

STRAP 2021

Highlights:



Import IFC file (AutoSTRAP):

- The import algorithm has been improved significantly.
- The IFC import parameters can now be revised after importing the file.
- Define or delete IFC levels prior to importing to AutoSTRAP.



Stage for dynamic analysis:

Select the construction stage for dynamic analysis, when more than one stage is defined in geometry.



Walls:

Curved walls can now be designed and drawn.

Detailed List of Enhancements:

General:

- Setup
 - Wall elements display: Walls can now be displayed with fill color and opacity.
 - Line width for screen display can now be specified for all objects: beams, elements, loads, and results.
 - The command "Limit display to a plane" default settings can be revised.
- Tool bar icons are now displayed as "pressed" when the display icons are active. (for example: node numbering, property by color, etc.).
- Support symbols have been revised.
- Torsion and axial force beam release symbols are now displayed.
- The cold-formed sections table is now separated from the hot-rolled tables.

Geometry:

- **General:**
 - Copy/delete, the new "Delete part of a model" command allows the deletion of several element types (beams, elements, walls, etc.) with one command.
 - "Model Wizard". The whole definition process is improved and now includes explanation drawings.
 - Node selection. An option to select only unrestrained nodes.
 - Create an IFC file: STRAP now exports vertical and/or inclined elements surfaces.
- **Beams:**
 - Offsets: the following options have been added -
 - Offset type: select beams and specify whether to generate additional moment due to offsets.
 - Remove all offsets at node: select a node and remove offsets from all beams connected to it.
 - Replace beam end node: select a different end node for beam with an offset. The beam geometry remains the same and an offset to the new node is created.
 - Change end node location: Move the end node to the physical end of a beam and delete the offset. New offsets are created at the ends of all attached beams.
- **Slabs:**
 - Mesh angle: rotate the mesh of generated rectangular elements.
 - Releases: continuous supports can now be defined along selected slab edges.
- **Walls:**
 - Curved walls can now be defined as a design unit for moment and/or shear.
 - Output: Display a table drawing of walls sections segment indices.
 - Create wall section from beams: an option to not delete the beams has been added.

- **Submodels:** When defining a submodel using "Select part of the model", the connection points type can now be specified.
- **Stages:** Select a stage for modal analysis. This option provides flexibility by deactivating/revising different elements for static analysis and dynamic analysis.

Loads:

- **Wall loads:**
 - Area load: A new option to define area load on selected wall segments or the entire wall in any global direction.
 - Line load: A new option to define line load at the top level of selected segments or the entire wall.

Existing load: The program now immediately displays the loads when a row in the Load case table is highlighted.

Self-weight: The self-weight now defines self-weight for the entire model. (beams, elements, etc.)

DXF Drawing:

- A DXF drawing can be displayed in the background. (similar to Geometry)
- The DXF points can be selected when defining Global loads.

Temperature load: A new option to use the beam section height to define the temperature gradient.

Results:

- **Punching:** Punching for wall edges/corners is now available. The user must insert the punching shear force and the program will calculate the punching based on the selected design code.
- **Contour map:** a new option to display spring stresses.
- **Element results along a line:**
 - The program saves the last defined lines after exiting the results module.
 - When defining a new line, a new option to delete all previously defined lines.
- **Force reactions - Envelope:** Display minimum or maximum reaction values for load cases or load combinations.
- **Beam result diagram:** The program now displays the positive and/or negative values with different colors.

Dynamic:

- **Weights:**
 - Static load: While defining mass by converting static loads, multiple load cases can be converted in a single command.
 - Total weight: The program now displays (at the bottom left corner of the screen) the weight defined on displayed nodes. (in addition to the total defined weight.)
- **Stage for dynamic analysis:** Now under the geometry tab > stages, the user can select a stage for modal analysis. This option provides flexibility regarding the model used for static analysis and dynamic analysis.
- **Mass radius of inertia:** The MMI and mass centers tables now include the calculation of mass moment of inertia at each level.

Concrete:

- **Beams/Columns:**
 - Horizontal shear for beams: An option to calculate reinforcement for horizontal shear. The program calculates A_v/s for vertical shear and A_h/s for horizontal shear stress. The maximum result is displayed.
 - Beam/column properties table: The table display has been improved and is now similar the properties table in the geometry module.
 - Solid section columns (defined with *CROSEC*): The program now calculates column shear reinforcement for any section with an arbitrary.

- **Walls:**
 - **Coupling beams:** The program now calculates diagonal reinforcement for earthquake design. The results are displayed in both the results summary table and detailed results table.
 - **Curved walls:** Curved wall sections can be designed and drawn.
- **Slabs:** A new option to remove the display of the element mesh when checking slab detailing .

Steel:

- **Section tables:**
 - All section tables (American, European, User, etc.) are now available simultaneously for steel design. The limit of two section tables has been removed.
 - Reloading the steel tables after revising or adding properties is not necessary.
 - The program saves the steel sections for a model. For example, when models are copied from one computer to another, the user defined steel sections are now copied.
- **Cold-formed steel:** Crosec line sections can be copied to the clipboard and then pasted in STRAP's geometry for analysis and design.
- **Section shapes for design:** Solid rectangle and solid round sections defined by dimensions in the geometry module can now be designed.
- **Composite:**
 - Stud spacing: An option to design the stud spacing in the positive moment region along the beam.
 - Partial connection: The minimum number of required studs for partial connection is now computed.
 - Parameters > composite: Sections defined as composite in the geometry module can now be revised to non-composite in the Steel design module.
 - The general arrangement display now shows both the steel and the concrete sections of composite columns.
- **Detailed results:** The detailed result tables now include a section drawing and local axes orientation.
- **Section optimization:** The selection menu and definition menu includes a section drawing.
- **Combined beam:** Parameters display of combined beam will be displayed once next to the first combined member.

Prestress:

- **Shear:**
 - The program calculates the minimum shear reinforcement based on specified code requirements.
 - Calculation of horizontal shear for prestressed composite sections.
- **Stresses:**
 - Time envelope. Display stresses envelope for all defined time steps.
 - The stresses envelope table now displays selected stress values at points along the prestressed beam. The time of the selected stress value will be displayed as well. The selected time step will be the time step with the greatest value of (Actual stress/Allowable stress) ratio.
- **Cable losses (pretension):** An option to define a factor for transmission length (lpt) loss.
- **Setup:** Default settings can be saved under File > Setup.

AutoSTRAP:

- **IFC:**
 - Define or delete IFC levels prior to importing to AutoSTRAP.
 - An option to go back from AutoSTRAP to the IFC model window to revise the IFC parameters.
 - The IFC model window tool bar now includes display/undisplay IFC elements, rotation options and limiting the display by coordinates.
 - Colors for the different IFC elements (beams, walls, removed elements, etc.) can be defined.