

Version 2015

Enhancements Guide

May 2015

© 2015 ATIR Engineering Software Ltd.

STRAP 2015

Highlights:

New

Increased program capacity:

The program capacity has been enlarged to 1,000,000 nodes and elements (the main model and each submodel) *

New

Walls:

The program now automatically adds nodes to the wall at the location of nodes in attached floor slabs and connects the walls directly to the slab. This eliminates the need to divide long segments, reduces the number of rigid links and improves the accuracy of the solution, particularly in long walls.

New

Wall drawings:

Wall sections with reinforcement may now be drawn.

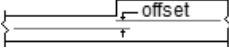
Detailed List of Enhancements:

General:

- The program capacity has been enlarged to 1,000,000 nodes and elements (the main model and each submodel).
- The program can now handle *RESLTnn.DAT* files that are larger than 2 GB.
- The following options have been added to node/beam/element selection:
 - all nodes/beams/elements in a circle
 - all nodes/beams/elements in a ring
 - beams: all beams on the same lines as selected beams
 - nodes: all nodes at one end of selected beams.
- Wall selection: walls are now selected by highlighting a segment instead of the end nodes.
- "Print/edit a saved drawing": the Move option has been improved and is now much faster in large drawings.
- "Display":
 - wall numbering and element numbering are now separate options.
- "Remove":
 - "Limit display by coordinate": nodes not in the current display may now be selected.
- Print geometry: beam "release" circles are now always printed with thin lines.
- "About" (in geometry): now displays plug number, maximum number of nodes, whether finite elements are included

Geometry:

- page 87

 - **General:**
 - "Check element definition": The program now checks whether
 - submodel "connection points" have a corresponding nodes in the Main model and allows the user to create new nodes at these locations in the Main model.
 - main model nodes in the area of a submodel are attached to connection points.
 - Section table: sections not used in the currently displayed submodel are now listed as "-used in a submodel-" (instead of "-not used-")
 - Output - submodel connection data table: The "Go to" option now allows the user to select a specific submodel and a node.
 - **Nodes:**
 - Unify: the program now unifies the nodes in the Main model and all submodels at the same time.
 - **Beams:**
 - x1 local axis: may now be defined in the direction of the closest global axis.
 - **Walls:**
 - The program now automatically add nodes to the wall at the location of nodes in attached floor slabs and connects the walls directly to the slab. This eliminates the need to divide long segments, reduces the number of rigid links and improves the accuracy of the solution, particularly in long walls.
 - A new "offset" option has been added. Connected segments may now be offset so that they are not connected at their center points. For example, align the faces of segments with different widths:
 
 - **Copy:**
 - beams connecting the copies can now be added only at selected nodes

Loads:

- The algorithm for applying Global Loads to nodes has been improved so that the applied loads are accurate for loads defined on large areas.

Results:

- If moment reduction around columns and/or average moments along a strip are requested, the program now remembers that these options were selected.
- If "geometry only" was selected after results were displayed and then display graphic results are selected again, the program now displays results according to the options previously specified.
- Reactions: for submodels the program displays the reactions at the connection points (graphic display & tables).
- page 111

- Foundation design: selected combinations only can now be transferred as loads to the foundation design module (e.g. service load combinations only)

Steel:

- The program does not display results for members that were previously designed but now have "illegal sections".
- Updated codes: Canada cold-formed
- Brazil code: composite section design has been added.

Concrete:

- Walls
 - page 12 ☐ Identical walls can now be specified (similar to identical beams)
 - page 13 ☐ Wall sections with reinforcement may now be drawn.
 - page 19 ☐ Walls can be designed as a 7"mm with the appropriate reinforcement
- Slab design and drawing:
 - page 15 ☐ Dividing lines:
 - a chain of nodes may now be selected.
 - walls may now be selected as dividing lines.
 - different dividing lines may now be defined for reinforcement in the X and Y directions.
 - page 15 ☐ dividing lines at "change of thickness" may now be applied to top and/or bottom reinforcement.
 - ☐ The algorithm has been improved so that there are fewer groups of reinforcement and the groups are organised more efficiently.
 - ☐ The placement of text on the drawing has been improved.
 - page 17 ☐ A minimum lap length may now be defined
 - ☐ Reinforcement areas to ignore: different limits may now be specified for top and bottom steel.
 - ☐ Add reinforcement: the location of the extent line perpendicular to the bar may now be specified by the user and it may be aligned with a similar line.
 - ☐ The cover at a slab edge may now be specified (distance to first bar parallel to the edge).
 - ☐ the program now extends reinforcement into walls located at the edge of a slab.
 - page 17 ☐ Bars may now be added where they are not required in one direction only.
 - ☐ when slab thickness changes, reinforcement from the shallow section is now extended into the deeper section:

**POSTTEN:**

- page 20 ☐ A table showing the cable elongations may now be displayed.
- ☐ Results along the beam may now be displayed at user-defined distances.
- ☐ Composite beams: the section with and without topping are now printed.
- page 20 ☐ Geometry table: the vertical cable coordinates can now be displayed relative to a zero coordinate specified by the user: top/bottom face of beam/topping.

AutoSTRAP:

- page 22 ☐ BEAMD files with beam geometry and loads may now be created.

* The increased program capacity will be supplied as a free update only to eligible users who have the current maximum capacity version.

Selection options

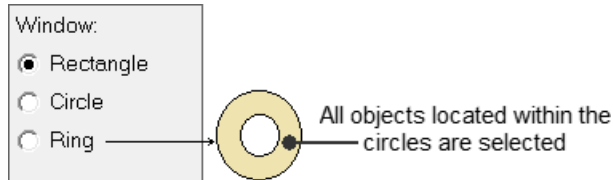
The following selection options have been added:

Node / beam / element selection

Select by window:

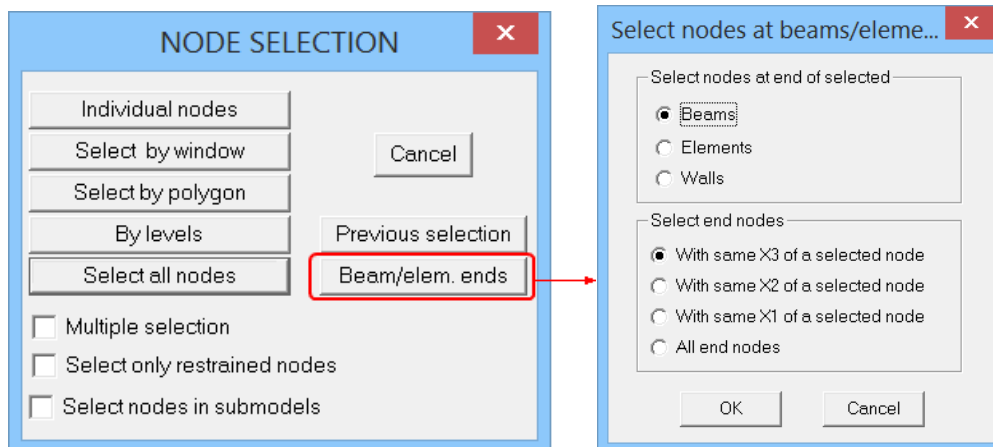
Define a window; the program automatically identifies all nodes located in the window.

The window may be either a rectangle, a circle or a ring.



Node selection

Select the end nodes of beams/elements/walls:



- Select the beams/elements/walls using the standard selection options.
- If you selected one of the **With the same Xn of a selected node**, select a single node in the display.

Example: define restraints at the base of columns (all at the same level). Select -

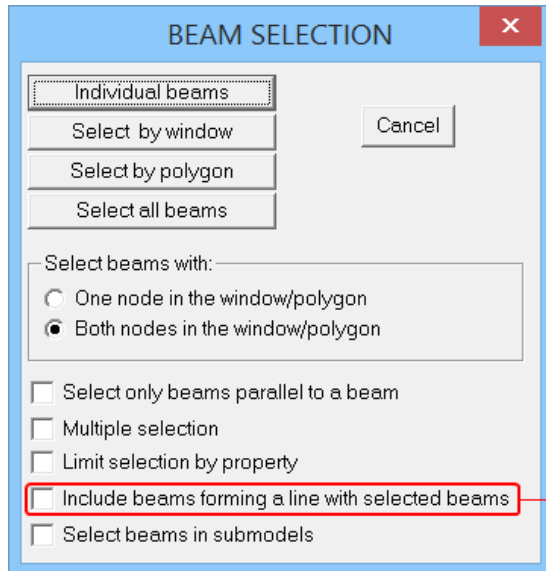
Beams

With the same X3 of a selected node

Then -

- **Select all beams**
- select the node at the base of one of the columns

Beam selection



All beams forming a chain on the same line as a selected beam are also selected

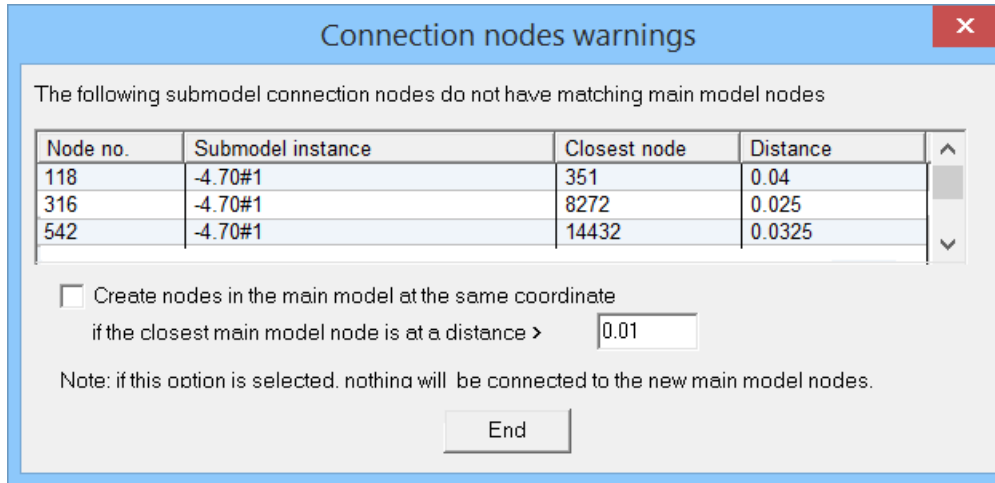
Wall selection

Walls are now selected by highlighting **segments** instead of end nodes.

Check element definition

The "Edit - check elements definition" option now checks the following in submodels:

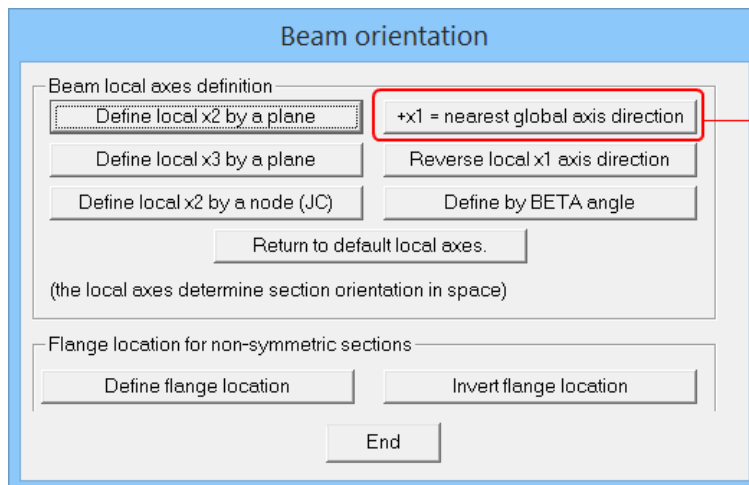
- two submodel instances added to the main model at the same location.
- main model nodes in the area of a submodel not attached to a submodel connection point.
- submodel connection points not attached to a main model node. The program can create new main model nodes at the connection points (note that they will not be connected to anything in the main model):



The latter two submodel problems are easily solved by using the Automatic connection points option.

Beam local axis

A new option has been added to define the local x1 direction:

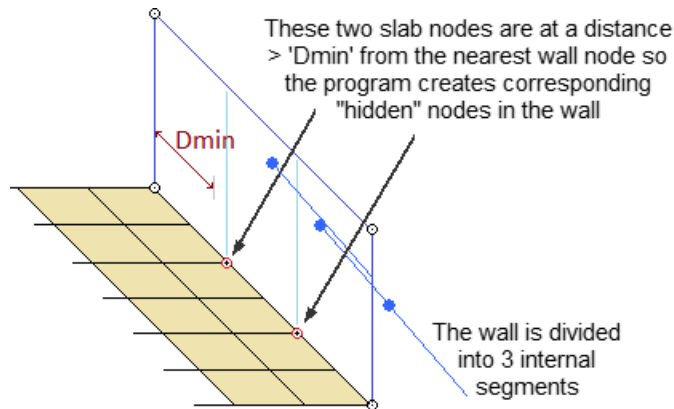


The local x1 axis is always parallel to the axis of the beam. Revise +x1 so that it points in the general direction of the closest positive global axis.


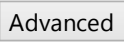
Wall definition

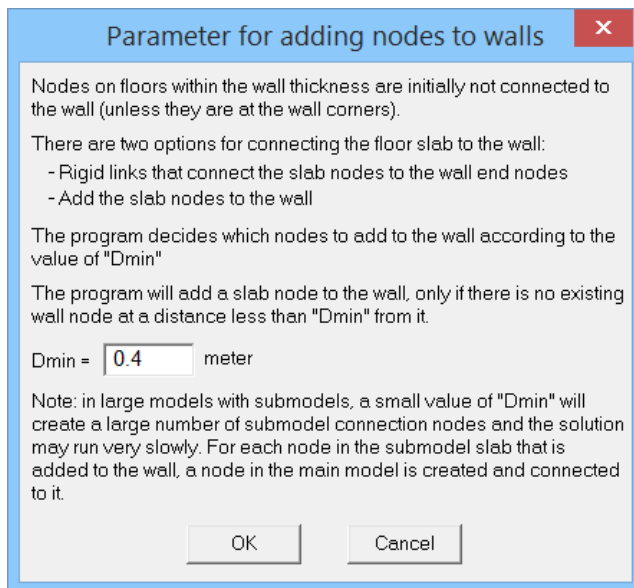
The wall definition option has been modified to increase the accuracy by improving the connection of the wall to the connected floor slabs.

Results become inaccurate in long wall segments with many rigid links connecting slab nodes to wall segment end nodes. To overcome this problem, the program now internally divides segments at existing nodes on the wall edges (usually from the attached slab) to create new segments (not seen by the user). For example:



The program uses a default value of $D_{min} = 0.4$ m. To revise the value:

- select the  Link option.
- click  and enter a new value:

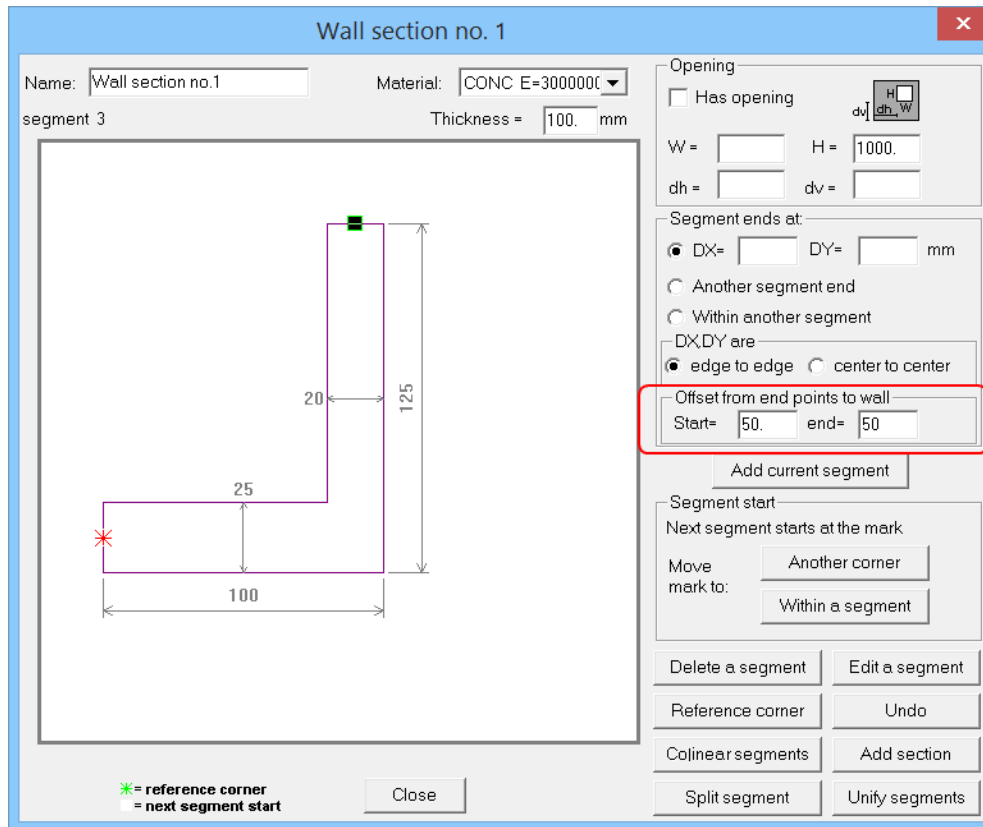


Note:

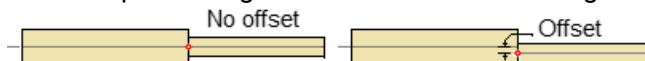
- If the value of **Dmin** is later revised so that the "hidden" wall nodes are at different locations, the program also automatically updates the rigid links and deletes unnecessary previous nodes from the Main model.
- This option will correctly create the nodes when the wall is in the Main model and the slab is in a submodel.
- The default value of **Dmin** was selected after testing to optimize accuracy vs solution time and changing the value is not recommended. Reducing the value of **Dmin** in large models with many walls will greatly increase the solution time.

Wall - offsets

Wall segments can now be offset from their center-line:



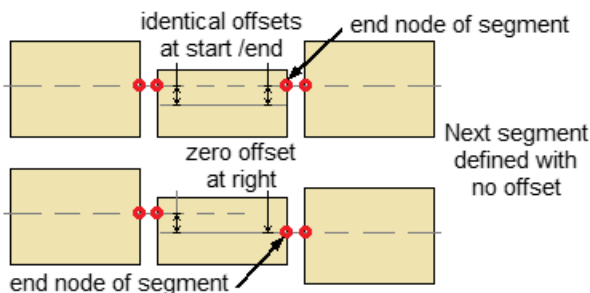
Use this option to align the faces of continuous segment with different thicknesses. For example:



Note:

- that the 'start' node of the segment is at the same location as the end node of the previous segment
- the 'end' offset defines the segment end nodes location. A zero offset at the right places the end node on the segment center-line.

For example:



Copy geometry

The program can connect the nodes in the original block to the corresponding nodes in the new block with beams. A new option has been added to connect only selected nodes:

Foundations

A new option has been added to transfer only the results from selected load combinations to the footing design module:

Skip disabled loads/combinations

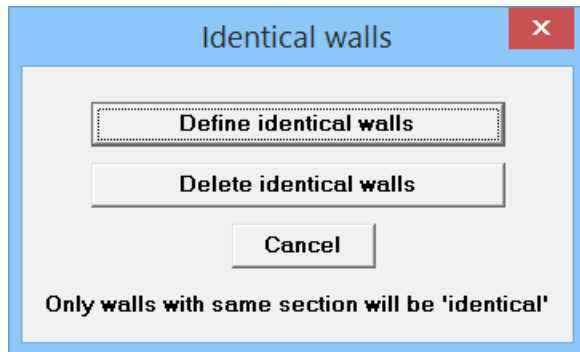
Load combinations may be disabled in the Display/print tabular results option. ☒ check this option to ignore the reactions from these combinations when transferring the reactions to the footing design. For example, transfer only the Service load reactions.

Concrete - identical walls

Identical walls can now be defined, similar to identical beams and columns; identical walls -

- must have identical wall sections
- must be stacked vertically

The program calculates the reinforcement required for each wall segment and assign the heaviest reinforcement to the corresponding segments in the "identical wall list".



For both options, select existing walls using the standard wall selection option.

Concrete - draw wall sections

Walls sections with vertical reinforcement can now be added to slab drawings:

Add an object to the drawing

Object to add

- ☒ Add a slab drawing
- ☐ Add a slab drawing - clipped.
- ☐ Add a bar schedule
- ☐ Add mesh schedule
- ☐ Add mesh details
- ☐ Add a wall section
- ☒ Add a wall section with reinforcement

Display

- ☒ Top reinforcement
- ☐ Bottom reinforcement
- ☐ No reinforcement
- ☒ All bars
- ☐ Bars in X dir.
- ☐ Bars in Y dir.

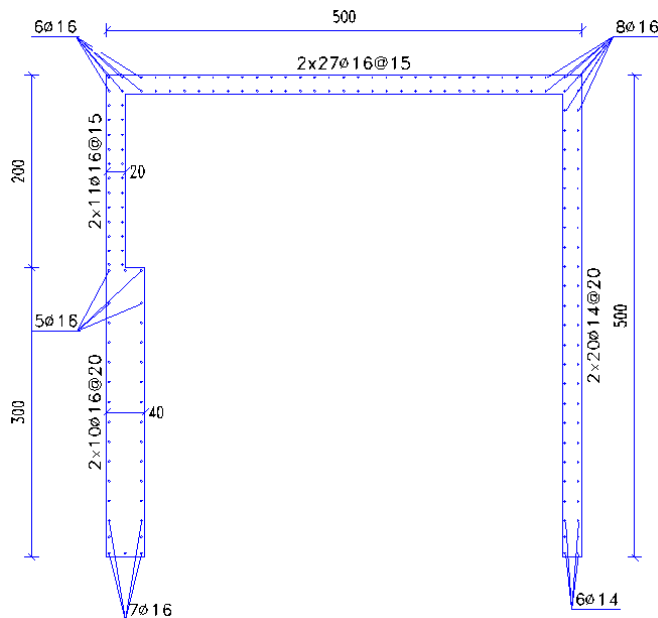
Detailing with fixed bars/mesh

- ☒ Display all bars
- ☐ Display only additional bars
- ☐ Display only fixed bars/mesh

Scale = 1: 20. Name

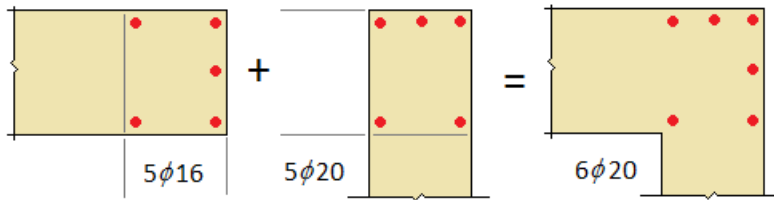
OK Cancel

This option creates a wall cross section drawing with reinforcement. The program currently draws only the vertical reinforcement. For example:

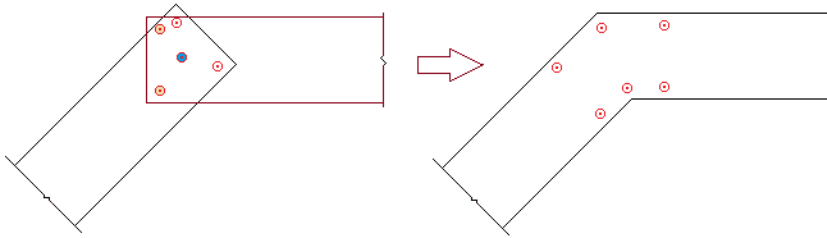


The program combines the reinforcement at the corners from attached segments:

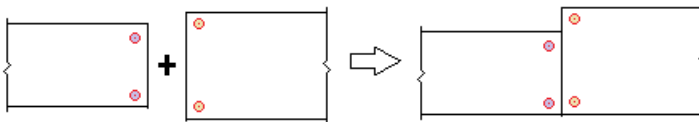
- the maximum diameter is used if the diameters in the attached segments are not equal
- the bars at the faces are combined. For example:



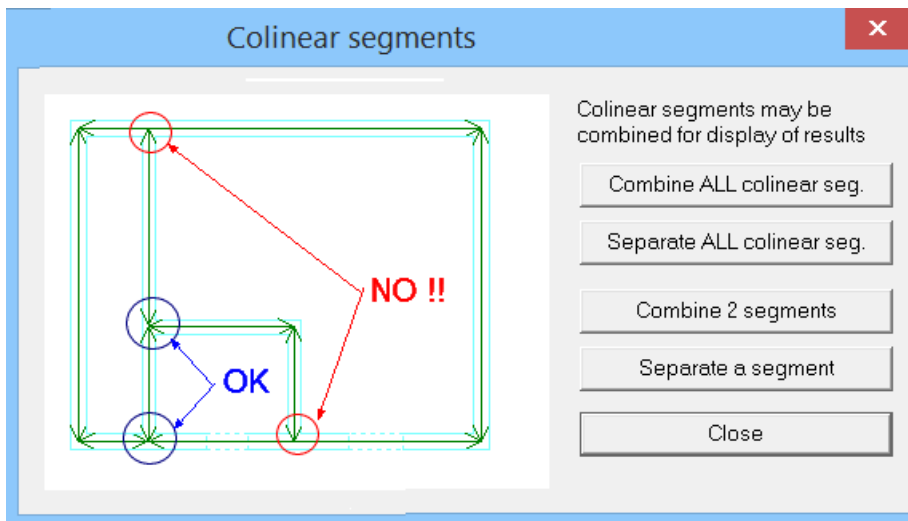
- attached segments that meet at angles $> 0^\circ$ and $< 90^\circ$ are combined as follows:



- attached segments that are a continuation (e.g. change of thickness, with or without offset) are combined as follows:



- segments should not be "colinear" at the junction with other segments because there will be no end reinforcement to combine:



Concrete - slab drawings

Dividing lines

New options have been added to the slab "dividing line" definition:

Define / change dividing lines:

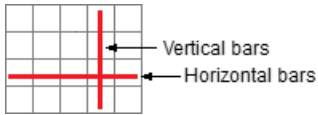
- subspaces can now divide vertical (on plan) reinforcement only, horizontal reinforcement only, or both.
- subspaces can now be defined by a chain of nodes.

To define/revise the line:

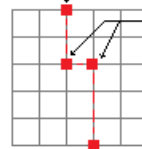
- select the dividing line type - it can divide top and/or bottom bars, bars in the X and/or Y direction:

When defining a line:

- select two nodes defining the dividing line or define a chain of lines to create an irregular space. Click on the last node twice to end the selection.
- "Horizontal" and "vertical" refer to the display (in plan):



(1) select the start node of the dividing line



(2) select the following nodes

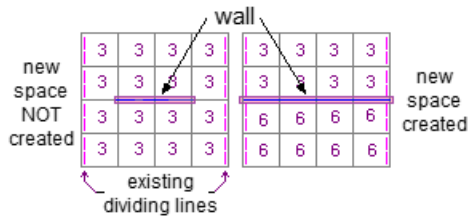
(3) click twice on the last node to end the line definition

Wall dividing lines:

Walls may be defined as dividing lines for top and/or bottom reinforcement. **All** walls are used as dividing lines if this option is activated (individual walls cannot be selected).

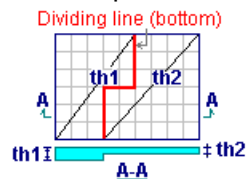
Note:

- a dividing line is not created if the wall does not extend to other dividing lines. For example:



Change in thickness

Create dividing lines for top or bottom reinforcement at element boundaries where the thickness changes. For example:



Parameters

New default parameters have been added to slab detailing:

General:

Bars added where not required can now be placed in one direction only.

General | Steel area | Bars | Mesh | Drawing

Height axis : X3 Code : BS8110 Level at 0.0 coord. = 0.

Reinforcement at top

- ☒ Bars
- ☐ Meshes (fabric)
- ☐ User defined mesh + bars where needed
- ☐ User defined mesh + meshes where needed
- ☐ User defined bars + bars where needed
- ☒ Put bars also where As=0

X direction: diameter= 10 spacing= 20. cm

Y direction: diameter= 10 spacing= 20. cm

Note: spacing=0 - no bars

Reinforcement at bottom

- ☐ Bars
- ☒ Meshes (fabric)
- ☐ User defined mesh + bars where needed
- ☐ User defined mesh + meshes where needed
- ☐ User defined bars + bars where needed
- ☒ Put bars also where As=0

X direction: diameter= 8 spacing= 20. cm

Y direction: diameter= 8 spacing= 20. cm

Steel area:

Reinforcement areas to ignore: different limits may now be specified for top and bottom steel.

General | Steel area | Bars | Mesh | Drawing

Concrete 30 Nominal steel strength 460. N/mm²

Cover

Top - X direction = 3. cm

Top - Y direction = 3. cm

Bottom - X direction = 3. cm

Bottom - Y direction = 3. cm

Minimum steel area

- ☐ Ignore
- ☐ Compute for slabs
- ☒ Compute for slabs or walls

- according to in plane force

Ignore required steel if less than

Bottom: 0.01 cm²/m Top: 0.01 cm²/m

☒ Use Wood & Armer moments

☐ Ignore in plane forces

☒ Use average moment over a strip of width = 1. m

Moments at columns

- ☐ Take moment at column center
- ☒ Compute average over column rectangle area
- ☐ Take moment at rectangle edge

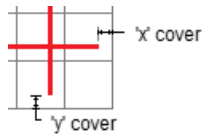
Note: Columns with reductions and their rectangles must be defined in the results module

Bars:

General	Steel area	Bars	Mesh	Drawing																								
<div> <div> Bar length Minimum length= <input type="text" value="200"/> cm Maximum length= <input type="text" value="900"/> cm Round length to: <input type="text" value="5"/> cm </div> <div> Spacing <table border="1"> <thead> <tr> <th></th> <th>Top</th> <th>Bottom</th> </tr> </thead> <tbody> <tr> <td>Optimum =</td> <td><input type="text" value="20"/> cm</td> <td><input type="text" value="20"/> cm</td> </tr> <tr> <td>Maximum =</td> <td><input type="text" value="30"/> cm</td> <td><input type="text" value="30"/> cm</td> </tr> <tr> <td>Increment when - spacing < optimum</td> <td><input type="text" value="5"/> cm</td> <td></td> </tr> <tr> <td>spacing > optimum</td> <td><input type="text" value="10"/> cm</td> <td></td> </tr> </tbody> </table> </div> <div> Diameter <table border="1"> <thead> <tr> <th></th> <th>Top</th> <th>Bottom</th> </tr> </thead> <tbody> <tr> <td>Minimum=</td> <td><input type="text" value="10"/></td> <td><input type="text" value="10"/></td> </tr> <tr> <td>Maximum=</td> <td><input type="text" value="25"/></td> <td><input type="text" value="25"/></td> </tr> </tbody> </table> </div> <div> Side cover x: <input type="text" value="2.5"/> cm y: <input type="text" value="2.5"/> cm </div> </div>						Top	Bottom	Optimum =	<input type="text" value="20"/> cm	<input type="text" value="20"/> cm	Maximum =	<input type="text" value="30"/> cm	<input type="text" value="30"/> cm	Increment when - spacing < optimum	<input type="text" value="5"/> cm		spacing > optimum	<input type="text" value="10"/> cm			Top	Bottom	Minimum=	<input type="text" value="10"/>	<input type="text" value="10"/>	Maximum=	<input type="text" value="25"/>	<input type="text" value="25"/>
	Top	Bottom																										
Optimum =	<input type="text" value="20"/> cm	<input type="text" value="20"/> cm																										
Maximum =	<input type="text" value="30"/> cm	<input type="text" value="30"/> cm																										
Increment when - spacing < optimum	<input type="text" value="5"/> cm																											
spacing > optimum	<input type="text" value="10"/> cm																											
	Top	Bottom																										
Minimum=	<input type="text" value="10"/>	<input type="text" value="10"/>																										
Maximum=	<input type="text" value="25"/>	<input type="text" value="25"/>																										
<div> <div> Divide bar group into 2 groups when Number of bars per group > <input type="text" value="8"/> Difference in As/s > <input type="text" value="0.5"/> cm²/m Length difference > <input type="text" value="150"/> cm </div> <div> Hook bars at slab edge - top <input checked="" type="radio"/> No hook at slab edge <input type="radio"/> Single hook A = <input type="text" value="15"/> cm A <input type="radio"/> Double hook B = <input type="text" value="20"/> cm A B </div> </div>																												
<div> <div> Nominal steel strength for bars <input type="text" value="460"/> N/mm² Increase positive moments by: <input type="text" value="1"/> <div> minimum lap length: <input type="text" value="0"/> cm </div> </div> <div> Hook bars at slab edge - bottom <input checked="" type="radio"/> No hook at slab edge <input type="radio"/> Single hook A = <input type="text" value="15"/> cm A <input type="radio"/> Double hook B = <input type="text" value="20"/> cm A B </div> </div>																												

Side cover:

Define the cover from the end of the bar to the perpendicular slab edge, in both directions:


**Minimum lap length:**

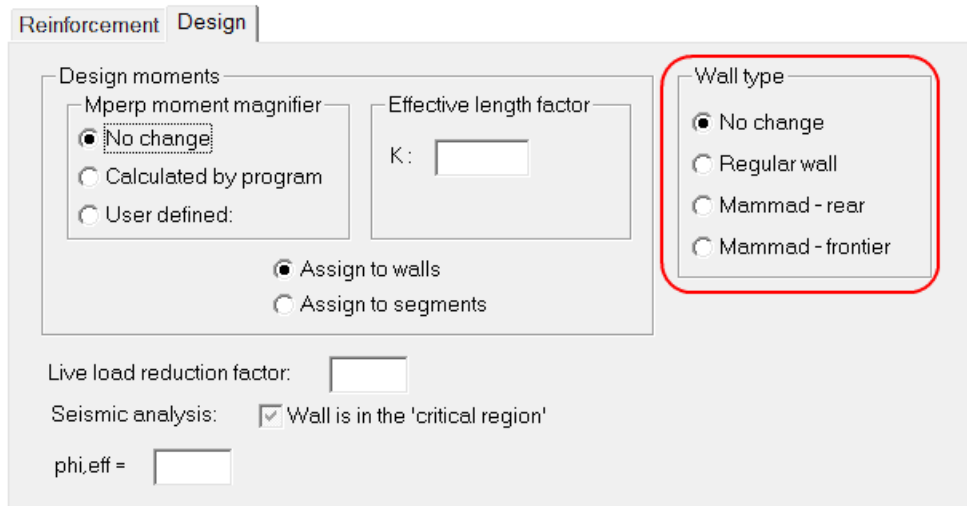
Specify a minimum lap length for all diameters.

ממ"ד - Concrete

The program can now design and detail walls in a ממ"ד according to the following document:

מנחה לסוגיות קונסטרוקטיביות בתכנון מקלטים ומרחבים מוגנים
דצמבר 2012

Select  Parameters in the side menu and the [Design] tab:



Select the type of wall:

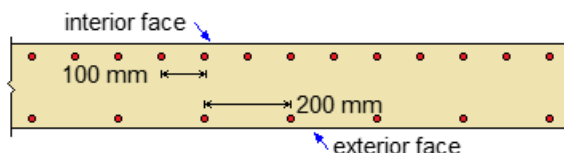
- ☒ **Regular** - ממ"ד not a
- ☒ **Mammad- rear** - ממ"ד ערפי
- ☒ **Mammad frontier** - ממ"ד קדמי

Note that the entire selected wall, i.e. **all segments**, will be designed as a ממ"ד.

The program provides **initial reinforcement** according to the requirements of the document; the program increases the reinforcement if the initial arrangement is not adequate for the design loads.

The relevant requirements in the document:

- Minimum diameter: according to Table 2 - $\phi 8$ - ערפי, $\phi 10$ - קדמי
and according to Appendix B if the governing wall dimension > 2.8 m (the program uses 3.0 m center-to-center)
- Initial difference between corner & distributed diameters: according to the Table in 6(a).
- Minimum corner bar diameter: $\phi 12$ - according to the examples in Appendix A.
- Maximum spacing : 100 mm - interior face, 200 mm - exterior face



POSTTEN

New options have been added to **Display/print tables**:

Display table - Composite beam

Select table to display:

- ☐ General project data
- ☐ Geometry Geom. display param.
- ☐ Stages data
- ☐ Shear
- ☐ Ultimate Moment
- ☐ Max. Deflections
- ☐ Min. Deflections
- ☐ Stresses at non-composite top/bottom
- ☒ Stresses at non-composite 1 * h and 0 * h
- ☐ Stresses at topping
- ☐ Additional forces applied by differential creep and shrinkage
- ☐ Additional forces applied by change of schema

Att = 30000 days

Maximum of t = 30000

Losses for selected cable

- ☐ Frictional and Draw-in losses
- ☐ Summary of losses

Cable elongation

- ☐ Elongation of cables from jacking

Display results at

- ☒ 1/n of span length, n = 2
- ☐ each 1 m

OK Cancel

Print tables

Select tables to print:

- ☒ General project data
- ☒ Geometry Geom. display param.
- ☒ Stages data
- ☒ Shear
- ☒ Ultimate Moment
- ☒ Max. Deflections
- ☒ Min. Deflections
- ☒ Stresses at non-composite top/bottom
- ☒ Stresses at non-composite 1 * h and 0 * h
- ☒ Stresses at topping
- ☒ Additional forces applied by differential creep and shrinkage
- ☒ Additional forces applied by change of schema

Att = 30000 days

Maximum of t = 30000

Losses for selected cables

- ☒ Frictional and Draw-in losses
- ☒ Summary of losses

Cable elongation

- ☒ Elongation of cables from jacking

Print results at

- ☒ 1/n of span length, n = 20
- ☐ each 1 m

OK Cancel

Geometry

Geometry display parameters

Height of cable drawing = 4 lines

Slab area

Display cables of : long and short cable lines only

Cable coord relative to :

Beam

- ☒ Top of beam
- ☐ Bottom of beam

Slab or slab area

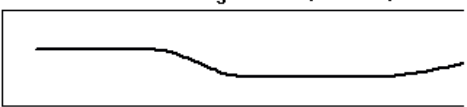
- ☐ Top of slab
- ☒ Bottom of slab

Composite beam

- ☒ Top of beam
- ☐ Bottom of beam
- ☐ Top of non composite

OK Cancel

MIDDLE CABLE starting at X=2.6, Y=0.25, Z=3.



For trapezoidal slab areas: display either:
- all cables
- the first and last cables only
(the middle cable is displayed if the first and last are identical).

CABLE COORDINATES (mm), relative to section bottom

x	30	530	650	1030	1270
y	100	71	70	81	100

Cable "y" coordinates can be displayed relative to the top or bottom of the beam/slab.

Jacking data

Display the cable jacking data: time, length, elongation, etc. For example:

Elongation of cables from jacking , beam no. 1						
Exit Print Copy						
Cable no.	Jacking time	Cab. len. before jacking	Jacking side	Jacking strands	Jacking force percent	Elongation at jacking time
	day	[cm]			%	[mm]
1	20.0	1801.1	Left	24	100	104

Time-steps analysis was performed.

Display results at

Results along the span can be displayed at user defined intervals:

Display results at

☒ 1/n of span length, n =

☐ each m

Select one of the following:

- ☒ 1/n of span: n=2 - results will be displayed at three points: start, middle and end.
n=3 - results will be displayed at four points: start, $(1/3)L$, $(2/3)L$ and end.
etc.
- ☐ each "n" m: results are displayed at the start, end and at every interval along the length.

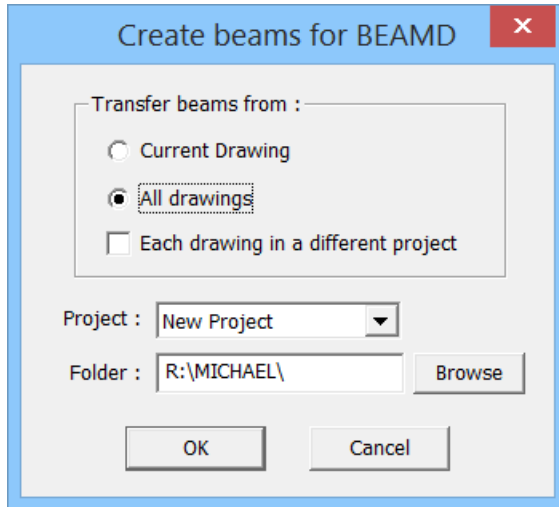
AutoSTRAP -> BEAMD

Create *BEAMD* files for beams on the drawings.

- the program transfers the beam geometry and the applied loads.
- design and detailing is done in the *BEAMD* program.

To create the files:

- select **Beamd** in the bottom side menu
- Select the drawings and the *BEAMD* project:



- Create *BEAMD* files for either the current drawing or all the drawings in the model.

If you select ☒ **All drawings**, the beams on each drawing can be created in separate *BEAMD* projects or they can all be created in a single project.

- Select an existing project from the list or create a new project; enter the new project name.

If you select ☒ **Each drawing in a different project**, the program automatically takes the project names from the list of drawing names in the Parameters - DXF files option.

- The beams can be added to those in a folder with existing beams (created either by *AutoSTRAP*, *BEAMD* or the *STRAP* concrete design module) or can be created in an empty folder.

- Click **OK** to continue
- select the beams to transfer
- Run the *BEAMD* program to complete the design of the beams.

Note:

- all slab loads must be applied as **beam loads** (bidirectional or unidirectional), not element loads.
- Select Display - BEAMD beam numbers to display the beam number in the *BEAMD* beam list.
- The program creates end supports as follows:
 - supported by another beam or column : pinned
 - supported by a wall - long wall dimension parallel to beam : fixed
 - supported by a wall - short wall dimension parallel to beam : pinned
- the program does not calculate reactions from a supported beam as a load on the supporting beam.
- when a beam is created again in the same folder (any project), the program **replaces** the previous beam.
- when a beam previously transferred is created in a different folder, the existing *BEAMD* file is not deleted. The number (**Bnnn**) of the last created version is shown on the *AutoSTRAP* display.