

	<h1 style="color: red; margin: 0;">STEEL</h1> <h2 style="color: red; margin: 0;">Main Modifications</h2>	
		<p>DEV Apr. 2017</p>
<p>DEV/UG/STEEL/M001 rev10</p>		

## Version 3.1 (April 2017)

### New functionalities

- Selection: Grow. Extend selection by adding connected beams.
- Menu/Selection/Load\_Beams/Like\_Current, two options are provided (for applying the current load on selected beams) :
  - 1) scaled by a fixed constant coefficient (user-s value).
  - 2) automatically scaled according to each selected beam loaded-width. This option requires that all loaded-width are defined but this allows to perform all of them at once.
- Post Check-list for Selection Only:
  - In Solver Setting, new check-box allowing to check stress-ratios for selected beams only. Checking this box On/Off is enough for refreshing (not necessary to solve again).
  - Print check (stresses and Max/min forces) is based on beam selection (when solver-setting is on, same as for check-table here above).
- Menu/Selection/Load\_Beams/OwnWeight allows to create Loads corresponding to Own-weight in order to target total Weight and COG for Cranes (builder's values).
- Backup file. At next run further a Steel crash, when backup-file exists, user may choose to re-load the temporary backup-file.
- Compute Settings / Allowable stress-ratio: display real values for selected material (info).
- Beams and Nodes input interface: new check-box to apply the next modified data value to all selected elements. In beams input interface, this new feature is not available for Hole and Nodes coordinates values.

### Enhancements

- Selecting a Beam (click a beam in general graph) automatically selects the first load-line (in table list of beam load-lines) loading this beam. Now, selecting again the same beam, leads to the next load-line, and so on....
- Delete all unused nodes. The first step of the Solver process is to identify all unused nodes. When Node selection is on, these unused nodes are selected (solver is canceled), then pressing "del-key" will remove all unused nodes. Improve is due to selecting unused nodes during solver process.
- Create new Node from Beam Panel input. The Node-List table was not increased. A new line is added in the Node-List table (and previous coordinates are kept).

## Version 3.0g (October 2016)

### New functionalities

- PLOT Beam curves: LY, LZ, TY, TZ, MY, MZ, are filled using Opacity.
- PLOT Background Color: provide input and save setting. Advice: first choose Black or White in order to set the legend color to White or Black.
- Tool selection, Apply load as current: provide scaling coeff.

### Enhancements

- Beam-Type = Bulb, associated plate should be considered as Continuous for local stresses (as well as for beam stiffness caract). Stresses induced by "Ty, Mz" are set to 0.

## Version 3.0f (April 2016)

### New functionalities

- Menu/Selection/Load Nodes/Load as current". Copy current node-load to all selected nodes.

### Bug corrections

- Combined Load-Case on Model with nodes restrained by forced displacement: All results are combined (reactions ...) except the node displacement itself. For better accordance, the resulting node displacement is also combined..

## Version 3.0e (January 2016)

### New functionalities

- Menu/Options/Graph general settings - text size for graphic display and legend.
- Menu/Options/Graph general settings - Colors for Materials and beam-types (when few colors <15).
- Menu/Analysis/Infos displays Weight and global centre of gravity for total model and selected beams, taking into account the material specific weight.
- Pop-up "Split beam", useful for Beam-type Linear scantling: When Beam selection mode is on, Right mouse click on a beam allows to split the beam into N beams, each partial beam gets a new beam-type for which the scantling values are set using linear interpolation between starting and ending beam-types selected. The beam selected cannot contain a hole definition. Both beam types at start and end must be the same standard section type, use same material...

### Enhancements

- Beam check (detailed): The local stresses components (i.e. normal stress due to compression, bending stresses due to My or MZ, shear stress due to torsion etc.) at X location along the beam can be displayed individually (not only as the total stresses).

## Bug corrections

- Status-bar: refresh title when "selection only".
- Show Beam Load Lines: refresh colors.

## Version 3.0d (January 2015)

### New functionalities

- Plot beam-type 2D and 3D: standard types are shown with their real size (all dim), a scroll-bar is provided for scaling from 0 to 1 (type Bulb-plate seems like an Angle with dimensions calculated according to Rule). The plot-3D is similar to plot-2D but stretched all over the beam-length, a scroll-bar allows to define the Transparency/Opacity.
- Tools/Clean/Unused Beam Type: remove unused beam types (when there is no existing beam in any model using that beam-type, this is useful for checking over all models), the number of deleted beam-types is given for information and "Edit/Undo" is available.
- Tools/Clean/Unused Beam Load: remove unused beam load lines (in the current model) when there is no existing beam loaded by the beam-load line sequence, the number of deleted lines is given for information and "Edit/Undo" is available. We advise to perform this command if "Tools/Copy Selected Beams" has been used in previous version (see bug correction).

### Enhancements

- Plot beam colors: material and beam-type colors are visible on the deformed model (if any) instead of intact model.
- Partial model: when selecting node/beam, the On/Off switch remains active, this allows better use for modeling.

## Bug corrections

- Beam selection : Ctrl+click on a beam already selected did not show it as unselected (problem was only a missing plot-refresh).
- Tools/Copy Selected Beams...: all beam loads were copied, now the copy-load is done for selected beams only (the beam-load sequence is lost, each beam must be loaded alone). The "Tools/Clean/Unused Beam Load" may be performed for cleaning the un-necessary beam loads.

## Version 3.0c (November 2014)

### New functionalities

- Beam Creation wizard: now if the keyboard SHIFT is down, last node from the previous created beam is the first node of the next created beam
- A new Tool has been added to copy a beam selection using a 3D translation vector (new Menu item: Tools → Copy Selected Beams). It is possible to choose at the same time whether the related Beam Loads and Node Loads are to be copied too. It is also possible, may the new copied beams rely on existing nodes, to choose a merging nodes option that

will use the already defined nodes instead of creating new nodes at the same locations (note that when the merging nodes option is ON, the node loads cannot be copied)

- The Check list and Print Check Synthesis contain now a trigger value expressed in stress (Sigma, Tau, Von-Mises) equivalent to the criteria threshold in %Re
- A new Input setting is now available (new Menu item: Options → Input Settings): it is now possible to define background colors for even & odd lines of the Data and Check lists (default is both white)

## Enhancements

- Operations on Model, Merge operation: it is now possible to specify the maximum gap (in m) used for merging nodes (all nodes separated by less than this gap will be merged)
- The possibility to "assign as current Beam hinges" over a selection was done before without affecting the hinge definition between local or global reference axis. This has been changed: now the reference axis of the current Beam hinges is assigned to all the beams in the selection (Menu item Selection → Assign as Current → Beam Hinges → Equal)
- The management of very large models (over 4000 beams) has been improved
- Plotting of the beam type sketches in the general graph: the scale for the sketch is fixed constant for all beams

## Bug corrections

- Input of a new HTI Beam Type from the data list spreadsheet: the error message that appeared at each cell validation when one of dimensions (H1, e1, etc.) was still equal to 0 has been removed. The correct definition of the dimensions of the beam type is checked anyway before running the solver.

## Version 3.0b (August 2014)

### New functionalities

- A new Tool has been added to connect all beams with a selection of beams: for each pair of beams in the selection, it calculates (and creates) intersection node and splits each beam if needed (new Menu item: Tools → Connect Selected Beams)

### Enhancements

- Plot Reactions: force value is now indicated
- Clicking on Node in the graphical area: the node is set to 'selected' so that the chosen node is now visible. This is useful when a node selection is needed in the various tools such as: "Create Beam", "Distance to node", "Limited model", "Angle 2 other Nodes", "Working Plan"

## Version 3.0a (July 2014)

*Version done for internal reasons*

## Version 3 (July 2014)



**There is a new license management system now activated for this version. Previous licenses will not be operational. Please refer to the appropriate document to know how to proceed.**

### Enhancements

- Compute Settings: the allowable stress ratios are now defined over 3 decimals
- Compute Settings: the default allowable stress ratio range limits are now 0.817 (= 192 / 235) for Sigma and 0.4085 for Tau (they were previously taken equal to 1 and 0.5)
- Check beam: at points 2 and 6 of a flanged beam type, as the flange has a thickness, the calculation points of Sigma and Tau are not at the same position. For the Von Mises stress now the calculation point of the Sigma component is taken same as the Tau calculation point at inner flange face. As a consequence the Von Mises stress may be a little bit lower than before, but it is now more precise.

### Bug corrections

- A correction has been made in the plotting of the deformed model; there was a problem with the beam release.

## Version 2.2b (Dec. 2013)

### New functionalities

- Allowable stress ratio range (stress criteria allowance for general Graph and check list of beams): now, for each one of the criteria Sigma, Tau & Von Mises, the allowable range is defined by a Stress/Re ratio, i.e. the stress-value is divided by the specific beam material Re:
  - Thus a common safety coefficient may be applied to various material having different Yield stresses (Re)
  - There is still one different safety coefficient for each criteria (Sig., Tau, V.M.)
- New plotting on the general Graph: there are new options to show the moments on nodes, both applied moments and results reactions:
  - The graphical representation of these moments on nodes is oriented to show sign and the radius is scaled to give an idea of the relative values
  - Note that the same scale is used for drawing the reactions and for the Node Loads
- New plotting on the general Graph: there are new options to view the axial force FX (kN) and torsional moment MX (kN.m) beam results values. They are displayed on the graph near the beam label:
  - This is useful to compare the critical buckling force for a beam

- FX is acting on the beam start node, so a positive value means a compressive stress, (an arrow-head on the beam shows the beam local x-direction)
- Colors are same as Shear force and Bending moment settings colors
- Check Beam display, there are new check-boxes available for removing Upper-flange and/or Lower-flange from stresses evaluation:
  - This is useful at beam-ends when the flange is not continuous
  - This option is available for "I" beam-types when both flanges are not equal or when the attached plating is not continuous
- Model operation - Add model : it is now possible to input the coordinates of a center of rotation (previously all rotations were done from the origin (0,0,0))
- A new Beam Selection is available based on the current material (from current beam-type): in order to check results (the user may take into account an appropriate criteria according to material)
- It is now possible to initialize the beam types loaded width with the attached plating value, taken as lower flange breadth when relevant (new Menu item: Tools → All Beam Types → Initialize Loaded Width)

## Enhancements

- Check Beam display, there is a new starting direction on the 3D-plot for the beam (reverse from previous) as it has been judged more obvious
- Now the 9 last opened files are available in the File menu

## Bug corrections

- A correction has been made in the licensing associated data and input form because of reported problems for computers with Japanese or Chinese (etc.) as "Current language for non-Unicode programs" (it's a Windows Regional setting). As a consequence of this change, previously input Client password will not be correctly re-read on all computers. **It will be necessary to re-type the correct Client Password before upgrading the license!**
- Check Beam display, there was a refresh problem in the stress values when there were overall stresses and a change of load cases. This has been corrected
- A correction has been made to avoid a critical crash of the program when creating a beam with the mouse mode wizard in the Graph while being displaying beam type sketch or beam type number or material number in "selection only" display mode
- In the data lists under MS Windows 7, several display problems were corrected: Nodes' displacements values are now with a white background and typed negatives values of the form "-0.xxx" are no more displayed as "--0.xxx"

## Version 2.2a (Aug. 2013)

### New functionalities

- Beam end release: any DOF (related to global axis) at a beam end may be released; it means that the beam will not transfer any corresponding force to the beam end node. For doing so use the "Global Axis (free DOF)" option in "Hinges" frame or the new "Release translations" frame in Beam Input form
- Beams for Overall Stresses in Model Main Data: it is now possible to select beams slightly oblique with respect of the axis X, where before only beams strictly oriented along axis X could be selected to have overall stresses applied. Now choose the angle to consider

(compared to X axis) for slightly oblique beams. Note that the overall stress is calculated at mid-z of the beam

- Check Beam display: when the Analysis option “calculate stresses at rigid ends” is unchecked, now no stresses are displayed in the rigid ends
- When deleting Beam(s), it is now possible to remove the beam(s) from the corresponding beam loads (and thermal loads): it will be asked when deleting beams
- A new option has been added in the Graph View Configuration form to display the beam type associated plating, for types H, T, I sections, Bulb Plate and Corrugated.
- New Beams Selection based on current beam direction (new Menu item: Selection → Based on Current → Beam Direction): all visible beams parallel (allowed gap of 10 degrees) to the current one are selected. The current beam may be chosen from the beam-list table (it is not necessary to use the select beam-mode from the graphic). This is an easy way for checking the “beam type attached plating” when using “Selection only” from the pop-up menu (mouse right-click in graphic area)

## Enhancements

- Print results: a summary at the end of the report on node displacements (by load-case) has been added to show the maximum deflection value for the whole model, the nodes and values corresponding to the maximum of magnitude and the maximum (absolute) for each global axis direction
- Plotting of My, Mz, Ty, Tz (on the general Graph): the display of the values at beam-start and beam-end locations has been added, attached to the corresponding curve, for the current selection of beams
- Plotting of concentrated loads:
  - Node concentrated loads on the general Graph: when several similar loads (with same global axis) are defined on one node (within one load case), the sum of all loads is now displayed (previously, each one was displayed separately, at the same place, so only the biggest one remained visible)
  - Beam concentrated loads in the Check Beam display (lower part): the total value of concentrated loads acting at a given x location along beam is now shown (previously each individual value was layered)
- "Ctrl+Q" Keyboard shortcut has been chosen to navigate between "Beam Selection", "Node Selection", "Beam Creation" and default mouse mode without using the toolbar buttons. A change of mouse cursor over the Graph has been added to clearly differentiate between the mouse modes

## Bug corrections

- There was a crash when deleting a beam while displaying stresses after a solving. This has been corrected
- Better Lists refresh after “assign as current” use

## Version 2.1c (Jan. 2013)

### Enhancements

- Display of the deformed structure: now if beam is with rigid-end = 1, then it will not be deformed
- Action to “Save As” done over a STEEL file that already exists (and different that the actual file) will be blocked

- Node input display : the list of the DOFs is now accessible in multi-select mode plus 3 buttons have been added to allow to clamp/un-clamp, to fix/free the translations and to fix/free the rotations

## Bug corrections

- Crash when "New Project" or "New Model" was clicked while the output display of "Bending Moment/Shear Force" was checked (no load-case). This has been corrected
- Beam Hole: the update of SF-BM values at points 1-2 when modifying xStart of hole has been added

## Version 2.1b (June 2012)

### Bug corrections

- A correction has been made to the licence load from file process that was bugged (report to changes done in versions 2.1a and 2.1 for the new licence management CERBERUS and to the "How to Register.pdf" document for more details on how to proceed if you haven't yet switched licence)
- Correction for critical crash of the software while saving projects on high capacity server drives
- Correction for crash when deleting the last node load of a load case
- Correction in the copy load case functionality (in the Load Manager) to make it clearer on how the coefficients are used

## Version 2.1a (Jan. 2012)

### Enhancements

- Following the change of licence management (CERBERUS) in the last version (2.1), if the direct access to the licence server doesn't work, now the program stays open (but disabled) to allow access to loading licence file menu. Refer to the document "How to Register.pdf" for more details on how to proceed
- Node Loads viewing option now shows external Bending Moment applied on node
- Two new viewing options have been added for displaying Materials
- Enhancement of the inserting node on beam tool: a modified value (either distance or percentage) now activates the corresponding radio button (case to be followed)

### Bug corrections

- When changing the standard type of beam-type, or recalculating the beam-type section data, both eccentricities (shear-centre to centre of gravity) are reset to zero



## Version 2.1 (Nov. 2011)

### Enhancements

- The main enhancement of this version regards the change of licence management. The previous system was too old to work properly with the last versions of Ms. WINDOWS. Therefore we have adopted the CERBERUS licence management already used in other software of the Marine Division like VeriSTAR Hull. Refer to the document “How to Register.pdf” for more details on how to proceed
- Enhancement of the inserting node on beam tool: the input of one the coordinates instead of percentage of length is now possible. It is also possible to input a distance instead of the percentage
- While loading selected beams with the hydrostatic selection tool, users were surprised to be shown the error message "No beam selected" (whereas there was clearly some selected beams). The problem was caused because no loaded width had been previously defined on the beams. The error message has been clarified
- Check Beam display: on the M-N-T plot the node indexes on X axis have been replaced by the node labels

### Bug corrections

- Crash when no node (on new model) and further input of a node coordinate (in the input window): corrected
- Aluminum default weight, previously set to 2.7, has been corrected to 27
- Crash on opening the Check Beam while no model solved: corrected
- Check Beam display, the X combo box had lost the default focus that was useful for users to scroll between x values directly on incoming the form: corrected

## Version 2.0b (Apr. 2011)

### Enhancements

- Like foretold at the release of the version 2.0, a redesign of the Check Beam feature has been completed. It is now accessible at all times and for all types of beams (including composite and user defined types). In the new pop-up window, it is now possible to view the curves representing the beam loads applied on the beam as well as the bending moments and shear forces if the model is solved
- NOTE: The restriction counselled since version 2.0 to access the Check Beam pop-up window only when the view parameter “Stresses values” selected is now lifted.
- NOTE: The profile of the beam displayed for a composite and user defined beam type will be based on the input values of Inertias and Moduli (and not on the real profile on the beam)
- New Tool for Nodes: measure the angle between 3 nodes. While in the select node mode, click one node (1), right-click on the graph to access the pop-up menu and click on "Angle between nodes", then select 2 more nodes (2) and (3). The result will appear in the status bar as the angle between the (1-2) and (1-3) directions
  - A new Selection feature has been added: the selection of All visible elements (found in Menu: Selection → Select All Visible or through the keyboard shortcut Ctrl+W). Equivalent to on “mouse squared area” which selects all beams displayed in a graphic area, the idea here is to first apply zooms to get only the desired beams visible on screen then use this feature.
  - The Zoom box has a new behaviour: it will automatically be unselected after 1 use (come back to normal mouse behaviour)

## Bug corrections

- Crash when Solve with no beam type: corrected
- Due to changes in version 2.0, some beams could get a label duplicate when proceeding with a Model Extend: this has been corrected, please check your models if you have used the Extend function

## Version 2.0a (Dec. 2010)

### Bug corrections

- Beam hole size may become wrong when input hole size first and then location

### Enhancements

- Beam hole 830x500 added in list

## Version 2.0 (Nov. 2010)

**NOTE:** Version 2.0 introduces a major overhaul in the STEEL interface. This is a consolidated version but there are still elements planned to be implemented for the next version, like:

- A better Check Beam display
- A new "Rebuild" tool which could perform renumbering, cleaning, etc., of a model

### Enhancements

► Inputs:

- Beam Type name: automatic name based on scantling data. New button "OK & Rename" in beam type input (Section dialog). Naming under the form:

Box	$\text{Box} - B \times T_1 \times T_3 + H \times T_2$
U section	$\text{U} - B \times T_1 + H \times T_2$
Tube	$\text{Tube} - D \times T$
Bar	$\text{Bar} - D$
Bulb	$\text{HP} - H \times T + H_{\text{plate}}$
Corrugated	$\text{CorrV} - D \times U \times V \times e - \text{width}$ or $\text{CorrU} - D \times U \times e \times 1 \times e^2 - \text{width}$
Composite	Composite "i"
H, T, I	$H, H - H_{\text{web}} \times T_{\text{web}} + H_{\text{flange}} \times T_{\text{flange}} + H_{\text{plate}} \times T_{\text{plate}}$ $H, H - H_{\text{web}} \times T_{\text{web}} + H_{\text{plate}} \times T_{\text{plate}}$ $T, T - H_{\text{web}} \times T_{\text{web}} + H_{\text{flange}} \times T_{\text{flange}}$ $I, \text{FB} - H_{\text{web}} \times T_{\text{web}}$

- Load input when no Load Case defined: automatic start new Load Case definition
- Holes: the beam hole data is saved in the project database, each beam can store only one hole (split the beam if more needed)

- Overall stresses related data now saved in project database. See Project and Model Main Data
- Model translation operation: includes model rotation (the rotation center is located at origin). It is highly recommended to check the beams local-axes, because the orientations are not kept when parallel to Y axis
- Materials: add default values for Aluminium and INOX steel
- Beam input: display the beam length (read-only field, in the beam input specific window)

► Results:

- Stress Allowance Criteria: provide allowable (min-max) values for each stress criteria and perform check according to multi-criteria (user's selection)
- Beam selection, new feature through Menu: Selection → Based on Check: all beams listed in the Check table in the Data list are selected
- New print option: Beams – Internal forces, in Menu: File → Print Check synthesis: prints the extreme values of "FX, TY, TZ, MX, MY+, MZ+, MY-, MZ-" for each beam

► Tools:

- New Beam Connection tool, through Menu: Tools → Create intersection node: for beams crossing over but not attached
- New tool for merging two continuous aligned beams (remove intermediate node), through Menu: Tools → Merge Beams. Not tested by the process: both beams should have their local axis with same direction
- Show List of load-lines for current beam(s) selection (Menu: Tools → Highlight beam loads...): in the Beam Loads table of the Data list window, all lines loading the selected beam(s) are highlighted. This menu item should work as an On/Off switch but needs sometimes to be refreshed

► Graphical area:

- Stresses values legend: option for auto-legend based on visible beams only (see Menu: Analysis → Settings)
- X, Y, Z toolbar buttons to 3D. There is no more 2D input mode, so it is advised to define a working plan, if needed, especially when creating new beams using the grid
- New Menu: Edit → Copy Graph to Clipboard, provides a copy to clip-board function

► Calculations:

- Bulb Beam Type: geometrical properties computed using the equivalent angle profile defined according to BUREAU VERITAS Rules for the Classification of Steel Ships – Pt B, Ch 4, Sec 3, [3.1.2]
- Allowable stress: default for maximum value is set to 192 N/mm<sup>2</sup> (it was previously 200) for Von Mises and Sigma criteria and 96 N/mm<sup>2</sup> for Tau

## Bug corrections

- Capacity exceeded for beams and nodes Labels. A crash occurred when an "Add" or "Extend" operation was performed on a model which contained already labels with high numbers, the new labels created were exceeding the maximum integer capacity 32 767. Corrected by reducing the label gap introduced in the automatic numbering of nodes and beams by taking into account the real amount of nodes and beams instead of just previous assigned labels
- Data list window: while editing a cell, the keyboard keys "INS", "DEL", "←", "→" will affect the cell contents instead of triggering the validation and exit of the cell
- Ratio in full Rigid-end beam, the ratio was calculated at the very end of the beam (last section), even when the corresponding check-box option was off
- Data list window: some label inputs (like Nodes/Beams: Label/Start/End) were limited to 4 digits. New limit fixed to 5 (number should be less than 32 768 though)
- Beam Type input, when changing the beam section type (i.e. Tube, I, Box ...), the unused data is automatically reset to zero

- Models combo-box selection: when 2 models had the same name, only the first appeared to be selected (even though it was the second one being displayed). Corrected
- File Save process, the current directory was not remembered in case of new Project. Corrected
- When there is no beam defined yet in the model, the nodes are non the less plotted
- The main Graph is refreshed when moving the detail Check Beam window on top (previously the Graph area was becoming blank)
- Save process: check disk-space (previously when not enough space, an empty data-file was saved)

## Miscellaneous

- Release of a document "Getting Started with Steel 2.0", complement to the Help, (document reference: DEV/UG/STEEL/M002).
- Demo file "bulk.stw" completed to show examples of Overall Stresses and Hole on Beams use

## Version 1.9b (Oct. 2009)

### Enhancements

- Stress criteria Check: all beams which are listed in the Check-tab (Data list window at screen bottom) are selected (red color) in the Graph model view. Previously, only those higher than the maximum allowable were selected
- Grid-points color set to white when Graph background color is Black
- Beam / Node / Thermal Loads input when no Load-case defined: automatic start Load Manager
- Distance tool: display distance between two selected nodes. When Node Selection mode is on, right-click on a first node, choose "Distance to node" in the pop-up menu, then select a second node (by left-click), the distance (in m) between the two nodes is displayed in the Status bar
- Model View: new feature allowing rotation of the model around a selected node. When the Node Selection mode is on, right-click on the node and choose the "Centre View on node" item in the pop-up menu
- Menu: Selection → Assign as current → Beam rigid-end: new feature allowing multiple rigid-end definition at once.
- Beam Type view: scaling for beam type sketches adjusted (size = beam length / 5)
- Allowable stress: default for maximum value set to 192 (before 200)
- Active Tab selection in Data list window: right-click on Tab to automatically display the corresponding input window and automatically switch the Node / Beam Selection related to the selected-tab in the Toolbar

### Bug corrections

- Stress ratios Check: the X-beam section located exactly at the end of rigid-end length is always checked (even when the rigid-end check is not required) Previously, that limit section was considered within the rigid-end area
- DOF view selection dialog: hidden when Steel is minimized

## Version 1.9a (June 2009)

### Enhancements

- Combo-box in Toolbar which allows current Model selection

### Bug corrections

- Menu: Selection → Manager → Delete: error when deleting last remaining set. Corrected
- Added beam types remaining when cancel solving (in case of connection problems)
- Uncheck ratio for rigid-end, but remains active for full rigid-beam