

New Features Release Version 23

While the change from release versions 99 to 21 was a large step due to the change of the design code selection based on [INI-Files](#), we have now a smaller step from version 21 to 23 as many parts have been developed in parallel and most parts of the software are still identical. If you have used SONAR for updates on a regular base, there are little changes in the CADiNP input files. However some old data formats for the 99 version have been withdrawn (e.g. MAXIMA) and the specification of the desired design code is now mandatory.

The most visible change is the new interactive user interface [SOFiSTiK Structural Desktop \(SSD\)](#). The SSD is the central control interface for all SOFiSTiK programs and controls Pre-Processing (system geometry), Processing (analysis) und Post-Processing (design/result evaluation) offering dialog-boxes and interactive graphical input. Of course creating and working with normal DAT-files is still possible. The SSD will not need an extra license, but can be used by anyone eligible to use the current 23-versions.

The SOFiSTiK- Editor [TEDDY](#) has been ported from [MFC](#) to [QT](#) (C++ libray classes). This will allow a generic [LINUX](#) version for TEDY in the near future. TEDDY will be an integrated part of the SSD and be available as a stand-alone version as well..With "SOFiSTiK → Options" all SOFiSTiK-settings are now to be controlled. This menu is available in most SOFiSTiK modules. The old menus of TEDDY-21 "SOFiSTiK → Settings" and "Extras→ Options" have been removed. Some of the SSD-tasks are also accessible form the native TEDDY-Version and allow direct insertion of text in the dat file (Menue: "SOFiSTiK" > "Tasks")

For [WinG](#) and [WinGRAF](#) there are additional system values available like beam sections and prescribed reinforcements in planar elements. New result types for nonlinear analysis from [TALPA](#), [ASE](#) and [HYDRA](#) are available. Stresses in beams may be visualized with the maximum from all materials. A new selection allows to select elements based on their properties like section or material number and local coordinate systems. Load cases may be controlled not only for the current layer but also for the current picture or the whole page. Further it is possible to define loops on the load cases within the trees. The default settings within WINGRAF.DEF are no longer overwritten automatically.

For the analysis programs it should be noted, that the [computing speed](#) of the MS-Windows versions has been increased significantly as we may now use vectorisation of modern processors. [DYNA](#) has been chosen for the implementation of new variants of [solvers](#). This is a new direct sparse-Solver (CTRL SOLV 3), which is considerably faster especially for spatial structures, if the internal renumbering has been selected for minimum fill in (CTRL OPTI 17 or similar). But also the classical skyline solver has been parallelized successfully and with a ISOL (iterative solver) licence will use now multiple processors.

All analysis program of the 23 release support [variable group sizes from SOFIMSHA/SOFIMSHB](#). This feature has not yet been implemented in SOFiPLUS/SSD completed however.

AQB has not only [additional design codes \(Russian, Swedish; current codes can always be checked in the log-files !\)](#) but also a new feature for the shear design of reinforced concrete where the longitudinal reinforcement has an effect. A special superposition is performed between bending and shear design allowing to transfer the maximum reinforcement from the mid of the span to the supports. A significant enhancement is also available for timber design where we have admissible stresses depending on the duration of the loading.

BEMESS allows now the definition of the reinforcement directions within a separate preprocessing run. Some additional design tasks for minimum reinforcement and fatigue complete now the design according the new DIN 1045-1. [The Swedish concrete code has been implemented in BEMESS as well \(also check the log-file for the latest codes !\)](#)

STAR2 has been renovated to account for new group definitions, excentric beams with a reference axis are now also supported if they do not have haunches.

The **Construction Stage Manager CSM** has been considerably enhanced for bridge design and allows now to iterate for forces also in earlier construction stages, e.g. cable forces in cable stayed bridges. A [precamber analysis](#) allows to define the requested camber to achieve a prescribed geometry at the time of inauguration. With a [design module within CSM](#) a default design for prestressed bridges may be created now, especially usable for the [SSD Bridge-Wizard](#).

ASE has been extended by a SLIP cable – this is a series of cable elements allowed to slip at their nodes to achieve a common cable force in all elements. Construction stages and time dependant behaviour is now also possible with pile elements

Finite Element analysis for [geotechnics](#) have been considerably enhanced with respect to speed and stability. Time dependant effects of creep, swelling and strength development are now available via viscous material models. The extended GRAN (Granular Hardening) material model has become more usable based on a complex automatic calibration with additional material parameters (Stiffness; Pressure coefficients). Now compressive loading (Oedometer test) and shear loading (Triaxial test) are treated properly. This extends the applicability significantly while reducing the calibration effort to a minimum.

Spring, cable and truss elements within **TALPA** allow now for the complete functionality including the use of prescribed work laws. The features of the [Construction Stage Manager Geotechnic \(CSG\)](#) have been enhanced with special consideration of load case sequences for time dependant analysis. The option to generate nodal loads from stresses allows the usage of dynamical results within TALPA in a consistent way. This feature also allows to transfer earth pressures from a non linear analysis to a standard design with partial safety factors.

DYNA has been reprogrammed in parts completely. Besides the already mentioned new solvers, there is now also available the Hughes-alfa-Method (changed item names in STEP!) and non linear spring elements. For the superposition of stationary loadings a new method to account for the exact inclusion of the phases has been implemented and the preservation of signs within the SRSS method has been extended for the superposition of different loading directions. The definition of an earthquake response spectra has been considerably enhanced by the seismic zone.

SOFIMSHB has now a new unstructured **tetrahedric mesh generator NETGEN** from the university of Linz. **SOFIMSHA** has been enhanced continuously and has achieved a maturity to let GENF become obsolete. Please, note that new features like the continuous beam will not be implemented in GENF.

HYDRA has received an implementation of the **material laws for hydration of concretes (Saul, Freiesleben, Wesche, Jonasson etc.)**. The calculated maturities and effective ages are use be TALPA and ASE for a reduction of the elasticity and strength accordingly.

Much more minor and major changes are continuously documented in the program's log-files (TEDDY-menue: "Help" > "SOFiSTiK Log Files"). The above listing is only a selection of those which we consider as the more important ones. If you have any questions, do not hesitate to contact us at info@sofistik.com !

Innovations in SOFiPLUS(-X) Version 16.4

System requirements:

SOFiSTiK FEA programs version 23

SOFiPLUS 16.4: AutoCAD/ADT 2004, 2005,2006

SOFiPLUS-X 16.4: will be shipped with an integrated AutoCAD 2006 OEM engine



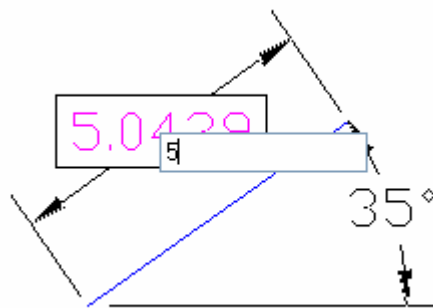
Database, Data Management

When using the same name for both the database and the drawing, the layer names all have the same ending “ASDWG”. The name of the database will now be changed if saving the drawing under a new name using the ‘Save As’ command.

When closing tabbed dialogs, the current tab and other settings will be automatically saved.

Dynamic Dimensioning

The “dynamic dimensioning” feature within AutoCAD can be activated when creating beams, trusses, cables or structure lines. The length and angle to the global X-axis of these objects can be defined.



Pressing the TAB key changes between these options.

The dynamic dimensioning can be switched off in the SOFiPLUS options dialog.

Beam Sections

The number of the beam sections along the length of every beam will be automatically determined. The number of cross sections created depends on the ratio of the beam length to the smaller cross sectional dimension (width or height). The number of automatically generated beam sections can be controlled via the option dialog (system options).

Export

The commands “automatic mesh generation” and “export” have been combined to a global “export” command. In drawings containing only structural elements, the new command “export” automatically exports these objects and all loads into the database and executes a mesh generation and a load distribution. In drawings containing only finite elements, the same command exports the finite elements directly into the database and also executes a load distribution. In drawings containing both structural elements and finite elements, the user is given the option to select which process is used or the ability to mix the two together.

System Data

The well known commands ‘Materials’, ‘Cross-sections’ and ‘Design parameters’ can be accessed within the section ‘System data’ (new toolbar).

Additionally the commands “Bore Profiles” (until now accessible via the dialog for structure lines) and “Working laws for springs” are available. Bore profiles can either be used for creating 3D pile elements (structural lines) or for constrained soil modulus calculations (structural points).

Displacement [mm]	Normal Force	
2.0000	1.0000	x
1.0000	1.0000	x
0.0000	0.0000	x
-1.0000	-1.0000	x
-2.0000	-1.0000	x

Structural Elements

Element numbers can be defined in advance when creating a model. In the same way the group number for structure lines and structure areas can also be specified.

The dialog boxes for structural lines and structural points have been standardised.

The same tabs 'General', 'Support conditions' and 'Constraints' have been implemented for both these dialogs.

Structural Points

Tab General:

Structural points can be attached to a structural area via the number of the structural area.

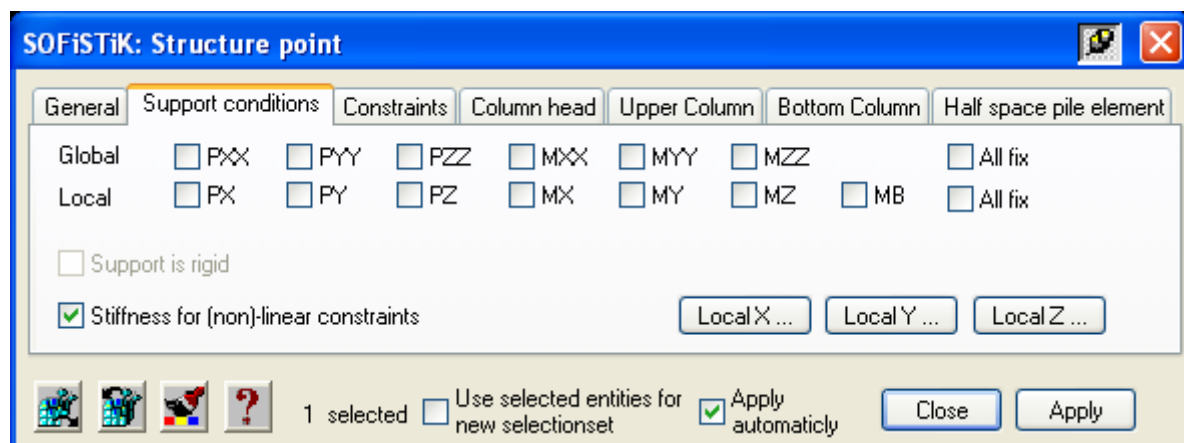
Tab Spring x to z:

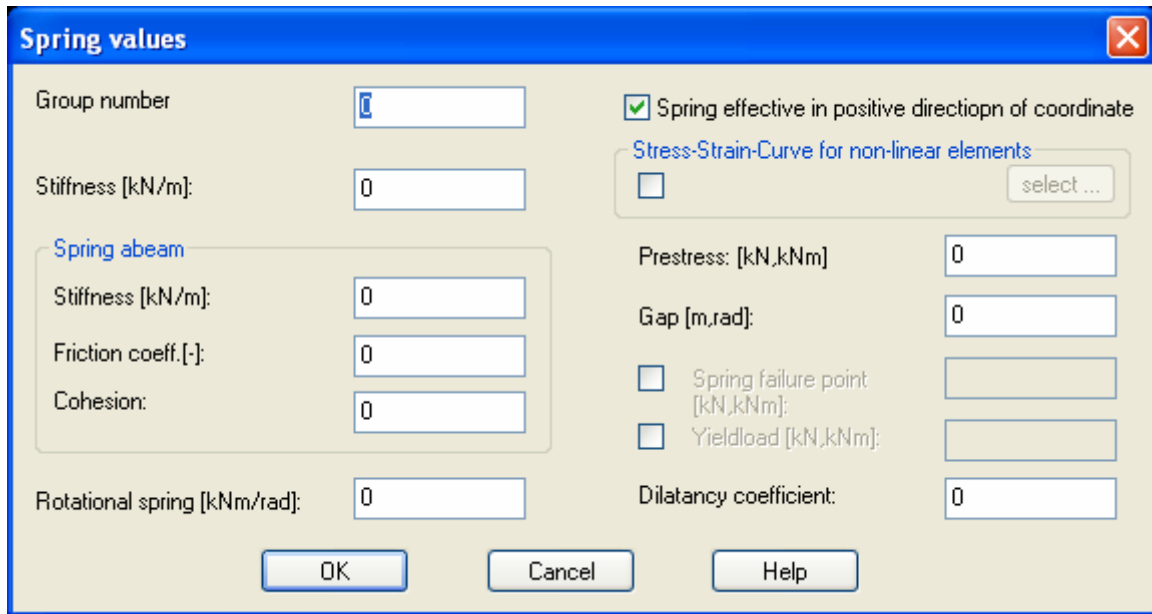
These tabs have become obsolete. The attributes from these tabs can be set in the tab 'Support conditions'. There are up to three buttons for springs depending on the current type of system.

Tab Support Conditions:

Support conditions, rigid or elastic, are defined similar to structure edges in this tab.

When using elastic support conditions a new dialog box appears for every direction is available.





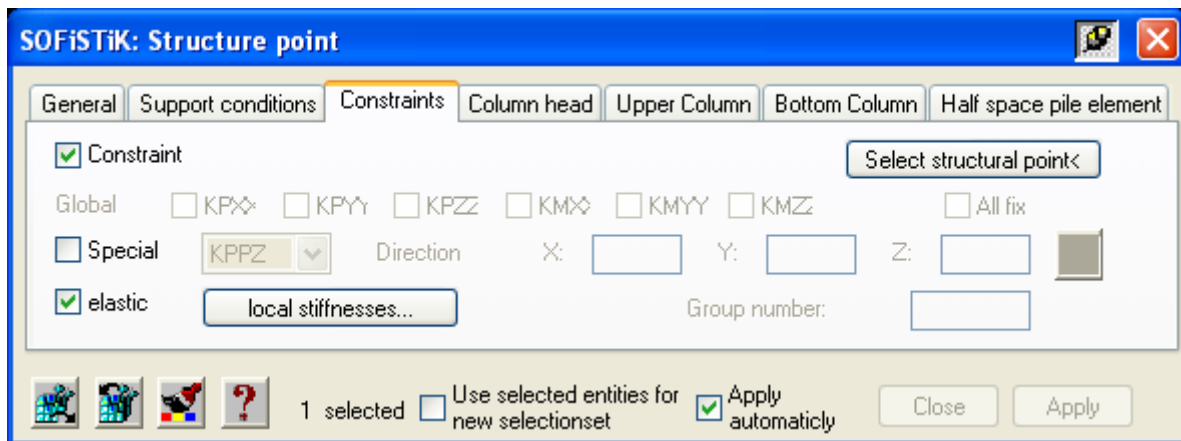
Additionally the direction of the spring can be inverted and a previously defined working law for springs can be selected.

Tab Constraints:

Rigid and elastic constraints will be defined in this new tab.

Unlike support conditions, constraints are defined between two elements.

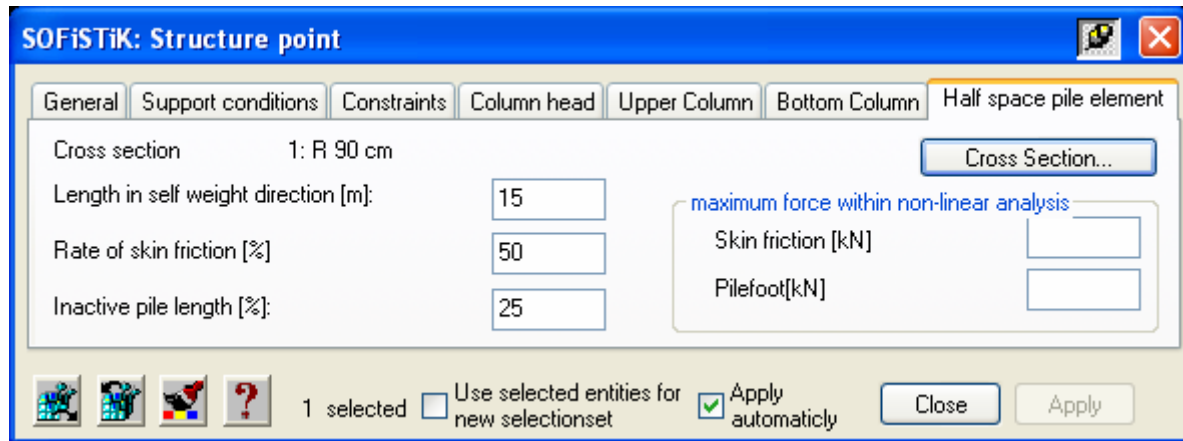
New special constraints (e.g. KP – hinged connection to rigid body at the second node) are now available. Additionally the vector of the constraint can have another direction other than the constraint itself.



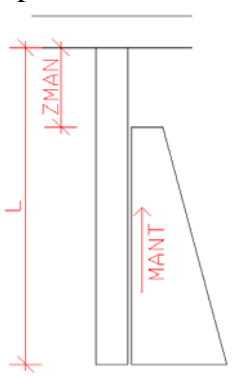
The button ‘local stiffnesses’ will open the dialog box ‘Spring Properties’.

Tab Half Space Pile:

For further calculations with the module “HASE”, the necessary properties are defined as follows:



Explanation of the attributes:



Vertical length (L)

Percentage of skin friction (MANT)

Percentage of inactive pile length (ZMAN)

Maximum force for skin friction / pilefoot

Total pile length

Defines how the load is distributed. The remaining load will be distributed via the pile foot.

Length from the top of the pile within which no skin friction will be activated.

With this value the force distributed via skin friction or the pilefoot can be limited for non-linear analysis.

Aligning of Structural Points

The alignment of structural points has been improved. A lot of new possibilities for aligning structural points have been implemented.

Structural Areas

When creating structural areas, two new options ‘Close’ and ‘Back’ are available.

Opening

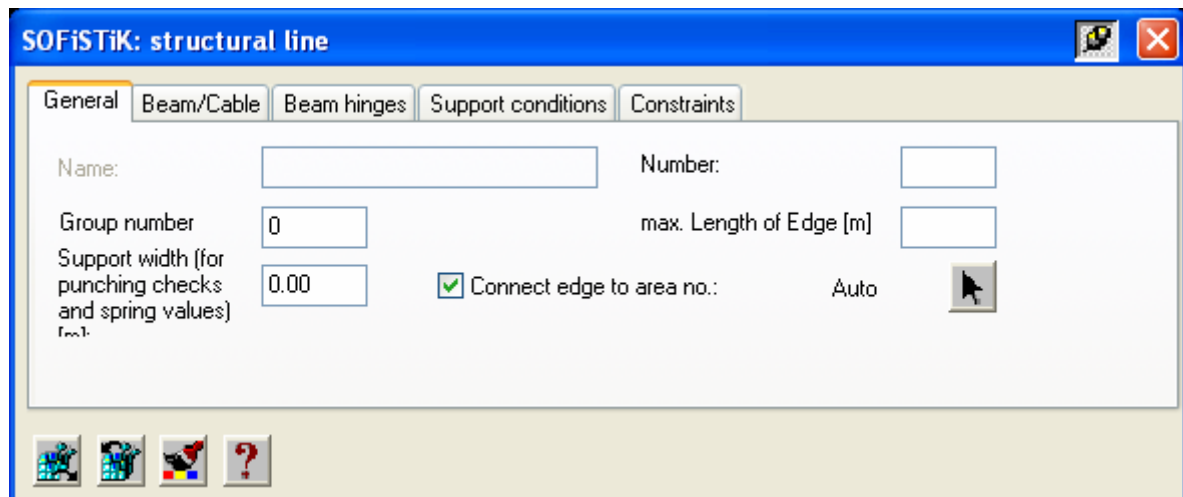
Additional option ‘Pick Corner points ‘ implemented.

Structural Lines

The new option ‘Single element definition’ (similar when creating beams) is available.

Tab General:

Structural lines can be attached to a structural area via the number of the structural area.

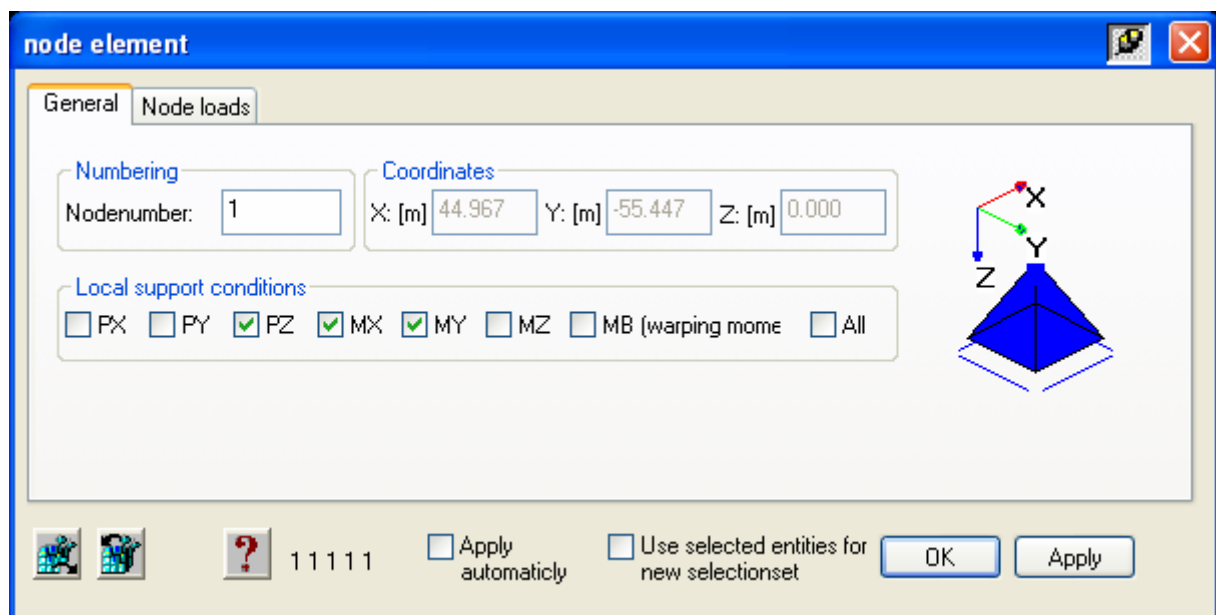


Finite Elements

The finite element dialog box has been completely revised and should now work in a similar manner to the beam element dialog box. The finite elements are objects with associated loads. Also the COM interface (characteristic dialog) is now available for all finite elements.

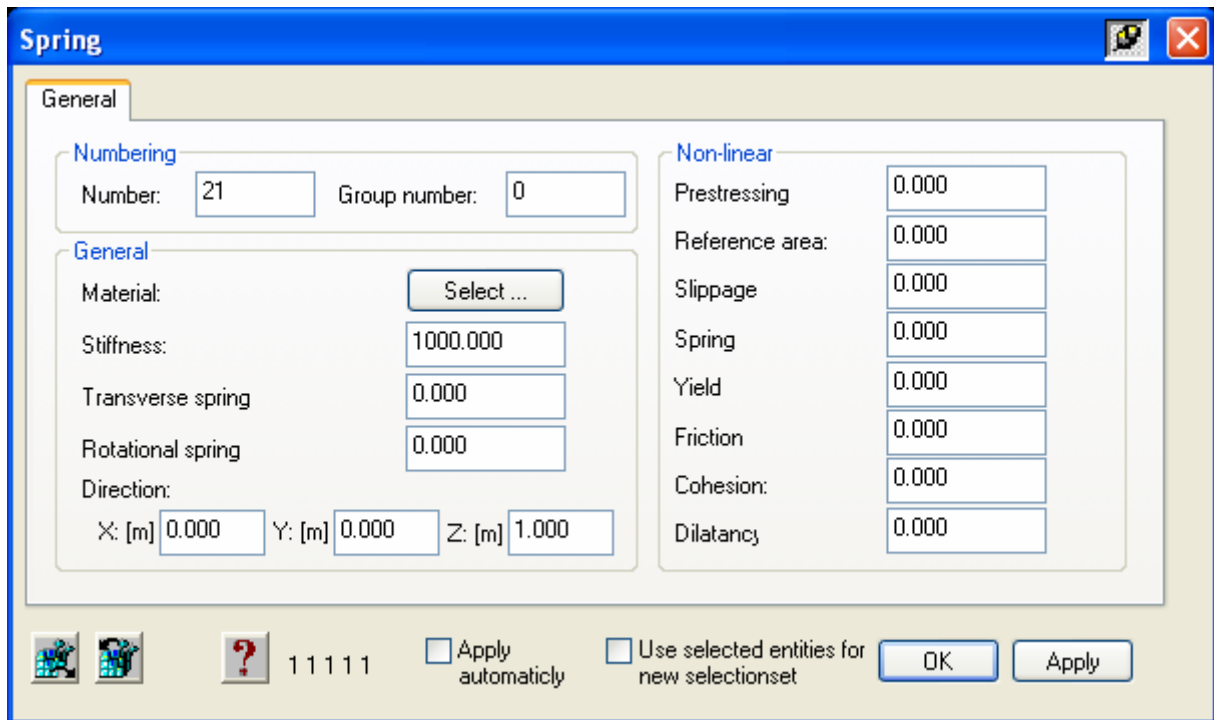
Node

The dialog box for nodes and nodal forces has been completely revised. For a better understanding, the support conditions are drawn directly in the dialog box as the selection is made.



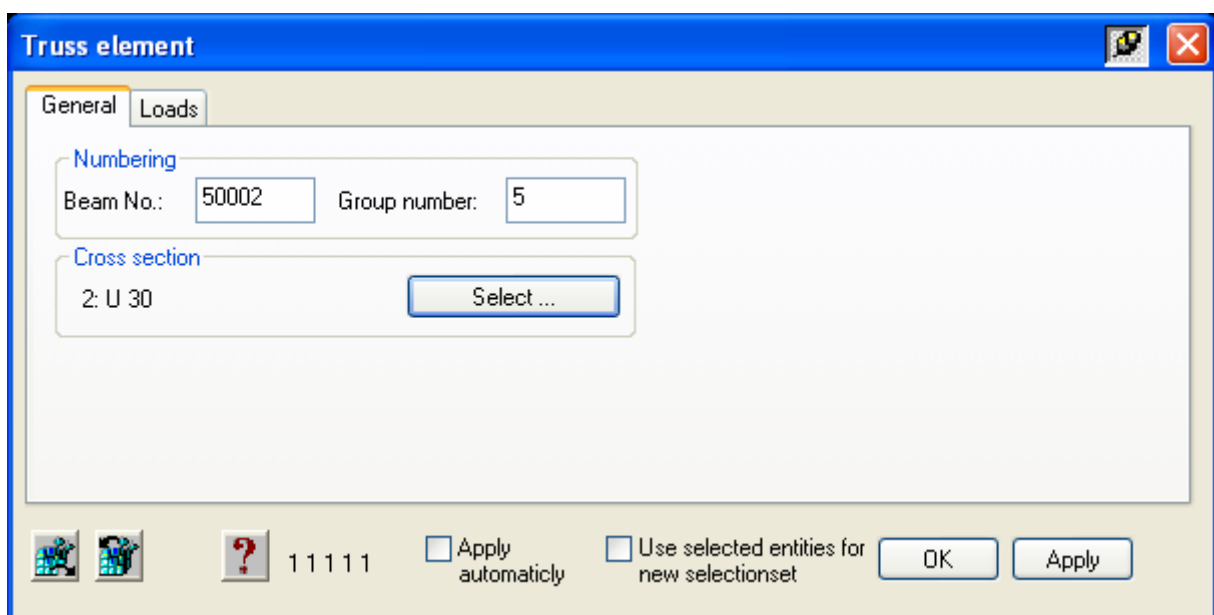
Spring/Constraint

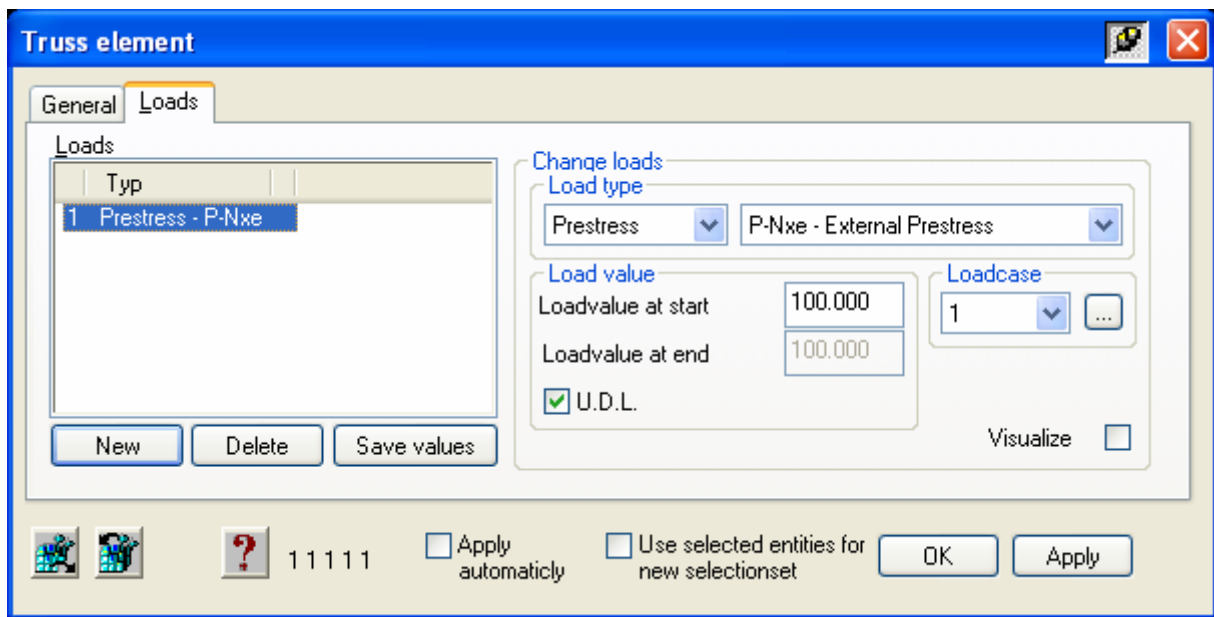
A differentiation between springs/constraints with one node (normal support spring) and with two nodes (spring between two nodes) is now possible. However, this differentiation, once made, cannot be reversed.



Truss Element/Cable

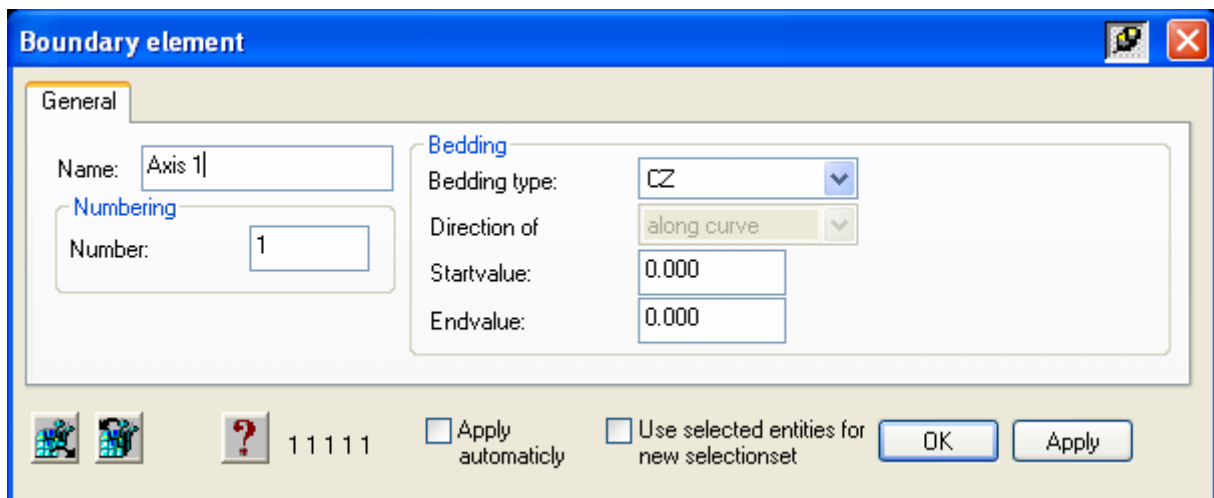
The element-bound pre-stress (without load case) is no longer supported.





Boundary Element

New dialog box for boundary elements:



Loading

Structural element associated loads

Structural element associated loads are now available for structural points, lines and areas. These loads are created using the same commands for free loads. Whilst creating the free load, the structural element to which it is associated can also be selected. The load gets its coordinates directly from the associated structural element. Therefore, when moving or stretching this structural element, the load follows its “master object”. When deleting the structural element, the load will also be deleted.

Loads on finite elements

Loads on finite elements are now applied using completely new dialog boxes. The important attributes of the loads are available in the properties dialog. Excentric nodal loads are now available.

Load distribution area

All attributes of the load distribution areas are available either via the AutoCAD properties dialog or a dialog box for these objects.

Running imposed Load (Moving Load)

The new load cases are copies from the original ones with load factors and type of action copied as well.

Output of Results

This command is available in three options:

- Total result
- Result mesh generation
- Result of loading

New Commands

BFE-Show

The command ' BFE Show ' displays the required reinforcement for detailing. This aids reinforcement detailing as missing reinforcement can easily be identified.

Cross Section Outlines

This command generates 3D-volume elements out of a defined cross section. These elements will be saved on layer X__SOLIDS. These solids can be exploded for further manipulation within AutoCAD.

Cutting lines

Cutting lines that have been defined within SOFiPLUS are available within WinGRAF after exporting or meshing the model.

Load export / mesh generation

Commands for mesh generation and load distribution are available separately. In case only the loads have been changed in a model there is no need to mesh up the system again. Just a load distribution is enough for the loads to become operative.

Align elements

New command for aligning beams, trusses and cables. Several possibilities for alignment are available.

Re-number elements

A new command allowing easy re-numbering of beams, trusses and cables is available. The numbering will increase from a given start number..

Visualisation

The visualisation of all elements is controlled using a new dialog box. Color, size and design for all elements can be changed here.

Ceased Commands

The following commands have become obsolete:

WINGRAF is called always by the command “graphical output”

The command “MACRO generation” can be used directly when creating area elements.

The command “input single elements” is available now for beams, truss elements, cables and structure areas.

Visualizing commands for finite elements are omitted. A new dialog box to cater for this is under construction

The command Info/Edit is no longer available.