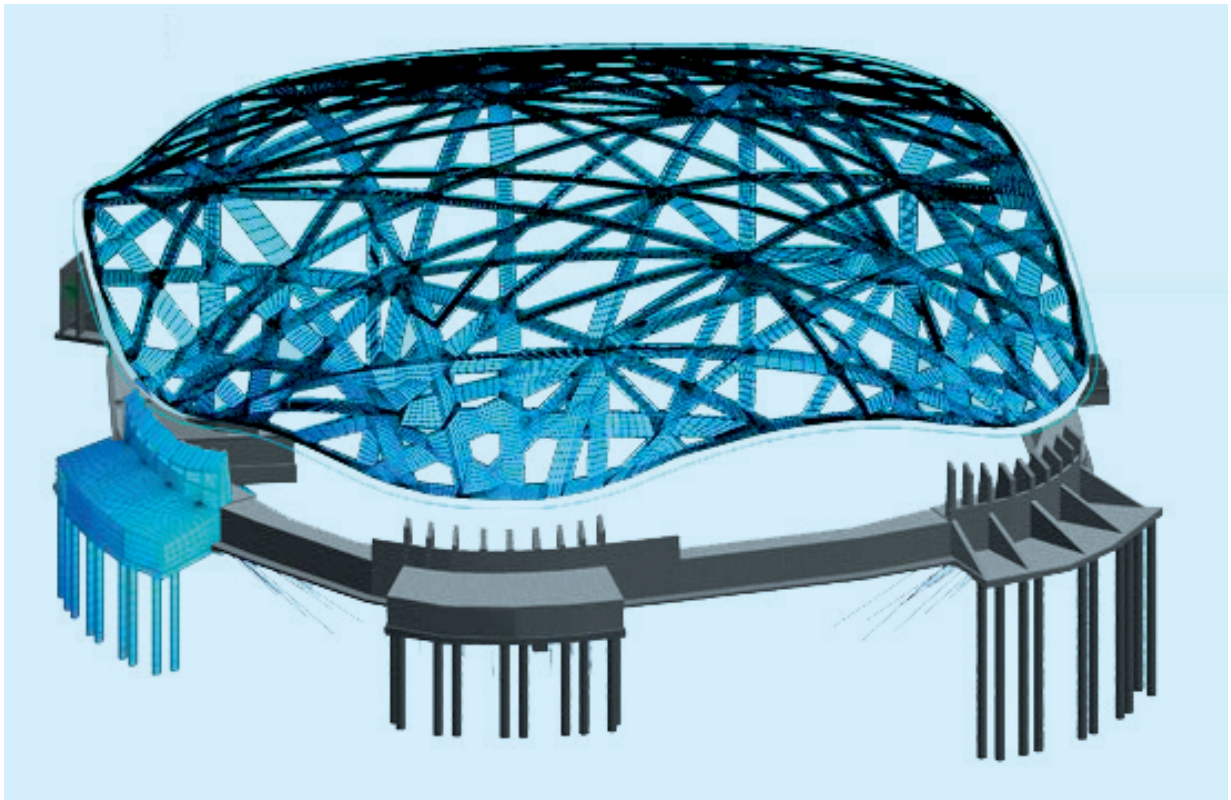
SOFiSTiK

FINITE ELEMENT SOFTWARE
VERSION 2012

New Features SOFiSTiK FEA Version 2012



Elephant park | Zoo Zurich

Walt + Galmarini AG Dipl. Bauingenieure ETH SIA USIC

1 General

System Requirements

The SOFiSTiK products in Version 2012 have been developed, tested and released for Windows XP Professional (SP3), the 32-Bit Windows Vista Business and Windows 7 Professional 32-Bit & 64-Bit.

Further information: System_Requirements

Compatibility

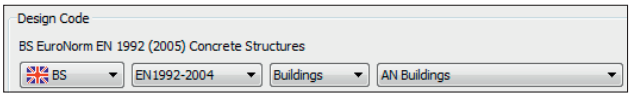
- SOFiPLUS 2012 18.2 runs on AutoCAD 2010-2012
- The SOFiSTiK Extensions require Revit Structure 2012/2012 64bit
- The Rhinoceros Interface needs Rhinoceros 4.0

Further information: Version_Matrix

2 FEA General / Text-Input

Eurocode Support

The SOFiSTiK software supports many national implementations (e.g. BS, NF, S, UNI, DK etc.) of the new EuroCodes. Officially available EuroCodes should be found listed below the respective institute (resp. country flag in the 'System Information' dialog). The description of the code contains also the year of the base-document, an additional info text contains the release year of the national amendment as well. The selection of the latest EuroCode 2 (i.e. NA to BS EN 1992-1-1:2004) for the U.K. should yield:



Further information on code support and features should be obtained from the manuals of the modules AQUA, AQB and BEMESS explanations and table for record NORM.

Improved handling of units

The text-input allows specifying the unit of an input value using brackets (e.g. 0.5[in] resp. 500[mm]). This option is available for all inputs where the manual shows a related description in the 'Unit' column.

Three categories of units are distinguished and described in the SOFiSTiK documentation:

m	Fixed unit. Input is always required in the specified unit
[mm]	Explicit unit. Input defaults to the specified unit, but may be provided with explicit assignment of a related unit (eg. 2.5[m]).
[mm]1011	Implicit unit. The actual unit for the identification number is assigned by the currently active (design code specific) unit set. Specified here is the default for unit set 5 (NORM UNIT 5). As for the explicit units, the default may be overridden by explicit unit assignment.

3.23. SREC – Rectangle, T-beam, Plate **SREC**

Item	Description	Unit	Default
NO	Cross section number	–	1
H	Total height	[mm]1011	–
B	Width for rectangular, T-beam	[mm]1011	1[m]
HO	Thickness of the plate (upper part)	[mm]1011	0
BO	Thickness of the plate (lower part)	[mm]1011	0
SO	Offset of top reinforcement	[mm]1024	H/10
SU	Offset of bottom reinforcement	[mm]1024	SO
ASO	Minimum top reinforcement- layer 2	[cm²]1024	0

```
prog aqua
head
norm bs en1992-2004

srec

end
```

<Neue Datei>1

NO H[mm] B[mm] HO[mm] BO[mm] SO[mm] SU[mm] ASO[mm2

AQUA / Units, Openings

Fixed Units for cross-section elements have been removed (PANE, PLAT etc.).

Openings can be generated more robust using automatic polygonal intersection (POLY MNO 0).

```
QPOL U MNR 0
QP A1   0.00 0.50 REFP OM ~OR
     A2   0.45 0.50 REFP OM ~OR
     A3   2.75 0.50 REFP OM +OR
     A4   2.50 -.50 REFP UM
     A5  -2.50 -.50 REFP UM
     A6   2.75 -.50 REFP OM +OL
     A7   0.45 -.50 REFP OM ~OL
     A8   0.00 -.50 REFP OM ~OL
```

SOFiMSHC Meshing

Improved possibility for the input and meshing of NURB-surfaces and surfaces of revolution.

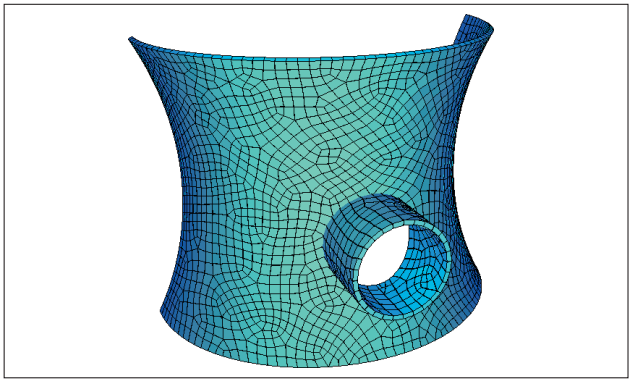
Extraction of sub-systems. Using the command XSUB, substructures can be extracted from a 3D model and the interface stiffness of vertical elements is applied as sprigs or fixed supports.

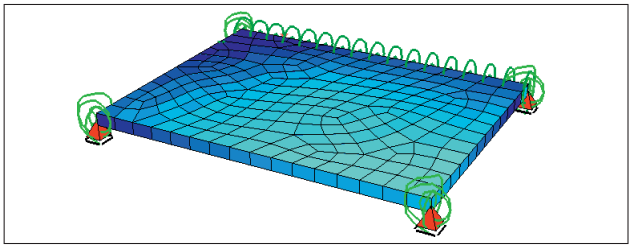
Sequential numbering of beam elements along a structural line allows easier definition of imperfection loads etc.

Definition of local coordinate systems/ working planes using the record COOR.

Double precision allows better accuracy for geometry operations (intersections, route mapping).

Improved numerical and graphical output for geometry elements (ECHO GEOM).



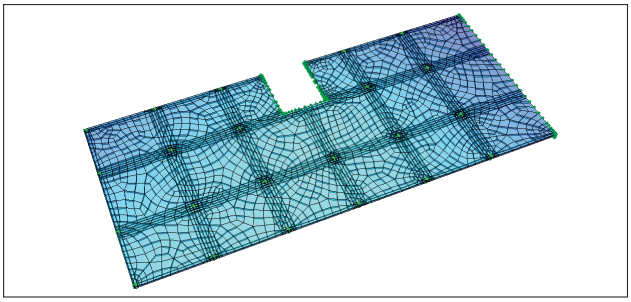


TENDON Pre- and Post-tensioning

TENDON is the successor of GEOS which is not available in 2012 anymore.

TENDON supports the axis-based CABD workflow, i.e. tendons can be defined related to bridge axes in addition to the classical reference to nodal and beam sequences.

The definition of tendons following cross-section points and general axes is now possible (PTUV CSP/ TGEO XREF).



SOFiMSHA Result Sets

Grouping several result into a new RSET ‚element' allows to easily extract e.g. coexisting bearing forces.

Application:
Input in SOFiMSHA (SYST REST)
Calculation in MAXIMA, ELLA and DYNA
Postprocessing using text output (MAXIMA ECHO RSET)

```
+PROG SOFIMSHA urs:3
KOPF 'Result Set Definition'

SYST REST

$ RSET - definition of co-existing results

RSET ID 1 BEZ 'Elastomeric Bearings S=1.00'

RSET ID 'PXR' SET SPRI_RES ITEM P NR 100001  BEZ 'P-Long_r_S=1.00'     $ Elastomeric Bearing +y
RSET ID 'PYR' SET SPRI_RES ITEM P NR 100002  BEZ 'P-Tran_r_S=1.00'
RSET ID 'PZR' SET SPRI_RES ITEM P NR 100003  BEZ 'P-Vert_r_S=1.00'
$
RSET ID 'VXR' SET SPRI_RES ITEM V NR 100001  BEZ 'V-Long_r_S=1.00'
RSET ID 'VYR' SET SPRI_RES ITEM V NR 100002  BEZ 'V-Tran_r_S=1.00'
RSET ID 'VZR' SET SPRI_RES ITEM V NR 100003  BEZ 'V-Vert_r_S=1.00'
$
RSET ID 'PXL' SET SPRI_RES ITEM P NR 100004  BEZ 'P-Long_l_S=1.00'    $ Elastomeric Bearing -y
RSET ID 'PYL' SET SPRI_RES ITEM P NR 100005  BEZ 'P-Tran_l_S=1.00'
RSET ID 'PZL' SET SPRI_RES ITEM P NR 100006  BEZ 'P-Vert_l_S=1.00'
$
RSET ID 'VXL' SET SPRI_RES ITEM V NR 100004  BEZ 'V-Long_l_S=1.00'
RSET ID 'VYL' SET SPRI_RES ITEM V NR 100005  BEZ 'V-Tran_l_S=1.00'
RSET ID 'VZL' SET SPRI_RES ITEM V NR 100006  BEZ 'V-Vert_l_S=1.00'
```

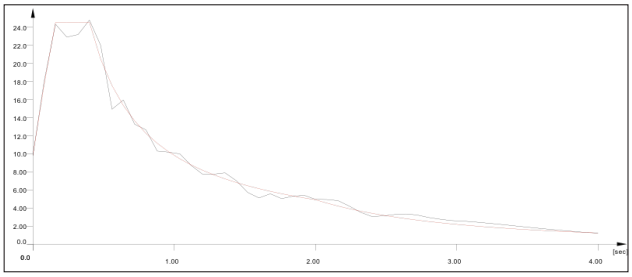
SOFILOAD Artificial Earthquake / ECHO LANE

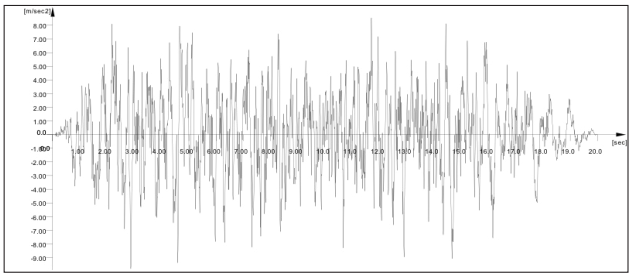
The new function SIMQ allows to generate artificial earthquake histories for further timestep simulation, the earthquakes may be defined as:

- Power spectrum
- Velocity spectrum
- Acceleration spectrum
- The transient character of an earthquake is considered using an intensity function.

The input of loadfunctions (FUNC) has been revised, especially w.r.t. the addition and multiplication of functions.

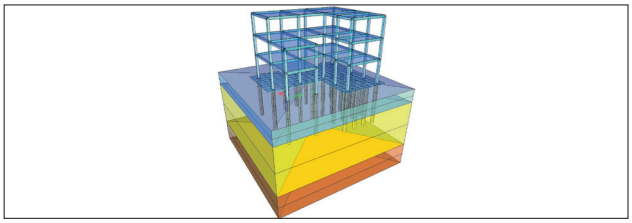
Output for traffic lanes has been extended and improved (ECHO LANE).





HASE/ ASE Half Space Soil-Structure-Interaction

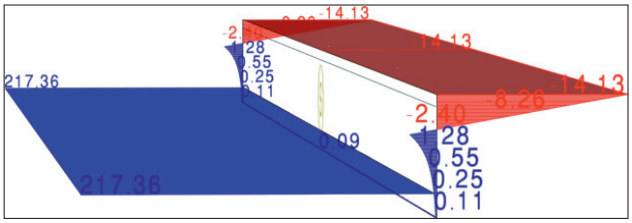
3D Interpolation of bore-profiles/soil-layers: Using input data of several bore-profiles/soil-layers the program is able to perform an automatic 3D 'layer'-interpolation to simulate the soil stiffness more realistic.



TALPA Fiber Beam (New 2D Beam Element)

Internal subdivision of the beam cross-section into ‚material-fibers' allows the consistent integration of the material response, this allows:

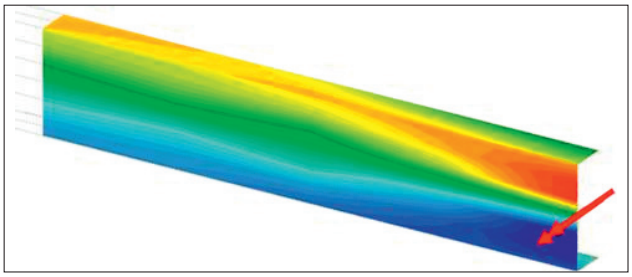
- Nonlinear material behavior for R/C and steel materials.
- Damage model for cracked concrete and bond simulation as we as tension stiffening.
- Temperature dependent material response and temperature strains following EN 1992–1–2:2004.



DYNA New Shell Elements and Nodal Results

New shell elements with drilling degrees of freedom allow the consistent connection of beam and shell elements.

Nodal results are now available for e.g. Eigenvalues.



Parallel Computing

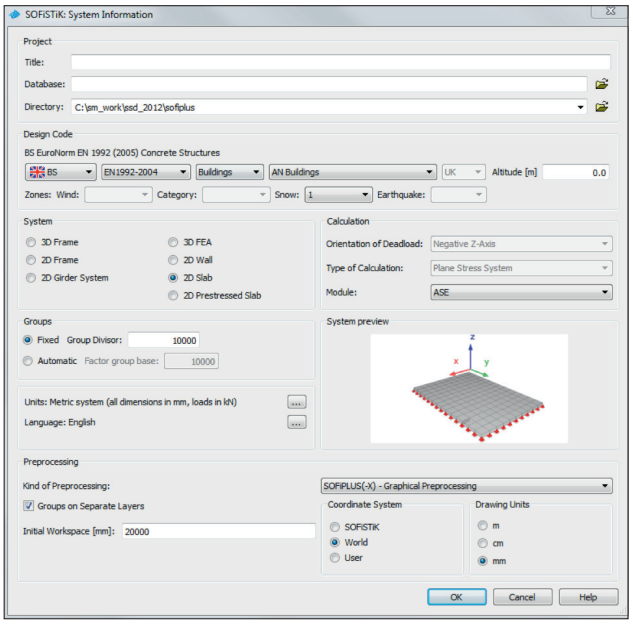
Optimized settings and overhead reduction for parallel processes provide faster computation time.

3 SSD / Graphical User Interfaces

System Information Dialog

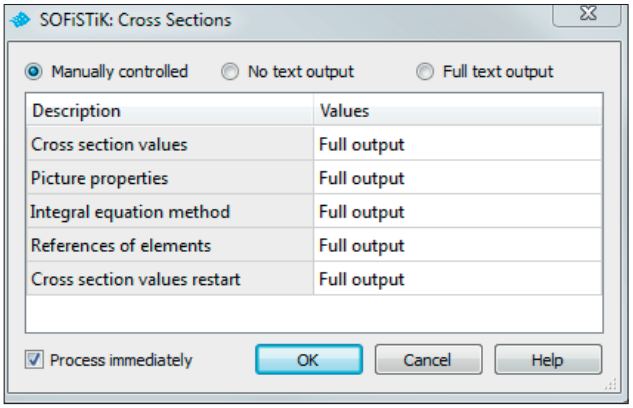
The System Information dialog comes with several improvements:

- Sorting of available .ini files according to their relevance and date. Latest and most relevant codes will be displayed first
- Automatic grouping of elements (free group base) is supported.
- Units can be chosen by the means of a new dialog with preview functionality.



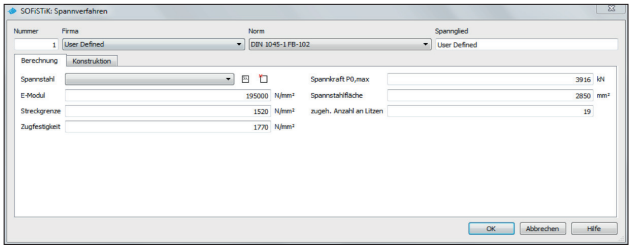
Task Cross Sections

Using the ‚Edit' option of the task's context menu the amount of cross-section output can be controlled.



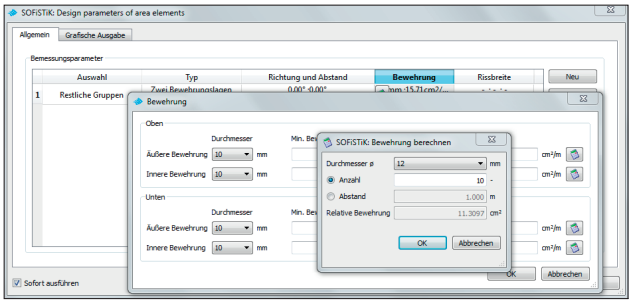
Task Prestressing System

The selection of ‚User Defined' in the Company edit box allows to specify the basic parameters for a prestressing system manually. The prestressing system is then available in the specific project.



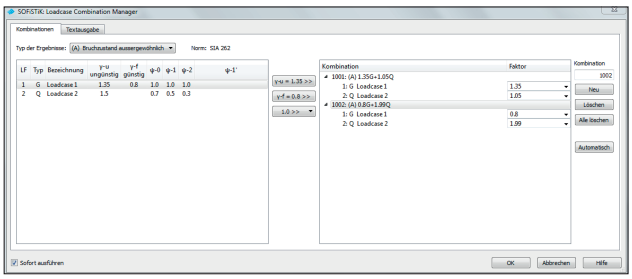
Task Design Parameters

This task has been completely redesigned. The basic input element is a table containing all relevant input data. Detailed input of the specific parameters can be done using additional dialogs and controls.



Task Loadcase Combination Manager

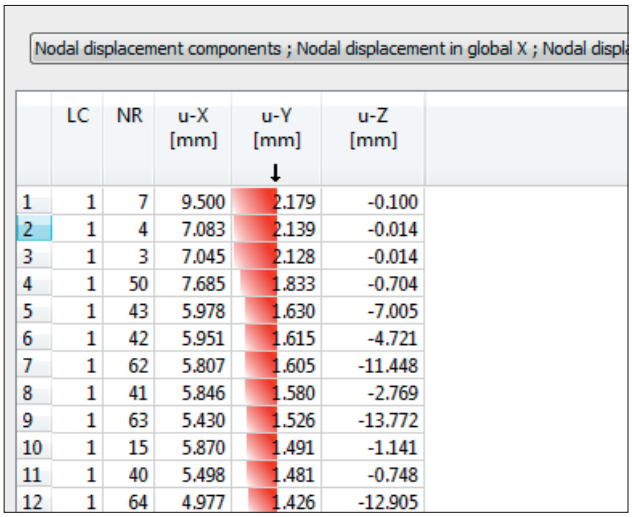
All result loadcase types are now available depending on the selected design code. Improved display of combination factors and coefficients.



GRAFiX for List/Spreadsheet Output and Cross-Section Results

The program GRAFiX allows interactive post-processing of data in a spreadsheet environment. Data can be filtered and sorted individually.

Additionally GRAFiX can be used for plotting of cross-sections and results on cross-sections.

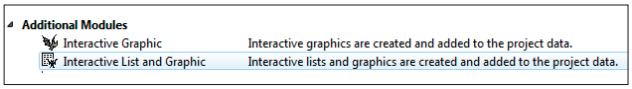


Nodal displacement components ; Nodal displacement in global X ; Nodal displ

	LC	NR	u-X [mm]	u-Y [mm]	u-Z [mm]
1	1	7	9.500	2.179	-0.100
2	1	4	7.083	2.139	-0.014
3	1	3	7.045	2.128	-0.014
4	1	50	7.685	1.833	-0.704
5	1	43	5.978	1.630	-7.005
6	1	42	5.951	1.615	-4.721
7	1	62	5.807	1.605	-11.448
8	1	41	5.846	1.580	-2.769
9	1	63	5.430	1.526	-13.772
10	1	15	5.870	1.491	-1.141
11	1	40	5.498	1.481	-0.748
12	1	64	4.977	1.426	-12.905

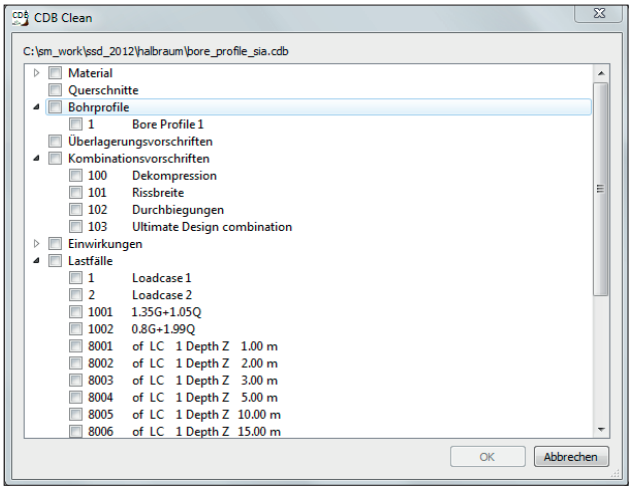
Task Interactive List und Graphic (GRAFiX)

This task can be used to insert a GRAFiX document into an SSD project.



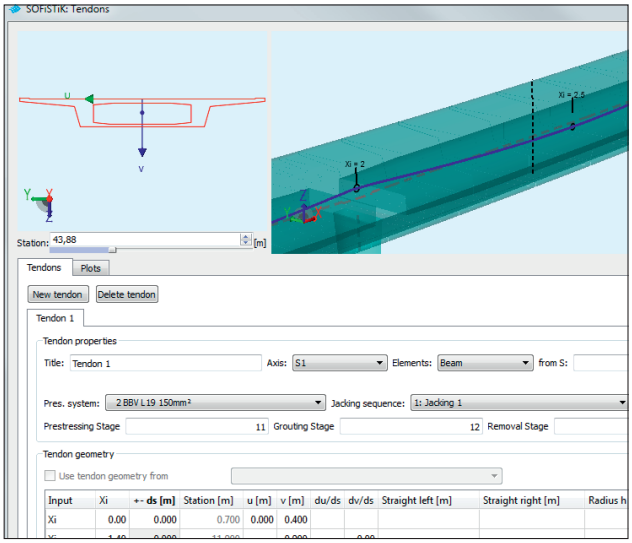
CDB Clean

A new dialog ‘CDB Clean’ for the selective removal of system data and loadcases from the database file is available in SSD.



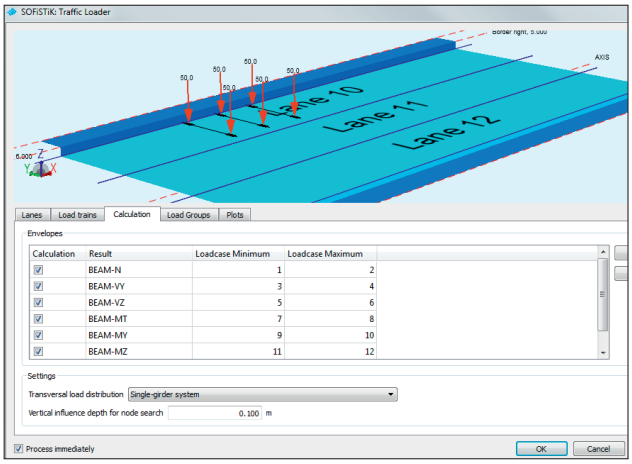
CABD Task Tendons

The CABD task for the input of tendons provides a 3D preview for the definition of beam tendons.



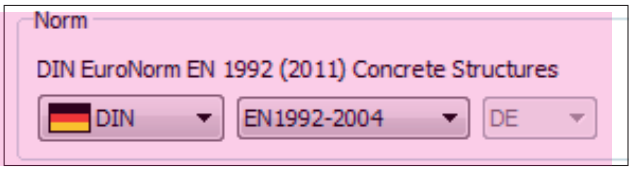
CABD Task Traffic Loader

The influence line evaluation task for traffic loading allows the calculation of QUAD systems with transverse influence lines. Further new options are the input of combinations of loading cases and the selection of result loadcases.



COLUMN

Das Programm COLUMN zur Stützenbemessung unterstützt mit der Version 2012 den aktuellen Eurocode 2 für Deutschland.

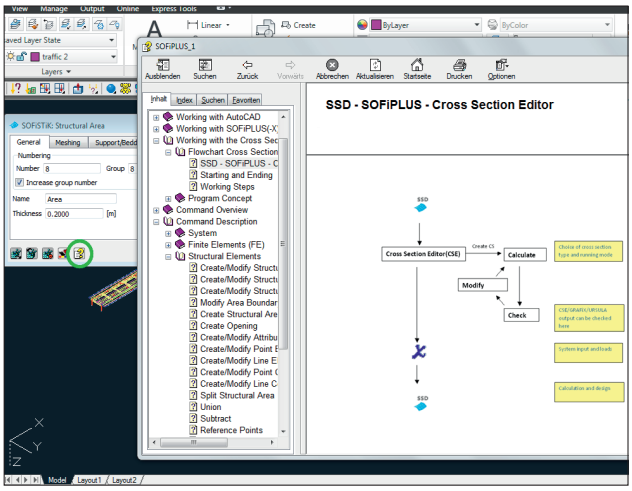


4 SOFiPLUS(-X)

The following versions are available:

1. SOFiPLUS 2012 18.2 for AutoCAD 2010-2012.
2. SOFiPLUS 2012 18.2 x64 for AutoCAD 2010-2012 64-bit.
3. SOFiPLUS-X 2012 deutsch
   w/ German OEM Kernel for 32-bit Operating Systems.
4. SOFiPLUS-X 2012 englisch
   w/ English OEM Kernel for 32-bit Operating Systems.
5. SOFiPLUS-X 2012 x64 deutsch
   w/ German OEM Kernel for 64-bit Operating Systems.
6. SOFiPLUS-X 2012 x64 englisch
   w/ English OEM Kernel for 64-bit Operating Systems.

All OEM versions contain the latest AutoCAD 2012 kernel.



General

The SOFiPLUS help and manuals have been extensively revised. The documents are available via F1 or the help button (.chm), a complete .pdf manual is available via the AutoCAD quick-access toolbar.

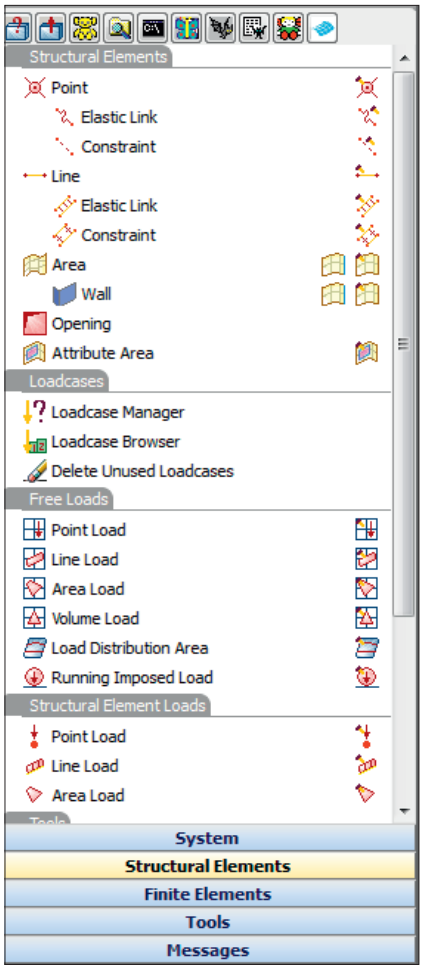
Geometry data is managed and saved with double precision to the .cdb, this provides improved accuracy for complex operations like intersections and route mapping.

User Interface

The complete SOFiPLUS(-X) user interface is now based on Auto-CAD CUI technology. User settings can be made using the CUI-Editor and stored in a partial CUI-File.

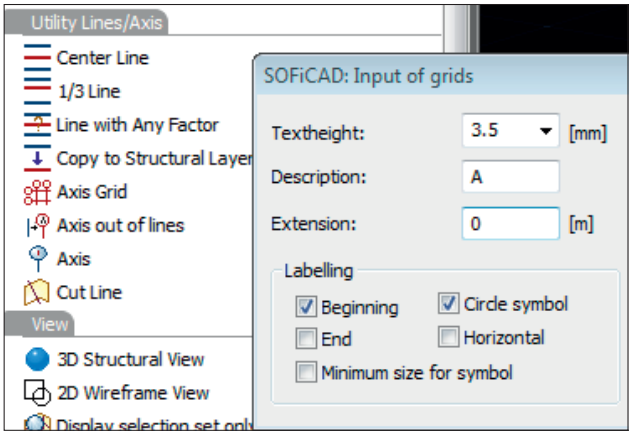
The sidebar layout has been redesigned, the sidebar also provides quick-access to commands like ‘Export‘ and ‘Open Explorer’.

The ‚Finite Elements Tab‘ can be hidden via ‘User Options’.



Axis / Axis Grid

Axes for post-processing (WinGRAF) can be generated more efficiently by converting AutoCAD lines.



Import of Structural Elements

The performance of the import has been increased, especially for structures being generated by the Revit Structure Interface.

Automatic Grouping / Free Group Base

Automatic grouping of elements has been enabled. The setting cannot be changed for the 'Finite Element' workflow..

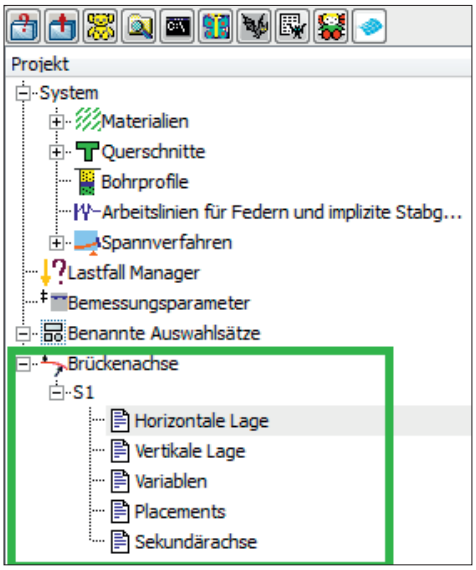
4.1 Parametric Bridge Design (CABD Technology)

Full Integration of Bridge Axes and Geometry Parameters in SOFiPLUS(-X)

Bridge Axes
- New Axis:
- Create Axis from 3D AutoCAD curves
- Import Axis from SOFiSTiK Database (.cdb)

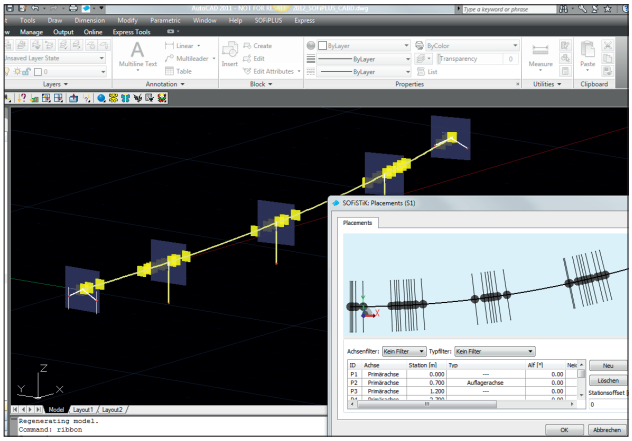
The following geometric parameters are available
- Horizontal Alignment
- Vertical Alignment
- Variables
- Secondary Axis
- Placements

Of course, the structure can be defined independent of an axis as before. In such a case the axes could be used as geometric reference for the definition of prestressing tendons and the arrangement of traffic loads.

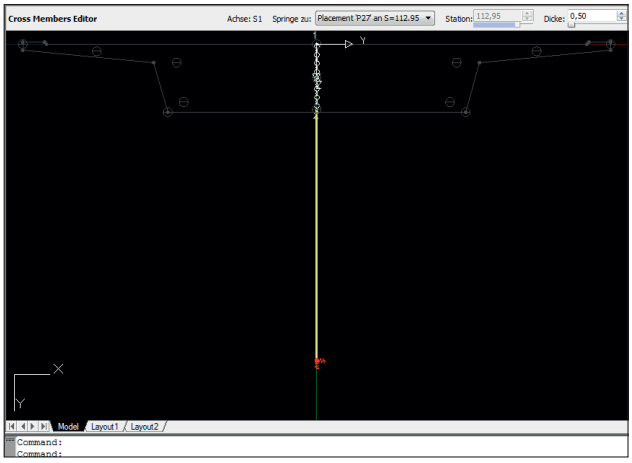


Structural Elements along an Axis

Along the defined axes, structural elements can be created. In the right click menu of the structural elements (context menu while the command is in use), the option "Segment on Bridge Axis" will be available if an axis has been created. Using this option, segments between placements can be selected and used for creation of structural elements.
If the geometrical parameters are modified, the structural elements referring to an axis will be automatically updated.



Cross Members Editor / Placement Editor



For each entered placement, a local working plane (placement plane) will be automatically created. Double clicking on a placement plane leads directly to the “Cross Members Editor”.

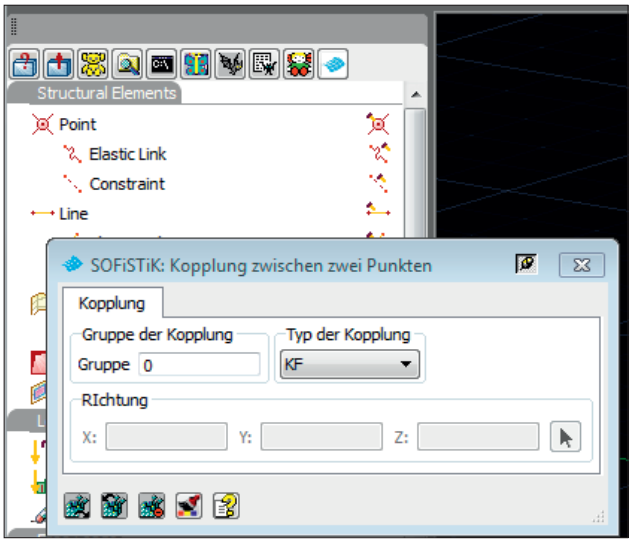
The “Cross Members Editor” allows the input and modification of structural elements in each placement plane, both in 2D or 3D.
The most important operation instructions are noted below:

- In “Cross Members Editor” the corresponding (interpolated) beam cross section at each station will be displayed (if a structural line with a cross section is defined along the line).
- Structural elements that are input in the “Cross Members Editor” will inherit a reference to the axis and each placement. For example, when changing the placement of a station, the structural elements are automatically adjusted to the new arrangement.
- The geometry of the structural elements can also be linked to cross section reference points. Changes to the height of the cross section, for example, will also modify/scale the linked structural line.
- Copying structural elements between placements can be done using the AutoCAD “Copy with Base Point” command.
- Using the Placement dropdown box in the editor toolbar, the user can navigate through the working planes.

The visualised depth in a 3D view can be controlled using the “Thickness” parameter.

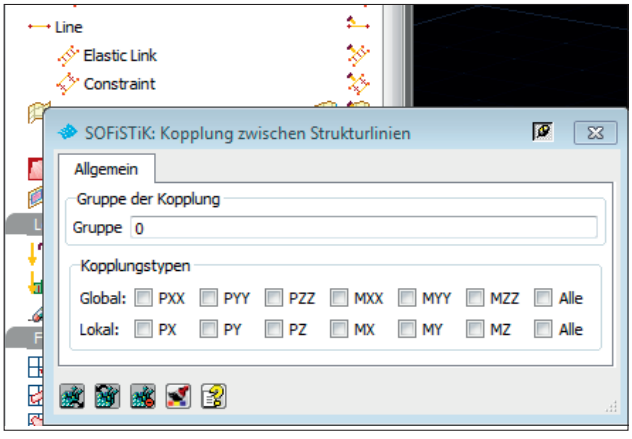
4.2 Structural Elements

Element: Constraint Between Two Points



This new command replaces the ‘Constraint’ tab in the ‘Structural Point’ dialog.

Element: Constraint Between Two Lines



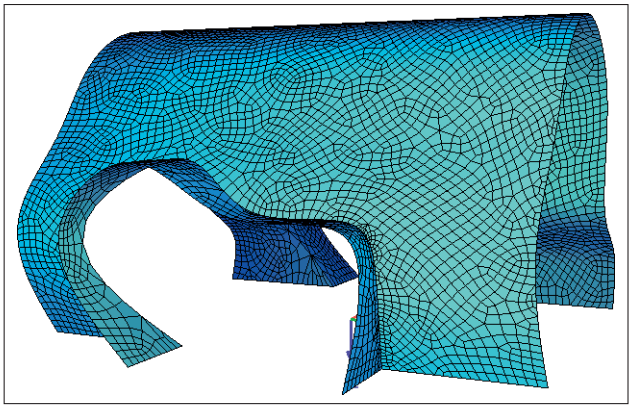
This new command replaces the ‘Constraint’ tab in the ‘Structural Line’ dialog.

Start and end of a structural line are marked by filled circles..

Structural Areas by Conversion of AutoCAD Elements

The conversion of AutoCAD Elements into structural areas is now possible with the 'Elements' option. The dedicated command for the generation of a curved surface is therefore not necessary anymore.

This option supports the generation of structural areas out of Regions, Solids, Nurb Surfaces, other 3D surfaces, Polymeshes (Legacy) and closed 2D and 3D Polylines. The conversion of Subdivison Surfaces is not supported directly. 2D and 3D Polylines which consist out of line segment with constant length and are the approximation of a circle will be converted into circles. .



4.3 Cross Section Editor

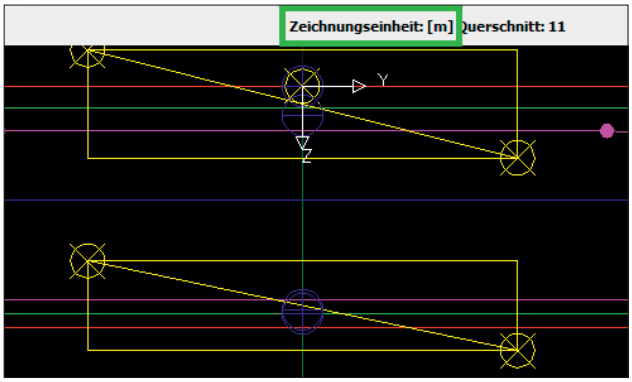
General / Units / Cross Section Types

The Cross Section Editor supports the drawing units m and mm now. The default depends on the selected design code.

Multiedit of cross section elements is now possible.

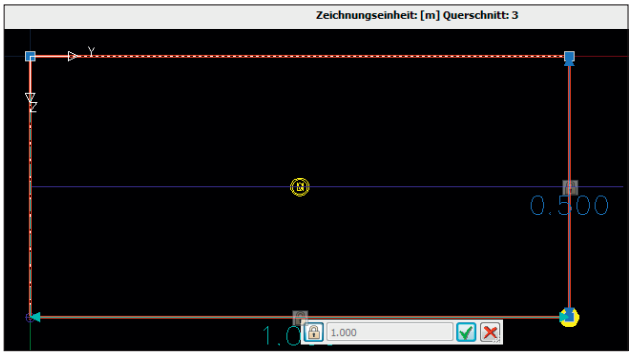
Materials and cross section can be copied using a new context menu option.

The input of new cross sections is supported by specific dialogs depending on the cross section type.



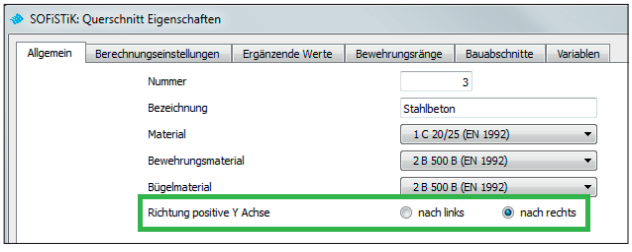
Input of References / Parametric Cross Sections (CABD)

The input of references and parametric cross sections has been redesigned. The improved visualization shows dependencies and type (constant / variable / function).



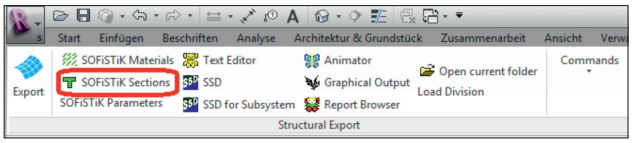
Orientation of Cross Sections

The direction of the local y-axis can be chosen for the cross section input (AQUA: CTRL FACE).

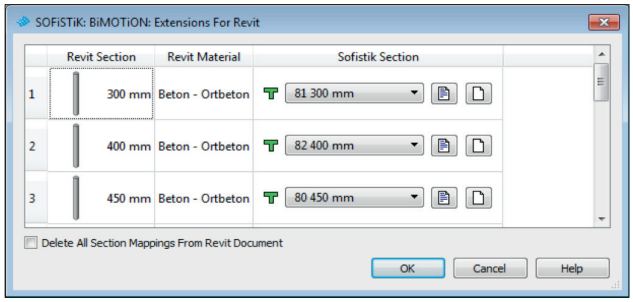


5 SOFiSTiK Extensions for Revit Structure 2012

Mapping Tables for Materials and Cross Sections



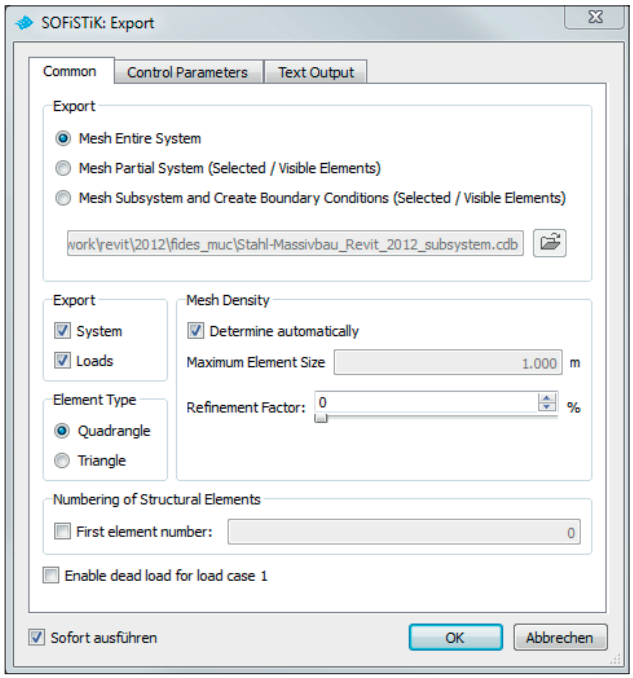
The cross sections and materials used in a Revit model will be automatically mapped to SOFiSTiK materials and cross sections. Standard sections which are available in the SOFiSTiK database will be automatically recognized (and not exported as polygonal sections). Individual changes and modifications are possible using the mapping table dialogs.



Enhanced Export Options

The export and meshing of the structural model provides three different options:

1. **Mesh Entire System**
   All elements with suitable structural properties will be exported and meshed.
2. **Mesh Partial System (Selected / Visible Elements)**
   Only selected / visible elements will be exported.
3. **Mesh Subsystem and Create Boundary Conditions (Selected / Visible Elements)**
   Only selected / visible elements will be exported to a separate database, boundary conditions will be generated using stiffness information of connecting elements (e.g. columns and walls). The Subsystem can be opened as a separate SSD project.



Load Division

For the generation of live load patterns, free or hosted element loads in Revit can be subdivided by model lines.

SOFiSTiK Parameters

Different SOFiSTiK parameters can be assigned to Revit elements

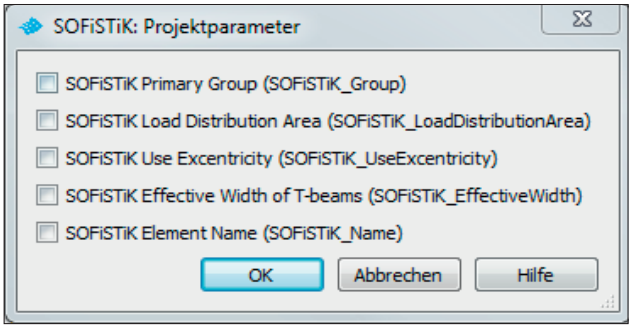
SOFiSTiK_Group
Primary group number

SOFiSTiK_LoadDistributionArea
Parameter to generate a load distribution area which fits to the respective area load.

SOFiSTiK_UseExcentricity
Parameter to consider element eccentricities.

SOFiSTiK_EffectiveWidth
Effective width for downstand beams. Rectangular Revit sections will be exported as suitable T-beam with effective slab width.

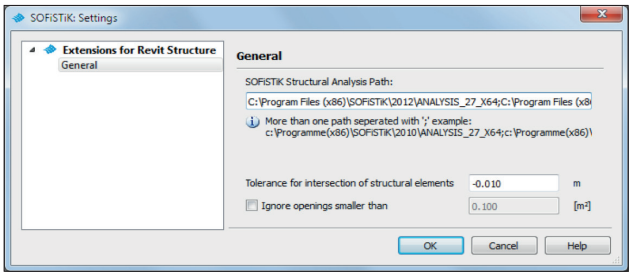
SOFiSTiK_Name
Descriptive name of an element.



User Settings

The tolerance for the intersection of structural elements can be set here.

Openings below a certain area can be ignored.



6 Rhinoceros Interface

New Produkt: Rhinoceros Interface

With version 2012 a SOFiSTiK interface for McNeal Rhinoceros 4.0 is available. The Rhinoceros Interface can be used to define and export structural elements (Points / Lines / Areas) from a Rhino model. The elements can be meshed and modified using SOFiMSHC.

A dedicated SOFiSTiK toolbox provides the necessary commands and some helpful tools (e.g. cross section visualization).

Structural element properties can be assigned and modified directly in the properties dialog in Rhino.