Open View parameters settings dialogue.
2. Select Tab Misc..
3. Set the option Drawing style for Model+Loads > Show add data, results at to:
   1. Reference line in order to see the real position of the hinge (the hinge is put on a short rigid arm that is not drawn in the screen).
   2. System line in order to see the schematic position of the hinge.
4. Confirm with [OK].

Results

Note: Results are always drawn in the system line. (Despite the specification, it was not done in this version.)

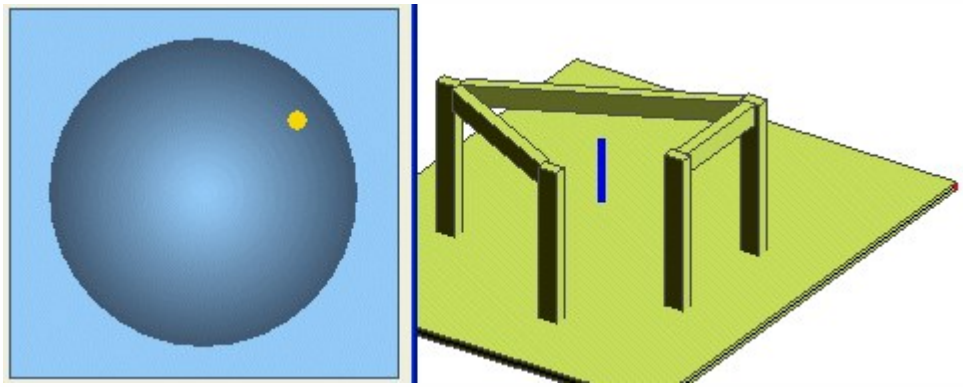
Structural model

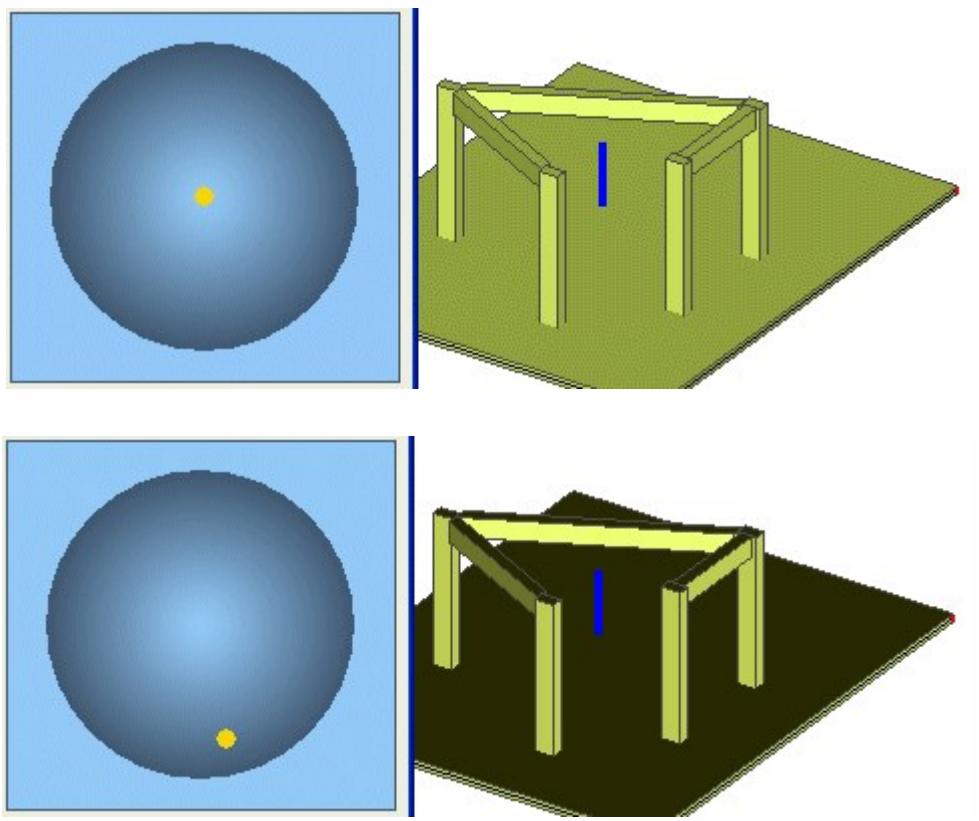
Note: The display of eccentric entities relates exclusively to the analysis model. It has no relation to the structural shape.

Lights

If rendering is switched on in View parameters, you may control the direction of the light used to illuminate the graphical screen.

The following examples give the idea of what the effect of the light direction is.





The dialogue for the adjustment of the light direction can be opened through menu function: View > Light. When the dialogue is opened, the light direction can be adjusted by a single click on the picture of the ball in the dialogue. The effect is immediately shown in the graphical window, so it is quite easy to find the required light direction. When the light is adjusted appropriately, close the dialogue.

Regeneration of view

Introduction to regeneration of view

It is a common phenomenon in CAD and similar "drawing" programs that once the drawing becomes excessive or is being edited and modified, the "current state" displayed on the screen may happen not to reflect completely the "reality". This is due to the fact that it is not possible to guarantee a flawless automatic regeneration of the view. If the automatic regeneration of the view had to be ensured, it would result in unbearably slow response of the program.

Therefore, SCIA Engineer, similarly to other graphically oriented program, offers the user the possibility to regenerate the view manually at any time when necessary.

Redrawing the active graphical window

This function redraws the graphical window if some changes affecting the display were made and the window has not been regenerated automatically.

The procedure to regenerate the contents of the graphical window

1. Press button [Redraw] () on toolbar View.
2. The contents of the active window is regenerated and redrawn.

Calculator

Calculator

Any time you enter a number into an edit box or command line, you may use the internal calculator. This calculator provides for basic operations: addition, multiplication, subtraction and division. You may use brackets, basic goniometric functions (tan, sin, cos) and it is possible to calculate powers of numbers. The calculator takes account of priorities of operators.

If you want to use the calculator to calculate the value in the input box, you must start with the equals sign (=).

As soon as you type the first character, a temporary field - "bubble" - appears just below the input box. This new field calculates the result of the input formula. If the field shows "error" than the syntax of the formula is invalid.

Valid operators and functions

=	obligatory, this character must start the formula
+	addition; e.g. 1+2
-	subtraction; e.g. 2-1
*	multiplication; e.g. 1*1
/	division; e.g. 2/1
^	power;; e.g. 2^3
()	brackets; e.g. 2*(3+3)
e	exponential notation, useful for large numbers; e.g. 1e5
sin()	sinus; e.g. sin(45)
cos()	cosine; e.g. cos(30)
tg()	tangent; e.g. tg(45)

The calculator may be used also in the situation when set of numbers is to be input, e.g. when point coordinates are defined. In such a case any of the coordinates can be input as formula, and any of the coordinates can be input as number.

Example 1

The input of point

1;=2*(3+2);sin(45)*5

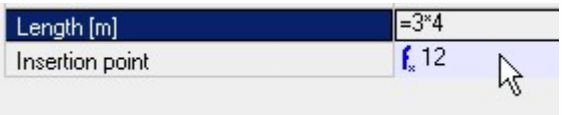
is "decoded" as:

X = 1

Y = 2*(3+2) = 10

Z = sin(45)*5 = 3,5355339

Example 2



Cleaner

Removing unnecessary data from the project

When you work for some time on a project, it may happen that some data you input at the beginning are not relevant any more. For example, you may be force to change the material grades, you decide on replacing certain types of cross-sections, you may opt for another type of reinforcement, the load the structure is subjected to may have been altered, etc.

In order to keep the project (especially a large one) "free of ballast", it is convenient to remove all the entities that are no longer necessary.

Sometimes, it may happen that you must completely abandon the solution you chosen and you must start from scratch – sometimes not exactly, but almost.

For all these situations, you may use tool called Cleaner. It is a simple tool that enables you to select which particular data should be removed from your project.

There are several groups defined within the Cleaner dialogue with each of them containing usually several items. The number and type of the items depends on the data that were defined in the project. The Cleaner dialogue offers only the data that really exist in the project.

Below, you will find an example of the groups and individual items in them (the complete list would be too long and it would contain all possible entities that can be defined in SCIA Engineer).

General	This group allows you to delete e.g. results, temporary solver data, mesh, etc.
Document	Here you can clear the document.
Model	This group allows you to remove e.g. supports, hinges, etc.
Loads	It is possible to remove all the applied loads or just the selected types of loads.
Sets	Defined sets such as load cases, combinations, bore holes, stressing beds, etc. can be deleted here.
Unused library entities	The unused items in specified libraries can be removed from the project to make the project file more compact.

Coordinate information

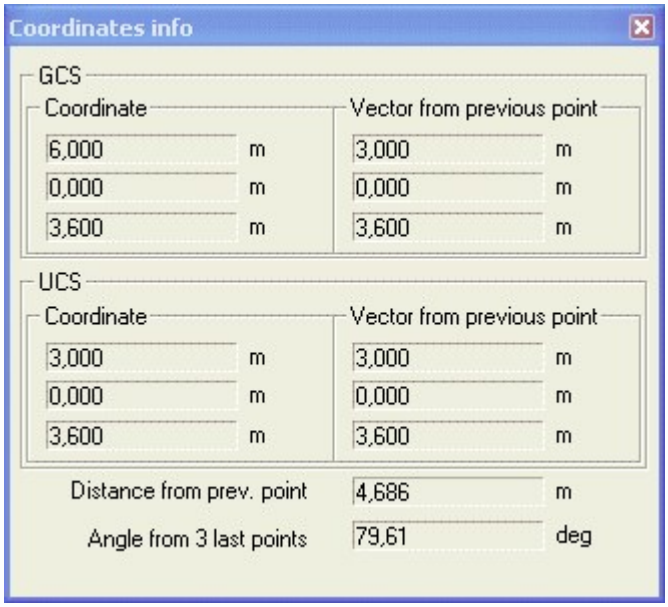
Information about coordinates of selected points

Function Coordinates info enables the user to review the coordinates of selected points in the model and to measure the distance between two defined points.

The function is easy to use. Once it is started, the user just selects (clicks) the required points (nodes) in the model and a simple dialogue shows the information:

- coordinates of the selected point in the global coordinate system,
- vector (in the global coordinate system) from the previous point to the last point,
- coordinates of the selected point in the current user coordinate system,
- vector (in the current user coordinate system) from the previous point to the last point,

- distance between last two selected points,
- angle defined by the last three selected points.



The procedure to obtain the information about coordinates

1. Start function Coordinates info:
   1. either through menu function: Tools > Coordinates info,
   2. or through tree menu function: Tools > Coordinates info,
   3. or through icon  on toolbar Tools.
2. The information dialogue is opened on the screen.
3. If necessary, position the dialogue so that it does not it does not hinder you.
4. Select (click) the points you are interested in.
5. When ready, use the close button in the top right corner to close the information dialogue.

Attributes

Attributes

Introduction

The attribute is an object in SCIA Engineer. It can display some additional information about members. It can be edited in the Property window in the same way as other properties.

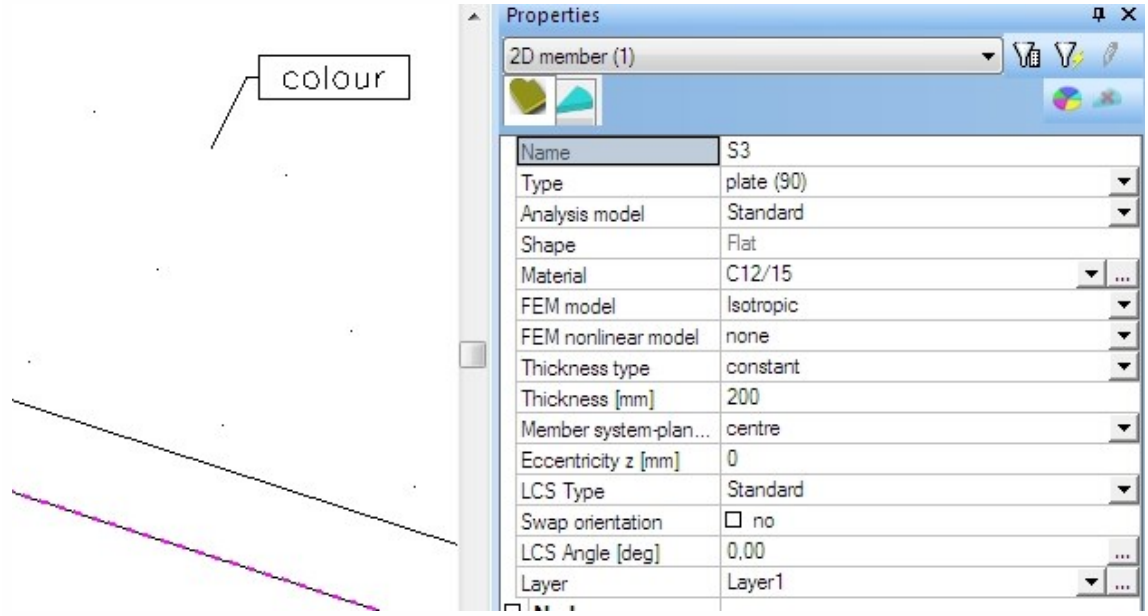
We have two different kinds of attributes.

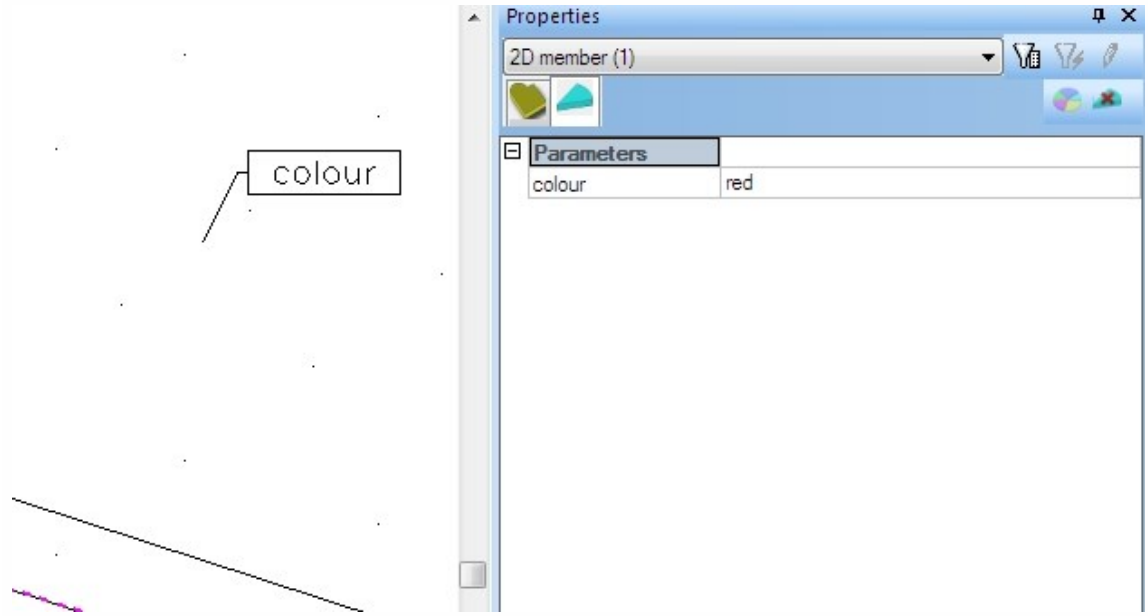
• User defined attributes

• Previous member data – now converted to attributes

When the user selects a member with an attribute, then the attribute is always displayed in the properties. It is not possible to select a member without attributes when any attributes are connected to the member.

There is a second tab in the properties when the member contains an attribute. This second tab displays the properties of the attribute.



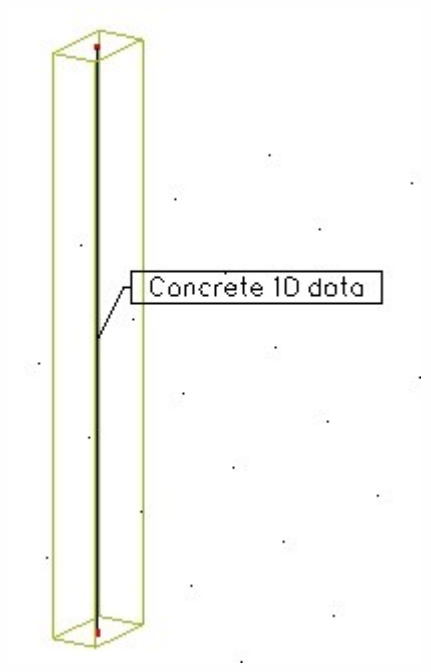


The user has also the possibility to select each attribute separately by selecting it by its label.

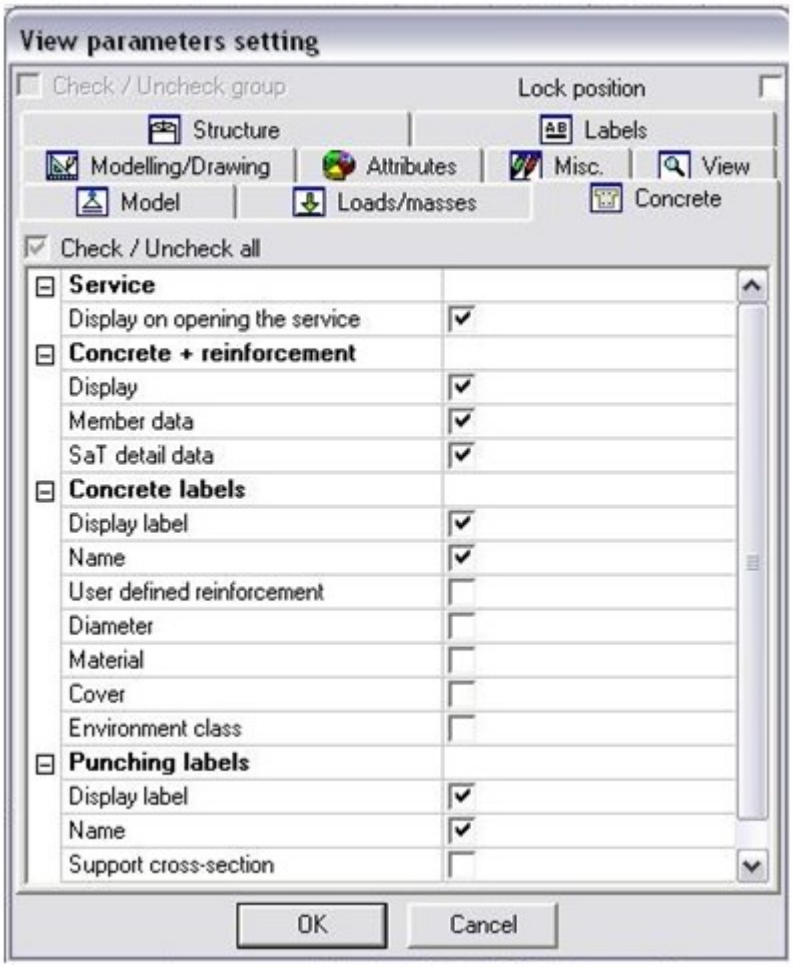
Previously Member Data - now Attributes

There are Member data in SCIA Engineer which can be considered as attributes already – Concrete member data, Steel member data, Fire resistance and more.

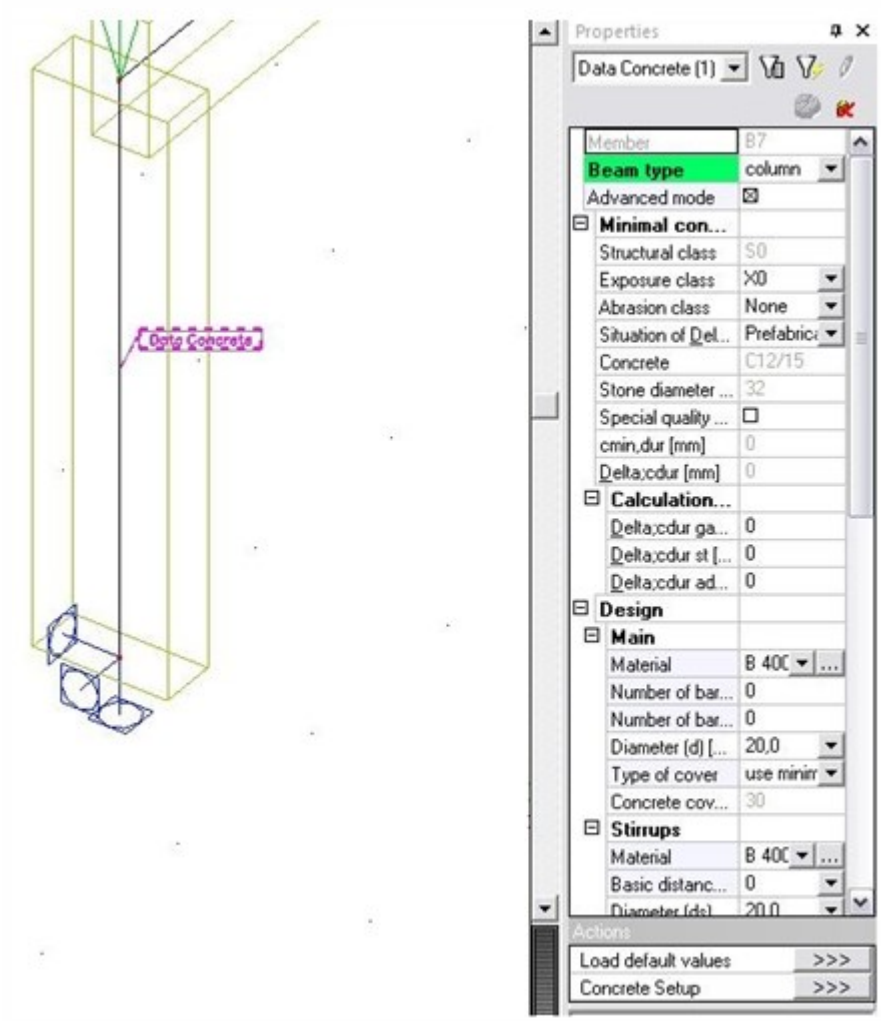
Those Member data which do not have special drawings of their flags have been already converted to attributes. Their behavior is the same as for attributes.



View flags for previous Member data stay the same – see example of concrete Member data:



The attributes that were Member data previously have the same dialogues and same properties. Only their displaying is different from now on.

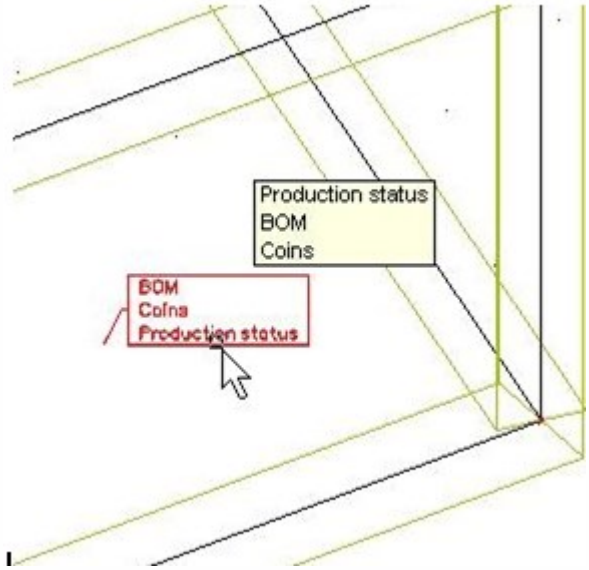


Example of converted Member data: Concrete member data, Steel member data, AMRD, Voided slabs, Fire resistance, Punching in node (Punching data).

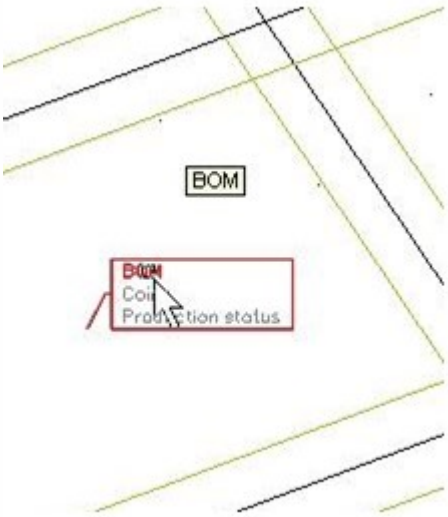
If the converted Member data are inserted to the project then they can be inserted to a new member in the same way as it is possible for user defined attributes.

Displaying of Attributes

Attributes are displayed in tooltips and in Labels.

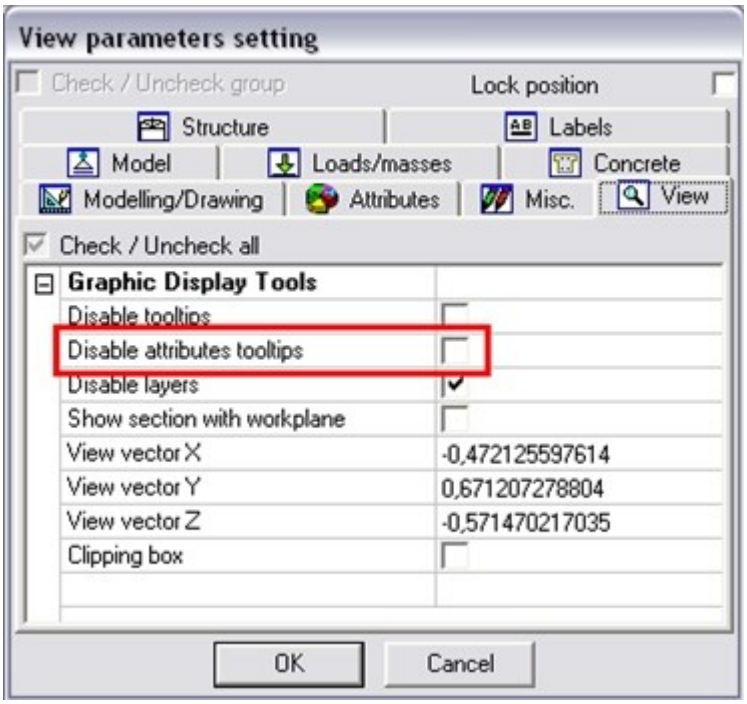


Attributes can be selected one by one or all together. One by one selection is by clicking the mouse on their description in label, All together are selected by clicking on the label itself.



The only exception is when the Attributes are not displayed – i.e. attributes have been inserted on non-geometrical objects (loads, foundation blocks, openings …).

Displaying of tooltips can be disabled in View parameters settings.

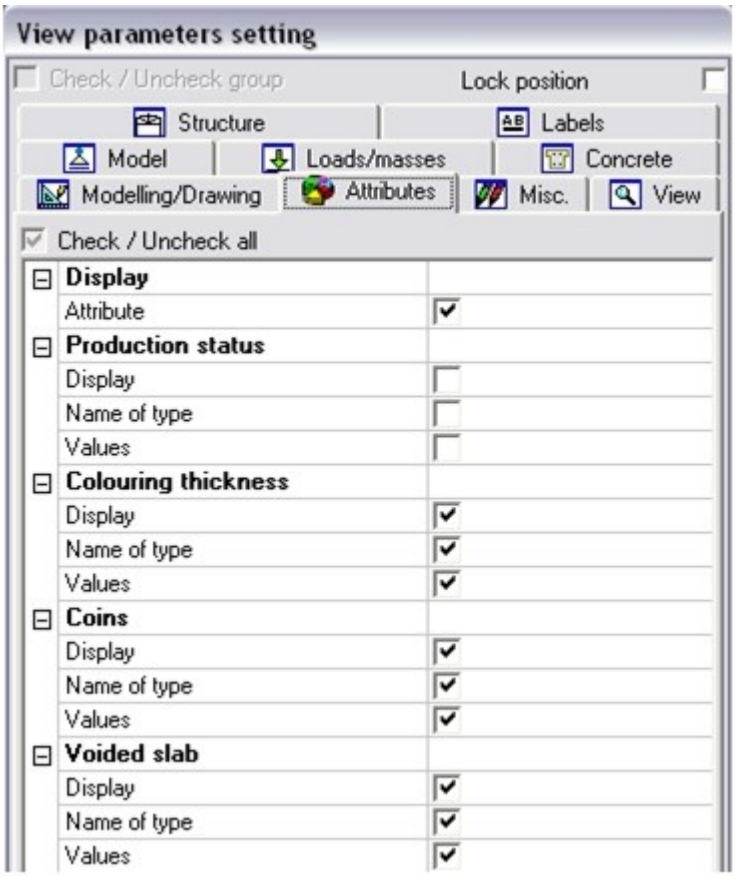


When the project contains at least one attribute, then a new tab in the view flags appears – Attributes. Here the user can switch the user-defined attributes on or off.

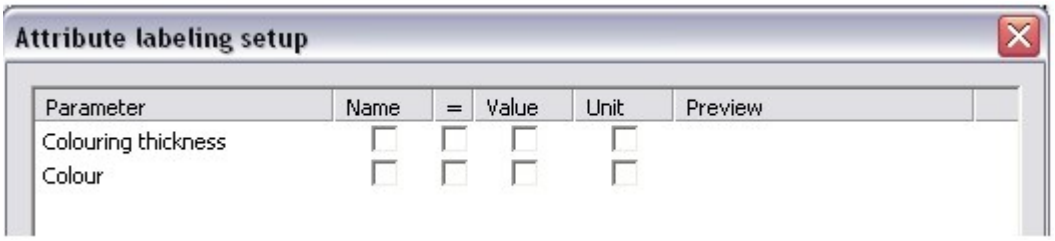
Display
This checkbox tells whether the attribute should be displayed in the whole window

Name of type
Displays the Name of the attribute (in the Attribute definition – the row Type name)

Values
Are set in the Attribute definition – Label setup



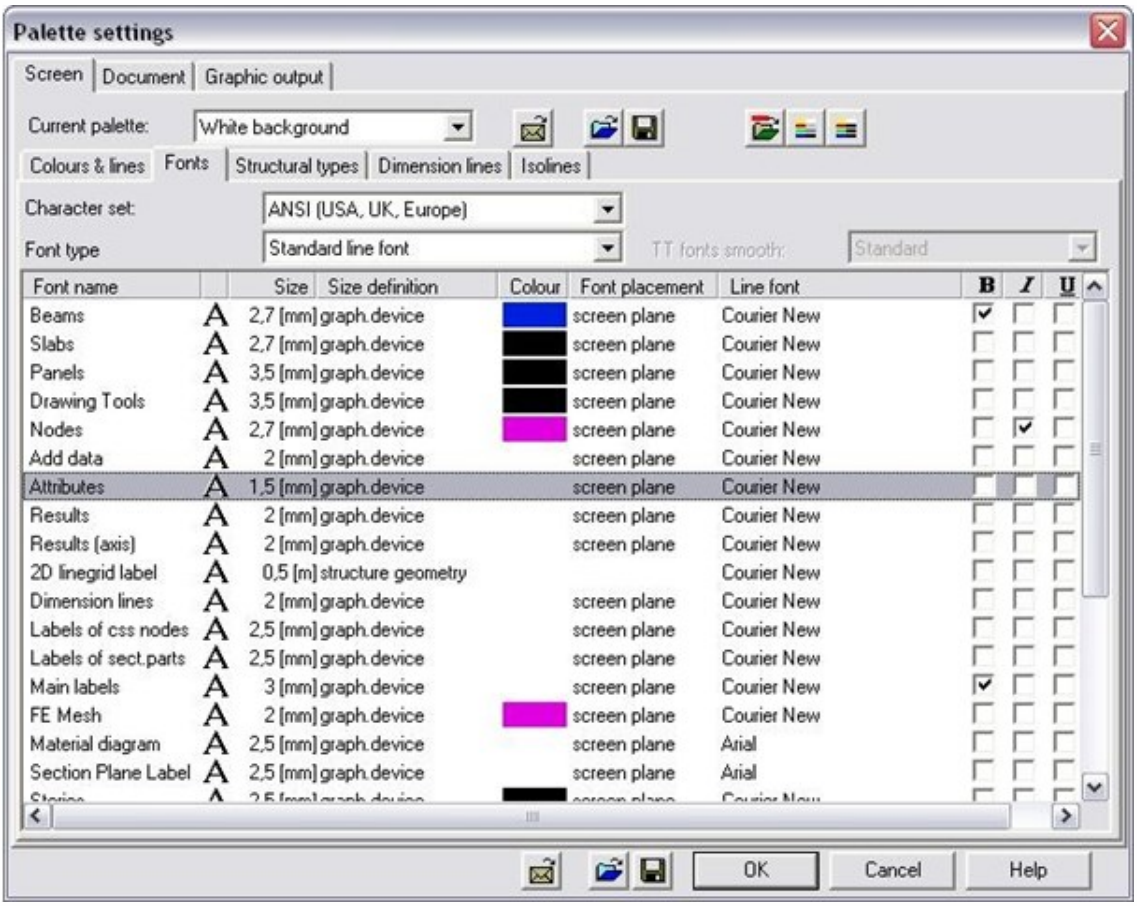
Label setup in the attribute definition



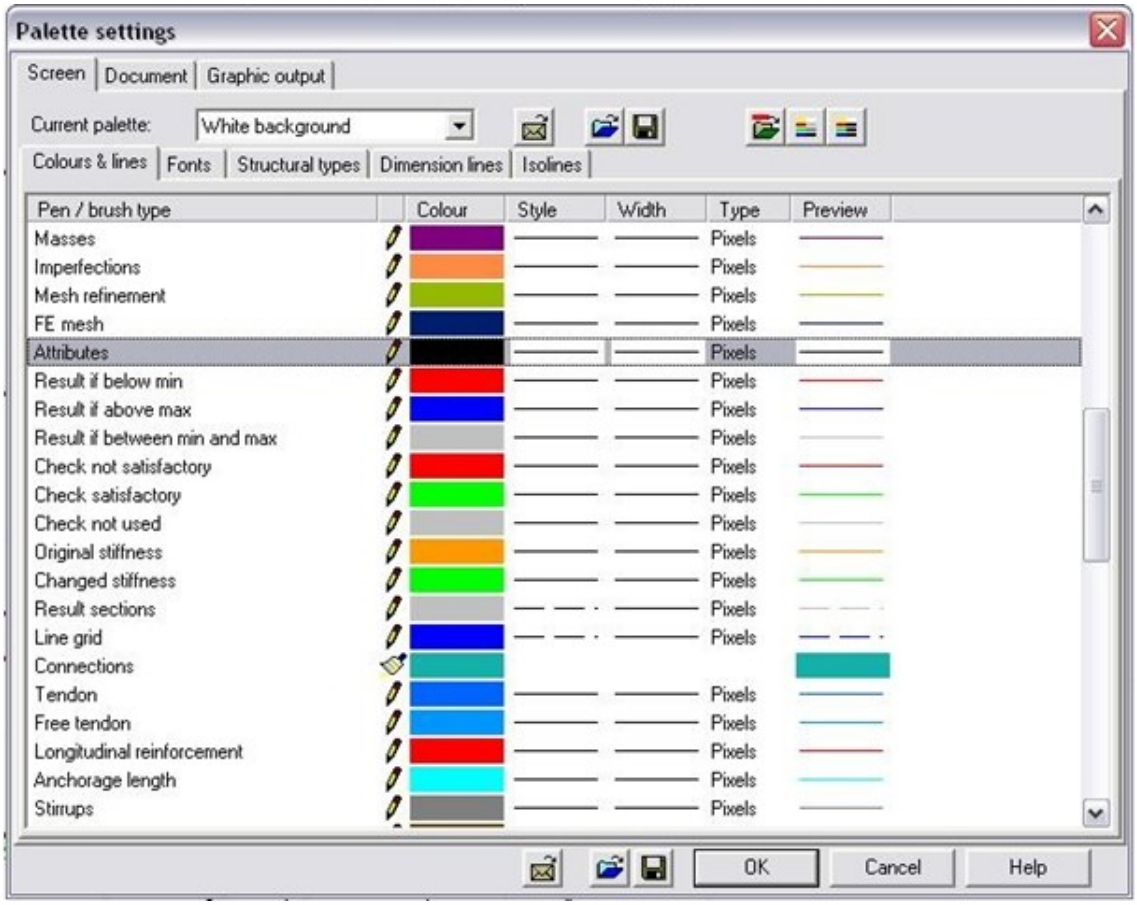
Here the user can define which values will be used for creating of label with Values.

Text size, fonts, colour, ...

Change of the size of the Label for Attributes can be set in dialogue Setup->Fonts->Attributes.



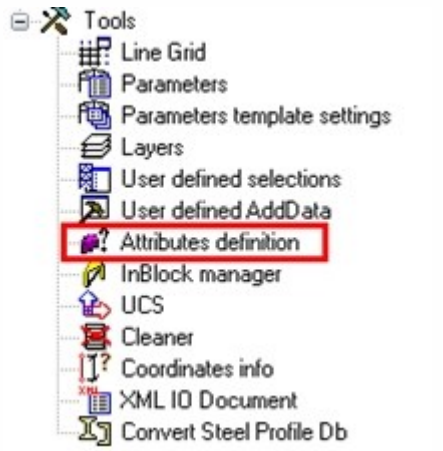
Change of the colour of the Label for Attributes can be set in dialogue Setup->Colours/Lines->Attributes.



User defined Attributes

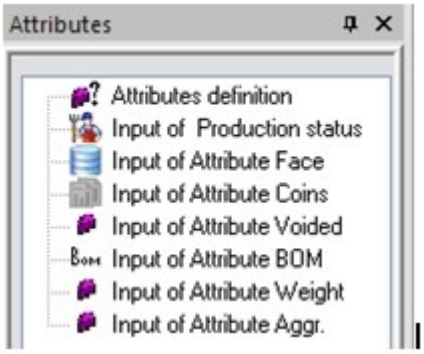
User defined attributes must first be defined and then they can be inserted to the project. The definition dialogue comes out of My Add Data dialogue. The user defines the String database, parameters, picture and icon. The pre-defined attributes appear in the new service menu – Attributes. The attribute can be inserted to the project from here .

The attribute definition:



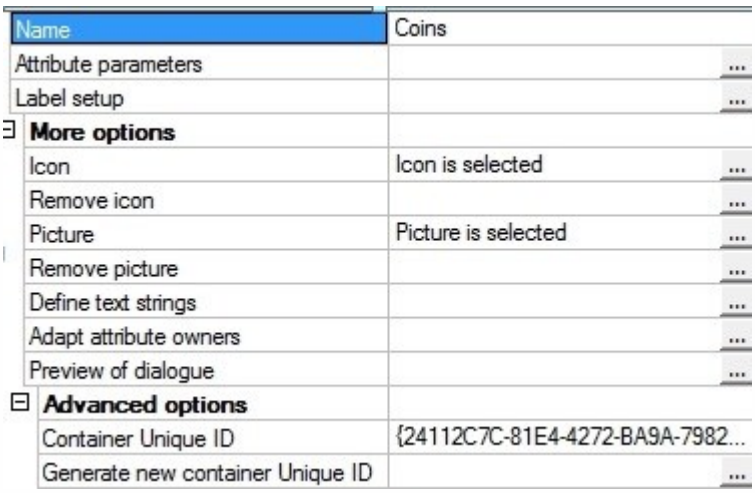
When at least one attribute is defined a new service in main tree appears:

In this service also the Attributes Definition dialogue is available.



The name of the service for a new attribute is set in the dialogue as Service name (example – Input of Attribute …) and to this name from string database is connected the name of the attribute in row Name in the Attribute definition (example – coins, face…).

It is much quicker to use an Attribute wizard for the definition.



Name
The user sets a name of the attribute

Attribute parameters
Here the user specifies parameters for Attribute

Label setup
The user can define which values from parameters will be displayed in the label of the attribute

Icon
The user can select an icon which will be used for this attribute in the service menu

Remove icon
Removes the icon

Picture
This item inserts a picture which will be displayed in a small window at the bottom of the dialogue

Remove picture
Removes the picture

Define text strings
All the text strings of the user interface are saved into a string database, this allows for an easy translation of the interface to different languages

Adapt attributes owners
In the list of all available owners, the user can select owners for his attribute. If he selects nothing, then the owner can be every member with geometry.

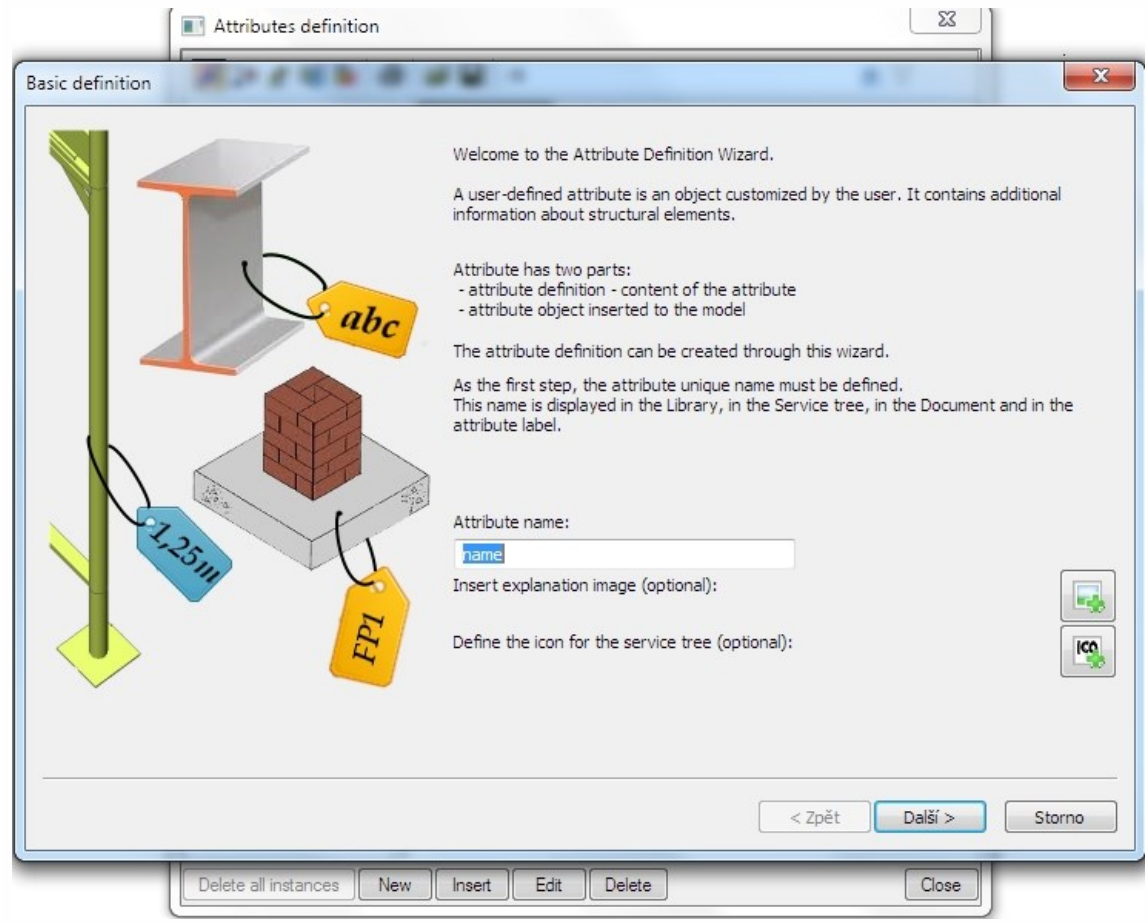
Preview of dialogue
Preview of the attribute dialogue

Container Unique ID
Tells which container is used for this attribute

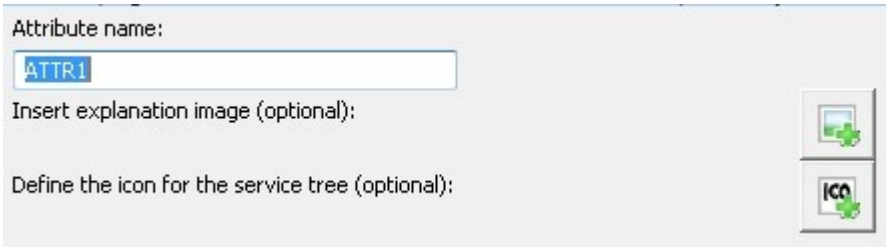
Generate new container Unique ID
The user can change the container for this attribute

User attribute wizard (version 2011)

The wizard is a new tool for creating User-defined attribute. The process is divided to the single steps which are completed by the button “Next”. The wizard can be used also for editing. Attribute parameters are stored on the second tab. User-attribute wizard is automatically launched when user open empty Attribute definition library.



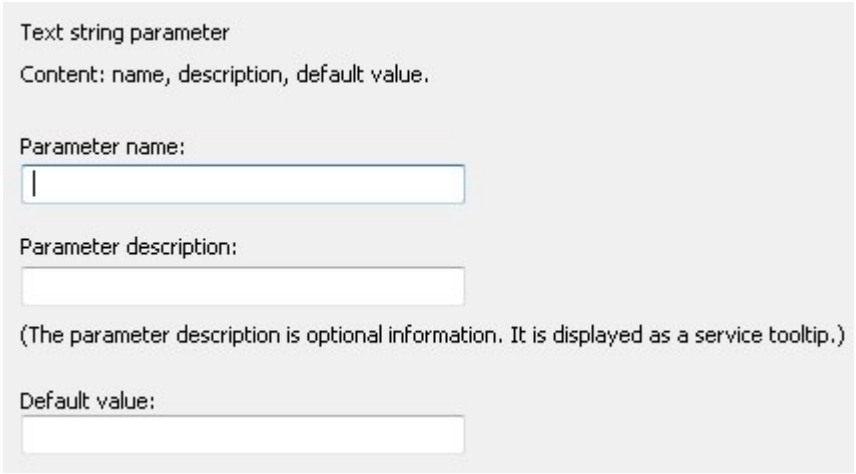
The first page defines the name of the attribute and the explanatory icon and picture.



Next step defines which parameter will be added to the attribute. Buttons for String, Number, Checkbox and Combobox are available. Each button switches the tab to the parameter specification.

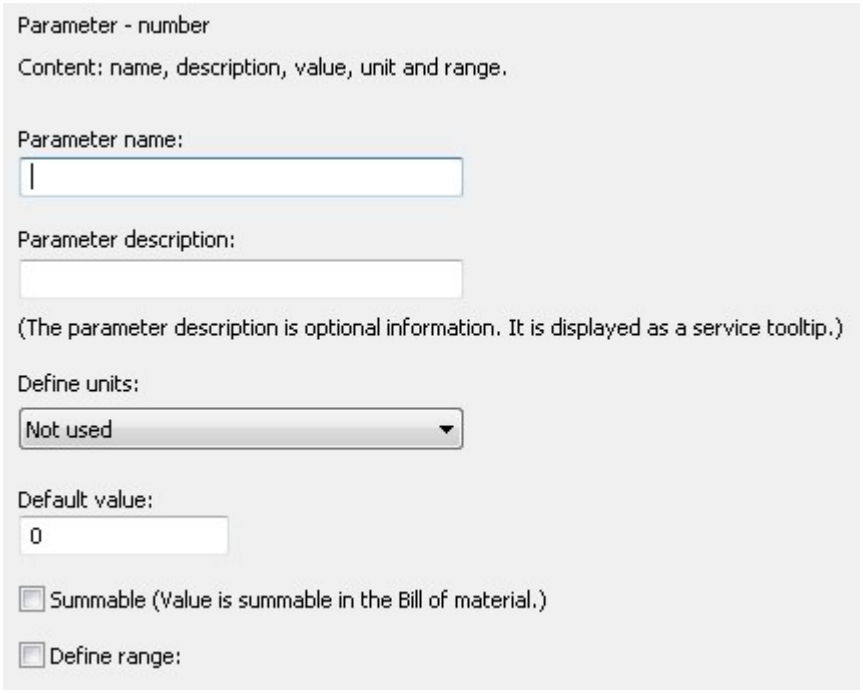
String

Text string is the simplest value for the attribute parameter.

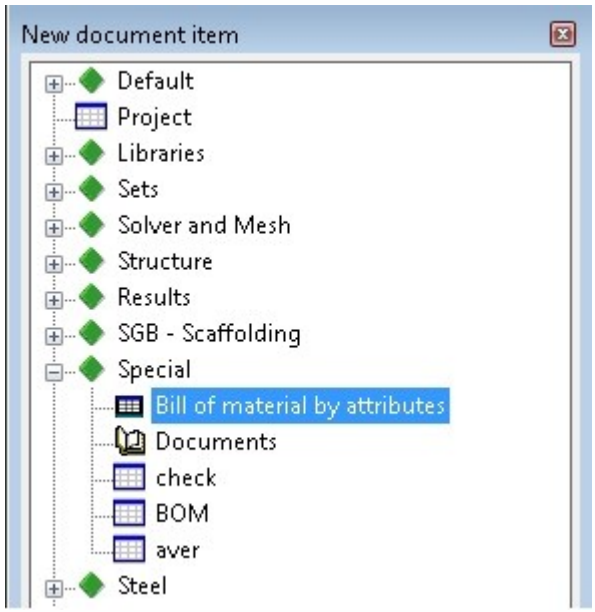


User set name, description and a default text string.

Number

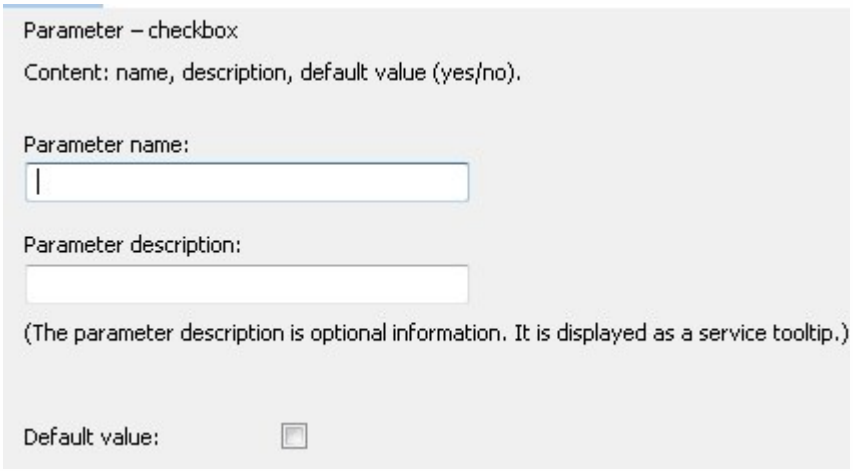


The number is defined by the name, description and default value. Additional information is unit, range and summable option. Summable option is a functionality of user-defined attribute. This checkbox add a column in the document table where this number displays its count. The table is available in the group Special in the New document item.



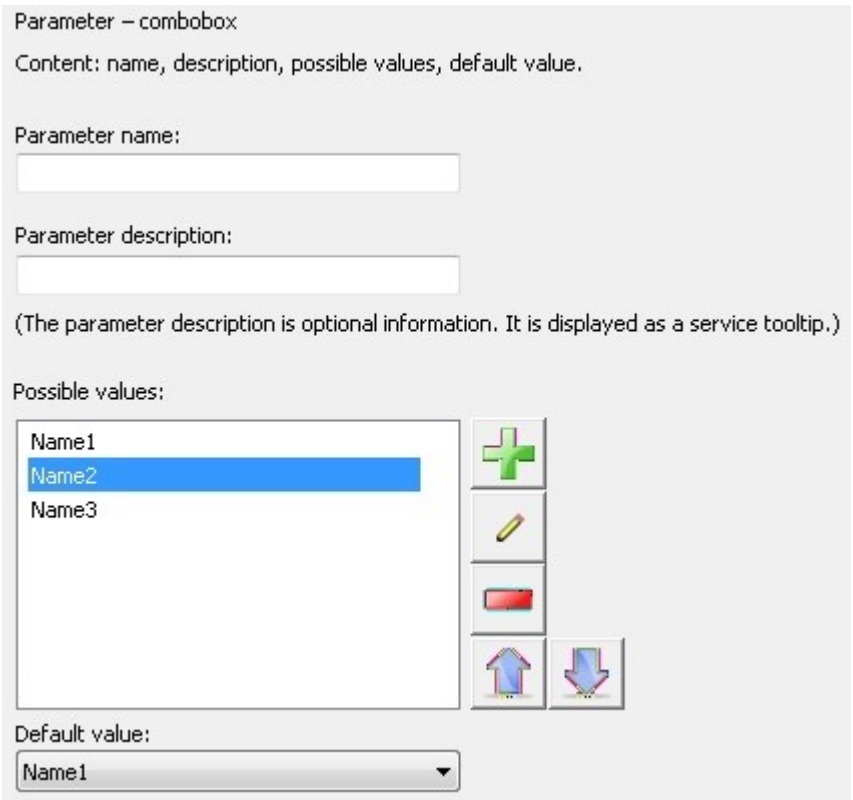
Type Name	Name	BOM	Total BOM	number-checkbox	Count
Bill BOM	BOM	2	4	✓	2
Bill BOM	BOM	3	3	×	1
Bill BOM	Total		7		3

Checkbox



The checkbox is defined by the name, description and default value – yes/no.

Combobox

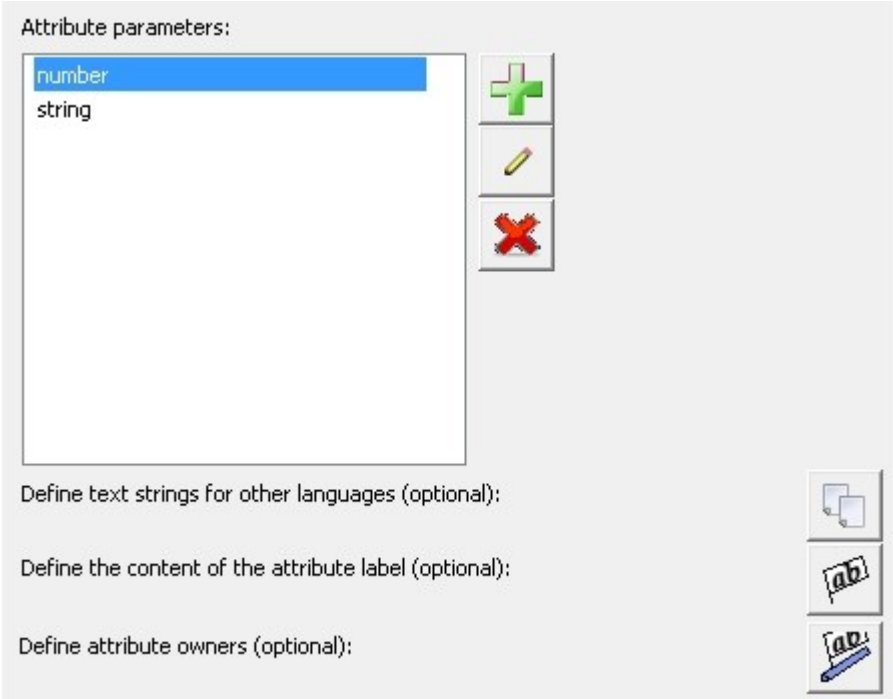


The combobox is defined by the name, description, the list of values and default value from the list.

Possible values are editable by the text editbox – it can be added, renamed, deleted and moved.

Definition of the owner, language and label

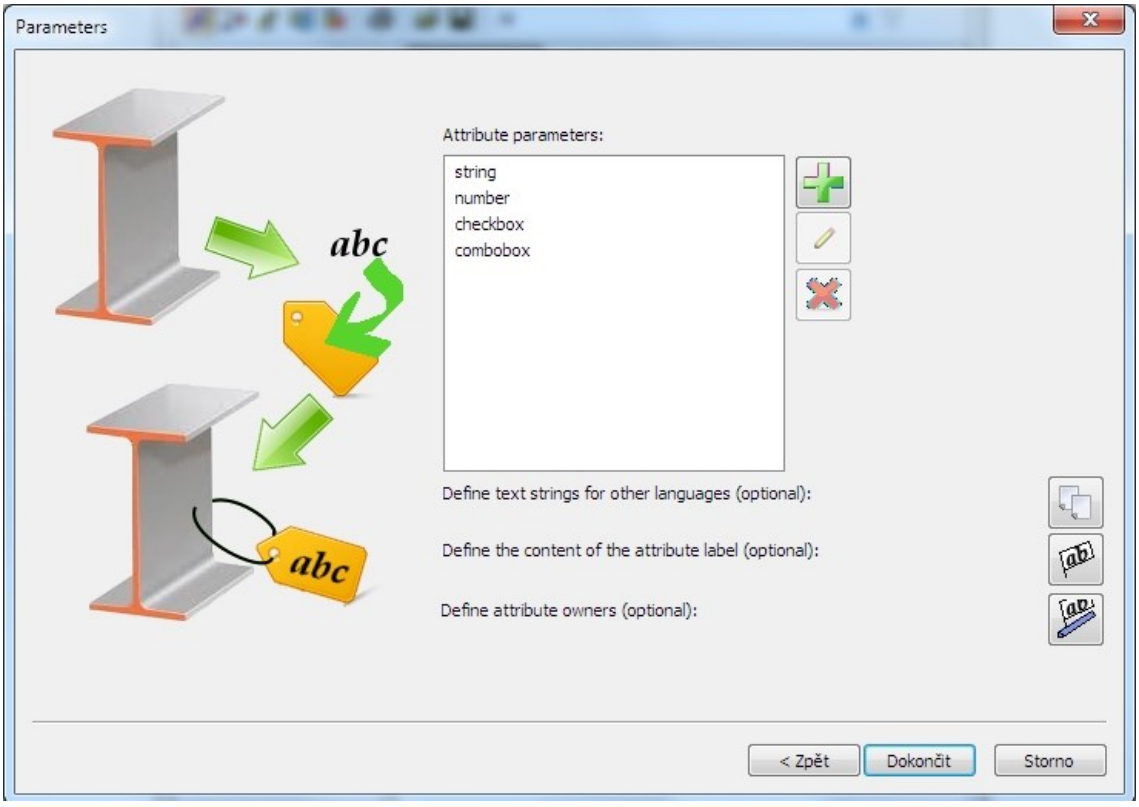
Definition of the owner, language and label is in the last step of the attribute wizard.



Each attribute parameter can be edited, deleted or added a new one. Language strings for localization, attribute label options and attribute owner can be edited for the whole attribute definition.

Editing by wizard

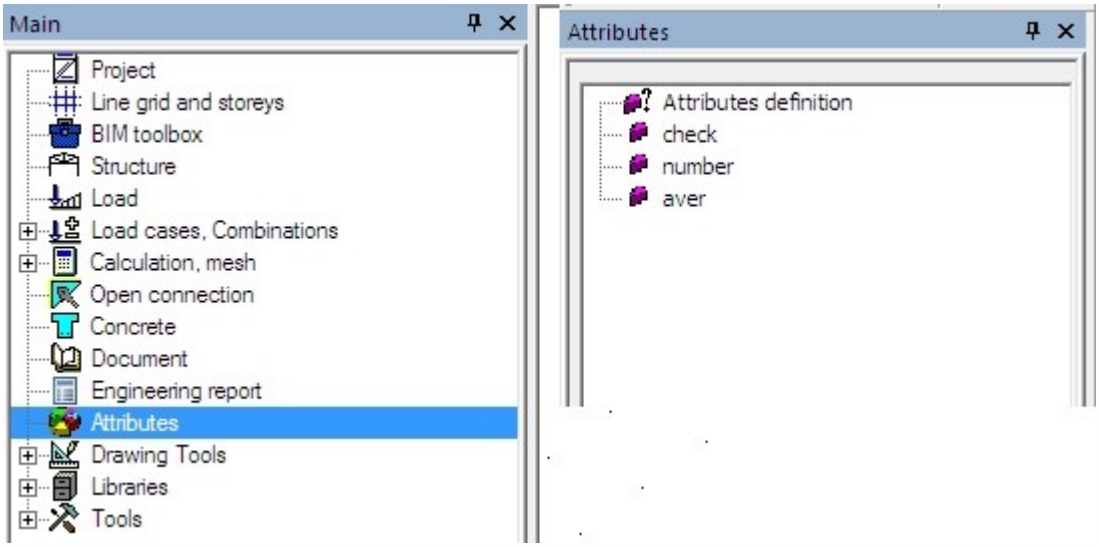
User-defined attribute can be edited by the wizard. Parameters are displayed on the second tab in the text editbox.



Note: The first parameter in the attribute define grouping in the document table. When there are 3 attributes inserted in the project and each of them has a different value for the first parameter then the document table displays 3 groups for this attribute definition.

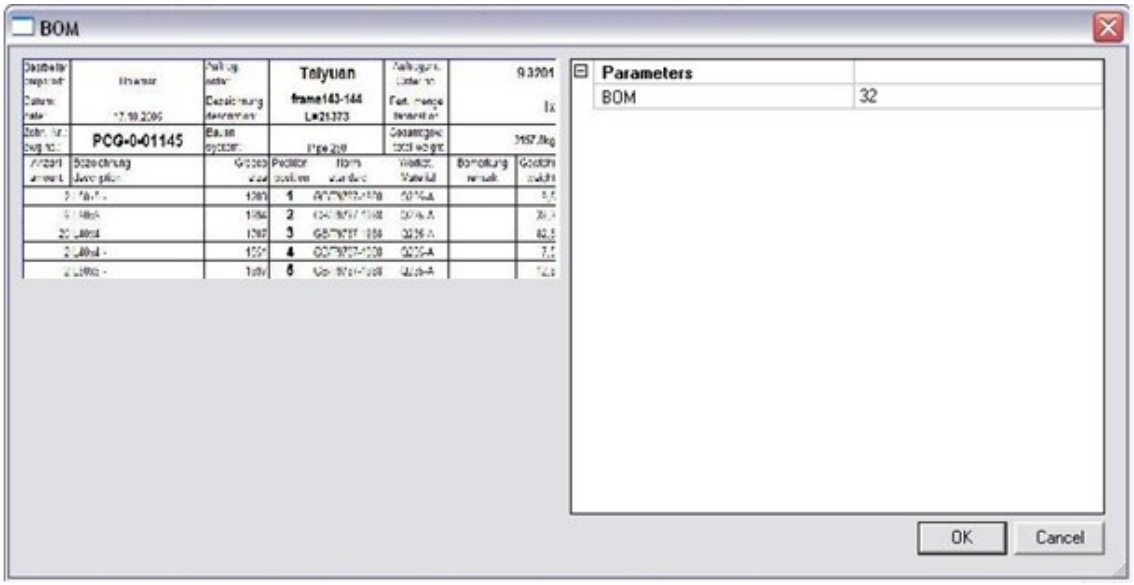
Attribute - service and inserting to the project

When the first attribute definition is created, the new service with attribute commands is displayed. Each definition has its own command and it can be inserted as attribute instance on the member.

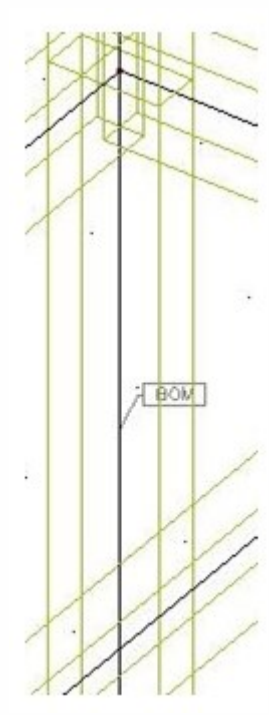


Inserting of user defined attributes

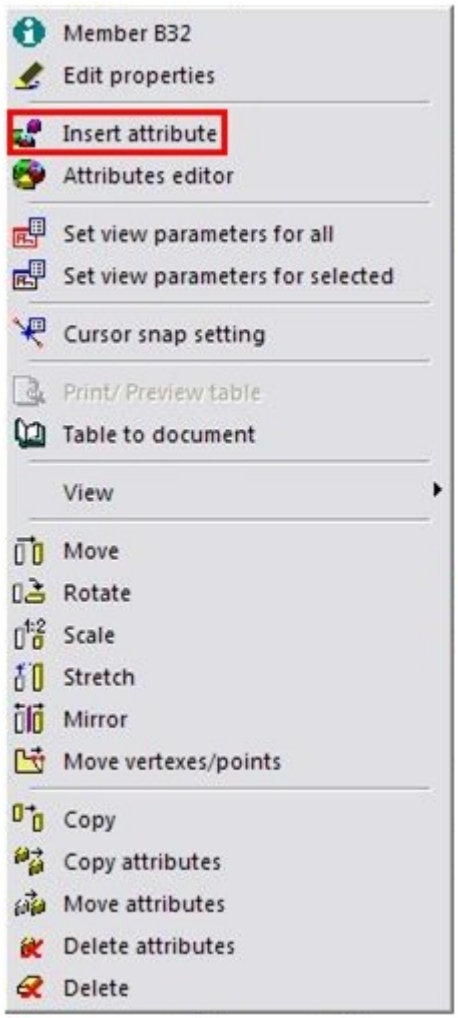
After launching the Input of Attribute a new dialogue window appears on the screen and the user can change the values for the new added attribute. Then he confirms his settings with the OK button and can select member(s) to which the new attribute will be added.



The user-defined attribute is displayed with a label containing the name according to the label setup.



An attribute can be inserted also in the Editor (see chapter about the Attribute editor) or through the context menu – Insert attribute.



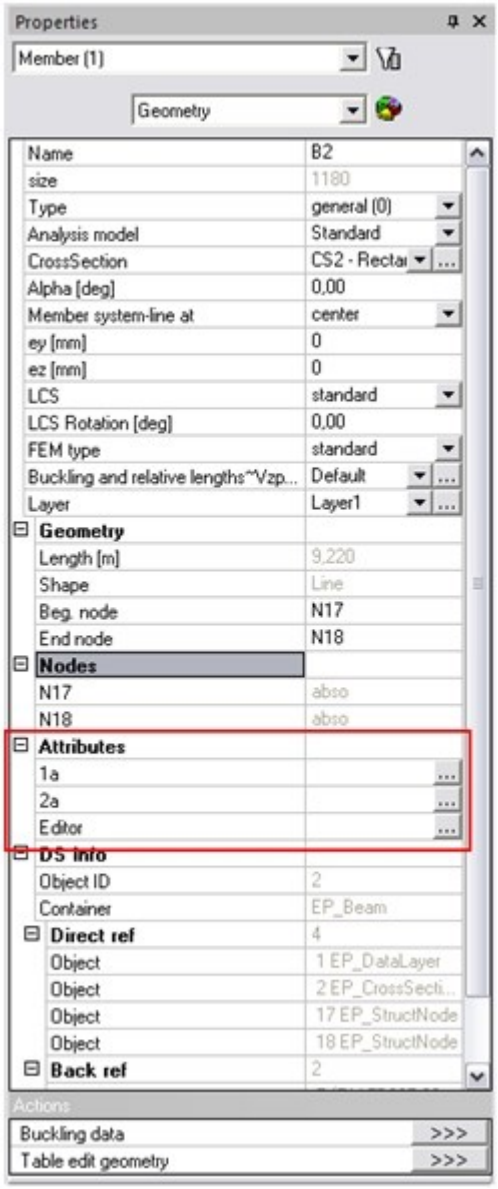
Editing of attributes instances

Editing by attribute properties

There are more ways to edit attributes. A list of attributes is pasted to the property dialogue of the appropriate member. Here a button can be used to see Instance setup with attribute properties.

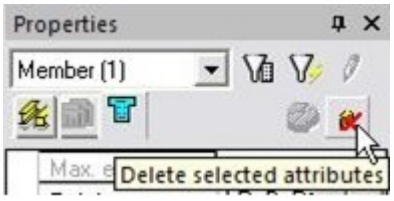
Alternatively the Editor in the same list can be started.

Both ways also works when the user displays the properties through a context menu.

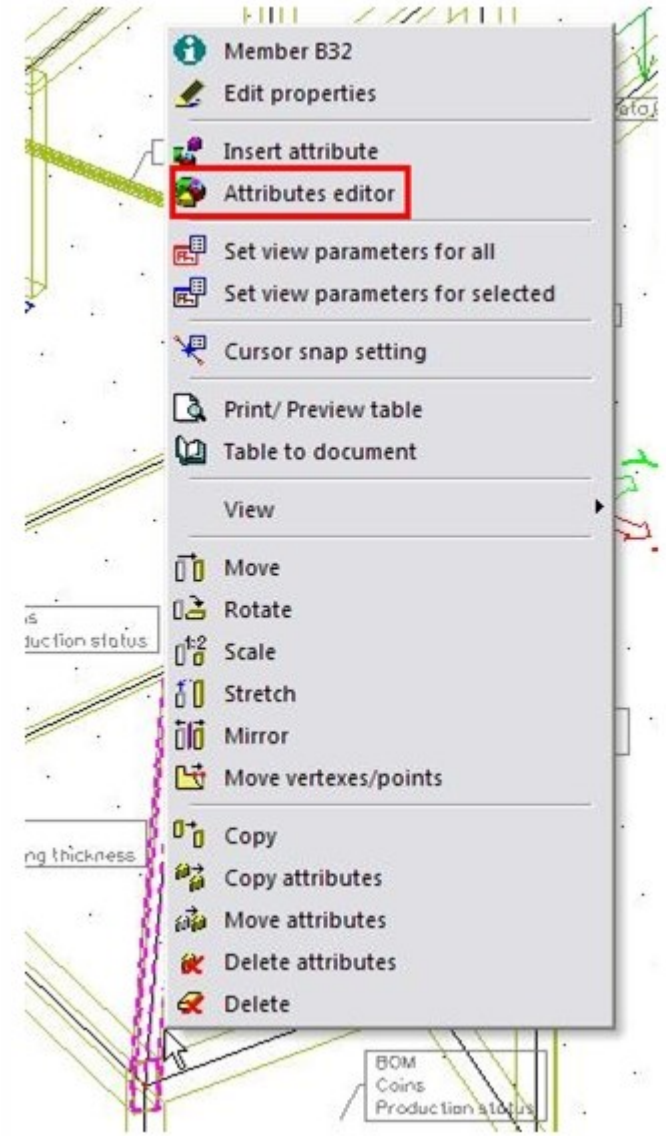


Each member with attributes has tabs with properties of the attributes in the property dialogue.

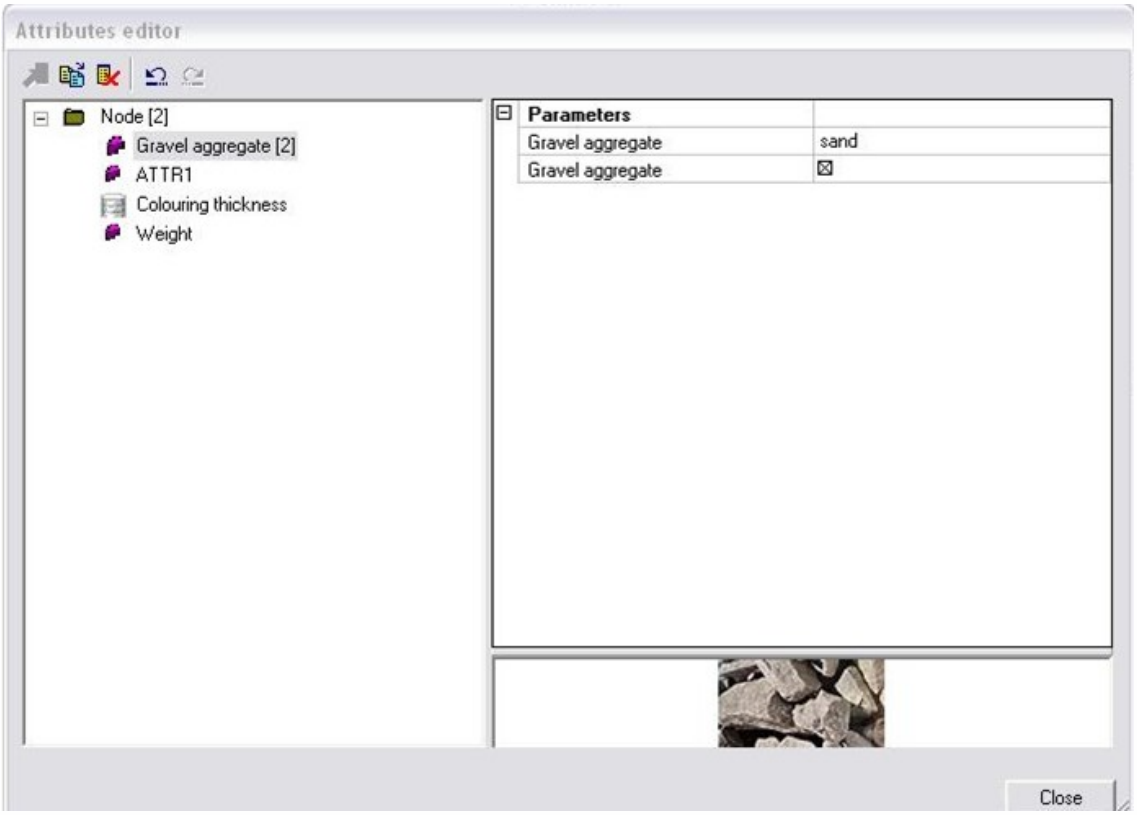
First tab is for member properties and others are for attributes. The last but one icon displays the editor and the last icon can delete active attribute in the properties.



The context menu can be used to start the Attribute editor.


Member B32
Edit properties
Insert attribute
Attributes editor
Set view parameters for all
Set view parameters for selected
Cursor snap setting
Print/ Preview table
Table to document
View
Move
Rotate
Scale
Stretch
Mirror
Move vertexes/points
Copy
Copy attributes
Move attributes
Delete attributes
Delete
BOM
Coins
Production status

Attribute editor



In the left part of the dialogue there is a list of selected members and attributes connected to them. The member without attributes has only a number in brackets showing the count.In the right part of the dialogue there is an “intersection” of the properties of selected member or attributes. The user can change the values here (change value and press ENTER). The user can change the value for multiselected attributes or members. When the user selects item without an attribute, he can insert a new attribute to this member – using button Append new attribute

The user can insert here only those attributes which are already connected to some items in the project (it does not matter to which type – member, node …). When the user has made a selection containing some items with attributes, then he can use button Copy attributes and Delete all attributes in the selected groups

Copy

The user selects Attribute he wants to copy. He clicks button Copy. A new dialogue with a list of available items appears. The user can (multi)select items to append selected attribute to.

Delete
The user (multi)selects a group of attributes and clicks button Delete.

Copy, move, delete

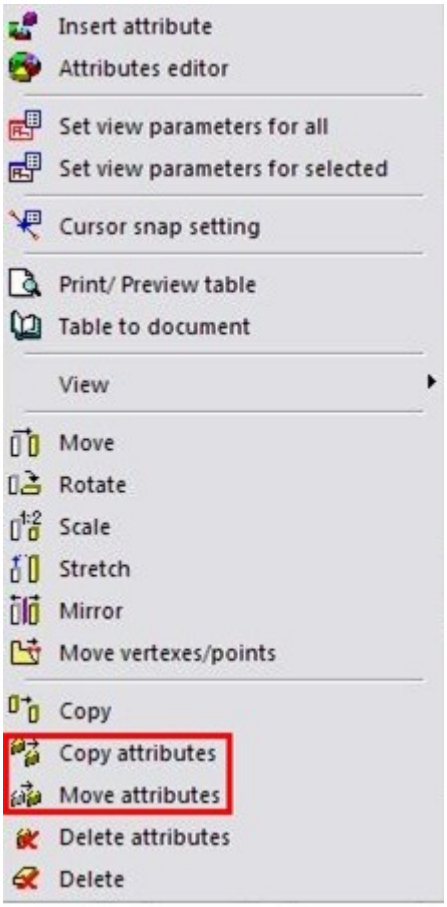
Copying and moving of Attributes

The first possibility to copy or move attribute is in the Attribute editor (previous chapter).

Another possibility to Copy and Move attributes are icons on a toolbar.

Copying and moving of attributes is possible only between the same types of item (from node to node, from 2D member to 2D member, etc.).

The same icons for copying and moving are added to the context menu.

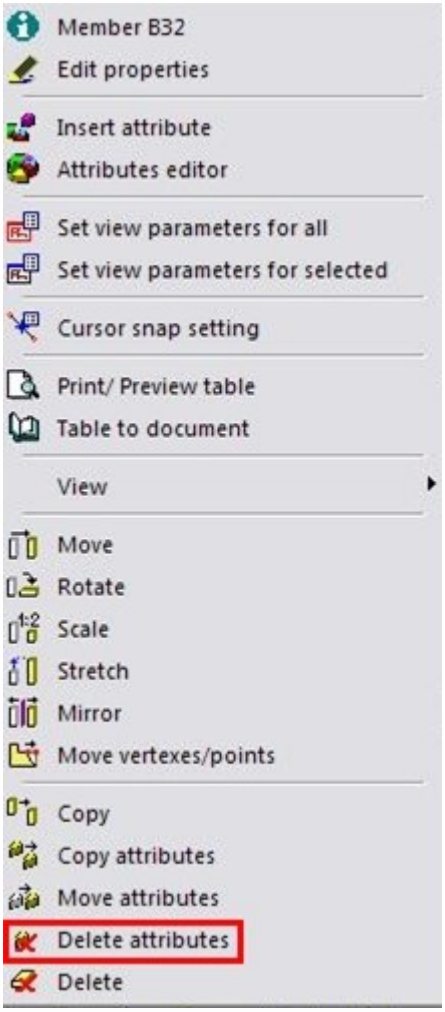


Deleting of Attributes

Deleting of Attributes can be done in the Editor (see chapter Attribute editor).

Alternatively, it is possible to use the icon in the property dialogue. This icon can be also used when no member is selected and the combo box contains only attributes.

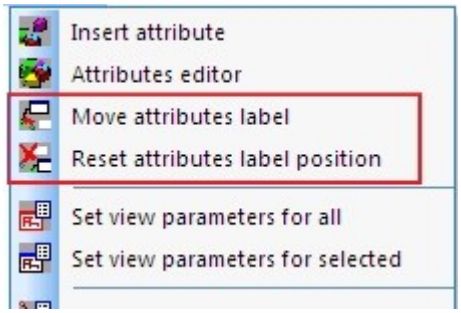
The third way is through the context menu and button Delete attributes.



Moving of attributes labels

Attribute label is automatically placed to the center of his owner – member, plate,... The attribute owner must be selected to move the label.

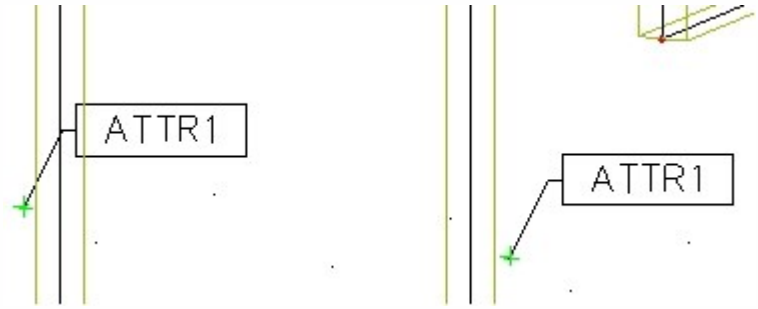
New commands for moving of labels are developed for version 2011 – “Move attributes label” and “Reset attributes label position”.



Attribute label can be moved in the same way as vertexes. “Move attributes label” display label with the green dot and by this dot is possible to move attribute label.

“Reset attributes label position” turns label back to its original position.

These two functionalities are possible to use also for more than one attribute label.

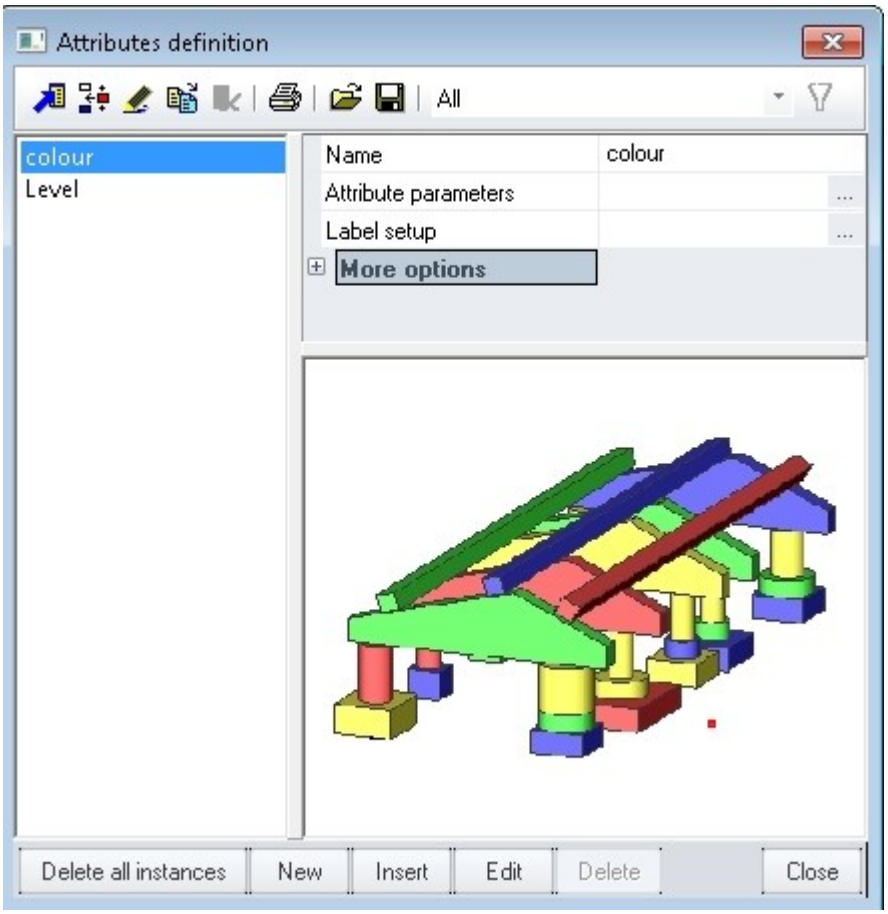


How to create and work with a User defined attribute

Attribute parts

Attribute definition

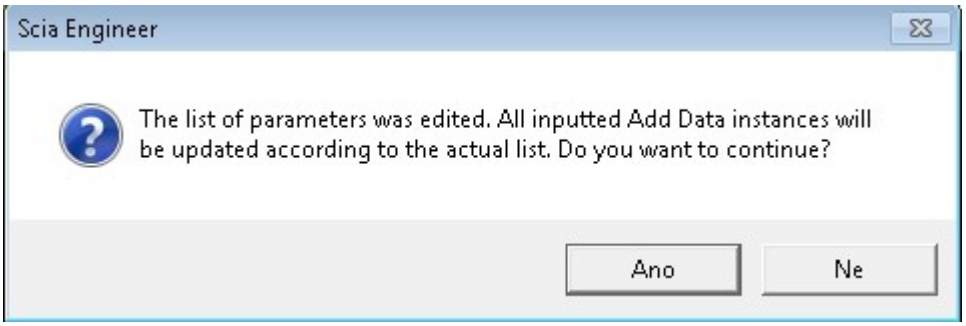
The definition of the attribute is a library item. This item specifies which parameters contains the user defined attribute. Each definition is only once in the project - in library. This is the specification of the attribute.



Instance

The instance is based on the attribute definition, but the values of predefined parameters can be changed by user. It is inserted to the project. It shows the label of the attribute and parameters are displayed in the properties. Each instance of the same attribute definition can have a different values but it has the same parameters.

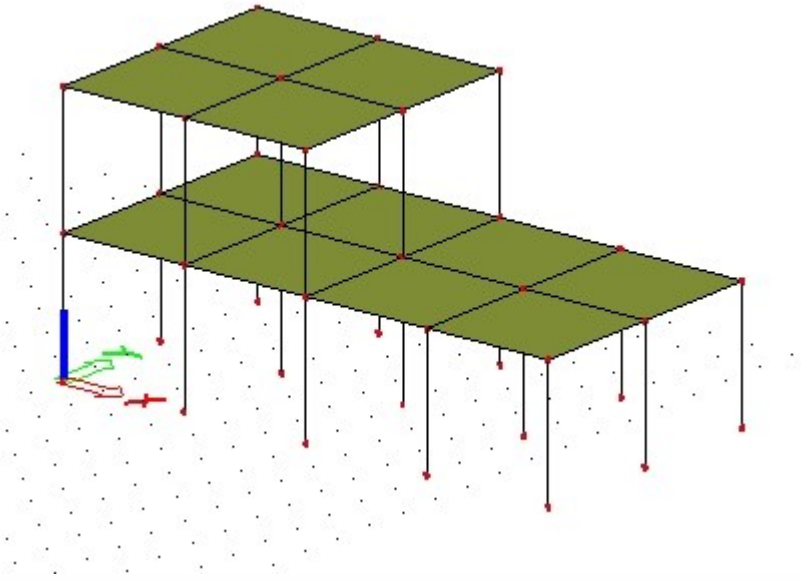
If the definition is changed, all its instances should be updated. There is a message:



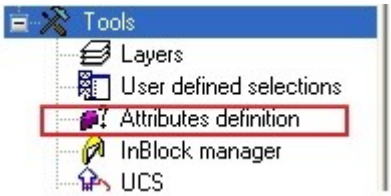
How to create the attribute definition

Definition of attribute Colour with two parameters - colour and thickness:

1. Open the attached project – Attributes.esa. You can see a simple structure composed of beams and plates in two levels.



2. Find service Attribute definition in Tree menu service Tools.

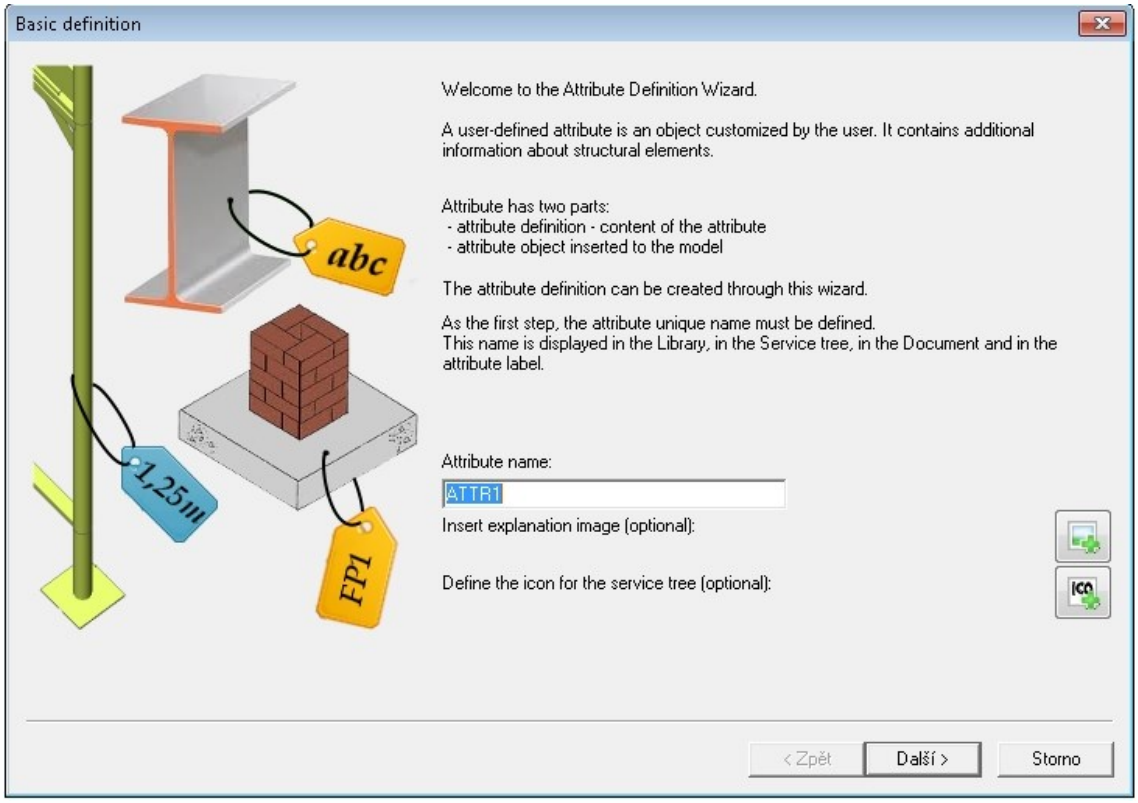


Definition of the attribute by Attribute wizard

It is possible to create the attribute definition by the library itself or user can use the attribute wizard. The wizard is much more user friendly tool and the next part describes its functionality.

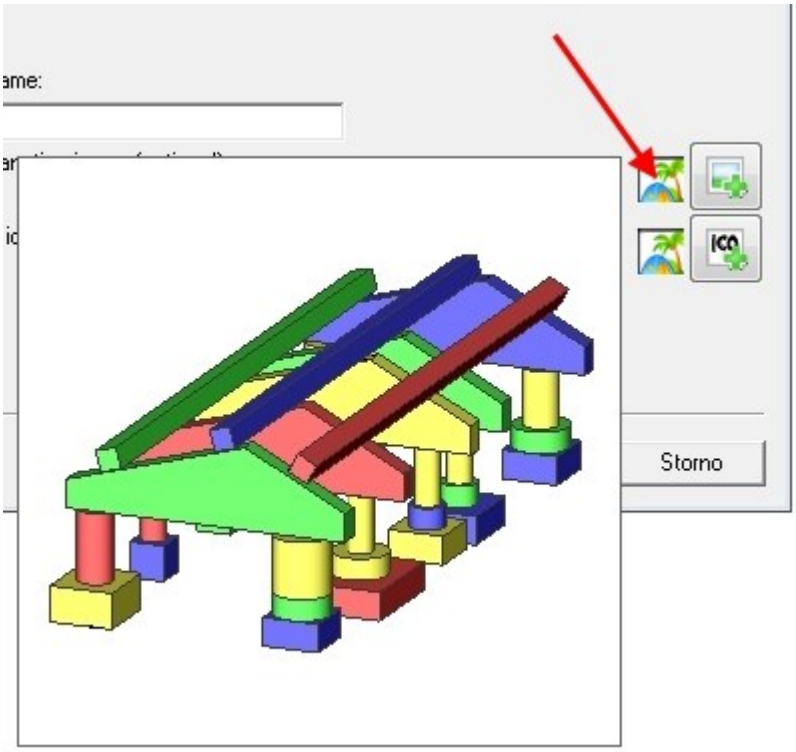
1. The attribute wizard is opened automatically when no attribute definition is already created in the library. If the library contains at least one definition, wizard starts automatically when user create a new item.

2. The first part contains the name, explanation image and icon which used in the service afterwards.



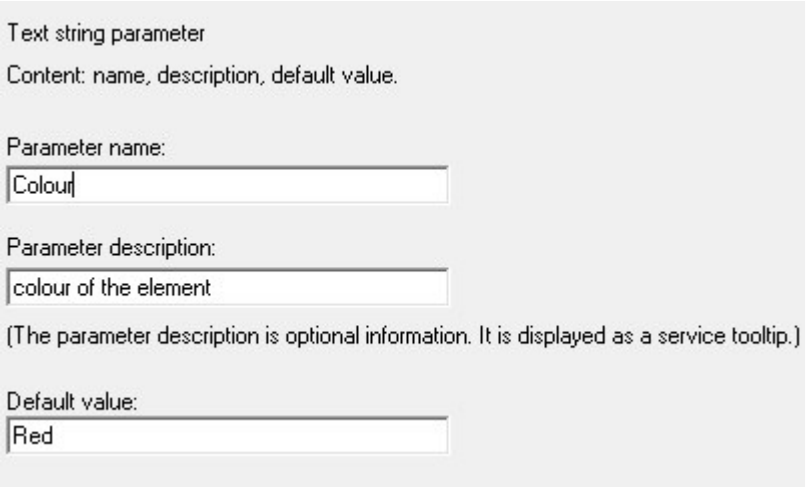
If there is no small picture next to the button for adding, it means that no image and no icon are added yet.

3. Define the name as "Colour", add picture "colour.bmp" and icon "ico_colour.bmp". Used picture and icon are displayed by small previews when user moves cursor over the picture next to the button for adding.

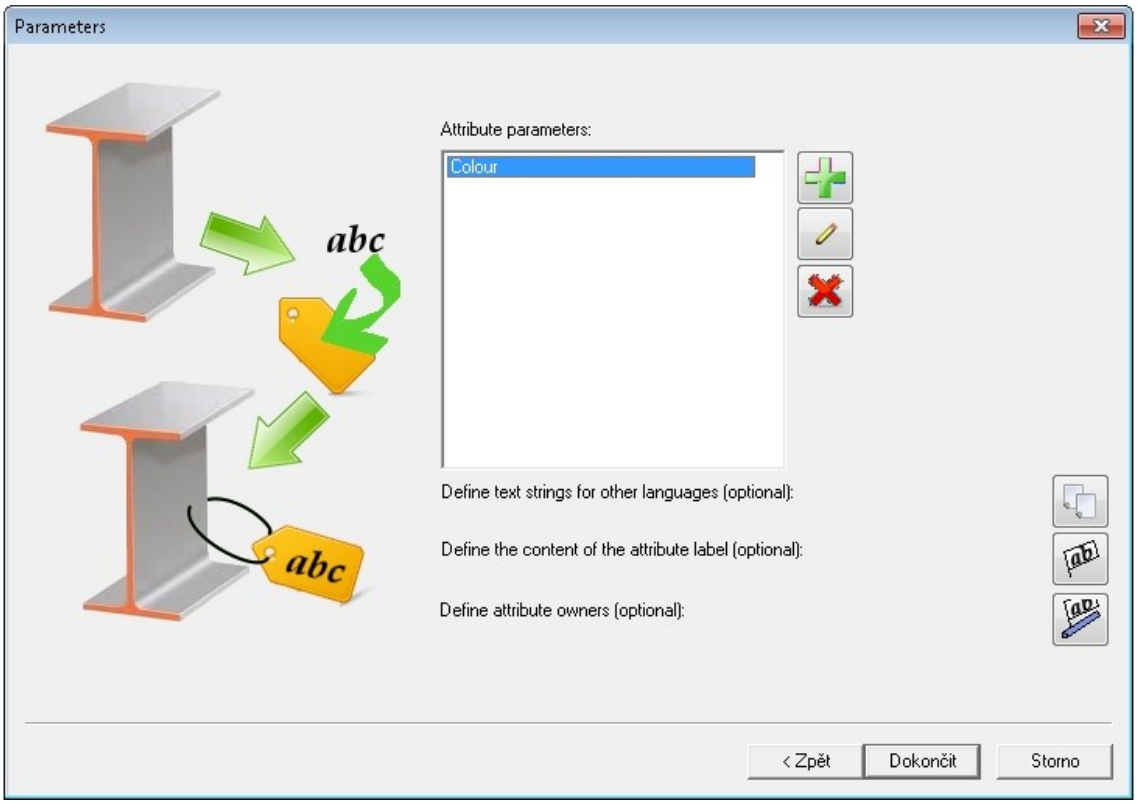


4. Use button Next and move to the second step.
5. Define the parameter type in the next step. The first parameter is String. Click on the big icon.

6. Fill in the String parameter definition. The name is "Colour", description is "colour of the element" and the default value is "Red".

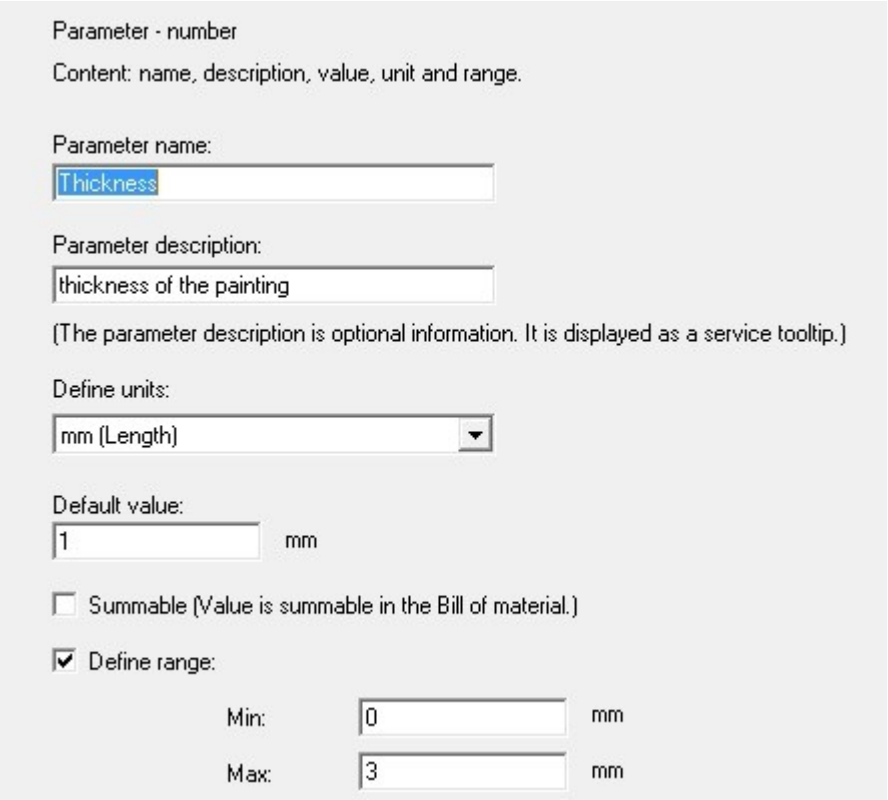


7. Continue to the next step.
8. There is a list of already created parameters for this User defined attribute.

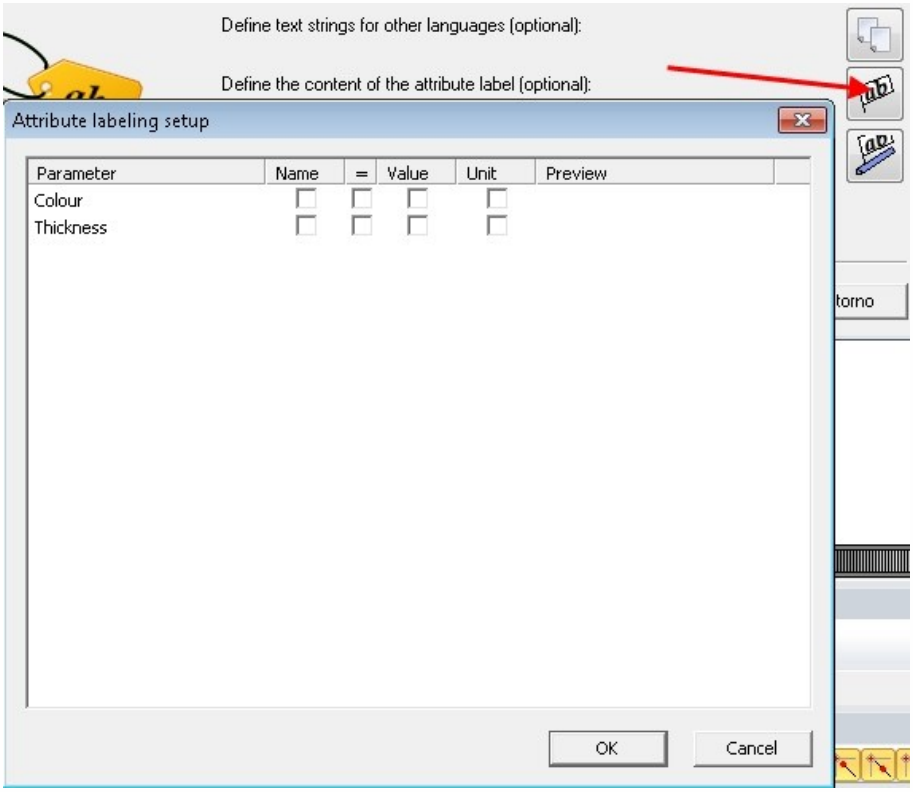


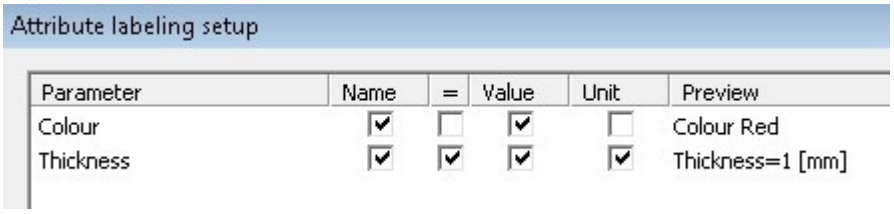
9. Add a new parameter by the button with green plus and then select the type Number.

10. Define the settings of the Number, name is "Thickness", description is "thickness of the painting" and default value is 1. Select the units - mm (Length) and define the range from 0 to 3mm.

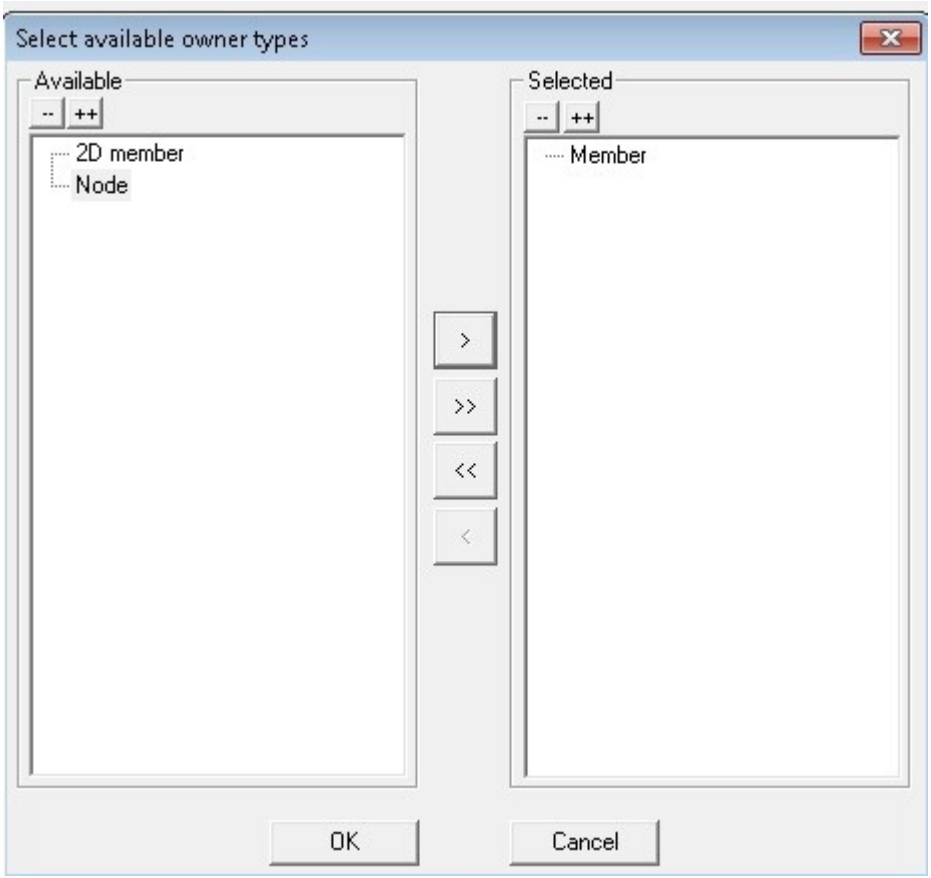


11. Finish this step and check if the list displays two parameters for this attribute.
12. Define the content of the attribute label. Label for colour - check the checkboxes for names and values. Label for thickness - check all checkboxes.

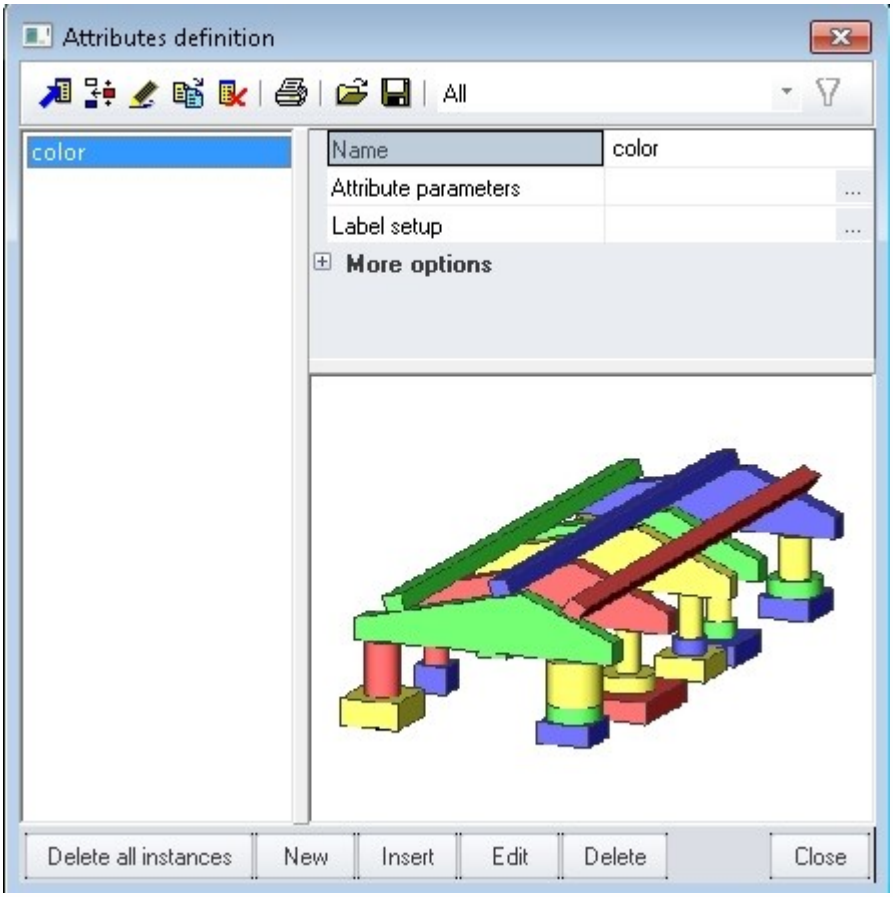

Define text strings for other languages (optional):
Define the content of the attribute label (optional):
Attribute labeling setup
Parameter
Name
=
Value
Unit
Preview
Colour
Thickness
OK
Cancel


Attribute labeling setup
Parameter
Name
=
Value
Unit
Preview
Colour
Colour Red
Thickness
Thickness=1 [mm]

13. Define the attribute owner - that means which type of elements can contain an instance of this attribute definition. The owner for this definition is 1D member. Select "Member" in the left list and move it to the right part.

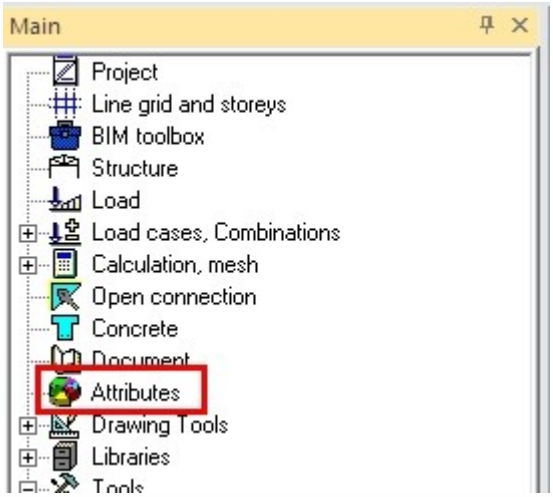


14. The attribute is ready to use. Finish the wizard and check the new attribute definition in the library.

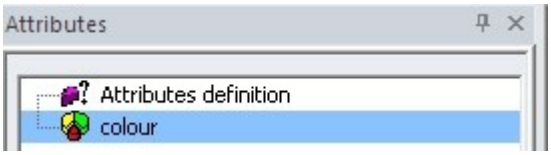


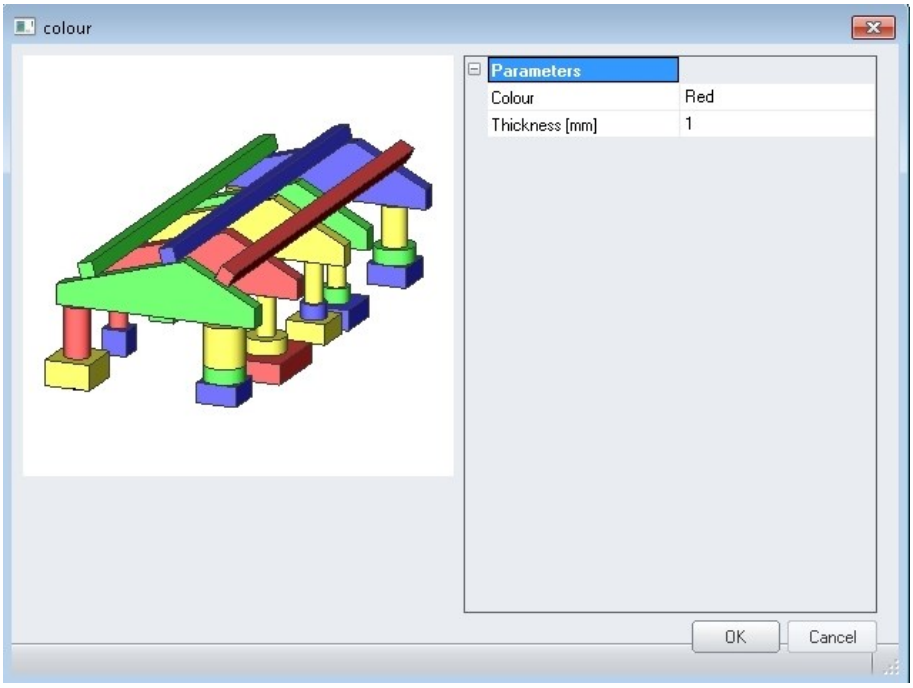
How to insert attribute instance to the project

1. Close the Attribute definition library
2. The new service is displayed in the main tree menu - Attributes.

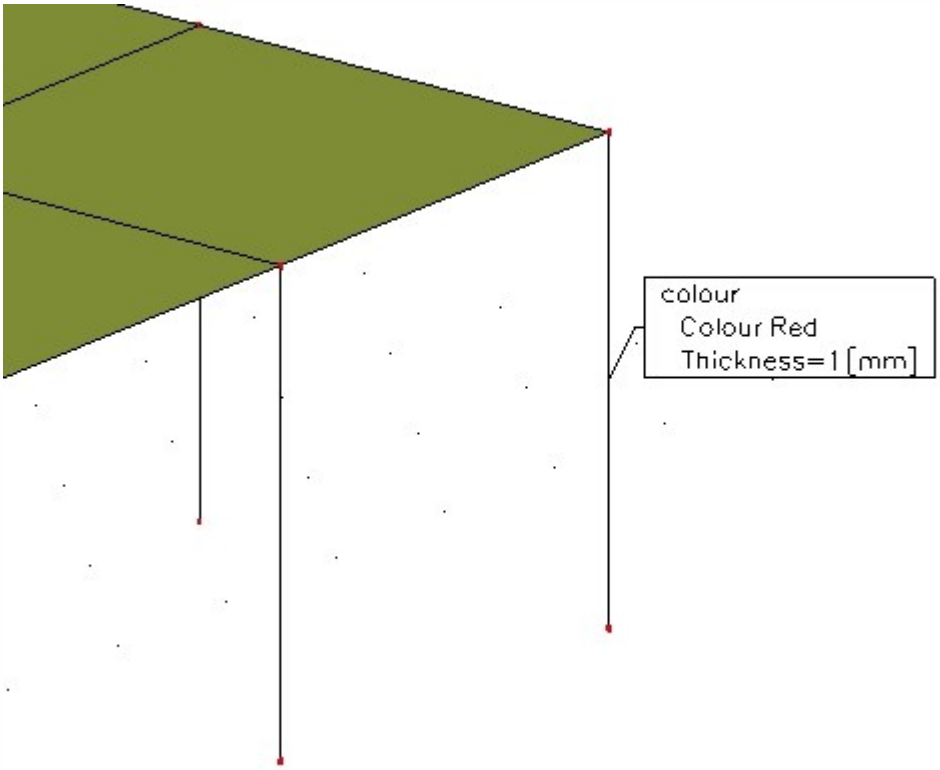


3. Open the service and start command Colour. The new dialogue is opened. The default settings are displayed.



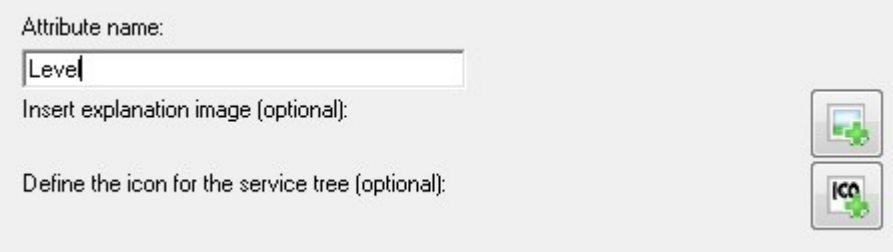


4. Confirm by OK and select one column. The attribute instance is added and the predefined label is displayed.



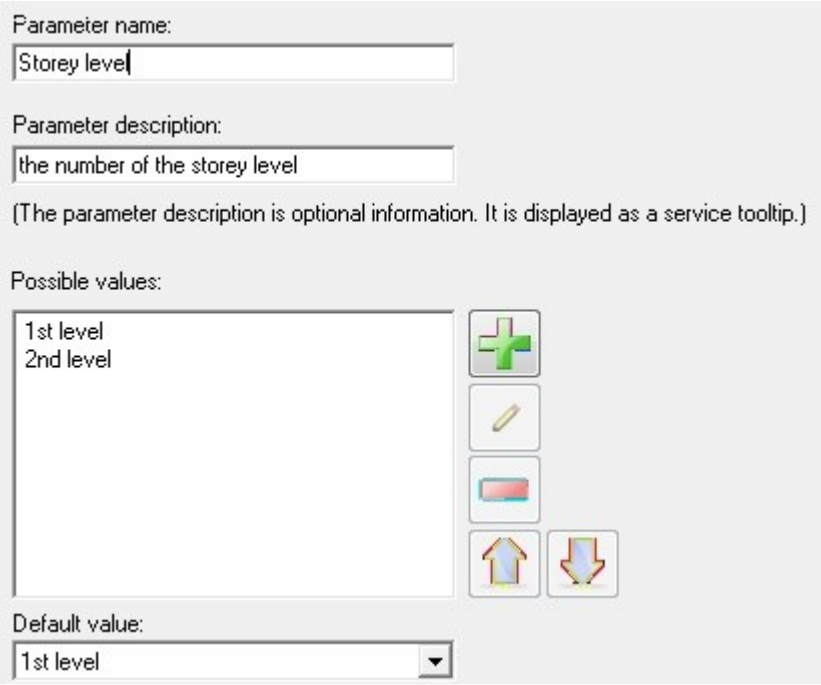
Add a new attribute definition

1. Open the Attribute definition library from the Attributes service.
2. Click on New. The application starts wizard.
3. The name is "Level". Don't add any picture or icon. The system defines the default. Continue to the next step.

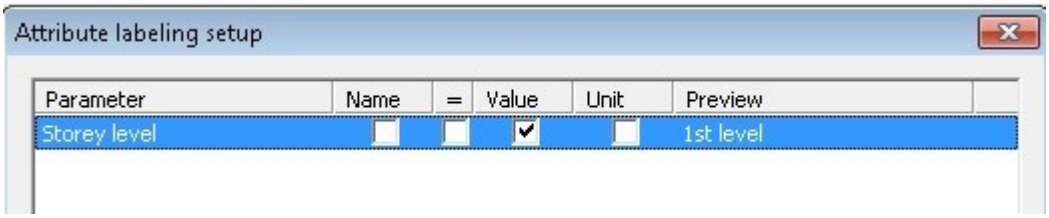


4. Select the parameter type Combobox.

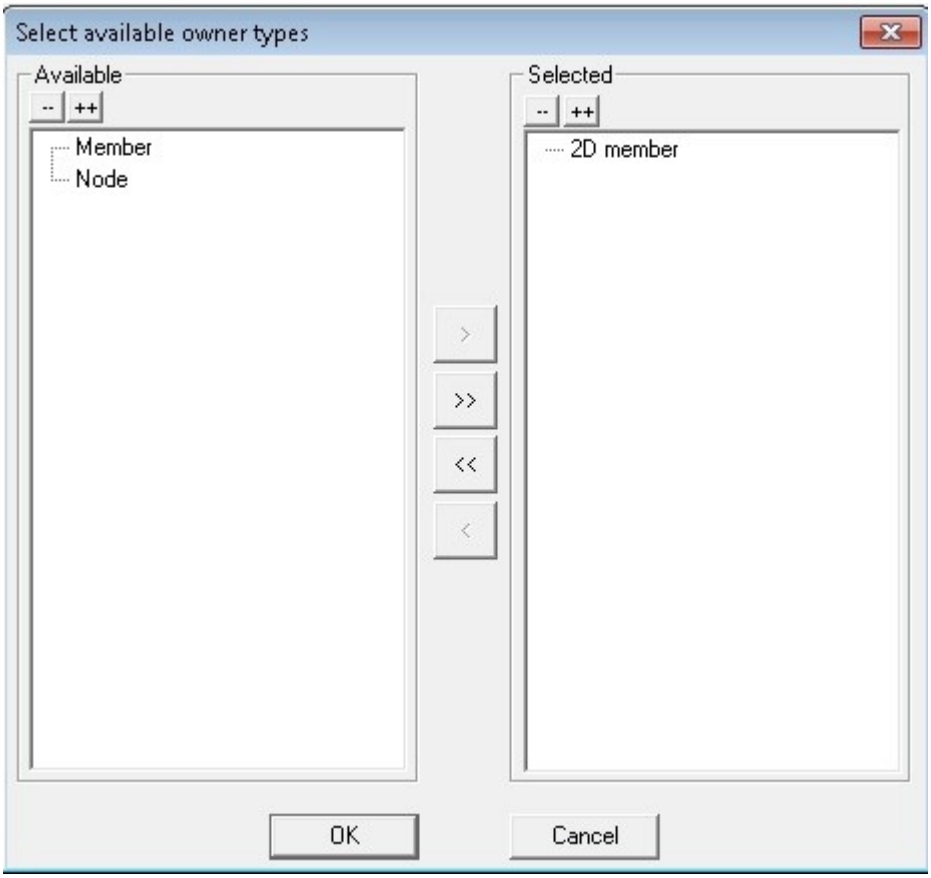
5. The parameter name is "Storey level", the description is "the number of the storey level", add two items to the list of possible values and name them "1st level" and "2nd level". The default value is "1st level".



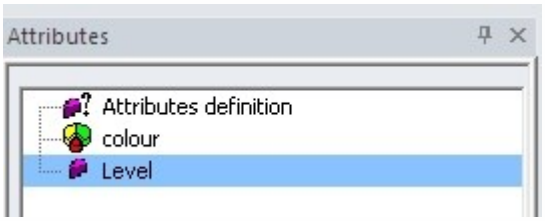
6. Finish the definition of the parameter.
7. Define the attribute label. Check only value.



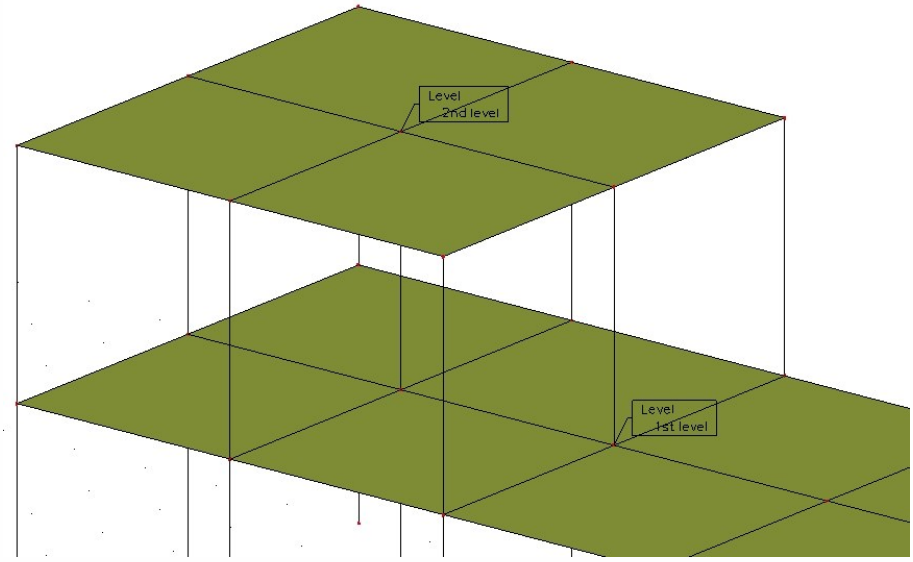
8. Define the parameter owner. Select 2D members.



9. Close the wizard and check that the new attribute is displayed in the service. There is a default icon used for the attribute.



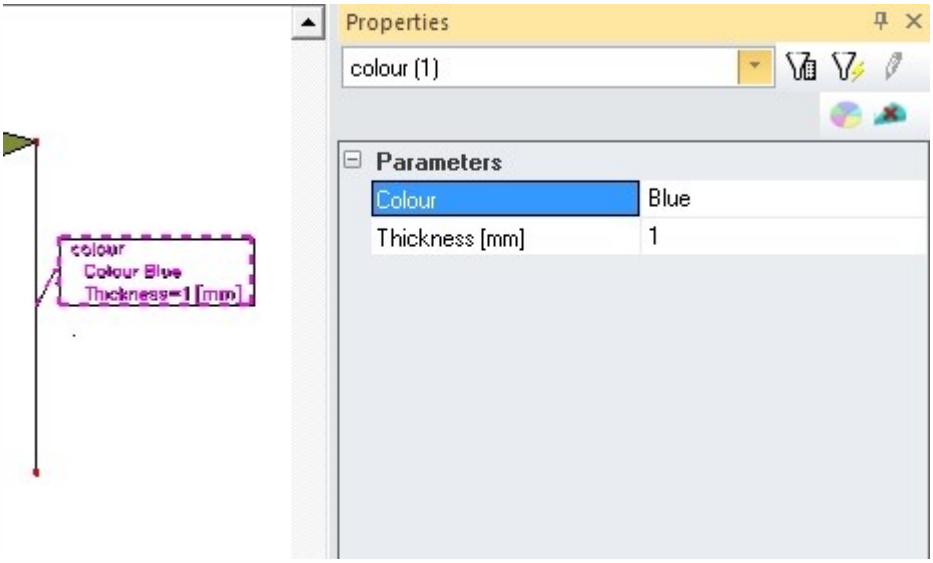
10. Insert the new attribute on both plates in the project. Change the value of the higher attribute to "2nd level" during inserting. The label is always placed to the plate center.



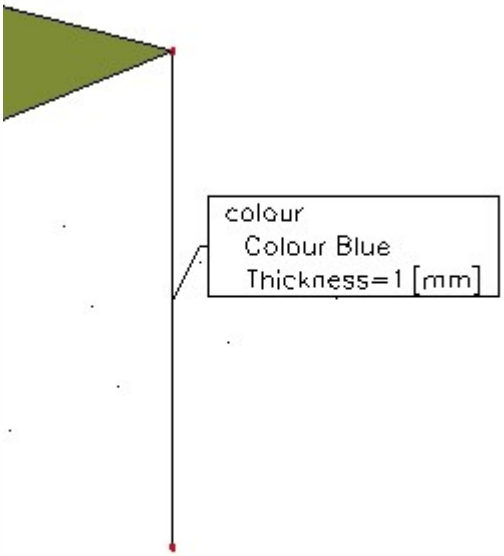
Modify the properties of the attribute

Properties are displayed if user selects attribute label or selects the owner. It is displayed on the second tab in the properties of owner.

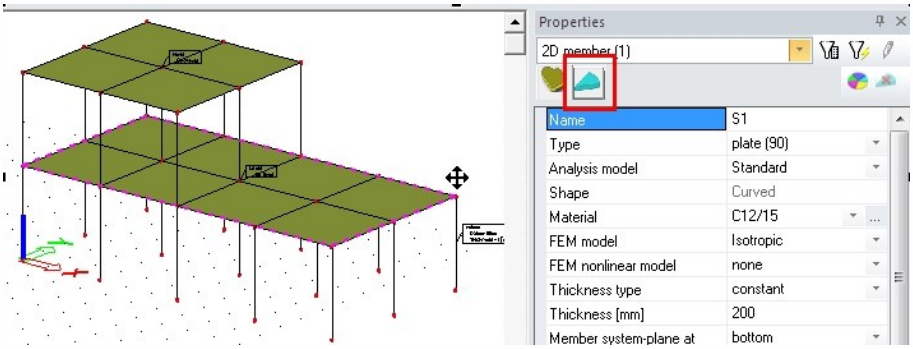
1. Select the Colour attribute label.
2. Change the parameter Colour from Red to Blue.

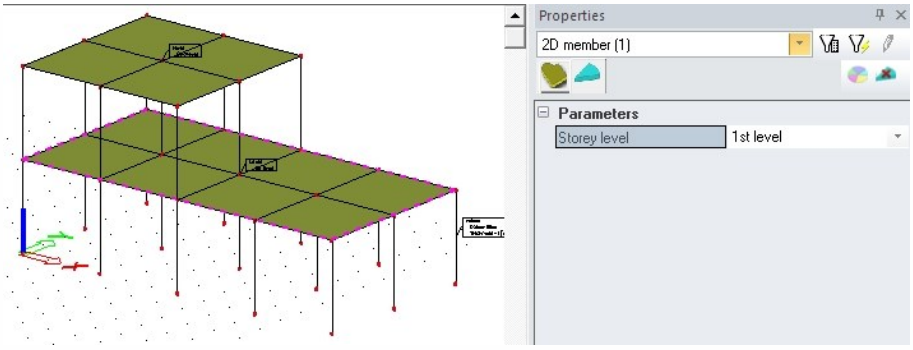


3. Check the changed value on the attribute label.

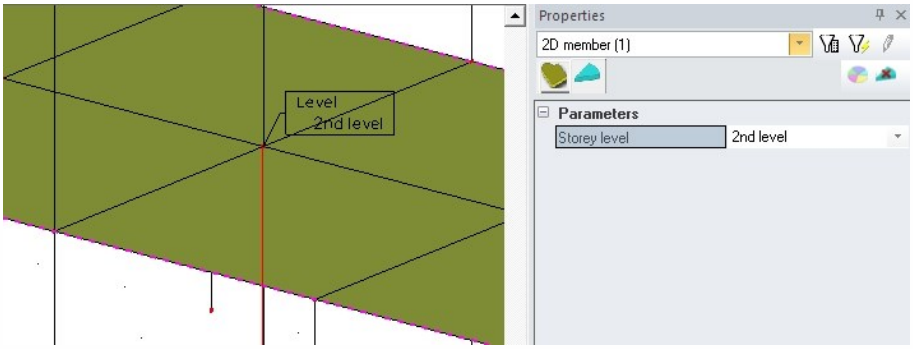


4. Select the bottom plate. Switch to the second tab in the property dialogue.



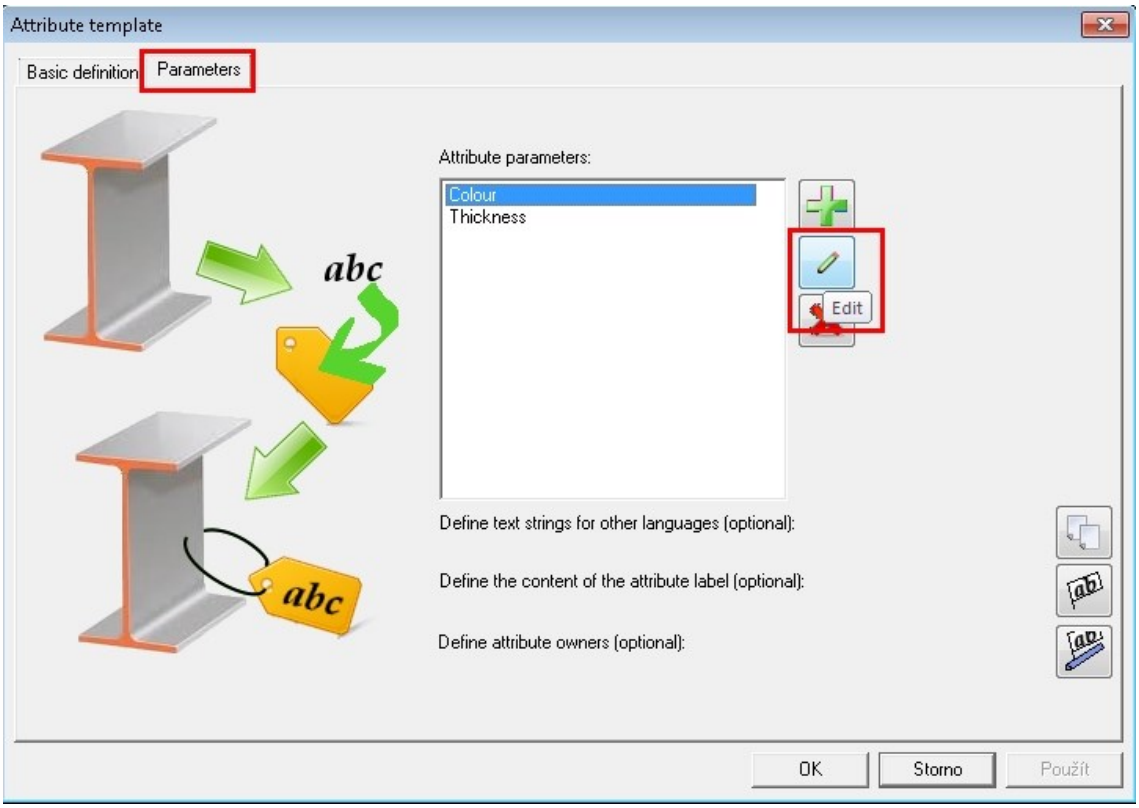


5. Change the parameter to "2nd level". Check if it is displayed on the label.



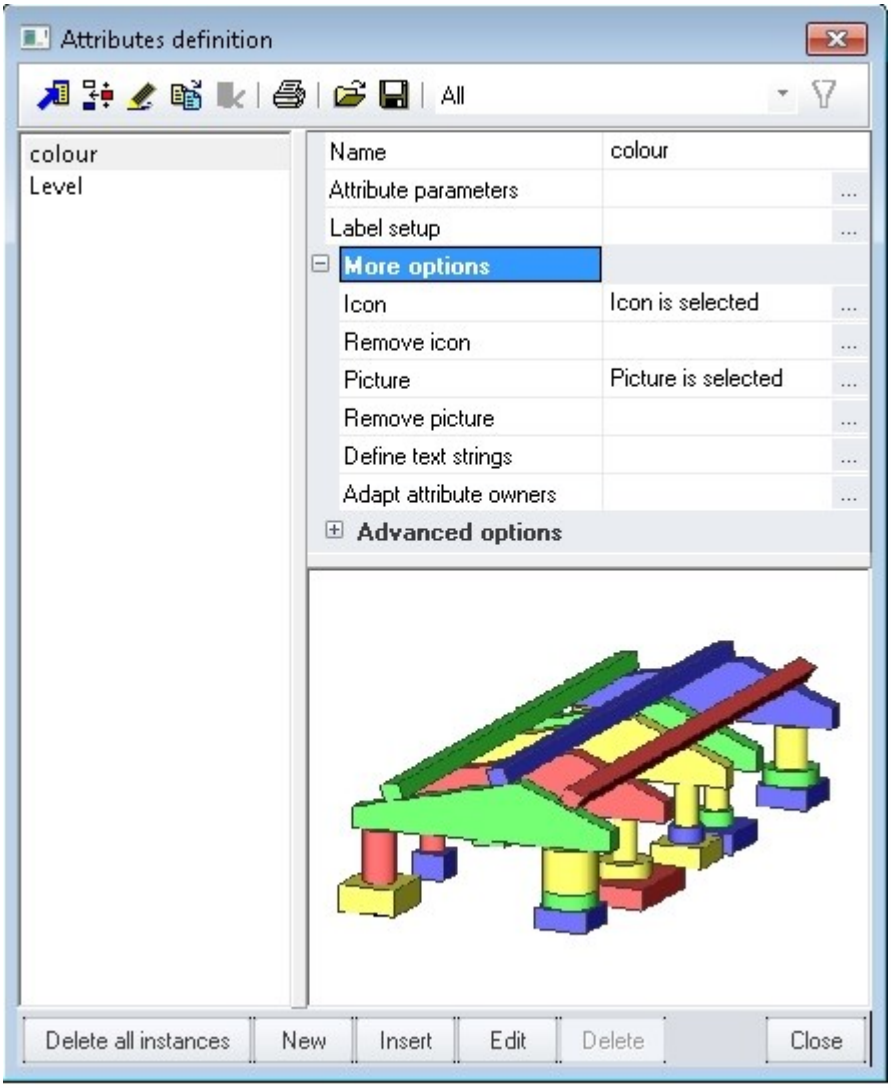
Modify the properties of the attribute definition

The attribute definition can be modified by the attribute wizard in the same way as it is created.

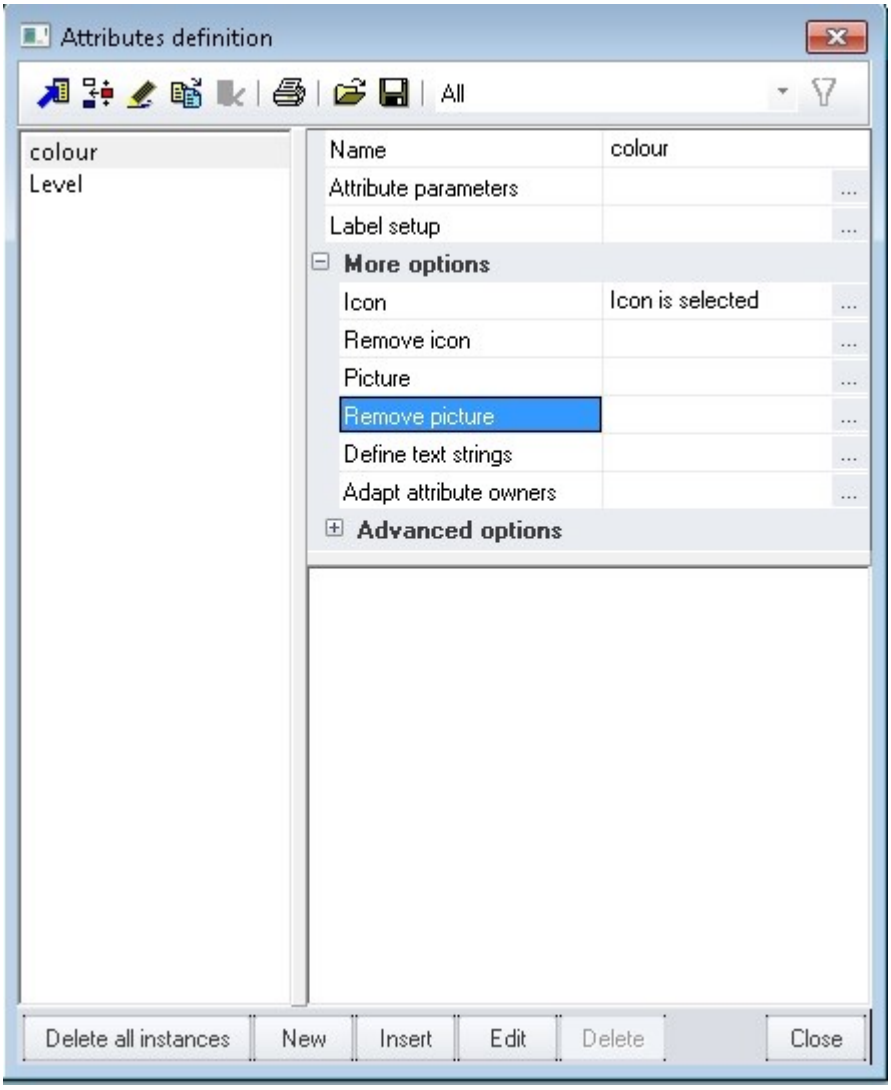


Sometimes it is faster to find the property in the library directly.

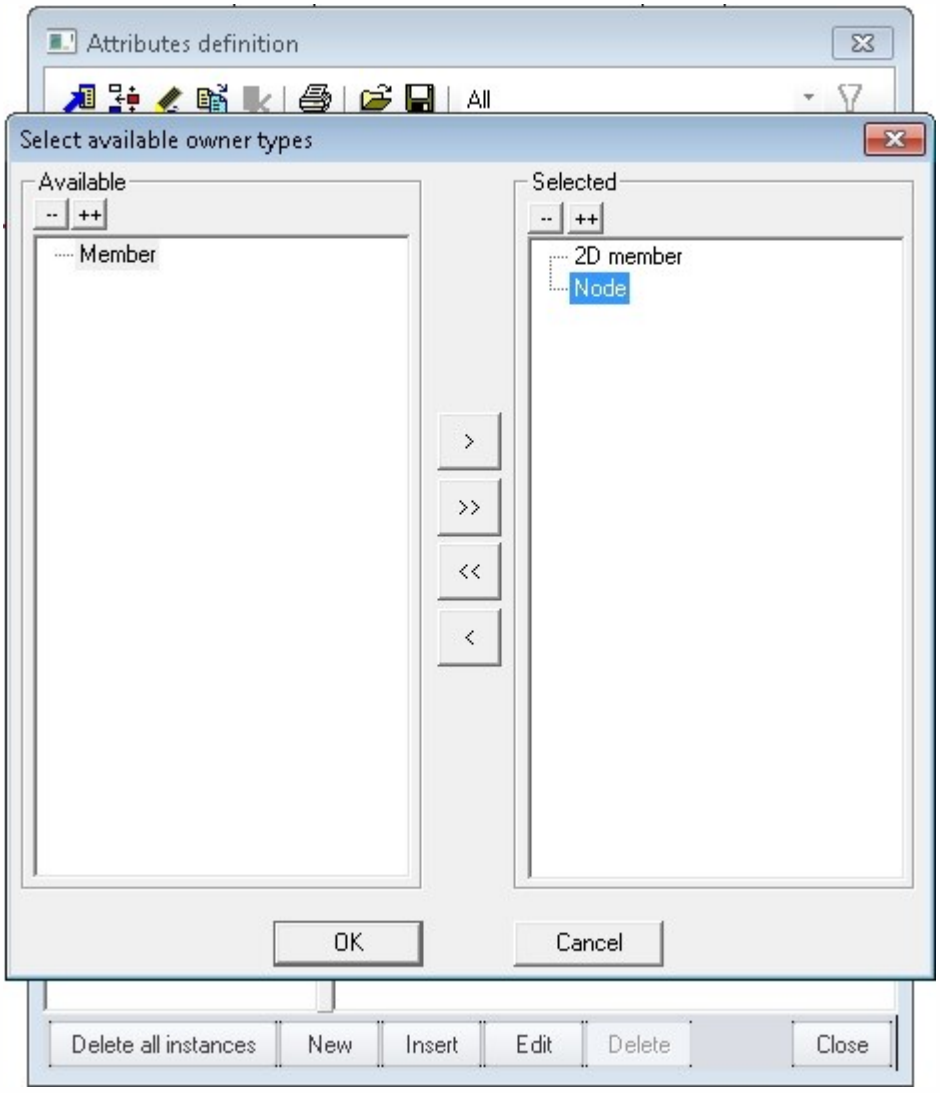
1. Open the attribute definition library.
2. Click on "More option" in properties of Colour. The group is expanded.



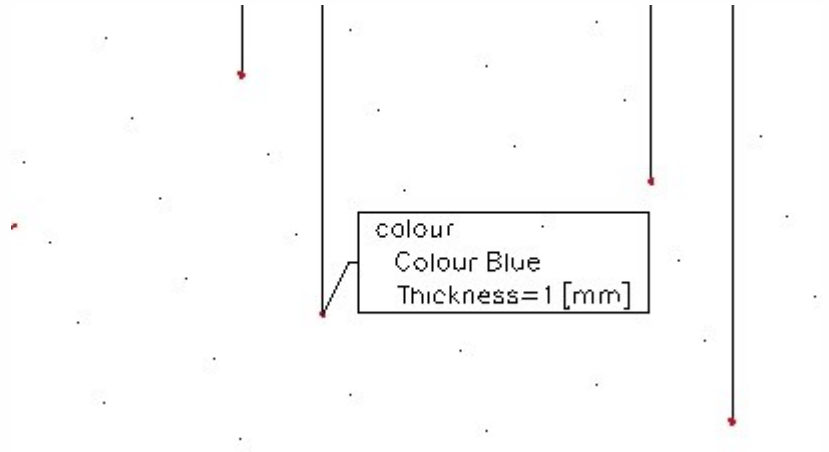
3. Click on button to remove the picture. The preview is empty.



4. Click on "Adapt attribute owners". Define one more owner for attribute - Node.



5. Close both dialogues and insert attribute Colour on node on the structure.



6. The label is now displayed on the node.

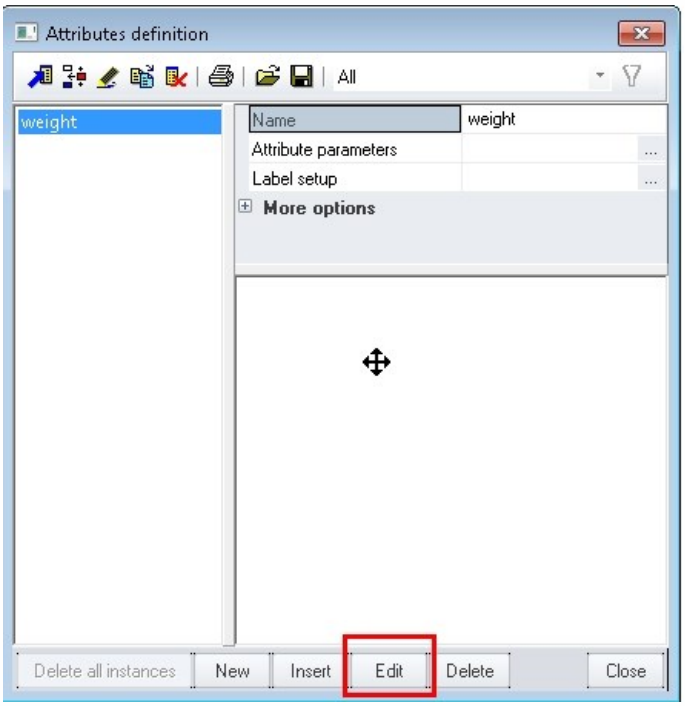
The final project with inserted Attributes is attached as "final_attributes.esa".

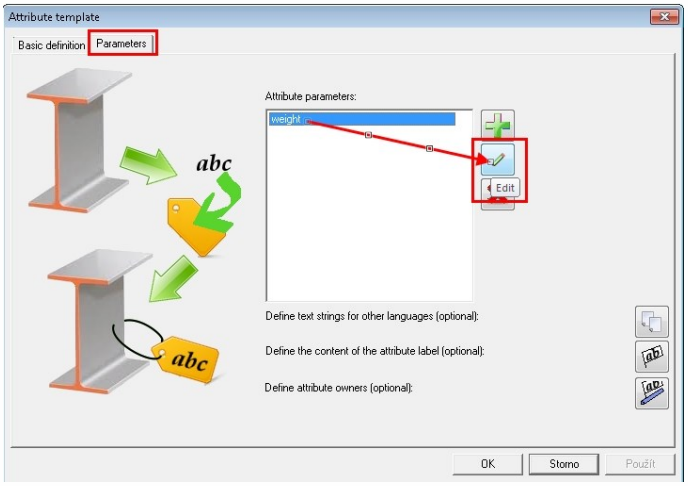
How to create summable attributes

The attribute parameter Number has an option summable. This option allows to add a table with sums of values to the document.

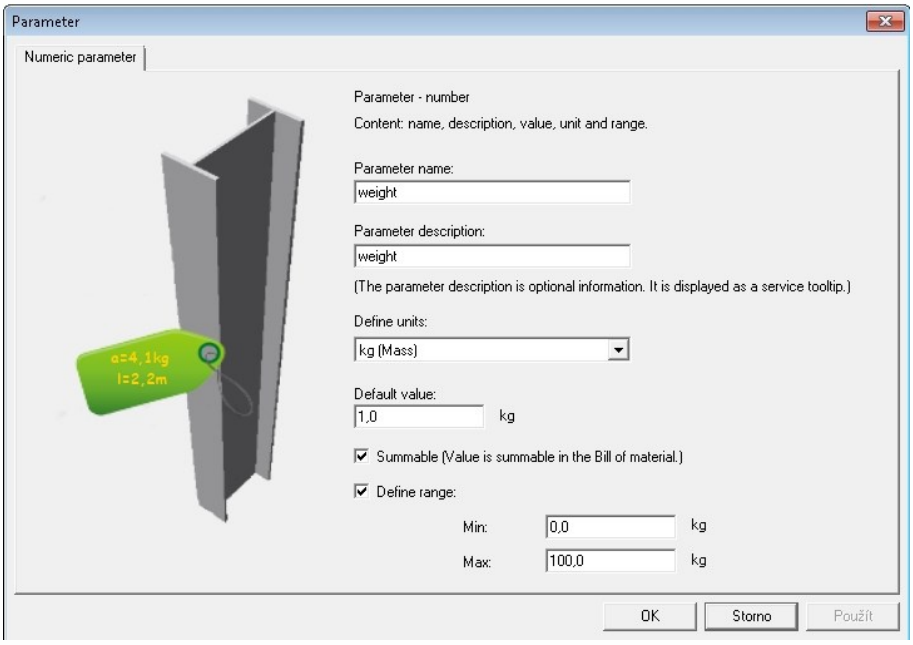
Create summable attribute

1. Open attached project "attribute_summable.esa"
2. Open service Attributes and open library Attribute definition.
3. Edit attribute definition "weight", open tab "Parameters" and select "weight" in the list. Start editing.

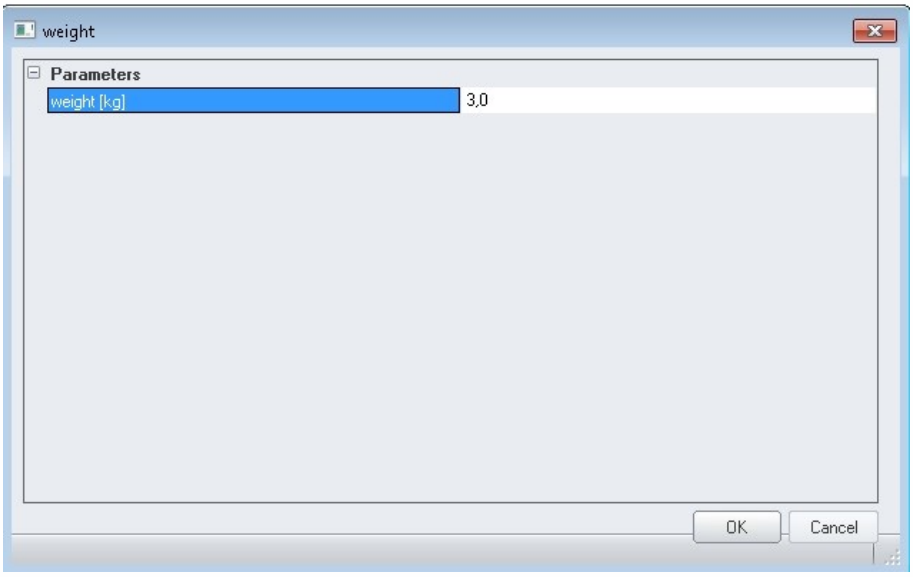




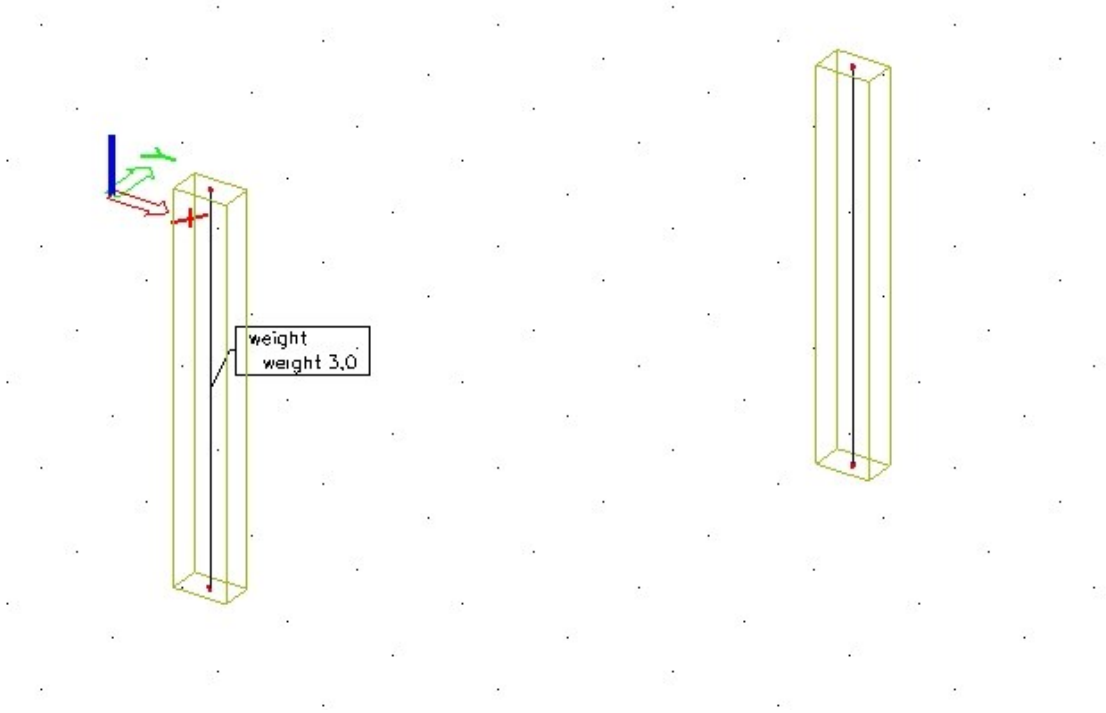
4. Check "Summable" option and close the dialogue.



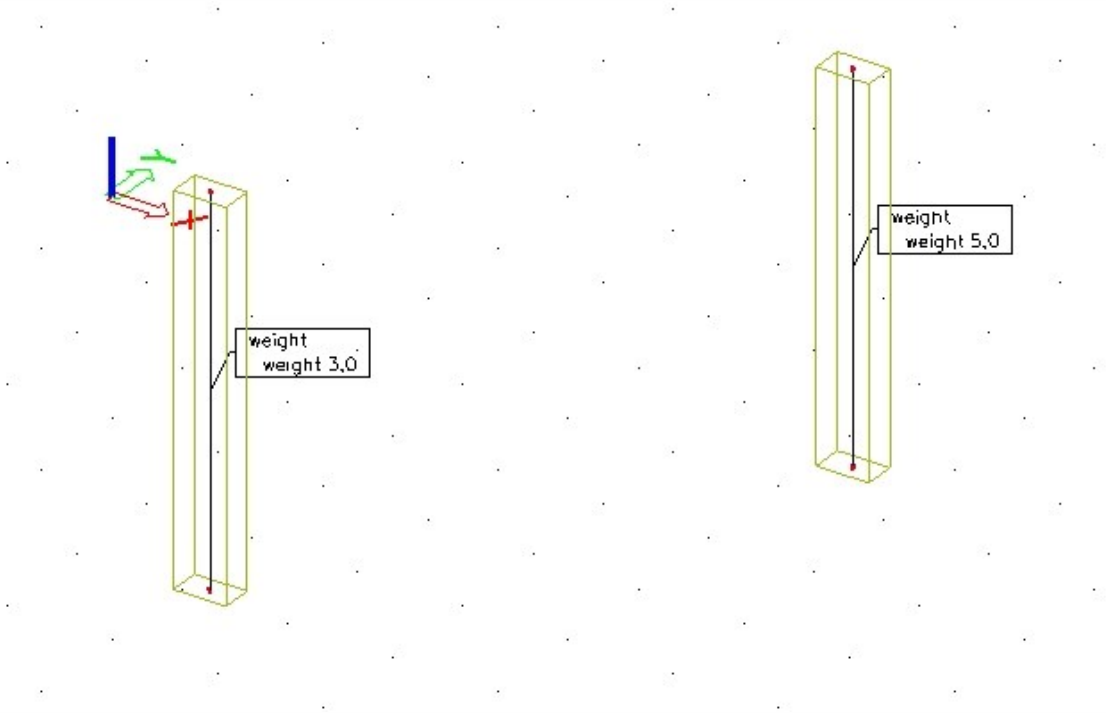
5. Start the command weight from the service menu. Change the value to 3 in the inserting dialogue. Confirm by OK.



6. Insert the attribute instance on the first column.

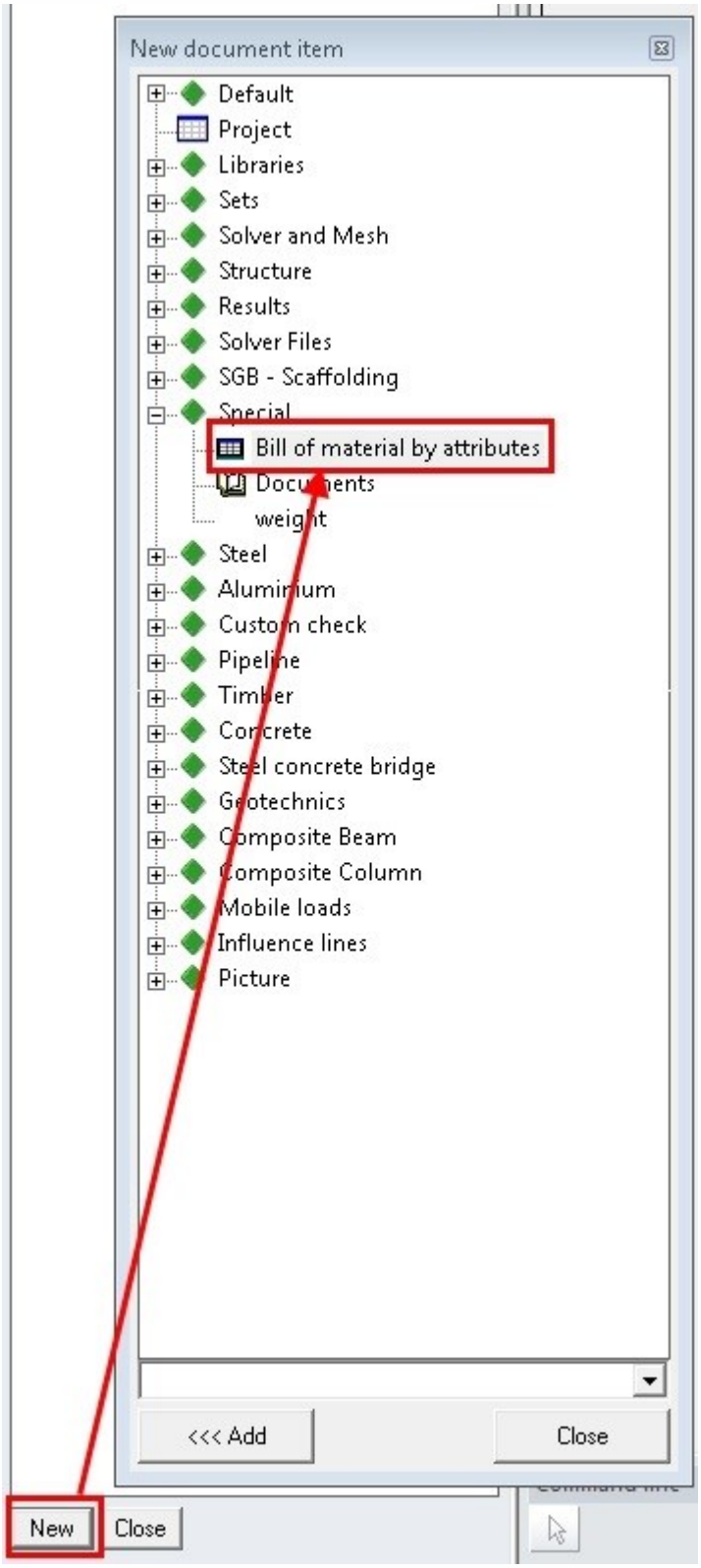


7. Start the command again. Change the value to 5 and insert it on the second column.

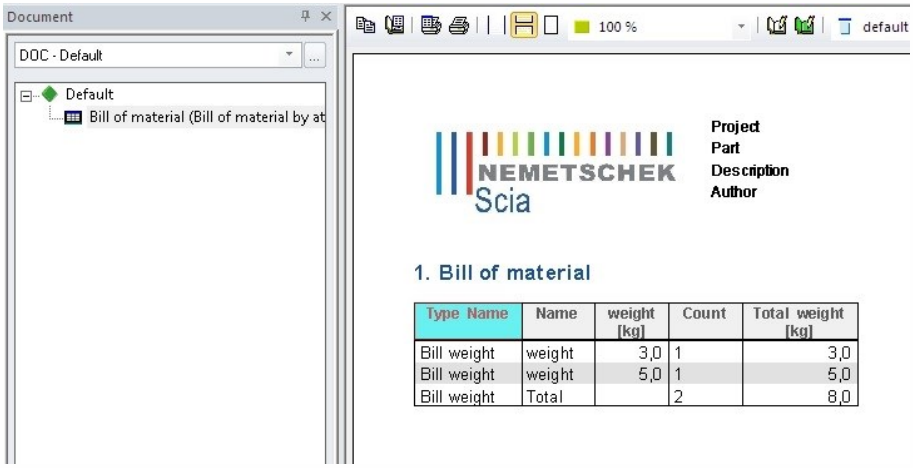


Add table "Bill of material to the old document.

1. Check the functionality for Old style document. Open document.
2. Open list of insertable items (button New). Select group "Special" and item "Bill of material by attributes".



3. Add it to the document and refresh.



4. There is a column Total weight (kg) which displays the summ of both attribute instances which is inserted to the project.

The final project is attached as "attribute_summable-finished.esa".

Storeys

Storeys

SCIA Engineer has two tools for better vertical orientation in the project. These are storeys and 3D linegrid. Storeys are defined by separate levels. Each level is defined by name, height, selection etc. The storeys are also connected with activity functionality, so user may display the level separately.

Storey is also used by the seismic calculation. See more in a manual about Seismic analysis.

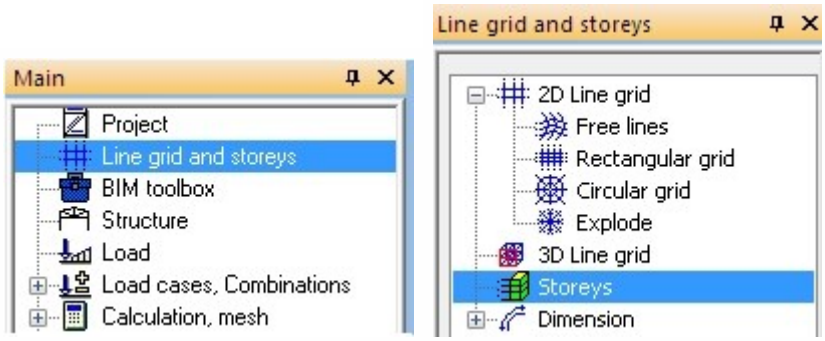
The storey can be created in XYZ type of project.

The storey is generated as an object with a predefined shape. The values for heights, the inserting point and name are defined by the Storey manager. The visibility of the storeys is controlled by view parameters.

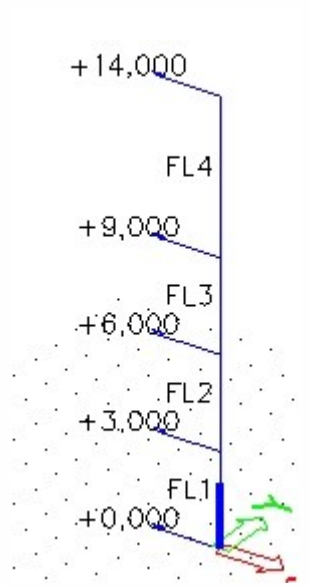
The storey can be used for special type of activity and also for the simple generation of the plan section entity.

The special settings is allocation. When any beam is allocated to the specific storey level, then it can be profitably used in the activity according to the storeys.

The storey manager is in the service “Line grid and storeys” in the main tree.



The storey object in the 3D window:



Storey manager

The dialogue for storey definition contains a simple grid with preview.

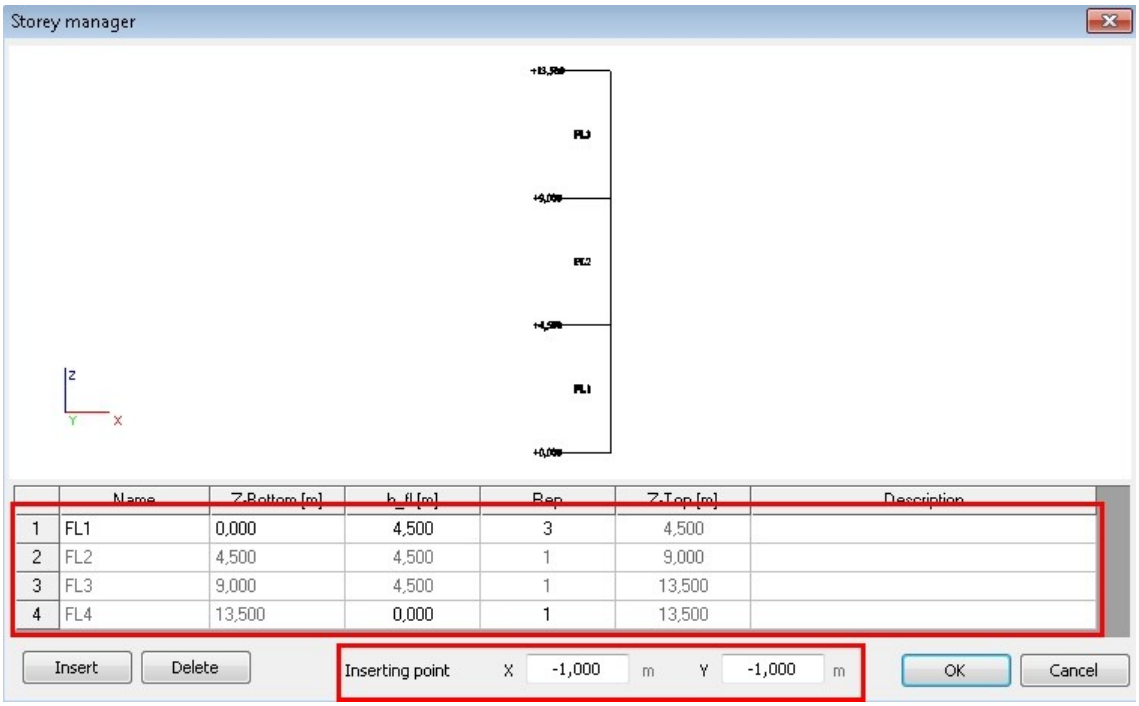
The storey is defined by storey name, its start, its height, number of repetitions and its top which is calculated automatically. User can fill some additional description in the last column.

The preview is updated automatically.

There are two buttons for adding a new row and deleting the existing one.

The inserting point is defined by X and Y coordinates.

The geometry of the storey, name and inserting point can be modified only in the storey manager.



The name of each level is automatically generated according to the first one. The number is added to the end of the string.

Example:

Floor1, Floor2, Floor3 ...

flr_1, flr_2, flr_3, ...

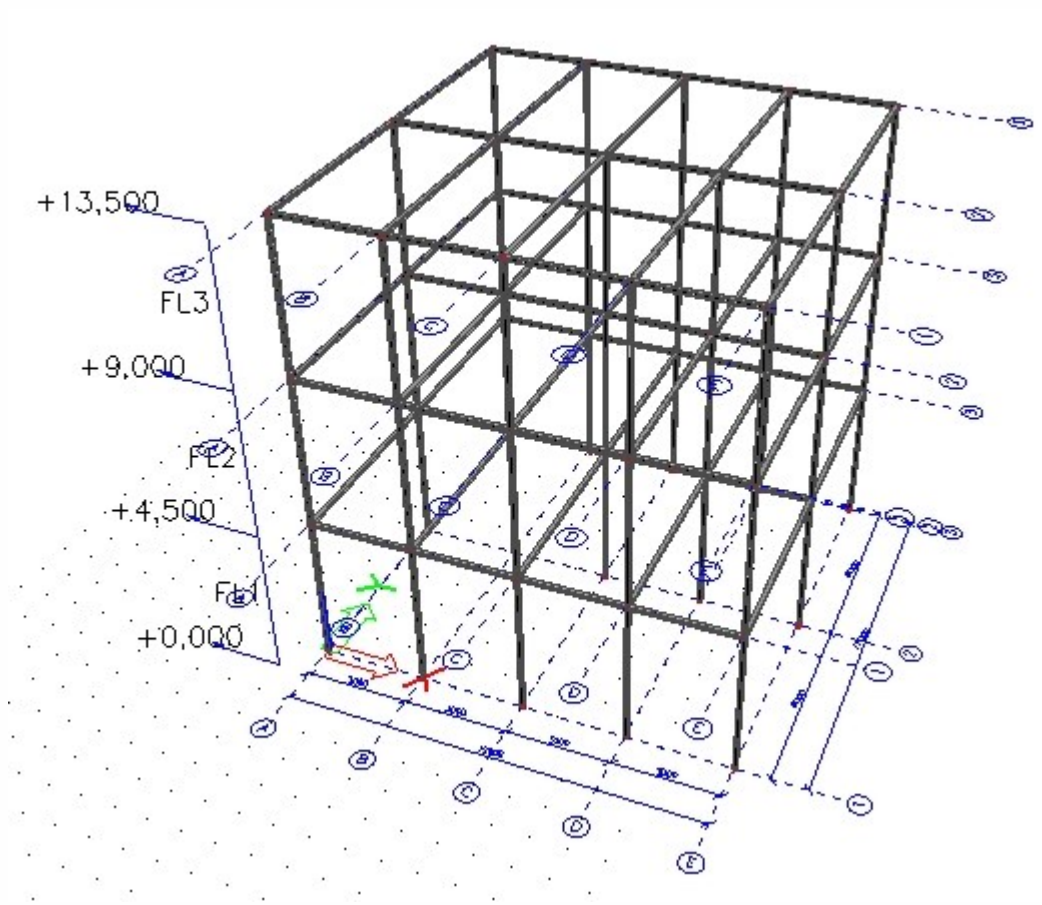
When user confirms the storey by OK, the new object is placed to the inserting point.

SCIA engineer has only one storey manager, so the storey can be only once in the project.

Storeys in the project

The storey is displayed as simple lines with labels in the 3D window. Each storey level is displayed like line with triangle and description – it contains the name and value for height (this is the default settings). User can add more info by the view parameters.

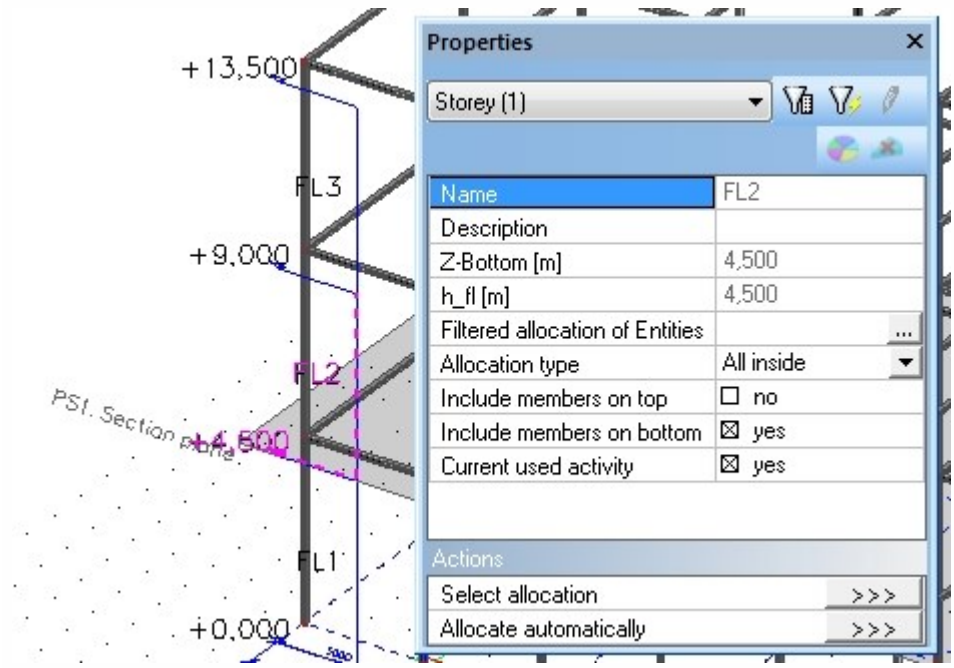
The 3D window with the perspective view on the structure and storeys:



Storeys are reused for seismic calculations.

Properties of storey level

The properties of the selected level are displayed in the property dialogue.



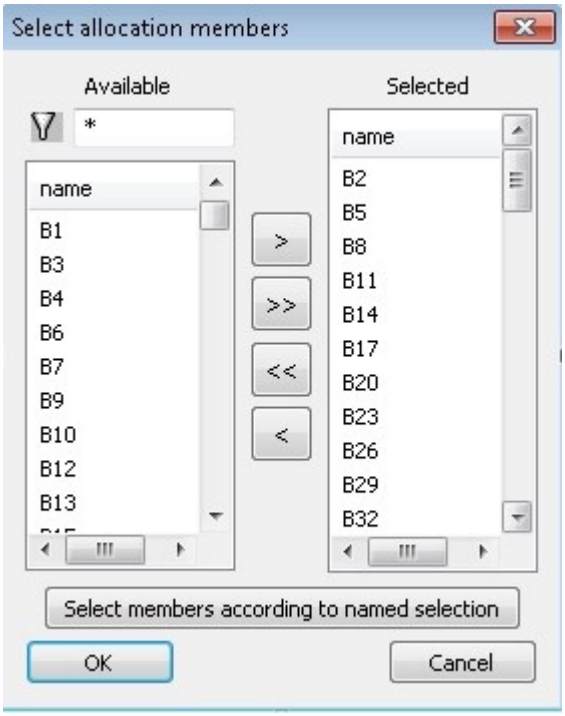
Name
the name of the storey level

Description
the user defined description

Z-bottom
where the storey level starts

h_fl
the height of the storey level

Filtered allocation
the list of the allocated members (to this storey level)



Allocation type: this is important for the automatic allocation, because it defines which members will be allocated, there are three possibilities:

1. **All inside** – the member must be whole inside the storey level (it means between Z-bottom and Z-top) to be automatically allocated
2. **Part inside** – at least a smaller part of the member must be inside the storey level to be automatically allocated
3. **Major part inside** – the bigger part of the member must be inside the storey level to be automatically allocated

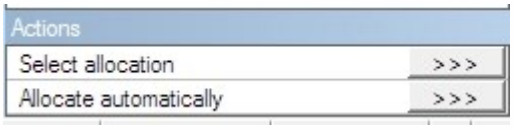
Include members on top: this is important setting for the automatic allocation, it defines if the members on the top border (member with Z coordinate=Z_top) should be allocated

Include members on bottom: this is important setting for the automatic allocation, it defines if the members on the bottom border (member with Z coordinate=Z_bottom) should be allocated

Current used activity
it is checked when the level is used in activity (see a special chapter)

Select allocation
the allocated members are selected in the model by this button

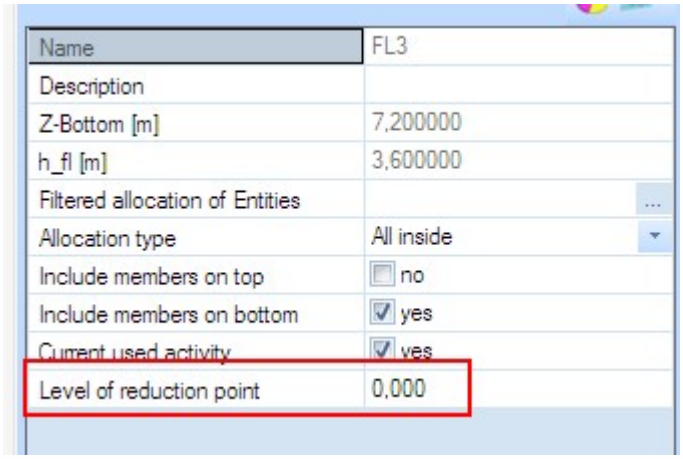
Allocate automatically
the system check the allocation parameters and allocate the members again by this button



When the member is added to the structure where storeys are already defined, the allocation is not updated automatically. User has to use the action button "Allocate automatically".

The special property for Seismic

Level of reduction point
This property is visible when the functionality Dynamics / Seismic is checked. See more about this property here.

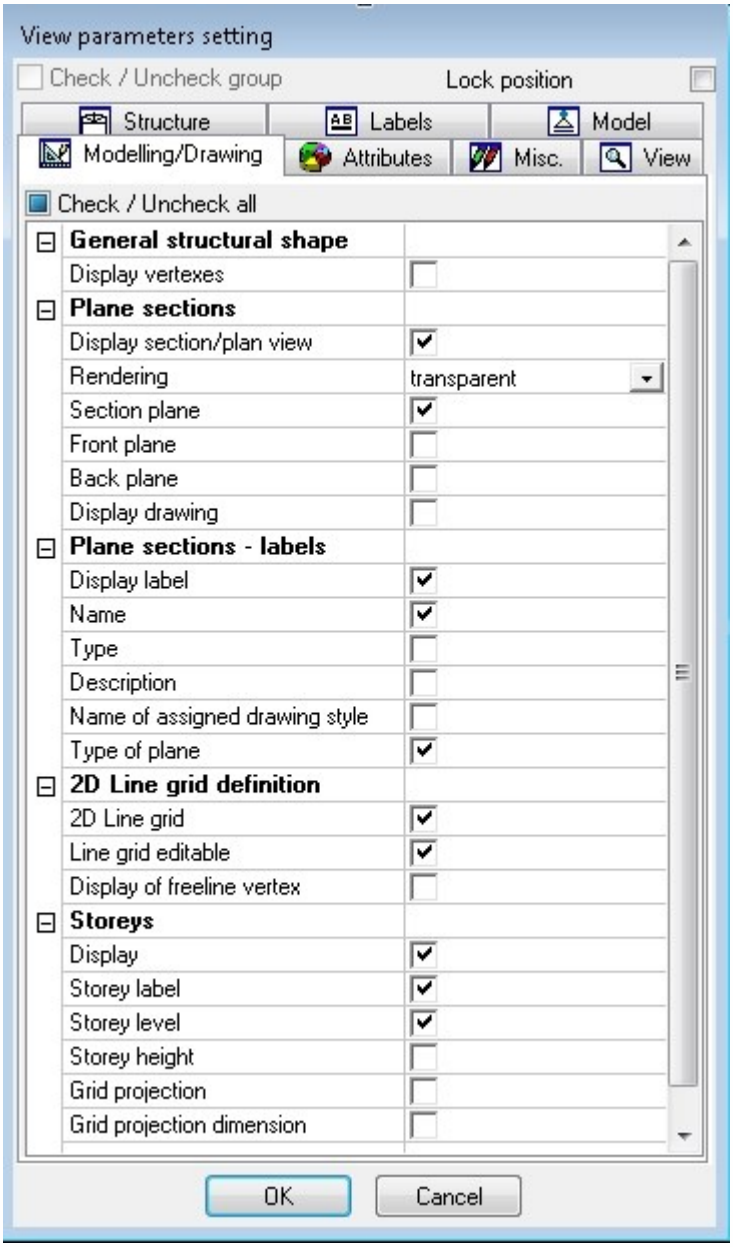


Storey view parameters

The view parameters for storeys are on the tab Modelling/Drawing.

The storeys displays:

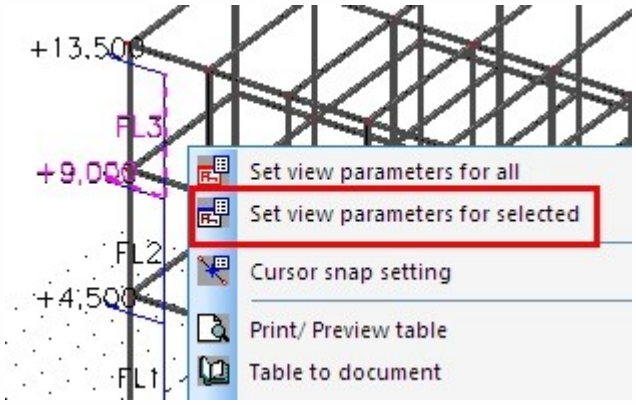
1. Storey label – name of the storey level
2. Storey level – the value from Z-bottom column in the storey manager
3. Storey height – the value h_fl in the storey manager
4. Grid projection – if the 2D linegrid should be projected to the storey level (grid with labels)
5. Grid projection dimension – if the 2D linegrid should be projected also with dimensions, this option can be used only when "Grid projection" is switched on, otherwise it is not visible in the project



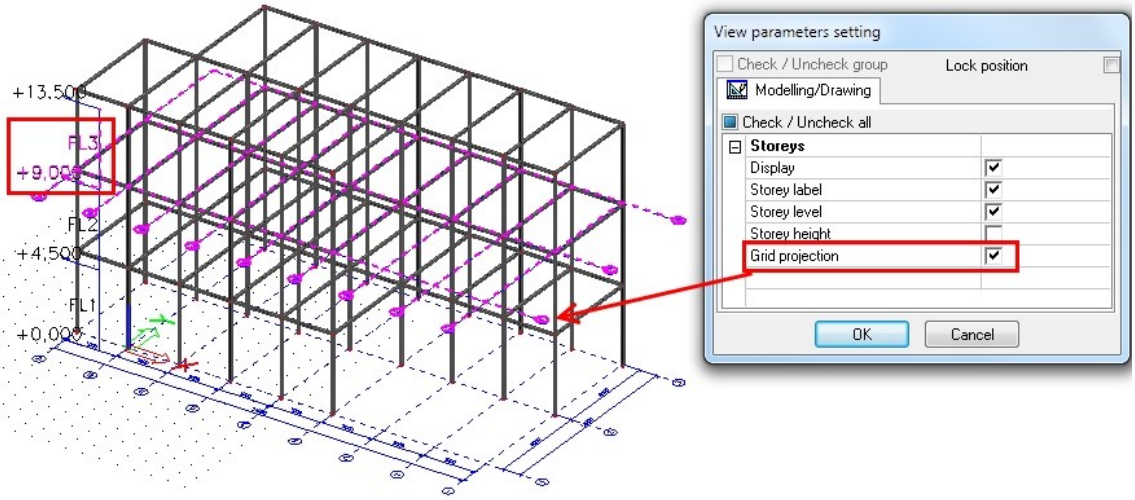
The view parameters for selected entities can be used for storeys as well. If the user wants to project the linegrid only in some storey levels, then he has to select them in the model and use view parameters for selected.

Example:

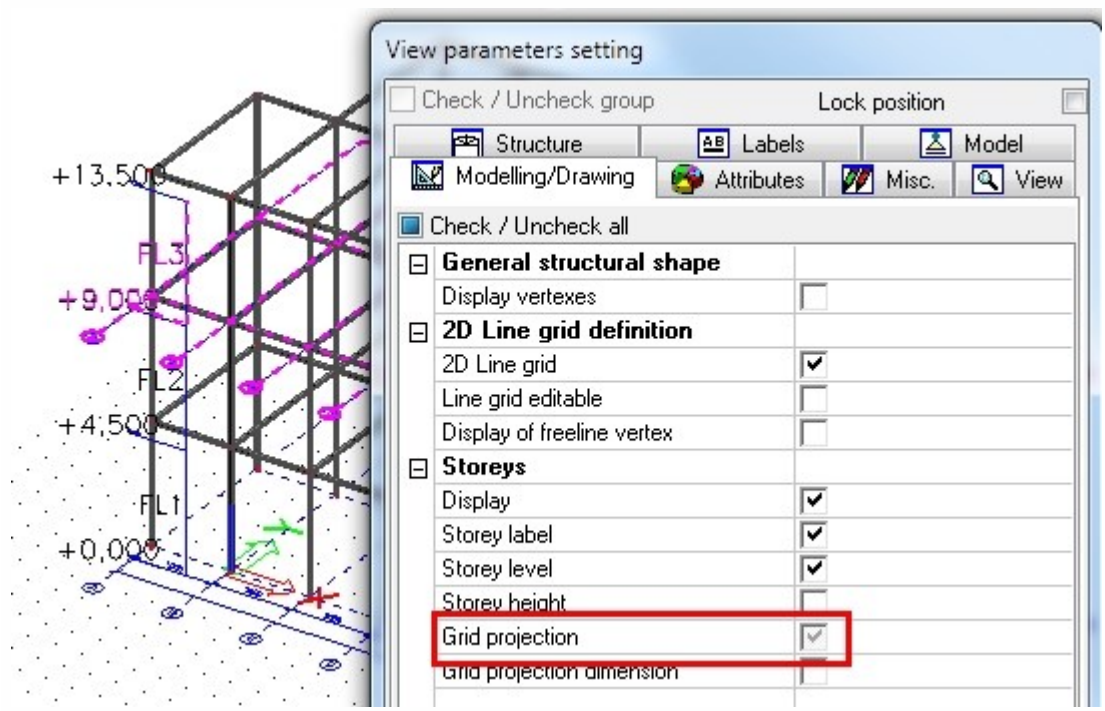
1. User wants to display linegrid on the FL3.
2. He select the storey level FL3 in the model and start view parameters for selected



3. He selects the grid projection in the dialogue.



4. The gray checkbox is displayed in the view parameters for all. That's because it is checked only for the selected part of the model. It can be changed by un-checking it.



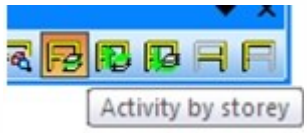
Activity by storey

'Storey activity' is a tool that allows for visualising of the model entities based on storey allocation. Allocated members on a (or multiple) storey(s) can be displayed and edited in the Scia modelling environment while all other entities are hidden.

This activity option is located on the Activity Toolbar and it contains three buttons – 'Activity by storey,' 'Move activity by storey up,' 'Move activity by storey down.'

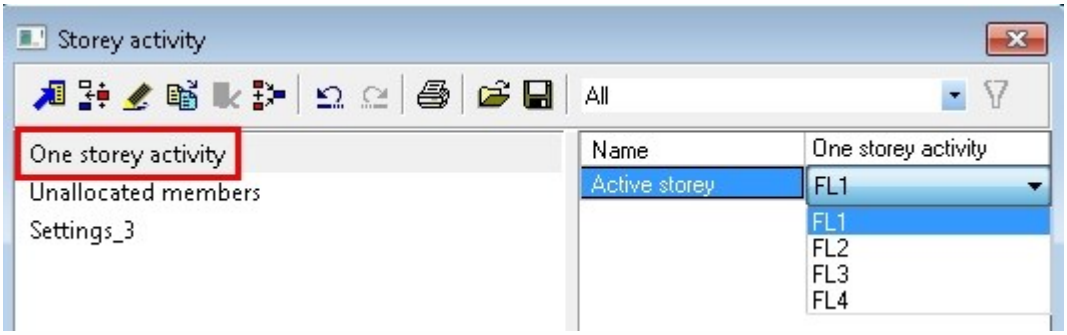
There are three ways to use 'Storey activity.'

The options linked to 'Activity by storey' define which type of activity is used – 'Single storey activity,' 'Unallocated members' or user-defined settings.



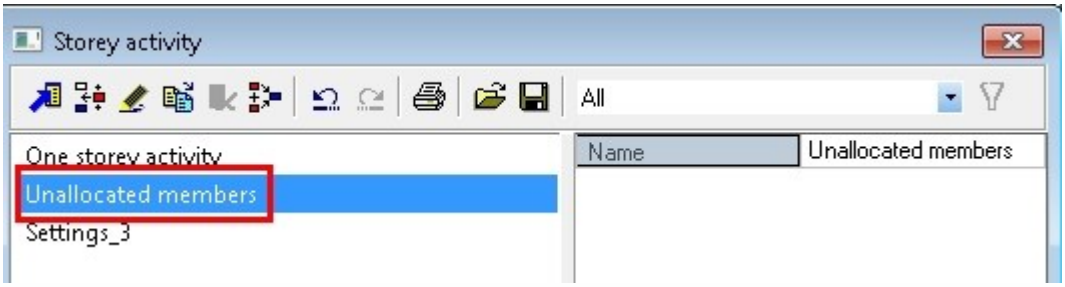
One storey activity

The user can only activate a single storey. All storey levels are available for selection via the combobox.



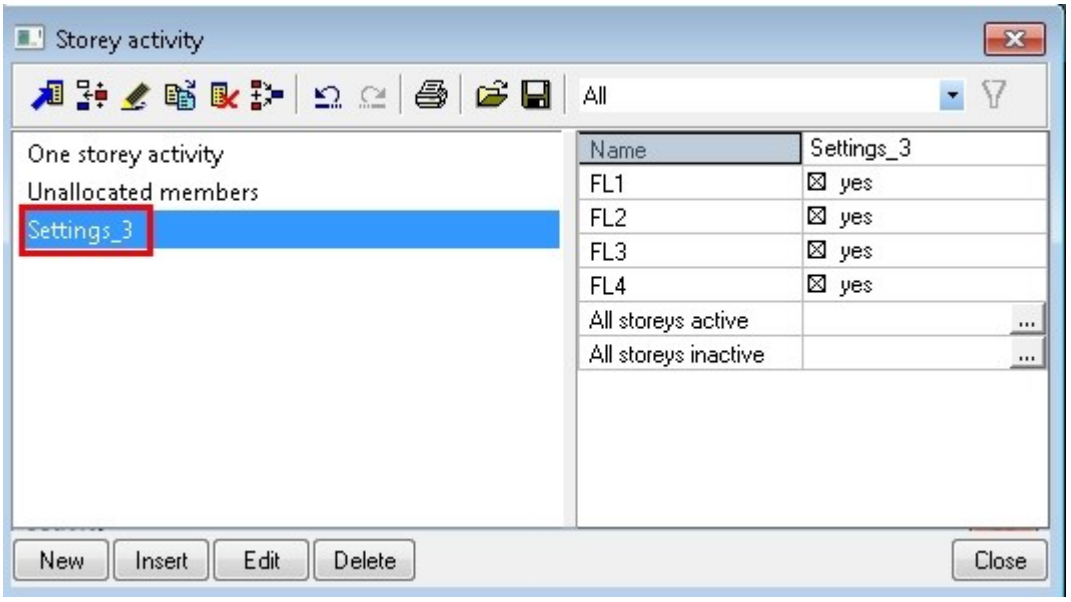
Unallocated members

A model may contain members that are not allocated to any storey. There are also object types which cannot be allocated to any level at all (linegrids, a storey itself, etc.). Those items can be displayed by the option 'Unallocated members.'



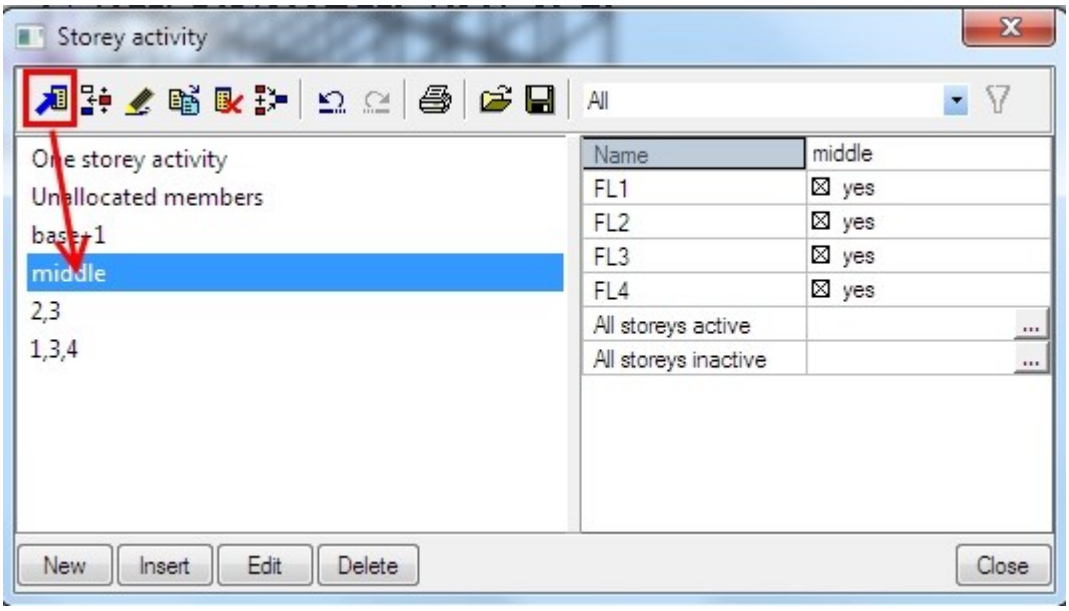
Multiple levels together – Settings

The third option displays selected levels together. This type can be created by user.



The user selects the active levels by checkboxes. All storeys can be activated and deactivated at once by the two buttons at the bottom right corner of the dialogue.

The 'Storey activity' dialogue may contain more user-defined settings to address different needs. New settings can be added by the button 'New' on the toolbar. The names of settings are user-defined.



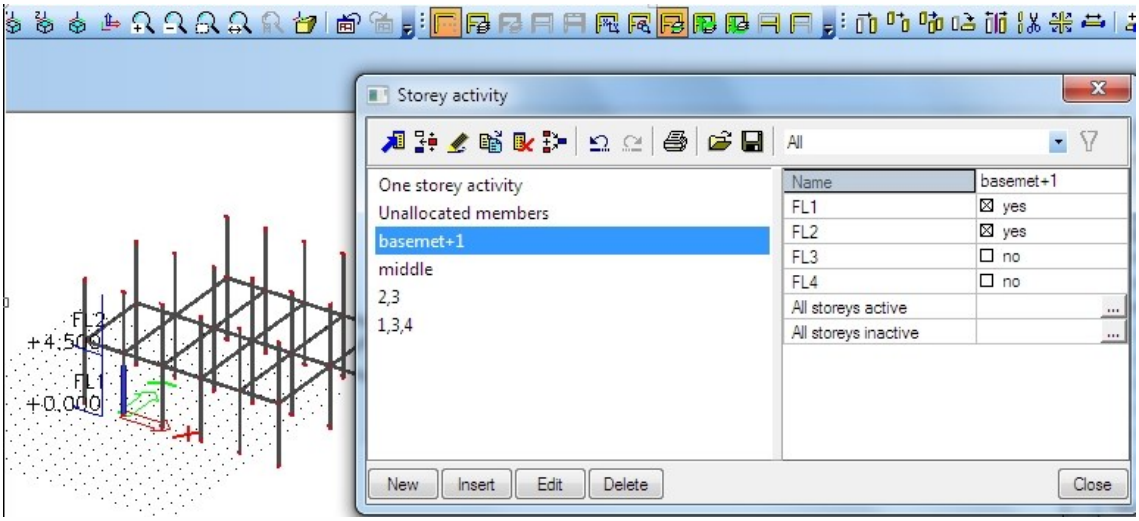
Move activity by storey up/down

The two additional buttons can be used only when the storey activity is selected.

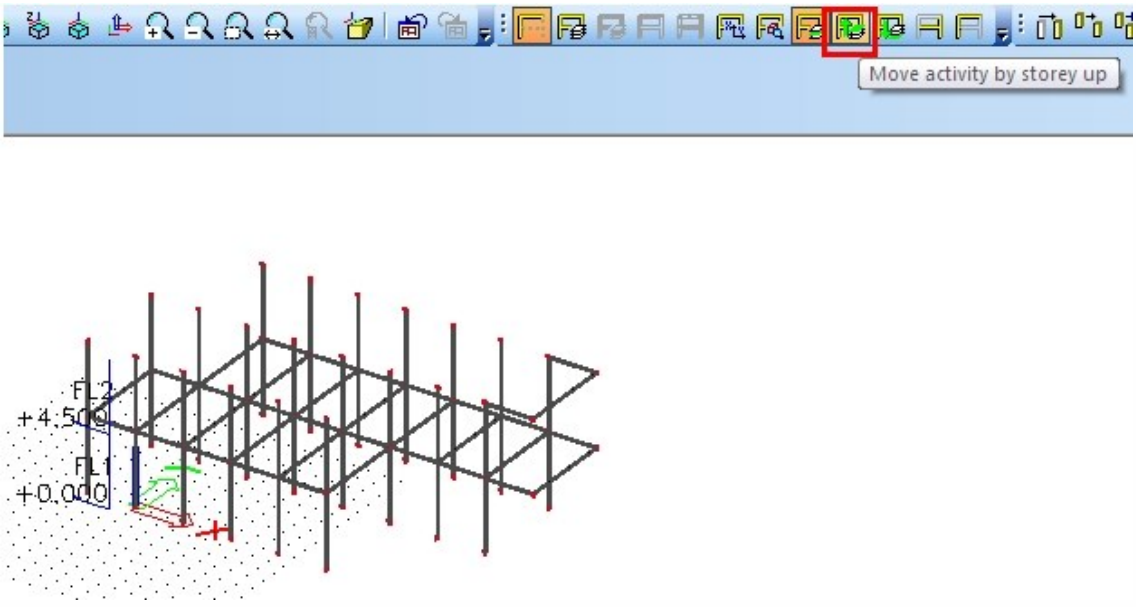
The first one moves the storey activity in the dialogue to the next one, the second to the previous.

Example:

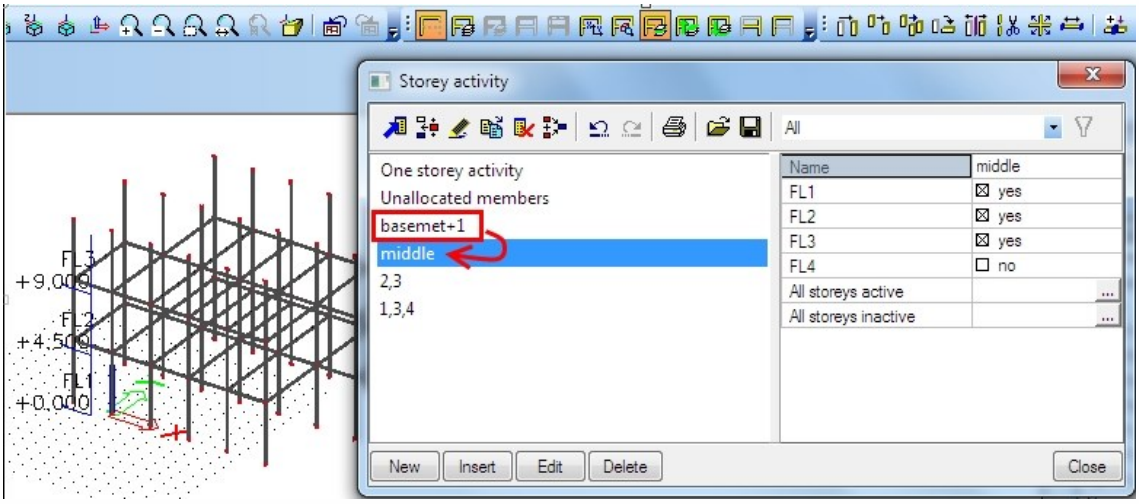
1. The initial setting – "basement+1" is active, thus, only FL1 and FL2 are visible in the 3D window.



2. User presses the "Move activity by storey up".



3. The model is changed. Three levels are now visible – FL1, FL2, and FL3. The active setting in the dialog is changed from "basement+1" to "middle."



Import and export

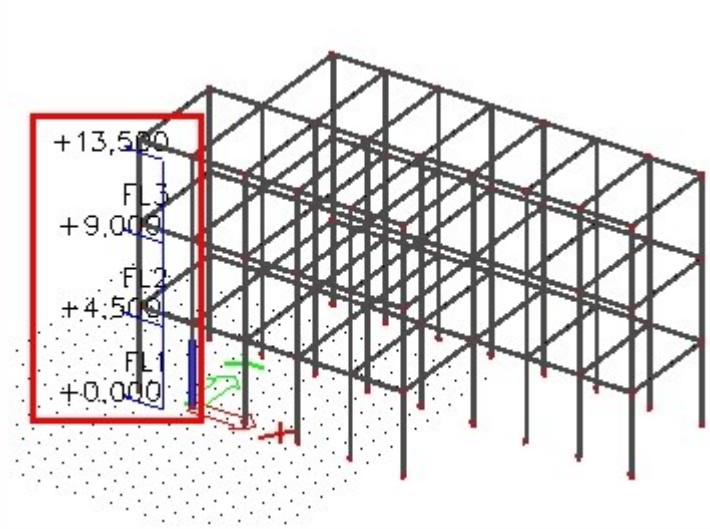
The storey can be only once in the project. This is also the only limitation of the import and export tool.

- If the existing project contains storeys -> the newly imported project is imported without storeys.
- If the existing project doesn't contain storeys -> the newly imported project is imported with storeys.

The inserting point for the imported storeys is placed correctly with the respect to the construction.

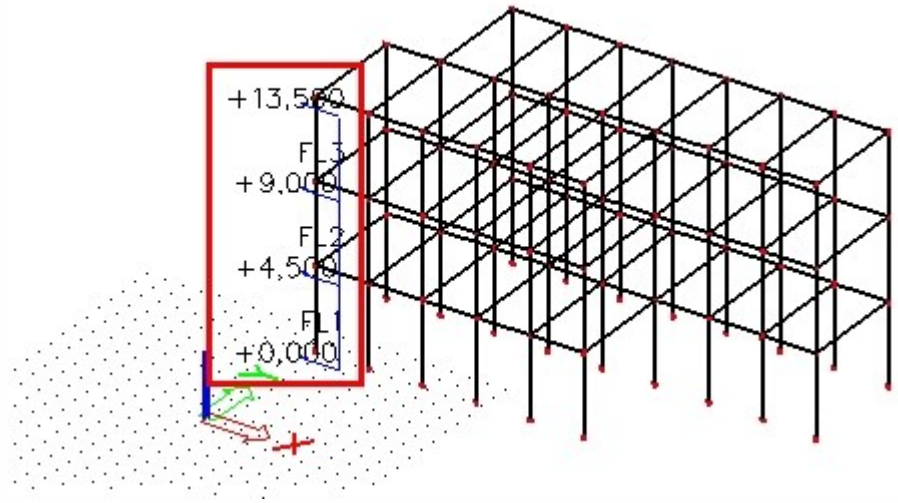
Project which will be imported:

- Storeys are placed next to the structure, GCS is under the second column.



Project imported to an empty project:

- Storeys are placed again next to the structure, GCS is located farther, because the inserting point for import wasn't set to 0,0,0.



How to define new Storeys

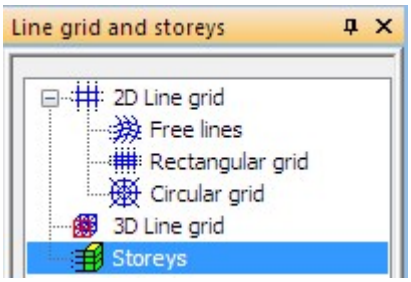
Introduction to Storeys

Storeys is a special entity in SCIA Engineer. It is defined by "Storey manager" dialogue. The Storeys object can be only once in the project. Any changes can be done in "Storey manager".

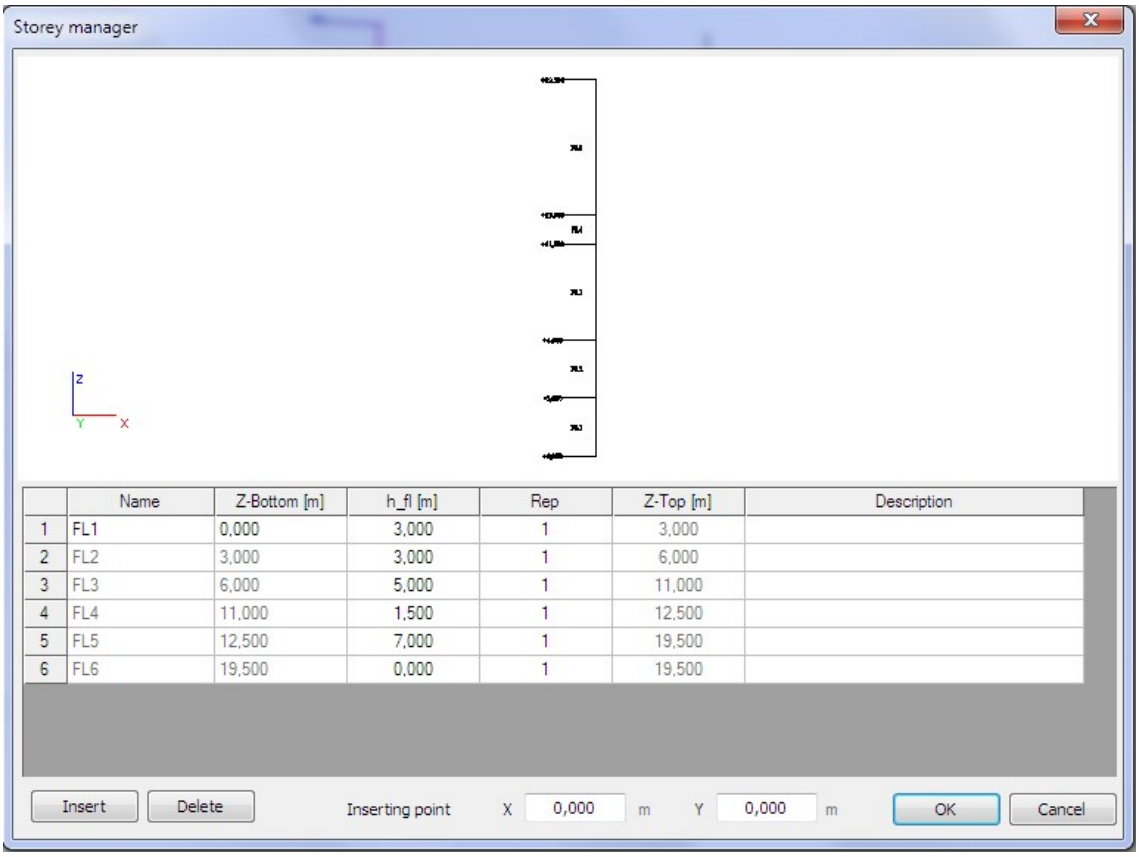
Storeys can be used with 2D linegrid. Together they create similar object as 3D linegrid. A visual copy of a 2D linegrid in each Storey is controlled by view parameters.

Location of Storeys in SCIA Engineer

Storeys dialogue can be started from service "Line grid and storeys".



Dialogue "Storey manager" starts. Each storey can be defined in an ordinary grid.

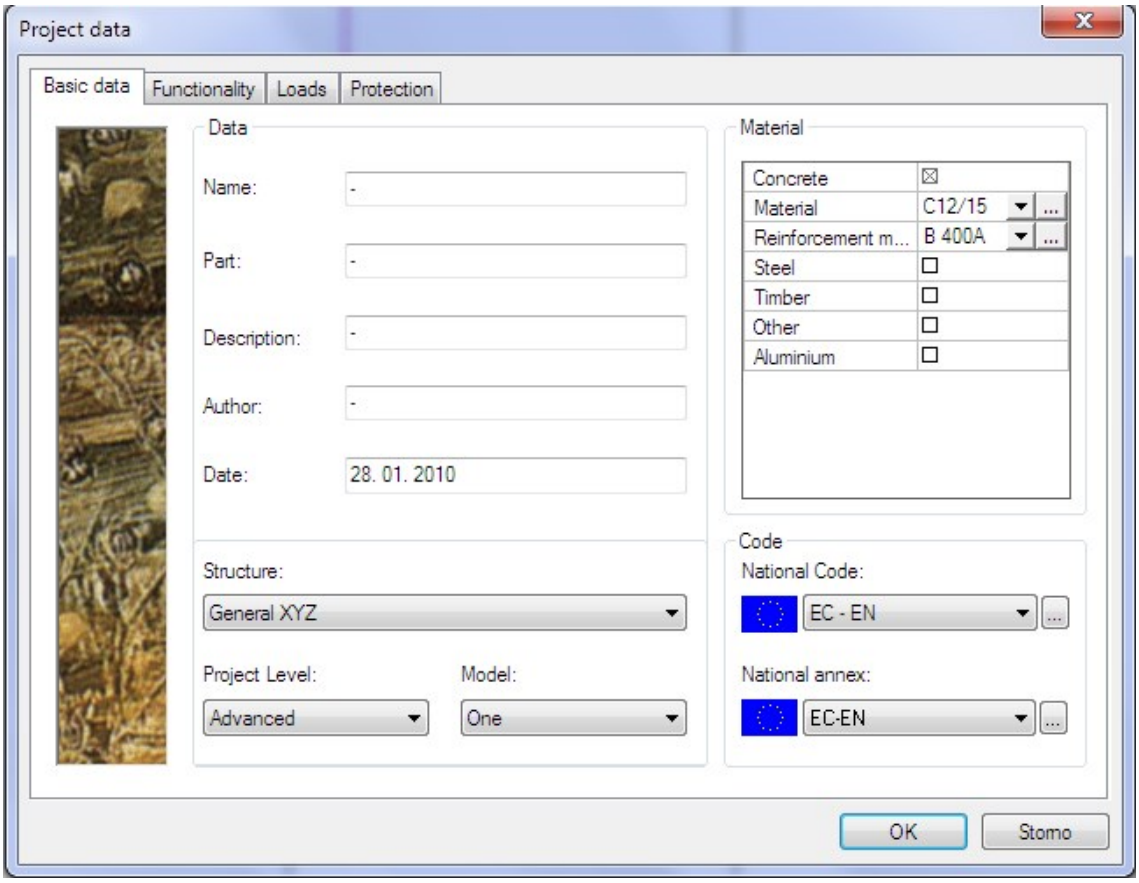


	Name	Z-Bottom [m]	h_fl [m]	Rep	Z-Top [m]	Description
1	FL1	0,000	3,000	1	3,000	
2	FL2	3,000	3,000	1	6,000	
3	FL3	6,000	5,000	1	11,000	
4	FL4	11,000	1,500	1	12,500	
5	FL5	12,500	7,000	1	19,500	
6	FL6	19,500	0,000	1	19,500	

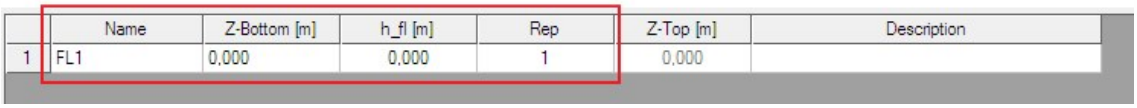
How to define Storeys

1. Open a new blank project in SCIA Engineer with this settings:
   i. Structure – General XYZ
   ii. Project level – Advanced
   iii. Model – One
   iv. Material – Concrete
   v. National code and annex – EN

No more functionality is needed for the Storeys input.

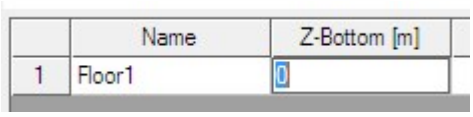


2. Find the command Storeys and start dialogue Storey manager.



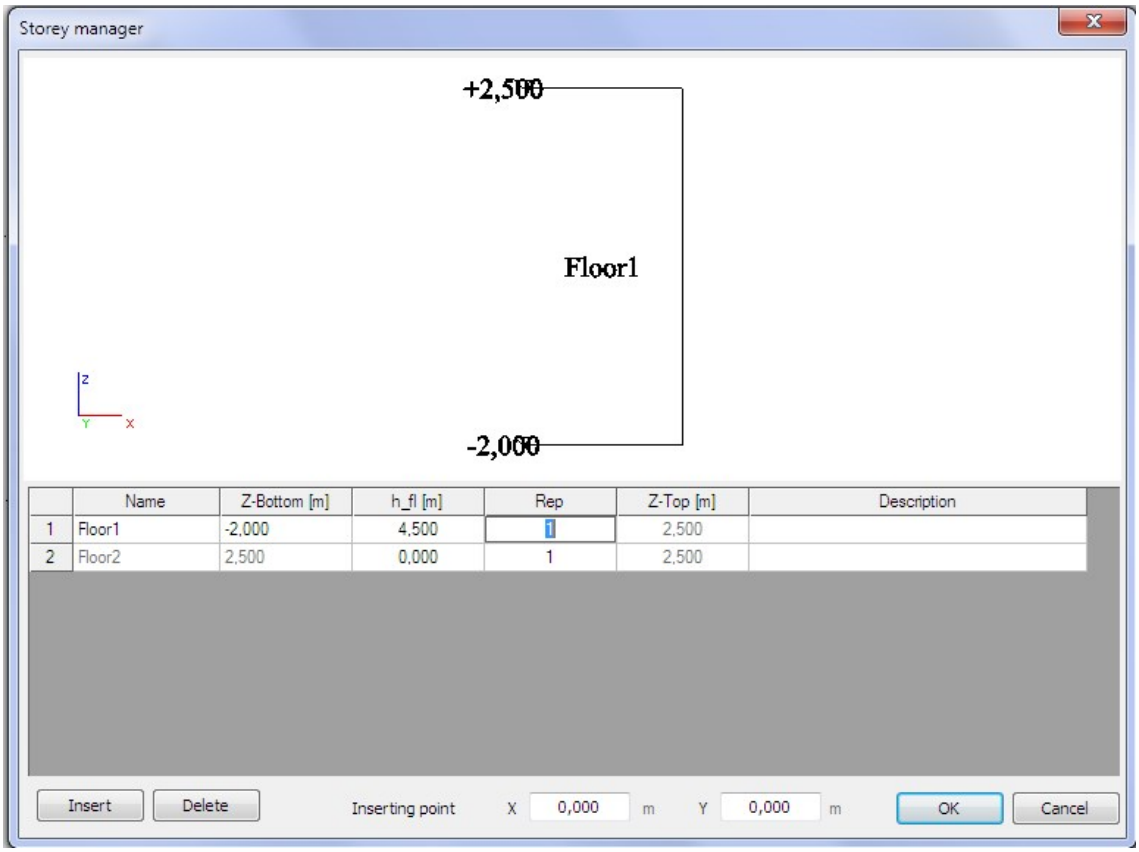
	Name	Z-Bottom [m]	h_fl [m]	Rep	Z-Top [m]	Description
1	FL1	0,000	0,000	1	0,000	

3. Define a new name for floors. Write down "Floor" in the column for the name. It is automatically renamed to "Floor1".



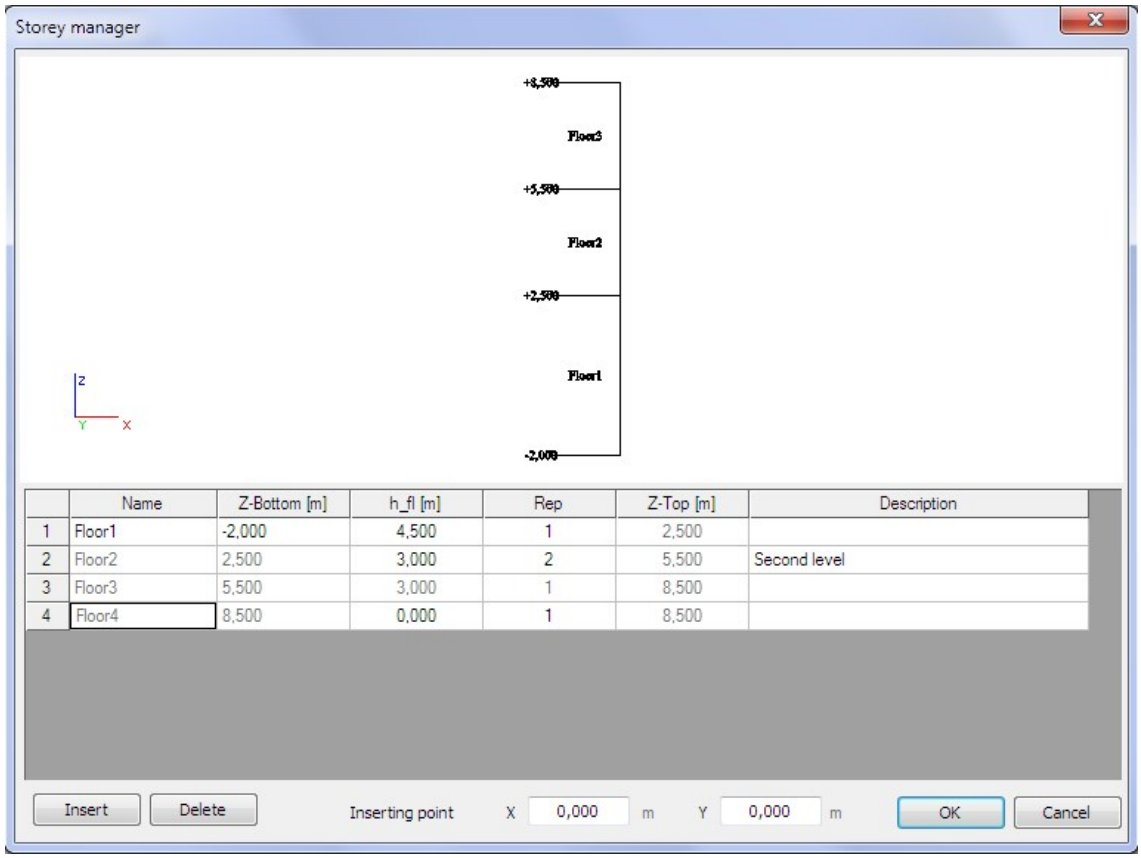
	Name	Z-Bottom [m]
1	Floor1	0

4. Define "Z-bottom" to -2.000m and "h_fl" to 4.500m. "Rep" stays 1.



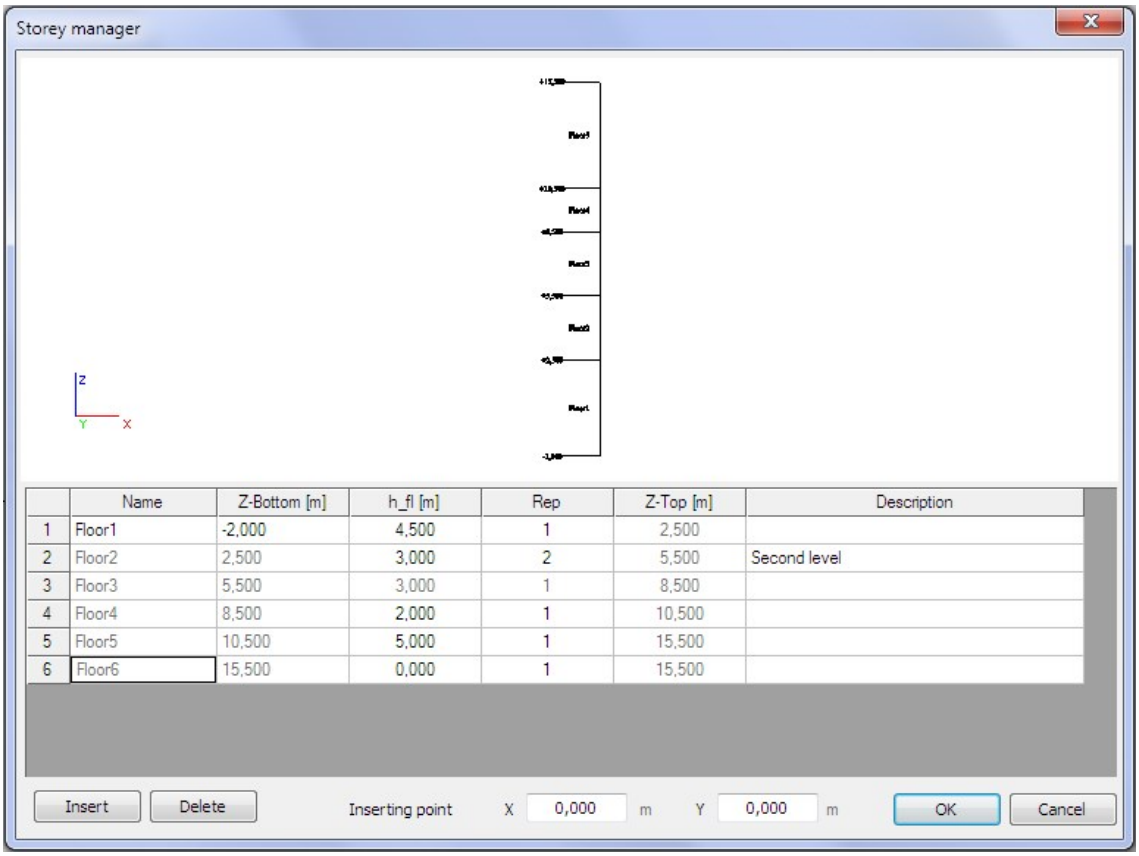
There is a scheme of Storeys in the picture above. One storey is defined.

5. Define another row = next level with these settings:
   i. "h_fl" = 3.000
   ii. "Rep" = 2
   iii. "Description" = Second level



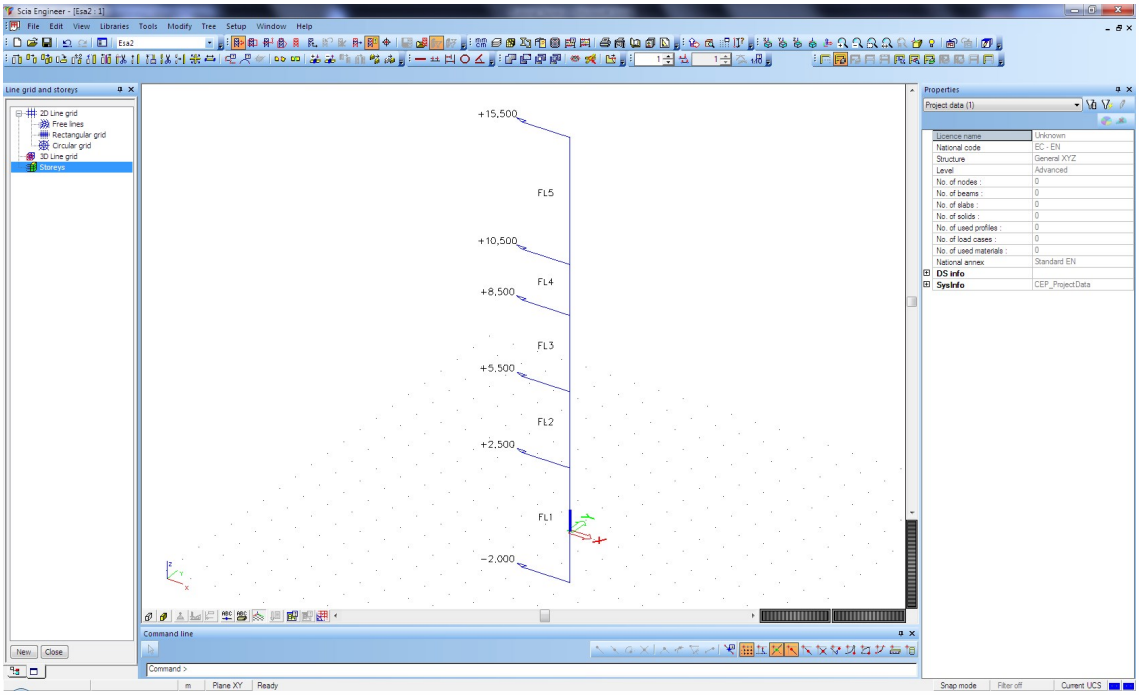
The preview is automatically updated according to the changes in the dialogue.

6. Add 2 more levels with height 2.000 and 5.000m.



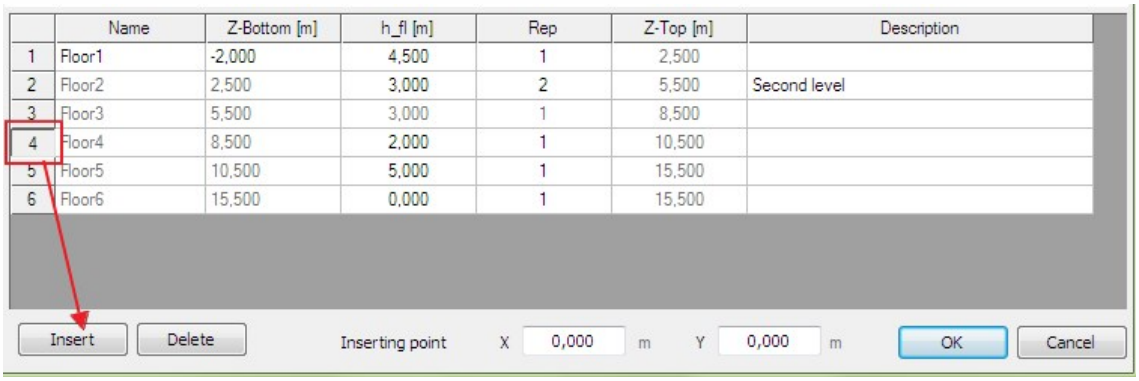
	Name	Z-Bottom [m]	h_fl [m]	Rep	Z-Top [m]	Description
1	Floor1	-2,000	4,500	1	2,500	
2	Floor2	2,500	3,000	2	5,500	Second level
3	Floor3	5,500	3,000	1	8,500	
4	Floor4	8,500	2,000	1	10,500	
5	Floor5	10,500	5,000	1	15,500	
6	Floor6	15,500	0,000	1	15,500	

7. Confirm the Storeys by OK and see the result in the 3D window.

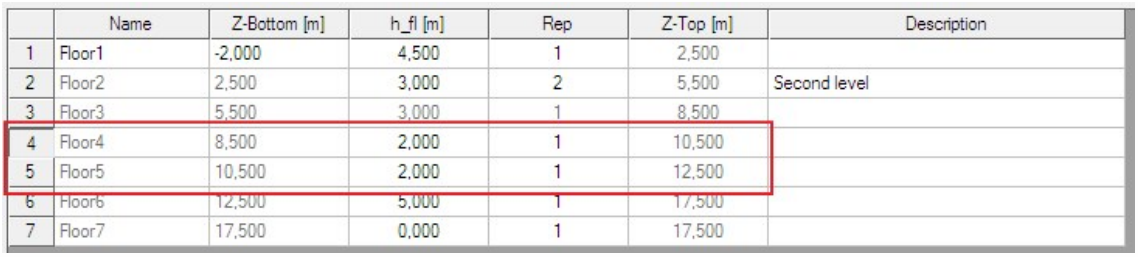


Modification of Storeys

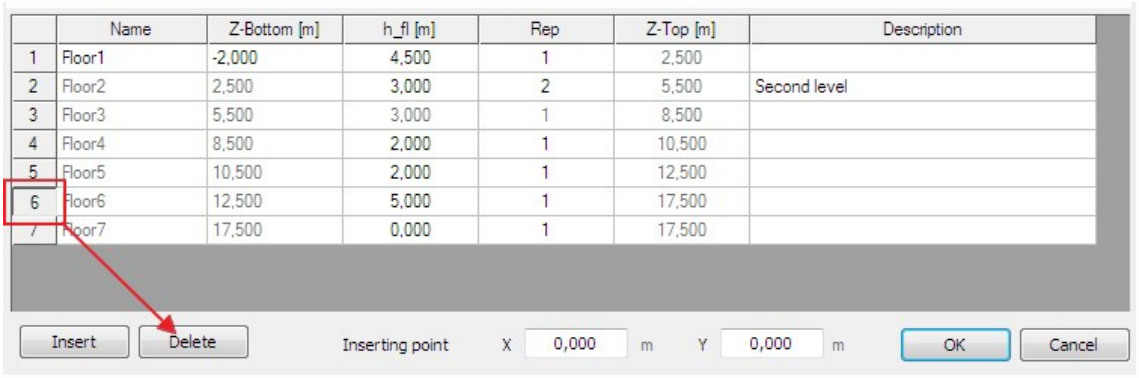
1. Storeys can be edited only in the "Storey dialogue". Start this dialogue again.
2. Buttons "Insert" and "Delete" modify the number of rows in the dialogue. Click on row number 3 and then on the "Insert" button.



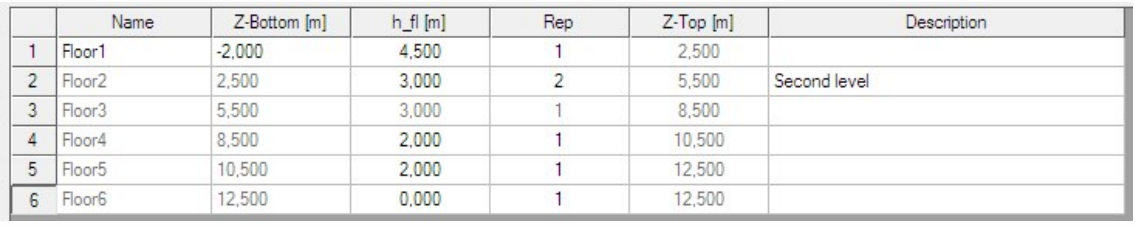
There is a new row with default values from row number 4.



3. Click on the row with number 6 and push button "Delete".



The 6th row is deleted from the grid.



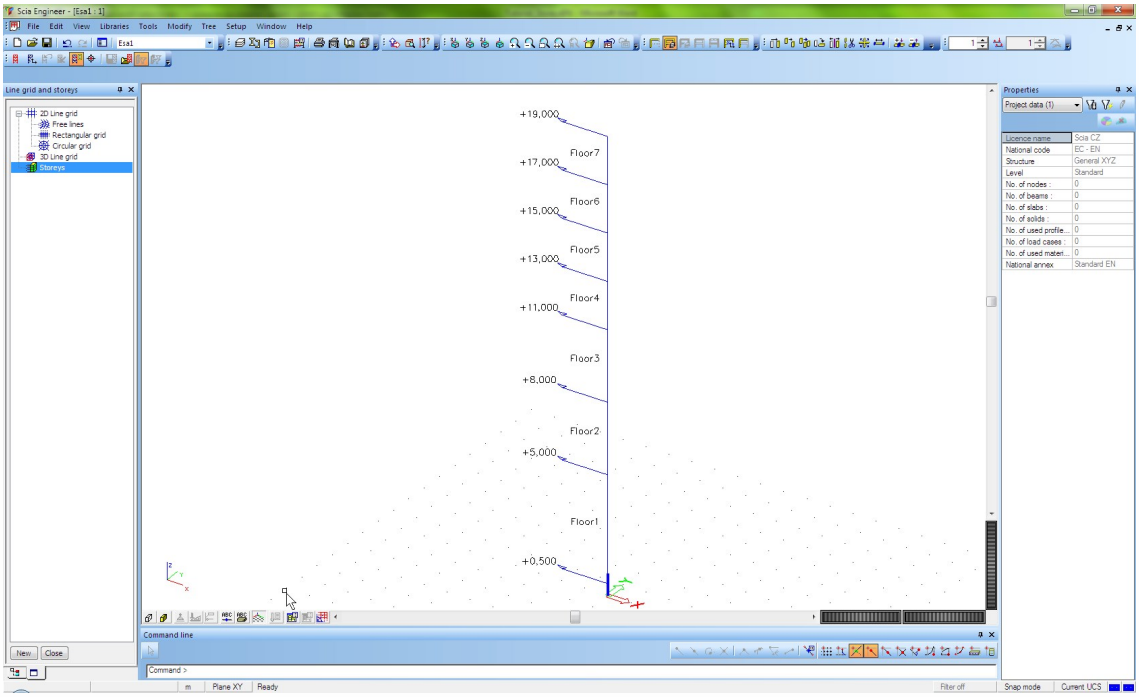
4. Edit value for "Z-bottom" from -2.000 to -0.500. All values under this cell are automatically recalculated.

	Name	Z-Bottom [m]	h_fl [m]	Rep	Z-Top [m]	Description
1	Floor1	-0,5	4,500	1	5,000	
2	Floor2	5,000	3,000	2	8,000	Second level
3	Floor3	8,000	3,000	1	11,000	
4	Floor4	11,000	2,000	1	13,000	
5	Floor5	13,000	2,000	1	15,000	
6	Floor6	15,000	0,000	1	15,000	

5. Edit value "Rep" in 4th row from 1 to 3.

	Name	Z-Bottom [m]	h_fl [m]	Rep	Z-Top [m]	Description
1	Floor1	0,500	4,500	1	5,000	
2	Floor2	5,000	3,000	2	8,000	Second level
3	Floor3	8,000	3,000	1	11,000	
4	Floor4	11,000	2,000	3	13,000	
5	Floor5	13,000	2,000	1	15,000	
6	Floor6	15,000	2,000	1	17,000	
7	Floor7	17,000	2,000	1	19,000	
8	Floor8	19,000	0,000	1	19,000	

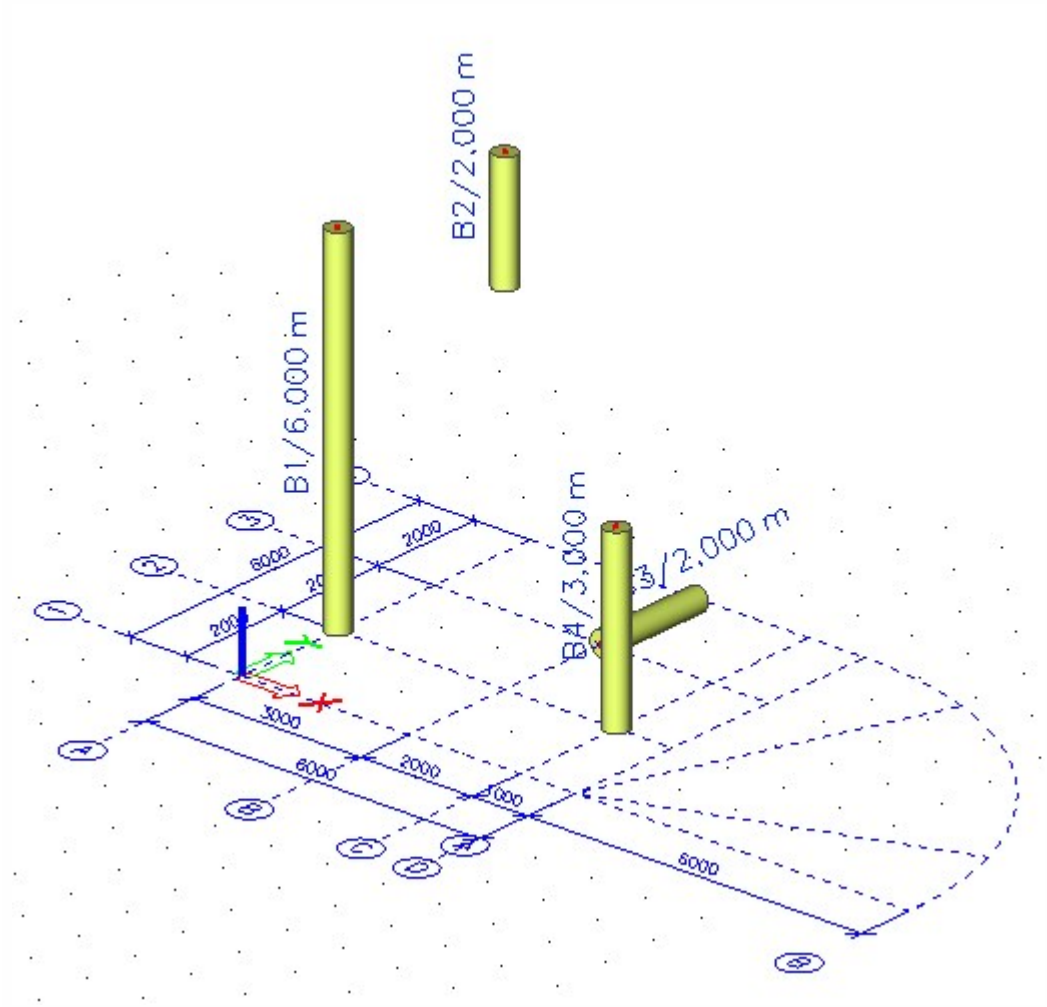
6. See the result in the 3D window.



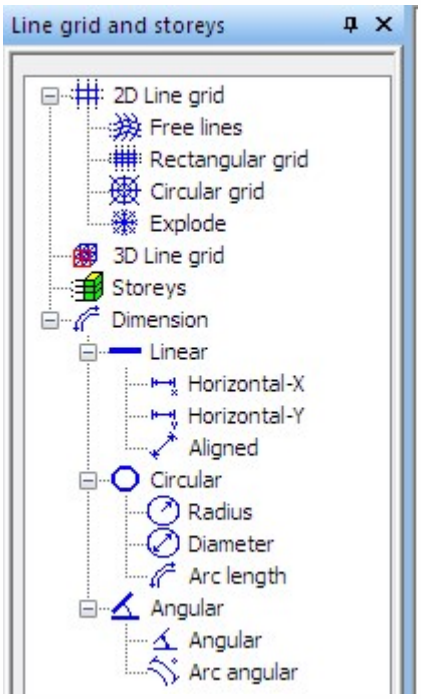
7. Result is also saved in project "storeys_final.esa".

Storeys and Linegrid in one project.

1. Open project "storeys+linegrid_start.esa".

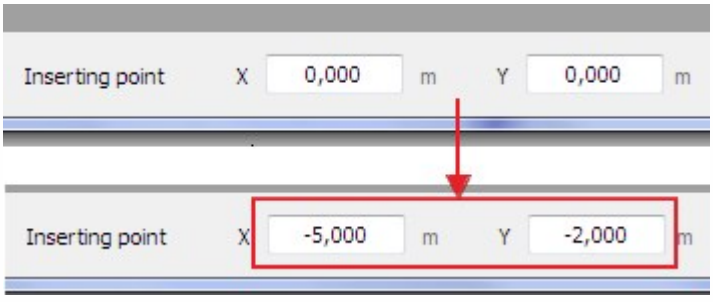


2. Open service "Line grid and storeys".

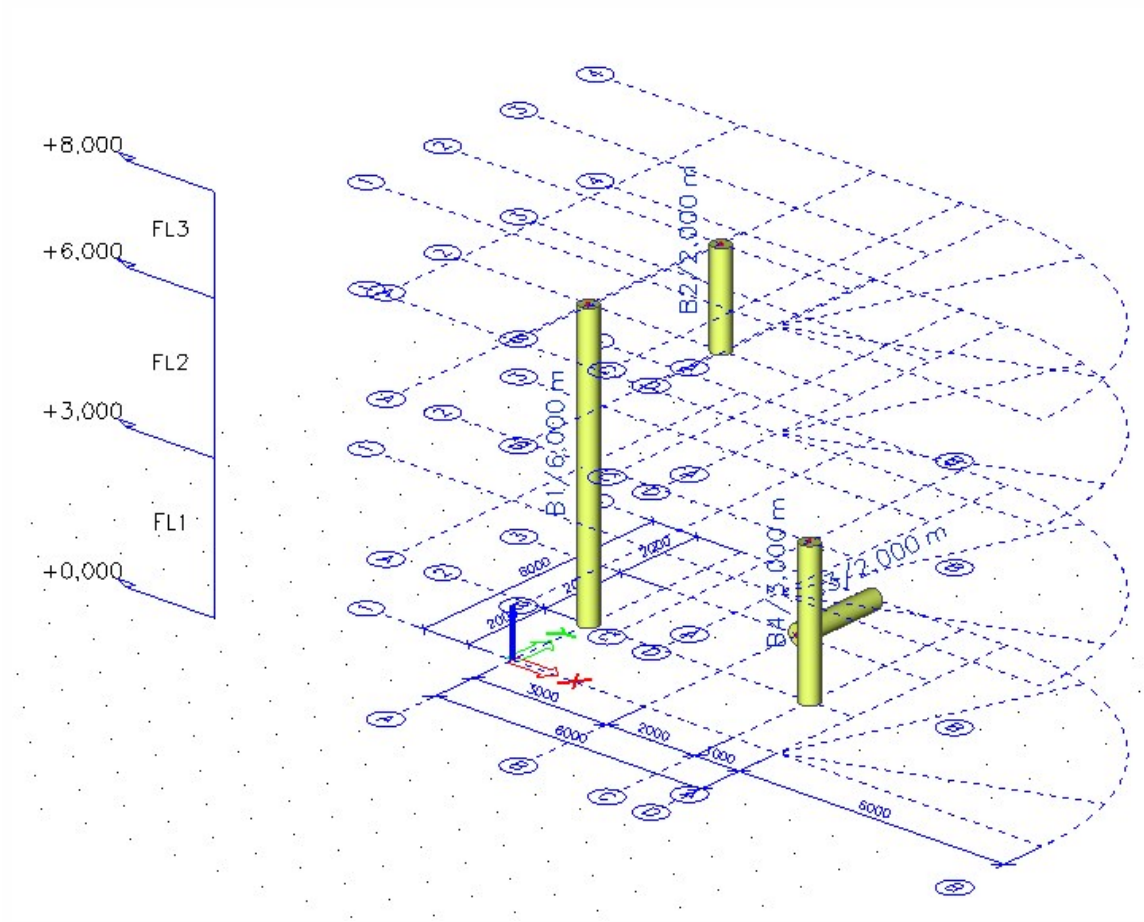


3. Define Storeys with these settings:
   i. "h_fl" in 1st row = 3, "Rep" = 2
   ii. "h_fl" in 3rd row = 2, "Rep" = 1
   iii. Inserting point is x=-5.000, y=-2.000

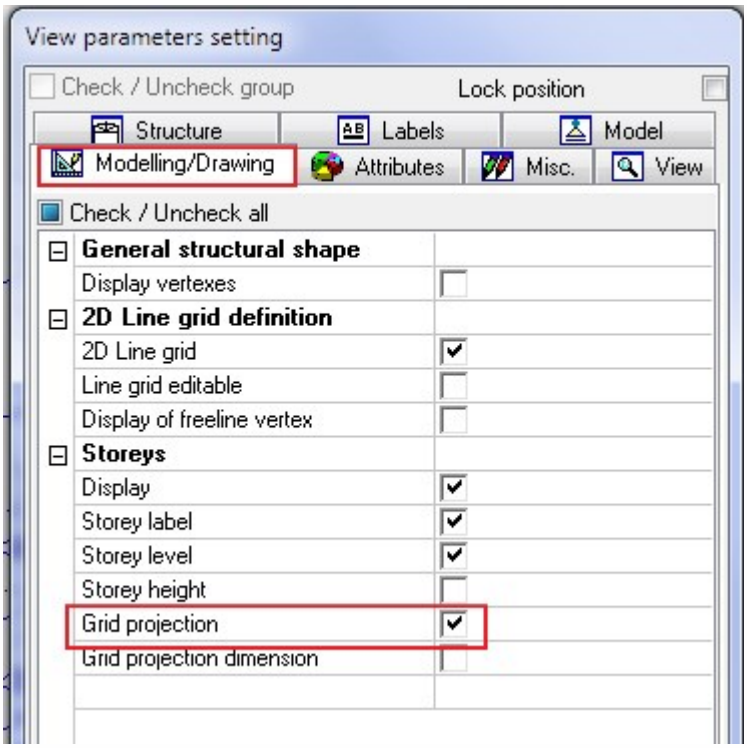
	Name	Z-Bottom [m]	h_fl [m]	Rep	Z-Top [m]	Description
1	FL1	0,000	3,000	2	3,000	
2	FL2	3,000	3,000	1	6,000	
3	FL3	6,000	2,000	1	8,000	
4	FL4	8,000	0,000	1	8,000	



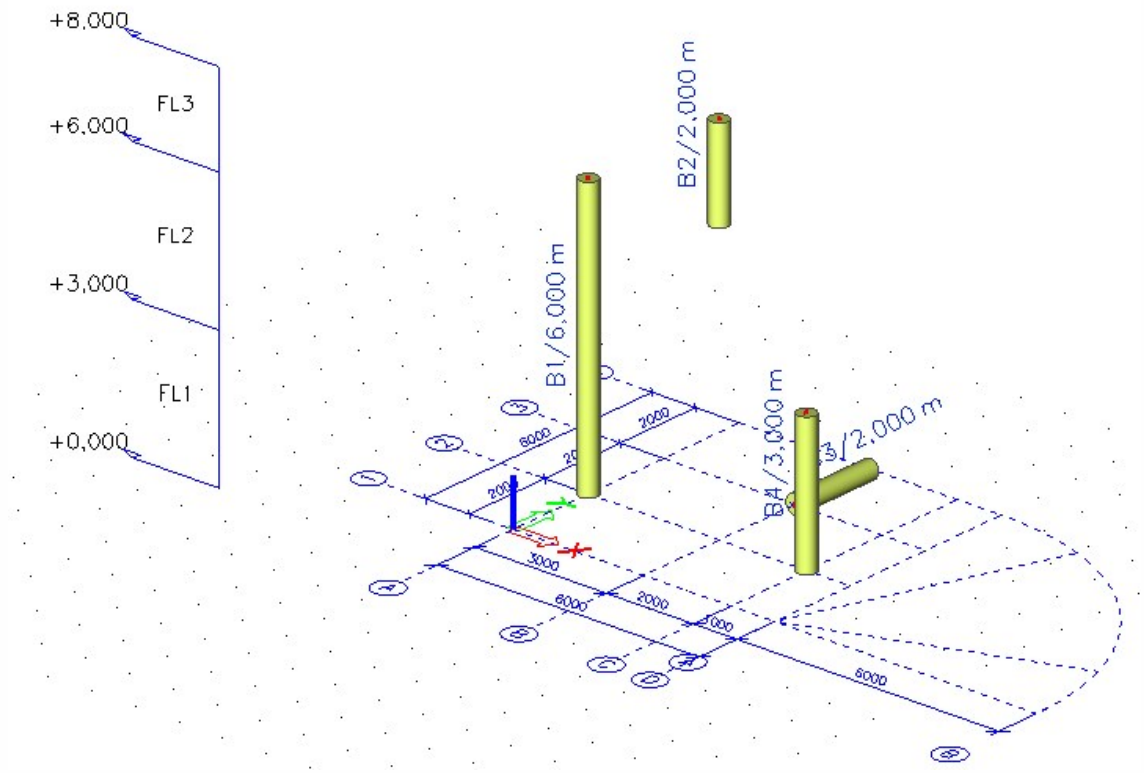
4. Close "Line grid and Storeys" service and see the changes in the 3D model. The 2D linegrid is copied to the each level of Storeys.



5. Change view parameters for the whole structure. Tab "Modelling/Drawing", item "Grid projection" -> uncheck the check-box and confirm it by OK button.



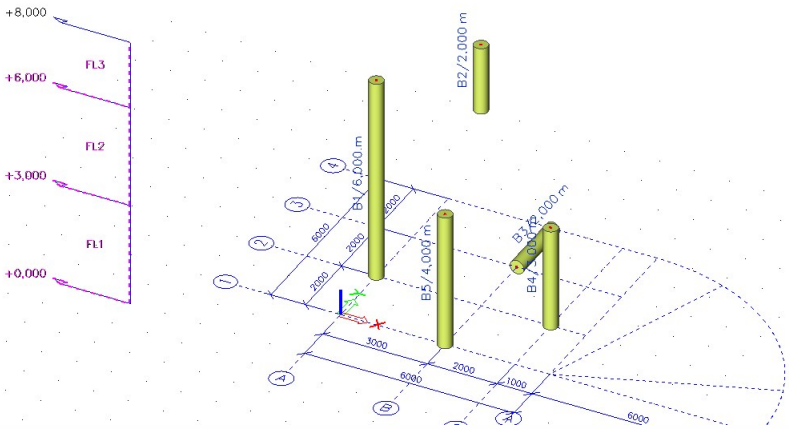
The 2D linegrid is displayed only in the working plane.



6. The result is in file “storeys+linegrid_final.esa”.

How to use storey activity

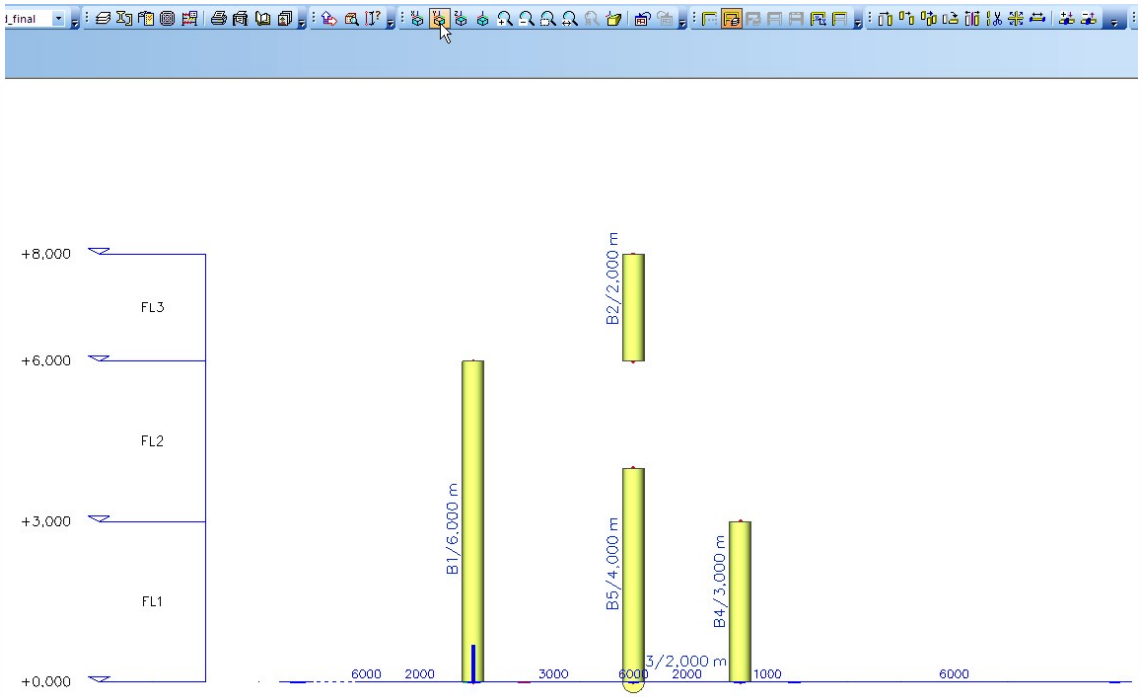
1. Open file “storeys+linegrid_final.esa”.
2. Graphically select all three storeys in the project and set their properties to “Include members on top” and “Include members on bottom” as checked. Then click action button “Allocate automatically”.



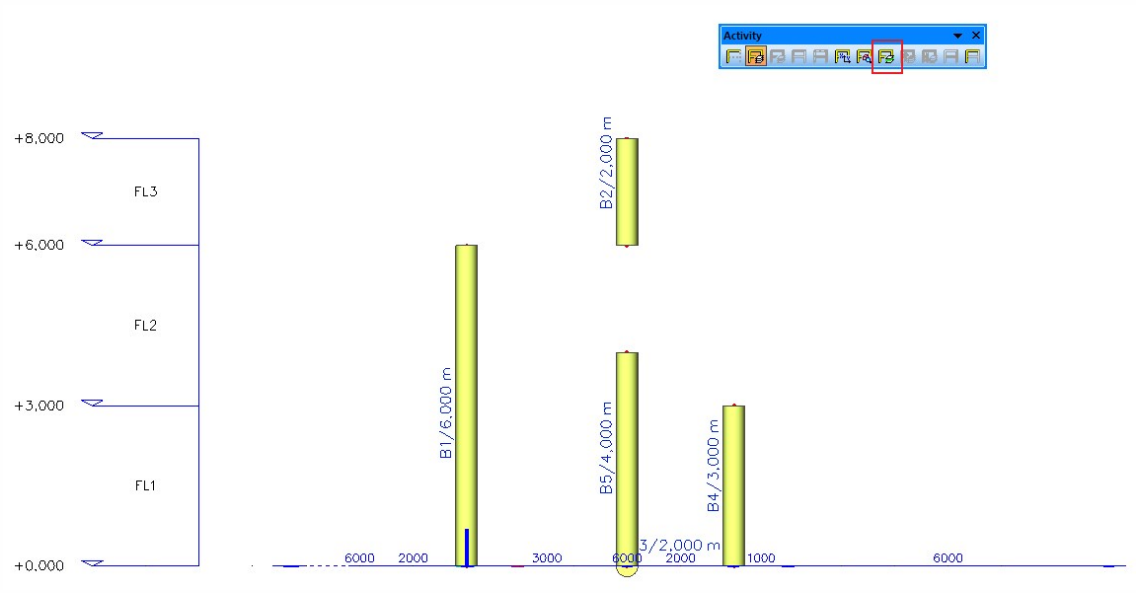
3. Deselect all storeys and now select only the second level. Set property "Allocation type" to Part inside. Click action button "Allocate automatically".

Name	FL2
Description	
Z-Bottom [m]	3,000
h_fl [m]	3,000
Filtered allocation of E...	...
Allocation type	Part inside
Include members on top	☒ yes
Include members on b...	☒ yes
Current used activity	☒ yes

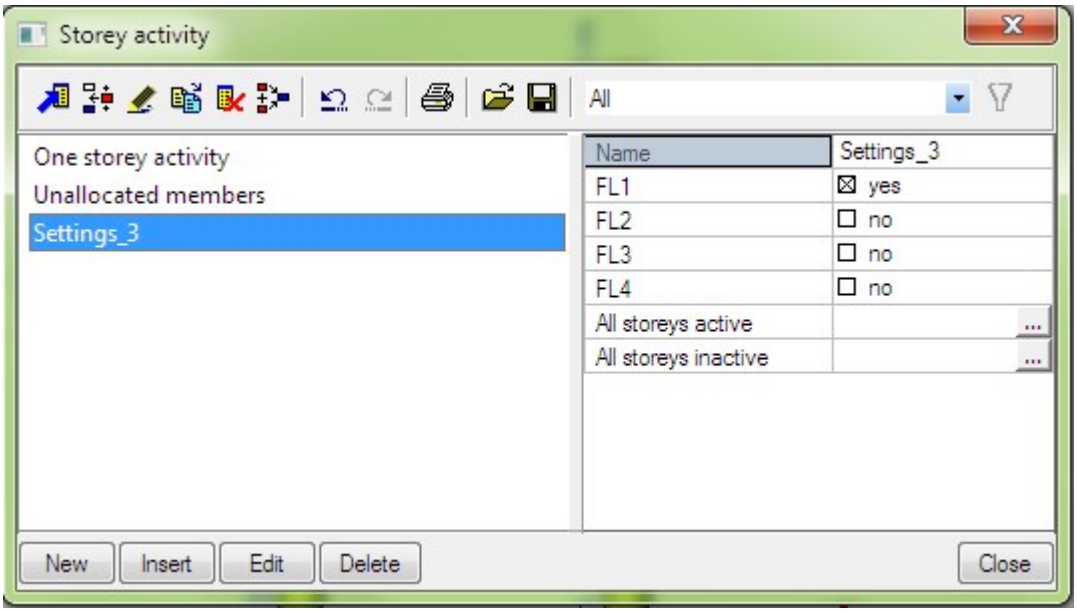
4. Use View Y.



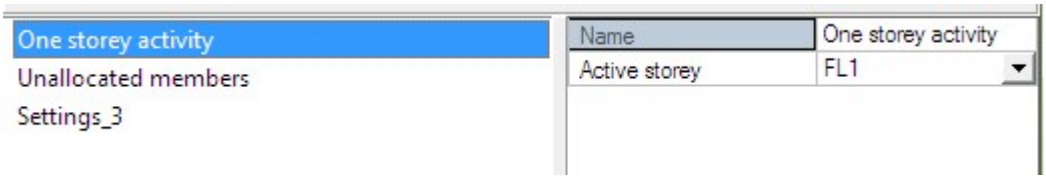
5. Switch activity to Activity by storey. Confirm the warning.



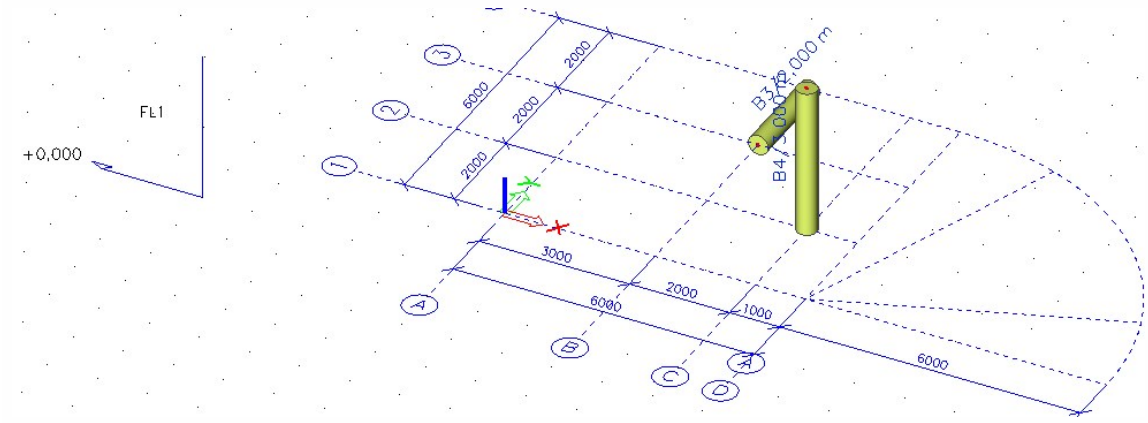
6. The new dialogue is opened. All possible activities that can be used for storeys are here.



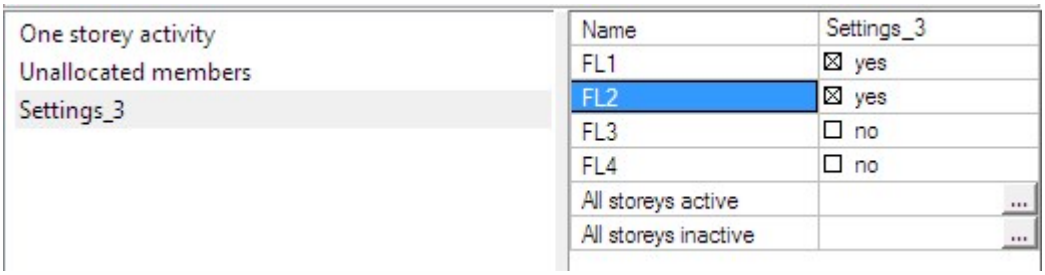
7. Select "One storey activity" and there only 1st level – FL1. Close the dialogue.



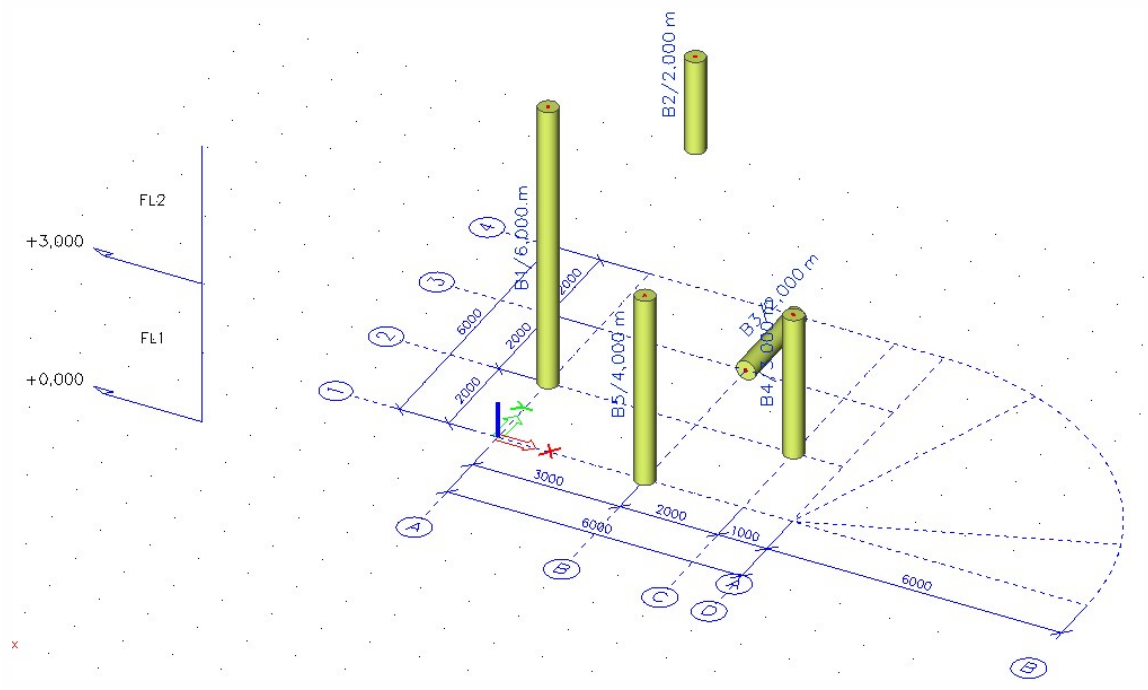
The result in the 3D window:



8. Go again to the "Storey activity" dialogue and select Settings_3. Check also the second storey – FL2. Close the dialogue.



The result in the 3D window:

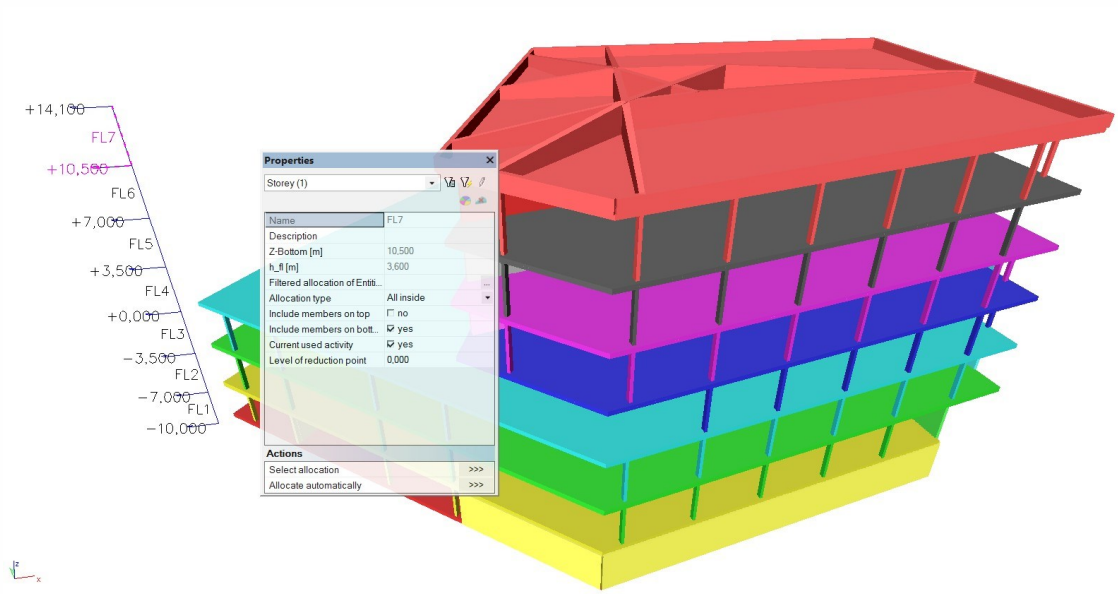


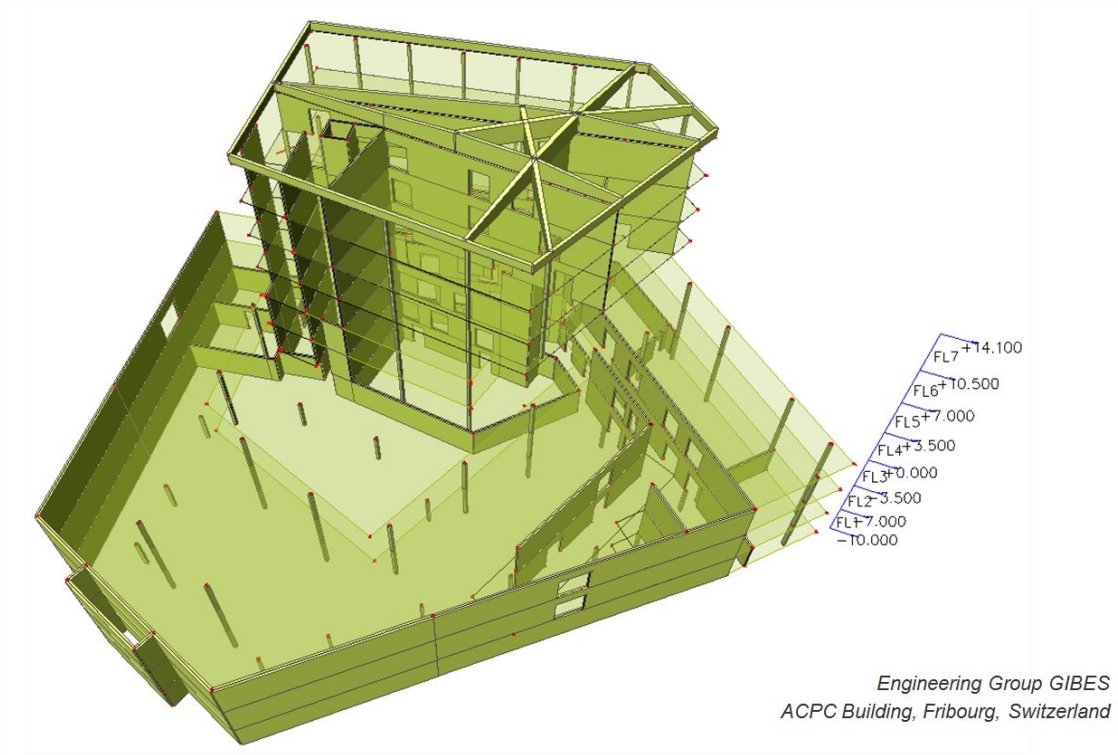
9. Conclusion: The activity by storeys works in the similar way as activity by Layers. It is useful functionality for higher buildings. The engineer has possibility to work in separate levels without defining layers.

Storey Results

For a detailed description of storey results and their usage, see the following pages:

- Summary Storey Results
- Detailed Storey Results





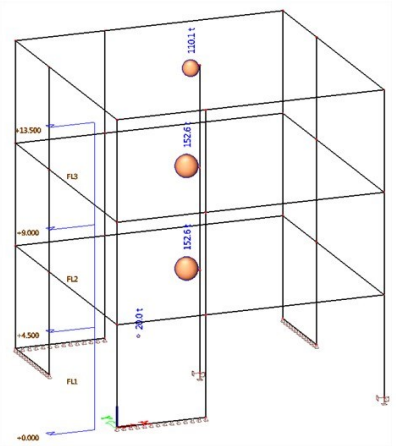
Summary storey results

The service "Summary Storey Results" provides results directly produced by the IRS analysis (see Reduced analysis model). At this time, this service is dedicated to result output for the seismic analysis of buildings. It provides single results per storey, such as mass, position of mass center, displacement, acceleration…

Summary storey results are available only when the reduced modal analysis is enabled and storey are defined.

Types of results:

For mass combinations	Storey data: mass & mass center of each storey Displacements of storey mass center per mode Accelerations of storey mass center per mode
For seismic load cases	Displacements of storey mass center Accelerations of storey mass center Inter-storey drift at storey mass centers



Summary storey result

Storey Displacements:
Eigen solution, Extreme: No, System: Principal
Selection: All
Mass combinations : CM1/1 - 2.07

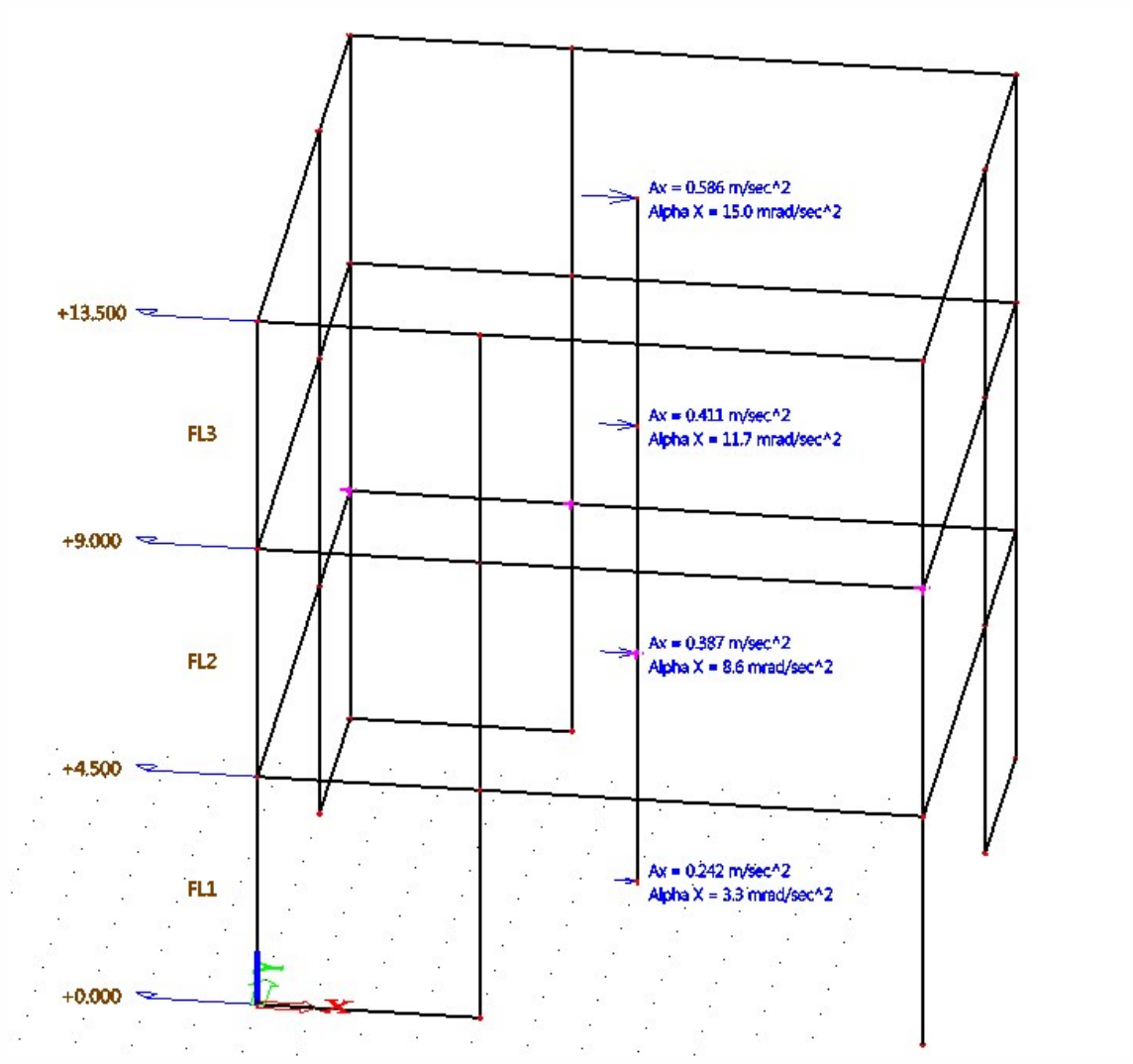
Name	Ux [mm]	Uy [mm]	Uz [mm]	Phix [mrad]	Phiy [mrad]	Phiz [mrad]
FL1	-6.1e-02	6.1e-02	0.0e+00	-2.0e-03	-2.0e-03	0.0e+00
FL2	-3.3e-01	3.3e-01	7.7e-02	-8.0e-03	-8.0e-03	0.0e+00
FL3	-9.9e-01	9.9e-01	1.1e-01	-1.1e-02	-1.1e-02	0.0e+00
FL4	-1.7e+00	1.7e+00	1.3e-01	-1.0e-02	-1.0e-02	0.0e+00

Summary storey result

Storey Accelerations:
Eigen solution, Extreme: No, System: Principal
Selection: All
Mass combinations : CM1/1 - 2.07

Name	Ax [m/sec^2]	Ay [m/sec^2]	Az [m/sec^2]	Alpha X [mrad/sec^2]	Alpha Y [mrad/sec^2]	Alpha Z [mrad/sec^2]
FL1	-0.010	0.010	0.000	-3.38e-01	-3.38e-01	0.00e+00
FL2	-0.056	0.056	0.013	-1.35e+00	-1.35e+00	0.00e+00
FL3	-0.167	0.167	0.019	-1.86e+00	-1.86e+00	0.00e+00
FL4	-0.291	0.291	0.022	-1.69e+00	-1.69e+00	0.00e+00

Picture: Summary storey results



Picture: Accelerations

Detailed Storey Results

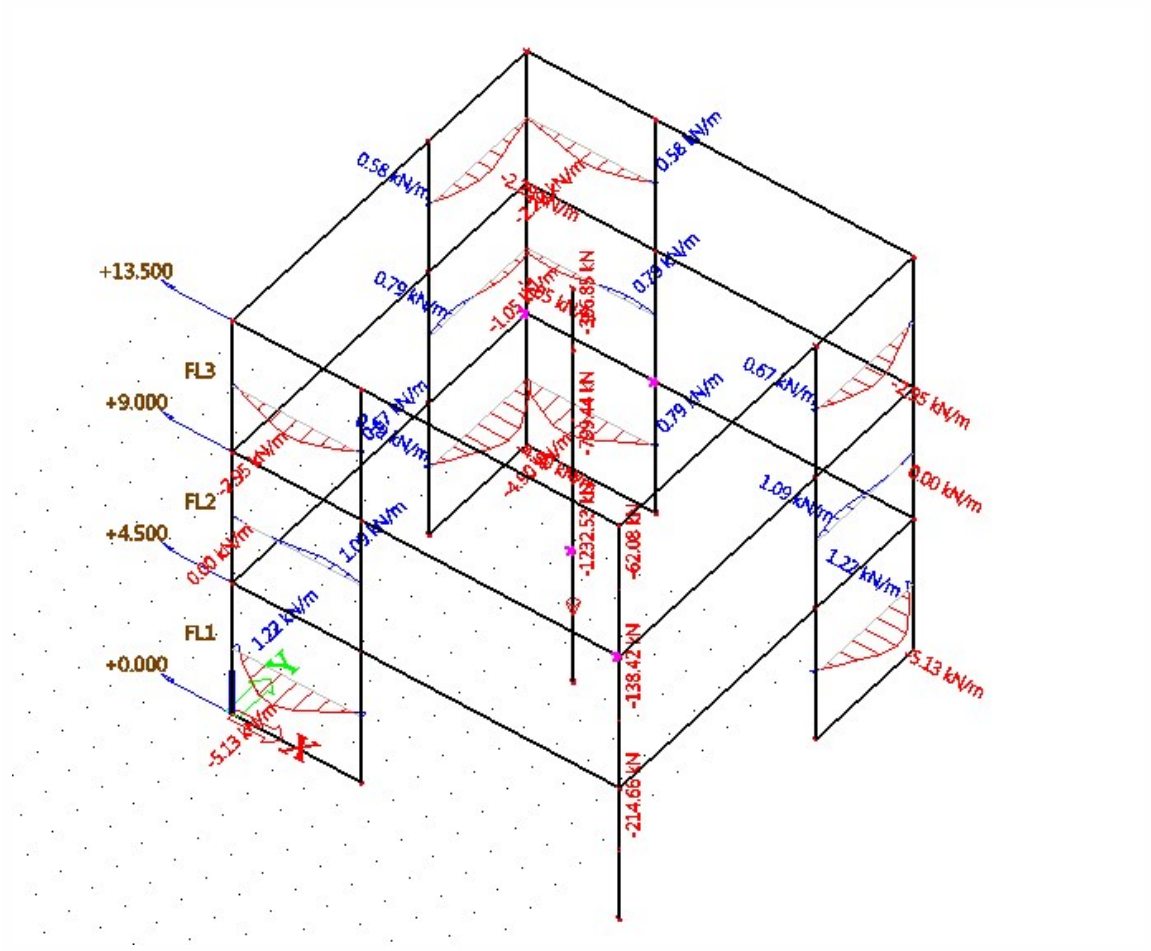
The service "Detailed Storey Results" provides results from the full mesh analysis. At this time, it may be used for results from any linear analysis, with or without dynamic analysis, with or without IRS analysis. It provides results in all supporting members, with easy selection of members per storey. Walls and columns may be represented on the same drawing. Typical provided results are: internal forces, resultants per wall or per storey…

> Tip: before using detailed storey results, make sure that all supporting members of the building are properly allocated to storeys. That information is essential for proper handling of storey results.

Type of results

Internal forces in supporting members	Selection by storey Extreme by member (also for walls !) Walls & columns on the same drawing

	Simple choice of section level Display of average & total value for walls Available for static & seismic results Also suitable e.g. for load descending
Resultant forces in supporting members	Resultant for each wall on the same drawing Clear display of more components + all the key points mentioned previously
Resultant forces in supporting members per storey	Resultant of all supporting members at once (walls + columns combined) + all the key points mentioned previously



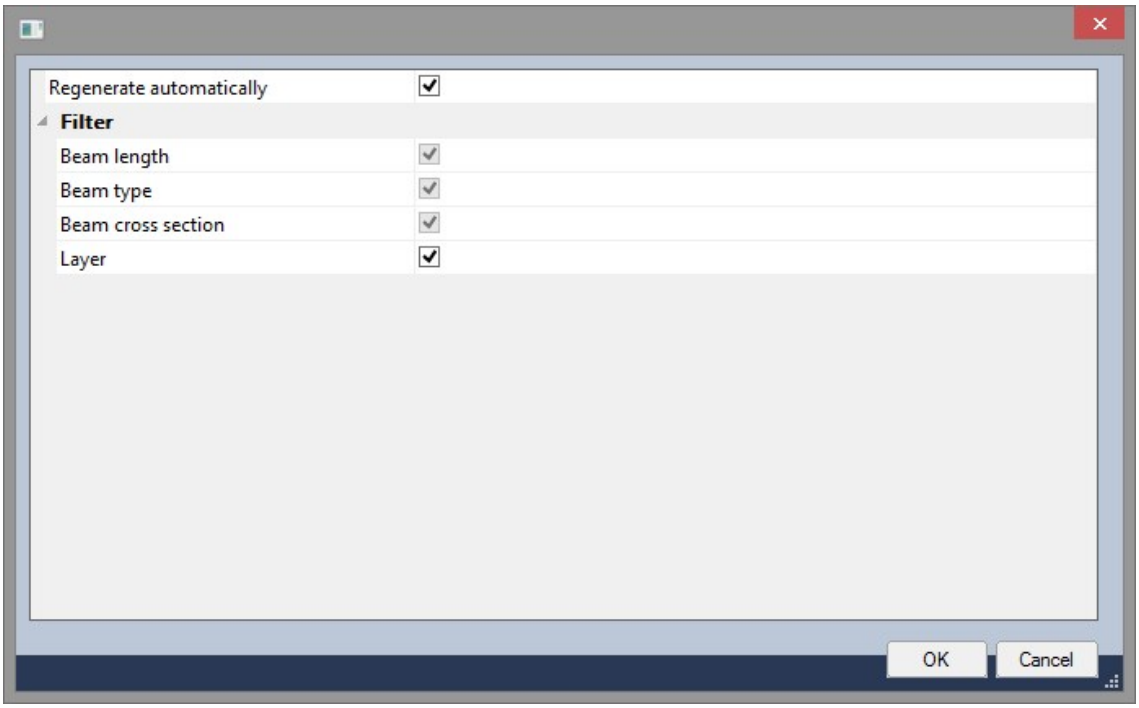
Picture: Internal forces

Design groups

Design groups is a tool which automatically sorts members according to predefined Filter and stores found groups in the Design groups list. This simplify design of a structure because the user can easily identify identical members and edit them at once. Next, design groups can be used for selection of members in services Results, Steel, Concrete, etc.

Design groups setup dialogue

This chapter provides description of available parameters from the Design groups Setup dialogue.



Setup parameters

Regenerate automatically	Activates Design groups functionality. If active, Design groups library is available.

Filter

Filter is a group of criteria which are used for generation of Design groups.

Beam length	Default parameter. Cannot be deactivated.
Beam type	Default parameter. Cannot be deactivated.
Beam cross-section	Default parameter. Cannot be deactivated.
Layer	If active, Layers are used

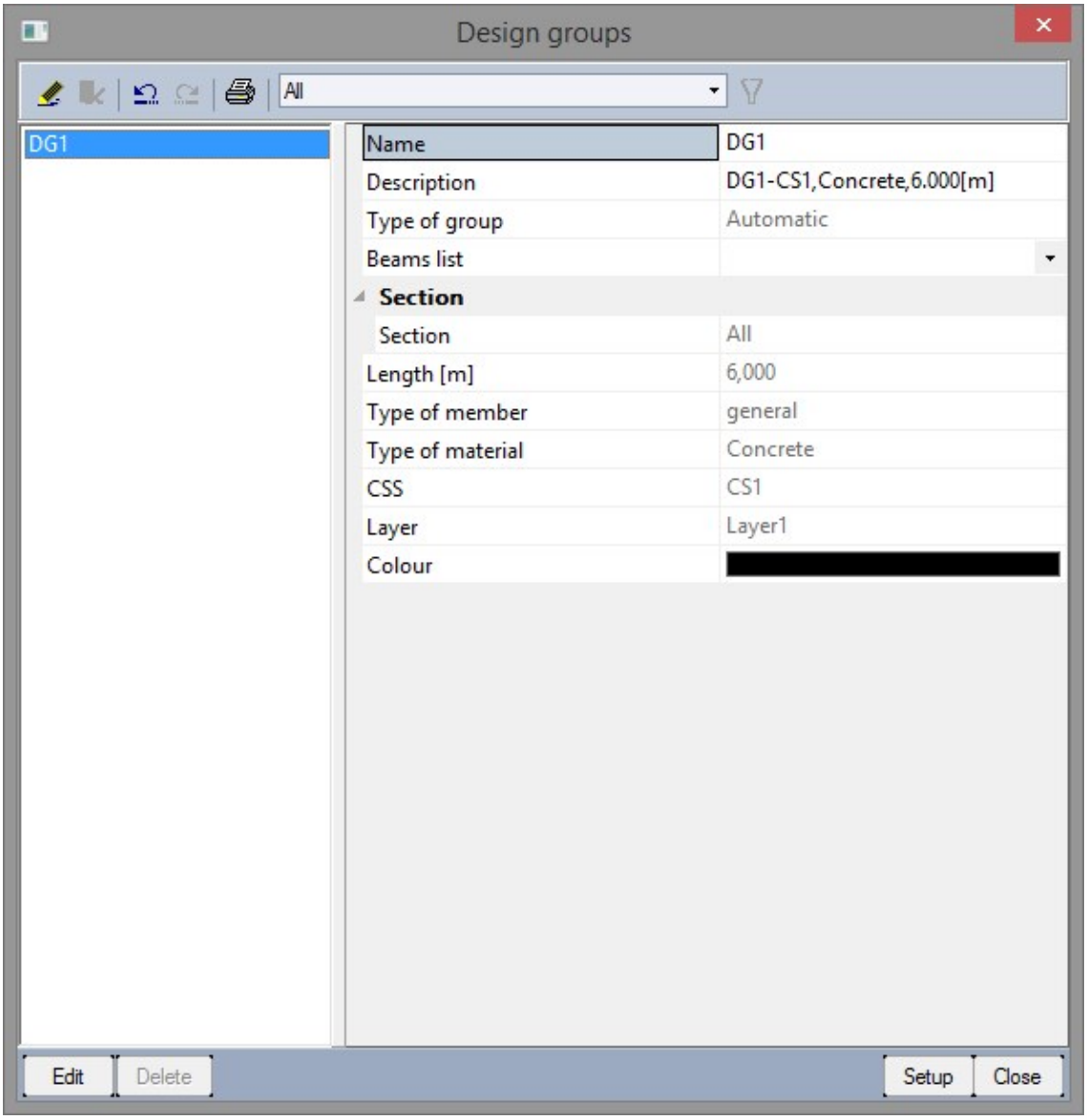
Design groups list

The Design groups list is a tool to control and edit existing design groups in the project.

It is a manager which uses the same "manager philosophy" as other SCIA Engineer managers.

Edit	It opens an editing dialogue where the layer-properties may be changed.
Undo / Redo	It performs the Undo or Redo operation.
Text Output	It opens a small document window with a table that summarizes properties of selected layers.

In order to open the Layers manager use tree menu function Design groups > Design groups list.



Design groups view parameters

Tab Structure > Group Structure

Style + colour

It controls the style and colour of members of the model (beams, plates, shells, etc.)

normal	settings made in Setup > Colour/Lines dialogue are used,
by layers	each member is displayed in the colour of the appropriate layer, all members assigned to the same layer are of the same colour,
by material	each member is displayed in the colour of the appropriate material, all members made of the same material are of the same colour,
by cross-section	each member is displayed in the colour of the appropriate cross-section, all members of the same cross-section are of the same colour,

by design group	each member is displayed in the colour of the appropriate design group, all members assigned to the same design group are of the same colour
according to structural type	each member is displayed in the colour corresponding to its structural type.

Note : If e.g. two materials, two layers, two cross-sections have assigned the same colour, than the same colour is used for members of different controlling property.

Tab Structure > Group Design groups

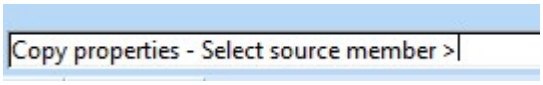
Display

The view flags with the design group name can be displayed or hidden.

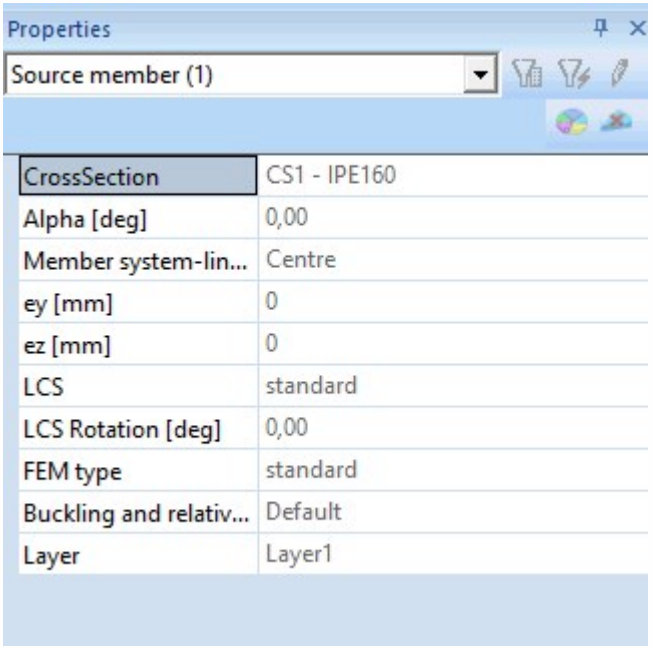
Match properties

Match properties is feature which will allow user to copy properties across elements.

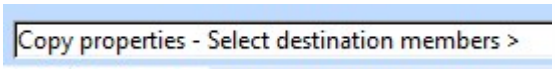
User can start Match properties simply from property window (or command line, keyboard shortcut), the tool tip in command line will appear. (Or user can choose source first and then start Match properties)



There will be shown list of properties which were picked to copy



And tooltip in command line will appear



Now user can choose as many elements he want to copy properties to. If some property was not possible to copy, error log will appear.

List of allowed elements for Match properties:

- 1D members
- 2D members
- Load panel
- Support in node
- Support on beam
- Line support on 1D member
- Line support on 2D member edge
- Point force in node
- Point force on 1D member
- Free point force
- Line load on 1D member
- Line load on 2D member edge
- Free Line load
- Surface load on 2D member
- Free surface load
- Moment in node
- Moment on 1D member
- Free moment

- Line moment on 1D member
- Line moment on 2D member edge
- Hinge on 1D member

User can start Match properties:

- From icon in property grid
- default keyboard shortcut CTRL+SHIFT+F
- "ma" to command line
- Match properties command and keyboard shortcut can be adjusted as any other command in Scia Engineer

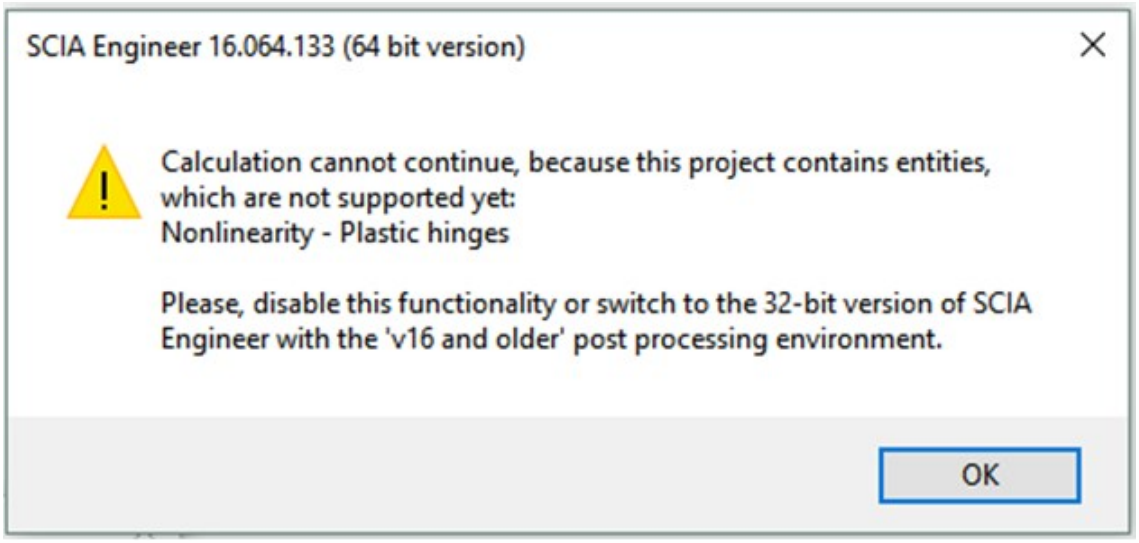
SCIA Engineer 64-bit

SCIA Engineer 64-bit main approach & overview

SCIA Engineer (from version v17.1) provides the possibility to choose between two versions (32-bit and 64-bit) for displaying results and performing the design.

Main approach for modelling – calculating - checking in 64-bit

The 64-bit version does not yet contain all functionalities which are available in the 32-bit build due to the fact that not all of these functionalities are 64-bit compliant yet. The main approach for 64-bit is to allow the user to input (almost) everything (even if it is unsupported). If an unsupported analysis type is used in the project an error message will appear before the calculation starts. For example, when using "plastic hinges" which is currently unsupported you will get the following error message before the calculation starts:



And if a result or check is unsupported it will be hidden in SCIA Engineer. In those cases the user has the possibility to continue his/her work in the 32-bit version of SCIA Engineer (with the V16 PPE). If an unsupported check/result item was already present in the Engineering Report, Picture Gallery or Table result the user will see the following message appear: **Not supported task. Please switch to PPE v16 (available in 32-bit version).**

Remark: PPE is the abbreviation for the term post-processing environment

Overview table of functionalities

Below you can find an overview of the supported or unsupported functionalities per version and per post-processing environment (PPE).

Functionality	Subfunctionality	32-bit version (R17.1)		64-bit version (R17.1)
		"V16 and older" PPE	"V17" PPE	"V17" PPE
Aluminium	Scaffolding	yes	yes	no (*1)
	LTB 2nd Order	yes	yes	no
Concrete	Code dependent deflection	no	yes	yes
	1D physical nonlinearity	yes	no	no
	2D physical nonlinearity	no	yes	yes
Steel	Plastic hinges	yes	no	no
	Fire resistance	yes	yes	yes
	Connection modeller	no (*2)	no (*2)	no (*2)
	Frame rigid connections	yes	yes	no
	Frame pinned connections	yes	yes	no
	Grid pinned connections	yes	yes	no
	Bolted diagonal connections	yes	yes	no
	Hollow section joints	yes	yes	yes
	Expert system	yes	yes	no
	Connection mono-drawings	yes	yes	yes
	Scaffolding	yes	yes	no (*1)
	LTB 2nd Order	yes	yes	no
	ArcelorMittal	no (*2)	no (*2)	no (*2)
	Girders with sinusoidal webs	yes	yes	no
Dynamics	-	yes	no (*5)	no (*5)
	Seismic	yes	yes	yes
	Harmonic band analysis	yes	no	no
	General dynamics	yes	no	no
	Non-proportional damping	yes	no	no
Initial stress	-	yes	yes	yes
Subsoil	-	yes	yes	yes
	Soil interaction	yes	yes	yes
	Soil loads	yes	yes	yes
	Pile Design (NEN method)	yes	no	no
	Pad foundation check	yes	yes	no
Non-linearity	Initial deformations and curvature	yes	yes	yes
	Geometrical non-linearity	yes	yes	yes
	Phys. non-linearity for reinf. concrete	yes	no	no

	Beam local non-linearity	yes	yes	yes
	Support non-linearity/Soil spring	yes	yes	yes
	Friction support/Soil spring	yes	no (*4)	no (*4)
	Membrane elements	yes	no	no
	Press only 2D members	yes	no	no
	General plasticity	yes	yes	yes
	Sequential analysis	yes	no	no
	Cable	yes	no	no
Stability	-	yes	yes	yes (*9)
Climatic loads	-	yes	no (*8)	no (*8)
Prestressing	-	yes	no	no
	Advanced	yes	no	no
Pipelines	-	no (*2)	no (*2)	no (*2)
Structural model	-	yes	yes	yes
BIM properties	-	yes	yes	yes
Parameters	-	yes	yes	yes
Mobile loads	-	yes	no	no
Automated GA drawings	-	yes	yes	yes
LTA - loadcases	-	yes (*2)	no (*2)	no (*2)
CADS composite checks	Composite column	yes	yes	no
	Fire resistance	yes	yes	no
External application checks	-	yes	yes	no
Slabs with void formers	-	yes	no	no
Property modifiers	-	yes	yes (*7)	yes (*7)
Bridge design	Load combinations	yes	yes	yes
	Concrete checks extensions	yes	no	no
Document	-	yes	no (*6)	no (*6)
Advanced concrete checks	-	yes	no	no
	Fire resistance	yes	no	no
	Hollow core slab	yes	no	no

*(*1): Only the scaffolding coupler check is not supported in 64-bit, the scaffolding check does work under 64-bit.*

*(*2): Functionality is obsolete and thus to be removed from the source code.*

*(*4): Only "friction support" is not supported in "V17" PPE and 64-bit.*

*(*5): Only modal analysis is supported (eigenmode shapes are available under 3D-displacements). Harmonic loads and Karman vibrations are not supported.*

*(*6): Hidden in new projects but available when opening old projects that contain the old document.*

*(*7): Property modifiers are not supported for composite*

*(*8): 3D wind generator is supported since 1st patch of SCIA Engineer 17.1.*

*(*9) : Buckling factors from stability are not supported yet in 64-bit*

Overview table of results service items

Results service items	32-bit version (R17.1)		64-bit version (R17.1)
	"V16 and older" PPE	"V17" PPE	"V17" PPE
Deformed structure	yes	no(*2)	no(*2)
Foundation table	yes	yes	no
Nodal space support resultant	yes	yes	no
Relative deformations	yes	no	no
Member stress	yes	no(*1)	no(*1)
Shear stress	yes	no(*1)	no(*1)
Connection forces	yes	yes	no
Integration strip	yes	no	no
2D/1D upgrade	yes	no	no
Bill of material	yes	no	no

*(*1): User can use 1d member stress and 3d stress*

*(*2): Deformed structure is drawn in wireframe mode of 3D displacement*

Overview table of model types

Below you can find an overview of the supported or unsupported model types per version and per post-processing environment (PPE).

Model type	32-bit version (R17.1)		64-bit version (R17.1)
	"V16 and older" PPE	"V17" PPE	"V17" PPE
One	yes	yes	yes
Absence	yes	no	no
Construction stages	yes	no	no

Overview table of import formats

Below you can find an overview of the supported or unsupported import formats per version and per post-processing environment (PPE).

Import	32-bit version (R17.1)		64-bit version (R17.1)
	"V16 and older" PPE	"V17" PPE	"V17" PPE
DSTV	yes	yes	no
Esa	yes	yes	yes
Esa In	yes	yes	no
Etabs	yes	yes	no
Graphic format (dwg,...)	yes	yes	yes
IFC 2x3	yes	yes	yes
SDNF	yes	yes	yes
Stepsteel	yes	yes	no
Tekla	yes	yes	yes

Revit	yes	yes	yes (*1)
XML	yes	yes	yes

*(*1): Direct data exchange is not supported yet in 64bit.*

Overview table of export formats

Below you can find an overview of the supported or unsupported export formats per version and per post-processing environment (PPE).

Export	32-bit version (R17.1)		64-bit version (R17.1)
	"V16 and older" PPE	"V17" PPE	"V17" PPE
Allplan (.ifc)	yes	yes	yes
CEA Plant-4D	yes	yes	yes
DSTV	yes	yes	no
Esa	yes	yes	yes
Esav	yes	yes	yes
Esa In	yes	yes	no
Etabs	yes	yes	no
Graphic format (dwg,...)	yes	yes	yes
IFC 2x3	yes	yes	yes
IFC 2x3 Compressed	yes	yes	yes
SDNF	yes	yes	yes
Stepsteel	yes	yes	no
Tekla	yes	yes	yes
Revit	yes	yes (*1)	yes (*3)
XML	yes	yes (*2)	yes (*2)

*(*1): Export of results is not supported in "v17" PPE.*

*(*2): Export of new results (results v17)) is not supported in "v17" PPE*

*(*3): Direct data exchange and export of results to r2s file is not supported yet in 64bit.*

SCIA Engineer 64-bit: Hidden items

Hidden items of unsupported functionalities in 64-bit (General)

The list of unsupported functionalities in the 64-bit version can be found below. This list will become shorter after each incremental release of SCIA Engineer in the future.

Functionality/Material	Subfunctionality	Hidden items in R17.1 (64-bit)
Advanced concrete checks	(not applicable)	- Concrete Advanced service containing all the checks (Overall Check, Crack control,...) - EC-tools (also not available in 32-bit with "v17" PPE) - Advanced concrete checks functionality in the project data dialog
	Fire resistance	- Concrete Advanced service containing all the checks

		(Overall Check, Crack control,...) - Fire resistance functionality in the project data dialog
	Hollow core slab	- Concrete Advanced service containing all the checks (Overall check, Crack control,…) - Hollow core slab functionality in the project data dialog
Aluminium	(not applicable)	- Check (ULS) - Relative deformation (SLS)
	LTB 2nd Order	- No hiding, FriLo-solver isn't used because the checks that use it are hidden (see above).
	Scaffolding	- Scaffolding – Coupler Check
External application checks	(not applicable)	- Custom check (i.e. Excel Check)
Bridge Design	Concrete checks extension	- Concrete Advanced service containing all the checks (Overall check, Crack control,…)
CADS composite checks	Composite beam	- Member data - ULS design check - SLS design check
	Composite column	- ULS design check
	Fire resistance	- Fire resistance (composite beam) - Fire resistance checks (composite column)
LTA - loadcases	(not applicable)	- LTA Load - LTA Conductor insulators (load service)
Prestressing	(not applicable)	- Concrete Advanced service containing all the checks (Overall check, Crack control,…) - Prestressing functionality in the project data dialog
	Advanced	- Concrete Advanced service containing all the checks (Overall check, Crack control,…) - Prestressing functionality in the project data dialog
Slabs with void formers	(not applicable)	- Concrete Advanced service containing all the checks (Overall check, Crack control,…) - Slabs with void formers functionality in the project data dialog

Steel	ArcelorMittal	- Cellular beam check
	Bolted diagonal	- Bolted diagonal functionality in the project data dialog - Bolted diagonal, Bolted diagonal – bolt operations - Action buttons in bolted diagonal property window (Refresh, Open preview, Results, Autodesign, Edit bolted diagonal) - Bolted diagonal (in the connections setup).
	Expert system	- Expert system functionality is hidden in the project data dialog - Action buttons in connection property window (Save to expert database, Load from expert database) - Expert system (in the connections setup)
	Frame pinned connections	- Frame strong-axis - Frame weak-axis
	Frame rigid connections	- Frame strong-axis - Frame weak-axis
	Girders with sinusoidal webs	- SIN beam check
	Grid pinned connections	- Grid pinned
	LTB 2nd Order	- No hiding, FriLo isn't used -> default method from steel setup is used + error message is shown in the header of the steel code check output.
	Plastic hinges	- No hiding, plastic hinges aren't accounted for. Error message is shown when calculating.
	Scaffolding	- Scaffolding – Coupler Check
Subsoil	Pad foundation check	- Geotechnics setup - Pad foundation – Pad foundation stability

Hidden items of unsupported functionalities in 64-bit (Non-Eurocode)

Remarks:

• Advanced concrete checks (old concrete menu) is not supported in 64-bit, only the new concrete menu. Therefore, the functionality Advanced concrete checks are only possible to obtain in the 32-bit version with the post processing environment set to "V16 and older".

• The same applies for the steel connections related functionalities (Bolted diagonal, Expert system, Frame pinned connections, Frame rigid connections, Grid pinned connections), these are only possible to obtain in the 32-bit version (available in both post processing environments).

Code	Functionality/Material	Hidden items in R17.1 (64-bit)
ABNT-NBR (Brazilian)	Steel	- Steel slenderness
BS (Great Britain)	Steel (BS)	- Setup - Steel slenderness - ULS design check - SLS design check

	Steel (BS 5950-1:1990)	- Setup - Steel slenderness - ULS Checks - Check - SLS Checks - Relative deformation
CSN (Czech Republic)	Steel	- Setup - Steel slenderness - ULS Checks - Check - SLS Checks - Relative deformation
	Timber	- Setup - Check - Deformation with creep
DIN (Germany)	Steel	- Setup - Steel slenderness - ULS Checks - Check - SLS Checks - Relative deformation - Characteristic beam factor
	Timber	Beams - Setup
ENV (Europe)	Steel	- Setup - Steel slenderness - SLS Checks - Relative deformation
	Timber	- Setup - Check - Deformation with creep
France	Steel	- Setup - Steel slenderness - ULS Checks - Check - SLS Checks - Relative deformation
IBC (U.S.A.)	Steel	- Setup - Steel slenderness - Check ASD - Check LRFD - SLS Checks - Relative deformation
India	Steel	- Setup - Steel slenderness - ULS Checks - Check - SLS Checks - Relative deformation
NEN (Netherlands)	Steel	- Setup

		- Steel slenderness - ULS Checks - Check - ULS Checks - Fire resistance - SLS Checks - Relative deformation
ONORM (Austria)	Steel	- Setup - Steel slenderness - ULS Checks - Check - SLS Checks - Relative deformation
SIA (Switzerland)	Steel	- Setup - Steel slenderness - ULS Checks - Check - Fire resistance - SLS Checks - Relative deformation
	Timber	- Beams - setup
Slovakia	Steel	- Setup - Steel slenderness - ULS Checks - Check - SLS Checks - Relative deformation
	Timber	- Setup - Check - Deformation with creep
Spain	Steel	- Setup - Steel slenderness - ULS Checks - Check - SLS Checks - Relative deformation

SCIA Engineer 64-bit: Frequently Asked Questions

Frequently Asked Questions

What is the difference between 32-bit and 64-bit SCIA Engineer?

The terms **32-bits** and **64-bits** refers to how the processor (CPU) of a computer processes data. The **64-bit version** of SCIA Engineer processes large amounts of RAM (memory) more effectively than the 32-bit version of SCIA Engineer. This gives the user the possibility to **allocate much more memory** (> 2^32 bytes = approx. 4 Gb) which **improves the stability** of SCIA Engineer when **modelling large projects**.

The 64-bit version of SCIA Engineer uses the “v17” post processing environment while the 32-bit version of SCIA Engineer can use the “v16 and older” post processing environment as well as the “v17” post processing environment. More information about the post processing environments can be found here: "v17 Post-processing environment " on page 209

What is the difference between "v16 and older" versus "v17" post-processing environment?

Usage of the v17 post-processing environment is recommended to get the benefits from some of the latest developments of SCIA Engineer 17: ability to run a selection of load cases or non-linear combinations, CDD (Code Dependent deflection) for concrete according to the Eurocode, upgraded result service. For more info: visit the page what's new SCIA Engineer 17

While in the "v16 and older" post-processing environment there are no new developments but it fully supports all the developments before version 17.

An overview table of supported and unsupported functionalities can be found in "SCIA Engineer 64-bit main approach & overview" on page 450

What is the interaction between the two post-processing environments?

The two post-processing environments are fully compatible. Existing projects from previous SCIA Engineer versions or using the "v16 and older" post-processing can be run using the "v17" post-processing and vice-versa. Every time the post-processing environment is switched, a recalculation of the model is required.

It may happen that you start your project in "v17" and you realize that an advanced functionality you need is missing. You can switch back to "v16 and older" post-processing by the "project data" dialogue.

If you change the post-processing environment, as mentioned above you will use another export of the model to the solver, but also another method of showing results and new algorithms which might lead to slightly different result values.

Another consequence when switching between post-processing environments is the fact that the calculated results will be deleted. So as indicated a recalculation after switching is needed.

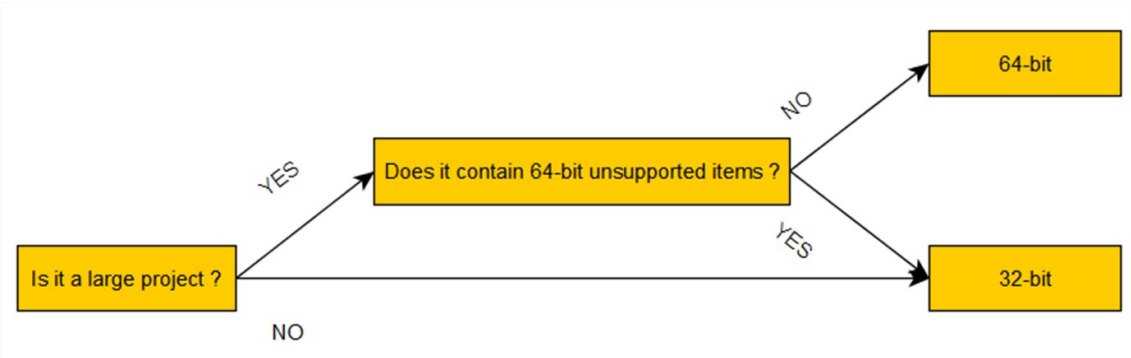
What happens with the Engineering report when switching to another version or post-processing environment?

In 32-bit version:
The Engineering Report is fully compatible with the two post-processing environments and can also contain images from both environments. Checking the Engineering Report might be required if content from the other post-processing environment is used, to avoid missing results from unsupported features.

In 64-bit version:
If the Engineering Report contains an unsupported item (see previous tables) it will not show the results of that item. Instead the following message will be displayed: *Not supported task. Please switch to PPE v16 (available in 32-bit version)*.

When to use the 32-bit or 64-bit version of SCIA Engineer?

The main advantage of 64-bit is the fact that it can process large amounts of data effectively than the 32-bit version. But the 64-bit version doesn't support all the features yet of the 32-bit version. So it depends per project which version is recommended. The flow chart below can offer some guidance:



This decision table doesn't need to be followed strictly during the start of the project as it can be sometimes unclear what functionalities will be used. The user can always start in both versions and in a later stage switch to the optimal version. This is possible due to the fact that SCIA Engineer projects can be opened in both versions (32-bit & 64-bit).