Scia Engineer

What’s New in Scia Engineer 2008.1

What’s New in Scia Engineer 2008.1

This document describes the main changes and amendments implemented in program Scia Engineer version 2008.1 in comparison with the previous version 2008.0. Not all changes are necessarily listed here and the document does not cover corrected bugs.

Table of contents

User interface

GUI improvements (esa.00)

Customisable User Interface

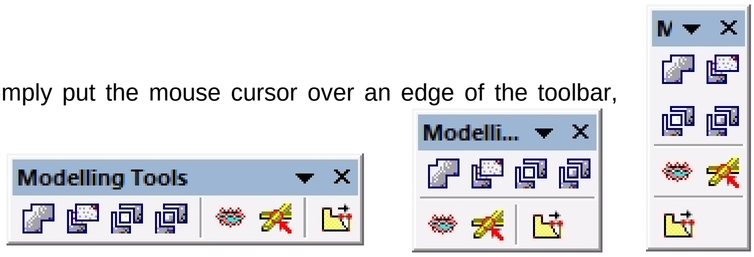
Customising the toolbars

It is possible to reshape the toolbars, add or remove buttons from individual toolbars and to define new tailor-made toolbars.

Reshaping the toolbar

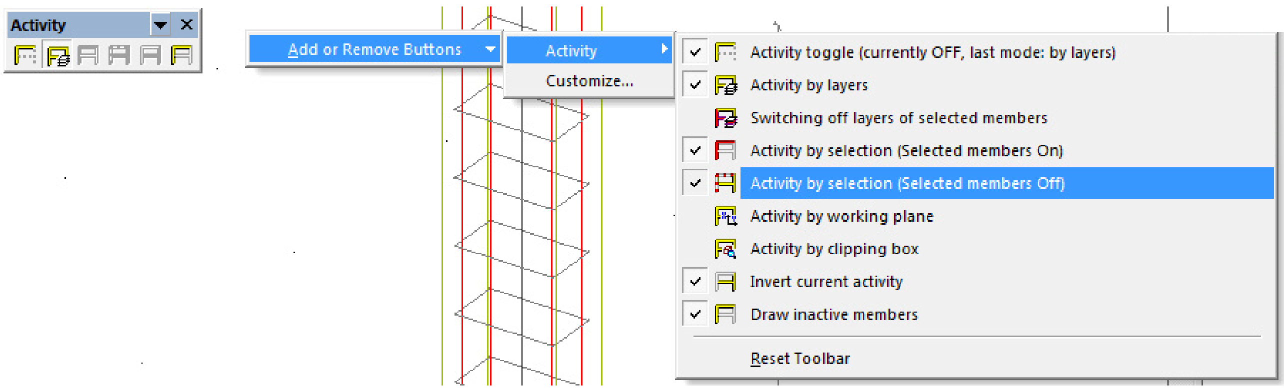
Each floating toolbar can be reshaped. Simply put the mouse cursor over an edge of the toolbar, click the mouse left button and drag.

Example:



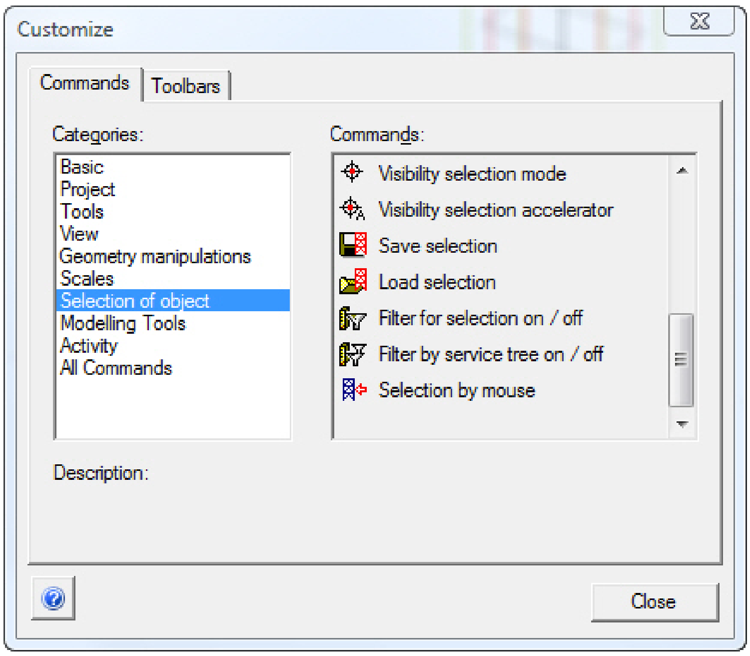
Hiding buttons from a toolbar

The users can hide from any toolbar those buttons that they do not use.



Dialogue "Customize"

The Customize dialogue can be used for a modification of any existing toolbar and for definition of new user-tailored toolbars.



Commands tab

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

When this tab is active, the user can drag-and-drop any command from the dialogue to any displayed toolbar.

Toolbars tab

On this tab you can:

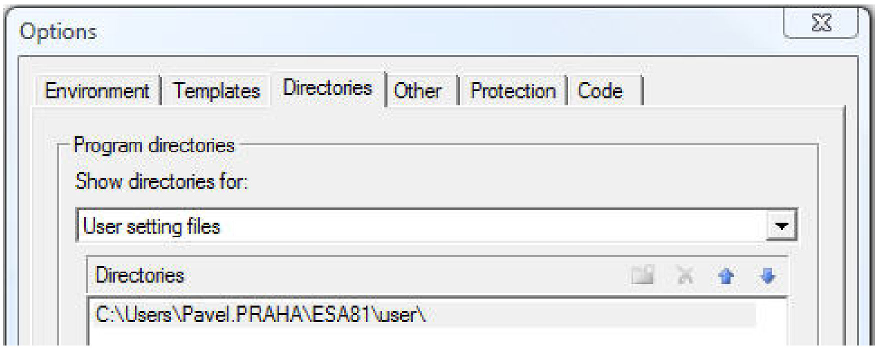
- display or hide any of the existing toolbars,
- reset the toolbar to the default configuration,
- create a new toolbar(s),
- delete you user-made toolbar(s),
- rename you user-made toolbar(s),

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

Copying the customised toolbars to a different computer

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.

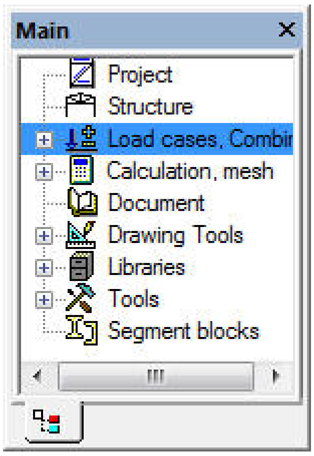
This folder contains sub-folder Toolbars with files for individual toolbars. Each toolbar has its own file with extension CTC: e.g. TB_Activity.CTC, TB_Basic.CTC, TB_Calculate.CTC, TB_Curves_Edit.CTC, etc.

If you want to transfer your settings to another computer, just copy these files to folder Toolbars in the User settings files folder on that second computer.

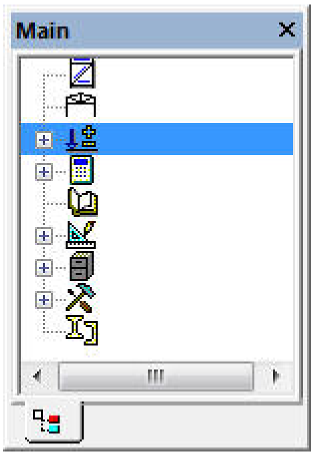
Customizing the tree menu

The tree menu can be customized using a local pop-up menu.

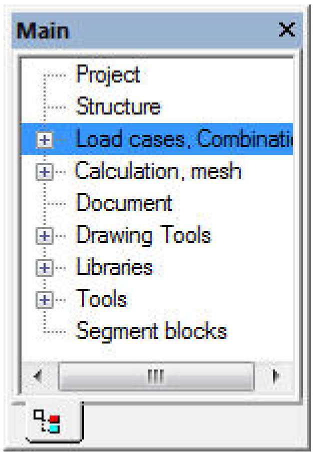
A) Icons + captions



B) Icons only

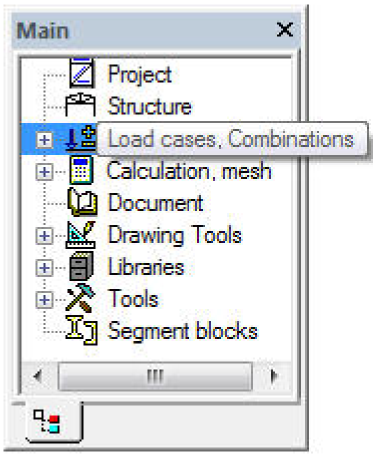


C) Captions only



D) Tooltips

If tool tips are ON and the window is to narrow to display the whole item, the full name of the selected item is shown as a tool tip.



Selections

Visibility selection mode

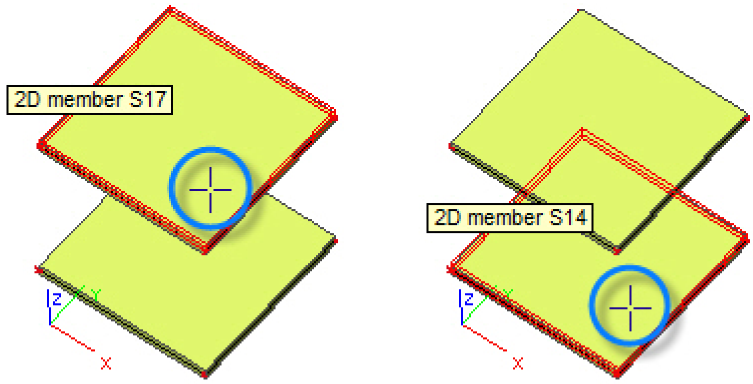
In the "normal" selection mode, you must select an edge of an entity in order to select it.

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.

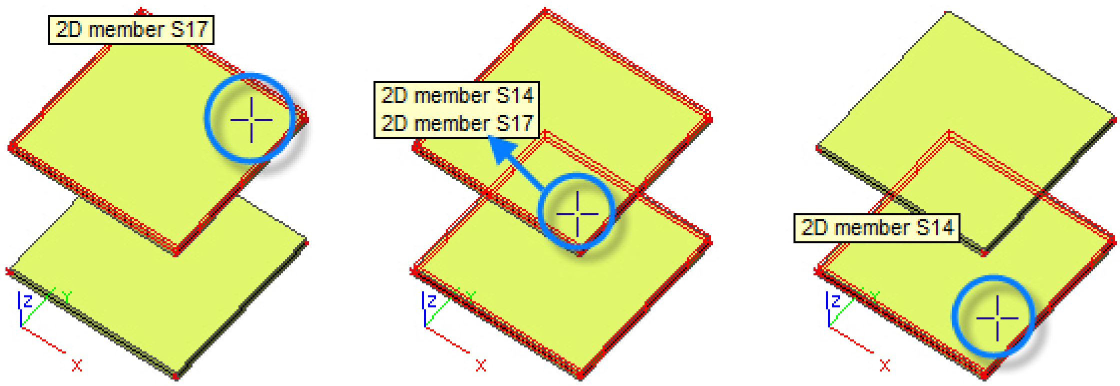
The Single selection mode toggle is taken into account in the Visibility selection mode.

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



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.

Small improvements

Remembering the state of tree in services

The program remembers the state (collapsed and expanded items) of every three menu. Whenever you re-enter any service in the tree menu, its items are expanded or collapsed as they were the last time you left the service.

Picture in the Property window

When an already defined entity is selected (e.g. a column, beam, slab, shell, support, load, etc.), its properties are displayed in the Property window. This feature has been now extended and the Property window displays also a picture of the selected entity. This helps especially better understand individual properties of the selected entity.

Slide-bar in the Property window

If the Property window is too narrow to display the whole names of items, you can use a newly implemented slide-bar to read the whole lines. Previously, only tooltip could be used to see the whole text of long items.

Zoom all by middle-button double-click

Function Zoom All can be easily invoked by a simple double-click on the middle button (or wheel) of your mouse.

Activity

Switching off layers of selected members

It is now possible to quickly switch off the activity of all entities that belong to the layer or layers of the currently selected entity or entities. You can simply select one entity from each layer and deactivate the whole layers.

Introducing OpenGL streams

When a rendered model is displayed, a new technology of OpenGL streams has been implemented. The result is that the model can be rotated, shifted and zoomed-in and out very fast – adjusting the view using key combinations Ctrl / Ctrl+Shift / Shift + right mouse button + drag gives almost immediate response even for extremely large models.

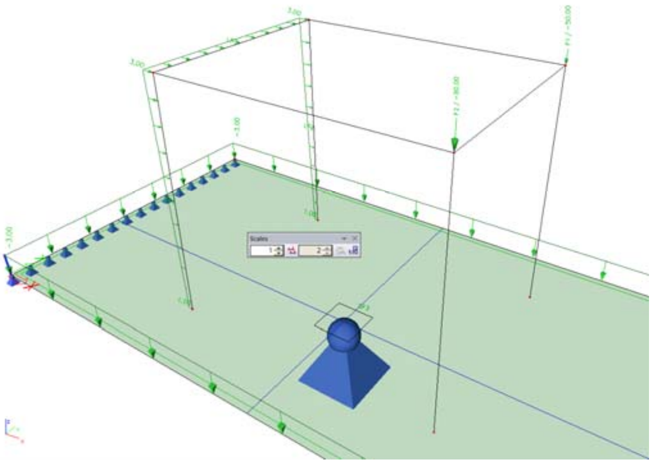
Scales improvements (esa.00)

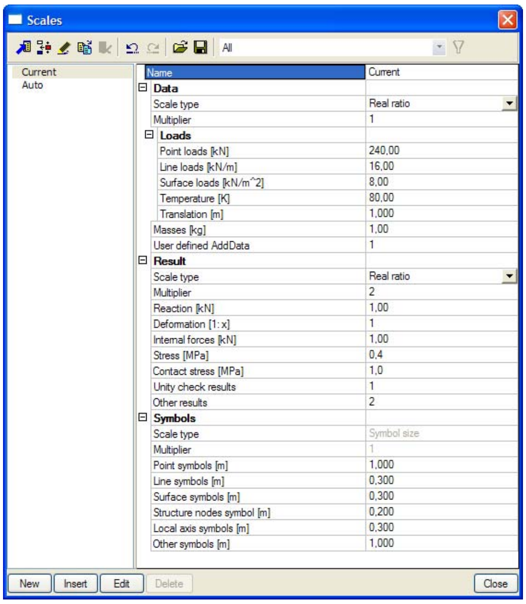
General

In Scia Engineer 2008.1 the Scales toolbar and functionality have been upgraded. In former versions this functionality was limited. Now the user is able to scale the items in the pictures more easily.

Add data

The user can define different scales for different add data. Some examples are shown in the figure. A different scale is used for line supports and point supports. Similarly, different scale can be applied to point load and line load, which means that loads are clearly visible.

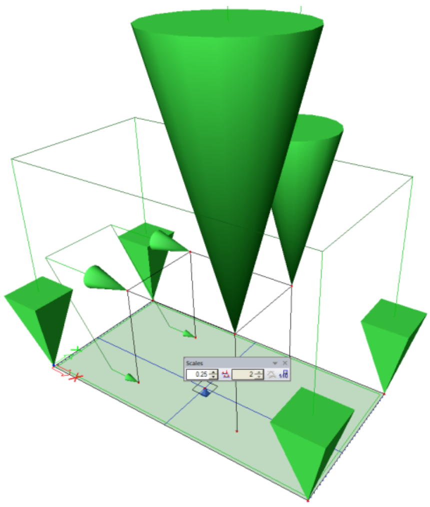


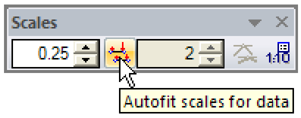


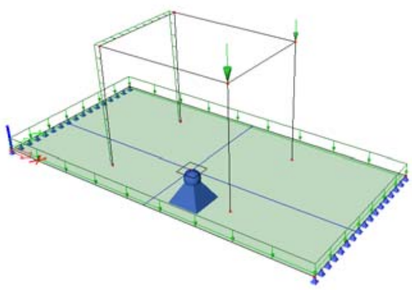
In order to manage all these scale relationships the user can use the scale manager. In this manager the user can create different sets of scales and store them so that they can be used in other projects.

Results (esas.00)

The same options as for the add data are available for the results. In the Scale manager different scale relations can be defined for different results.

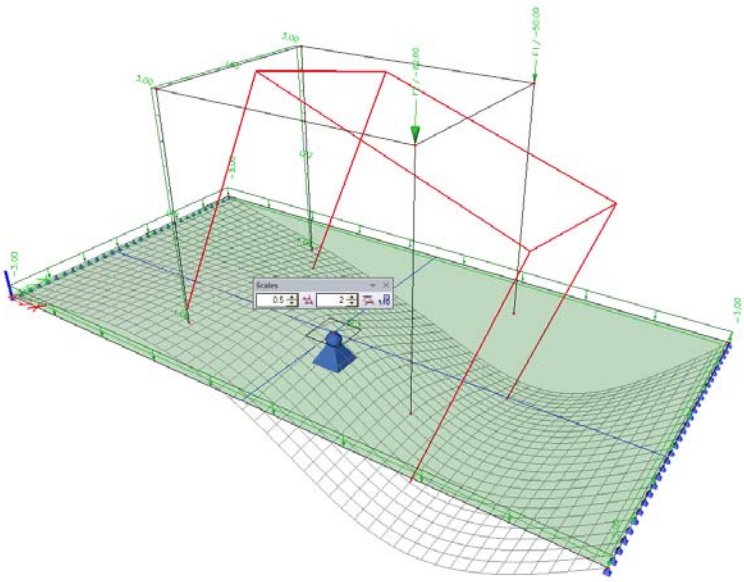






Auto scaling

When in a certain situation the chosen scale is too big to for the screen, the user can use the auto scale option. This option will scale the add data and the results in such a way that they safely fit the screen.



Add data together with results

In the new “Scales” toolbar the user can set a different scale for the add data (e.g. supports, loads, etc.) and the results. This can be very useful to point out some specific results in the picture without overwhelming the screen with the add data.

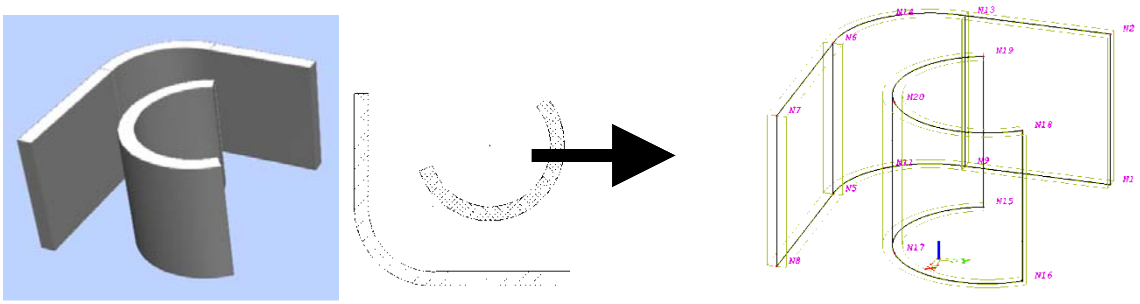
Modelling

Member recognizer improvements (esa.26)

As a part of our BIM Engineering toolbox for engineers, the member recognizer functionality for converting imported or ‘freely’ modelled solid CAD elements to 1D and/or 2D analysis members, or vice versa, has been extended with the possibility to recognize and convert circular walls and cross-sections.

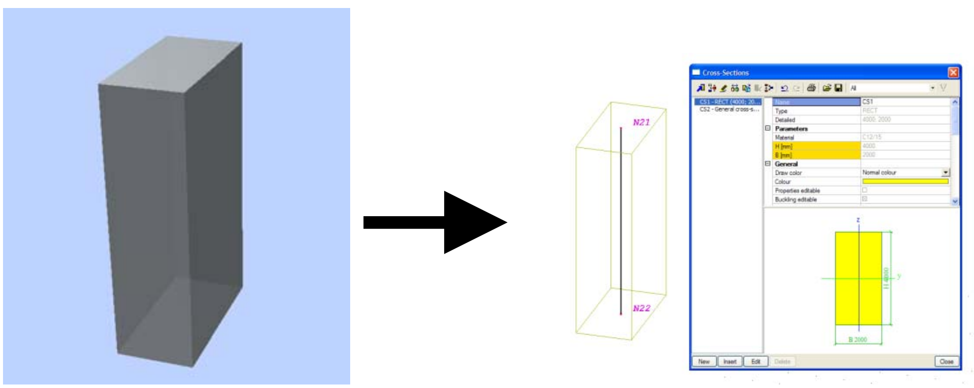
Recognition of circular walls

In version 2008.0 the member recognizer of 2D members was able to recognize only flat walls and slabs. In 2008.1 also walls of shape of circular arc are recognised.



Recognition of rectangular cross-sections

Recognition of 1D members from solids creates geometry of cross-section as an intersectional polygon of the solid and a certain plane perpendicular to expected member axis. The result of this operation is always a cross-section of the shape of a general polygon (general cross-section) in Scia Engineer 2008.0. In version 2008.1 this procedure has been extended with additional recognition of cross-section shape which results in cross-section of type “geometric-rectangle” instead of general polygon. This enables the users to handle the member as a rectangular profile which is much easier than working with a general polygon: for example, editing of cross-section dimensions by means of retyping the values, or also inputting and editing of reinforcement in Scia Engineer using standard functions and templates. Rectangular profile also allows for the application of the automatic reinforcement generator “AMRD” in Scia Engineer.



Creating lines from defined text (esa.24)

Function Lines from text allow you to type a text message anywhere in the graphical window and treat the text as lines.

It means that you can move it, rotate it, resize it, etc. like any other entity.

Document

Miscellaneous improvements

There are only small improvements introduced in a Scia Engineer Document in version 2008.1.

Automatic names of pictures

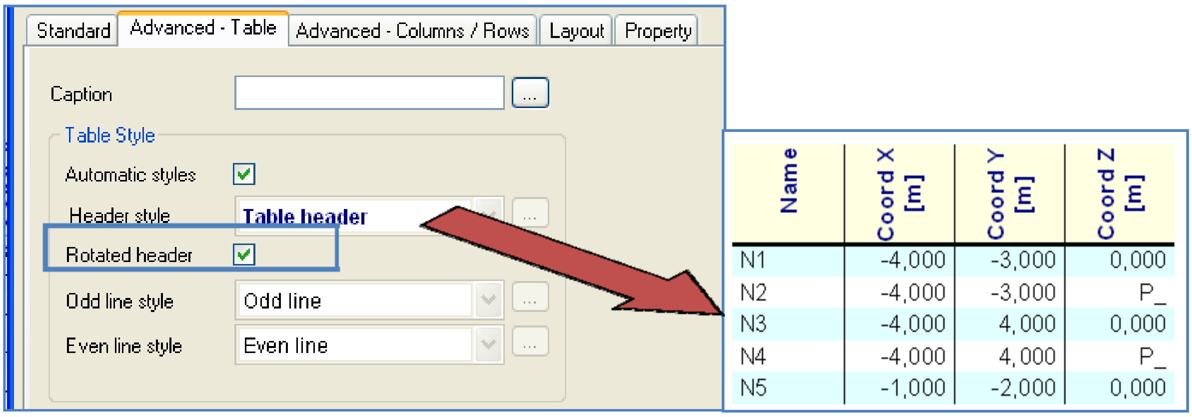
Scia Engineer defines a name of the picture when it is sent to the document. The name corresponds to the contents of the picture, e.g. which load case or type of result is displayed.

Possibility to copy a picture from the document to a clipboard

This feature works also for pictures generated by ChapterMaker.

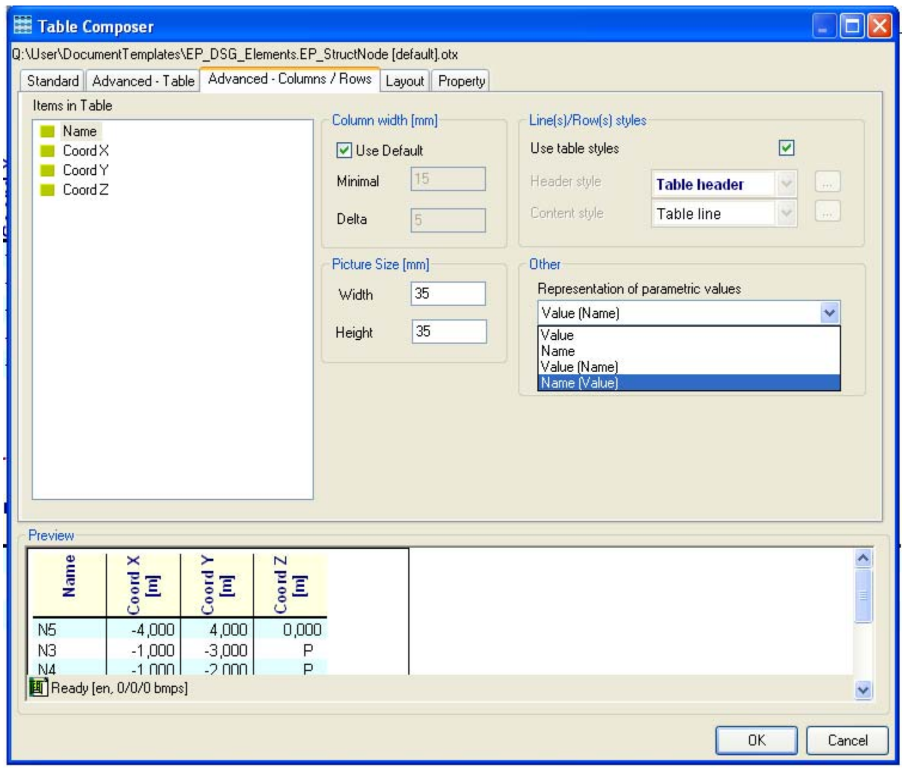
Rotated header

New advanced functionality makes it possible to rotate the text in a table header to reduce the width of the page.



Parameters

Properties defined by parameters can be represented by parameter names or by their real values in table output in Scia Engineer 2008.1 Document.



Load and other additional data

Free loads become real load generators

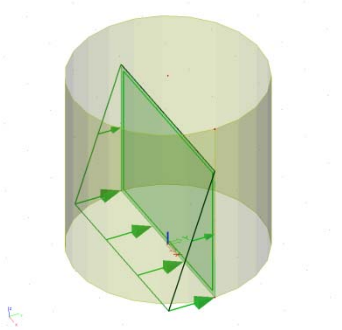
"Free loads" are a powerful tool for loading of both flat and curved 2D members such as walls, slabs and shells. Definition of free loads is composed of their geometry, which is independent on geometry of structural members, direction of load effect and a list of 2D members which are influenced by the free loads. Free loads are in fact easy load generators.

Original load / generated load

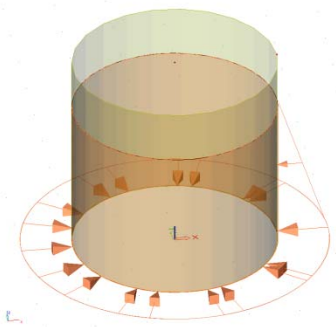
Inserting of free loads is easy to use. Scia Engineer 2008.0 didn't provide clear view which 2D members are loaded and in which direction. Scia Engineer 2008.1 introduces possibility to generate loads directly on 2D members and display them. Generated loads are displayed in the same way as the rest of generated loads (wind load, snow load, load transferred from load panels to members). They use different colour for their drawings and their properties are visible but disabled - the users cannot change their values.

The users can easily check if their input is correct or some adaptations are needed. Free loads which are not projected to any 2D members are highlighted to warn users, that their definition is probably not correct.

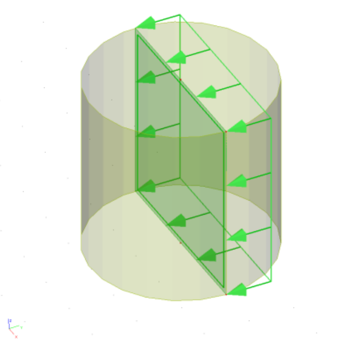
Below there are examples of usage of free loads in the next pictures.



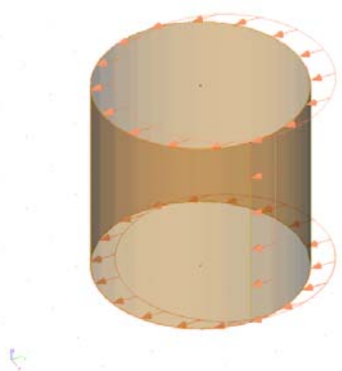
Surface free load on a cylinder



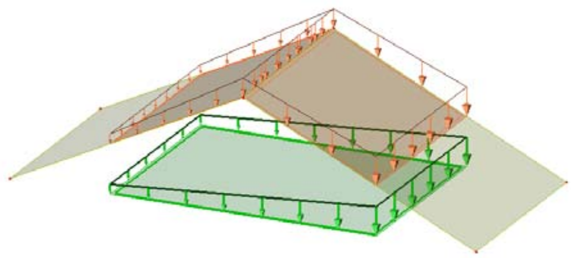
Generated load on a cylinder



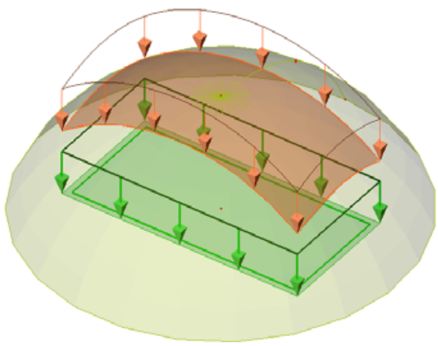
Definition of surface free load on a cylinder – wind load



Generated load on a cylinder



One definition of the free load can load more 2D members



Defined and generated load on a cupola

Generated loads in the Document

The side effect of the generation of "real loads" from free loads is ability to print them into a document. Both possibilities are kept. The user can print definition of free loads and/or generated loads related to 2D members.

Improvements in definition of validity of free loads

Projection of a free load is done in its local axis Z. The user can influence which 2D members are affected by the free load by definition of "validity" of the free load. This property of the free load has been extended by new settings:

- "Z=0": only 2D members, which are placed in the same plane as the free load, are loaded,
- "Z>=0": 2D members, which are placed in the same plane as the free load and 2D members above this plane, are loaded,
- "Z<=0": 2D members, which are placed in the same plane as the free load and 2D members below this plane, are loaded.

Formerly defined settings (all members in the projections, definition Z level "from –to") are kept.

FEM Analysis

Analysis after analysis (esas.45)

A new type of staged analysis is available, i.e. analysis after analysis. The purpose is to perform an analysis which starts with initial condition from a preciously calculation. Different types will be available for the user

(A) A stability analysis with initial condition from a nonlinear analysis. Hence the stability analysis is preformed taking into account of removed members, removed supports, initial stress and deformed initial mesh.

(B) A modal analysis with initial condition from a nonlinear analysis. Here the mode shapes are computed taking into account of removed members, removed supports, initial stress and deformed initial mesh.

(C) Sum the result a nonlinear analysis with the result of a linear one. For example results of removed bracings ware added with results of a seismic analysis.

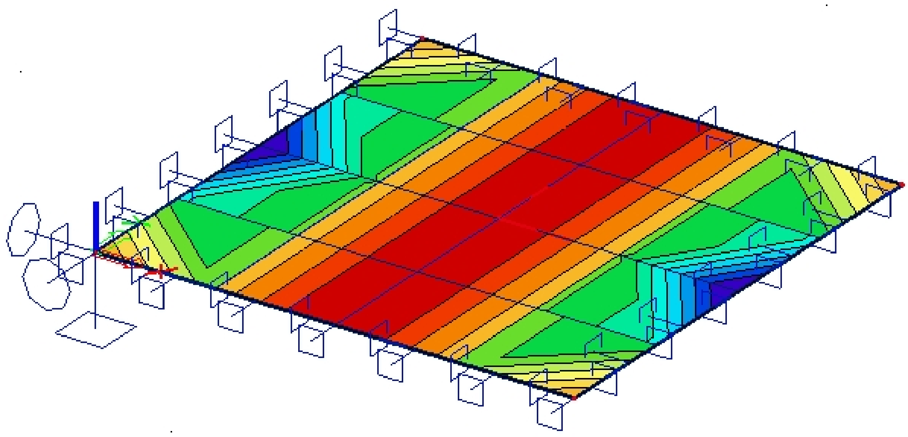
Results

Results improvements (esas.00)

The following improvements concerning results handling and interface have been done in the new Scia Engineer release. At first, the results tree has been restructured into a more logical way taking into account the new improvements on the deformations. We can take a short look at the following items from the list:

Averaging Strip for Stresses

Averaging strips were available for internal forces. Now the same tool has been developed for stresses.



2D Member name in results

The 2D member name is now shown in the table output to see exactly in which 2D slab the FEM element with corresponding result is located.

2D member - Internal forces

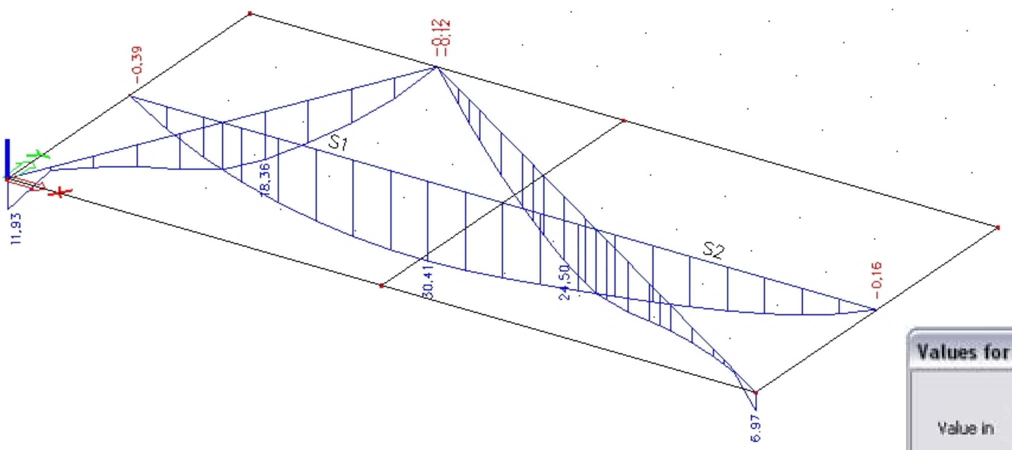
Linear calculation, Extreme : Local
Selection : All
Combinations : CO1
Basic magnitudes. In nodes, avg. on macro.

Case	Member	elem	mx [kNm/m]	my [kNm/m]	mxy [kNm/m]	vx [kN/m]	vy [kN/m]
CO1	S1	1	**11,94**	**11,93**	-2,99	-33,48	-33,61
CO1	S1	10	1,77	1,53	1,76	**18,25**	17,96
CO1	S1	91	**11,94**	**11,93**	6,26	-33,48	67,03
CO1	S1	100	1,77	1,53	-1,28	**18,25**	-8,78
CO1	S2	101	**-1,56**	**-1,61**	1,37	13,44	9,03
CO1	S2	101	-1,15	-1,15	1,88	18,18	**18,29**
CO1	S2	110	**6,98**	**6,97**	3,86	38,77	-12,70
CO1	S2	191	**-1,56**	**-1,61**	-1,88	13,44	**-18,29**
CO1	S2	200	**6,98**	**6,97**	-1,21	38,77	38,80

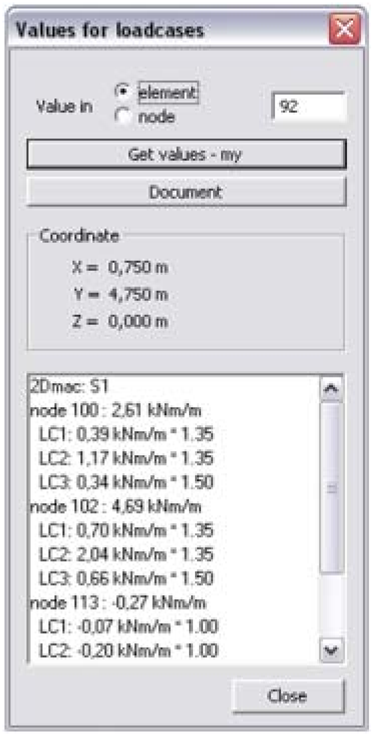
2D Member Shortcut buttons

In the same way as for 1D members, shortcut buttons are now available for 2D members.

2D Member Local extremes

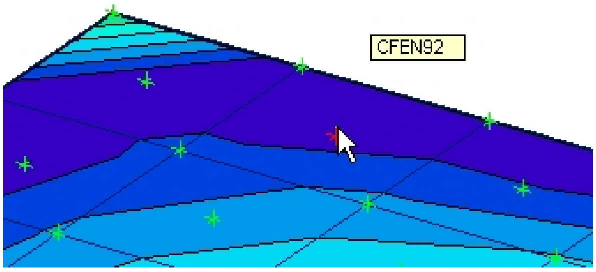


Visualization of results using local extremes is now available for sections in 2D elements. The max/min values for local parts of a section are now visible.

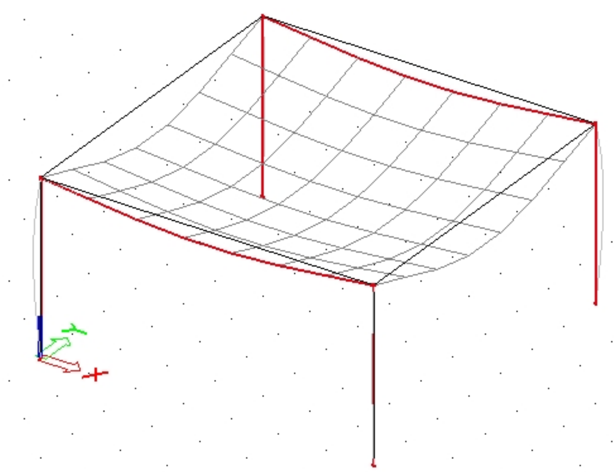


2D Member Detailed results in mesh elements or vertexes

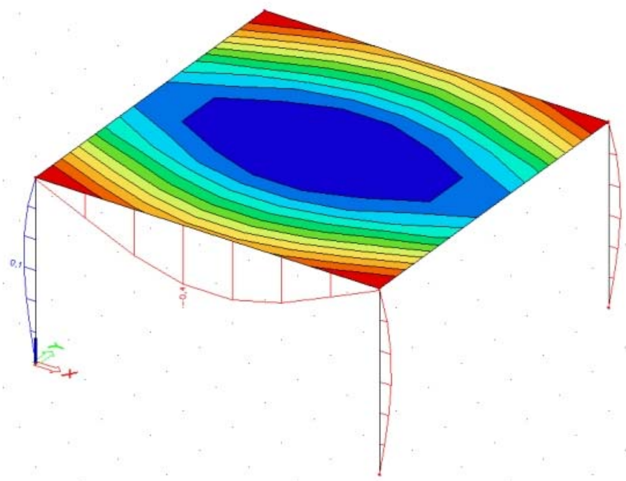
Results of a FEM mesh element or mesh node can now be viewed directly by simply clicking on a finite element node (FEN) of the mesh in a 2D member or on a centroid finite element node (CFEN) represented by a vertex point.



Combined 1D and 2D Deformations

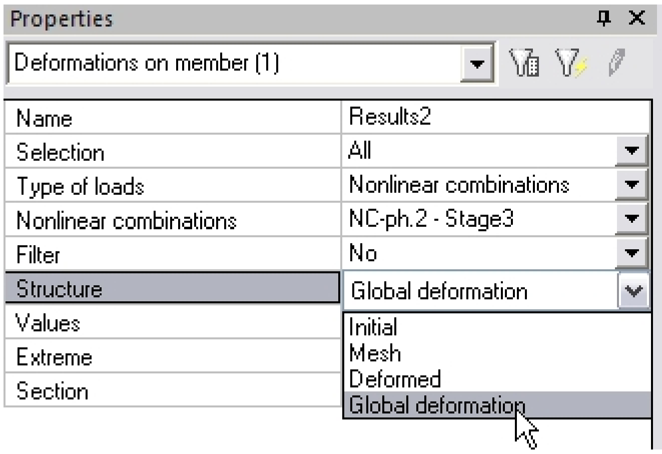


A new service has been developed, showing the combined deformation of 1D members and 2D members. In the past, only separate deformations for 1D and 2D elements were represented graphically. Now both can be shown at the same diagram.

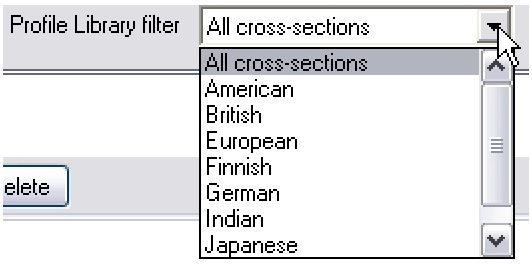


Absolute 1D member deformations & Resultant

A new structure definition is available in the service 'deformation on beams' showing absolute deformation or global deformation results instead of relative.



Filter for Profile Library (esa.00)



Because of the growing number of steel cross section types from various suppliers and/or countries in the steel section list, a filter is added to the profile library to be able to limit the number of selectable cross-section types depending on the selected region.

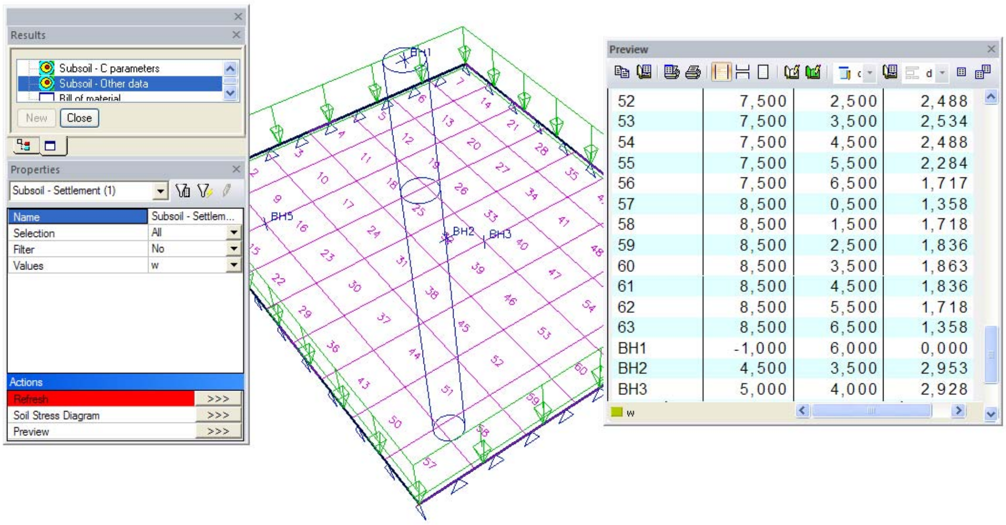
The specific cross-section user list of cross-section names or the dimension list for the optimization of cross-sections, to do the design of 1D-elements, is maintained as before.

Soilin

Soilin Improvements (esas.06)

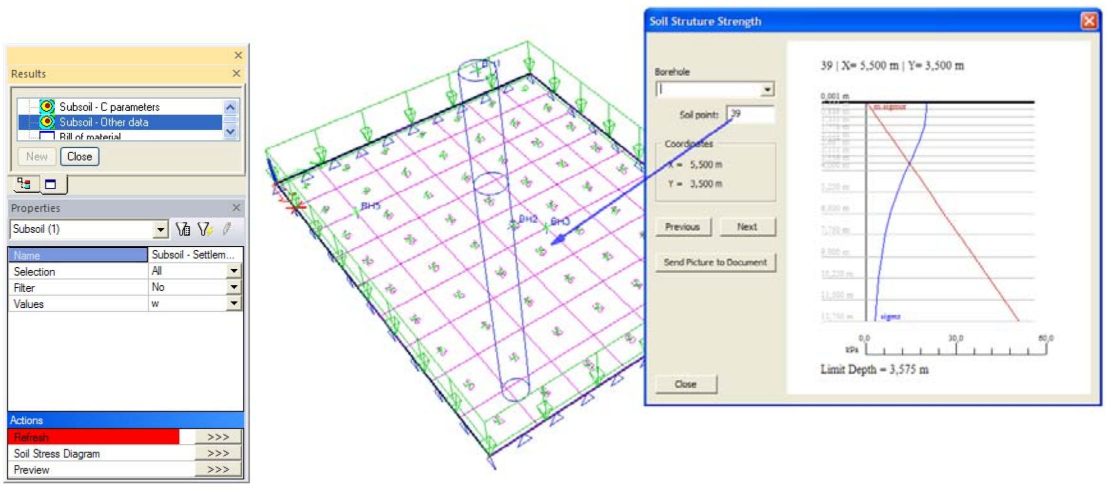
Settlements

In the results branch in Subsoil – Other data the user can view the settlements. In a preview the user can review the settlements in each finite element.



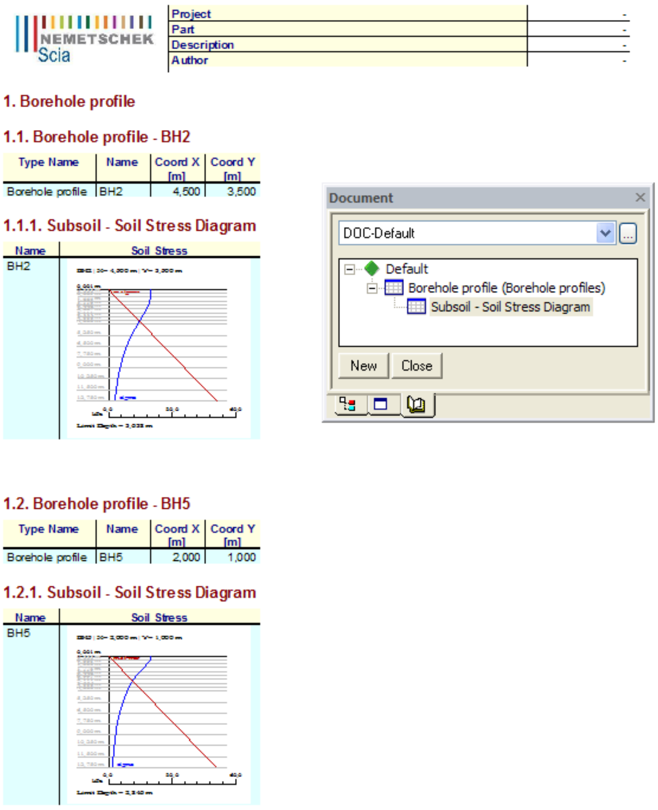
Generation of Vertexes (Soil Points)

When the user clicks on the button 'Soil Stress diagram' in the 'Soilin – Other Data service' Scia Engineer generates vertexes (Soil Points). These vertexes are generated in centroids of 2D mesh elements. Vertexes are only generated on those 2D elements which have Soilin support data defined. After generation of the vertexes, Scia Engineer prompts the user to select a vertex for which the Soil Stress Dialog should be displayed.



Document ChapterMaker

The soil stress dialog can be inputted in the document of Scia Engineer. The ChapterMaker can be used in combination with the table borehole profiles.



Design of concrete structures

Miscellaneous improvements

Improved Recalculated Internal Forces for Design and Checks (ESACD.01)

In 2008.1 the recalculated internal forces are available in each branch of the concrete design and checks. It means that the recalculated shear force due to support conditions is used in design and checks or that the recalculated internal bending moment M_y is used in design, crack proof and response check.

Improvements in Column Design (ESACD.01)

Series of improvements have been done to solve issues regarding the design of reinforcement in columns. Defaults should now automatically lead to an exact design of theoretical reinforcement areas. The user can clearly distinguish the amount of required reinforcement in the corners and in the directions y and z (for rectangular shaped cross-sections).

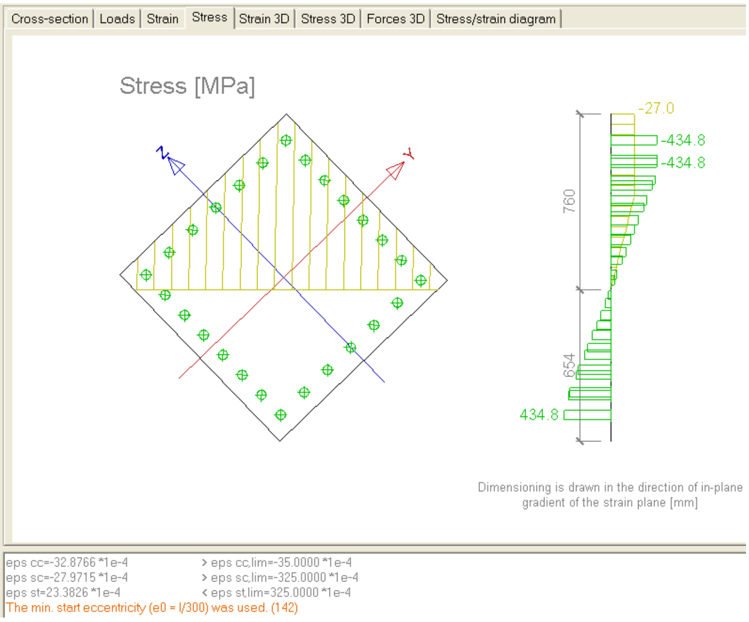
Design As NEN 6720

Linear calculation, Extreme : Member
Selection : All
Load cases : LC2

Main reinforcement for selected columns

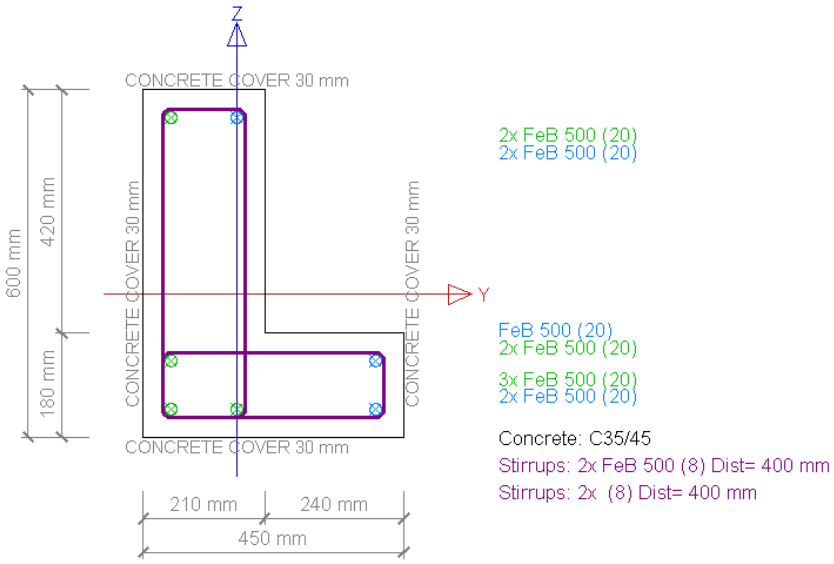
Member	d_x [m]	Case	N_d [kN] M_{zd} [kNm]	M_{yd} [kNm]	$A_{s,req}$ [mm^2] $A_{s,user}$ [mm^2]	$Reinf_{cor}$	$Reinf_{ed,y}$ $Reinf_{ed,z}$	$Reinf_{tot}$	W/E
B1	0,000	LC2	-10000,00	4156,02	0	4d40(FeB 500)	10d40(FeB 500)	26d40(FeB 500)	142
			4156,02		32673		12d40(FeB 500)		

Secondly, it is now possible to perform checks including the recalculated internal forces due to code-required 2^{nd} order effects. Correctly recalculated forces are taken into account automatically and the user can check his design for the required internal forces.



Additionally, there many improvements have been done in the calculation of eccentricities. Now the eccentricity in one direction is influenced by the eccentricity in the other direction. Special rules have been implemented to define the maximum eccentricities per direction, $e_{y,max}$;$e_{z,max}$. Also a new reinforcement design method for uni-axial calculation has been introduced. In the summaring method the reinforcement will be separately designed in both directions and summarized. In the maximizing method, the reinforcement will be separately designed in both directions and only maximum will be evaluated.

Finally, it is now possible to perform a design of L- and T-shaped cross-sections. In a reinforcement template, the user defines the locations of the bars. By gradually increasing the reinforcement diameters for the bars in the corners the program checks the cross-sections and if it satisfies. Therefore, Scia Engineer has now an effective solution for the design of columns and shear walls in seismic regions.



Main reinforcement for selected columns

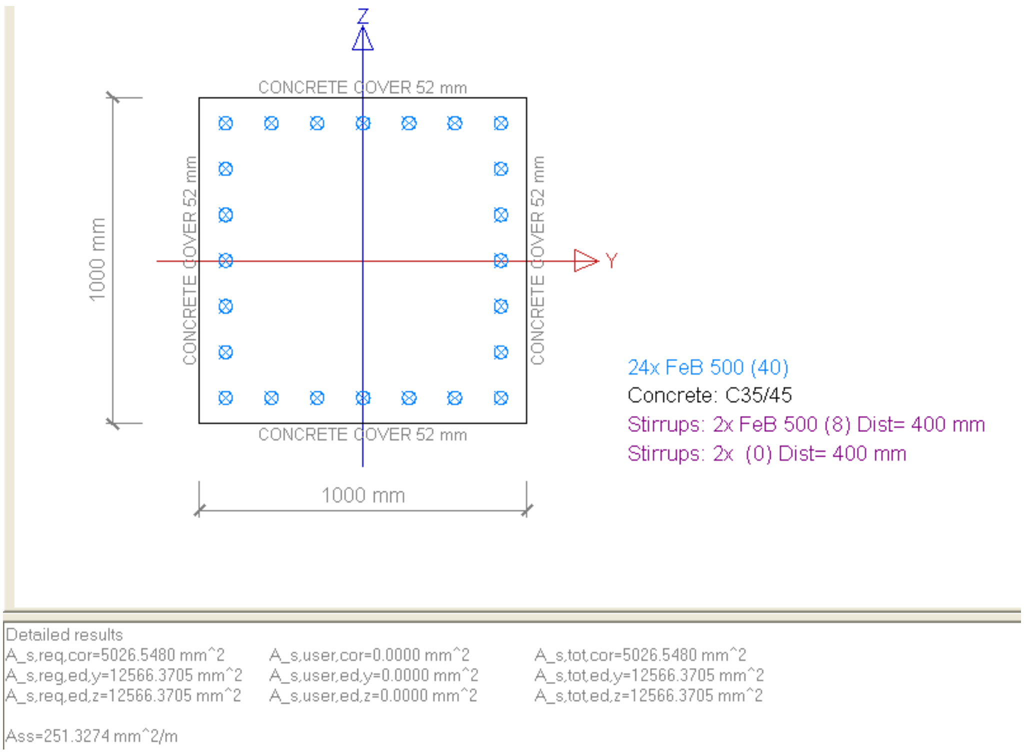
Member	d_x [m]	Case	N_d [kN]	M_{yd} [kNm]	M_{zd} [kNm]	Calc. type	$A_{s,req}$ [mm^2]	$A_{s,user}$ [mm^2]	$Reinf_{req}$	$Reinf_{tot}$	W/E
B1	0,000	LC1	-500,00	31,87	30,17	O	1571	2199	5x20,0	5x20,0(1571)+7d20(FeB 500)	142

Explanation of warnings and errors

142	The min. start eccentricity (e0 = l/300) was used.

Single Check for Column Design (SnapCheck) (ESACDT.01)

The check for single sections in a column has been improved. Today, the user can clearly see the locations of the reinforcement scheme and check the number of bars per edge of the cross-section. A numerical overview is given with the reinforcement amount per direction.



Improvements for EN 1992-1-1 (ESACD.01.01)

The omega value (reinforcement ratio), acc. to Clause 5.8.3 is now taken into account in the SnapCheck of columns. This results in more adequate recalculated internal forces. Moreover, if the user estimate is underestimated relatively to the final design, the user obtains a warning that the estimate was incorrect.

Improvements for EN 1992-1-2 (ESACD.07.01)

A new interaction diagram has been introduced for the capacity checks according to EN 1992-1-2 (fire resistance of buildings). Due to some flaws in the EN code, Scia Engineer had responded sometimes with really strange results. With the new diagram the user can clearly see that the results in Scia Engineer are accurate, although not always as expected. (The declining branch in the concrete stress/strain diagram causes the problems with the diagrams design.)

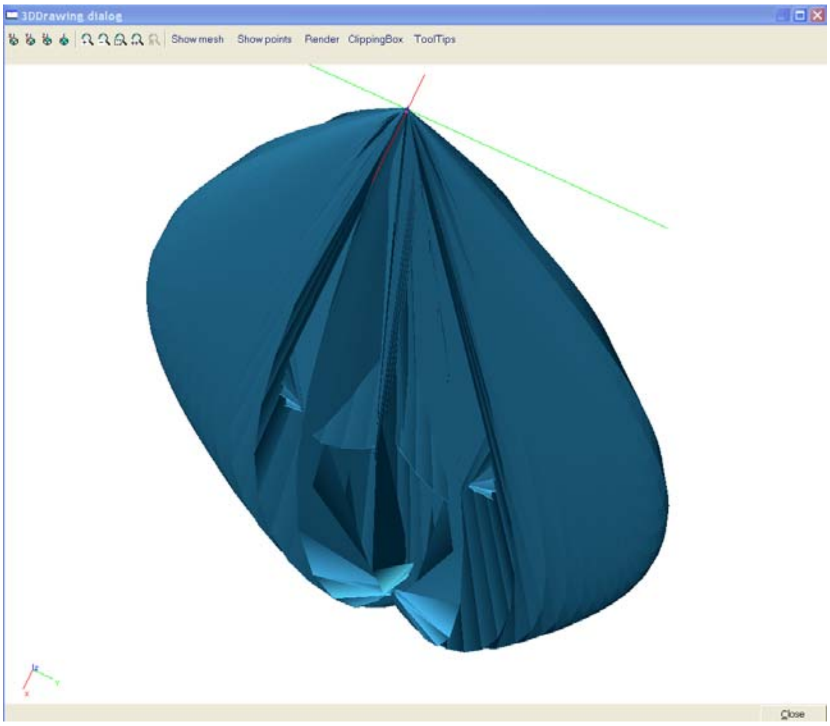
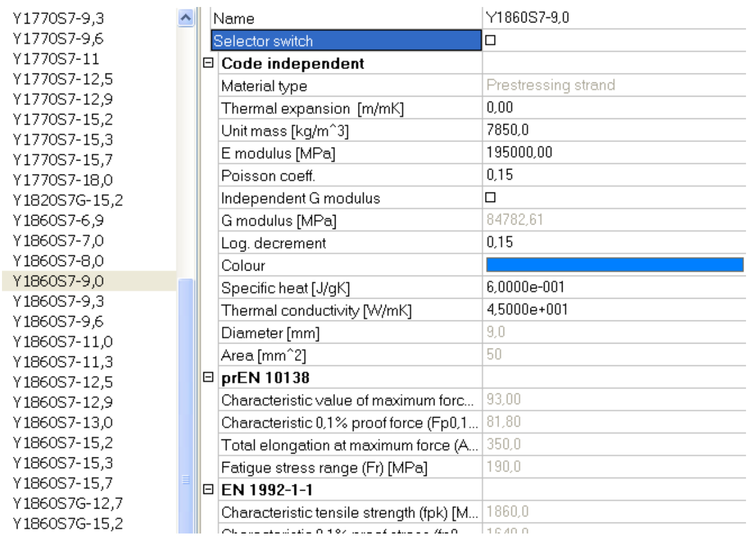


Figure 1 Interaction Diagram for a capacity check according to EN 1992-1-2.

Improvements for prEN 10138 (ESA.00)

Although still in prEN status, the diameters and material grades for the prestressing materials according to prEN 10138 have been fully implemented in Scia Engineer. If the prEN 10138 becomes official, the code will be finalized.



Improvements for EN 1168 (ESACD.06.01)

The new shear formula has been introduced, according to the revision of EN 1168.

For prestressed single span hollow-core slabs without shear reinforcement, the shear resistance of the regions uncracked by bending (where the flexural tensile stress is smaller than $f_{ctk0,05}/\gamma_c$), the shear resistance should be calculated with the following expression:

$$V_{Rdc} = \frac{I b_w(y)}{S_c(y)} \left(\sqrt{(f_{ctd})^2 + \sigma_{cp}(y) f_{ctd}} - \tau_{cp}(y) \right)$$

where

$$\sigma_{cp}(y) = \sum_{t=1}^{n} \left\{ \left[\frac{1}{A_i} + \frac{(Y_c - y)(Y_c - Yp_t)}{I} \right] \bullet P_t(l_x) \right\} - \frac{M_{Ed}}{I} \bullet (Y_c - y) \quad \text{(positive if compressive)}$$

$$\tau_{cp}(y) = \frac{1}{b_w(y)} \bullet \sum_{t=1}^{n} \left\{ \left[\frac{A_c(y)}{A_i} - \frac{S_c(y) \bullet (Y_c - Yp_t)}{I} + Cp_t(y) \right] \bullet \frac{dP_t(l_x)}{dx} \right\}$$

Figure 2 Clause 4.3.3.2.2.1 according to EN 1168 revision A1.

Improvements for 2D Concrete Elements (ESACD.02)

Today it is possible to assign 2D-concrete member data to various subregions. Therefore, it is much easier to perform reinforcement design for different areas of a floor system. Also the position of the view flag can be changed easily, so the problem with the display of adjacent walls and floors is solved.

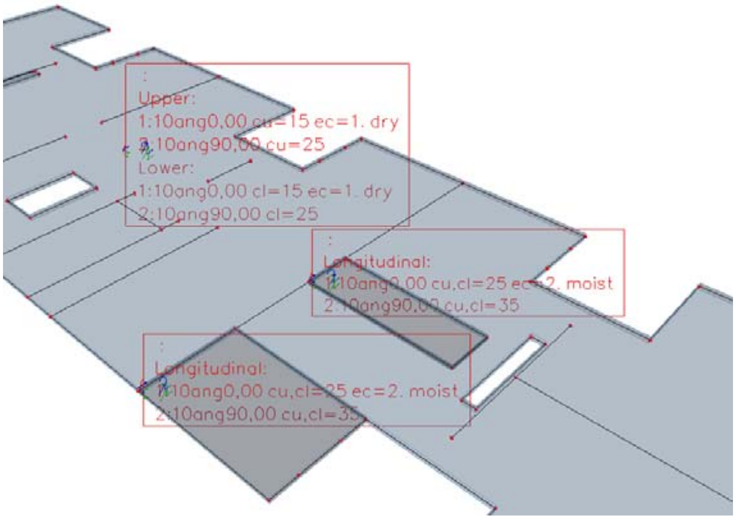


Figure 3 Concrete Member Data for the Slab and its Subregions.

Another improvement we can find in user scale isolines. The user-defined isolines can be directly accessed by the user using a library of user scales.

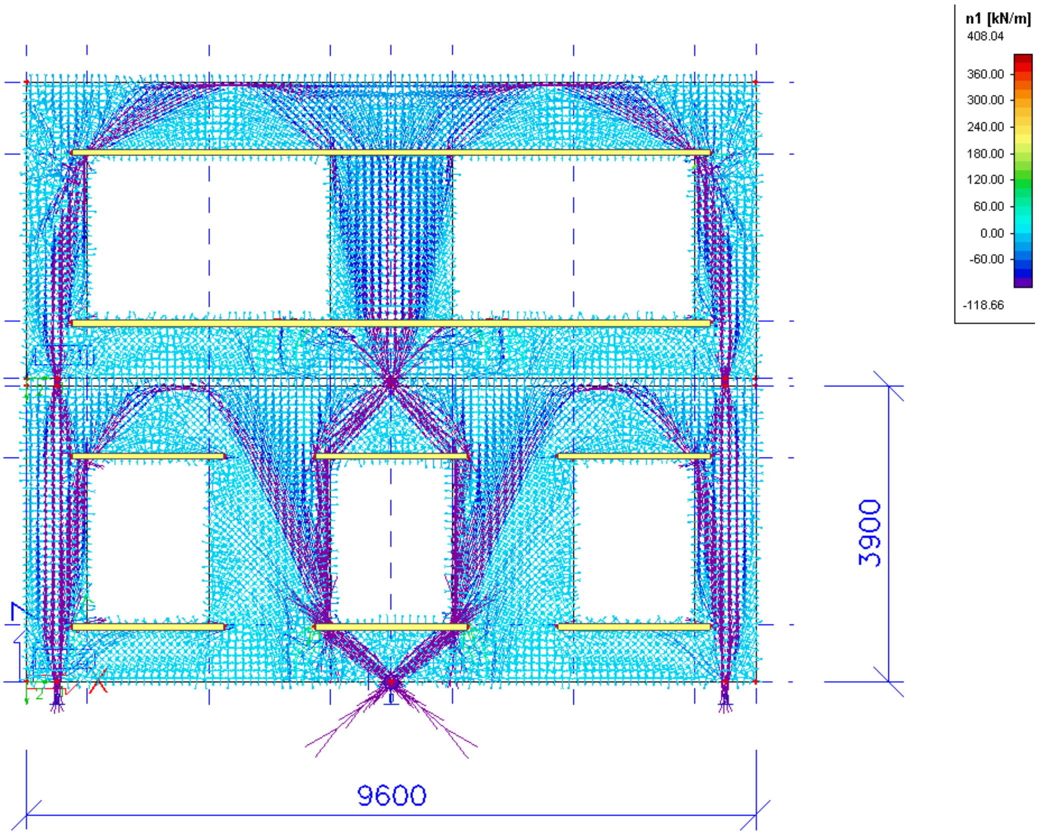


Figure 4 Pressure only analysis (using esas.44) for a precast wall.

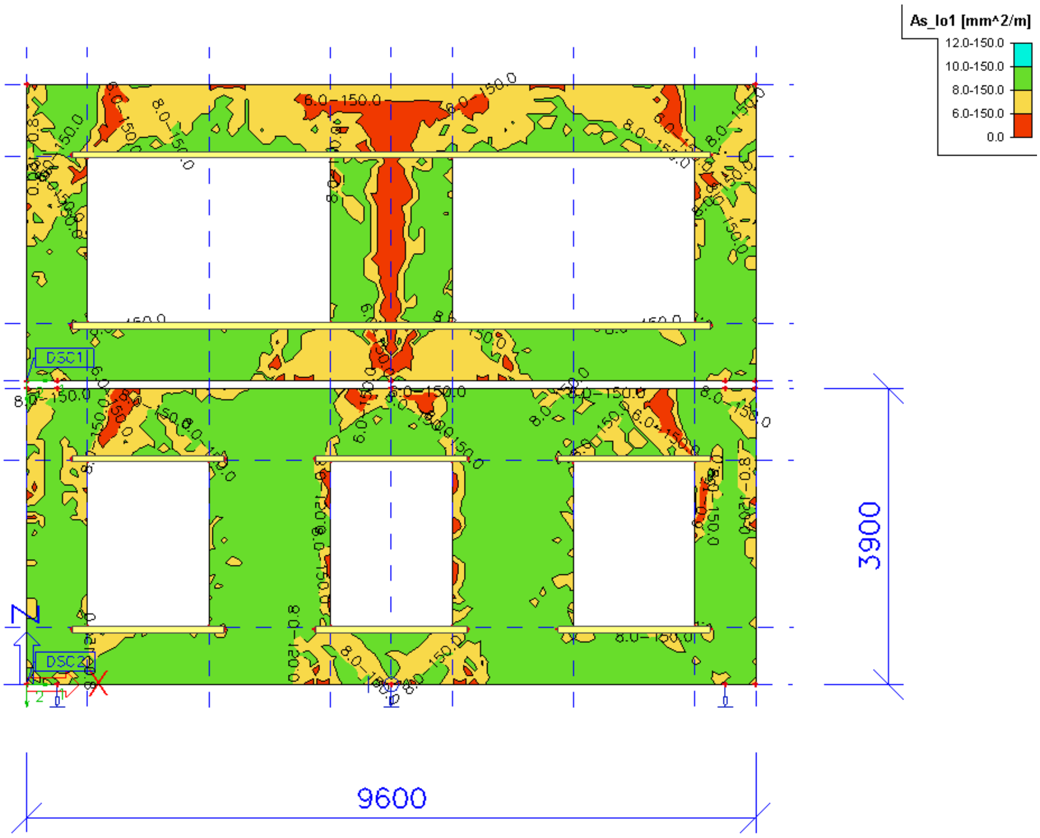


Figure 5 User scales isolines for reinforcement according to EN 1992-1-1. (after pressure only analysis).

Code Dependent Deflections 1D/2D (ESAS.18/ESAS.19)

The deflections calculated according to stipulations of the code have been renamed from Physical Non-Displacement to Code Dependent Deflections. We hope that this terminology will cause less confusion with the module Physical Non-Linear Internal Forces (ESAS.16). In the code dependent deflections, the deflections of a beam/slab are calculated using stipulations of the code, in the module PNL by using the new stiffnesses of the cracked cross-sections.

For the EN code the additional and total deflections are now calculated for 1D and 2D-elements. Both deflections (additional and total) can now be checked according to the values in the concrete setup.

The calculation for 2D-elements with or without practical reinforcement areas always takes into account the total maximum amount of reinforcement per finite element.

Check of Joints for Composite Cross-section according to NEN 6720 (ESACD.01.03)

Today it is possible to define a joint and set the appropriate properties (smooth, rough) for a phased cross-section and perform a check according to clause 8.2.5. of NEN6720.

Allplan

Allplan Improvements (esa.28)

More ways to import the structure from Allplan, also solids are implemented in interface

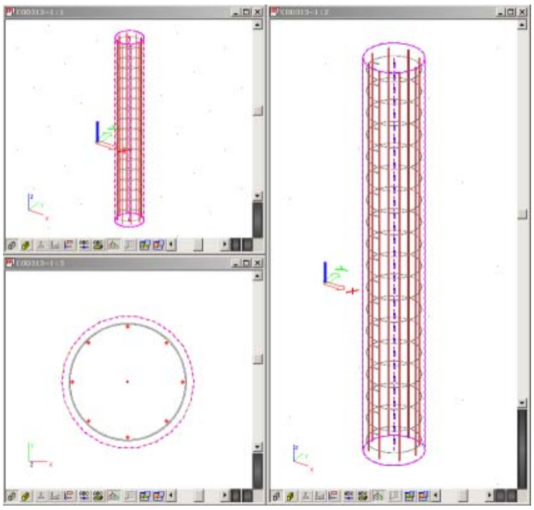
Member recognizer allows the user to handle shapes created in Allplan using clipping easier than direct transfer, because it works primarily with the whole 3D volumetric shape. Therefore, e.g. skew walls and slabs, as well as walls or slabs with large openings, are handled with respect to their final shape, and not to the way how they were created in Allplan. Member recognizer has been introduced to Allplan – Scia interface to be run directly during import.

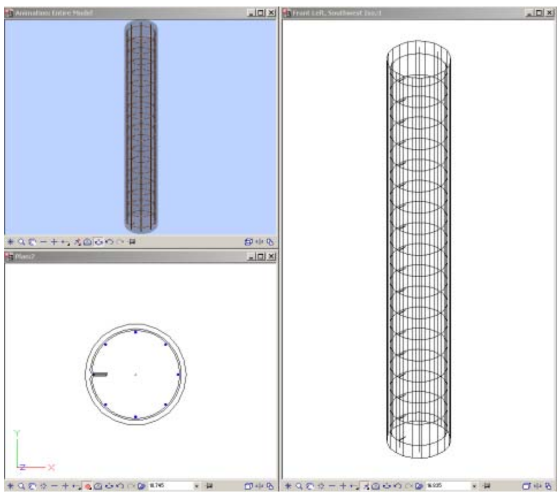
It is possible to choose way, how to import the structure from Allplan with respect to recognizer function. It is possible to choose one of following options:

- standard import (same as in Scia Engineer 2008.0 and previous versions),
- import all architectural members as solids. It is expected that the user makes members from solids using member recognizer functions. This is the safest way of creation of analysis model from the shape imported from Allplan, because the procedure is fully under control of the user and he can decide about the most suitable analysis model for each entity (for example if a narrow architectural wall is to be modelled as a wall or as a column etc.) When handling with large and complex structures consisting of many parts, it is possible to use multiple selections with filtering with respect to member type or materiel etc.
- input architectural members as solids and run recognizer directly during the import. This fastest way is useful in case of very good quality of imported architectural model with respect to its topological correctness, precision and proper dimensions.

Export of circular stirrups to Allplan (in templates)

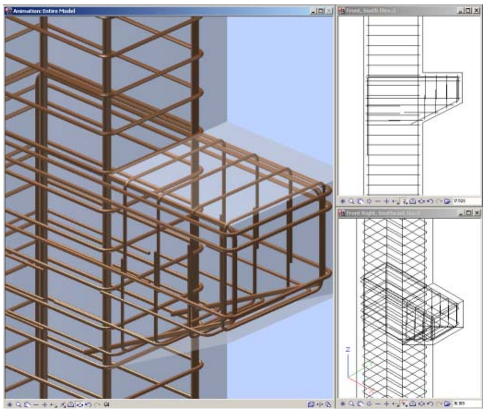
Circular stirrups are allowed in Scia templates.

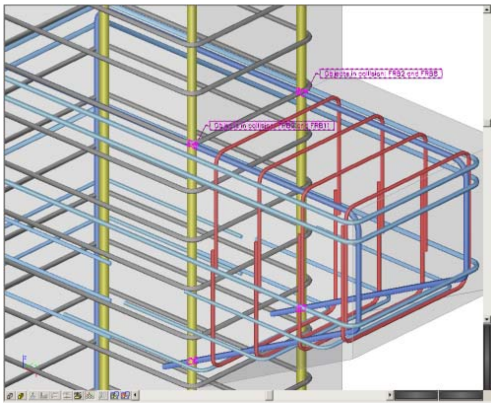




Clash check improvements

Clash check has been extended in version 2008.1. It is possible to run clash check not only for structural members and solids, but also for reinforcement bars. So, it is possible to import structure with reinforcement bars from Allplan to Scia Engineer and perform clash check for the whole geometry of structure (formwork) as well as for reinforcement bars.





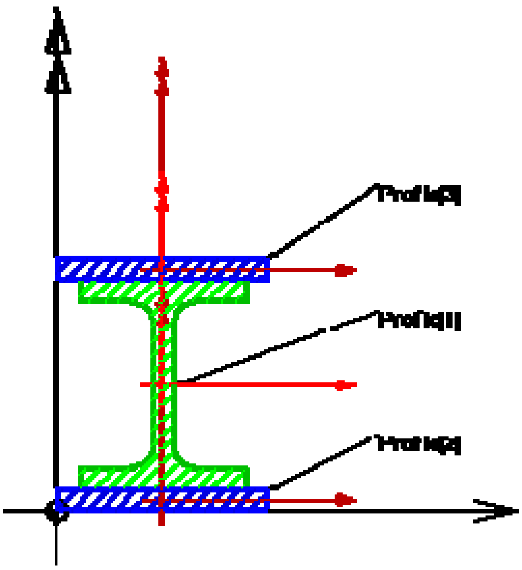
IFC improvements

Import / export improvements (esa.00)

Import and export of welded and composite cross-sections

Composite cross-sections can now be imported into Scia Engineer.

All cross-sections defined in Scia Engineer and consisting of several partial cross-sections (welded, composite, built-in, etc.) are exported as composite cross-sections.



Import of rolled cross-sections

If a rolled cross-section is stored in the IFC file including its name, it is imported into Scia Engineer as a rolled cross-section.

Code selection

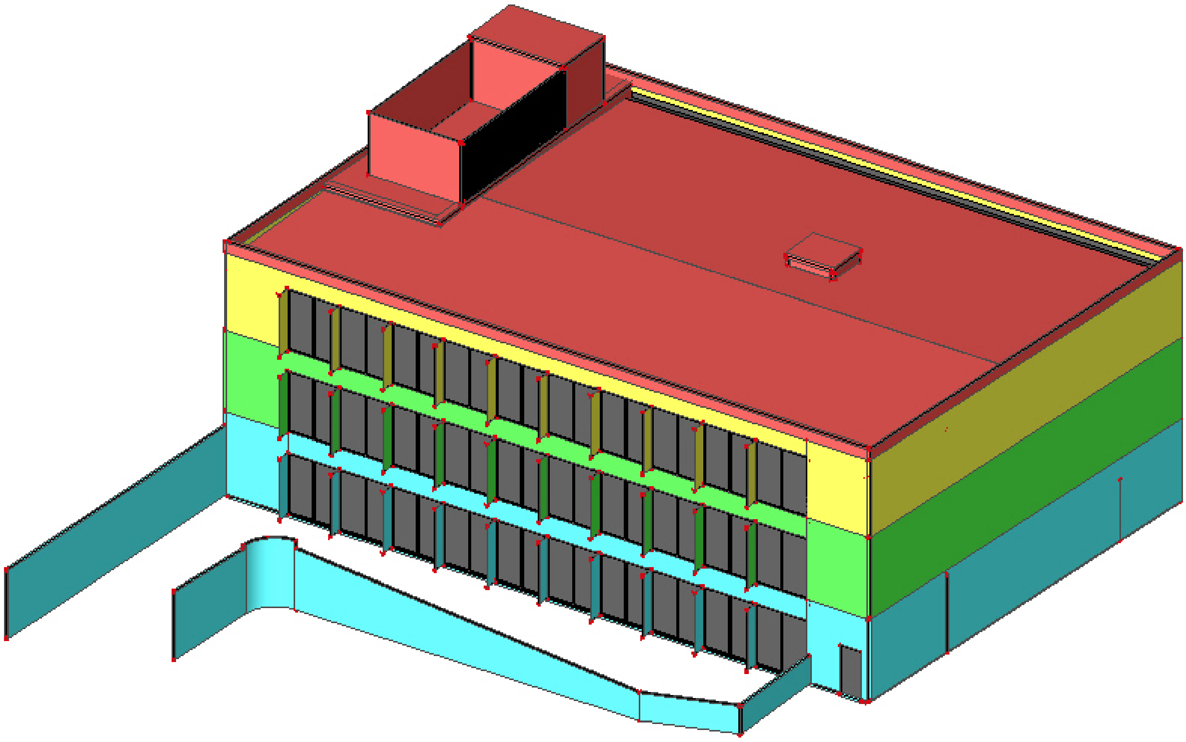
During import, the user can select national standard for the project.

Export of 2D members with cut-out

2D members defined in Scia Engineer with cut-outs are now exported to IFC file.

Layers as storeys

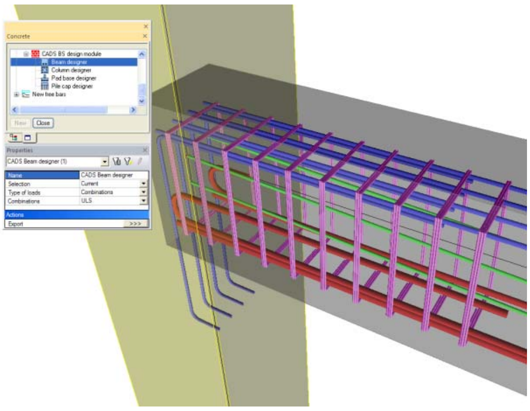
IFC building storeys are now imported as layers. Similarly, when a project is exported into IFC, layers are exported as IFC building storeys.

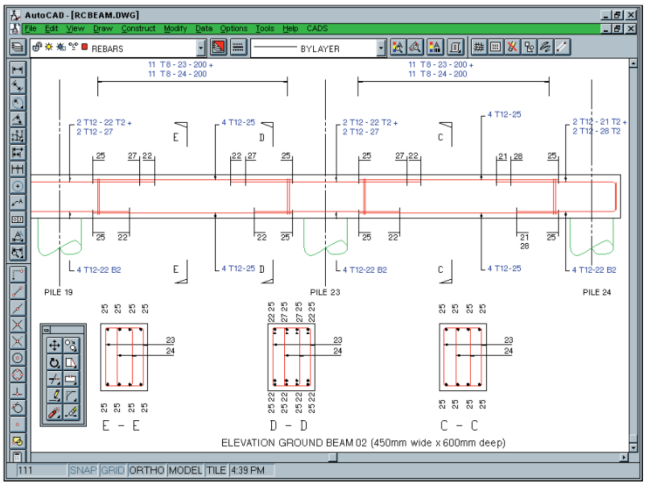


CADS RCBD modules integration

Reinforced Concrete Beam Designer (ESACD.01.08)

With this updated feature it is now possible to use Scia Engineer together with CADS RC software. After making the design in Scia Engineer the user can use the CADS RC software to make the detailing as shown on the figure below (more info can be found on http://www.cadsglobal.com/cadsrc.html).

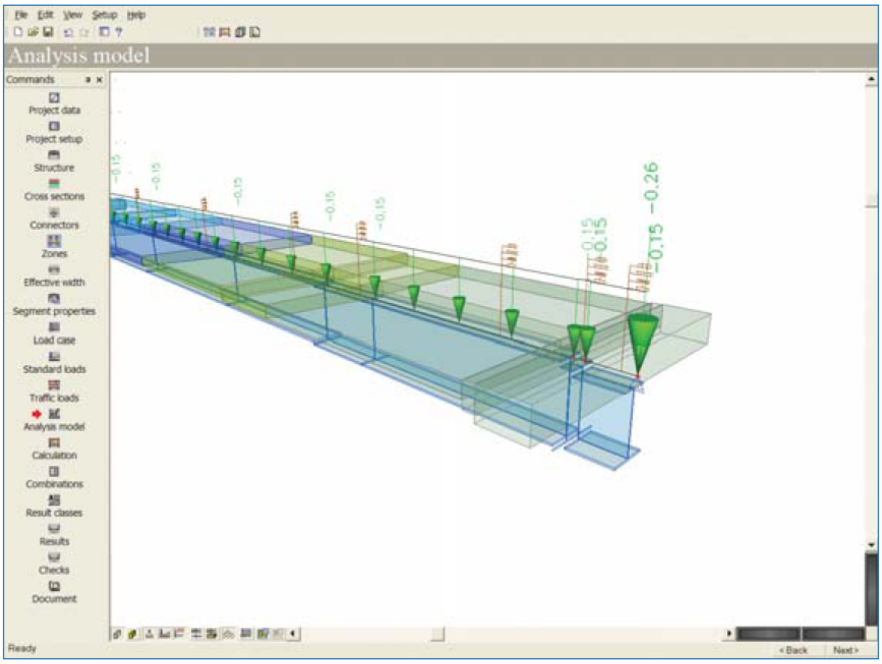




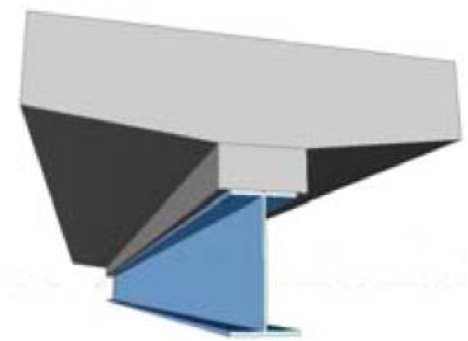
ODA applications

Mixbeam (esamd.00, esamd.01.06)

Scia Mixbeam is a specific program made for the calculation of composite beams. It calculates internal forces and determines automatically the stresses in the composite sections in construction stages. Specific checks for composite beams can also be performed.



Based on the environment of Scia Engineer with state of the art technologies, Mixbeam represents a simple interface dedicated for the calculation and design of composite bridges. Before calculation and checks, the program helps the user input easily the structure, loading, combinations and construction stages.



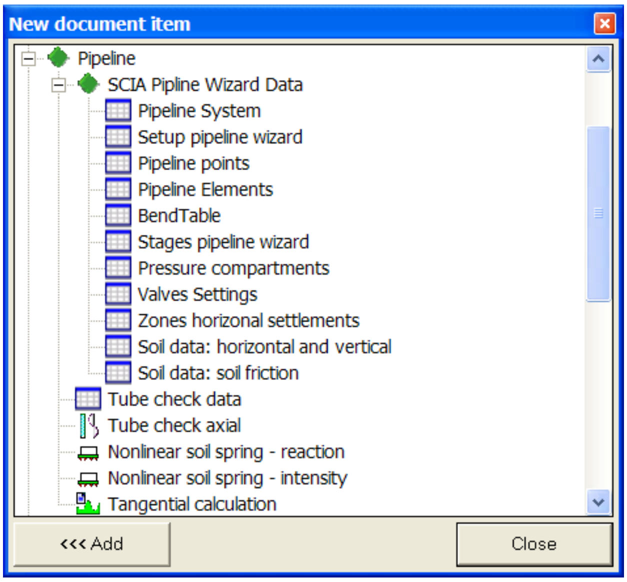
The whole study can be done in a very structured way and graphically. This gives the user a method to work faster and with fewer errors. The project made in Mixbeam can also be imported to Scia Engineer. In Scia Engineer the user is able to adapt the whole project in the well known Scia Engineer environment.

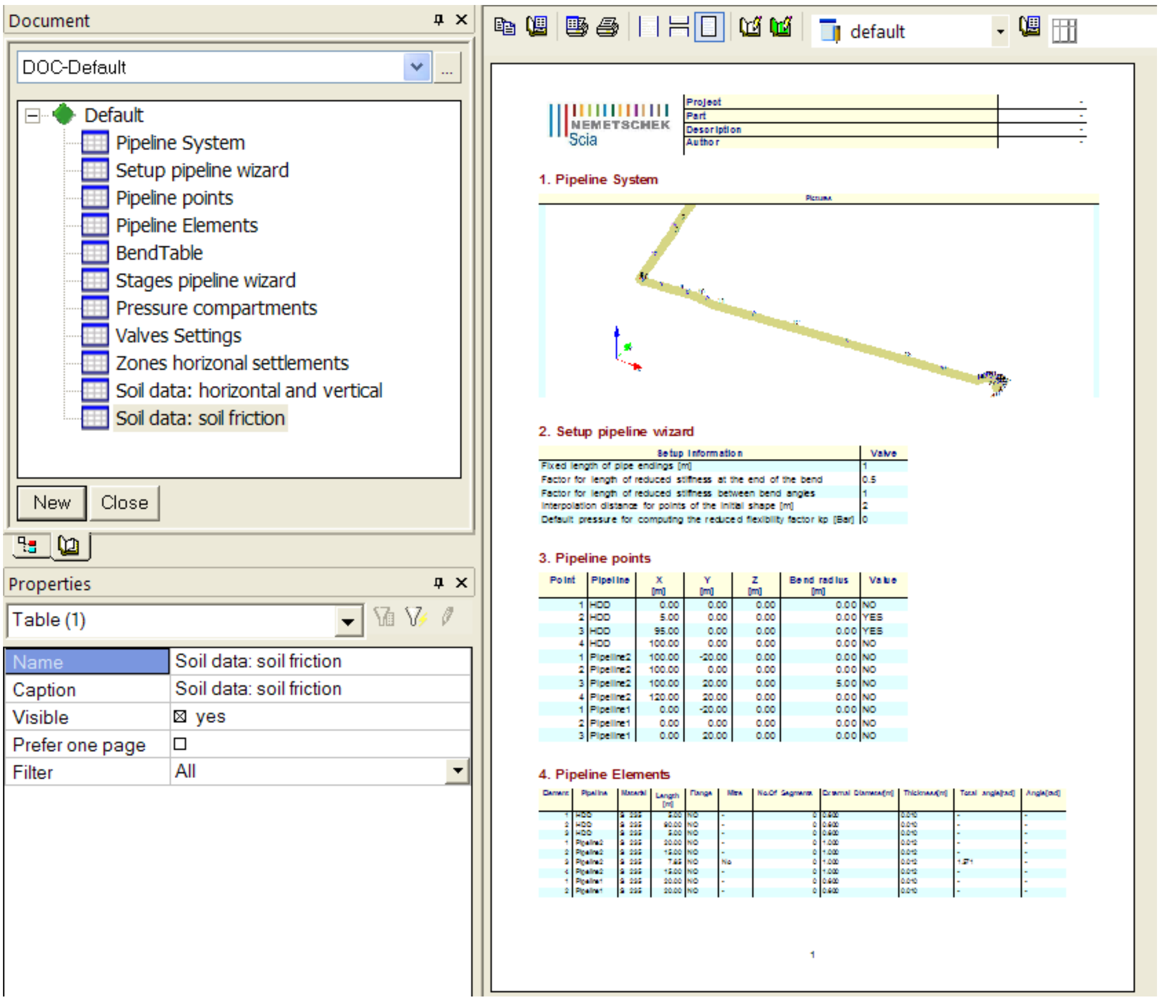
Special modules

Pipelines improvements (esas.31, esas.28)

Document

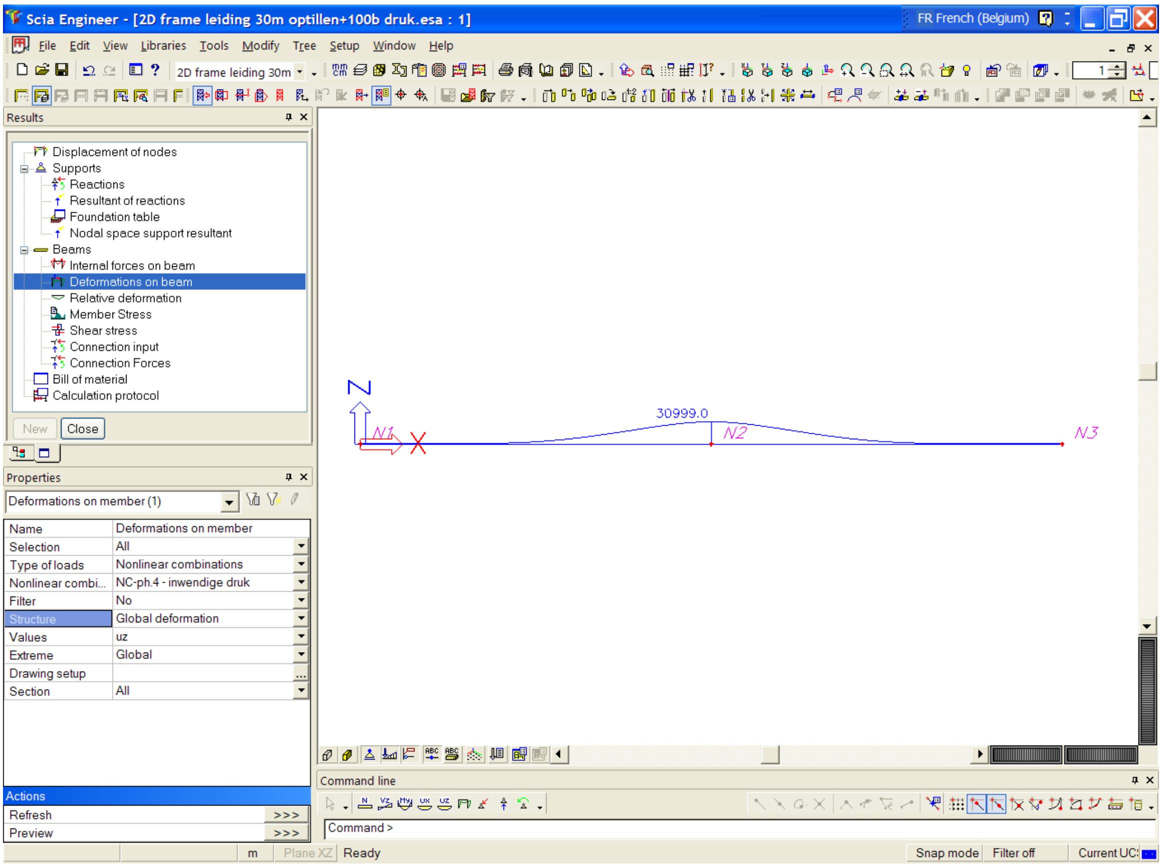
New Pipeline items can be inputted in the document of Scia Engineer.





Deformations

It is now possible to review the real (absolute) deformation of the pipeline in a certain stage.

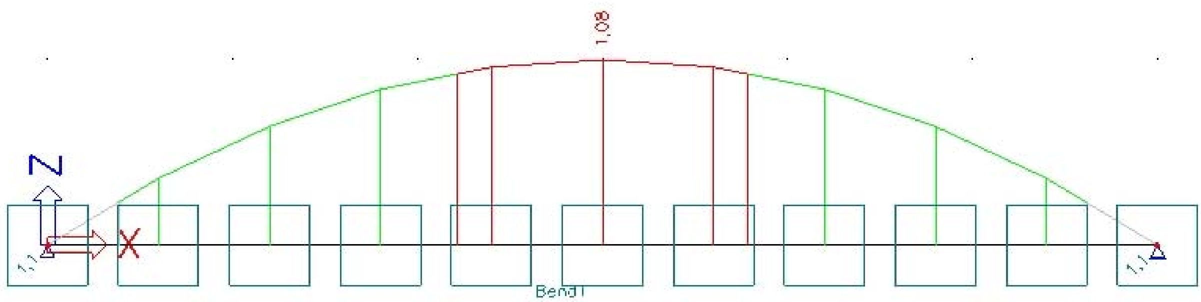


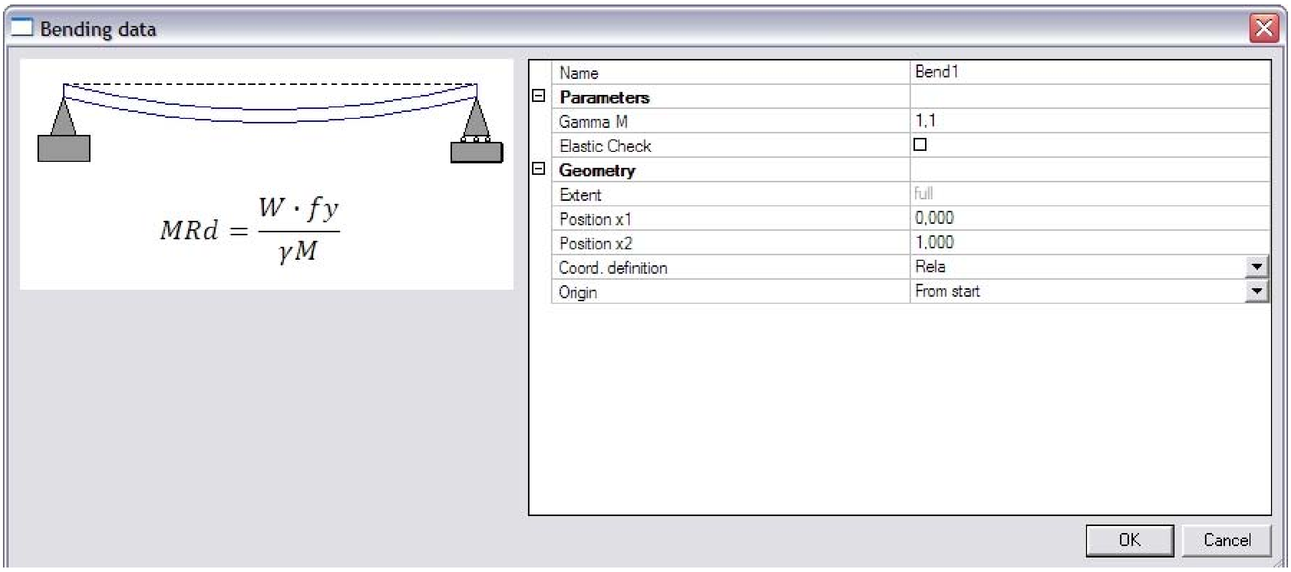
External application checks for MS Excel (esa.06)

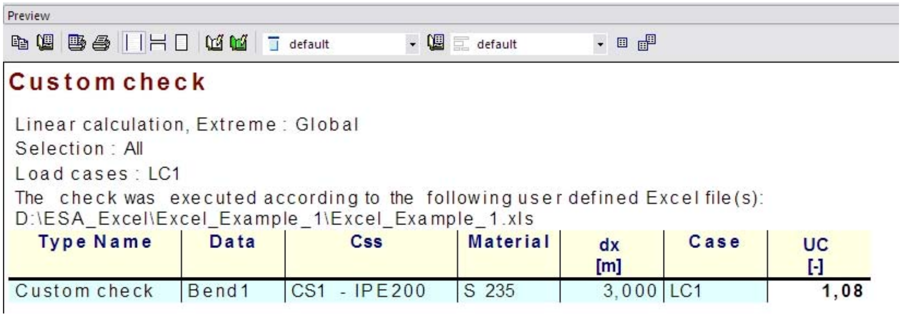
With this brand new module, a Scia Engineer user can integrate input for and calculation output from their MS Excel files for nodes and 1D members into the general Scia Engineer environment. The final outcome will be similar to the actual advanced checks in Scia Engineer and input and results can be visualized in the current graphical Scia Engineer user interface and in the document.

A lot of engineers still perform a number of (advanced) company specific or user checks into MS Excel. The intention of this module is to enable these special checks into Scia Engineer. The user is able to send input data and intermediate results from Scia Engineer (like internal forces, members data, dimensions …) to the Excel file(s) and to obtain the results of this MS Excel calculation graphically on the beam or in a table in the document. A copy of a part (a range of cells) of the MS Excel file can be added into the document.

The external application check is created as additional data on a 1D member, containing all the necessary mapping information for defining the connection to specific cells in the user appointed MS Excel files. Once the link to a MS Excel check is defined it can be stored in a library and used in all future projects.







Custom check

Linear calculation, Extreme : Global
Selection : All
Load cases : LC1
The check was executed according to the following user defined Excel file(s):
D:\ESA_Excel\Excel_Example_1\Excel_Example_1.xls

Type Name	Data	Css	Material	dx [m]	Case	UC [-]
Custom check	Bend1	CS1 - IPE200	S 235	3,000	LC1	**1,08**