

STRAP 2009

List of Enhancements

Highlights

- New** **Linear rigid links:**
A node can now be linearly linked to two different nodes; its displacement will be proportional to its relative distance from the two nodes.
 - refer to "Detailed list of Enhancements" for more information.
- New** **Castellated and cellular beams:**
Castellated and cellular steel beams may now be defined in geometry and designed in the steel design module.
- New** **Edit concrete slab detailing:**
The automatic slab detailing may now be edited and modified by the user. The program can check whether the revised detailing complies with the Code requirements.
 - refer to "*Detailed list of Enhancements - Concrete*" for more information.
- New** **Identical concrete beams:**
Concrete beams with the same spans and sections can now be designated as "identical" and are designed with the same moment and shear reinforcement.
 - refer to "*Detailed list of Enhancements - Concrete*" for more information.
- New** **Concrete slab punching results display:**
Slab punching results may now be displayed graphically.
 - refer to "*Detailed list of Enhancements - Results*" for more information.

Detailed List of Enhancements:

General:

- Support for 64-bit Windows has been added.
- If the "results" pressed after revisions to geometry or loads the program asks the user whether to solve the model.
- Select nodes, beams, etc.: The user may now request a previous selection.
- Select nodes: a new option to select supports nodes only.
- Display:
 - local axes: the axes for either beams or elements may now be displayed.
 - Springs: may now be displayed with the symbol only and without the value may now be displayed in all *STRAP* modules (loads, results, etc).
- Remove:
 - display a plane: the display is now rotated to the selected plane.
 - Limit by coordinates: the coordinates can now be selected by moving the cursor.
- Submodels: 'Display/remove submodels' menu
 - new option that assigns 'Current view' to all submodel instances.
 - new option to display all submodels with the Main model 'Display' options
- Print: long printer names are now displayed in the dialog box
- Magnifying glass: the window center may now be moved with the arrow keys.
- Icon bar: a "Draw wall" icon has been added in all modules

- Setup:
 - Dimension line options:
 - number of digits after the decimal point for angles
 - round-off parameter for level values.
 - Grid lines: initial values for all grid line parameters.

Geometry:

- Submodels:
 - springs may now be defined.
 - Connection points: new "Automatic" option; the program automatically identifies common nodes in the submodel and the main model.
- Rigid links: new option to define "Linear links":

A node can now be linearly linked to two different nodes; its displacement will be proportional to its relative distance from the two nodes. This option gives more accurate results when attaching a wall element to a floor slab that has multiple nodes along the face of the wall and is the default option when the "Link walls" option is selected.
- Define nodes - "Line center" option: the program now remains in this mode so that multiple nodes may be easily defined.
- Beams - properties - combined sections: the following steel sections may now be defined:
 - castellated beams
 - cellular beams
 - new combined cold-formed sections: back-to-back double section, front-to-front and four-sections.
- Beams - properties - steel sections: an exact sketch of each section type is now drawn for the "Major axis direction" option.
- Display - DXF drawing:
 - the reference point may now be displayed by moving the cursor on the graphic display.
 - the program now remembers which DXF files were previously selected for the model and their parameters.
 - the DXF file can now be rotated to any angle.
- Display - rigid links: the rigid links direction is now displayed in the title bar of the display window
- Display data: new option to display submodel instance connected to the selected node.
- Copy:
 - rigid links may now be copied
 - new option to select beams/elements that are not to be copied even though they have been selected.
- Element mesh: the property number may now be specified when defining the mesh.
- Model type: the program automatically modifies the restraints when a plane frame/grid is changed to a space frame.

Loads:

- Wind loads:
 - lattice loads: if two panels have a common member the program now applies a wind load to it only once.
 - new 'tolerance' option: all members within this distance from the panel will now be loaded.
 - Russian wind code has been added.
- Global loads: line loads may now be defined, either as a load/length or as load/area together with a strip width.
- Beam loads: the load sketch in the dialog box is now updated instantly as the data is entered.
- Display: support displacements may now be displayed
- Display - loads: a minimum arrow height may now be defined for all loads.

Results:

- Tables: Beam stresses - M/S+P/A - for all four section corners may now be displayed.
- Tables: element reinforcement: Code minimum area are now denoted by ** adjacent to the values.
- Graphics - spring stresses: the soil coefficient is now displayed.
- Element reinforcement: the Russian SP52 Code has been added.
- Long combination names are now displayed in the main window.
- Right-click - beams: the program now displays in addition to the results:
 - the combination for the maximum value of each result type
 - the end node numbers
- Punching - graphic display has been added:
 - Nodes are color coded according to status:
 - no reinforcement required
 - reinforcement required
 - stress exceeds maximum Code value
 - The following values may be displayed:
 - Area required
 - critical perimeter
 - punching shear force
 - punching stress
 - column dimensions
 - Column head dimension
- Remove: "Remove nodes without elements" option has been added.

Dynamic:

- seismic design according to ASCE/SEI 7-05 has been added.

Steel:

- Brazil steel code NBr 8800 : program updated for 2008 version.
- the following steel sections may now be designed:
 - castellated beams
 - cellular beams
 - four new combined cold-formed sections.

Concrete:

- Identical beams:
 - Selected beams may now be specified as 'identical'; the program creates combined moment/shear envelopes for all of the beams in an identical list and designs identical reinforcement (main and links) for all of the beams.
- Display wall numbers: 'w' prefix added
- Detailed results - walls: section title added to header
- Display property numbers: wall section no. now displayed
- Reinforcement area: units according to STRAP setup
- General arrangement drawing: new setup option for section units
- Draw: "Draw walls"/"Draw columns" options added
- Remove: "Display selected levels" option added
- Display/revise beam/column menus: now moveable
- Max. no of STRAP members in continuous beam increased from 40 to 80
- Solid sections menu: external X/Y section dimensions now displayed
- Column/wall parameters: maximum bar spacing may now be defined
- Column tables:
 - new option to draw elevations automatically in new tables
 - new option to draw all sections in a column according to the scale of the largest section
 - Column dimension format: new option in setup - 'm / n' (e.g. 200/400) or 'm x n' (e.g. 200x400)
 - Links: new option to draw only first and last links in every group
- Slab detailing:
 - The automatic slab detailing may now be edited. The following options are available:
 - combine reinforcement bars
 - move reinforcement text
 - move all lines describing location and extent of a bar group
 - modify bar lengths
 - modify extent of bar group
 - modify diameter and spacing
 - add/remove bars
 - add/remove hooks and ends of bars
 - the program tries to maintain all modifications even if the the slab detailing is recomputed.
 - the user can check whether the modified reinforcement complies with all Code requirements.
 - elements may be color-coded according to the reinforcement provide/required ratio.