

What's New in Advance Design 2024



Table of contents

1.	Welcome to Advance Design 2024	4
2.	Advance Design - Quick list	5
3.	New computing capabilities	11
3.1.	Carbon emission estimations	11
3.2.	Cost estimations	. 16
3.3.	Displacement control nonlinear analysis	. 20
3.4	Possibility for defining initial constraints on linear elements per load case	. 26
3.5.	Better formulation of drilling rotation in shell elements	. 28
4.	Enhance steel structure design capabilities	. 30
4.1.	Verification of steel cold-formed sections according to AISI S100	. 30
4.2	.Shear web buckling analysis for I sections (EN 1993-1-5)	. 33
4.3	.User definition of buckling and LT buckling lengths for cold-formed sections for EC3 .	. 39
4.4	. Modelling welded tube connections	.43
4.5	.Deflection in value (cm or mm) in the shape sheets and diagrams	.46
5.	Enhancing the analysis of timber structures acc. Eurocode 5	. 48
5.1.	Design of spaced rectangular sections (EC5)	. 48
5.2.	Verification of bearing pressure on the support acc. EC5	. 52
5.3.	Verification of beams with a notch at the support acc. EC5	.54
5.4	.Optimization of timber elements according to the deflection (EC5)	57
6.	Enhanced user experience and the comfort of program operation	. 59
6.1.	User templates for Selection	. 59
6.2.	New options for selecting identical elements	63
6.3.	Possibility for filtering inactive elements	64
6.4	.Filtering profiles from databases	64
6.5.	New commands on the ribbon for adding stages	65
6.6	Improved mesh generation for Pushover analysis	66
6.7.	The next stage of unification of dialog windows	66
6.8.	Improvements to the selection by criterion window	67
7.	Other novelties and small improvements	69
7.1.	New database with cold-formed steel profiles for North America	69
7.2.	New steel material database for North American cold formed sections	.70
7.3.	New parametric section – spaced shafts	.70
7.4.	Improved combination creation with seismic cases for Canada	71

7.5.	Defining Link at node on selection	72
7.6.	Possibility of material definition for supports	73
7.7.	Dedicated system for master-slave links	73
7.8.	Improvements to the program defaults for North America	. 74
7.9.	Shorter time of generation of load combination	75
7.10	. Improvements in localization for Spain and Portugal	75
8.	Design Modules - Common improvements	76
8.1.	Allow applying project templates when module is already opened	76
8.2.	Display the steel ratio in the Info Panel	77
8.3.	Small improvements	77
9.	RC Beam	79
9.1.	Import geometry and internal forces from Rib design	79
9.2.	Possibility to define the position of supports (columns/ walls)	. 80
9.3.	New support type "beam"	81
9.4.	Ability to define secondary beams	. 82
9.5.	New chapter in report on openings reinforcement	. 83
9.6.	New diagrams for reinforcement for torsion	. 84
9.7.	Small improvements	. 84
10.	RC Column	. 89
10.1.	Different upper beams geometry	.89
10.2	. Possibility to disable the interaction curves checks	.90
10.3	.Minor improvements to ribbons	.90
10.4	. Minor improvements in Info Panel	91
10.5	. Improvement to transverse reinforcement on Edit dialog	91
11.	RC Footing	. 93
11.1.	Representation of the substitute footing	. 93
11.2.	Additional information on concrete cracking in Info Panel and report	.93
11.3.	Auto correction of reinforcement when crack widths are too large	.94
11.4.	Improved editing of the distribution of bars	.94
11.5.	Information in the report about the final reinforcement area	95
11.6.	Improvements to seismic bearing capacity calculations for France	.95
11.7.	New method of determining soil pressure	. 96
12.	RC Wall	. 98
12.1.	.Generation of drawing with schedules for bars and fabrics	.98
12.2	. Import and export forces using Excel file	.99
13.	Masonry Wall	100

13.1.Loads representation in viewports	
13.2. Expanded report for RC6 calculations	
13.3.Possibility to choose the section for the stress diagram	
13.4.Saving the masonry databases data per model	102
13.5. Presentation of partial factors	103
13.6.Small improvement: Support depth representation in viewports	103
14. RC Slab	104
14.1. New bar shapes	104
14.2.Possibility to manage hooks for reinforcement zones	105
14.3 New options for defining reinforcement solution	
14.4.New methods for defining reinforcement zone graphically	107
14.5. Set of improvements to drawings	
14.6 Set of smaller improvements	
15. Steel Connections	
15.1.Welded truss tube connections for square/rectangular tubes	113
15.2. Improvement on the Info Panel for Shear plate joint	117
15.3New option to ignore bending moment in calculation for hinged beams	

1. Welcome to Advance Design 2024

GRAITEC is pleased to present the latest version of the leading structural analysis software – **Advance Design 2024**, part of the Graitec Advance suite.

GRAITEC has continuously strived to provide first-rate advancements for innovative software solutions to its valued customers, and the recent launch of its new and upgraded product range for 2024 is no exception, proving they are still on top of its game in terms of providing top-level Construction, AEC, and Building Design software solutions worldwide.



This version 2024 of Advance Design is enhanced with a lot of users-centric new functionalities with high-end benefits, and is articulated around a few main subjects:

- **New computing capabilities** directed towards **sustainable design**, including carbon emission estimations, cost estimations, but also displacement control nonlinear analysis.
- Enhancing the analysis of timber structures acc. Eurocode 5 by increasing the scope of verification, including new Eurocode checks for bearing, notches, and compound sections, and the optimization of timber elements according to the deflection.
- Enhance steel structure design capabilities mainly by allowing design checks of cold form sections acc. American AISI code, web buckling analysis sections acc. Eurocode 3 and new welded truss tube connections.
- Enhance concrete structure design capabilities with new options and improvements to RC design modules, including new shapes of bars for RC Slab, new drawing capabilities, or transferring rib results to RC Beam.
- Enhanced user experience and the comfort of program operation by introducing improvements to facilitate everyday work, including templates for selections, and new commands for faster selecting, or filtering sections on databases.

Version 2024 of Advance Design also comes with a vast number of improvements and adjustments following the feedback received from thousands of users worldwide.

2. Advance Design - Quick list

This is a condensed/short list of new features for Advance Design 2024:

New computing capabilities

- **Carbon emission estimations** Determination of CO2 emissions based on carbon factors entered for materials and/or elements. Results in graphical form and reports.
- **Cost estimations** Determination of costs based on unit prices for materials and/or elements. Results in graphical form and reports.
- **Displacement control nonlinear analysis** The ability to conduct nonlinear calculations using displacement increment steps, which allows to correctly analyze highly nonlinear problems with the post-peak behaviors and can easily manage the snap-through problems.
- **Defining initial constraints on linear elements per load case** A new way of defining and considering initial constraints, making it easier to include initial constraints in combinations, as well as considering their impact on the rest of the structure.
- A better formulation of drilling rotation in shell elements Thanks to this new formulation, shell elements are now capable of capturing torsional moments transmitted from perpendicular beam elements.

Enhance steel structure design capabilities

- Verification of steel cold-formed sections according to AISI S100- The possibility of performing the standard verification of cold-formed steel profiles according to the AISI S100- 16 American standards.
- Shear web buckling analysis for I sections according to EC3 Verification of the stability of I-section webs (including the impact of transverse and longitudinal ribs) according to EN 1993– 1-5.
- User definition of LTB lengths for cold-formed sections for EC3 Next step of improvements to cold-formed design acc. Eurocode 3 for analyzing buckling and lateral-torsional buckling, especially applied when the Advanced Stability analysis is not used.
- **Modeling welded tube connections** Ability to define welded truss connections in the model for rectangular hollow sections, which can then be sent for analysis in Steel Connections design module.
- **Deflection in value in the shape sheets and diagrams** When presenting the results of verification of deflection of steel elements in graphical form and in reports, deflection values can now be presented both in ratio form and as values in a unit of displacement (e.g. cm).

Enhancing the analysis of timber structures acc. Eurocode 5

- **Design of double rectangular sections acc. EC5** Possibility of verification of timber beams and columns from compound profiles (consisting of two, three, or four rectangular section branches) according to EN 1995-1-1.
- Verification of bearing pressure on the support acc. EC5 Possibility of performing additional verification for timber elements the check of the compression perpendicular to the grain according to support sizes, according to EN 1995-1-1 (6.1.5).

- Verification of beams with a notch at the support acc. EC5 Possibility of performing additional verification for timber elements the check of the notch on an element ends, according to EN 1995-1-1 (6.5).
- **Optimization of timber elements according to the deflection (EC5)** Ability to automatically select the optimal timber section due to the deflection condition according to EN 1995-1-1.

Enhanced user experience and the comfort of program operation

- User templates for Selection The ability to save and easily select templates with object selection. The templates can contain both the current selection and the selection criteria.
- Selecting identical elements A new option that allows you to quickly select objects that are identical to a previously selected one, with easy consideration of the range of properties being compared.
- **Possibility for filtering non-active elements** A new filter in the selection by criteria window allows the selection of elements excluded from the creation of the analytical model.
- **Filtering profiles from databases** Easily search for profiles from the library by filtering content by profile type or text from a name, as well as by separating cold-formed profiles.
- **Commands on the ribbon for adding stages** A set of icons on the ribbon related to adding new construction stages and assigning elements to them.
- **Improved mesh generation for Pushover analysis** When generating a calculation model with elements with defined plastic releases for pushover analysis, the finite element mesh at the ends of the elements is now regular, which significantly improves the mesh distribution in adjacent surface elements.
- The next stage of unification of dialog windows Another set of dialog windows has been updated, giving them a unified appearance and components (Concrete settings, Steel settings, Timber settings, Design templates, Element selection, NL setting, Pushover settings)
- Improvements to the selection window by criterion Several changes have been made to the Element selection window that, among other things, make it easier to choose a selection method.

Other novelties and small improvements

- New database with cold-formed steel profiles for North America A new library of cold-formed steel profiles typical for the North American market.
- New steel material database for North American cold-formed sections A new library of materials for cold-formed steel profiles according to the ASTM standard.
- **Spaced shafts parametric section** Possibility to define parametrically profiles composed of two, three, or four identical rectangular shafts.
- Improved combination creation with seismic cases for Canada During the automatic generation of combinations with seismic cases according to the NBC standard, seismic combinations are now considering both directions (EX+, EX-, EY+, EY-).
- **Defining Link at node on selection** An additional way of quickly defining multiple links to node at the same time has been introduced, allowing to point to primary and secondary elements independently.



- **Possibility of material definition for supports** The option to assign a material (concrete) to supports has been introduced. This makes it possible to both include foundations in the calculation of costs and CO2 emissions, as well as to send the concrete class to RC Footing module.
- **Dedicated system for master-slave links** Master-slave links automatically generated (where the slab connects to the column or support) are now automatically placed in a dedicated system.
- Improvements to the program defaults for North America Better customization of some of the program defaults when selecting a localization for the US or Canada.
- Improvements in localization for Spain and Portugal The translations for Spanish and Portuguese languages have been improved and expanded on missing areas (like the Section editor).

Common improvements to all Design modules

- **Applying project templates when the module is already opened** Ability to update settings from a template for a currently open project.
- Display the steel ratio in the Info Panel Quickly available information on the steel ratio.
- Small improvements Set of various small enhancements.

RC Beam design module

- Import geometry and internal forces from Rib design Consideration of effective section and design internal forces when importing beams with rib option active.
- **Possibility to define the position of supports (columns/ walls)** Column supports now distinguish whether they support a beam from below, above, or from both sides. This data is now imported from the Advance Design model and affects the drawings.
- **New support type "beam"** New support type enables the correct generation of drawings when the support is another beam.
- **Ability to define secondary beams** Possibility for generating better drawings by considering the position of the secondary beams approaching perpendicularly.
- **New diagrams for reinforcement for torsion** The ability to display two new diagrams the longitudinal reinforcement from torsion and transversal reinforcement for torsion.
- **A new chapter in the report on openings reinforcement** In the detailed report for beams, if there are openings in the beam, there is now a new chapter on the reinforcement of openings.
- Small improvements
 - $\circ~$ A better description of extreme dimensions in the drawing
 - o Improvements to Reinforcement assumption dialog
 - o Modification of the default Wmax value for France
 - o Adjustment of waterproofing data entry for France
 - The additional coefficient for modification of the theoretical area of the longitudinal reinforcement



- Possibility to choose the way of distribution of reinforcement for torsion.
- o Improvement in defining suspended loads with link distribution.
- $_{\odot}$ $\,$ Support conditions are added to the Design Assumptions dialog.

RC Column design module

- **Different upper beams geometry** Possibility to individually specify the length and width of the top beams and to use that data for automatic determination of buckling lengths.
- **Possibility to disable the interaction curves checks** A new option makes it possible to skip checking the interaction curves.
- **Minor improvements to ribbons** Separation of icons for defining geometry and specifying section type.
- **Minor improvements in Info Panel** Adding information in Info Panel on whether second-order effects have been included.
- Improvement to transverse reinforcement on Edit dialog Quickly and precisely define and edit the distribution of transverse bars in the table with new definition methods.

RC Footing design module

- **Representation of the substitute footing in viewports** A visual representation of the substitute footing when this analysis was activated.
- Additional information on concrete cracking in Info Panel and report Greater control over calculations with additional concrete cracking information.
- Auto correction of reinforcement when crack widths are too large A new option allows you to keep the cracking within the limit by automatically increasing the reinforcement.
- **Improved editing of the distribution of bars** Changes in the Edit reinforcement dialog to facilitate the definition and modification of bar distributions on a pad.
- Information in the report about the final reinforcement area Additional information in the report to better describe the final reinforcement area.
- Improvements to seismic bearing capacity calculations for France Possibility to select the method for Nmax calculation: according to EN1998-5 or according to NF P94-261
- A new method of determining soil pressure A new algorithm for determining active and passive earth pressure, following annex C of EN 1997-1.

RC Wall design module

- Generation of drawing with schedules for bars and fabrics New drawing style added to allow automatic simultaneous generation of schedules for bars and fabrics.
- Import and export forces using Excel file For Shear walls, it is possible to import and export resultant forces to an excel sheet.

• Masonry Wall design module

- Loads representation in viewports To make it easier to view defined loads as well as imported internal forces, it is now possible to display loads and internal forces graphically.
- **Expanded report for RC6 calculations** A new section has been added in the reports for verification of a wall subjected to in-plane bending according to the Romanian CR6 standard.
- **Possibility to choose the section for the stress diagram** When displaying the results in the form of stress diagrams, it is now possible to select the position of the section (bottom/middle/top)
- Saving the masonry databases data per model To facilitate the management of data from databases used by the Masonry Walls module, databases have been separated from the data of the current project. This allows convenient management of data used in each project.
- **Presentation of partial factors** To make it possible to verify the determined partial factors for the current masonry section, they are now available in the Design Assumptions window.
- **Support depth representation in viewports** Displaying the depth of the slab support on viewers to easily check the data.

RC Slab design module

- New bar shapes Possibility to generate u-shaped and two bent bars on the edges of the slab.
- **Possibility to manage hooks for reinforcement zones** Possibility for editing hooks separately for each reinforcement zone.
- **New options for defining reinforcement solution** Possibility to define reinforcement coverage with reinforcement areas.
- New methods for defining reinforcement zone graphically Faster modeling of reinforcement zones with additional graphic definition methods.
- Set of improvements to drawings
 - Possibility of drawing generation simultaneously or separately for each reinforcement direction
 - Improvements to bar annotations (better automatic generation to avoid collisions, the possibility to move individually the bar annotation, and the possibility to hide the text above the dimension line)
 - New drawing style for simultaneous generation of schedules for bars and fabrics
 - Better representation of openings
 - New option to hide grids in drawings.
- Set of user improvements
 - Allow the second layer to have a bigger diameter than the first one.
 - o Displaying local axes for each slab
 - Possibility for disabling reinforcement layer by entering 0 coverage.
 - New option to generate one-way reinforcement zones.

Steel Connections design module

- Welded truss tube connection New welded truss connection type with multiple diagonal configurations of square and rectangular tubes.
- **Improvement on the Info Panel for Shear plate joint** A new column on the Info panel for the indication of the element in which the verification is made.
- New option to ignore bending moment in the calculation for hinged beams More control over the type of forces used for vérifications.

3. New computing capabilities

A series of new features and improvements related to structural calculations and additional building analysis.

3.1. Carbon emission estimations

Calculation of CO2 emissions based on carbon factors entered for materials or elements, with results in graphical form and reports.

Following its engagement to build a sustainable future, Advance Design is now equipped with a new CO₂ emission calculator. The main purpose of this new tool is to facilitate the determination of embodied carbon values for structural elements. This new tool will help designers to better assess the environmental impact of the choices they make on structural elements and construction materials.

The fundamental principle of an **embodied carbon calculation** is typically to multiply the quantity of each material by a carbon factor (normally measured in kgCO₂e per kg of material). As the embodied carbon associated with the production stage is the largest contributor to the embodied carbon of a structure, **production stage carbon factors** (carbon factors for the production stage – modules A1–A3), are used for calculations.

NOTE: The mechanism for defining the data, the calculation, and the presentation of the results are identical to Cost estimations (more info about the Cost estimations you can find in the next paragraph).

Defining carbon factors

The general procedure for defining data for CO₂ estimation is to prepare a list of data sets with specific carbon factors, and then assign them to individual materials or optionally to structural elements.

Preparing the list of data sets is done with the help of a dedicated Carbon Factor Manager. It can be opened from the *Materials and Sections* group in the *Manage* ribbon.



The Carbon Factor Manager window is divided into two parts. The upper part contains data sets added and used in the project, while the lower part shows the entries available in the selected database.

Data Set Name	Weight	Volume	Area (Planar Elements)	Reinforcement
Concrete	0.119 kgCO2e/kg	0.000 kgCO2e/m3	0.000 kgCO2e/m²	0.000 kgCO2e/kg
Concrete + Reinforce	0.119 kgCO2e/kg	0.000 kgCO2e/m3	0.000 kgCO2e/m ²	1.990 kgCO2e/kg
Steel (Europe)	1.130 kgCO2e/kg	0.000 kgCO2e/m3	0.000 kgCO2e/m ²	0.000 kgCO2e/kg
oon factors	/ Add	× Delete	× Purge	► Export
abase Ibase Path		ign/2024/Resources/Cat	talogs (Carbon Factors \De	faultCarbonFactors.x
pase base Path	Waidd	ign\2024\Resources\Cat	talogs\Carbon Factors\De	faultCarbonFactors.x
base Path Data Set Name Concrete	Weight 0.119 koCO2e 4ko	ign)2024/Resources/Cat Volume	alogs (Carbon Factors (De Area (Planar Elements)	faultCarbonFactors.x
oase base Path Data Set Name Concrete Concrete + Reinforce	Weight 0.119 kgC02e/kg 0.119 kgC02e/kg	Volume 0.000 kgC02e/m3 0.000 kgC02e/m3	alogs (Carbon Factors (De Area (Planar Elements) 0.000 kgCO2e/m ² 0.000 kgCO2e/m ²	FaultCarbonFactors.x
oase base Path Data Set Name Concrete Concrete + Reinforce Steel (Europe)	Weight 0.119 kgC02e/kg 0.119 kgC02e/kg 1.130 kgC02e/kg	Ign\2024\Resources\Cat Volume 0.000 kgC02e/m3 0.000 kgC02e/m3 0.000 kgC02e/m3	Area (Flanar Elements) 0.000 kgCO2e/m ² 0.000 kgCO2e/m ²	FaultCarbonFactors.x Reinforcement 0.000 kgCO2e/kg 1.990 kgCO2e/kg 0.000 kgCO2e/kg
pase base Path Data Set Name Concrete Concrete + Reinforce Steel (Europe) Day brick + Gypsum	Weight 0.119 kgC02e/kg 0.119 kgC02e/kg 1.130 kgC02e/kg 0.213 kgC02e/kg	Ign 2024 Resources\Cat Volume 0.000 kgCO2e/m3 0.000 kgCO2e/m3 0.000 kgCO2e/m3 0.000 kgCO2e/m3	Area (Flanar Elements) 0.000 kgCO2e/m² 0.000 kgCO2e/m² 0.000 kgCO2e/m² 1.320 kgCO2e/m²	FaultCarbonFactors.x Reinforcement 0.000 kgC02e/kg 1.990 kgC02e/kg 0.000 kgC02e/kg 0.000 kgC02e/kg
base Path Data Set Name Concrete Concrete + Reinforce Steel (Europe) Clay brick + Gypsum Lightweight concrete	Weight 0.119 kgCO2e/kg 0.119 kgCO2e/kg 1.130 kgCO2e/kg 0.213 kgCO2e/kg 0.280 kgCO2e/kg	Ign 2024 Resources\Cat Volume 0.000 kgCO2e/m3 0.000 kgCO2e/m3 0.000 kgCO2e/m3 0.000 kgCO2e/m3 0.000 kgCO2e/m3	Area (Flanar Elements) 0.000 kgC02e/m ² 0.000 kgC02e/m ² 0.000 kgC02e/m ² 1.320 kgC02e/m ² 0.000 kgC02e/m ²	FaultCarbonFactors.x Reinforcement 0.000 kgC02e/kg 0.000 kgC02e/kg 0.000 kgC02e/kg 0.000 kgC02e/kg 0.000 kgC02e/kg
base Path Data Set Name Concrete + Reinforce Steel (Europe) Clay brick + Gypsum Lightweight concrete Timber - no carbon st Timber - no carbon st	Weight 0.119 kgCO2e/kg 0.119 kgCO2e/kg 1.130 kgCO2e/kg 0.213 kgCO2e/kg 0.280 kgCO2e/kg 0.493 kgCO2e/kg	Ign 2024 Resources\Cat Volume 0.000 kgCO2e/m3 0.000 kgCO2e/m3 0.000 kgCO2e/m3 0.000 kgCO2e/m3 0.000 kgCO2e/m3 0.000 kgCO2e/m3	Area (Planar Elements) 0.000 kgC02e/m ² 0.000 kgC02e/m ² 0.000 kgC02e/m ² 1.320 kgC02e/m ² 0.000 kgC02e/m ² 0.000 kgC02e/m ²	FaultCarbonFactors.x Reinforcement 0.000 kgC02e/kg 0.000 kgC02e/kg 0.000 kgC02e/kg 0.000 kgC02e/kg 0.000 kgC02e/kg 0.000 kgC02e/kg
ioase Path Data Set Name Concrete Concrete + Reinforce Steel (Europe) Clay brick + Gypeum Lightweight concrete Timber - including car Timber - including car	Weight 0.119 kgC02e/kg 0.119 kgC02e/kg 0.213 kgC02e/kg 0.230 kgC02e/kg 0.493 kgC02e/kg -1.030 kgC02e/kg	Ign (2024) Resources (Cat Ign (2024) Resources (Cat Volume 0.000 kgCO2e/m3 0.000 kgCO2e/m3 0.000 kgCO2e/m3 0.000 kgCO2e/m3 0.000 kgCO2e/m3 0.000 kgCO2e/m3	Area (Flanar Elements) 0.000 kgC02e/m² 0.000 kgC02e/m² 0.000 kgC02e/m² 1.320 kgC02e/m² 0.000 kgC02e/m²	FaultCarbonFactors.x Perinforcement 0.000 kgC02e/kg 0.000 kgC02e/kg 0.000 kgC02e/kg 0.000 kgC02e/kg 0.000 kgC02e/kg 0.000 kgC02e/kg 0.000 kgC02e/kg
pase base Path Concrete Concrete + Reinforce Steel (Europe) Cay brick + Gypsum Lightweight concrete Timber - no carbon st Timber - including car Foundation	Weight 0.119 kgCO2e/kg 0.119 kgCO2e/kg 1.130 kgCO2e/kg 0.213 kgCO2e/kg 0.280 kgCO2e/kg 0.493 kgCO2e/kg 0.000 kgCO2e/kg	Volume 0.000 kgC02e/m3 0.000 kgC02e/m3	Area (Planar Elements) 0.000 kgC02e/m² 0.000 kgC02e/m² 0.000 kgC02e/m² 0.000 kgC02e/m² 1.320 kgC02e/m² 0.000 kgC02e/m² 0.000 kgC02e/m² 0.000 kgC02e/m² 0.000 kgC02e/m² 0.000 kgC02e/m² 0.000 kgC02e/m² 0.000 kgC02e/m²	FaultCarbonFactors.x Reinforcement 0.000 kgCO2e/kg 0.000 kgCO2e/kg 0.000 kgCO2e/kg 0.000 kgCO2e/kg 0.000 kgCO2e/kg 0.000 kgCO2e/kg 0.000 kgCO2e/kg 0.000 kgCO2e/kg
base Path Data Set Name Concrete Concrete + Reinforce Steel (Europe) Clay brick + Gypsum Lightweight concrete Timber - no carbon st Timber - including car Foundation	Weight 0.119 kgC02e/kg 0.119 kgC02e/kg 1.130 kgC02e/kg 0.213 kgC02e/kg 0.280 kgC02e/kg 0.493 kgC02e/kg -1.030 kgC02e/kg 0.000 kgC02e/kg	Ign (2024) Resources (Cat Ign (2024) Resources (Cat Volume 0.000 kgCO2e/m3 0.000 kgCO2e/m3 0.000 kgCO2e/m3 0.000 kgCO2e/m3 0.000 kgCO2e/m3 297.500 kgCO2e/m3	Area (Flanar Elements) 0.000 kgC02e/m²	FaultCarbonFactors.x Reinforcement 0 000 kgCO2e/kg 0 000 kgCO2e/kg 0 000 kgCO2e/kg 0 000 kgCO2e/kg 0 000 kgCO2e/kg 0 000 kgCO2e/kg 1 990 kgCO2e/kg

Carbon Factors Manger

In this top part, we can either import data from the database entry, or add a new data set using the *Add* button, and then we fill in the name of the data set, and CO₂ emission per unit of weight, volume, or area (for planar elements), and values corresponding to the weight of reinforcement.

Although in most cases we use carbon factors based on the weight of the material (such data is usually provided by manufacturers and other institutions), in special cases we can use factors based on volume or area. What is important, we can combine factors – for example, if for a wall or slab, we want to consider cladding elements, then in addition to the value according to weight we can add in the same position a value depending on the surface area. For concrete elements, we can simultaneously enter the carbon factor for reinforcement. This allows us to obtain a CO_2 value including the reinforcement, based on the quantity of reinforcement calculated during the concrete design analysis for that element.

In the bottom part of this manager, we can see data sets available on a selected library. Each user can independently prepare and manage the data content in the library.

NOTE: Advance Design provides a sample library with a list of examples of averaged values of the production stage carbon factors for selected materials. It is important to remember that the factors for many materials strongly depend on additional information, such as place and technology of production, use of recycling, type of ingredients (e.g. cement), etc. Therefore, for accurate estimation, adequate data provided by manufacturers should be used. Also note that for some localizations, the country's local regulations may dictate the use of other official data.

Assigning carbon factors

There are two methods of assigning carbon factors: by assigning to materials or by assigning to elements (linear/planar/support/load area).

When Carbon factors are assigned to **Materials** (for example concrete C25/30) then such values are used for all objects from this material.

For this, the materials manager now includes a new Carbon Emission section where we can assign a specific CO₂ data set for each material by selecting from a drop-down list of available data sets.



Assigning a CO2 emission data set to material

By default, each structural element inherits its material CO₂ emission. However, we can override this and impose another data set for selected elements. Assigning data sets with carbon factors to selected elements is done using new dedicated options available on the list of element properties.

Interestingly, it is also possible to determine carbon factors for foundations, although they are not separately defined or presented in the model. For this purpose, the parameters available in the properties of supports are used, including foundation dimensions and material. Note that in case of foundations, the reinforcement ratio that could be used for estimations needs to be set manually, while for other elements (beams, columns, slabs) it is calculated automatically during concrete design calculation analysis.

		Footing Material	
Releases		- Material	C25/30
- E Total releases		Footing Dimensions	
- 🗄 Elastic releases		- Width A	1.00 m
Plastic hinges		- Width B	1.00 m
Cost and CO+ calculations		- Height H	0.30 m
- Cost estimations		E Punching	
		Design Module Temple	ate
 Settings by material 	Disabled	□ Cost and CO _± calculat	tions
- Carbon factors		Cost estimations	
	B Concrete + Neinforcement	⊂⊟ CO₂ emisson	
	None	 Settings by materia 	al 🗖 Disabled
v gradient of avial stress Svy	0.00 MPa	Carbon factors	Foundations
y gradient of axial stress SXX	0.00 MPa/m	Reinforcement ratio	0.02

Overwriting default carbon factors of a linear element (left) and a support (right)

Results

The results of the calculations are available in graphical form and in the form of report tables.

For the graphical results, on the *Results* ribbon, a new section for Cost and Carbon emission is used.



We can display the CO₂ emission results of point and linear supports, linear and planar elements, and load areas. The results values can be displayed per element, per unit of length (for linear elements), per unit of area (for planar elements), per unit of volume, and unit of weight.



CO₂ emission of linear elements displayed per element



CO2 emission of planar elements displayed per element

For tables and reports, Advance Design can generate CO₂ emission tables. The tabulated results can be presented by material, by element type, and by system.

		CO ₂ emiss	sion estimation for	materials		
Material	Emission/weig ht (kgCO ₂ e/kg)	Emission/volu me (kgCO ₂ e/m²)	Emission/area (kgCO ₂ e/m²)	Volume (m ³)	Weight (kg)	CO ₂ emission (kgCO ₂ e)
C25/30	0.112	0.000	0.000	283.00	707500	79240.000
C25/30	0.000	297.500	0.000	16.20	40500	9784.476
(Foundations)						
C30/37	0.119	0.000	0.000	904.20	2260500	268999.500
D27	-1.290	0.000	0.000	3.79	2313	-2984.373
S235	1.500	0.000	0.000	2.77	21747	32620.141
S355	1.130	0.000	0.000	10.20	80070	90479.100
			Total	1220.16	3112630	478138.844

Report table of CO2 emission estimations by material

	CO ₂ emission estima	tion by element type	
Туре	Volume (m ³)	Weight (kg)	CO ₂ emission (kgCO ₂ e)
Linear elements	191.96	542130	169170.868
Planar elements	992.20	2480500	293639.500
Foundations	36.00	90000	15328.476
Total	1220.16	3112630	478138.844

Report table of CO₂ emission estimations by element type

	CO ₂ emission estimation by system	
System No.	System name	CO ₂ emission (kgCO ₂ e)
5	L1 - Columns	22107.539
6	L1 - Beams	9172.800
7	L1 - Slabs	168682.500
8	L2 - Columns	11424.000
9	L2 - Beams	30610.187
11	L3 - Columns	1400.000
12	L3 - Steel part	90479.100
13	L3 - Slabs	16065.000
14	L4 - Timber beams	-1967.250
15	L4 - Steel part	6961.615
16	L4 - Slabs	52836.000
17	Supports	15328.476
18	RC Walls	56056.000
19	L4 - Roof	-1017.123
20	Windwalls	0.000
	Total	478138.844

Report table of CO2 emission estimations by system

3.2. Cost estimations

Calculation of costs based on unit prices for materials or elements, with results in graphical form and reports.

Advance Design is now equipped with a new cost calculator. This new tool will help designers to better assess the economic impact of the choices they make on structural elements and construction materials.

The principle of **cost estimations** is simple – we need to multiply the quantity of each material by a unit price. To do this, we need to define specific unit costs, and then assign them to individual materials or optionally to structural elements. Results of calculations in the form of report tables are available both in the descriptive and analytical models, while results in the form of graphs are available in the analytical model. Please note that the values from the reinforcement of concrete elements are only available if we have performed a reinforcement analysis on these elements.

NOTE: The mechanism for defining the data, the calculation method, and the presentation of the results are identical to Carbon emission estimations (more info about the CO₂ estimations you can find in the previous paragraph).

Possibility to set a currency unit

To be able to correctly set the currency used for cost estimations, a new unit-type, Currency is now available in the Unit Settings window.

Reinforcement Area	Millimeter ²	0.00
Translation Stiffness	kN/m	0.00
Rotational stiffness	kN*m∕°	0.00
Currency	Euro	 ✓ 0
	Reinforcement Area Translation Stiffness Rotational stiffness Currency	Reinforcement Area Millimeter* Translation Stiffness kN/m Rotational stiffness kN'm/* Currency Euro

Defining unit prices

Preparing the list of data sets with unit prices is done with the help of a dedicated Unit Prices Manager. It can be opened from the Materials *and Sections* group in the *Manage* ribbon.



- Q	Home	Objects	Utilities	Analysis	Result	Docu	ments	BIM	Manage	De	sign Mo	dules		
Application	Pro Co Str	iject Settings nfiguration ucture	Mesh <u>Q1</u> Units	REBAR Calco	ulator ses	Used Cross Sections	Used Materials	Used Soils	Steel Dec Profiles	S € Unit Costs	CO2 arbon actors	Section Editor	Display/Hide the Project Browser	Properties
		Set	tings					Materia	Is and Secti	ons	•		Comp	ponents

The Unit Prices Manager window is divided into two parts. The upper part contains data sets added and used in the project, while the lower part shows the entries available in the selected database.

Data Set Name	Weight	Volume	Area (Planar Elements)	Reinforcement
Concrete	1500 €/T	0.€/m3	0 £/m ²	0€/T
Steel	2000 €/T	0 €/m3	0 €/m ²	0€/T
Timber	0 €/T	500 €/m3	0 €/m²	0€/T
	A A	dd X De	lete X Purge	► Export
vrary atabase atabase Path		DefaultUnitPrices.x	ml Resources \Catalogs \Unit Price	s\DefaultUnitPrices.xm
Data Set Name	Weight	Volume	Area (Planar Elements)	Reinforcement
New Unit Price(1)	0€/T	0 €/m3	0 €/m²	0 €/T

Unit Prices Manger

In this top part, we can either import data from the database entry or add a new data set using the *Add* button. When we add a new data set, we need to fill in the name, and the price per unit of weight, volume, or area (for planar elements), and optionally a value corresponding to the weight of reinforcement.

We can combine unit prices - for example, if for a wall we want to consider the cost of cladding elements, then in addition to the price per weight we can add in the same set a value depending on the surface area. For concrete elements, we can simultaneously enter the unit price for reinforcement, which allows us to obtain the cost of the reinforcement, based on the quantity of reinforcement calculated during the concrete design analysis.

In the bottom part of this manager, we can see data sets available in a selected library. Each user can prepare and manage the data content in the library. Note that by default, the library with unit costs does not contain any values.



G GRAITEC

Assigning unit prices

There are two methods of assigning unit prices: by assigning to materials or by assigning to elements (linear/planar/support/load area).

When unit prices are assigned to Materials then such values are used for all objects from this material.

For this, the materials manager now includes a new Cost Data section where we can assign a specific data set for each material by selecting from a drop-down list of available data sets.

	Designation	Family	Standard	Туре	Color
Þ	S235	STEEL	EN 10025-2	S235	8c0
	S355	STEEL	EN 10025-2	S355	e60
	C25/30	CONCRETE	EN206	C25/30	6c9
	C30/37	CONCRETE	EN206	C30/37	8da
	D27	TIMBER	EN 338 - 2016 (Har	D27	000
	GL22c	TIMBER	EN 14080 - 2013 (GL22c	000
	C24	TIMBER	EN 338 - 2016 (Soft	. C24	000
м	lechanical Properties >>	Prope	Add erties EN 10025-2 >>		ibraries >>
M	lechanical Properties >>	Prope	Add arties EN 10025-2 >>		ibraries >>
М	lechanical Properties >> Cost Data <<	Prope	Add erties EN 10025-2 >> arbon Emission >>		ibraries >>
M	Nechanical Properties >> Cost Data <<	Prope	Add erties EN 10025-2 >> rbon Emission >>	Delete	ibraries >>
M	Iechanical Properties >> Cost Data << st Data Data Set Name We	Prope Ca	Add erties EN 10025-2 >> rbon Emission >> Volume Ar	Delete	ibraries >> Reinforcement
M	t Data Set Name We Steel 7200	Prope Ca ight 0€/T	Add erties EN 10025-2 >> rbon Emission >> Volume Ar 0 €/m3 0 0	ea (Planar El E/m ²	Purge ibraries >> Reinforcement 0 €/T
M	Iechanical Properties >> Cost Data << t Data Data Set Name We Steel 200 Concrete	Prope Ca ight 0€/T	Add erties EN 10025-2 >> rbon Emission >> Volume Ar 0 €/m3 0 +	ea (Planar El	Purge ibraries >> Reinforcement 0 €/T
M	Iechanical Properties >> Cost Data << tt Data Data Set Name Steel Steel Steel Steel	Prope Ca ight 0€/T	Add erties EN 10025-2 >> rbon Emission >> Volume Ar 0 €/m3 0 €	ea (Planar El £/m²	Purge ibraries >> Reinforcement 0 €/T
M	Iechanical Properties >> Cost Data << at Data Data Set Name Steel Concrete Steel Timber	Prope Ca ight 0€/T	Add erties EN 10025-2 >> rbon Emission >> Volume Ar 0 €/m3 0 €	ea (Planar E £/m ²	Purge ibraries >> Reinforcement 0 €/T

Assigning a cost data set to a selected material

By default, each structural element inherits its material unit price. However, we can override this and impose other data sets for selected elements. Assigning data sets of unit prices to selected elements is done using new dedicated options available on the list of element properties. As it is available for the Carbon estimations, it is also possible to determine the cost for foundations, using the parameters available in the properties of support.

Releases						
Cost and CO ₂ calculations						
Cost estimations						
 Settings by material 	Disabled					
Unit costs	🗾 Rafters	~				
E CO= emisson						
El Initial constraint						
F Mesh						

Overwriting default unit prices on the property list of a linear element

Results

The results of the calculations are available in graphical form and in the form of report tables. For the graphical results, on the *Results* ribbon, a new section for Cost and Carbon emission is used.

- []-	Home	Objects	Utilitie	s Analysis	Г	Results	Docur	nents	BIM	Manag	e De	sion Modules							
	同~	을 Value on dia;	rans	100	٠	Displacer	ients	- 4	None	-	N	CO ₂ emission	· >	14	-	👍 Select by Criteria	- 48	- 🚮 🧯	iii ii
		- Q		1000 J		10		· /	D	*		Point Support	- / 5			Select by	- 🛋	100.1	n i
View * :	Settings	0	_	Calculation*	R			10	D	° P	poessing	CO ₂ emission of ele	emission	Wesurt Jurves *	Configuration *	None	-	Hiter	8
	Set	ttings						PEM Re	suffs			Cost & CO ₂ emis	sion results	Pn	# Processing	Selection		Isolate / Di	splay

Similarly, as for carbon estimations, we can display the cost estimation results on point and linear supports, linear and planar elements, and load areas. The results values can be displayed per element, per unit of length (for linear elements), per unit of area (for planar elements), per unit of volume, and unit of weight.



Cost of planar elements displayed per surface unit

For tables and reports, Advance Design can generate cost estimation tables. The tabulated results can be presented by material, by element type, and by system.

	Cost estimation for materials							
Material	Cost/weight	Cost/volume	Cost/area	Volume (m ³)	Weight (kg)	Cost (€)		
	(€/kg)	(€/m³)	(€/m²)					
C25/30	0.15	0.00	0.00	299.20	748000	114621.90		
C30/37	0.17	0.00	0.00	904.20	2260500	384285.00		
D27	0.24	0.00	0.00	3.79	2313	555.23		
S235	1.30	0.00	0.00	2.77	21747	28270.79		
S355	1.45	0.00	0.00	10.20	80070	116101.50		
			Total	1220.16	3112630	643834.42		

Report table of cost estimations by material

Cost estimation by element type									
Туре	Volume (m ³)	Weight (kg)	Cost (€)						
Linear elements	191.96	542130	210627.52						
Planar elements	992.20	2480500	417285.00						
Foundations	36.00	90000	15921.90						
Total	1220.16	3112630	643834.42						

Report table of cost estimations by element type



	Cost estimation by system	
System No.	System name	Cost (€)
5	L1 - Columns	25660.08
6	L1 - Beams	12285.00
7	L1 - Slabs	240975.00
8	L2 - Columns	15300.00
9	L2 - Beams	32817.31
11	L3 - Columns	1875.00
12	L3 - Steel part	116101.50
13	L3 - Slabs	22950.00
14	L4 - Timber beams	366.00
15	L4 - Steel part	6033.40
16	L4 - Slabs	75480.00
17	Supports	15921.90
18	RC Walls	77880.00
19	L4 - Roof	189.23
20	Windwalls	0.00
	Total	643834.42

Report table of cost estimations by system

3.3. Displacement control nonlinear analysis

The ability to conduct nonlinear calculations using displacement increment steps, which allows to correctly analyze highly nonlinear problems with the post-peak behaviors and can easily manage the snap-through problems.

Technical background

Until present, Advance Design conducted nonlinear analysis by using a force control Newton Raphson approach. This numerical method is the classical and most used technique in nonlinear analysis. It consists of dividing the loads applied on the structure into equal force increments. Then, at each step of the analysis, one additional force increment is added, and the structure is analyzed nonlinearly. Hence the name force control.



(F - Load, D - Displacement, f - Equal force increments)

The force control analysis has limitations. Since the applied loads on the structure are equally increased with every step, this technique cannot be used for structures that have a load drop (due to buckling, strength loss, or support failure ...). The figure below presents the nonlinear force Vs displacement curve of a structure with load drop. Since it must always keep increasing the load, the

force control solver exhibits a snap-through behavior and cannot capture the sagging part of the structural response (dashed line in the figure below).



To face snap-through problems in nonlinear analysis, Advance Design is now equipped with a new displacement control solver. With this new approach, loads are applied on the structure and a target displacement is set for a chosen control node on the structure. This target displacement is divided into equal displacement increments. For each analysis step, the nonlinear solver will try to scale up/down all the loads applied on the structure to achieve the required displacement increment at the chosen control node (hence the name displacement control, refer to the next Figure). At each step, all the loads are scaled proportionally. In other words, the same scale factor is applied on all the loads. Thus, the initial load pattern distribution is always respected.



Displacement control solver capturing the entire structural response (F - Load, D - Displacement, d - Equal displacement increments)

It is particularly important to note that displacement control analysis is different from imposed displacement loads. For an imposed displacement load on a node, a point load is automatically created on the concerned node so that it reaches the required imposed displacement. While for displacement control analysis, all the loads on the structure are proportionally scaled up/down to satisfy the target displacement of the control node (no automatic point load is created on the control node).

Now by controlling the nonlinear analysis with displacement increments, the loads on the structure can be increased or decreased without affecting the calculation procedure. Thus, structures with load drop can be completely studied using the displacement control nonlinear solver.

How to use displacement control analysis in Advance Design

Setting up a displacement control nonlinear analysis in Advance Design is similar to creating the already existing force control Newton Raphson analysis. The user should conduct the following steps:

- 1. Place a point on the structure to serve as an indicator of the displacement control node position.
- 2. Create a nonlinear static analysis.
- 3. In the static nonlinear load case, the set calculation is controlled by displacement increments.

Properties			×
🗐 💽 🚡 All properties			*
Static non linear load case			
— Title			
— Name	NL		
— No.	3		
Code			
Settings			
 Calculation controlled by 	Displacements increments		•
- Reference	Load increments		
 Large displacements 	Displacements increments		
Combinations			
- Effect	Leading		
- Coefficient γ EQU	1.00		

Selecting displacement control nonlinear analysis

4. In nonlinear analysis options, add the loads/combinations to be considered in the nonlinear.

1 D (ECG)		102 1.35x[1 D]+1.5x[2 L] (ECELUSTR) : 1	
2 L (ECQ) 101 1.35x[1 D] (ECELUSTR) 102 1.35x[1 D]+1.5x[2 L] (ECELUSTR) 103 1x[1 D] (ECELSCQ) 104 1x[1 D]+1x[2 L] (ECELSCQ) 105 1x[1 D] (ECELSOP)	Add >> Remove <<		
Coefficient 1			

Selecting Load cases/combinations for nonlinear analysis

5. For each selected load case/combination set the parameters of the nonlinear analysis. The new analysis parameters for displacement control concern the displacement control node. The user indicates the control node number (ID number of point created in step 1), specifies on which global degree of freedom the controlled displacement/rotation is occurring, and then sets the target value of the corresponding displacement/rotation.

						NOT F	OR RESALE versi	ion - Non Linear A	nalysis Options						1	3
	Load case or	combination		Incrementation		Save results	D)isplacement control	node			Conver	rgence			1
	Identifier - name	Coefficient	Number of steps	Iterations by step	Stabilize iteratio	Frequency / step	Point	Degree of free	Displacement	Ener	σy	For	00	Displa	cement	
1	102 1.35x[1 D]+	1.00	10	50	20	last	1	DX	• 0.000 cm	V	1.00e-08 J	V	1.0e-06 kN	1	1.000e-01 cm	
								DX	-							
								DZ								
								RX								
								RZ								
										1						
									Select analysis							
	Add/Remove ana	lyses	Apply to se	elected items	Coefficient		0		All	None						
														ок	Cancel	
															001100	

Nonlinear analysis parameters for displacement control

• Application example 1: Von Mises truss snap-through buckling

To start let us consider a textbook example of a simple structure made with 2 bars and a vertical load applied at their connecting node. This structure is called the Von Mises truss.



Von Mises truss model and its progressive deformation

The structure will deform progressively and a snap through buckling will occur as indicated in the figure above. An analytical solution (neglecting dynamic effects) exists for the Von Mises truss and can be used as a benchmark for Advance Design results.

For this purpose, the example was recalculated in Advance Design using both available methods of nonlinear calculations, and the following graph shows the resulting force Vs node displacement curve. The **red** dashed line presents the curve from the nonlinear **force control** analysis with large displacement. The **blue** line presents the curve from nonlinear **displacement control** analysis with large displacement, and it is identical to the curve from the analytical solution.



Force control analysis curve (dashed red) and Displacement control analysis curve (blue)

It is clear that the force control solver had a snap-through problem and could not calculate the sagging part of the force-displacement curve, while the displacement control analysis was capable of managing the snap-through buckling and captured accurately the entire structural response.

Application example 2: Support failure

Let us consider a uniformly loaded beam with fixed support (point A) and 2 vertical nonlinear spring supports (at points B and C). Each spring behaves linearly until a certain bearing capacity limit. Once this limit is reached, the spring will fail and drop all its support reactions.



Beam on nonlinear springs (points B and C)

Using Advance Design, we want to conduct a step-by-step nonlinear analysis to study the progressive failure of springs and its effect on the structure. Both springs have the same linear stiffness. However, Spring at point C has double the bearing capacity. Thus, it is expected that spring at point B will fail first then spring at point C will follow later. That is, during successive iterations of nonlinear calculations, we expect to get 3 different systems: the first in which all supports work, the second in which the middle support has stopped working, and finally the third in which the right elastic support has also stopped working.



System 1 - All supports are active



System 3 - Failure of the right support

The dynamic effect of spring support failure is neglected, and the midpoint of the beam (location of spring B) is considered for measuring vertical displacement.

The example was recalculated in Advance Design using both available methods of nonlinear calculations.

First, a nonlinear **force control** analysis is conducted in Advance Design. The figure below presents the curve of total force applied on the structure Vs measured displacement (the vertical deflection at the middle of the beam).



Force Vs displacement curve resulting from force control nonlinear analysis

It is remarkable that for the failure of springs B and C, the force control solver could not drop the load applied on the structure but instead, a snap-through behavior is observed.

Next, a nonlinear displacement control analysis is conducted in Advance Design. The figure below presents the curve of total force applied on the structure Vs measured displacement.





Force Vs displacement curve resulting from displacement control nonlinear analysis

During the failure of springs B and C, the displacement control solver was capable of dropping the load applied to the structure and avoiding snap-through problems.

Of course, this is just a specific example, but it shows that some projects can give different results in certain ranges, depending on the chosen method of nonlinear analysis. Thanks to the new solver, Advance Design can get correct analysis results for highly nonlinear problems with post-peak behaviors and can easily manage the snap-through problems.

3.4. Possibility for defining initial constraints on linear elements per load case

A new way of defining and considering initial constraints, making it easier to include initial constraints in combinations, as well as considering their impact on the rest of the structure.

In previous versions of Advance Design, initial constraints for line elements served to locally increase the element's internal forces/stresses for design calculation purposes. Since the main objective was to serve the design calculation of the element, this increase of internal forces/stresses affected only the element carrying them, and no interactions with other structural components were considered. In addition, initial constraints were defined as a global property of a given linear element, which caused problems if they were not to be included in all combinations.

With a new version, the influence of initial constraints for linear elements is studied on the entire structural system and it is considered during finite element analysis. For this, a linear element with initial constraints is trying to deform in such a way as to relieve this initial stress. Three scenarios are possible:

- The element is blocked and cannot deform In this case, initial constraints cannot be relieved and are exhibited as internal stresses in the element.
- The element is connected to other structural components In this case, the element will deform in a way to partially relieve the initial constraints. However, by doing so, the other structural components are getting stressed. An energy equilibrium is established between initial constraints relieve of the line element and stress increase in other structural components.
- The element is free to deform In this case, the element will freely deform in a way to relieve the initial constraints.



Starting with Advance Design 2024, initial constraints of linear elements are linked to load cases. Previously, they were defined as the properties of a linear element, but now we have the option of choosing which load case it belongs to.

Initial constraint	
 Load case 	2-L
 Uniform axial stress Sxx 	10.00 MPa
 y gradient of axial stress Sxx 	0.00 MPa/m
z gradient of axial stress Sxx	0.00 MPa/m

Defining initial constraints

Three types of initial constraints are available:

- Uniform axial stress Sxx Initial Axial stress distributed uniformly all over the section.
- y gradient of axial stress Sxx Initial axial stress variation per unit length of the section in the local y direction (this is similar to axial stress gradient generated by bending moment around local y direction).
- z gradient of axial stress Sxx Initial axial stress variation per unit length of the section in the local z direction (this is similar to axial stress gradient generated by bending moment around local z direction).

Let us look at the following example that will help you understand how initial constraints work.

Let us consider a simple beam truss assembly with pre-tensioned bars via turn buckles – it is a single-span horizontal beam with two pre-tensioned bars and a vertical bar.



As a result, part of the stresses from these tendons were distributed through the vertical element to the entire system, causing the beam to bend upward. Using other words - under the effect of initial constraints (pretension), the beam will have an initial upward deflection.



Beam deflection due to pretension

When gravity loads are applied to the beam, this beam will present a lower deflection and smaller bending moment compared to a system without pre-tensioned bars.

In the comparison below you can see the deflections and bending moments in the beam for two cases - with pre-tensioned bars (left) and without pre-tensioned bars (right).



Deflections (top) and bending moments (bottom) for example with pre-tensioned bars (left) and without pre-tensioned bars (right)

3.5. A better formulation of drilling rotation in shell elements

Improved definition of shell finite elements, thanks to shell elements are now capable of capturing torsional moments transmitted from perpendicular beam elements.

The drilling rotation of a shell node is the rotation degree of freedom around the local axis perpendicular to the shell finite element.



Advance Design 2024 uses a new and better formulation for calculating this drilling rotation. This change slightly improves the accuracy of calculations and has a positive effect on the convergence of nonlinear calculations; however, the biggest benefit is that shell elements are now capable of capturing torsional moments transmitted from perpendicular beam elements.



Example of beam transmitting moment to shell element in Advance Design 2024

4. Enhance steel structure design capabilities

A series of novelties and improvements related to the verification and optimization of steel element structures.

4.1. Verification of steel cold-formed sections according to AISI S100

The possibility of performing the standard verification of cold-formed steel profiles according to the AISI S100-16 American standard.

With Advance Design 2024, the possibility of standard verification for cold-formed profiles for North America has been introduced. The calculations are based on AISI S100-16 Specification (American Iron and Steel Institute - North American Specification for the Design of Cold-Formed Steel Structural members, 2016 Edition) and with the use of AISI Manual -- Cold-Formed Steel Design, 2017 Edition.

The verification process involves evaluating the structural capacity of cold-formed steel sections and ensuring they meet the necessary strength and stiffness requirements and includes bending, compression, and tension verifications.

Verifications according to these regulations are conducted if AISC 360-16/AISI S100-16 or CSA S16-19/AISI S100-16 is selected as the standard for steel verification in the project configuration:

Standards	
	Method
Combinations	ASCE 7-16 ASD -
Seismic	ASCE 7-16
Climatic	ASCE 7-16
Reinforced Concrete	ACI 318-14
Steelwork	AISC 360-16/AISI S 👻
(I+ICSA S16-09
Timber	NTC 2008
	INTO 2010
Masonry	
	CSA S16-19/AISI S100-16

Choosing the standard for the project

Cold-formed profiles can be analyzed, both those that are defined in the program's profile libraries (those that have the lamination type set as Cold - Formed) and parametric cold-formed profiles.

NOT FOR RESALE vers	ion - Cross section	libraries		P 23		NOT FOR RESALE version - Defined	\$ 53
Library manager							
Section-Type AI -	-	2.25		ок	Type: 2	Sigma	- ОК
Name				ancel	Subtype :	Sigma	Cancel
European Protes Her Sections	8	0.05		telp	5	Grama	
North America Timber Prefiles	80		Type of la	nination	Label:	, ageno	
North America Cold Formed Profiles	0.01		Cold-Form	red Bole -	2	Sigma with inclined lips	
C-Sectione With Lipe	0.0				Description 5	Sigma asymmetric	
-C-Sections With Lips (joist/studs)	17	20			Height (*)	8.00 0.021	
- Equal Leg Angles With Lips					Width (")	200 0.03	
- Equal Leg Angles Without Lips	Designation	Area	b.	lz 🔺	Inner web beight (") 400	
- Hat Sections Without Lips	8ZS2.75x070	1.05 sq in	10.30 ''4	2.05	Outer web beicht	() 190 T	• +
- Z-Sections With Lips	8ZS2.75x065	0.97 aq in	9.55 "4	1.9(COLOT WOD THOIGHT	0.00	
Contract Parties	8Z52.75x058	ni pa 88.0	8.69 "4	1.72	vveb depression () 0.63	
Gratec Protes	8Z52.25x105	1.46 sq in	13.60 14	1.90	Lip (*)	0.80	
Cold Formed Profiles	8ZS2.25x085	1.19 sq in	11.10.14	1.57	Thickness (")	0.08	Т
E-Compound Profiles	8ZS2.25x070	ni pe 89.0	9.18 "4	1.28	Inper radius (*)	0.08	-
	8ZS2.25x065	0.91 aq in	8.54 "4	1.15	Calar		9
	▶ 8ZS2.25x058	0.82 aq in	7.76 *4	1.08	Color		0
	9Z52.25x105	1.57 sq in	18.10 '4	1.96	Type of lamination	Cold-Formed Rolled	
	9252.25x005	1.2/ sq in	14.7014	1.5		00	V
	9252.25x070	1.05 sq in	12.2014	1.2			08
	9252.238065	0.97 sq in	11.30 4	LB		+ <i>c</i> ⁰ .	00 ±
	3232,238053	0.88 aq m	10.30 4	1.02			•
	10253.25x105	1.83 sq m	28.4014	4.6.			1
	10253258005	1.03 sq in	23.10 4	3.75			
	10253256070	1.25 sq in	12 00 14	3.04			2.00
	10253250065	1.17 sq in 1.05 eq in	17.00 4	2.6:			
	4	Constant and	10.10	A DEC			

Example of a cold-formed profile from the database (left) and defined as parametric (right)

To facilitate the definition of cold-formed profiles for the North American market, new dedicated steel grades (ASTM Cold Formed library) have been added, as well as an additional new cold-formed steel profile library (acc. AISI Cold-Formed Steel Design Manual). For more information, see the separate dedicated paragraph in this document.

To be able to properly verify cold-formed profiles according to the AISI standard, the list of steel design properties has been expanded to include a new set of Cold-Formed design options.

🗖 Design					
E Steelwork Design					
 To calculate 	Enabled				
 Design Results 	Available				
 Work ratio 	0 %				
 — General Design Template 	None				
 Cross section class 	auto				
- Advance stability (2nd order)					
- Deflections					
- 🗄 Buckling					
- E Lateral-torsional Buckling					
Specific Properties					
Cold-Formed design options					
 Exact CFD calculation 	Enabled				
 Overlapped section at support 	Disabled				
 Overhang length 	0.00 m				
— Rotational stiffness K-phie(N.m/rad/	0.00				
 Bearing length 	0.00 m				
Exterior support					
Support condition	Fastened				
 Flange condition 	Stiffened or partially stiffen				
Flange loading type	One flange loading or reac				
🛛 🗗 Interior support					
 Support condition 	Fastened				
 Flange condition 	Stiffened or partially stiffen				
Flange loading type	One flange loading or reac				

While most of the parameters are directly related to the provisions of the standard, the first of the options (*Exact CFD calculation*) is used to toggle whether the effective cross-section is computed for each load combination only on several points along the element (the option is off) or the analysis must be performed at multiple points between each finite element mesh node (the option is on). This has a direct impact on the calculation time.

It should also be mentioned that with the latest version of Advance Design when analyzing steel profiles according to AISC 360-16/AISI S100-16, we can conduct verification considering 7 degrees of freedom (2nd order analysis including warping). For this purpose, the Advance Stability option is activated.

- E Design	
Steelwork Design	
 To calculate 	Enabled
 Design Results 	Outdated
 Work ratio 	0 %
 General Design Template 	None
 Cross section class 	auto
- Advance stability (2nd order)	
 2nd order with warping and imperfe. 	. 🗹 Enabled
 Number of iterations 	10
 Stability - 2nd order parameters 	Definition
Use for deflection checks	Disabled

After the profile analysis, the calculation results are available for other types of steel profiles in graphical form and in the Shape Sheet window and related reports.



Results in graphical form for steel element - Work ratio for web crippling

	Unfavorable case	Verification	Work ratio	
Compression (E2)	o*101	Prid = [1 - 0.25*(Prid / Py) ^ 0.6(*(Prid / Py) * 0.8*Py Prid = 11.54 kN	11%	
Bending /yy (F2.1)	n*101	Mind = (1 - 0.22*(Mord / My) * 0.5)*(Mord / My) * 0.5 My Mur(MndHad) + 1.0 - 1.06 Min/m(1.37 Min/Mord 90) + 1.0		
Bending /zz (F2.1)	n*101	Mind = [1 - 0.22*(Nerd / My) ^ 0.5]*(Nerd / My) ^ 0.5 Ny Nar(Vind/tect) < 1.0 0.05 kV*n/(0.23 kV*n/0.90) < 1.0	13%	
Shear G	n'101	Ve = (Ver * Vy) * 0.5 104 52 M = (140.56 M * 116.86 M) * 0.5 Vr : V/ 1764 59 20 M = 104.52 M/ 10.55 Vr : 142.54	1%	
Web Crippling (G5)	n*101	Pf ≤ Pr 1.42 kN < 7.64 kN	19%	
Combined Moment And Astal (H1)	n*101	Mx / Ma x + My / Ma y + P / Pa ≤ 1 0.00 < 1	0%	
Combined Moment And Shear (12)	n*101	((N / Melo) ^ 2 + (V / Va) ^ 2) ^ 0.5 < 1 0.03 < 1	86%	
Combined Moment And Web Crippling (H3)	o*101	$\begin{array}{l} 0.91^{*}(P/Pn) + \left(0.7Mnln \right) \leq 1.37\Omega \\ 0.16 \leq 1.20 \end{array}$	18%	

Shape Sheet widow with a summary of performed verifications

3) Cross sections stree	ngth			
Tension (D2)	not done (-)			
Compression	Case no 102 : 1x[1 D]+1x[2 L], Mesh No. 13.1 0/4			
(E2)	Ac = 0.91 sq in txc = 8.64 "4 tyc = 0.34 "4			
	$P_{n1} = A_n F_n \le P_{nn} (E3.1-1)$			
	9921.59 lb = 0.91 sq in * 10940.68 lb/in* < 12031.97 lb (0 %)			
Hending /yy	Case no 102 : 1x[1 D]+1x[2 L], Mesh No. 13 8: 4/4			
(F2.1)	Ae - 1.10 sq in Se - 2.45 cu in Set - 2.45 cu in Ie - 9.79 *4			
	$M_{rd} = (1 - 0.22^{\circ} \sqrt{\frac{M_{uad}}{M_{u}}})^{\circ} \sqrt{\frac{M_{uad}}{M_{u}}}^{\circ} M_{y} (F4.1.2)$			
	Mu/(Mnd/fact) < 1.0			
	106 23 b*ft/(10808 18 b*ft/0 90) < 1.0 (1.%)			
Bending /zz	Case no 102 . 1x[1 D]+1x[2 L], Mesh No. 13.1 0/4			
(F2.1)	Ac = 1.10 sq in Sc = 0.39 cu in Sct = 1.06 cu in Ic = 0.57 "4			
	M ₁ ≤ M, (F2.1)			
	13.93 b"ft < 1515.20 b"ft (1 %)			
Shear G	Case no 102 : 1x[1 D]+1x[21], Mesh No 13 8: 4/4			
	$V_n - V_{\alpha} - A_{\alpha} \Gamma_{\alpha} (G2.3-1)$			
	Vu < fact * Vn			
	11728.70 lb = 0.83 sq in * 18504.18 lb/in*			
	52 79 lb < 0.95 * 11728 70 lb (0.%)			
Web Crippling	Case no 102 . 1x[1 D]+1x[2 L], Mesh No. 13.8 4/4			
(G5)	$P_1 \leq P_1$ (G5)			
	52.79 b < 869.70 b (6 %)			
Combined Moment	Case no 101 : 1x[1 D], Mesh No. 13.1 0/4			
And Axial	$\frac{M_{12}}{M_{12}} + \frac{M_{12}}{M_{12}} + \frac{P_{12}}{M_{12}} \le 1.0412.11$			
(H1)	$M_{ax} = M_{ay} = P_a = 1000.2400$			
	24 74 lb*ft / 9320 74 lb*ft + 3 47 lb*ft / 1515 20 lb*ft + 2 51 lb / 8433 35 lb< 1 (0 %)			
Combined Moment	Case no 102 . 1x[1 D]+1x[2 L], Mesh No. 13.8 4/4			
And Shear (HZ)	$\sqrt{(\frac{M}{M_{w}})^{2} + (\frac{M}{M_{w}})^{2}} \le 1 (12.1)$			
	((106.23 lb*tt / 9727.36 lb*tt)/2 + (52.79 lb / 11142.26 lb)/2) ^ 0.5 < 1 0.01 < 1 (1%)			
Combined Moment	Case no 102 : 1x[1 D]+1x[21], Mesh No: 13.8, 4/4			
And Web Crippling	on Pr. Meralowan			
(H3)	$(917\frac{1}{10}) + \frac{1}{M_{40}} \le 1.37 \ \Omega(H3.1)$			
	0.91 * (52.79 lb / 1023.18 lb) + (106.23 lb*tt / 11217.16 lb*tt) <1.20 0.06 < 1.20 (6 %)			
Combined Moment	not done (-)			
And Torsion	and another of t			
(114)				

Shape Sheet report with details of performed verifications

4.2. Shear web buckling analysis for I sections (EN 1993-1-5)

Possibility for running a verification of the stability of I-section webs according to EN 1993-1-5.

Advance Design 2024 now checks webs for shear buckling according to EN 1993-1-5. This verification can be critical for deep I-sections, which can be used on many types of structures, including bridges.



Such beams usually carry heavy loads, and because bending is typically taken by flanges, the web is kept very thin, making it prone to plate buckling from the shear force, also known as shear buckling.

Shear buckling is mentioned in §6.2.6(6) from EN 1993-1-1, as a verification to perform when the width-to-thickness ratio of the web plate (h_w/t_w) exceeds a limit value.

§5.1(2) from EN 1993-1-5 is more specific, introducing two limit values for web slenderness:

- For an unstiffened web.
- For a stiffened web.

The EN 1993-1-5 offers two methods when to account for buckling in steel plates:

- The effective width method, specified in sections 4 to 7.
- The reduced stress method is briefly covered in section 10.

The method implemented in Advance Design 2024 is the **effective width method**. The reduced stress method is not available yet, but it is planned to be implemented soon.

NOT FOR RESALE version - Calculation Settings			
Verification	Verification		
Safety factors Optimisation Sort profiles Buckling Angle verification Calculation Sequence	Verification Verification Verification Verification Time Omin Verification Verifica	Calculation type tension - compression combined bending oblique bending	
		Beams	
	Columns buckling Igth = properties list Columns buckling neglected	Beams buckling lgth = properties list Beams buckling neglected	
	Plate (web) buckling Effective width method (EN 1993-1-5 §5.6.7) Reduced stress method (EN 1993-1-5 §10) 	OK Cancel Help	

Theoretical background - the Effective width method

In the effective cross-section method, the plate-like behavior of the web is accounted for by reducing the gross width (b or h_w) to an effective width (b_{eff}), with a full f_y yield strength.

Note that the shear buckling verification of the web is performed with the gross web height (h_w) (not the effective height).

• Verification

The shear buckling verification consists of ensuring that the design shear force (V_{Ed}) does not exceed the design resistance for shear ($V_{b,Rd}$).

The design shear resistance $(V_{b,Rd})$ is estimated from:

- The contribution from the web ($V_{bw,Rd}$)
- The contribution from the flanges (V_{bf,Rd})

$$V_{b,Rd} = V_{bw,Rd} + V_{bf,Rd} \le \frac{\eta \ f_{yw} \ h_w \ t}{\sqrt{3} \ \gamma_{M1}}$$
(5.1)

• Contribution from the web



The contribution of shear resistance from the web

The contribution from the web requires the determination of a shear buckling factor (χ_w) (EN 1993-1-5, 5.1).

This shear buckling factor (χ_w) depends on:

- The relative slenderness of the web (λ_w)
- Whether the transverse stiffeners at the end of the web panel are **rigid** on **non-rigid**

NOTE: In Advance Design 2024, the end stiffeners are considered non-rigid.



Rigid end post (left), non-rigid end post (right)


The relative web slenderness (λ_w) is given in eq. (5.3).

$$\overline{\lambda}_{w} = 0.76 \sqrt{\frac{f_{yw}}{\tau_{cr}}}$$
(5.3)

It involves the stress and the shear buckling coefficient (k_c), defined in Annex A from EN 1993-1-5. This shear buckling coefficient (k_c) depends on the distance between transverse stiffeners as well as the inertia about the z-z axis of the longitudinal stiffeners:



1 - Rigid transverse stiffeners, 2 - Longitudinal stiffeners, 3 - non-rigid transverse stiffener

NOTE: In Advance Design 2024, with the effective width method only transverse stiffeners can be defined and they are assumed as non-rigid.

Once web slenderness (λ_w) has been determined, the shear buckling factor (χ_w) is computed according to Table 5.1, with Advance Design considering the end posts as **non-rigid**.

• Contribution from the flanges



The contribution of shear resistances from the flanges

The flanges may only contribute if they have some remaining capacity after taking the bending moment (EN 1993-1-5, 5.4). For this, we need to calculate the moment of resistance of the cross-section consisting of the effective area of the flanges only $(M_{f,Rd})$.

$$M_{f,Rd} = b_f \cdot t_f \cdot \left(h_w + t_f\right) \cdot \frac{f_y}{\gamma_{M0}}$$

GRAITEC



Note that the presence of an axial force (N_{Ed}) may reduce the resisting moment of the flanges through the factor:



We need also determine the c length for the flange's contribution:

$$c = a \left(0,25 + \frac{1,6 b_f t_f^2 f_{yf}}{t h_w^2 f_{yw}} \right)$$

Note that the contribution from the flanges, when it exists, is usually significantly lower than the contribution from the web.

• Shear buckling verification

The shear buckling verification consists of ensuring the design shear force V_{Ed} does not exceed the design resistance for shear $V_{b,Rd}$ from web and flanges:

$$\eta_3 = \frac{V_{Ed}}{V_{b,Rd}} \le 1,0 \tag{5.10}$$

When η_3 exceeds 0,50, extra verification is performed to account for the interaction between shear force, bending moment, and axial force (EN 1993-1-5, 7.1).

Definition and results in Advance Design

On Advance Design, a new 'Web buckling' section has been added to the steel design properties, for managing parameters related to the shear buckling verification.

□ Lateral-torsional Buckling	
 Elem. Verification 	Enabled
Lateral-torsional buckling length	
- E Web buckling	
 Web buckling verification 	Enabled
Transverse stiffeners	
- Enable	Enabled
 Relative location along element 	0.0; 0.2; 0.4; 0.6; 0.8; 1.0;
Sides of the web	Both sides
□ Longitudinal stiffeners	
Imperfections EC3	
Beams equivalent forces	

Web shear buckling parameters (Effective width method)

The *Web buckling verification* checkbox governs whether Advance Design performs the verification or not. The *Transverse stiffeners* parameters let us position the transverse stiffeners along the member, with various definition modes being available. For this, a dedicated dialog is opened.

	Relative	Add
0.00 m	0.00	
2.00 m	0.20	
4.00 m	0.40	Equal spacings
6.00 m	0.60	
8.00 m	0.80	X Remove
10.00 m	1.00	
		× Remove all
-f:-11		
erinition		
_		
r₃ Graj	phical definition	Auto detect
r⊊ Graj Type	phical definition	Auto detect
rs Graj Type	phical definition	Auto detect
rs Graj Type	phical definition	Auto detect Please input a number of intermediate
rs Graj Type	phical definition	Auto detect Please input a number of intermediate stiffeners
rs Graj Type ● Number ○ Abscissas (phical definition	Auto detect Please input a number of intermediate stiffeners

Transverse stiffener definition

Advance Design allows many ways to define transversal stiffeners, including by equal distribution after entering their number, by entering their relative or absolute coordinates, but also by graphical definition (indicating points by the length of the element in the 3D view) or by using the command for automatic detection based on intersecting elements.

After calculation, the basic results from the web shear buckling verification are available in the shape sheet:

	case	section Class	Verification	ratio	
Shear on Z direction (6.2.6)	n*101	Class 1	Fz < Vpiz 646.4 < 2605.0 kN	25%	
Bending on Y-Y (8.2.5)	="101	Classe 4	My€d < MycRd 2457.81 < 4347.18 M*m	57%	
Shear buckling of web (EN1993-1-5, 5.10)	n*101	-	Ved + Vb,Rd : 646.4 < 671.7 kN	96%	

Shape sheet

The detailed version of the shape sheet report provides more information regarding:

• The shear buckling checks itself, mentioning the buckling coefficients ($k_{c,sB}$, and k_c), the slenderness parameter (λ_w), the shear buckling factor (χ_w)...

Shear buckling of	Case no 101 : 1x[1 L], Mesh No. 1.1 0/4 Point x = 0.00 m
web	Cross section : Class -
(EN1993-1-5, 5.10)	
	$k_x = 5.34 \ k_{x,sB} = 0.00$
	σ _E = 3.04 MPa λ _w = 2.89 χ _w = 0.29 V _{bw Rd} = 623.1 kN
	M _{f.Rd} = 37.61 kN*m k _{NEd} = 0.00 c = 77.35 cm V _{bf.Rd} = 48.6 kN
	h _{w 2} 31
	t < π ε\/κ _x : 250.00>59.70
	V _{Ed} < V _{b.Rd} : 646.4 < 671.7 kN (96 %)

• The interaction between shear force, bending moment, and axial force (if relevant)

Interaction - shear, bending, axial force (EN1993-1-5, 7.1)	Case no 101 : 1x[1 L], Mesh No. 1.1 0/4 Point x = 0.00 m Cross section : Class -
	$\eta_{-1} = 0.00$ $\eta_{-3} = 0.96$ M _{pl.Rd} = 5715.39 kN*m M _{f.Rd} = 3797.60 kN*m
	$\eta_1 \leq \frac{M_{f,Rd}}{M_{ol,Rd}}$ Bending moment can be carried out by flanges alone. No need to satisfy
	eq. (7.1) : 0.00 < 1.00 (0 %)

• The position of the stiffeners

Γ	Stiffeners	Transverse stiffeners - relative positions: 0.00; 0.20; 0.40; 0.60; 0.80; 1.00;
		Location: Both sides

4.3. User definition of buckling and LT buckling lengths for cold-formed sections for EC3

Next step of improvements to cold-formed design acc. Eurocode 3 for analyzing buckling and lateral-torsional buckling.

Advance Design 2024 now allows for a more accurate definition of buckling lengths and lateraltorsional buckling lengths on cold-formed structural components.





The hat section is used as a top chord of a lightweight steel structure

Buckling length definition

The buckling dialog now lets the users define the critical length for torsional buckling and torsionalflexural buckling, involved in the calculation of the Ncr,T, and Ncr,TF critical forces, respectively.

	subject to buckling			
		-	z	
			Y	
			*x	
xz plan	e (strong axis)	_		
Cmy,0	Auto calc.	\sim	0	
LO	Auto calc.	~	0 m	
Lfz	Imposed value	~	3.16 m	
Lfz fire	= Lf	~	0 m	
~	Auto			
Curve	Auto	*		
Curve	Auto	Ŷ		
Curve	Auto	Lfz = 3.16	m	
Curve	Auto	Lfz = 3.16	m	
xy plan	e (weak axis)	Lfz = 3.16	m	
xy plan Cmz,0	e (weak axis)	Lfz = 3.16	m 0	
xy plan Cmz,0	e (weak axis) Auto calc. Auto calc.	Lfz = 3.16	m 0 0 m	
xy plan Cmz,0 L0	e (weak axis) Auto calc. Auto calc. Imposed value	Lfz = 3.16	m 0 0 m 1.58 m	
xy plan Cmz,0 L0 Lfy	Auto calc. Auto calc. Imposed value = Lf	↓ Lfz = 3.16 ↓ ↓ ↓	m 0 0 m 1.58 m	
xy plan Cmz,0 L0 Lfy Lfy fire Curve	e (weak axis) Auto calc. Auto calc. Imposed value = Lf Auto	Lfz = 3.16	m 0 0 m 1.58 m 0 m	
xy plan Cmz,0 L0 Lfy Lfy fire Curve	Auto calc. Auto calc. Imposed value = Lf Auto	Ufz = 3.16	m 0 0 m 1.58 m 0 m	
xy plan Cmz,0 L0 Lfy Lfy fire Curve	Auto calc. Auto calc. Imposed value = Lf Auto	Lfz = 3.16	m 0 0 m 1.58 m 0 m	
xy plan Cmz,0 L0 Lfy Lfy fire Curve	e (weak axis) Auto calc. Auto calc. Imposed value = Lf Auto	Lfz = 3.16	m 0 0 m 1.58 m 0 m	

Buckling settings

Indeed, the complex shape of these open-walled sections, often asymmetrical, results in low torsional resistance, with a significant offset between their shear center and gravity center.



S - Shear center, G - Centroid

This makes them prone to **torsional buckling** (where compression causes a twist of the element) as well as **flexural-torsional buckling** (a combination of bending and twisting of the compressed member), as explained in §6.3.2 from EN1993-1-3.

This new option ensures a precise estimation of the corresponding critical forces N_{cr,T_r} and $N_{cr,TF}$.

After calculation, the detailed shape sheet will mention:

- The critical lengths for torsional and torsional-flexural buckling ($L_{cr,T}$ and $L_{cr,TF}$)
- The corresponding critical forces (N $_{\rm cr,T}$ and N $_{\rm cr,TF}$)
- The corresponding buckling coefficients χ_T and χ_{TF}

Buckling	Case no 101 : 1x[1 S], Mesh No. 2.1 0/4
(6.2.2)	Cross section : Class 4
	N _{Ed} < N _{b.Rd} : 200.8 kN < 210.3 kN
	$L_{fy} = 1.58 \text{ m}, \ \lambda_{bar, y} = 0.43, \ \alpha_y = 0.34$
	$L_{f_7} = 3.16 \text{ m}, \ \lambda_{\text{bar}, z} = 0.16, \ \alpha_z = 0.34$
	L _{cr.TF} = 3.16 m, N _{cr.TF} = 312.1 kN, L _{cr.T} = 3.16 m , N _{cr.T} = 312.1 kN
	$\chi = \min(\chi_{y}, \chi_{z}, \chi_{T}, \chi_{TF}) = \min(0.91, 1.00, 0.50, 0.50)$
	$N_{\rm Ed} \le \frac{\chi * A_{\rm eff} * f_{\rm yb}}{2}$ (6.48 EN1993-1-1)
	YM1
	200.8 kN < 0.50 * 11.86 cm ² * 355.00 MPa (95 %)
	1.00

Lateral-torsional buckling length definition

The Lateral-torsional buckling dialog now lets the users define intermediate restraints on the top or bottom sides of cold-formed members.

GRAITEC

	Upper Flang	Lower Fl	ange					
Co	intinuous restra	nt along flange						
Rest	traints/Only for	elements with su	me length)					
-				Contrather				
d	definition	Auto detect		flange			1	
	Abscissa	Restr	aint	Lor		A		
(0.00 m	Pinned	0.00	m	1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1			
	3.16 m	Pinned	3,16	m		1		
	6.32 m	Pinned	3.16	m				
L	12.64 -	Pinned	3.16					
						tt		-
ł	Add Equ	al spacings	Remove	Remove all	Pinned	Fixed		Continuous
Defi	Add Equ inition type :	al spacings	Remove	Remove all	Pinned	Fixed	0 kN'm	Continuous
Defi	Add Equ inition type :) Number	al spacings	Remove s (m) C	Remove all Ratio	Pinned Critical moment Mcr Critical moment Mcr0	Fixed Auto calc.	0 kN'm 0 kN'm	Continuous
Defi	Add Equ inition type :) Number Use space or ?	Abscissa separator be 5	Remove s (m) C	Remove all	Pinned Critical moment Mor Critical moment Mor0 Select curve	Fixed Auto calc. Auto calc. Curve b	0 kN'm 0 kN'm	Continuous
Defi C L Res	Add Equ inition type :) Number Use space or ',' example: 1 3.5 straint type :	Abscissa Separator be Separ	Remove s (m) C	Remove all Patio m Define	Pinned Critical moment Mor Critical moment Mor0 Select curve Load type	Fixed Auto calc. ~ Auto calc. ~ Curve b Auto	DkN'm DkN'm ~	Continuous
Defi C L Res	Add Equ inition type :) Number Use space or ',' example: 1 3.5 straint type : eral-torsional bu	el spacings Abscissa as separator be 5) Pinned okling length	Remove s (m) C tween values	Remove all Ratio m Define	Pinned Critical moment Mor Critical moment Mor0 Select curve Load type Choose method	Fixed Auto calc. ~ Auto calc. ~ Curve b Auto General case (Dause	0 kN'm 0 kN'm ~ * : 6322	Continuous
Defi C L Res	Add Equ inition type :) Number Use space or example: 1 3.5 straint type : eral-torsional bu Auto calc.	Abscissa Abscissa separator be 5) Pinned ckling length v	Remove s (m) C tween values	Remove all Ratio m Define	Pinned Critical moment Mor Critical moment Mor0 Select curve Load type Choose method Reduction Factor f	Fixed Auto calc. ~ Auto calc. ~ Curve b Auto General case (Clause None ~	0 kN'm 0 kN'm • 6.3.2.2) 0	Continuous
Defi C L Res Late	Add Equ inition type : Number Use space or '? example: 1 3.5 straint type : eral-torsional bu Auto calc.	el spacings Abscissa as separator be 5) Pinned ckling length Lds = to calc	Remove s (m) C tween values > 0 m	Remove all Ratio m Define	Pinned Critical moment Mor Critical moment Mor Critical moment Mor0 Select curve Load type Choose method Reduction Factor f Load application point	Fived Auto calc. ~ Auto calc. ~ Curve b Auto General case (Dause None ~	0 kN'm 0 kN'm • 5322	Continuous
Defi C L Res Late	Add Equ inition type : D Number Use space or V example: 1 3.5 strait type : arait onsional bu Auto calo.	Abscissa Abscissa S Abscissa as separator be 5) Prined Abking length Lds = to calc tors	Remove (m) C tween values) m avlate	Remove all Ratio m Define	Pinned Critical moment Mor Critical moment Mor Critical moment Mor Select curve Load type Choose method Reduction Factor f Load application point Neutral fibre	Fixed Auto calc. ~ Auto calc. ~ Curve b Auto General case (Clause None ~ 2020 m	0 kN'm 0 kN'm • • • • • • • • • • • • • • • • • • •	Continuous
Defi C L Res Late	Add Equ inition type :) Number Jee space or V example: 1.35 straint type : real-tonional bu Auto cale. Imposed value	al spacings Abscissa as separator be 5) Pinned ckling length Lds = to calc tors e [Remove (m) C tween values (m) (m) (m) (m) (m) (m) (m) (m) (m) (m)	Remove all Ratio m Define	Pinned Critical moment Mor Critical moment Mor Select curve Load type Choose method Reduction Factor f Load application point Neutral fibre	Fixed Auto calc. ~ Auto calc. ~ Curve b Auto General case (Clause None ~ Zg: 0 m ?: ()r	0 kN'm 0 kN'm v 6 3 2 2) 0 n	Continuous
Defi C L Res Late Effect kz: kw:	Add Equ inition type :) Number Jee space or ? example: 1.35 straint type : real-tonional bu Auto calc. Imposed valu Imposed valu	Absciss Absciss S Absciss S Pinned cking length C Lds = to calc tors e [e [e]	Remove s (m) C tween values v () m vulote 1	Remove all Ratio m Define	Pinned Critical moment Mor Critical moment Mor Select curve Load type Choose method Reduction Factor f Load application point Neutral fibre ~ Loading and end supporting c C1: Auto calc. ~	Fixed Auto calc. ~ Auto calc. ~ Auto calc. ~ Curve b Auto General case (Clause None ~ 7 27:0m n 0	0 kN'm 0 kN'm < \$322 0 n	Continuous
Defi C L Res Late Effec kz: kw:	Add Equinition type : D Number Jase space or % example: 13.5 straint type : real-tonsional bu Auto calc: Imposed valu Imposed valu	el spacings Abscissa as separator be 5) Pinned biting length Lds a to colu tors e v [Remove s (m) C tween values v [) m vulate 1	Remove all Ratio m Define	Pinned Critical moment Mor Critical moment Mor Critical moment Mor Select curve Load type Choose method Reduction Factor f Load application point Neutral fibre ~ Loading and end supporting of C1: Auto calc. ~ C2: Auto calc. ~	Fixed Auto calc. V Auto calc. V Curve b Auto General case (Clause None 20:0m; i: 0: conditions 0 0 0	0 kN'm 0 kN'm • • • • • • • • • • • • • • • • • • •	Continuous

Lateral-torsional buckling settings

The users may even neglect the lateral-torsional buckling effects entirely by ticking the 'Continuous restraint along flange' box, which assumes the torsion effects have been avoided by other means (by using batten plates, by ensuring a fixed connection from a slab to the flange...).

teral-torsional B	uckling	
Upper Flai	Lower Flange	
Continuous res	traint along flange	
Restraints(Only f	or elements with same len	gth)
Graphical definition	Auto detect	Copy to other flange

Neglecting the lateral-torsional buckling effects

Alternatively, the users may rely on the Advanced stability engine to compute the critical moment (M_{cr}) in case of a complex section shape.



Option for selecting the method of Critical moment calculation

In this case, the $N_{\text{cr},\text{T}}$ and $N_{\text{cr},\text{TF}}$ critical forces will also result from the Advanced stability analysis.

After calculation, the detailed shape sheet will return the various parameters involved in the calculation of the critical moment (M_{cr}), conducted according to Annex I from EN 1999-1-1. Such parameters include:

- The cross-section monosymmetric factor $(\Psi_{\rm f})$
- The C_1 , C_2 and C_3 coefficients
- The Wagner factor (z_j) (which reflects the asymmetry of the section)

Lateral-torsional Buckling (6.2.4)	Case no 101 : 1x[1 S], Mesh No. 2.2 4/4 Cross section : Class 4 M _{y,Ed} < M _{by,Rd} : 6.00 kN*m < 25.45 kN*m
	$ M_{y,Ed} + \Delta M_{Ed} < \frac{\chi_{LT} \star W_{eff,y} \star f_{yb}}{\gamma_{M1}}$
	-6.00 kN*m + 0.00 kN*m < <u>0.76 * 94.78 cm3 * 355.00 MPa</u> 1.00
	Eccentricity favorable effects are ignored.
	$k_z = 1.00, k_w = 1.00, k_{wt} = 1.18, \psi_t = 0.81,$ C1 = 1.13, C2 = 0.46, C3 = 0.53
	Lds = 12.64 m, Ldi = 0.97 m, Lcr = 12.64 m,
	zg = 0.00 cm, <u>zj = 18.20 cm</u> , <u>Mcr = 60.23 kN*m</u> , λ _{bar L1} = 0.75, _{χL1} = 0.76 Mcr was computed analytically. (24 %)

4.4. Modeling welded tube connections

Ability to define welded truss connections for rectangular hollow sections.

In the latest version of Advance Design, welded tube connections have been added to the set of many types of steel connections that can be created in the model and then sent for verification to the Steel Connection design module. These are 2d joints that allow the connection of welded rectangular and square steel tubes, i.e. connections typical of trusses.





New connections defined on trusses in the Advance Design model

New connections can be defined using new commands from the ribbon or the context menu.

X 🗶 🍊 🎽 🔛 🖥	Load Anus / Selection Points / Selection Slage / Selection		
Design Se	Connections Create / selection	Base Plates	
Jinks (Stoup	Restraints + Group	Splices	
	Select identical	Fixed connections	
	Selection + Auto systems	Tube Connections with Planges	4
N	Super stamped	Turbe connections with exercise state	I
M M	Jupat element.	Plate connections with gusset plate *	

We can define 3 category of welded tube joints that differs with the number of secondary bars - with 1, 2 or 3 secondary bars. With them, we can model the most common typical joints supported by Eurocode 3 (T type, Y type, N type, K type, X type and KT type):



All these connections allow to join open steel profiles with a square or rectangular cross-section. Such sections can be used as main elements (chords) and as secondary elements (diagonals). In addition, for main elements (chords) it is possible to use I sections.



NOT FOR RESA	Ever	tion - Cross section	I bovies		\$ 53		NOT FOR RESAU	Ever	dan - Cross section H	bowlet		¥ 12
Ubray manager Seator holice Sector Hype Seator holice Runc Runce Runce 1	•	2'00	t	1.30 C Son Type of lond Calul Paleot Restan	K cod	Cubrany I Sector Name I I I I I I I I I I I I I I I I I I I	narago nr 1994 Hectangular holon Auropean (hollea HR20 HR20 HR20 HR2 HR2 HR2 HR2 HR2 HR2		EN EN		Con Con Type of land Cold Palled Thitston	8 and a second s
Contact Profiles Reflect-Corporated Treffiles ACC A10051155 square ACC A10051155 square Action 20155 square Action 20155 square ACTH A500 ACSC 111155 Square ACTH A500 A	-	Designation SH5435-30 SH5535-2-30 SH5535-2-30 SH5535-30 SH5535-40	Area 6.35 cm² 4.55 cm² 3.75 cm² 5.41 cm² 6.55 cm²	2 30 cm3 15 30 cm4 14 10 cm4 15 30 cm4 23 70 cm4 23 70 cm4	12.2 76.1 16 78.1 23.7 23.7	34	Jahlee Hoffes - Non-Compound Hoffes - ASC: A1005 HB3 rectangular - ASC: HB5 rectangular - Mis Bised He3 angular - Australian Rectangular HB1 on Socion - CISC: T1 HS3 rectangular CANSH 20 - CISC: T1 HS3 rectangular CANSH 20 - CISC: T1 HS3 rectangular CANSH 20 - CISC: T1 HS3 rectangular - CISC: T1 HS3		Designation R1570x53x32 RH580x43x2 RH580x43x2 RH580x43x2 RH580x43x32 RH580x	Area 12.00 cm ² 6.09 cm ² 4.04 cm ² 6.01 cm ² 8.05 cm ² 10.40 cm ²	70.50 cm/ 45.10 cm/ 52.40 cm/ 52.30 cm/ 52.30 cm/ 54.50 cm/ 54.50 cm/	41.1 15.8 12.7 177 21.3

Square and rectangular open steel profiles on Advance Design databases

As in the case of other available connections, new connections can be defined individually or many at once, and groups can be created from them. After defining new connections we have their properties, in which we can check the most important parameters.



Properties of a selected new welded tube connection

New connections can be opened or exported to the Steel Connection design module for further analysis. Along with data on geometry, profiles, and materials, internal forces are also transferred. Further details, including welds' parameters, can then be defined in the Steel Connection design module and the load capacity analysis of these connections can be conducted.

Note: More details about new connections in the Steel Connection design module you can find in a separate dedicated chapter.



Welded truss connection opened on Steel Connection design module

4.5. Deflection in value (cm or mm) in the shape sheets and diagrams

Easier to check results and create documentation for steel member deflection verification.

Previously, deflection values from the verification of steel elements were displayed only as a ratio (e.g.,1/574) to make comparison with limit values (e.g., 1/500) easier. Now the deflection can also be displayed in displacement values (e.g., cm).

For graphical presentation, the new display mode is available in the Result Settings window for Deflection results, as a new 'Values' item in the Display Mode list.









Graphic results (max deflection in cm)

In the case of the Shape Sheet window and report, the values in the displacement unit are displayed just below the ratio values. In addition, the Mesh column displays information about the location of the verification point - whether it is in the span or the extremity of the element.

		Unfavorable case	Mesh	Verification	Deviation max / all defi	
ement	Max deflect. y: n*104 : 1x(1 D)+1x(2 L) 1st criterion 2: n*104 : 1x(1 D)+1x(2 L)		2.7 (Span)	L / 678.40 < L / 500.00 (1.47 cm < 2.00 cm) L / 576.69 < L / 500.00 (1.73 cm < 2.00 cm)	74 % 87 %	L = 10.00 m
	Max deflect. 2nd criterion					

Shape Sheet

5. Enhancing the analysis of timber structures acc. Eurocode

5

A series of novelties and improvements related to the verification and optimization of timber element structures according to Eurocode 5.

5.1. Design of spaced rectangular sections (EC5)

Ability to verify timber columns and beams with compound sections according to Eurocode 5.

Advance Design 2024 now enables the design of spaced members using Eurocode 5.

This includes:

- Spaced columns
- Spaced beams



Spaced columns



Spaced beams

Spaced shaft profiles are defined using a new parametric profile type *(see details in a separate paragraph on new parametric profiles).*

Spaced columns

A spaced column is a built-up column made of 2 to 4 identical sections (shafts) separated and connected by packs or gussets fixed by glue or mechanical fasteners. They are often used as compression chords or in frame construction due to their high load-carrying capacity.

Spaced columns are covered in Annex C.3 from EN1995-1-1.





A vertical element with a cross-section defined as a spaced shaft will be recognized as a column, which will give access to additional parameters in the timber design section, where the users can define the distance between connectors as well as their type.

E Design	
E Timber Design	
 To calculate 	Enabled
 Design Results 	Available
— Work ratio	57 %
 General Design Template 	None
 — Service class 	Class 2
 Humidity percentage 	12 %
 Systems effect coefficient Ksys 	1
- 🗗 Spaced columns	
 Length of unrestrained bays (midlines) L1 	0.68 m
Connectors	Nailed packs
-⊞ Fire Design	
Deflections	

Spaced columns are designed according to annex C from EN 1995-1-1 which is intended for simply supported members. This annex describes two checks related to spaced columns:

- Verification of the axial load-carrying capacity (§C.3.2)
- Verification of the load on fasteners, gussets, or packs (§C.3.3)

After calculations, the results of the analysis can be viewed on the 'Stability' tab of the Shape sheet window as well as in the report. The detailed version of the shape sheet additionally provides the intermediate terms involved in these verifications.

4) Elements stability	,
Verification	Case no 103, Mesh No. 5.5
	$\lambda_{\rm v} = 86.6 \ \lambda_{\rm z} = 64.1$
	Lfy = 5.00 m Lfz = 5.00 m
	Ky=1.504 Kcy=0.475 Kz=1.056 Kcz=0.490 Km=0.700 Kcrit=0.994
	$\lambda_{rel,v} = 1.378 \ \lambda_{rel,z} = 1.019 \ \lambda_{rel,m} = 0.755$
	$\lambda_1 = 32.332$ $\eta = 3.000$ $\lambda_{ef} = 85.080$ Kc=0.475
	Ldy = 5.00 m Ldz = 5.00 m
	Work ratio Verification:
	Case no 103, Mesh No. 5.5
	6.23: σ _{c0d} /(K _{cy} F _{c0d}) + σ _{myd} / F _{myd} + K _m σ _{mzd} / F _{mzd} ≤1
	0.11695 < 1 (12%)
	Case no 103, Mesh No. 5.5
	6.24: $\sigma_{c0d} / (K_{cz} F_{c0d}) + K_m \sigma_{myd} / F_{myd} + \sigma_{mzd} / F_{mzd} \le 1$
	0.11335 < 1 (11%)
	Case no -, Mesh No,
	6.33: σ _{md} /(K _{crit} F _{md}) ≤1
	not done (-)
Axial Load-carrying	Case no 103, Mesh No. 5.5
capacity	(C.1): $\sigma_{c,0,d} \le k_c * f_{c,0,d}$
(C.3.2)	0.96 MPa < 0.48 * 17.28 MPa : 0.96 < 8.21 MPa (12 %)
	Simply supported column under concentric axial load, as per Annex C from EN1995-1-1
Shear Force on	Case no 103, Mesh No. 5.5
connectors	$(C 5): V_{c} = \frac{f_{c,d}}{f_{c,d}} = \frac{28.8 \text{ kN}}{100000000000000000000000000000000000$
(C.3.3)	(0.3) . $v_d = 60 * k_c = 60 * 0.48 = 1.0 \text{ km}$
	(C.13): $T_d = \frac{V_d^* L_1}{a_1} = \frac{1.0 \text{ kN} * 700 \text{ mm}}{75 \text{ mm}} = 9.4 \text{ kN}$
	$F_{v,Ed} = T_d = 9.4 \text{ kN}$

Shape sheet (detailed version)

Spaced beams

Spaced beams are commonly used as collar ties or rafter ties. Therefore, they are meant to resist the tension or compression forces caused by gravity loads and uplift loads.



Spaced beams as rafter tie and collar tie

The design of spaced beams according to EN 1995-1-1 is conducted in the same manner as for a usual rectangular section, but with adjusted parameters to account for the spacing between the rectangular shafts.

The following parameters are modified:

- Depth factor k_h
 For members in tension, depth factor k_h uses section width. For spaced beams width = b (width of a single rectangle)
- Critical bending stress σ_{m,crit}
 It is used for lateral-torsional buckling effects. It is calculated using simplified (6.32) formula, with b and h of a single rectangle.
- Shape of the cross-section factor k_{shape} k_{shape} is used in the torsion check. It is calculated based on the b dimension of a single rectangle.
- Effective width b_{ef}
 For shear verification, the effective width b_{ef} is computed with width = 2b (width of the two rectangles).

The results of the timber design are available graphically or through the shape sheet:



Stability work ratio (%)

🔳 S	■ Shape sheet - Linear element n°2 ×							
Se	ction Deflections (20%) Strength (29%) Sta	bility (26 %) Fire Resistance () Fire stability ()						
			_					
	Buckling Slend. and Length	$\lambda y = 60.622 \ \lambda z = 33.627$ Lfy = 3.50 m Lfz = 3.50 m						
	Ltors.buckl. Slend. and Length	Ldy = 3.50 m Ldz = 3.50 m						
	Unfavorable case	Case no 102 : 1.35x[1 G]+0.9x[3 V]+1.5x[2 N]						
	Magnification factor	Ky=1.101 Kcy=0.669 Kz=0.690 Kcz=0.928 Km=0.700 Kcnt=1.000 λrel.y=1.028 λrel.z=0.570 λrel.m=0.540 WR=26.303% λ1=0.000 η=0.000 λef=0.000 Kc=0.669						
	Verification	Case no102 : 1.35x[1 G]+0.9x[3 V]+1.5x[2 N] (6.23): s c.0,d /(K c.y * F c.0,d) + s m.y.d / F m.y.d + K m * (s m.z.d / F m.z.d) ≤ 1 0.263 < 1 (26 %)						
	Verification	Case no102 : 1.35x[1 G]+0.3x[3 V]+1.5x[2 N] (6.24): s c.0,d /(K c,z * F c,0,d) + K m * (s m.y,d / F m.y,d) + s m,z,d / F m,z,d ≤ 1 0.207 < 1 (21 %)						
	Venfication	Case no101 : 1.35x[1 G]+1.5x[2 N] (6,33): s m, d / (K cnt *F m, d) < = 1 0.249 < 1 (25 %)	•					
		Edit report Edit detailed report	ОК					

Shape sheet

5.2. Verification of bearing pressure on the support acc. EC5

Possibility of performing additional verification for timber elements according to Eurocode - the check of the compression perpendicular to the grain according to support sizes.

Advance Design 2024 now checks the compression perpendicular to the grain on the end supports of a beam, according to §6.1.5 from EN 1995-1-1.



Member on discrete support

A new 'Bearing at supports' section has been added to the timber design properties for users to define the actual contact length (L) as well as a potential overhang (a).

Bearing at supports	
- 🗗 Start support	
 Verification 	Enabled
 Support width (L) 	190 mm
Beam edge to support edge distance (a)	0 mm
End support	
 Verification 	Enabled
 Support width (L) 	190 mm
Beam edge to support edge distance (a)	25 mm



The verification is intended to ensure that, in the effective contact area, the compressive stress perpendicular to the grain ($\sigma_{c,90,d}$) does not exceed the compressive strength ($f_{c,90,d}$) magnified by a $k_{c,90}$ factor.

$$\sigma_{c,90,d} \le k_{c,90} f_{c,90,d}$$

The compressive stress perpendicular to the grain depends on the compressive load on the end support ($F_{c,90,d}$) and the effective contact area (A_{ef}),

$$\sigma_{\rm c,90,d} = \frac{F_{\rm c,90,d}}{A_{\rm ef}}$$

The effective contact area (A_{ef}) is determined by extending the actual contact length (i.e. support width) by a 30mm distance on each side, without exceeding the length of the overhang (a).



Effective contact width if long overhang ($a \ge 30$ mm)



Effective contact width if short overhang (a < 30mm)

The $k_{c,90}$ magnification factor accounts for the fact that the compression capacity is larger than the pure material strength perpendicular to the grain when a beam is subjected to concentrated pressure over a small surface.

This effect is even bigger for glued laminated timber than for solid timber, resulting in a smaller $k_{c,90}$ for solid timber.

(4) For members on discrete supports, provided that $\ell_1 \ge 2h$, see Figure 6.2b, the value of $k_{c,90}$ should be taken as: - $k_{c,90} = 1,5$ for solid softwood timber - $k_{c,90} = 1,75$ for glued laminated softwood timber provided that $\ell \le 400$ mm

After calculation, the results from this check are available in the shape sheet for the selected member.



	Shape sheet - Linear element	n°6			×
Se	otion Deflections () Strength	(87%) Stability (87%)	Fire Resistance () Fire stability ()		
		Unfavourable case	Verfication	Work ratio	-
	Share Not-have (6.5)	n°101	Start Notch: r d = 1.5V / (s eff *h ef) <=k v *f v,d. (5.60) 1.250 < 2.215 MPa	56%	
	Sheer Notches (6.5)	n'101	End Notch: 1 d = 1.5V / 15 eff "h ef) <= k v "f v.d. (6.60) 1.250 < 2.215 MPa	56%	
	Surface of a second of 1.5	n'101	Start support: or c.90 d = F c.90 d / A ef <= k c.90 °F c.90 d (6.3) 0.523 < 2.123 MPa	25%	
	econing as supports (s. 1.)	n'101	End support or c.90 d = Fic.90 d / A efic= k.c.90 * fic.90 d (6.3) 0.664 < 2.123 MPa	302	J
_			Edit report Edit detailed report		ĸ

Shape sheet

The detailed version of the shape sheet will also provide the various parameters involved in this verification ($F_{c,90,d}$, A_{ef} , $k_{c,90}$ and $f_{c90,d}$).

Bearing at supports	Case no 101, Mesh No. 6.1
(6.1.5)	Start support: Fc.90.d = 4.186 kN Agr = 80.00 cm ² kc.90 = 1.50 fc.90.d = 1.415 MPa
	$\sigma_{c,80,d} = \frac{F_{c,80,d}}{A_{cd}} \le k_{c,80,d} (6.3)$
	0.523 < 2.123 MPa (25 %)
	Case no 101, Mesh No. 6.4
	End support: F _{c.80,d} = 4.186 kN A _{cf} = 65.00 cm ² k _{c.80} = 1.50 f _{c.80,d} = 1.415 MPa
	$\sigma_{c,80,d} = \frac{F_{c,80,d}}{A_{ed}} \le k_{c,80f_{c,80,d}}$ (6.3)
	0.644 < 2.123 MPa (30 %)

Shape sheet (detailed)

5.3. Verification of beams with a notch at the support acc. EC5

Possibility of performing additional verification for timber elements according to Eurocode - the check of the notch on an element ends.

Advance Design 2024 now manages the support verification of notched beams according to §6.5.2 from EN 1995-1-1.



End-notched beams

A new 'Notches' section has been added to the timber design properties.

- Divide Notches	
E Start Notch	
 Notch location 	On the same side as the support
 Beam start height (hef) 	150 mm
Support to notch distance (x)	100 mm
Inclined part length (Li)	0 mm
End Notch	
 Notch location 	On the same side as the support
 Beam end height (hef) 	150 mm
- Support to notch distance (x)	100 mm
Inclined part length (Li)	0 mm

Notch definition at the start of an element

The 'Notch location' parameter allows the user to specify whether the notch is:

- On the opposite side of the support
- On the same side as the support

A notch located on the same side as the support (i.e., tension side) can lead to brittle failure due to stress concentration at the notched corner. Thus, the design shear strength of the member shall be reduced by a k_v factor. On the contrary, for a beam with the end notch at the compression side (i.e. opposite side to the support), the k_v factor is taken as 1,0.

This k_v reduction factor is determined based on:

- The effective depth of the beam (h_{ef}) at support location
- The distance from the line of action of the support reaction to the corner of the notch (x)
- The notch inclination (i), if any



The α and k_n parameters, also involved in the determination of k_v , are computed automatically during the timber design sequence:

 $\alpha = \frac{h_{\rm ef}}{h}$ $k_{\rm n} = \begin{cases} 4.5 & \text{ for LVL} \\ 5 & \text{ for solid timber} \\ 6.5 & \text{ for glued laminated timber} \end{cases}$

The verification aims to ensure that the shear stress at the notch support (c_d) does not exceed the design shear strength ($f_{v,d}$), reduced by the k_v factor.



$$\tau_d = \frac{1,5V_d}{b_{ef}h_{ef}} \le k_v f_{v,d}$$

After calculation, the results from this check are available in the shape sheet for the selected member.

		Unfavourable case	Verification	Work ratio
		n°101	Start Notch: τ d = 1.5V / (b eff * h ef) <= k v * f v,d (6.60) 0.728 < 2.462 MPa	30%
Shear Notches (6.5)	n°101	End Notch: τ d = 1.5V / (b eff * h ef) <= k v * f v.d (6.60) 0.728 < 2.462 MPa	30%	
E	Bearing at supports (6.1.5)	-	not performed	-
			not performed	-

Shape sheet

The detailed version of the shape sheet also provides the various parameters involved in this verification (k_v , $f_{v,d}$...).

Tension	Case no -, Mesh No,
Compression	6.1: σ _{i0d} ≤F _{i0d} : not done (-)
Shear	Case no 101, Mesh No. 2.1
	6.13: τ _d ≤F _{vd} : 0.485 < 2.462 MPa (20 %)
Oblique bending	Case no 101, Mesh No. 2.3
	6.17: σ _{t0d} / F _{t0d} + σ _{mrd} / F _{mrd} + K _m σ _{mrd} / F _{mrd} ≤1 : 0.48900 < 1 (49 %) Case no 101, Mesh No. 2.3
	6.18: σ _{10d} / F _{t0d} + K _m σ _{mvd} / F _{mvd} + σ _{mzd} / F _{mzd} ≤1 : 0.34230 < 1 (34 %)
Torsion	Case no -, Mesh No,
	6.14: τ _{tor d} ≤K _{shspe} F _{vd} : 0.000 = 0.00 * 0.000 MPa: not done (-)
Shear with Torsion	Case no 101, Mesh No. 2.1
	(τ _d / F _{vd}) + (τ _{tor d} / (K _{shape} F _{vd})) ≤ 1 : 0.19706 < 1 (20 %)
Shear Notches (6.5)	Case no 101, Mesh No. 2.1
	Start Notch: k _y = 1.00 f _{v.d} = 2.462 MPa
	$\tau_d = \frac{1.5V}{b_{ut}h_{ut}} \le k_v f_{v,d} (6.60)$
	0.728 < 2.462 MPa (30 %)
	Case no 101, Mesh No. 2.5
	End Notch: k _v = 1.00 f _{v,d} = 2.462 MPa
	$\tau_{d} = \frac{1.5V}{b_{e}+b_{e}} \le k_{e}f_{e,d}$ (6.60)
	0 728 < 2 462 MPa (30 %)

Shape sheet (detailed)

5.4. Optimization of timber elements according to the deflection (EC5)

Ability to automatically select the optimal timber section due to the deflection condition according to EN 1995-1-1.

Advance Design 2024 can now find the optimal dimensions for timber members based on their deflection results.

New dimensions are searched if the appropriate option is selected on the Timber Calculation setting window, and the current deflection ratio exceeds a limit, set as 100% as default.

NOT FOR RESALE version - Ca	Iculation Settings	×
NOT FOR RESALE version - Ca Verification Optimisation Buckling Calculation Sequence	Iculation Settings Optimisation Optimisation mode by element	×
	Image: Second structure Image: Second structure <th>łelp</th>	łelp

Optimization parameters on the Timber Calculation Settings window

In this dialog, we can define the section size limits and the increments to apply on the base and height of the initial section.

The results of the optimization are given in the Suggested shapes dialog. The members for which the deflection is over the limit are displayed in red. A more appropriate, bigger section is then suggested.

	Element	Cross sections	Strength/stability work ratio	Deflection work ratio	Suggested solutions	Strength/stability work ratio	Deflection work ratio	Accepted solutions
	1	R75*200	52.8 %	57.1 %	R75*200	52.8 %	57.1 %	
10	2	R75*200	57.7 %	61.8 %	R75*200	57.7 %	61.8 %	
	3	R75*200	33.8 %	38.7 %	R75*200	33.8 %	38.7 %	
11 all	4	R75*200	44.7 %	49.2 %	R75*200	44.7 %	49.2 %	
	5	R75*200	50.1 %	54.4 %	R75*200	50.1 %	54.4 %	
	6	R75*150	89.1 %	133.6 %	R85*160	69.1 %	97.1 %	R85*160
ement ection esign template ame system							1	

Suggested shapes dialog

This feature ensures that none of the various deflection components exceeds the limit values defined in §7.2 from EN 1995-161 (or its national appendix).

Shape sheet - Linear eler	ment n°6			>
Section Deflections (97%)	Strength (69%) Stability (69%) Hire Resistance	e () Fire stability ()		
	Unfavourable case	Verification	Wor	k ratio
W inst Q	n*104 : +1x[2 L]	L/352 < L/300 (1.278 cm -	< 1.500 cm) 8	5%
Winst	n*104 : 1x[1 D]+1x[2 L]	L / 264 (1.704 cm))	-
W creep	n*106 : 1x[1 D]+0.3x[2 L]	L / 927 (0.486 cm))	-
W fin		L / 206 < L / 125 (2.190 cm -	< 3.600 cm) 6	1%
W net fin		L/206 < L/200 (2.190 cm -	< 2.250 cm) 9	7%
W inst Q - SE				
Winst - SE				
W creep - SE				
W fin - SE				
W net fin - SE				
ł		Wfin = Winst + Wcreep Wnet,fin = Wfin + Wc		
		Edit report	Edit detailed report	OK

Results of deflection verification for a timber element

6. Enhanced user experience and the comfort of program

operation

A series of novelties and improvements related to user experience, resulting in increased efficiency and comfort of use.

6.1. User templates for Selection

The ability to save and easily select templates with object selection.

The selection of objects is one of the more frequently performed operations. For this purpose, we have several tools available, including selection using the Project browser or using the window to define selection criteria. Starting with Advance Design 2024, we also can save and use previously saved object selections. This makes it easier to quickly select any set of objects at any time during work.

We can save a template with selections for two types:

- saving a selection of objects that is, saving any selection (e.g., graphic) of any objects
- saving selection criteria that is, saving of selection defined by criteria in the Element Selection window.

Templates can be added at any time when working with a structure. Creating a new template is done using the commands at the bottom of the Element Selection window. Using these commands, we can add, rename, and delete a template, and in the case of templates for criteria, we can also export and import templates to a file.

Combination method: Any of selected criteria All selected criteria X Deable all Selection mode: Image: Add to selection Apply to selection Image: Add to selection Image: Apply to selection Templates Image: Selection of objects Image: Provide the selection Image: Provide the selection Image: Provide the selection None Image: Provide the selection Image: Provide the selection Image: Provide the selection	- 🥵 Section Cu - 🎉 Traffic Pan - 🎉 Traffic Lan - 🚜 Crane Run	si Kay	•	 ✓ Select all × Deselect all
Selection mode: Add to selection Apply to selection Remove from selection Templates Selection of objects Officers None New Save Rename X Delete Export Import 	Combination method:	O Any of selected criteria	Al selected ortera	× <u>D</u> isable all
Templates Selection of objects @ Criterta None New Save Rename Delete Export Import	Selection mode:	 Add to selection Replace selection 	 Apply to selection Remove from selection 	
	Templates Selection of objects None	© Criteria ▼ New ✓	Save Rename X Delete	Export K Impo

New options for selection templates on the Element Selection window

Templates for the selection of objects

Object selection templates allow multiple quick selections of any object selection, regardless of how the selection was created and regardless of the type of object. For example, we can graphically select several different parts of a structure (walls, beams, supports, ...) and save this selection as a template. This type of template is based on saving the ID numbers of the elements.

The procedure for adding a template is as follows:

• To begin with, we create a selection. You can use any method (such as a graphical selection of elements) and it can contain various types of objects.



 Then open the Element Selection window (ALT+S) and set the template type to 'Selection of objects' at the bottom.

 Templates © Selection of objects None	Criteria	v Sava	✓ Rename X	Delete +	Epot 4	Import
				ОК	Cancel	Help

• Click on the New button and enter the name of the template

Templates Selection of objects	O Criteria		
None	 New Save 	Rename X Delete Export	+ Import
	Selection Template - New	Cancel	Help
Name:	External Walls	OK	

• The template is saved when the Save button is used or when the window is closed with the OK button.

Selecting a selection from a template can be done in several ways.

The fastest way is to select a template from the list available directly on the ribbon. Once selected from the list, object selection is automatic. As presented in the image below, templates work in both descriptive and analytical models.



A list selection from a dedicated selection toolbar works on a similar basis, and you can also recall a selection using the Element selection window (ALT+S).

-	1	 m	Π.	Q.	邤	ø
_						
		 ·	1 11 10 10 IN		. 176. AD 100 101. AS	. 1999 - 100 100 100 100 100

Renam

Since object selection templates are based on storing the ID numbers of selected elements, it should be noted that modifying the numbering of objects in the model may affect the selection effect of a previously saved template. Templates of this type are available only in the current project.

Templates for selection criteria

The selection template by saving criteria is used to save the current set of criteria set in the Element Selection window. This type of template is not based on ID numbers and thus is more versatile, although it requires a precise definition of criteria. Templates of this type can be used in the current project but can also be saved/exported to an external file and easily loaded and used in other projects.

The procedure for adding a template is as follows:

• First, we open the Element Selection window (for example, using the ALT+S shortcut) and select criteria in the tabs.

- We make sure that the template type is set to 'Criteria' at the bottom of the window (this is the default choice).
- Click on the New button and enter the name of the template.

- Ter C	nplates) Selection of ob one	jeda 🛞 Criteria	Save	A Bename	X Delete	F Expot	1 Inset
		Criteria Template - New		23	ок	Cancel	Help
	Name:	Slabs t22cm C30		OK Cancel			

• The template is saved when the Save button is used or when the window is closed with the OK button.

The application of the template is done in the same way as discussed earlier for object selection templates.



In the case of criteria selection templates, we additionally can save them to an external file. This can come in handy when we define several of our universal templates and want to use them in other projects. To do this, use the dedicated export and import buttons.



6.2. New options for selecting identical elements

Much faster and easier object selection.

To facilitate the graphic selection of elements, a new tool has been added for the quick selection of elements identical to the current selected one. To use the new function, select a single element (linear element / planar element/load area/support/load) and open the right-click context menu. The new 'Select identical' command contains several comparison criteria conveniently selectable from the menu, so you can quickly narrow down your search.



The following criteria are available:

- All main Properties selects elements with the same main properties. The list of main properties depends on the element type:
 - o linear elements: geometry (length), material, and section
 - o planar elements: geometry, thickness
 - o load area: geometry
 - supports: category (Rigid Point, Elastic Point...) and the definition of restraints (values of TX, TY, ...).
 - o loads: category (Point / Linear / Planar) and the definition (values of FX, FY, ...).
- **Geometry** selects elements with the same geometric properties (for example the length for linear elements)
- **Type** selects elements with the same Type property (like S beam or Tie for linear elements)
- Material selects elements with the same material.
- Cross Section selects elements with the same cross-section (linear elements)
- Thickness selects elements with the same thickness (planar elements)
- Name selects elements with the same Name property.
- **Design Template** selects elements with the same general design template.
- System selects elements from the same subsystem.

NOTE: Depending on the selected element, some of the criteria may not be available. For example, the Thickness criterion is available for surface elements, but not for supports, loads, or linear elements.

6.3. Possibility for filtering inactive elements

Faster search for inactive items.

To facilitate the search for elements that have been excluded from the generation of the calculation model (i.e., have the Active state property turned off), a new filter has been added to the Element Selection window. This filter allows you to quickly select elements whose state is active or inactive.



Faster search for inactive items

6.4. Filtering profiles from databases

Faster profile search from databases.

GGRAITEC

To increase the speed and convenience of finding the cross sections you need, several improvements have been made to the window for selecting profiles from databases.

The first change is the addition of a filter by section type. The filter in the form of a drop-down list contains a set of available profile types, for example Rectangular, Rectangular hollow, U, C, T, Z, Zeta, etc. Selecting a type from the list automatically narrows down the list of the displayed profile libraries.



Another improvement is the addition of a text filter. The principle of operation is simple - you should enter any string of characters (it can be the beginning or any fragment of the library name) and press Enter. The list of available libraries will be automatically narrowed to those which contain the entered text in their names.

	NOT FOR RESALE version - Cross section libraries									
Library manager Section-Type Name Br European HEA HEA	All THEA			10.	00		OK <u>C</u> ancel <u>H</u> elp Type of laminati	on		
⊡. Graitec Pro ⊡. Non-Co HE HE	nnes ompound Profiles ;A :A - half DIN 997 :A - half DIN 18800-1			0.80	0.50		Rolled		•	
HE	A DIN 997			Designation	Area	ly		lz		
HE	AA		►	HEA100	21.24 cm ²	349).20 cm4	133		
HE	HEAA - half DIN 997			HEA120	25.34 cm ²	606	5.20 cm4	230		
HE	AA - hait Din 18800-1			HEA140	31.42 cm ²	103	3.00 cm4	389		
His	Span Header			HEA160	38.77 cm ²	167	73.00 cm4	615		
				HEA180	45.25 cm ²	251	0.00 cm4	924		

In addition, to make it easier to search for cold-formed steel profiles in the *Graitec Profiles* database, such profiles have been separated into a new group 'Cold-Formed Profiles'.

		NOT FOR RESALE ver	sion - Cross section lib	raries		8	23
Library manager							
Section-Type	All	•			ОК		
Name					Cancel		
⊡ European	Profiles				<u>H</u> elp		
USER	ons				Type of laminatio	n	
Graitec Pro	ofiles				Rolled	-	ור
⊞ ·· Non-C ⊞ ·· <mark>Cold-F</mark> ⊞ ·· Compo	ompound Profiles ormed Profiles ound Profiles	←			Rotation		
			Designation	Area	lv.	17	

It should be noted that the cold-formed profiles in this group are profiles that contain a set of data required for verification calculations according to steel standards. However, in the *Graitec profiles* database, in the group 'Non-Compound Profiles' there are also other profiles which by their shape and name suggest that they are cold-formed profiles. However, these profiles cannot be verified according to the standard due to a lack of necessary data for verification calculations.

6.5. New commands on the ribbon for adding stages

Easier definition of construction stages.

To make it easier to define construction stages, four commands have been added to the Analysis ribbon:

- Add stage adds a single new construction stage.
- Add multiple stages adds many new construction stages (based on the number entered)
- Automatic generation automatically generates construction stages one for each story.
- Assign to stage calls a window to assign selected objects to stages.



6.6. Improved mesh generation for Pushover analysis

For the elements with plastic hinges, the finite element mesh is now regular, which significantly improves the mesh distribution in the adjacent surface elements.

The modeling of plastic hinges in line elements is enhanced in Advance Design 2024. Previously, to capture plastic rotations/translations of hinges, additional nodes were automatically generated on both sides of the plastic hinge. These nodes disturbed the regular meshing of structural elements, especially when linear elements were in contact with surface elements and increased the total number of Degrees of Freedom (DOF) in the structure. Better modeling of plastic hinges in Advance Design 2024 made it possible to capture plastic rotations/translations without the need for these additional nodes.



Example of plastic hinges modeling on beams (blue dots – plastic hinges, red dots – additional nodes)

6.7. The next stage of unification of dialog windows

More convenient operation in the program thanks to a clearer interface with a uniform window design.

In this version of Advance Design, another set of dialog windows has been updated, giving them a unified appearance. Although the layout and content of the windows have usually remained unchanged, the modifications concern the appearance and the components used. These changes have two main goals - to standardize the appearance of the content of windows to improve the user's perception during everyday work, as well as to modernize the program by switching to the use of newer technological components.



6.8. Improvements to the selection by criterion window

Improvements to the Element selection window for easier choosing selection methods.

The window for selection by criteria (opened from the ribbon or via the keyboard shortcut ALT+S) has been slightly expanded to expand the possibilities and make the work easier.

The first improvement is the change of the window content selection from tabs to a list. These speeds up the selection of criteria categories and eliminates the problem of scrolling through the tabs, which occurred especially with long names for some language versions.

The second change is new selection modes and a clearer definition of criteria combinations. For this purpose, several options have been added at the bottom of the window.

	N	OT FOR RESALE version - Elem	ents Selecti	ion	53
Types Materials Cross sections Thickness Coordinates Names Groups Estimates Design Templates Systems Design parameters Work Ratio	Kontesential	DT FOR RESALE version - Elem stem ve tents two node raint k ment r Super Bement r Super Bement	ents Selecti	ion Identifiers: Systema: None GTC Identifiers: Super elements identifiers: Active state :	23
	Combination method: Selection mode: Templates Selection of objects None	Any of selected offerla Any of selection Replace selection Offerla New Vitterla	 AI ○ Ap ○ Re Save 	selected oritera ply to selection move from selection Rename × Deleta OK	× Destelect an × Destelect an e ▶ Export ← Import Cancel Help

Improved window for selection by criteria

The first set of options is called the Combination method. There are two choices:

- Any of the selected criteria it is a previous 'union' method. It selects all elements which correspond to at least one of the defined criteria. That is, for example, when you choose two criteria: Type=*Linear* elements and Material=*Concrete*, then with this method all linear elements (including non-concrete elements) and all concrete elements (including planar elements) will be selected.
- All selected criteria it is a previous 'intersection' method. It selects only the elements which correspond to all defined criteria at the same time. That is, for example, when you choose two criteria: Type=*Linear* elements and Material=*Concrete*, then with this method only linear concrete elements will be selected.

The second set of options is called the Selection mode. These are used when the window for selection by criteria was called to create a new selection when any object selection already existed in the model.

There are four choices:

- Add to selection adds new selection to the existing one. This is a default option.
- **Replace selection** replaces the existing selection with a new one.
- **Remove from selection** removes new selection from the current selection.
- Apply to selection applies new criteria to the existing selection.

G GRAITEC

7. Other novelties and small improvements

Several various new features and small improvements introduced in the newest version of Advance Design.

7.1. New database with cold-formed steel profiles for North America

Ability to model structures using cold-formed profiles typical of the North American market.

A new list has been added to the list of profile databases – **North America Cold-Formed Profiles**. It includes cold-formed steel profile data from tables I-1 to I-8 from the *AISI Cold-Formed Steel Design Manual (2017 edition*).

There are 8 profile families available:

- > C-Sections with Lips
- C-Sections with Lips (joist/studs)
- > C-Sections without Lips
- Z-Sections with Lips
- Z-Sections without Lips
- > Equal Leg Angles with Lips
- > Equal Leg Angles without Lips
- > Hat Sections without Lips

	NOT FOR R	ESALE ver	sion	- Cross section li	braries		la 1	3
ibrary manager								
Section-Type	All	-		Т (7		ОК		
					0.0			_
Name					<u> </u>	<u>C</u> ance		
FI- European	Profiles			2	+r	Help		_
	ons			2		<u> </u>		
	rica Timber Profiles			1	ят	Type of laminat	ion	
⊡ · North Ame	rica Cold Formed Profiles					Cold-Formed F	Rolle	÷
···· C-Sect	tions With Lips				1.60 40	Cold Formed F		-
···· C-Sect	tions With Lips (joist/studs)				4			
··· C-Sect	tions Without Lips				2			
Equal	Leg Angles With Lips			D		1		
Equal	Leg Angles Without Lips			Designation	Area	ly	IZ	-
	tions With Line			4CS4x105	8//.4/mm²	1610813.97 mm4	121	
- Z-Sect	tions Without Lins			4CS4x085	709.72 mm²	1323614.58 mm4	986	1
Fl- Graitec Pro	otiles	- 1		4CS4x070	584.55 mm²	1098849.84 mm4	815	2
				4CS4x065	542.61 mm ²	1023928.26 mm4	/5/	
				4C54x059	492.29 mm²	936519./5 mm4	686	
				4CS2.5x105	6/1.01 mm²	1111336.//mm4	389	
				4CS2.5x085	545.19 mm ²	915/08.20 mm4	317	
				4CS2.5x070	449.06 mm ²	761702.73 mm4	262	
				4CS2.5X065	4 16.80 mm ²	/11/55.01 mm4	243	
				40.52.5X059	3/8.09 mm*	049320.36 mm4	221	
				40.52X105	618.10 mm*	35/331.30 mm4	239	
				40.32X083	412 29 mm ²	/000/0.00 mm4	100	
				40.32X070	412.20 mm ²	63/644.36 mm4	162	
				40.32X003	302.00 mm²	561011 05 mm 4	127	,
				4C32X033	1012.96 mm ²	4012466.84 mm4	1/1	
				6CS4x105	919 /0 mm ²	2270000 20 mm 4	114	
				6CS4x065	677 /6 mm ²	2717988 / 3 mm /	944	-
				0034X070	077.40 000-	2717300.43111114	344	P
				1111				

7.2. New steel material database for North American cold formed sections

Ability to model structures using steel materials for North American cold formed sections.

To the material library, a new standard has been added to the STEEL family list: *ASTM Cold Formed*. It contains new materials for use for North America cold-formed profiles. Each material has a specific application which is described in detail in Table 1.2 of the *AISI Cold-Formed Steel Design Manual*.

Family STE	EL 🔹	Standard	ASTM Cold Fo	med		-
A1003 ST33H	H A1008	33 Type 2	A1008	HSLAS 60 Clas	s ,	
A1003 ST37H	A1008	40 Type 1	A1008	HSLAS 65 Clas	s 🔺	Import
A1003 ST40H	A1008	40 Type 2	A1008	HSLAS 65 Clas	s 🔚	
A1003 ST50H	A1008	HSLAS 45 Cla	ISS1 A1008	HSLAS /0 Clas	s 👻	Export
A 1003 S 155H	A 1008	HSLAS 45 Cla	ISSZ A1008		s	
A1003 ST5/H	Δ1008		ee2 Δ1008	HSLAS-F 50 CI		Delete
A1003 ST70H	A1008	HSLAS 55 Cla	ss1 A1008	HSLAS-F 70 CI	a	
A1003 ST80H	A1008	HSLAS 55 Cla	ss2 A1008	HSLAS-F 80 CI	a	
A1008 33 Typ	e 1 A1008	HSLAS 60 Cla	iss1 A1008	SS-25		
4						

7.3. New parametric section – spaced shafts

Ability to easily define wooden compound profiles.

A new type - Spaced Shafts - has been added to the list of profile types that can be defined in a parametric way.

It allows the modeling of a profile consisting of 2, 3, or 4 identical rectangular sections, set in parallel at the same spacing. Profiles with two elements are most often used to model wooden roof elements, while 2, 3, and 4 element configurations are usually used to model wooden columns.

NOT FOR RESALE version - Defined 🔋 😒				NOT FOR RESALE version - Defined				P 23	
<u>T</u> ype:	II Space	ced Shafts	•	<u>о</u> к	<u>T</u> ype:	II Spaced	Shafts	•	<u>O</u> K
Subtype :			•	Cancel	Subtype :			-	Cancel
Label:	2R33*11	0 33			Label:	3R33*110 3	3		
Description		Value	33	00.	Description		Value		
Number of ident	tical s	2	Т		Number of ide	entical shafts	3 🔹	33.00	
Height (mm)		110.00		•	Height (mm)		110.00		
Width (mm)		33.00			Width (mm)		33.00		
Spacing between sha		33.00	9		Spacing betw	een shafts (33.00	8	
Color		FFCC00	1		Color		FFCC00	01	
Type of lamination		Welded	-		Type of lamin	ation	Welded		
				<u>33</u> .00				33.00	

The axis convention used in Advance Design is with the local z axis parallel to shaft height (h).





Axes convention for spaced columns



Axes convention for spaced beams

The number of shafts and spacing between them have an impact on the section parameters of the member:

Area for n shafts:

$$A_{tot} = n \cdot A = n \cdot b \cdot h$$

In-plane inertia for n shafts:

$$I_{y,tot} = n \cdot I_y = n \cdot \frac{b \cdot h^3}{12}$$

Out-of-plane inertia

for 2 shafts:

$$I_{z,tot} = \frac{h \cdot [(2 \cdot b + a)^3 - a^3]}{12}$$

for **3 shafts:**

$$I_{z,tot} = \frac{h \cdot [(3 \cdot b + 2 \cdot a)^3 - (b + 2 \cdot a)^3 + b^3]}{12}$$

for **4 shafts:**

$$I_{z,tot} = \frac{h \cdot [(4 \cdot b + 3 \cdot a)^3 - (2 \cdot b + 3 \cdot a)^3 + (2 \cdot b + a)^3 - a^3]}{12}$$

7.4. Improved combination creation with seismic cases for Canada

Automatic consideration of the occurrence of opposite directions of seismic forces for Canada.

During the automatic generation of combinations with seismic cases according to the Canadian NBC standard, seismic combinations are now considering both directions for forces from seismic load cases (EX, EX, EY). Now seismic cases are considered in combination once with a positive sign and once with a negative sign.

This prevents the problem of force cancellation when we have concomitant loads on the structure, for example, soil pressure in one direction and seismic force in the opposite direction.

GRAITEC
Project Browser		₽ × USER v	iew							
ADVANCE DESIGN										
A A A								Cor	nhinati	0.04
Model	Combinatio	ns Concrete Steel	Timbe	r						
Loading	ld.	Туре	Case	Coeff	Case	Coeff	Case	Coeff	Case	Coeff
Dead Loads	101 102	ULS ULSA	1 1	1.40 1.00	2	1.00				
E 🚱 Seism NBC 2015	103	ULSA	1	1.00	2	-1.00				
54 2 - EX	104	ULSA	1	1.00	3	1.00				
	105	ULSA	1	1.00	3	-1.00				
4 - EZ	106	ULSA	1	1.00	4	-1.00				

7.5. Defining Link at node on selection

Better control over modeling with a precise selection of the elements to be linked.

An additional method of quickly defining multiple links at node at the same time has been introduced. Now, after calling the command 'Link at Node' in the command line, you can choose to define it by selecting elements (by pressing x on the keyboard).

Comman	nd l	.ine										
Press	р	to	input	а	point	or	press	х	to	select	elements	>

You can then graphically indicate the main and secondary elements separately. You can include multiple main elements and multiple secondary ones.

Command Line
> Select primary elements and press enter > 1
Command Line
> Select secondary elements and press enter > 5 4 2 3 7 8 9 6

This allows for a precise and clear definition of links.





7.6. Possibility of material definition for supports

Ability to set the type of concrete for the foundation in the 3d model.

The option to assign a material (concrete type) to supports has been introduced. It is available in the list of properties of all types of punctual and linear supports.

Properties	₽ ×
🗐 💽 🚡 All properties	•
E General	
- Identifier	5
— Name	Rigid Point Support
 Active state 	Enabled
- Systems	0
- Comment	
- GTC Identifier	0
Footing type	None
Coordinate System	
- Option	global coordinate system/us
Coordinate System	1
E Footing Material	
🖵 Material	C25/30
El Footing Dimensions	
- Width A	1.00 m
- Width B	1.00 m
- Height H	0.30 m

Assigning material to a foundation currently has two uses. First, together with information on foundation dimensions, it allows cost and CO2 emissions estimation. Secondly, it is used when exchanging information between the Advance Design model and the RC Footing design module.

7.7. Dedicated system for master-slave links

Better organization of system content by grouping automatically generated master-slave connections.

One of the existing configurations of the program is the ability to automatically generate master-slave connections in the slab, to avoid the effect of point concentration of forces over a column or support.

Smoothing of extreme values		
Smoothing of extreme values of the following elements:		
Point Supports	× /	
☑ Linear elements (Columns)		
The distance of the smoothing area with respect to $X = 0.5 \times h$ the element:		

Such connections are generated automatically when preparing the calculation model, and from this version are also automatically placed in a dedicated system in the Project browser window. This increases order and clarity of the project structure.





7.8. Improvements to the program defaults for North America

Better customization of some of the program defaults when selecting the localization of the US or Canada.

To make things easier for users who have chosen US or Canada as their localization in the program configuration, a few minor changes have been made to the default settings.

The first change concerns printing parameters. Now the default paper format for printing for the US and Canada is Letter format instead of A4.

NOT FOR RESALE version - Document prope ?							
Report <u>N</u> ame FTDoc38_ndc01.rtf							
Custom	ized margins	Paper size					
Тор	1.75	Letter					
Bottom	1.75			_			
Left	1	Width 0					
Right	1	Height 0					

Another change concerns the default design settings. For both countries, the default material has been changed, and for the US, additionally, the default section and units of the project.

	1	y and y
		* + +
v 💌		
		-
00 ~		
R12*20 ~		U.
	l U	
M. DE.		
	00 ~ R12*20 ~	00 V R12'20 V



7.9. Shorter time of generation of load combination

Faster generation of load combinations.

The algorithm for generating load combinations using the concomitance matrix has been improved so that for large tasks the combinations generation time has been reduced.



7.10. Improvements in localization for Spain and Portugal

Full interface translations in all modules are provided for Spain and Portugal.

The translations for Spanish and Portuguese languages have been improved and expanded on missing areas (like the Section editor).



Section Editor

8. Design Modules - Common improvements

New features and improvements common to design modules.

8.1. Allow applying project templates when the module is already opened

Ability to update settings from a template for a currently open project.

One of the ways to speed up and streamline your work while designing the next element is to use project templates with our predefined settings. Project templates can be selected when creating a new project for an element (for standalone versions of design modules) or before exporting/opening an element in a design module in the Advance Design environment.

Starting with the newest version, it is possible to apply a project template to an already opened element. That is, at any time during the design, for example, after opening or importing an element from Advance Design, we can use a new command to apply a template.

When working in the standalone version of the modules, the 'Apply template' command is available from the menu. When working in the Advance Design environment, the new command is available on the ribbon.



Location of the new command in the standalone version (left) and on the ribbon in Advance Design (right)

Similarly, when starting a new project with a template, when assigning a template to an open project, we can also select the scope we want to overwrite from the template.

🔁 Apply template		×
Template File		
My template for footings	~	Browse
Apply from template Project Settings and User Data Geometry Design Assumptions Reinforcement Assumptions Reinforcement Diameters Soil Assumptions Load Cases & Combinations		
	OK	Cancel

Selecting the scope of application of the template for RC Footing module

This functionality is available for RC Beam, RC Column, RC Wall, RC Footing, RC Slab, and Masonry Wall modules.



8.2. Display the steel ratio in the Info Panel

Quickly available information on steel ratio.

One of the basic qualitative results regarding the reinforcement of concrete elements is the reinforcement steel ratio. Until now this result has been presented only in the report and in the drawings. With the latest version of the program, this value is visible immediately after the calculation and also in the info panel window, making it possible to check it quickly.

In all RC modules (Beam, Column, Foundation, Wall, and Slab), the global steel ratio value for the entire element is visible at the bottom of the Info Panel.

Verifications	Тор х	Top value	Top WR	Bottom x	Bottom value
σ_crq Concrete	3500 mm	2,8 MPa	11,38%	0 mm	0,5 MPa
σ_crq Steel	0 mm	48,2 MPa	12,06%	3500 mm	157,5 MPa
σ_qp Concrete	3500 mm	2,8 MPa	11,38%	0 mm	0,5 MPa
Calculation results Frrors and w	arnings		1		Steel ratio = 51.7 kg/n

Calculation results Errors and warnings

In addition, for multi-span beams, the corresponding steel ratio value is presented in the header separately for each span.

					(Steel Ratio = 108	,4 kg/m³)				
Reinforcement	x	Design effort	Resist. effort	Comb.	Reinf. min.	Reinf. theo	. Reinf. real	Work Ratio	x	Desigr
Longitudinal top left	0 mm	-28 kN ⋅m	-35,5 kN⋅m	104	178 mm²	178 mm²	180 mm²	78,94%	0 mm	-96,2
Longitudinal top right	7000 mm	-92,6 kN·m	-123 kN·m	104	178 mm²	508 mm²	653 mm²	75,26%	7000 mm	-17
Longitudinal bottom	3010 mm	112 kN·m	119,6 kN·m	104	178 mm²	596 mm²	628 mm ²	93,66%	4200 mm	68,1

8.3. Small improvements

A set of smaller improvements affecting many design modules.

Better notification of report generation •

In order not to miss the information about the generation of the report, the small window indicating the generation of the report is now always visible on top. In addition, it can be easily moved to a new location.



• Grouping icons on a dedicated ribbon in Advance Design

On the Design Modules ribbon in the Advance Design environment, some of the commands have been grouped to make it easier to find the right commands, especially when the ribbon is short for smaller screen resolutions.

Save Template As - Settings - A	Project Settings*	Wat Section	Reinforcement Diameters	
Save Template As	C Project Settings	Wall Section Wall Sections Database	Reinforcement Diameters	
		Onits List		

• Multiline descriptions on drawings

When adding custom annotations on drawings, it is now possible to enter text with multiple lines (using the Enter key).



9.RC Beam

New features and improvements implemented in the latest version of the RC Beam module.

9.1. Import geometry and internal forces from Rib design

Consideration of effective section and design internal forces when importing beams with rib option active.

One of the options available for reinforced concrete beams supporting a concrete slab is the ability to consider them as ribs. Activation of this option causes that during the design analysis of a reinforced concrete element, the interaction of the beam with the slab is considered, and the design efforts are not just the forces occurring only in the rectangular beam part but are the combinations envelop of FEM (Finite Element Method) internal forces occurring on the entire monolithic T or L section.

Beam calculation					
 To calculate 	☑ Enabled				
- Reinforcement					
■Ribs design					
🚽 🗖 Ribs design	🛛 🗹 Enabled				
Effective width	Definition				
 Calculate deflection 	Disabled				
- Allowable defl. = 1/	250				
Simple bending Disabled					
Ribs design option for reinforced concrete beam elements					

With Advance Design 2024, when a rib beam is exported from Advance Design to RC Beam module, instead of a rectangular beam section, the equivalent monolithic T or L section is imported.



Initial rectangular section (left), effective section on Advance Design (middle), and imported section (right)

Also, instead of the FE internal forces from a rectangular beam, the equivalent integrated design internal forces are imported.



Design bending moment for the entire monolithic rib beam in Advance Design



Design bending moment for rib beam imported to RC Beam module

9.2. Possibility to define the position of supports (columns/ walls)

Better quality of drawings due to showing the real support position.

Supports (columns and walls) now distinguish whether they support a beam from below, above, or from both sides.

For this, an additional option is available on the 'Geometry' tab, where we can set the position independently for each support. This data is now also imported from the Advance Design model.



🔁 Geometry [NOT FOR RESALE versi	on]			×
 Span 1 (Cast-in-Place) Span 2 (Cast-in-Place) Main Geometry Additional settings 				
Section Openings Depressions	z bi bi	In	Section	b2 bw
Opened Links	Name	T 1.2	Type Cast-in-	Place ~
Secondary beams	Upper fiber offset (dz): 0 mm	Width	(bw): 200 mm
	Length ([): 2050 mm	Height	(h): 500 mm
	Left Support		Right Support	
	Name		Name	
	Туре	Column ~	Туре	Column ~
	Width (b1):	: 300 mm	Width	(b2): 300 mm
	Beam Height	550 mm	Beam Height	550 mm
	Beam eccentricity (z1)	: 0 mm	Beam eccentricity	(zz): 0 mm
	Eccentricity (e1)	0 mm	Eccentricity	(e2): 0 mm
	Position	Bottom + Top 👻	Position	Top only ~
ADVANCE DESIGN		Bottom only Top only Bottom + Top	Ok	Apply Close 👔

New options for the supports position on the Geometry dialog

The main purpose of this new possibility is to get correct drawings, where we can see the position of support (wall or column).



Example of a drawing with an elevation of a two-span beam with different support configurations

9.3. New support type "beam"

Possibility for generating correct drawings when the support is another beam.

With the latest version of the RC Beam module, it is possible to define as support another beam (perpendicular beam that acts as a support). For this, on the 'Geometry' dialog, the list of support types is expanded with a new entry - *Beam*. This data can be edited manually, but also, when importing

data from Advance Design, beams supporting an element (beams of higher height) are recognized as supports of beam type.



This enables the correct generation of drawings when the support is another beam.



Drawing for a beam elevation with the center support modeled as a perpendicular beam

9.4. Ability to define secondary beams

Possibility for generating better drawings by considering the position of the secondary beams approaching perpendicularly.

In the latest version of the module, secondary perpendicular beams can be defined. Such beams have a lower height than the analyzed beam and are not treated as a support.

Secondary beams are added using a new tab in the Geometry window.

GRAITEC



New tab for defining secondary beams

NOTE: When importing data from the Advance Design model, secondary beams are automatically detected and added to the list of secondary beams.

Secondary beams can be seen on the views and reinforcement drawings.



9.5. A new chapter in the report on openings reinforcement

Easier verification of required reinforcement for openings.

In the detailed report for beams, if there are openings in the beam, there is now a new chapter on the reinforcement of openings.



View of the beam with opening reinforcement

16 Openings reinforcement (required reinforcement)

Openings Reinforcement									
Span Opening	Lintel	At	Tu	A _{top}	A _{bot}	A _{LL}	A _{LR}		
opan	Opening	Lance	(mm²/m)	(MPa)	(mm²)	(mm²)	(mm ²)	(mm ²)	
,	1	Тор	792	1.1	53	208	196	167	
1		B	Dottom	7.42	1.2	156	0	130	107

New table in report for opening reinforcement

9.6. New diagrams for reinforcement for torsion

Easier verification of reinforcement for torsion.

In the diagrams with information on the theoretical area of reinforcement now we can display two new diagrams – for the longitudinal reinforcement from torsion (distributed on top and bottom beam faces or distributed on all beam faces) and for the transversal reinforcement from torsion.



9.7. Small improvements

Improvements to Reinforcement assumptions dialog

For improving the user experience, two types of changes are applied to the Reinforcement assumption dialog – splitting of selected tabs and providing more pictures with explanations.

Splitting tabs was aimed at better grouping options to find them more quickly, as well as reducing the need to scroll through the contents of the window. These changes concern three tabs: Transversal

bars, Anti-crack bars, and Anchorage. New explanatory pictures have appeared at the top of some tabs as well as in some selection lists.

Reinforcement Assumptions [NO	T FOR RESA	LE version]				
	10	A-A	B-8	^-+	B ≠I	
Longitudinal Bars 4 Transversal Bars General Settings Additional Settings Assembly Bars		Number of Bars	w +	A	B+	
 Anti-crack Bars 						
General Settings		Condition			Bottom	Top on supports
Additional settings		0 mm	≤ Width <	170 mm	2	2
Splice Bars					-	
Sliding & Strut Bars		170 mm	≤ Width <	270 mm	3	3
Continuity Bars	_	270 mm	≤ Width <	370 mm	4	4
 Anchorage General Settings 		370 mm	≤ Width <	470 mm	5	5
Anchorage at supports		470 mm	≤ Width <	570 mm	6	6
Seismic Assumptions	_	570 mm	≤ Width <	670 mm	7	7
Bar Splitting		670 mm	≤ Width <	770 mm	8	8
 Bar Constraints 		770 mm	< Width <	870 mm	0	9
Number of Bars			- 11001 5		,	-
Number of Lavers		1870 mm	is Müdele z	1000 mm	10	10

The Reinforcement Assumption dialog with split tabs (left) and one of new picture (top)

Drawing – improvement to side dimension lines for reinforcement

Dimension lines for bottom beam bars are now referenced from the inner edge of the support, making it easier to place bars accurately when the beam is supported by walls.



Modification of the default Wmax value for France

When the French national annex to Eurocode 2 is selected, then the max crack value (Wmax) for class XA3 is now set to 0.2 mm. The change was done according to the recommendation from the FD P 18-717 (published in August 2021).

GRAITEC

C Design Assumptions [NOT FOR RESALE version]						
	Cracking					
	Exposure class	ХАЗ ~				
General Design Beam Design	W max	0,2 mm				
Reinforced Concrete	Imposed crack width	0,2 mm				
Cracking	Imposed steel tensile stress limit	0 MPa				
Concrete Covers	Load duration	Long term v				

Adjustment of waterproofing data entry for France

To increase the comfort and clarity of the data, several minor modifications have been made related to the input of waterproofing data (available on the Longitudinal Bars tab of the Reinforcement Assumptions dialog and a new Additional settings tab on the Geometry dialog).

Waterproof Coating (available for im	permeable coating only)	
Provided reinforcement		Ø>9 mm or spacing>150
Automatic determination		\checkmark
Settings	on the Reinforcement Assum	ptions dialog
Connection Direct FOR RECALE and an		
Geometry (NOT FOR RESALE Version)	0	,
Span 1 (Cast-in-Place) Main Geometry Additional settings Section Openings Depressions		be

Span 1 (Cast-in-Place) Main Geometry Additional settings Section Openings Depressions Opened Links Secondary beams			bc	
Beam 1	Coating		Seismic Geometrical Constraints	
Beam 2	Type Coated faces	Impermeable ~ Upper ~	Column dimension (bc): 30 cm	

Settings on the Geometry dialog

The additional coefficient for modification of longitudinal reinforcement

A new coefficient has been added to the beam longitudinal reinforcement settings to easily modify the required theoretical area of longitudinal reinforcement. The amplification factor is applied only to the theoretical reinforcement area resulting from the loads in the vertical plane (not for torsion). It applies to both bottom and top theoretical reinforcement.

CRainforcement Assumptions (NOT FOR RE	SALE version]	,	×
Empirical Bas Construction Bas Convertings Additional Settings Additional Settings Austribly Bata Anth-crack Ban Splice Bas Splice Bas	x x → y Different number of bars in last bar layer Beam monol this with its supports Detailed calculation for each half-span Extend bottom layers across the entire span length Impose bottom bars minimum length	AA	~
Seismic Assumptions Bar Solitting	Longitudinal reinforcement amplification factor	1.00	
Bar Constraints	Generate torsion bars on all beam faces	One package -	

Possibility to choose the way of distribution of reinforcement for torsion

A new option is added to the Reinforcement Assumption window to decide if distribute the theoretical reinforcement area from torsion on the entire length of the beam or to generate two different packages (having different diameters / different numbers of bars) on each half of the beam. This might be helpful when the theoretical reinforcement area for torsion on one half is dramatically different (much smaller or bigger) than on the other half. Note that the torsion bars will be generated with two packages only when they are needed on the lateral sides of the beam due to the torsion. Note also that if bars are needed also for anti-crack reasons, full-length bars will be generated.

The new option is available for Eurocode and NTC standards.



New option for torsion reinforcement separation



Torsion reinforcement in two groups

Improvement in defining suspended loads with link distribution

When a concentrated load is defined as suspended, but the Depth (h') value is not entered, the load is nevertheless applied as suspended if the Links distribution option is selected, assuming that it is above the middle fiber (and thus it is not necessary to reassemble the load).



GRAITEC

Support conditions on the Design Assumptions dialog

To the Design Assumption dialog, a new Support Conditions tab has been added. It contains the options used for defining left and right-end support conditions, which were available on the ribbon of the standalone version of the module. This allows modifying such data when working in the Advance Design environment.

ESALE versio	on]		
-9	Left Support Conditions		
	Method β : Minae ~	β1 value 0,25	
	Right Support Conditions		
	Method	Value	
	Degree or restraint .	80%	
	19	EALE version] Left Support Conditions Method Right Support Conditions Method Degree of restraint	ESALE version] Left Support Conditions Method β1 value β + M max v 0.25 Right Support Conditions Method Value Degree of restraint v 80 %

10. RC Column

New features and improvements implemented in the latest version of the RC Column module.

10.1. Different upper beams geometry

Possibility to individually specify the length and width of the top beams and to use that data for automatic determination of buckling lengths.

In the column geometry definition window, the scope of definitions for top beams has been expanded. For this, a new tab dedicated to these parameters has been added. Now, in addition to the possibility of defining the height and selecting the number of beams, the possibility of individual definitions of the width of beams and their length has been added.

Geometry [NOT FOR RESALE	versionj		
Geometry	Upper Beams		V
Upper Column	Height		(h): 300 mm
Upper Beams Buckling length		Width	Length
	Front	✓ 300 mm	300 mm
	Back	✓ 300 mm	300 mm
	Left	✓ 300 mm	300 mm
	Right	✓ 300 mm	300 mm
	Display real beam I	engths	
	Content of th	new tab	

Importantly, this information is now also imported from the Advance Design model along with the rest of the information for the column.

Beam geometry data from the new tab is now also automatically transferred to the window for the automatic determination of column buckling lengths if Auto is selected as the calculation method. In this case, moments of inertia and beam lengths are automatically filled in according to the geometric data. To be able to edit the values independently of the geometric data, you can disable the automatic synchronization option at the top of the window.

C Geometri (NOT EOR RESALE ini	tan'ny fanisa	Buckling Length Calculation (NOT FOR RESALE version)
	Column Height	Automatic adjacent beams parameters
Geometry Upper Column Upper Beams Buckling length	Column height 0-0: [3100 mm] Backling Length Along X Calculation method Auto	
	200 200 <th>E 67500000 B In 67500000 C</th>	E 67500000 B In 67500000 C
	$\begin{array}{c} \begin{array}{c} \\ \\ \\ \\ \\ \\ \\ \\ \\ \\ \\ \\ \\ \\ \\ \\ \\ \\ \\$	I to 0 mm ⁴ I to 0 mm
	Ok Apply Close D	

Data completion during automatic determination of buckling length

10.2. Possibility to disable the interaction curves checks

Ability to better customize verification.

On the Column Design tab of the Design Assumptions window, there is a new option to deactivate the verification of column resistance using interaction curves.



10.3. Minor improvements to ribbons

Separation of icons for defining geometry and specifying section type.

To accelerate the selection of options for geometry definition, two geometry-related icons are now separated on the ribbon of the standalone application - the first for selecting cross-section types, and the second for calling the window for defining other column geometry parameters.





10.4. Minor improvements in Info Panel

Faster information on the consideration of second-order effects.

On the Info Panel there is now displayed a piece of short information on whether the second-order effects are not included, or are included in both planes or in one plane (XOZ, YOZ).

Max Mx		104: 1.35s[1 G]+1.5z[2 Q]			13,8 kN·m			
Max My			110: 1.35	(1 G]+1.5z[3 V]+1.05z[2 Q]		-99,5 kN-m		
Buckling			хог			YOZ		
Buckling coefficient			1			0,7		
Buckling length			2600 mm			1820 mm		
Slenderness				25,73			21,02	
				Second order	effects are taken int	to account in both planes		
Reinforcement	Rea	ы	Theoretical	Ratio	0	ombination	Amin	Amax
Longitudinal top	1963 r	nm ²	1511 mm ²	76,95%		101: 1x[1 G]	210 mm ²	4200 mm ²
Longitudinal bottom	1963 (num ²	1511 mm ²	76.95%	108 1.3	5x[1 G]+1.5x[3 V]	210 mm ²	4200 mm ²

10.5. Improvement to transverse reinforcement on Edit dialog

Quickly and precisely define and edit the distribution of transverse bars in the table with new definition methods.

In the latest version of RC Column module, the possibility of specifying the method of distribution definition has been added, which significantly increases the possibility of editing and custom defining the distribution of transverse bars.

For this purpose, in the Reinforcement editing window, on the Transversal Reinforcement tab, a dropdown list 'Define distribution by' has been added, which contains four definitions: Quantity, Quantity and Length, Spacing, Spacing, and length.

Reinforcement [NOT FOR RESALE version]								×
66	Transversal distrib	ution						
D. Longitudinal Beinforcement	18 b	ars from 50 mr	n to 2250 mm (Col	umn height – 2600) mm)			
 Transverse Reinforcement 	General Settings							
Main Reinforcement Top Starter Bars	Bar mark				3	02		1
Bottom Starter Bars	Ber colour				<u> </u>	+		Ŧ
	Diameter				H8 ~	_Q3x53		в
	Configurations				Steel Binders	÷ _ ∎		A [‡]
	Along a				Stimups ~	• •		т
	Along b				Stimups -	- L - L		12
	Define distributio	n by			Quantity ~	Q2x52		
	Distribution				Quantity	I		
	Excertion length	Marte			Quantity and length	l î		+
	B u	all record		10.11	Spacing Spacing and length	1		
	Bottom offset			(01):		- T M - H		<u> </u>
	Top offset			(O 2):	350 mm	I		
	Add Del	Package	Q	5	L		A-A	
	+ ×	1	6	100 mm	600 mm			-
	+ ×	2	6	167 mm	1000 mm			ь
	+ ×	3	6	100 mm	600 mm			
						Î' 🖡		-
							-	
ADVANCE DESIGN						Ok	Apply Clo	15e 🚯

Selecting a given item changes the accessibility to editing the fields in the following reinforcement table and allows a comfortable definition of the distribution according to current needs. The method of operation is as follows:

- first select how to place the bars by using the new list (for example by Quantity)
- next select the type of distribution / how should be grouped (for example 3 Zones Distribution)
- next define offsets and what offsets are affected by exceeding the length.
- and finally define values on table / change the packages.

Although this may sound like a complicated procedure, in practice it is quite an easy and effective method that automatically determines all other parameters when editing selected values.

Also worth mentioning is another additional improvement in this window - now the distribution category parameter (3 Zones, 2 Zones,) is now automatically completed based on the resulting from calculation bar distribution.

11. RC Footing

New features and improvements implemented in the latest version of the RC Footing module.

11.1. Representation of the substitute footing

A visual representation of the substitute footing.

When we have defined a multilayer soil and activated the verification of bearing capacity using the substitute foundation method, then the location and dimensions of such a substitute foundation are now presented in the 2D elevation views automatically after calculations.



11.2. Additional information on concrete cracking in Info Panel and report

Greater control over calculations with additional concrete cracking information.

One of the available verifications according to Eurocode for the foundation is cracking analysis. If the crack width exceeds the limit value, an appropriate warning message appears.

With the latest version, the most important information about crack verification is also presented in the info Panel.

Verification type	Combination	Value	Limit	Work Ratio
Bearing resistance	111: 1x[1 G]+1x[2 Q]	314,80 kN	467,78 kN	67.3%
Seismic bearing capacity	118: 1x[1 G]+1x[101 COMB]+0.3x[2 Q]	0,59	1,00	59,02%
Sliding	117: 1x[1 G]-1x[101 COMB]	10,00 kN	71,64 kN	13.96%
Settlement	115: 1x[1 G]+0.3x[2 Q]	0,36 cm	5,00 cm	7,18%
Punching	118: 1x[1 6]+1x[101 COMB]+0.3x[2 Q]	0,33 MPa	0,96 MPa	34,43%
Cracking in x direction (bottom)	115: 1x[1 G]+0.3x[2 Q]	0,299 mm	0,300 mm	96,34%
Cracking in y direction (bottom)	115: 1x[1 G]+0.3x[2 Q]	0,241 mm	0,300 mm	80,5%
Reinforcement	Real	Theor	etical	Ratio
Bottom Along X	7,07 cm²	6,35	cm²	89.77%
Bottom Along Y	7,85 cm²	7,09	cm²	90,21%

Info Panel

In addition, the report adds a table with details of the calculation of cracking in the footing.





12 Crack width check

			Crack width	verification			
Direction	Position	Combi	Srmax	$\epsilon_{\rm sm} - \epsilon_{\rm c}$	Wk	Wlin	WP
Direction	rostion	comor	(mm)	(%)	(mm)	(mm)	, "A
Along X	Bottom	115	333	0.79	0.262	0.300	87.41 %
Along Y	Bottom	115	335	0.86	0.288	0.300	96.03 %

11.3. Auto correction of reinforcement when crack widths are too large

Keeping the cracking to the limit by automatically increasing the reinforcement.

In the latest version of the module, in such cases, an automatic modification of the reinforcement in the foundation can be conducted so that the crack width does not exceed the limit value. This is useful in cases when it is more beneficial to keep a smaller footing thickness instead of more reinforcement.

The choice of whether the cracking should be automatically corrected is in the Design Assumptions window on the Cracking tab:

🔁 Desig	gn Assumptions [NOT FOR RES	SALE version]	>	<
	e	Cracking		
(General Design	Verify concrete cracking	\checkmark	
F	ooting Design	Imposed crack width maximum limit	0,3 mm	
E	Bearing Verification Geotechnical Verifications	Cracking correction	\checkmark	
1	Multilayered Soil Verification	Imposed stress limit maximum limit	0 MPa	
9	Seismic Design	Exposure class	XC2 ~	
F	Reinforced Concrete			
C	Cracking	Load duration	Long term 🔍	
(Concrete Covers			

New option to correct cracking

As shown in the example below, enabling this option automatically reduces cracking at the expense of more reinforcement.

Vertification type	Combination	Velue	Linit	Work Ratio	Verification type	Combination	×	alaw -
Bearing resistance	108-135(1-0)+1.5(2-0)	657,7 88	963/181	61,27%	Bearing resolution	108(1.554) (9)(1.542)(9)	- 997.4	an -
Siding	105:1x(1.0)	22.85	306,6 kN	6.54%	Siking	105; fa(10)	2013	4
Overtaining	101-03-01-5]	. д	15	26,8%	Overturning	101:025(1.0)	2,9	
Settlement	110; 10[10] + 10[20]	5 mm	S0 nm	9.02%	Settlement	$\{1,2,1,1,1,0\} \in \{1,1,2,1\}$	See.	
Panahang	206-285 (218)	$0, CMR_{\rm B}$	0.0 MIG	10/07.	Punching	136 1.356[1.0]	Q7 MPs	
Crecking in a direction (bottom)	114: b(16)-03(2.0)	C29 mil	C3 mm	90,21%	Overlang in orders from (hold one)	124.141.91694642.03	041.00	- 0,
Crecking inly direction Brottomi	114 1517 BL 035 2 GL	C20mm	C.Simm	93,655	Cracking is y direction (bottom)	112:141 C]+03:(2 C]	0,44 mm	0,3 (
Reinforcement	Real	Theo	retical	Fatio	Beinfastement	Real	Thes	etical -
Rettorn Along X	1492 mor ²	N IS	hana'	95.1%	Sottom Along X	1100 mm ²	12.2	nn'
Bottom Along Y	1257 mm ²	1160	laar'	23.92%	Battan Along V	942 mm ²	903	n ni
Calculation results Franciscul sea	tine.			Steel ratio = 47,9 kp/m ¹	Calculation on day Dependent of	a minute a		

Cracking correction on (left) and off (right) for the same example

11.4. Improved editing of the distribution of bars

Easier definition and modification of bar distributions on a pad.

In the newest version of the program, the way of editing the distribution of top and bottom bars in the pad has been modified. The changes concern the Pad tabs on the Reinforcement edit window and are related to an option to select the method of distribution definition that has been added. We can now choose to define the bar distribution either by entering the number of bars or by entering the spacing.



Reinforcement [NOT FOR RESALE version]								×
6 6 9	Reinforcement Sections				Q	1 1 1	1.1	101
	Theoretical reinforcement			6,58 cm²	-+ -	╋┥╋	┥╼╊╼	+
 Pad Bottom Bars Along X 	Real reinforcement			7,07 cm ²	-	1		S -
Bottom Bars Along Y	General Settings				-			s
Top Bars Along X	Bar mark			1				(中)
Top Bars Along Y	Bar color			<i>i</i>		1	i i i	s
Supported Element Longitudinal				0		T		s
Main Bars Along X	Define distribution by			Quantity ~	L			+
Main Bars Along Y	Exceeding length affects			Both offsets ~~~	Y			02
Secondary Bars Along X				-	Ť x	-		
Secondary Bars Along Y	Quantity		(Q):	9	ц,			
# Supported Element Transversal	Diameter			я10 ~				
Links	Consister							
Binders Bars Along X	spacing		(5):	149,1 mm				
Binders Bars Along Y	Start / End offset	(01) / (02): 70 mm		70 mm				
	Covers				11			12
	Тор		(T):	0 mm	÷.			Ě

11.5. Information in the report about the final reinforcement area

Additional information in the report to better describe the final reinforcement area

In the calculation report in the section on reinforcement results, new information about the final required reinforcement area was provided.

This is the final theoretical reinforcement area, based on which the provided reinforcement is selected. This area may be larger than the original theoretical reinforcement area determined from the formulas because it considers possible additional adjustments resulting from the need to meet additional conditions/verifications.

This can be an iterative increase of the reinforcement area to meet the limit of cracking or stresses in the reinforcement bars, but also a recalculation of the theoretical reinforcement resulting from a change in concrete cover after the application of other than the originally assumed diameters of the real reinforcement.

Theoretical reinforcement	$A_{thee} = \frac{2N_{the} \cdot \delta}{l_1 \cdot f_{yd}} = \frac{2 \times 689.3}{400 \text{ mm}}$	kN×113 mm ×434.8 MPa = 832 mm ²
Minimum reinforcement	$A_{min} = 1.3 \% \cdot L_2 \cdot d = 1.3 \%$ $A_{min} = 336 \text{ mm}^2$	‰×1200 mm×210 mm
Required reinforcement	$\Lambda_{rest} = max \{ \Lambda_{thes}; \Lambda_{min} \} = 1$	nax{832 mm ² ; 336 mm ² } = 832 mm ²
Final required reinforcement	$A_{rept,f} = 952 \text{ mm}^2$	
(after correction for other verification to be satisfied - stresses, cracking etc.)		
Provided reinforcement	$A_{prov} = 1021 \text{ mm}^2$	13 × ø10 (Spacing = 88 mm)

11.6. Improvements to seismic bearing capacity calculations for France

Possibility to select another method for Nmax calculation for France, and more details on related checks on reports.

An additional method for determining the ultimate seismic bearing capacity of the footing under vertical axial load is available for French project settings. Until now, the Nmax value was always determined following EN 1998-5. Now we can determine this value following NF P94-261, using the formula:

$$N_{max} = \frac{q_{net} \cdot A}{\gamma_{R,v} \cdot \gamma_{R,d,v}}$$

GRAITEC

The choice of method is available as a list on the Seismic Design tab of the Design Assumptions window. Note that it is available for editing only for French settings.

E 19	Seismic Bearing Capacity Verification	
General Design	Activate	v
Footing Design	Soil type	Medium - dense to dense sand
Bearing Verification	Yes	1.00
Geotechnical Verifications Multilawared Soil Verification	New order define	NE 094-261
Seismic Design	Pimax calculation	NP 194 201
Reinforced Concrete	Nmax on purely cohesionless soil	Least favorable
Cracking	Υ.	0.80
Concrete Covers	Seil estes au	Category C v

Choice of method for determining Nmax

It is worth mentioning that this method requires user-defined bearing pressure/resistance.

If seismic bearing capacity verification is performed, details of calculations using the selected method are available in the relevant section of the report.

6.2 Seismic bearing capacity	
Along X:	
Combination	116: 1x[1 G]+1x[101 COMB]
Ultimate bearing capacity of the footing under vertical axial load	$N_{max} = \frac{q_{sct} \cdot A}{\gamma_{R,v} \cdot \gamma_{R,a,v}} = \frac{1.00 \text{ MPa} \times 22500.00 \text{ cm}^2}{1.4 \times 1.2} = 1339.29 \text{ kN}$
Normalized strength inertia	$\overline{F} = \frac{\gamma_{soil} \cdot a_s \cdot S \cdot B}{g \cdot \overline{c}}$
	$\overline{F} = \frac{19.0 \text{ kN/m}^3 \times 0.56 \text{ m/s}^2 \times 1.50 \times 1500 \text{ mm}}{9.81 \text{ m/s}^2 \times 0.03 \text{ MPa}} = 0.10$
EN 1998-5, Annex F, (F.2)	$\overline{N} = \frac{\gamma_{gd} \cdot N_{Ed}}{N_{max}} = \frac{1.00 \times 894.77 \text{ kN}}{1339.29 \text{ kN}} = 0.67$

Excerpt from the report with details on used method

The latest version of the program has also expanded the report's content when seismic bearing capacity cannot be calculated. Details of the calculation of the associated condition from the French National Annex are now provided.

6.2 Seismic bearing capacity	
Seismic bearing capacity cannot be calc satisfied. Along X:	ulated. Condition on N normalised, F.5 from EN1998-5, is not
Combination	119: 1x[1 G]-1x[101 COMB]+0.3x[2 Q]
Ultimate bearing capacity of the footing under vertical axial load	E N _{max} = $(\pi + 2)\frac{\bar{c}}{\gamma_{M}} \mathbf{B} \cdot \mathbf{L}$
	$N_{max} = (\pi + 2) \times \frac{0.08 \text{ MPa}}{1.40} \times 1000 \text{ mm} \times 1000 \text{ mm} = 293.81 \text{ kN}$
EN 1998-5, Annex F, (F.2)	$\overline{N} = \frac{\gamma_{Ed}}{N_{max}} = \frac{1.00 \times 311.53 \text{ kN}}{293.81 \text{ kN}} = 1.06$
Verification	$0 < \overline{N} \le 1: 0 < 1.06 \le 1$
EN 1998-5, Annex F, (F.5)	Failed

Excerpt from the report with details on not satisfied condition on normalized N

11.7. A new method of determining soil pressure

Determining active and passive earth pressure according to Eurocode 7 provisions.

In the latest version of the module, a new algorithm for determining active and passive earth pressure has been introduced. Earth pressures are considered during sliding verification and are now calculated

following annex C of EN 1997-1. The changes apply to calculations according to the Eurocode, excluding the German National Annex (using a different method).

NOTE: A consideration of earth pressure for sliding verification is not enabled by default and can be enabled in the Design Assumptions window.

	cieforced Concrete	Passive earth pressure mobilisation	100 %
N Se	lultilayered Soil Verification eismic Design	Consider earth passive pressure for sliding resistance	\checkmark
G	earing Verification eotechnical Verifications	Consider earth pressure for sliding resistance	V
G Fe	eneral Design ooting Design	Design friction angle (δ)	 δ = φ' δ = 1 × φ'
Ē	6 19	Sliding Verification	

Option to enable consideration of earth pressure for sliding

Detailed data and results of earth pressure calculations are presented in the report.

Combi	Condition	Layer	H _d	R _{h,d}	Vd	WR		
			(kN)	(kN)	(kN)			
106	Drained	None	307.6	451.6	743.8	68.1 %		
Drained cond	itions							
Combination:			106: 1.35x[1	G]				
Horizontal loa	d		$H_d = \sqrt{H_s^2 + H_s^2}$	$\overline{\mathbf{I}}_{y}^{2} = \sqrt{(300 \text{ kN})^{2}}$	$^{2} + (67.8 \text{ kN})^{2} =$	307.6 kN		
Vertical load (initial)		$V_{d,0} = 760.8 \text{ k}$	N				
Vertical load (including the eff	ect of earth	$V_d = Max \{V_d,$	$_0 + \gamma_G \cdot P_{av,x} - P_p$	$\gamma_{x,x} + \gamma_G \cdot P_{av,y} - P$	$p_{pv3}; 0$		
pressures)			$V_d = Max \begin{cases} 760 \\ +1 \end{cases}$	0.8 kN + 1.35×0. .35×0.1 kN - 8.0	1 kN - 8.6 kN - 5 kN; 0	+}		
Sliding resistar	nce (initial)		$V_d = 745.6 \text{ km}$ $V_d \cdot \tan \theta$	(δ _ν) 743.8 kN	×tan(30°)			
EN 1007.	1 (653)		$R_{d,0} = \frac{\gamma_{R,h}}{\gamma_{R,h}}$	$\frac{1}{l_0} = \frac{1}{l_0}$	$\frac{429}{1} = 429$	9.4 kN		
The resistance	will be modified	by lateral activ	e and passive	earth pressure o	n footing:			
Active earth m	mar de mouthet	at at a second dette	K = 0.34	en al pressure o	a roomig.			
C1/216	The same coefficient	tuner C	$R_0 = 0.54$					
C.1 (2) J	Om EN 1997-1, 2	unnex C	V 262					
Passive earth p	bressure coefficie	nt	$K_{p} = 3.03$					
C.1 (2) Jr	OM EN 1997-1, 1	Annex C	D 0.2131					
Horizontal com pressure on X Vertical compo	nponent of the activ	tive earth e earth pressur	$P_{ab,x} = 0.2 \text{ kN}$ $e P_{av,x} = 0.1 \text{ kN}$					
on X Horizontal con	nponent of the pa	assive	$P_{ph,x} = 18.5 \text{ kM}$	Į				
Vertical compo	onent of the pass	ive resistance	$P_{pv,x} = 8.6 \text{ kN}$					
Horizontal con pressure on Y	nponent of the ac	tive earth	$P_{ab,y} {=} 0.2 kN$					
Vertical compo	onent of the activ	e earth pressur	$e P_{av,y} = 0.1 \text{ kN}$					
on Y Horizontal con resistance on Y	nponent of the p	assive	$P_{ph,y} = 18.5 \text{ kM}$	ł				
Vertical compo on Y	onent of the pass	ive resistance	$P_{pv,y} = 8.6 \text{ kN}$					
Applied load c	component on X	with load from	$\mathbf{H}_{s} = \left \mathbf{H}_{ds} \right + \gamma_{G} \cdot \mathbf{P}_{dt,s} =$					
earth trust			$H_x = 299.7 \text{ km} $	$N +1.35 \times 0.2 \text{ kN}$	= 300 kN			
Applied load c	component on Y	with load from	$H_{y} = \left H_{dy} \right + \gamma_{0}$	$_{5} \cdot P_{ah,y} =$				
earth trust			$H_y = 67.5 \text{ kN} $	+1.35×0.2 kN =	= 67.8 kN			
Resultant from	passive earth pr	essure	$R = \sqrt{R_{p,d,x}^2 + }$	$\overline{R_{pol,y}^2} = \sqrt{(18.5)}$	$(kN)^{2} + (18.5 kN)^{2}$	$\overline{)^2} = 26.2 \text{ kN}$		
Resistance loa along X	d from passive e	arth pressure	$R_{p,d,x} = \frac{P_{pb,x}}{\gamma_{Rd}} = \frac{18.5 \text{ kN}}{1} = 18.5 \text{ kN}$					
Resistance load along Y	d from passive e	arth pressure	$R_{p,l,y} = \frac{P_{ph,y}}{\gamma_{w,z}} = \frac{18.5 \text{ kN}}{1} = 18.5 \text{ kN}$					
Passive resista resultant betwe	nce projected alc een Hdx and Hdy	ong the axis of	$R_{p,d} = R \cdot \cos(\alpha R_{p,d} = 26.2 \text{ kN})$	x – β) [xcos(12.7 ° – 45	°) = 22.1 kN			
Sliding resista	nce (final)		$R_d = R_{d,0} + R_a$	4 = 429.4 kN + 2	22.1 kN = 451.6	kN		
Sliding verific	ation		H. < R.: 307.6	kN < 451.6 kN		estrili		
Work Ratio			69 1 84 (P	- 1)				
TOIL Ratio			00.1 % (Pass	ea)				

12. RC Wall

New features and improvements implemented in the latest version of the RC Wall module.

12.1. Generation of drawing with schedules for bars and fabrics

Automatic simultaneous generation of schedules for bars and fabrics.

Previously, when generating drawings for walls, a bar schedule was generated, while if there were also reinforcement meshes, their schedule had to be manually generated and added to the drawing.

In the latest version, this process has been simplified, and depending on your needs, you can use either a drawing style for members only or a new template - for members and meshes. The same mechanism is also available for RC Slab module.



Selecting a drawing style on RC Wall (left) and RC Slab (right)

A new drawing style was added to allow the automatic simultaneous generation of schedules for bars and fabrics.



Automatically generated drawing for wall with schedules for bars and meshes

12.2. Import and export forces using Excel file

Easy entry of loads from external data using an Excel sheet.

For Shear walls, it is now possible to import and export resultant forces using an Excel sheet. For this purpose, two new icons are available in the window of load definitions – one for export and the other for import from an Excel spreadsheet.

Lea	ds De	finit	on [NOT	POR RESALE version]						×
ertica	Loa	ds i	Resultant	Forces						
Wall n	esulta	ent fa	intes							
-	Wal	11		× 📥						
				_	_					
Positi	on i	Botto	m of wal	 Eccentricity a 	n 0 X gnole	om				
Envel	ope s	hear	force 0	- KN			_		-	
🗆 Ta	ke in	to ac	count the	e out of plane moment	(Mod		- I	X3 X8		
_	_		_						N>0 MZ>0	
Add	Dyl	D	Code	Lowd care	N	Mz	Vx	18.2		
+	×	1	ECG	1 - Dead loads 1 1111	653,5 kN	2,5 kN-m	12,5 kN	0 kN-m	Marti	
+	×	2	ECQ	2 - Live loads 1 $^{}$ \simeq	345,5 kN	46,6 kN-m	134,5 kN	0 kN-m		
+	×	3	ECQ	3 - Live loads 2 👘 🐃	44,4 kN	22,3 kN-m	0,5 kN	0 kN-m		
+	×	4	ECV	4 - Wind loads 1 🔗	36,6 KN	0,2 kN-m	1245,5 kN	0 kN-m	AY	
									Z	
									x	

Import and export to Excel commands

With the ability to import resultant forces per wall or wall group, you can easily use the module to analyze walls when load data is available from external sources, as well as quickly edit load data.

A	12	\checkmark : $\times \checkmark f_x$					
	А	В	С	D	E	F	G
1	Code	Load case	N (kN)	Mz (kN∙m)	Vx (kN)	Mx (kN·m)	
2	ECG	Dead loads 1	653,5	2,5	12,5	0	
3	ECQ	Live loads 1	345,5	46,6	134,5	0	
4	ECQ	Live loads 2	44,4	22,3	0,5	0	
5	ECV	Wind loads 1	36,6	0,2	1245,5	0	
6							
7							
	< >	Wall Resultant Forces	Group R	esultant For	ces	+	

Loads exported to an Excel spreadsheet

13. Masonry Wall

New features and improvements implemented in the latest version of the Masonry Wall module.

13.1. Loads representation in viewports

Easy verification of defined loads.

To make it easier to view defined loads as well as imported internal forces, it is now possible to display loads and internal forces graphically

We can now see loads in the 2d and 3d views.



View of external loads on the wall

Depending on the needs, we can easily enable or disable the display of loads directly from the window of a particular view. From the window, with graphic settings, we can also control their display range – for example, for which load case.

In the case where the wall was imported from Advance Design, we can also graphically display the calculated internal forces at the bottom middle, and top of the panels for each load case.



Resultant forces determined for vertical design sections



13.2. Expanded report for RC6 calculations

Richer project documentation for Romanian clients.

A new section has been added in the reports for verification of a wall subjected to in-plane bending according to the Romanian CR6 standard.

		Verification of wall	l subjected to i	n-plane bendi	ng	
Leaf	Comb.	Critical Section	M _{Ed}	M _{Rd}	WR	Status
			(kN	·m)		
-	102	Tep	23.6	145.6	16.2 %	Passed
Design value o Design value o <i>(CR6-2013</i>	f the applied f the resistant (6.6.3.2, eq. (bending moment () (bending moment () (5.18))	$M_{Ed} = 23.6 \text{ kN} \cdot x_c$ $M_{Ed} = N_{Ed} \cdot y_c$	n 129.1 kN+1128	5 mm = 145.6 k9	4-m
Compressed an (CR6-2013	ea (6.6.3.2, eq. (5.17))	$\Lambda_c = \frac{N_{tot}}{0.85 \cdot f_d} = 0$	129.1 kN 0.85×1 MPa =)	148784 mm ²	
Axial load		1	N _{EC} =129.1 kN			
Design compre	ssive stress o	f masoury j	f _d =1 MPa			
Lever ann			$y_c = 1128 \text{ mm}$			
Verification		1	M ₆₀ < M ₈₆ 23.6 16.2 % (Passed	kN·m < 145.61)	kN∙m	

13.3. Possibility to choose the section for the stress diagram

Better control over the display of results.

When displaying the results in the form of stress diagrams, it is now possible to select the position of the section (bottom/ middle/ top). For the selected section, the height of the section level is also presented.

Out of Plane 👻 🗹 Extreme Values	Minimum Stress at Rigł 👻	O Bottom Middle	○ Top h = 2400 mm	✓ Fill area under curves	Show wall section
	Section	n position sele	ction options		
Leaf Section					
·		250 mm			
Leaf Stress [MPa]				
					0,4
1,3					
Out of Disea y	Sure Combination 102:125v11/31-15v2 v	Panel 1 v @ Rottom () Middle	O Ton ha Dana III Fil year under	oner 🗹 Showwell rection	
St St	ress diagrams fo	or the section	at the bottom o	of the wall	



GRAITEC

13.4. Saving the masonry databases data per model

A new way of selecting data from databases to make it easier to manage their contents.

To facilitate the management of data from databases used by the Masonry Walls module, databases have been separated from the data of the current project.

The changes apply to databases for mortar, masonry units, and wall sections.

Thanks to these changes, the change of data, e.g., of a given section in the masonry wall section database will not affect other projects that use the same section.

In the case of Masonry Units and Mortars, the data is managed in the same way as, for example, the management of reinforcement fabrics or soils in the other modules. For example, if you click on the Unit List icon, you will see a window with a list of masonry units loaded/available in the project. This list can of course be defined individually and depends on the project template. To delete from the list unwanted entry, you can click on the icon on the first column. To add new entries to the list, you need to use the Unit database button.

🤾 Uni	l fat	[NOT FOR RESALE varian]								
Del	No	Name	Color	Unit Material	Unit grou	.p	Unit category	Unit weight (y)	Compressive strength (fb)	
×	1	Solid clay brick, class 10		City	1		E	18.kN/m²	10 MPA	
×	2	Solid clay brick, class 15		Clay	1		1	18 kN/m ⁴	15 MPa	
×	1	Solid clay brick, class 20		City	1		1 · · · · ·	16.kN/m ²	20 MPa	
×	4	Clinker brick, class 25		Clay	1		1 · · · ·	22.kN/m ⁴	25 MPa	
×	5	Clanker brick, class 30		Clay	1		1	22 kN/m*	30 MPA	
×	6	Clinker brick, class 35		Clay	1		1 ×	22 kN/m ¹	35 MPa	
×	7	Clinker brick, class 40		Clay	1		1	22 kN/m*	40 MPa	
×	8	Vertical hollow brick, class 10		Clay	2		1	13 kN/m ³	10 MPa	
×	9	Vertical hollow brick, class 15		Clay	2		1	13 kN/m*	15 MPa	
×	10	Vertical hollow brick, class 20		Clay	2		1	13 kN/m ³	20 MPa	
×	11	Horizontal hollow brick, class 5		Clay	4		1	14 kN/m ⁴	5 MPa	
×	12	Parous ceramic hollow block, class 7.5		City	3		1	9 kN/m ³	7,5 MFa	
×	13	Porous ceramic hollow block, class 10		Clay	3		1	9 kN/m ⁴	10 MPa	
×	14	Parous ceramic hollow block, class 15		City	2		1	9 kN/m ³	15 MPA	
×	15	Porous corumic hollow block, class 20		Clay	2		1	9 kN/m²	20 MPa	
¥	16	Girata Marin Fare 15		Calcium cilcula	 1		1	15.0M/m ²	15,000	
U G	nit D	CE DESIGN	-						Ok Apply Cit	ose

In this way, another window is available, in which you can see a list of all available Masonry units in the databases. Using checkboxes, we select the items we want to add to the current project and then we can use the Import button.

th.	0	Y(Correl)	mb//	AppDate (Local Gratter) Advance Design	modu/er(2524)5	ettings/Data/Mesonry Doteboost					
lato	iŋ N	el Units,	жц	A_Vitang: *						Check All Check No	204
dd	Del 1	Import	No	Neme	Color	Unit Material	Unit group	Unit category	Unit weight (v)	Compressive strength (fb)	
-	×		26	Ytong PP4/0,6 S 17,5cm		Autoclaved acreted concrete	1 ~	1 *	6 kN/m ²	4 MRa	
-	×		27	Ytong PP4/0,6 S 20cm		Autoclaved acreted concrete	1 ~	1 ×	6 kN/m ²	4 MRa	
	×		28	Ytong PP4/0,6 S + GT 24cm		Autoclaved acreted concrete	1 ~	1 ×	6 kN/m ²	4 MPa	
	×	¥	29	Ytong PP4/0,6 S+GT 30cm		Autoclaved acreted concrete	1 ~	1 ×	6 kW/m ²	4 MPa	
	×	V	30	Ytong PP4/0,6 S+GT 36,5cm		Autoclaved acreted concrete	1 ~	1 ×	6 kW/m*	4 MPa	
	×		X	Ytong Solid PP5/0,6 S+GT24cm		Autoclaved sensited concrete	1 ~	1	6 kWm*	5 MRa	
	×		32	10 pg P95/0,7 GT 24cm		Autoclaved sensited concrete	1 ~	1	7 kWm*	5 MPa	
	×		33	Yton, YumboUltra+ P#2,2/0,3 S 36,5cr	n	Autoclaved sensited concrete	1 ~	1	3 kW/m*	2,2 MPa	
	×		34	Ytong AmboUltra+ P#2,2/0,3 3 48cm		Autoclaved sensited concrete	1 ~	1	3 kN/m*	2,2 MFa	
	×		35	Ytong Jumbo PMI/U/6 S 24cm		Autoclaved sensited concrete	1	1	b kN/m ¹	4 MPa	
	×		35	Ytong Panel G4/600 / Som		Autoclaved aerated concrete	1 *	1	b,5 kN/m ¹	4 MPa	
	×		37	Ytong Panel G4/600 10cm		Autoclaved aerated concrete	1 *	1	b,5 kN/m ¹	4 MPa	
	×		35	Ytong Panel G5/750 10cm		Autoclaved aerated concrete	1 *	1	BJ3 kN/m ¹	5 MPa	
	×		39	Yong Panel SWL P4/600 20cm		Autoclaved aerated concrete	1 *	1	6,5 kN/m*	4 MPA	
	×		40	Vitang Panel SWE P4/600 24cm		Autoclaved sensited concrete	1	1	6,5 kN/m ²	4 MPA	

As it was available in the previous version, in this window we can, of course, select a database from the disk, edit the contents of the database itself, including extending it with our own items.

13.5. Presentation of partial factors

Quick verification of the applied partial factors.

To make it possible to verify the determined partial factors for the current masonry section, they are now available in the Design Assumptions window. Note that the values of the partial factors are determined automatically, as most national annexes depend on the parameters and structure of the wall cross-section. Therefore, they may be different for different walls.

6	General	
ieneral	Automatic selection of a partial factor for material	4
artial Factors	Class of execution control	1 *
	Removal of the wall panel affects the overall stability	
	Partial Factors	
	ULS Compression	(yu): 2,6
	ULS Shear	(yw): 2,5
	ULS Flexural Tension	(yn): 2
	ULS Accidental	(yu): 1,3
	ULS seismic	(yu): 1,73
	SLS	And 1

13.6. Small improvement: Support depth representation in viewports

Displaying the depth of the slab support on viewers to easily check the data.

The depth of the slab support is now visible to viewers, so you can more easily make sure that the data you entered is correct.

Fe 10	Upper Wall		
	Wall		
Vall Geometry	Height	(h1): 3000 mm	
Jopper Elements Lower Elements	Upper Floor		
Openings	Type	Slab on left	
Supports Conditions	Continuous floor		h1
	Thickness	(D1): 200 mm	
	E	30000 MPa	(11.1
	Unit Weight	25 kN/m*	
	Support depth equal to leaf thickness	150 mm	
	Floor slab spanning in 2 directions	V	720

14. RC Slab

New features and improvements implemented in the latest version of the RC Slab module.

14.1. New bar shapes

Possibility to generate u-shaped and double-bend bars.

The latest version of the program has expanded the list of types of rebar shapes that can be used in concrete slabs. All these shapes refer to the method of bar end, i.e. bar anchoring solution. Now there are 4 anchoring methods available: straight bars, hooks, double-bent bars, and using u-shaped bars.



The choice of anchorage type is available in the settings (Reinforcement Assumptions window), on the new Edge Reinforcement tab. We can define different settings for the first and second layers of reinforcement, independently for free edges and edges on supports.

Rei	nforcement Assumptions [NOT	FOR RESALE version]			
Ē	Fin 19	- 3D-view 1st layer	1st layer	Super	ior layers
0 0 0	General Edge Reinforcement Minimum Reinforcement Solutions Parameters Bar Splitting	Eda Anthonas Solutions			
		Luge rationage solutions	F	ïrst løyer	Superior layers
		Free edge anchorage solution		U-shaped bar 🖂	Hooks "
		Edges on supports anchorage solution		Double Bent E 🕤	None "
		Extend main bars inside edge support			
		U-shaped bars			
		Diameter			ø8 ·
		Associated to			Top bars v
		Imposed spacing			20,00 cm
		Arms length			Auto •
		Top arm length			0,50 m
		Bottom arm length			0,50 m
G	WANCE DESIGN		C	% Apply	Close

In addition, this window allows us to set parameters for u-shaped bars, including diameter, and assignment to bottom or bottom reinforcement, and we can also impose spacing or arm length.

During the generation of reinforcement drawings, bar anchorage is presented as projected, so you can easily recognize its shape.



Automatically generated reinforcement plan for top reinforcement

14.2. Possibility to manage hooks for reinforcement zones

Possibility for editing hooks separately for each reinforcement zone.

In the latest version of the program, the window for editing generated reinforcement has been expanded with a set of new columns for modifying anchorage parameters. Separately for each reinforcement zone you can decide whether the anchorage is to be generated, you can set the diameter (for anchorage by U-shape bar), set the cover value, and also modify the start or end hook length.

[Slab - G	enerated Reinf	forcement (NO	T FOR RESAL	version	4									×
Filter														
Solution:	Тор			3			Layer	Layer 1			U.			
🔿 Bar zon	es													
Anchorage 3	X Anchorage	Y U-Shaped	X Diameter	U-Shap	ed Y Diameter	X Cover	Y Cover	Start hoo	k length X	End hook leng	gth X Start P	ook length Y	End hook k	angth Y
V	\checkmark	ø8	۷	88	v	0,03 m	0,03 m	Auto	~	Auto	" Auto	~	Auto	۰ ر
c								0,06 m						2
Fabrics	zones							Auto						
Del Layer	r Name	Fabrics Type	Lap Splice P	osition	Diameter X	Diameter Y	Spacing X	Spacing Y	Overlap X	Overlap Y	Real reinf X	Real reinf Y	Offset X	Offset Y
ADVANC	E DESIGN											Ok	Apply	Cancel

Part of the Generated Reinforcement dialog with anchorage parameters



Editing example - removing the hooks in one direction and changing the diameter of the U-shaped bars in the other

14.3. New options for defining reinforcement solution

Possibility to define reinforcement coverage with reinforcement areas.

For the automatic definition of reinforcement zones, we can use the setting (on the Reinforcement Assumption window) so that when defining reinforcement layers, they cover a fixed percentage of the theoretical reinforcement area. That is, for example, for the bottom reinforcement, the first zone (covering the entire surface) should cover 70% of the required reinforcement, and the remaining 30% should be covered by the second layer.

Reinforcement Assumption	ns [NOT FOR RESALE version]			
re re	Reinforcement So	lutions		
General	Reinforcement pa	ttern	Bars	~
Edge Reinforcement	Maximum number	r of density layers	2	Ŷ
Minimum Reinforcemer Solutions	nt Coverage relative	to area of	Max theoretical reinfo	orce ~
Top Bottom	Coverage method		Percentage	v
Parameters	Layer	Туре	Minimum Coverage	
Bar Splitting	1	Bars	70,00 %	
	2	Bars	30,00 %	

Minimum coverage defined as the percentage of maximum theoretical reinforcement area

In the latest version, we can specify the coverage method not only by percentages but also by absolute values of the reinforcement area. This facilitates the precise selection of coverage.

To be able to select the method, in the *Reinforcement Assumption* window, under the *Solution* tab, you must set the **Coverage** method to **Area**. Then in the table below, we can specify the minimum coverage area separately for X and Y directions.



C Reinforcement Assumptions [NG	OT FOR RESALE vers	sion]			
	Reinforcemen	nt Solutions			
General	Reinforcemer	nt pattern		Bars	~
Edge Reinforcement	Maximum nu	mber of density layers		2	~
Minimum Reinforcement Solutions	Coverage rela	ative to area of	Max theoretical reinforce $$		
Тор	Coverage me	thod		Area	~
 Bottom Parameters 	Layer	Туре	Minimum Coverage on X	Minimum Coverage on Y	
▷ Bar Splitting	1	Bars	6,500 cm²/m	10,000 cm²/m	
	2	Bars			

Minimum coverage defined as the value of reinforcement area

14.4. New methods for defining reinforcement zone graphically

Faster modeling of reinforcement zones with additional graphic definition methods.

When defining reinforcement in concrete slabs based on the calculated theoretical reinforcement, we can use either automatic determination of reinforcement zones, or we can define such zones by ourselves, as well as we can mix both methods (e.g., one zone on the entire surface of the slab is defined automatically and zones of additional reinforcement are defined manually).

Until now, the graphical definition of a zone consisted of entering the zone area via a closed polyline. In the latest version, two additional methods of drawing rectangular reinforcement zones are available - by 2 points and by the centerline.

The new method of defining rectangles by 2 points is quite simple. We just need to pick 2 opposite points of the rectangle's diagonal. This model is now a default one.



The second new method is a definition of a rectangle by a centerline and side offsets. We need to pick the start and end point for the line, and next we pick a point for defining the right and left edges of the rectangle. This mode is especially useful for defining rectangular zones over a line of supports.



Note that the drawing mode can be changed by pressing the Space key when adding a zone.




Zone definition with centerline and side offsets (left), and defined reinforcement zone (right)

14.5. Set of improvements to drawings

Increased attractiveness and greater customization of automatic reinforcement drawings for RC slabs.

Separate drawings for reinforcement directions

With this version, we can generate drawings either simultaneously or separately for each reinforcement direction. For this purpose, in the properties of the plans, we can choose whether a given view presents rebars in the X or Y or in both directions.





GRAITEC

Improvements to bar annotations

To increase the functionality of the drawings and improve their reception, several improvements have been made to the generation and editing of reinforcement descriptions.

First, the algorithm for the automatic placement of descriptions has been improved to increase their readability and avoid bar collisions.



Secondly, in situations where you want to modify the positions of the distribution descriptions, you can move the entire description, as well as the bar mark and the dimension line independently. You can also use the new option to disable descriptions above the dimension line.



Initial position / moved the whole description / moved bar mark / without dimension line's description

Better representation of openings

Another improvement of RC slab drawings is the ability to display a special graphic representation for openings (as shown in the image). In case you do not want such an additional symbol for openings, you can disable it with a new option in the drawing settings.



Option for changing the display style for openings

New option to hide grids in drawings

To increase the functionality of the parameterization of slab drawings, an option has been added to decide whether the structure axes are to be presented in the drawings.



Option to exclude axes and application example

14.6. Set of smaller improvements

Increased functionality and comfort with a set of small improvements.

Allow the second layer to have a bigger diameter than the first one

When defining bars in each direction in multiple layers, it is now possible to use any configuration of bar diameters in different layers, including adding bars with larger diameters in the next layer than in the first.



Displaying local axes for each slab

Since the reinforcement for slabs refers to their local axis system, to facilitate orientation during zone definition and reinforcement analysis, the views now show a graphical symbol for the local axis system of the slab.



Possibility for disabling reinforcement layer

In case you want to generate reinforcement automatically, but not in all layers, you can now enter minimum coverage as zero for a layer when setting parameters. An example application is the quick automatic generation of the bottom reinforcement, while the top reinforcement is then defined manually only in selected areas (e.g., over supports).

	Reinforcement Solutions				
General	Reinforcement pat	Reinforcement pattern Maximum number of density layers Coverage relative to area of			~
Edge Reinforcement	Maximum number				~
Minimum Reinforcement Solutions	Coverage relative t				~
Тор	Coverage method			Percentage	
Bottom					
Parameters	Layer	Туре	Minimum	Coverage	
Bar Splitting	1	Bars	0,00 %		

New option to generate one-way reinforcement zones

A new option 'Allow one-way reinforcement zones' is available in the Reinforcement Assumptions window. This option is used during automatic determination of reinforcement zones and causes some of the zones to be generated not as two-way, but as one or two one-way zones. Although not all zones can be modified in this way, in many cases the result is a smaller number of bars.



GRAITEC

Reinforcement Assumptions [NOT	FOR RESALE version]	×
	Reinforcement Calculation	
General	Theoretical reinforcement area	 Imported
Edge Reinforcement Minimum Reinforcement		 Calculated by Strips
Solutions	Reinforcement Generation	
 Parameters Bar Splitting 	Placement at cover	X direction ~
	Minimum hook length	5 x ø
	Generate bars inside neighbor supports	\checkmark
	Extend hooks outside slab thickness	\checkmark
	Minimum bar length	0,20 m
	Reinforce within the slab's analytical contour	
	Allow one-way reinforcement zones	\checkmark
	Openings in Fabrics	
	Don't cut openings if area is smaller than imposed value	2500,00 cm ²

Location of a new option



Zone with two-way reinforcement (left) and two zones with one-way reinforcement (right)

15. Steel Connections

New features and improvements implemented in the latest version of the Steel Connection module.

15.1. Welded truss tube connections for square/rectangular tubes

New welded truss connection type with multiple diagonals configurations of square and rectangular tubes.

We are excited to introduce a new functionality in our Advance Design Steel Connection module - the analysis of welded connections for rectangular/square tube sections, providing engineers with more flexibility in their design process. With this new connection category, users can confidently analyze and optimize connections using rectangular/square tubes, further expanding the capabilities of our software.

The calculations are based on the provisions of Eurocode 3 (EN 1993-1-8).

In the current version 2024, the users can verify welded truss connections for steel profiles for the following configurations: K Type, KT Type, N Type, T Type, X Type and Y Type. When creating a new connection in the standalone environment, the connection type can be selected from a list of available predefined types.



Selection of a connection type

The Welded Tubular Truss connection allows a chord (main beam) and brace elements (diagonals) to join. The main beam can be square, rectangular tubular or I steel cross-section, while the diagonals can be only square or rectangular tubular steel cross sections.

GRAITEC



A chord modeled by a square hollow section (left) and by an I section (right)

New connections can be modeled directly on the Steel Connection module but can be also imported from the Advance Design model. In the second case, along with data on geometry, profiles, and materials, internal forces are also transferred.

NOTE: As Eurocode mentions, the angle between the chord and diagonals cannot be less than 30 degrees. This Eurocode restriction is imposed in Advance Design and the standalone module - if the angle is smaller than 30 degrees, the connection is not generated in Advance Design, while in the standalone module, the angle is recalculated.

Whether the connection is defined fully in the module or imported from a FEM model, the details related to the position of the diagonals can be changed from the module.

For example, the gap/overlap offset between the axes/edges of the members, which may be different from that resulting from typical axial modeling of elements in a finite element model, can be changed from diagonals member definition.



Modification of the gap between diagonals

Also, the eccentricity of the main beam can be controlled from the member's dialog.



The eccentricity of the main beam

The elements in this category of connections are joined by welds, so the user can easily set their parameters.

		Middle diagonal	<u> </u>
Welds Diagonals First diagonal Second diagonal Third diagonal	Geometry Weld size: Weld type: Weld quality:	5 mm Fillet Shop contir	

Weld parameters

In the Welds dialog, the user can define which diagonal is cut or not by setting the "Cut Priority" option.

Welds [NOT FOR RESALE ven	sion]		
Cut Priority			
Priority:		Middle diagonal	~
		Outer diagonals	
∃- Welds		Middle diagonal	
	Cut Driarity		

Cut Priority

For the Design Options a new parameter has been added: the partial safety factor for resistance of joints in hollow section lattice girder γ_{M5} , with the default value of 1. The value that can be changed if needed.



	IS [NOT FOR RE!	SALE version]		
	Ste	el corrosion:	EN10025_5	\sim
	Control	shear plane:	Threaded	\sim
	T-stub mo	ode 1 failure:	Method 2	\sim
1	T-stub: tensione	ed zone limit:	Auto	\sim
	Reduction fac	tor - threads		
	compliance w	vith EN 1090:	1	\sim
	Weld des	sign method:	Simplified	×
	Weld ca	alculation by:	Real moment	\sim
	C	ombinations:	All	Ň
	S	ombinations: afety factors:	All Imposed ~]
γмо:	5. 1	ombinations: afety factors: γ _{M2s} :	All Imposed ~ 1.25	•]]
γмо: γм1:	Co Si 1 1	afety factors: γ _{M2s} : γ _C :	All Imposed ~ 1.25 1.5]
Υ M0 : Υ M1 : Υ M2 :	Co 5. 1 1.25	afety factors: γ M2s : γ M2s :	All Imposed ~ 1.25 1.5 1.1	•]]
Υм0: Υм1: Υм2: Υм3:	Co 5 1 1.25 1.25	afety factors: Y M2s : Y C : Y M3ser : Y M7 :	All Imposed ~ 1.25 1.5 1.1 1.1	•]]]

Once the loads are entered (or imported from a model in Advance Design), calculations can be performed. Basic information about verification results is available in the Info Panel.



Quickly accessible basic results in the Info Panel table

The analyses include all required verifications according to EN 1993-1-8, including weld verifications considering the geometry of the joint and the effective width of the weld.

More details of the calculations can be found in the report. It contains a detailed description of the data, as well as a set of results, for most of which detailed descriptions and formulas are available.

	Maximum Work Ratio	49.36 %	Passed
	Table of Conten ⊕	ts	
1	Joint description		
2	Load combinations description		
3	Design Assumptions		6
4	Validity limits		6
5	Chord face failure		
	5.1 Chord face failure (K node type)		
	5.2 Chord face failure (Y node type)		8
б	Brace failure		9
	6.1 Brace failure (K node type)		9
	6.2 Brace failure (Y node type)		9
7	Welds verification		10
	7.1 Calculation of welds - explanatory notes		
	7.2 Welds group connecting first diagonal and main me	mber	
	7.3 Welds group connecting second diagonal and main	member	
8	Warning and error messages		15
0	Summary		15





Selected pages from the results report

NOTE: This is the first version of the module with welded truss connections, and it is planned to further develop the range of available options, configurations, and results in future versions of the module. One of the options not available for these connections in the current version is drawings.

15.2. Improvement on the Info Panel for Shear plate joint

Easier connection verification with critical component information.

In the case of Shear plate connections (beam end connections in which the connecting element is a plate or rib working in shear), a new column is added to the Info panel for the indication of the element in which the verification is made. Thanks to this, it is much easier to verify which component is decisive for a given verification, without having to check the report.

Verification type	Objects	Combination	Force	Relitance	Wark liatio			
Searing verification of bolts	[Sec. Dears] (PC240 S235	(I): ULS encelope 1	-32,23 kN	40,53 kN	79.52%			
Boltshear	[Solb] M16 83 on see beam	DJ: U.S. erweispe 1	SSO M	602314N	53,745			
Bending and cheer interaction	[Sec. Deam] (P0240 S235	(I): UIS envelope 1	4,00 kH m	12,2 kH m	35.53%			
An accession of half going	[Sub] Mitrix Convertision	[1]: U.D. evenings - 1	220×10^{10}	10.04518	76,01%			
Blocktasting	(Sec. Deam) (PC240 S215	(I): ULS envelope 1	25,5 kN	103,02 (4)	26.19%			
Street inste	[Sec. Report] (FR3D 5245	(1) U.S. envelope 1	28,55.0	185,015.0	20/672			
Sharryle ding	[Sec. Deam] (P1240 S235	(1): ULS envelope 1	22,5 kN	150,54184	18.515			
Web1 owne	(Web) Greenershove plate	[1] U.S. envelope 1	\$4(13).N/m	811,811,00,64	11,39%			
Tension ultimate	(Sec. Seem) (P1240 S235	(1): ULS envelope 1	12,2 M	235,55164	5.16%			
Tension pieting Cale bits and a	[See, Reven] (FFRd) 5245	(Q-UIS eveninger 1	12,268	265(318	6,305 v			

New column in Info Panel for Shear connections

15.3. New option to ignore bending moment in the calculation for hinged beams

More control over the type of forces used for verifications.

For connections where the beams are hinged on the column's flanges or to beams, such as Shear Plate or Clip Angle, the position of the V force influences the calculation. This situation is often encountered especially when the forces are imported from an Advance Design model.

Generally, in practice, the moment is considered zero at the face of the column/beam. To control this behavior in the Steel Connection module, a new option has been added in the Loads definition dialog:



New option for hinged connections

If this option is checked, the value of the bending moment from the Loads definition tab will not be considered in the joint calculation.

