What’s New

ADVANCE DESIGN 2021
# What’s New in GRAITEC Advance Design 2021

## Table of contents

**Welcome to Advance Design 2021** .......................................................................................................................... 5

**New Options & Improvements - Calculation** ........................................................................................................ 6

- Pushover analysis .................................................................................................................................................. 6
  - Introduction to the Pushover method .................................................................................................................... 6
  - Pushover analysis on Advance Design ................................................................................................................ 7
  - Definition of plastic hinges ................................................................................................................................. 7
  - Definition of Pushover loads .............................................................................................................................. 9
  - Calculations ....................................................................................................................................................... 11
  - Results ............................................................................................................................................................... 11
  - Improvements to design workflow .................................................................................................................... 13
  - New commands on the ribbon for faster analysis definition ............................................................................. 15
  - Mesh size as a property of elements .................................................................................................................. 15

**New Options & Improvements - Loads** ............................................................................................................ 16

- Possibility for quick isolating loads ..................................................................................................................... 16
- Notional loads generator ..................................................................................................................................... 16
- Update of the ASCE 7 code to the 7-16 version (US/CAN) ................................................................................. 18
- Displaying resultant forces .................................................................................................................................. 19

**New Options & Improvements - Results** ......................................................................................................... 21

- Diagrams on section cuts in 3D views .................................................................................................................... 21
- New symbols for support moment reactions ...................................................................................................... 22
- New options for quick changes of settings for diagrams .................................................................................. 23

**New Options & Improvements – Steel Design** ............................................................................................... 25

- Superelement for steel design (Eurocode) ........................................................................................................... 25
  - Definition .......................................................................................................................................................... 25
  - Design calculations ......................................................................................................................................... 27
- Separate results for deflections in both local directions ....................................................................................... 29
- Displaying Lateral-buckling lengths on mesh ...................................................................................................... 30
- Improvements to Advanced Stability ................................................................................................................ 30
- Possibility for using the LBT length as a buckling length (Eurocode) ................................................................. 31
- Update of the AISC code to 360-16 (US/CAN) ................................................................................................. 31
- Execution class parameters (Eurocode) .............................................................................................................. 32
- Improvements in intermediate restraints definitions ........................................................................................ 34
- Steel Connections – New tubular column base plate joints ............................................................................... 35
- Steel Connections – Asymmetric pinned base plate connection ..................................................................... 36
- Additional result on reports for the local bow imperfections on beams (Eurocode) ......................................... 36
- Additional result on the detailed Shape sheet report (Eurocode) ..................................................................... 36

**New Options & Improvements – Reinforcement Concrete Design** ................................................................. 37
What's New in GRAITEC Advance Design 2021

Punching shear improvements (Eurocode) ........................................................................................................ 37
New option for the minimum reinforcement on slabs (Eurocode) ................................................................. 40
The maximum crack value separately for the bottom and top side of the slab (Eurocode) .......................... 42
Additional info about areas per unit length when defining a real reinforcement for slabs ....................... 43
Possibility for selecting SLS combination types for selected verifications (Eurocode) ............................... 43
Report for concrete – New column on the Concrete deflection table ........................................................... 44
Update of the design ribbon ............................................................................................................................ 44
Main improvements to the Concrete design modules ..................................................................................... 46

NEW OPTIONS & IMPROVEMENTS - MISCELLANEOUS .............................................................................. 50
New start page .................................................................................................................................................. 50
Update of the ASTM steel material database (US/CAN) .................................................................................. 51
Update of the AISC section database (US/CAN) ............................................................................................... 52
Improvement of icons for enabling the snap mode and tooltips ................................................................... 53
Faster moving between cells on the Load Combination dialog .................................................................. 53
Additional formats for imperial units (US/CAN) ............................................................................................ 53
New About window with the network license status ....................................................................................... 54

MISCELLANEOUS IMPROVEMENTS & CORRECTIONS ............................................................................. 55
Welcome to Advance Design 2021

GRAITEC is very pleased to present the latest version of the leading structural analysis software - Advance Design 2021, 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 2021 is no exception, proving they are still top of their game in terms of providing top-level Construction, AEC and Building Design software solutions worldwide.

Version 2021 of Advance Design is enhanced with a lot of new functionalities, bringing high benefits to the end user, and is articulated around few main subjects:

- **New FEM calculations capabilities** – Pushover analysis;
- **New possibilities for steel design**, including a possibility for defining superelements, new steel connections and update of the design codes for the North American countries;
- **New possibilities for reinforcement concrete design**, including improvements to the shear punching verification and a new way for defining drawings on Reinforced Concrete modules;
- **New workflows for design calculations**, including the possibility of calculations on selection.

Version 2021 of Advance Design also comes with a big number of improvements and adjustments following the feedback received from thousands of users worldwide:

- the Notional loads generator,
- easy definition of the Fem mesh size for planar elements,
- diagrams on section cuts in 3D views,
- and many others.

Advance Design 2021 is the invaluable tool for all your projects.
New Options & Improvements - Calculation

This chapter contains a list with the main new options and improvements related to the general calculation possibilities.

Pushover analysis

Main features & benefits:
- New Pushover FEM analysis type
- Extended manual and automatic definition of Plastic Hinges
- Automatic generation of Pushover loads with extensive parameterization capabilities
- Wide range of available Results

A new advanced analysis type is available on Advance Design 2021 – the Pushover analysis.

The pushover is a method to predict the non-linear behavior of a structure under seismic loads. It can help demonstrate how progressive failure in buildings really occurs, and also identify the mode of final failure. The advantage of the pushover analysis is that the material nonlinearity and plastic hinging are considered but without the complications of the dynamic behavior.

The principle of the pushover method is applying lateral loads to the structure in an incremental manner and monitoring the occurrence of non-linear behavior (at fixed points called plastic hinges) in order to finally obtain a base shear versus control node displacement diagram.

Introduction to the Pushover method

The pushover analysis consists of several steps of calculations that need to be conducted in the following order.

- Determination of the seismic lateral load pattern

In order to perform the pushover analysis, we need to increment the lateral loads following a specified fixed pattern. There are many possible load patterns described in the literature and seismic standards. For example, loads can be applied on the gravity center of each story linearly increasing in height, where load values are based on the seismic base shear force.

- Defining plastic hinges in the model at locations where plasticity is expected to occur

During the pushover analysis the loads are incremented on the structure while plastic deformations are being constantly monitored. As plastic deformations are most likely to occur at specific locations, we define the non-linear behavior locally, on elements, via the plastic hinges, whilst maintaining the elastic behavior on all other elements. Generally, the behavior of plastic hinges is provided by seismic codes, in the form of tables or formulas that make it possible to construct the characteristic curves for plastic hinges. In the case of concrete elements, characteristic curves strongly depend on the provided reinforcement. For this reason, an initial classic linear seismic analysis should be conducted prior to the pushover analysis in order to provide an initial value for sections reinforcements.

- Pushover calculations

The pushover analysis is a list of sequential actions. First, linear finite element analysis is run. One of the results used further on is the reinforcements of elements, used in defining the characteristic curve of plastic hinges. Next, the lateral load pattern is obtained and it applies to the structure. Then, in an iterative process, these loads are gradually increased. At every increment the internal forces at the location of potential plastic hinges, the base shear and the control point displacement are monitored. When the internal forces at a potential plastic hinge reach a yielding level, the plastic hinge is activated according to its characteristic curve previously defined. The stiffness matrix is adjusted accordingly, and the finite element calculation is continued. The incrementing lateral load is continued, and the matrix update process is repeated for all activated plastic hinges. Calculations are continued until either: the target displacement is reached; the structure becomes a mechanism; analysis does not converge anymore; or a maximum number of steps is reached.
At every step of incrementation the displacement of a control point on the structure is recorded with its corresponding base shear value. This data is then plotted on a curve, called the pushover curve. It is initially linear at relative low values of base shear (the structure is still elastic), then becomes non-linear for higher values of base shear due to plastic deformations occurring in the structure.

Pushover analysis on Advance Design

Main features of the Pushover analysis in Advance Design 2021:

- Extended definition of Plastic Hinges
  - Plastic hinges (linear elastic-perfect plastic) can be easily defined on linear elements;
  - Available on the axial (Tx) and flexural (Ry and Rz) degrees of freedom;
  - Can be defined automatically and fully customizable with respect to FEMA 356 and EC8-3;
  - Automatic definition can be done for steel I - cross sections (IPE, HEA, W, ...) and concrete Square, Rectangular and T-shaped cross sections;
  - For concrete element plastic hinges can be computed using the real reinforcement (for Eurocode) or the theoretical reinforcement (North America codes);
  - Can be user defined – allows for applying plastic hinges on any type of cross section, for both steel and concrete linear elements.

- Automatic generation of pushover loads with extensive parameterization capabilities
  - Pushover point & surface loads are defined at each floor;
  - Possibility for selecting the load distribution on the height of the structure within several types: Concentrated, Uniform distributed, Triangular distributed, Parabolic distributed, User defined (fully customizable);
  - Possibility for computing the maximum total lateral load by using the Percentage of the total gravity loads, by Seismic base shear force on X, and by Seismic base shear force on Y;
  - Up to 8 load cases can be defined: 2 distributions (as required by FEMA356 and EC8-3) and 4 directions (+/-X, +/-Y).

- Wide range of available Results
  - FEM results and reports;
  - The pushover force-displacement curve;
  - Reports tables with status of hinges and the overstrength ratio (αu/α1);
  - Graphical results showing the status of hinges at each load step.

Let's take a closer look at the next steps of the process. We start from the stage when a model is already prepared for linear statics calculations (including defined geometry, levels, loads, etc.).

Definition of plastic hinges

In order to perform the pushover analysis, the user first needs to define the plastic hinges at locations where they are expected to occur (ends of beams), or at locations where their arise needs to be monitored (ends of columns). The plastic hinges can be defined on individual linear elements from the properties panel.
The user is able to select the degrees of freedom for which this hinge is applicable, separate for each extremity. The ID name of a plastic hinge is generated automatically, and it consists of prefix PLH-L (plastic hinge on linear element), ID of the element, the extremity (1 or 2) and the type of the element (B – for beams, C for columns). The definition of parameters of the plastic hinge can be done by using a dialog opened by a button on the Definition property.

In a case when the user decides that parameters should be calculated automatically, then he can select the code (EC 8-3 or FEMA 356) and plastic hinge type. The available types (steel or concrete beams and columns) depend on the selected code and degree of freedom. Note that some of the parameters are computed only during the next stage, during the pushover analysis. In a case when the user decides to define the properties of the plastic hinge manually, the Definition should be set to User defined. Then, each property can be unlocked and edited individually.

When plastic hinges are applied to elements, they can be presented graphically (on the descriptive model) by using a grey symbol.
Definition of Pushover loads

The next stage is the creation of pushover load cases and generation of pushover loads. For this, a new Pushover load case family type can be defined from the Create load case family. On its property list we can set the basic data for load generation such as: the distribution type, the point of application and the directions of the loads.

Looking on the distribution types - there are several distribution types of the pushover forces on the height of the structure available:
What's New in GRAITEC Advance Design 2021

<table>
<thead>
<tr>
<th>Concentrated</th>
<th>Uniform distributed</th>
<th>Triangular distributed</th>
<th>Parabolic distributed</th>
<th>User defined</th>
</tr>
</thead>
</table>

(\(V_b\) is a maximum total lateral load, \(F_n\) is the maximum lateral load applied on level \(n\)).

Using the right click menu on the Pushover load case family we can then automatically generate the pushover load cases and loads. On the property list of each generated pushover load case we can set details related to the maximum total lateral load.

The maximum total lateral load is the cumulated sum of the lateral loads applied on the last step of the pushover analysis. This load can be defined either as the imposed value or as a percentage of the load applied on the structure, prior to the pushover. For each load case, a different definition of the maximum lateral load can be selected.

The \textit{Master node} is used for tracking the displacement of the structure and generating the pushover load-displacement curve. This node can be either defined (as an ID of a mesh node), or the \textit{Max displacement} option can be used. In this case, the maximum displacement, on the direction of the pushover load case, at each step of the analysis will be used for plotting the pushover curve.

Similar to the classical NL analysis, additional calculation conditions can be set for the PushOver Analysis as well. The analysis could either run until the total lateral load is applied (last step) or it could be stopped earlier due to the instability of the non-linear calculations – usually when a mechanism state is reached. In this case the results will be available for the calculated steps.
Calculations

The pushover analysis is a list of sequential actions, activated by a dedicated Pushover checkbox control in Calculation sequence dialog.

During the process several steps are performed automatically, including:
- a standard linear static and seismic calculation;
- the design of steel linear elements / design of concrete linear elements (including the real reinforcement); and finally, the main non-linear static calculation for the pushover load cases with incrementing lateral loads and an appropriate activation of plastic hinges.

Results

After successful completion of pushover calculations, a set of different types of results is available.

FEM results

As with normal static calculations, FEM results such as displacements and internal forces are available. The results can be checked as for the non-linear calculations for each of the subsequent calculation steps.

The pushover force-displacement curve

Using a new Pushover results curve command, available on the Results ribbon, a pushover capacity curve can be generated. It displays a relationship diagram of the displacement of the node with respect to the total applied lateral load.
Reports tables

For the results from the pushover analysis a set of new dedicated report tables is available, including:

- Flexural plastic hinges status by load step

<table>
<thead>
<tr>
<th>Load case id - name</th>
<th>Load step</th>
<th>Plastic Hinge</th>
<th>Rx (°)</th>
<th>My (kN·m)</th>
<th>Ry_pl (°)</th>
<th>My_pl (kN·m)</th>
<th>Status</th>
<th>Rz (°)</th>
<th>Mz (kN·m)</th>
<th>Rz_pl (°)</th>
<th>Mz_pl (kN·m)</th>
<th>Status</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td></td>
<td>Rx4.470.1 B</td>
<td>-1.06</td>
<td>314.47</td>
<td>0.88</td>
<td>54.12</td>
<td>Dp.0D</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Rx4.470.2 B</td>
<td>0.00</td>
<td>364.86</td>
<td>0.86</td>
<td>274.47</td>
<td>0.88</td>
<td>54.12</td>
<td>Dp.0D</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Rx4.700.1 B</td>
<td>-0.88</td>
<td>136.29</td>
<td>1.82</td>
<td>152.26</td>
<td>0.88</td>
<td>54.12</td>
<td>Dp.0D</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Rx4.700.2 B</td>
<td>-0.02</td>
<td>660.84</td>
<td>0.71</td>
<td>152.26</td>
<td>0.88</td>
<td>54.12</td>
<td>Dp.0D</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Rx4.700.3 B</td>
<td>-1.76</td>
<td>908.95</td>
<td>0.85</td>
<td>152.26</td>
<td>0.88</td>
<td>54.12</td>
<td>Dp.0D</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
</tbody>
</table>

- Axial plastic hinges status by load step

<table>
<thead>
<tr>
<th>Load case id - name</th>
<th>Load step</th>
<th>Plastic Hinge</th>
<th>Ax (cm)</th>
<th>N (kN)</th>
<th>Tension</th>
<th>Compression</th>
<th>Status</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td></td>
<td>Rx4.470.1 B</td>
<td>0.00</td>
<td>18.45</td>
<td>1.05e6</td>
<td>660.26</td>
<td>-0.05</td>
</tr>
<tr>
<td>Rx4.470.2 B</td>
<td>0.00</td>
<td>18.45</td>
<td>1.05e6</td>
<td>660.26</td>
<td>-0.05</td>
<td>-32.23</td>
<td>34</td>
</tr>
<tr>
<td>Rx4.700.1 B</td>
<td>0.00</td>
<td>15.83</td>
<td>1.05e6</td>
<td>660.26</td>
<td>-0.05</td>
<td>-32.23</td>
<td>34</td>
</tr>
<tr>
<td>Rx4.700.2 B</td>
<td>0.00</td>
<td>15.83</td>
<td>1.05e6</td>
<td>660.26</td>
<td>-0.05</td>
<td>-32.23</td>
<td>34</td>
</tr>
<tr>
<td>Rx4.700.3 B</td>
<td>0.00</td>
<td>15.83</td>
<td>1.05e6</td>
<td>660.26</td>
<td>-0.05</td>
<td>-32.23</td>
<td>34</td>
</tr>
</tbody>
</table>

- The overstrength ratio (αu/α1)

Graphical results showing the status of hinges at each load step

A new Pushover Results entry is available on the FEM results selection that allows selecting the Hinge status result for linear elements. When activated, it shows the status of defined plastic hinges for selected step of the selected pushover case. The status is displayed by using colors.
Improvements to design workflow

Main features & benefits:

- Possibility for running design calculation for a selection
- Full flexibility while design calculations

For Advance Design 2021 major changes have been made to the internal mechanism of storage of calculation results, which results in new and much more flexible working scenarios for design calculations. Let’s look at two of the most visible changes and their impact on everyday work.

Saving design results for elements

After the design calculations (for steel, reinforced concrete or wooden elements), the results for these elements are kept until the next FEM calculation. This allows multiple design calculations to be performed for different ranges of structures, without losing the results for the remaining elements. This means that design calculations can be carried out many times, e.g. by iteratively changing design parameters for one element only, without worrying that results for the remaining elements will be lost.

To quickly find out what the current status of the calculation is for a given element, the best way is to use the status of the Design Results parameter. Let’s see an example for a steel element.

- When the element has not been calculated yet, there are no design results available and the Design Results parameter has a Not available value.

- Once the element has been calculated, the design results are already available, and the Design Results parameter has an Available value.
What's New in GRAITEC Advance Design 2021

- If there is any change in the Design parameters of this element (e.g. change of template), then the parameter takes the Outdated status. This means that the results for this element are still available but may be out of date.

Of course, the design recalculation will update results and restore the Available status, while the FEM recalculation will remove design results and restore the Not available status.

Please note that disabling the To calculate parameter before the design calculation, as before, excludes the element from the calculation, but if you disable this parameter after the calculation, it additionally disables the display of results for this element (it is treated as if it were not calculated). Enabling the parameter again restores displaying results for this element.

Design calculation for selection

Another related new functionality is the possibility of running design calculations for selected elements. This allows for quick (e.g. graphical) element selection and design of these elements, as well as, for example, quick recalculation for even one element after changing its design parameters.

This makes the whole process easier - for example, you can perform calculations system by system.
New commands on the ribbon for faster analysis definition

Main features & benefits:
- Easier addition of new types of advanced analysis

To simplify the process of defining new advanced analyses, a small improvement has been made to the Analysis ribbon, where a new group Analysis is added that contains three commands to add analyses: Modal, Static Nonlinear and Generalized buckling.

These commands are used to add 3 types of advanced analyses to the current project, as an additional and faster method to add analyses from the Project Browser context menu.

Mesh size as a property of elements

Main features & benefits:
- Very quick mesh size setting for surface elements

To make the definition of the FEM mesh for planar elements easier and faster, you can now easily set the size of the desired mesh for the selected planar elements. For this purpose, a new Element size category has been added to the Density list in the mesh properties of the planar elements, with a corresponding field for entering the mesh size value.

This allows you to quickly set the mesh size for a selected element only, regardless of global settings and without the need of using much more labor-intensive methods for detailed mesh setting.
New Options & Improvements - Loads

This chapter contains the list of main new options and improvements related to the displaying of calculation results.

Possibility for quick isolating loads

<table>
<thead>
<tr>
<th>Main features &amp; benefits:</th>
</tr>
</thead>
<tbody>
<tr>
<td>▪ Smoother work with fast load selection</td>
</tr>
</tbody>
</table>

A small, but very useful improvement has been made to the load selection methods - when selecting loads from the Project Browser (either load cases or load definitions), we can use the *Isolate* command.

It works similar to the same commands available for other structural elements on the Project Browser and allows for very quick (by one click) selection of loads to be displayed.

Notional loads generator

<table>
<thead>
<tr>
<th>Main features &amp; benefits:</th>
</tr>
</thead>
<tbody>
<tr>
<td>▪ Easy to use generator of horizontal loads based on vertical loads</td>
</tr>
<tr>
<td>▪ Flexibility of application due to configuration possibilities</td>
</tr>
</tbody>
</table>

Notional loads are fictitious horizontal loads that create a horizontal deformation of the structure which causes an amplification of the forces and deformations. They are used especially by the North America steel codes, where notional loads are applied to the structure for steel design to consider the P-Delta effects.
The notional loads are calculated as a portion of the vertical loads, by using a conversion coefficient.

There are various considerations used in each standard for defining notional loads. Differences relate to the way the coefficient is determined, the direction of the force or the way it is combined with other loads. For example, the Canadian CSA-S16 steel code uses 0.005 factor to vertical loads whereas Eurocode 3 employs a notional load calculated with consideration of the number of columns in a story and the height of the frame. Also depending on the standard, notional loads should be applied in orthogonal directions or only in the direction of sway and are used in combination with all or selected loads.

Advance Design offers a universal and simple solution that allows users for a high flexibility for using notional loads in various scenarios.

A new command Create Notional Loads is available on two locations - on the Object ribbon and on the context menu, available when clicked on load cases in the Project Browser.

When called, it opens a new dialog, where the user can set which load cases are converted and the value of the conversion coefficient - by default is equal to 0.005 (0.5%).

The Notional loads direction allows for selection up to all 4 directions for new converted forces while the Load case family list is used for selecting a family for newly created cases. After using the Apply button a set of new load cases is generated, one per selected direction (X+, X-, Y+, Y-).

Each new load case includes converted loads from all selected load cases. Converted loads are punctual, linear and planar having FZ value < 0 (only downward loads are used). New load values are created on the corresponding horizontal direction, for example for X- direction values are calculated as FX = - |FZ| * coefficient. In the case of a dead load that contains a self-weight (defined with the Gravity field) on a new case the Z value of Gravity field is converted in a similar way as above loads, as a value on X or Y direction with a corresponding sign.
What's New in GRAITEC Advance Design 2021

Example:
Loads on a source loads case:

Notional loads on X+ direction, created using the 0.005 coefficient:

Update of the ASCE 7 code to the 7-16 version (US/CAN)

Main features & benefits:
- The latest version of the load standard ASCE 7-16 is available

A new version of the American standard for loads ASCE 7-16 has been introduced to Advance Design 2021. The ASCE 7-16 code (Minimum Design Loads for Buildings and Other Structures, ASCE/SEI 7-16) is now available for selection on the standards list for Combinations, Seismic and Climatic:

<table>
<thead>
<tr>
<th>Standards</th>
<th>Method</th>
</tr>
</thead>
<tbody>
<tr>
<td>Combinations</td>
<td>ASD</td>
</tr>
<tr>
<td>Seismic</td>
<td>ASD</td>
</tr>
<tr>
<td>Climatic</td>
<td>ASD</td>
</tr>
</tbody>
</table>

The changes with respect to the previous version of the standard (ASCE 7-10) mainly relate to climatic and seismic loads, including, among others:

- For wind loads, a new \( K_e \) factor for the wind velocity pressure is added (available on the property list of wind load families), to account for variations in air density at different altitudes.
- For wind loads, a new \emph{Partially open} selection for the building enclosure category is available on the dialog with parameters for internal pressure coefficients (available on the property list of load areas).
For seismic loads, values of site coefficients $F_a$ and $F_v$ have been updated. And as values for class B and D can change depending on if a site-specific velocity measurement was performed or not, the list of site classes, available on the seismic load case families, have been expanded.

For wind loads, the list of regions with load speed has been expanded with new entries.

**Note:** The map with snow load values available in the program has not changed, however, for 7 states the data with snow values are now given in the ASCE 7-16 standard in tabular form, separately for each county. Therefore, for the states listed below it is recommended to enter the values according to the table in the standard: Colorado, Idaho, Montana, Washington, New Mexico, Oregon and New Hampshire.

**Displaying resultant forces**

**Main features & benefits:**
- Easier verification of load definitions by displaying the resultant forces

A new annotation attribute called *Intensity (resultant)* is added to the Display settings. It works with loads (Point Load, Linear Load and Planar Load) and displays a resultant value for defined forces or moments. The resultant is calculated as the square root of the sum of squares of the force components: $F = \sqrt{F_x^2 + F_y^2 + F_z^2}$.

A new annotation type is especially helpful for displaying values of forces from wind if perpendicular to the sloped part, as in the exemplary pictures below. In the left picture the annotation type for planar loads is set to
**Intensity**, thus load values are displayed separately for all components. In the right picture the annotation type for planar loads is set to a new **Intensity (resultant)** type, thus only one value of the resultant force is displayed.
**New Options & Improvements - Results**

This chapter contains a list of the main new options and improvements related to the display of calculation results.

**Diagrams on section cuts in 3D views**

<table>
<thead>
<tr>
<th>Main features &amp; benefits:</th>
</tr>
</thead>
<tbody>
<tr>
<td>- Possibility for displaying diagrams with FEM results from planar elements on section cuts in 3D views</td>
</tr>
<tr>
<td>- Easier to postprocess graphical results</td>
</tr>
</tbody>
</table>

A new display method is available for FEM results on planar elements – diagrams on section cuts on model view. It allows for the display directly on the model (3D) view diagrams on any section defined by the user. Diagrams are generated based on predefined section cuts (Results/Postprocessing/Section cuts) - the same ones that are used to display Result Curves (Results/Postprocessing/Result Curves).

The settings for section cut diagrams are independent of the settings for the results of the planar elements but can be combined with them - this allows the diagrams and e.g. maps or isolines to be displayed simultaneously.

In addition, in order to facilitate the presentation of graphs in section cuts, e.g. on plane views, it is possible to set their display in the plane of the element.
As before, it is possible to manage the visibility of defined section cuts with the Project Browser. But there is also a quick method available to display the diagram only for selected cuts - simply select them and use the Post Processing button.

New symbols for support moment reactions

Main features & benefits:
- Easier graphic distinction of support reactions

The graphic symbol for moments vectors as support reactions has been modified to make it easier to distinguish forces from moments. Now the symbol for forces is an arrow and the symbol for moments is a double arrow.
New options for quick changes of settings for diagrams

Main features & benefits:
- Results for supports directly from the ribbon
- Faster access to configuration of graphical results

To increase the ease of access to typical graphical results and facilitate their configuration, three new commands have been added to the Result ribbon:

The first one is a button to switch on/off values on diagrams:

The second one is a slider to easily change the size of fonts used for values on diagrams:
The third one is a slider to easily change the scale of the diagram:

All above options were already available on the result configuration window, but thanks to being added to the ribbon the comfort and speed of these settings have been significantly increased.
New Options & Improvements – Steel Design

This chapter contains a list of the main new options and improvements related to the Steel design.

Superelement for steel design (Eurocode)

Main features & benefits:
- Possibility for grouping linear elements by using the superelement concept
- Possibility for using superelements for steel verifications (Eurocode)

Starting with Advance Design 2021 a new type of compound object is available - Superelement. A superelement is an element which consists of a set of individual linear elements. It's a kind of a group of linear elements, which is used as a single element for various checks.

A simple example of where the use of superelement is beneficial: verification of limit deflection for a roof that is modeled by several linear elements.

Definition

The definition of a superelement can be done in many ways, including by using the Create command from the right-click menu or from the ribbon, as well as by using the List property, available on the property list of linear elements:
For the elements to be combined into one super element, they must meet several conditions, including joining at the ends, having the same material category, having the same cross-section type and the same orientation (Z axis of the local system facing the same direction).

Each newly created super element has its own unique ID number. It can be used, among other things, for selection or for displaying on a model view, thanks to the new type of annotation for linear elements.

In addition, there is the possibility to set the color for each super element separately. With the new *Super element* display mode you can show them by color - the elements forming a super element are displayed in a common color.

To make the graphic selection of super-elements easier, a new command is available in the right click menu: *Super element selection*. It can also be enabled by using the shortcut Alt+S. When this mode is activated, then by clicking on any of the elements included in the super element, the remaining elements are also selected.
Design calculations

The superelement concept is used on Advance Design 2021 for the standard check of steel elements; therefore a number of new options are available on design parameters of steel elements.

Note: Please be aware that superelement options are for the moment considered only for Eurocode 3 design calculations.

On the property list of steel linear elements, a set of new options dedicated to the superelement verification are available on the Deflection group. When the Super Element Verification option is activated then the Super Element Deflection group of properties is available for editing. Properties are the same as on the above Deflection group (used for a single element verification) but apply to the superelement.

The results of the deflection verification can be checked separately for the element and the superelement either Graphically, using the postprocessing diagrams for deflections, or on the Deflection tab on the Shape sheet dialog.
On the Buckling dialog, the list for the selection of a calculation method of buckling lengths has been modified. New entries for supporting a super element are added and the naming of some of the existing ones has been modified. For example, on the list for the Lfz there are the following entries:

- **Auto calc.** - calculates the buckling length automatically;
- **Imposed value** – uses the entered buckling length value;
- **Element ratio** – calculates the buckling length as a ratio of the element length;
- **Mesh size ratio** – calculates the buckling length as a ratio of the finite element length;
- **Super element ratio** – calculates the buckling length as a ratio of the super element length;

In a similar way the list of available options for the calculation of the Lateral-torsional buckling length (on the Lateral-torsional buckling dialog) has been updated. Note that the content on the list depends on whether the dialog is opened for a superelement (left side image) or an element (right side image) that is not part of any superelement. When opened for a superelement, the list contains only two items *Auto calc.* and *Super element ratio*.

A new option is also available on the Calculation settings dialog, on the Optimization tab. When the *Apply new sections to the whole super element* option is activated, then during the steel section optimization, when the user accepts a new section for an element that is part of super element, the other elements are updated as well.

Additionally, the *Detect intersection along a super element for automatic buckling lengths* option, available on the Buckling tab of the Calculation settings dialog, with the name change has also been developed. If it is not activated, then the auto calculation is performed considering the super element as a single member, no matter the intermediate intersections with the other members. If it is activated, then it calculates the buckling lengths separately for each segment/element of superelement, considering the intermediate intersections.
Separate results for deflections in both local directions

Main features & benefits:
- Separated presentation of the deflection for both local directions

Deflection results coming from the steel analysis are now available independently for both local directions - local Y and local Z. These results are particularly useful for spatial analysis of columns or beams bending in 2 directions.

Separate deflection result can be seen in the Shape sheet, on the Deflection tab and also on the Deflection verification report table.
Displaying Lateral-buckling lengths on mesh

Main features & benefits:
- Better control thanks to the more detailed presentation of LTB data and results

Values of Lateral-buckling lengths and related results when displayed graphically are now presented separately for each mesh elements (similar as that available for buckling lengths).

This change is intended to improve the presentation of results, especially where the distribution of the finite element mesh is not uniform.

Improvements to Advanced Stability

Main features & benefits:
- Enables the Advanced stability method on compound sections.
- Better control of the process thanks to more detailed messages

In Advance Design 2021 several improvements have been made to the Steel Design with Advanced Stability solver (2nd order FEM calculation with 7DOFs).

Compound sections

Advance Design 2021 now offers the possibility to use the compound sections in Steel Design with Advanced Stability solver. The output of such calculation is like for any single section calculated with Advanced Stability, i.e. a general verification of combined stresses.

More detailed warnings

If during the calculation with the Advanced Stability solver for selected degrees of freedom at the ends of a given element the correct information about the stiffness is missing (or is too small), then an appropriate warning message is displayed. With Advance Design 2021 such messages are more detailed and contain the information about the corresponding DoF (degree of freedom). Messages are now also displayed separately for each DoF. This makes it much easier to correct the settings and control the calculations.
Possibility for using the LBT length as a buckling length (Eurocode)

Main features & benefits:
- Easier and quicker definition of buckling parameters

When defining the buckling parameters for steel elements (according to the Eurocode), it is possible to set the buckling length in the plane of smaller inertia ($L_{fy}$) as equal to the LTB length, either calculated for the lower or upper flange of the element. This makes it easier to parameterize the structure and speeds up work.

Although this new option was already introduced in the previous update 2020.2, it is worth mentioning because it was not officially announced and the need for such functionality has been reported by many users.

Update of the AISC code to 360-16 (US/CAN)

Main features & benefits:
- The latest version of the steel standard AISC 360-16 is available

A new version of the American standard for steel design AISC 360-16 has been introduced to Advance Design 2021. The ANSI/AISC 360-16 code (An American National Standard ANSI/AISC 360-16; Specification for Structural Steel Buildings; published July 7, 2016, by the American Institute of Steel Construction) is now available for selection on the list of steel codes.
The scope of changes made with respect to the previous version of the standard (ASIC 236-10) includes:

- Modification of naming and designation of selected parameters in accordance with AISC.
- Modification of calculation procedures and parameter conditions, including among others:
  - Calculation of slender element members in compression
  - Design HSS, T and 2L members for flexure
  - Design I-shaped members for shear.
- Updated steel material database with new ASTM steel materials.
- Updated steel section database with new ASTM steel sections (HSS acc. ASTM A1085).
- Updated steel section database in accordance with AISC Shapes database v15.0.

### Execution class parameters (Eurocode)

<table>
<thead>
<tr>
<th>Main features &amp; benefits:</th>
</tr>
</thead>
<tbody>
<tr>
<td>- Possibility for assigning the classification of steel elements according to the EN1090-2 standard</td>
</tr>
<tr>
<td>- New types of Bill of quantities for steel elements</td>
</tr>
</tbody>
</table>

Two new parameters are introduced for steel linear element: the **Execution class** and **Family of elements**. None of the parameters affect the calculations, and are only parameters for the classification of steel elements in accordance with the EN1090-2 standard.

There are 4 **Execution classes** introduced with the EN1090-2: EXC1, EXC2, EXC3 and EXC4. Each class mandates its own set of requirements, with complexity increasing as the number rises. EXC2 is the most common / default specification.

The definition of the Execution class could be done either globally or locally. The global definition is available on the **Options-Application** dialog and defines the class for the whole structure – it’s used as default for all steel elements.

In case of a local definition, the user can select one of the classes directly from the **Execution class value** list, available on the list of design parameters (acc. Eurocode) of linear steel elements. The Execution class type parameter is used for selecting which global/local definition is used individually.
The **Family of elements** property is introduced in French comment to NF EN1090-2. We have 8 families, marked by the letters A, B, C, D, E, D+, E+. The Family of elements is an auxiliary parameter, used in practice to select the Execution class.

To make the work easier, the new parameters can be used for element selection.

Four new tables have been added to the list of available report tables with the Bill of quantities of linear elements to use the entered classification data.

The bill of quantities of linear elements by execution class / by family of element generates separate tables for each execution class / family of element within defined on the model.
Summary tables generate a summary list for all elements by execution classes / family of element.

<table>
<thead>
<tr>
<th>Execution class</th>
<th>Quantity</th>
<th>Length (m)</th>
<th>Surface (m²)</th>
<th>Weight (T)</th>
</tr>
</thead>
<tbody>
<tr>
<td>BC12</td>
<td>10</td>
<td>50.95</td>
<td>109.22</td>
<td>5.85</td>
</tr>
<tr>
<td>BC23</td>
<td>4</td>
<td>22.25</td>
<td>44.50</td>
<td>1.50</td>
</tr>
<tr>
<td>Total</td>
<td></td>
<td></td>
<td>154.21</td>
<td>8.35</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Family of element</th>
<th>Quantity</th>
<th>Length (m)</th>
<th>Surface (m²)</th>
<th>Weight (T)</th>
</tr>
</thead>
<tbody>
<tr>
<td>E</td>
<td>10</td>
<td>50.95</td>
<td>109.22</td>
<td>5.85</td>
</tr>
<tr>
<td>Total</td>
<td></td>
<td></td>
<td>154.21</td>
<td>8.35</td>
</tr>
</tbody>
</table>

### Improvements in intermediate restraints definitions

Main features & benefits:
- Faster and more effective method of graphical definition of intermediate restraints

One of the available methods for defining intermediate restraints for the lateral-torsional buckling analysis is the possibility for a graphical indicating points on the axis of the selected element. Until now, the definition process had to be carried out individually for pinned and fixed restraints, separately for the lower and the upper flange.

In the latest version, this process has been greatly improved by adding to the right click menu a new entry, **On points**, that allows changing the flange and restraint type during each point selection. Thanks to this it is possible to define pinned and fixed restraints on any flange configuration at one calling of the command.

When indicating the points, on the command line a prompt is continuously visible for selecting the flange (U – for upper, L for lower, B – for both) and restraint type (P – for pinned, F – for fixed). Note that all the above key letters are universal for all language versions.

Any time during the definition the user can press any of the P/F/U/L/B keys to modify parameters for next points (next restraints). Each time any of the keys is pressed, the command line changes marking the modified key by a rectangular bracket.
Steel Connections – New tubular column base plate joints

Main features & benefits:
- Possibility of modeling base plate connection for hollow section columns
- Possibility of design for new types of connections

Advance Design 2021 allows definition and export for design of the base plate connection for tubular columns. New connections are created by using the Base Plate connection button that can be found on the Connection panel on the Objects ribbon.

New connections can be designed using the built-in Steel Connections module. The design calculations can be done for tubular columns defined with square, rectangular or round section. In case of circular sections there are two shapes of a base plate available: rectangular and circular.

This new connection category is analyzed as a 3D connection - for a full set of forces: a vertical force (N), shear forces (Vy, Vz) and bending moments (My, Mz) in both horizontal directions and a torsional moment (Mt) in column.

Different types of stiffeners can be used for the new connection type, depending on the section of the column. Diagonal stiffeners are available for all hollow section types, while side and outer stiffeners are available for rectangular and square columns.
Steel Connections – Asymmetric pinned base plate connection

Main features & benefits:
- Possibility for defining base plate as asymmetrically shortened
- Allows for modeling asymmetric configuration with sloped or vertical stiffeners

When designing a pinned base plate connection for I-section columns using the built-in Steel Connections module, it is possible to define the geometry of the base plate as shortened separately for each direction. This allows modeling and calculation of the asymmetrical system with sloped or vertical stiffeners.

Additional result on reports for the local bow imperfections on beams (Eurocode)

Main features & benefits:
- Better control thanks to more detailed documentation

To the EC3 Local bow imperfections on beams report table, an extra column for k value is added. The k value is used for adopting the imperfections when considering the lateral torsional buckling of a member in bending for a second order analysis. The k value can be imposed by the user or automatically determined in accordance with the National Annex to EN 1993-1-1.

Additional result on the detailed Shape sheet report (Eurocode)

Main features & benefits:
- Better control thanks to more detailed documentation

On the detailed shape sheet generated for the steel beams calculated according to Eurocode, the $M_{cr0}$ value is added to the list of displayed auxiliary values. The $M_{cr0}$ value is used for calculation of the $\lambda_0$ value during the lateral-torsional buckling analysis.
New Options & Improvements – Reinforcement Concrete Design

This chapter contains a list of the main new options and improvements related to the Reinforcement Concrete Design.

Punching shear improvements (Eurocode)

Main features & benefits:

- Shear punching over punctual supports and below concentrated forces
- Detailed calculation of the punching reinforcement

In Advance Design 2021 the shear punching calculations according to Eurocode have been greatly expanded. The changes cover many aspects, including punching verification on slabs under concentrated forces and over punctual supports, the ability to set parameters separately for each slab, a wider range of verifications, calculations of punching reinforcement, more detailed reports and other improvements.

New types of verification points

The shear punching can now be checked also over punctual supports and below concentrated forces. The parameters required for shear punching calculations appear in the properties of these elements, including the possibility of automatic or manual setting the load eccentricity $\beta$ factor value.

Possibility for deactivating analysis

On the property list for columns, punctual forces and supports, a new None item has been added to the list with the $\beta$ factor, which allows deactivating the punching verification on a given point.

Improvements to control perimeters determination mechanisms

The mechanism for automatic detection of a position of punching object and determining the control perimeter has been improved. Improvements are related mainly to considering the real distance to the edge of the slab (considering a projection) during determining the control perimeter for loaded areas close to or at an edge or a corner of a slab, when the Auto detection of punching object position is used.

The calculation of punching reinforcement per perimeter

A new detailed calculation of punching reinforcement per perimeter has been added. Below is a summary of the successive steps of the calculation and a description of the results obtained concerning the reinforcement. All calculations are according to §6.4.3 from EN1992-1-1.

- Shear checks at the perimeter of the loaded area and at the basic control perimeter
  
  In this step the punching shear resistance is calculated along the perimeter of the loaded area ($u_0$) and along the basic control ($u_1$). After checks performed in this step it is determined if punching shear reinforcement is required.

- The calculation of the outermost control perimeter ($u_{out}$): it is the perimeter at which shear reinforcement is no longer required.

- The calculation of the outermost (last) perimeter of shear reinforcement ($u_{last}$). It is placed at a distance not greater than $kd$ within the outermost control perimeter ($u_{out}$)
• The position of first perimeter of shear reinforcement. It is placed at a 0.5d distance from column face.
• The number of punching perimeters and the radial (equal) spacing between punching reinforcement perimeters. The maximum spacing is equal to 0.75d.
• The required reinforcement per each perimeter of punching reinforcement (Asw). The reinforcement is calculated using the 6.52 formula from amendment A1 (February 2015) to EN1992-1-1. During this step it is verified if the resistance of the slab with the calculated punching shear reinforcement $V_{Rd,cs}$ is not exceeding the limit value $k_{max} \cdot V_{Rd,cs}$. If the condition is not met, an appropriate error message is displayed.

The reinforcement calculation is implemented in two main scenarios - when the tangential spacing or link leg diameter is imposed. After the calculation the following results are available per each perimeter (results can be checked using a new report table):

- position of perimeter as distance from column face,
- length of perimeter,
- tangential spacing between shear reinforcement links,
- number of link leg around perimeter,
- provided diameter of one link leg,
- min area of one link leg,
- requested reinforcement per perimeter,
- provided reinforcement per perimeter.

Separate shear punching settings for each slab

On previous versions of Advance Design, the punching verification parameters were defined globally for all slabs. Now such settings are defined locally, separately for each planar element, using a new set of new parameters on the Property list:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Verification</td>
<td>Enabled</td>
</tr>
<tr>
<td>Method</td>
<td>With longitudinal reinforcement</td>
</tr>
<tr>
<td>Correction of long. reinforcement</td>
<td>Disabled</td>
</tr>
<tr>
<td>Punching reinforcement design</td>
<td>Enabled</td>
</tr>
<tr>
<td>Punching reinforcement angle</td>
<td>90.00 °</td>
</tr>
<tr>
<td>Provided reinforcement</td>
<td>Tangential spacing (st) imposed</td>
</tr>
<tr>
<td>Tangential spacing (st)</td>
<td>Max allowable (st, max)</td>
</tr>
<tr>
<td>Value (st)</td>
<td>0.0 mm</td>
</tr>
<tr>
<td>Link diameter</td>
<td>Auto</td>
</tr>
</tbody>
</table>
Some options are the same as they were previously available globally, including:

- The selection of the method for verification with or without longitudinal reinforcement;
- The option for activating the design of punching reinforcement;
- The angle of the punching reinforcement.

And there are new options:

- The Verification checkbox
  This option lets the user deactivate the punching verification on specific elements.
- The Correction of longitudinal reinforcement checkbox
  Automatically increases longitudinal reinforcement (Ax and Ay, up to the limit ratio = 0.02) when the shear check is not fulfilled (\(v_{Ed} > v_{Rd,c}\)).

\[ \rho_L = \sqrt{\rho_{Lx} \cdot \rho_{L_y}} \leq 0.02 \]

It is used to apply the maximum allowable longitudinal reinforcement ratio to avoid defining the punching reinforcement.

Note: If both Correction of longitudinal reinforcement and Punching reinforcement design options are enabled, Advance Design will first try to satisfy the punching verification by increasing the longitudinal reinforcement.

- The Provided reinforcement list
  It allows for deciding the method of calculating the provided reinforcement. Two options are available:
  - Tangential spacing (st) imposed – allows for setting the link spacing (st) and letting automatically find out the diameter and the number of link legs
  - Link leg diameter imposed – allows for setting the link leg diameter and letting automatically find out their number and spacings.
- The Tangential spacing list
  In case the user wants to impose the spacing (st), the option lets him select how to determine the value:
  - Max allowable (st,max) - for all perimeters the maximum allowable spacing value is used
  - Variable (st) – spacings are calculated separately for each perimeter to obtain an optimum provided area.
  - Value imposed - for all perimeters the same imposed value is used. If selected, the Value (st) property is available for entering the spacing value.
- The Link diameter
  In case the user wants to impose the link leg diameter, the option lets him select how to determine the value by using a small dialog:

If the Auto link diameter option is active (default), then it automatically finds the first (smallest from list) diameter that satisfies all requirements. If the option is not active, then the diameter value can be either selected from the list of available diameters (activated for transversal reinforcement in global Concrete Design assumptions) or entered manually.

**Improvements to the report table for calculated punching reinforcement**

The table with details about the punching reinforcement has been completely modified by adding new columns and removing or modifying existing ones. It now shows new information on perimeters, more details for verification and quantities of required reinforcement per perimeter.
New report table for provided punching reinforcement

A new report table is available for the provided reinforcement on each perimeter. It now shows information on the details required to place the reinforcement for each perimeter, including its position from the loaded area, the length of a perimeter, the number of links, spacings, diameters and related reinforcement areas.

Other changes

In addition to the above changes, various types of corrections have been introduced, including:

- For a calculation of the reinforcement percentage used during a calculation of the punching shear resistance of a slab without punching shear reinforcement along the basic control perimeter, the reinforcement areas are taken into account over a distance equal to Column width + 3d (instead 2d) on each side (§6.4.4(1) from EN1992-1-1).
- Fixed a problem of punching effort sign related to the effective height calculations in the case of slabs having different top and bottom concrete covers.

New option for the minimum reinforcement on slabs (Eurocode)

Main features & benefits:
- Greater flexibility in the reinforcement calculation for planar elements

To extend the control over the definition of minimum reinforcement in slabs a new Coverage of minimum reinforcement option has been added to the property list of concrete planar elements (Eurocode). The new option is available for selection only if the Minimal longitudinal reinforcement option is enabled.
Case 1 - no consideration of minimum reinforcement

You can see areas without the required reinforcement (0.00, blue color), areas with higher values (red color) and other areas with intermediate values.

Case 2 - the minimum reinforcement is applied to the entire element

The minimum reinforcement is applied to all areas where the calculated reinforcement area is smaller than the minimum, including also areas without the required reinforcement.

Case 3 - the minimum reinforcement is applied only to tensioned areas

The minimum reinforcement is applied only to areas where the calculated reinforcement area is bigger than zero.
What's New in GRAITEC Advance Design 2021

The maximum crack value separately for the bottom and top side of the slab (Eurocode)

Main features & benefits:
- Possibility for the verification and correction of the maximum crack value separate for the top and bottom side of slabs

To allow the verification of the maximum crack width value separately for the top and bottom side of concrete slabs, which is particularly important for slabs having contact with ground or water from one side, the $W_{max}$ value has been separated for the top and bottom value. Both values are available on the property list of concrete slabs.

When the $W_{max}$ is set to Auto, then both values have the same calculated value. When the $W_{max}$ is set to Imposed, then both values can be edited and can have different value.

Crack verification is now carried out independently for each side and display separate warning message in case one is exceeded.

**WARNING:** Planar No 1, Case 106: Cracking limit exceeded - bottom face ($W_{max} = 0.250 \text{ mm}$)

**WARNING:** Planar No 1, Case 106: Cracking limit exceeded - top face ($W_{max} = 0.150 \text{ mm}$)

Thanks to the separate limits, the cracking correction, if activated, is acting correspondingly using different limits. Also, the Work ratio for cracking is displayed considering the corresponding limits:
- bottom (negative) $wk / W_{max}$ bottom
- top (positive) $wk / W_{max}$ top
Additional info about areas per unit length when defining a real reinforcement for slabs

Main features & benefits:
- Better control when defining the real reinforcement for slabs

In the Reinforcement definition dialog, available for concrete planar elements, an additional info about reinforcement areas per unit length has been added. It is available on new columns separately for the global definition part and for parts for defining additional areas with using fabrics or bars.

All values on new columns are read only and are automatically updated if diameter or spacing is modified for reinforcement bars or a fabric type is changed.

Possibility for selecting SLS combination types for selected verifications (Eurocode)

Main features & benefits:
- Possibility for selecting SLS combination types for the stress verification
- Possibility for selecting SLS combination types for the maximum crack verification

For the linear concrete elements, a possibility for selecting SLS load combination types for selected verifications (according to Eurocode) has been added.

Stress limits

A new selection has been added to the property list of concrete elements (for Eurocode): SLS type for stress limitation having three available options: SLS Characteristic, SLS Frequent and SLS Quasi-permanent.
New options are used for deciding which SLS load combination types are used for the verification of maximum stresses. As usually, users want stresses limits to be considered only for the SLS types relevant to cracking verification (i.e. SLS Quasi-permanent); by default the stress limitation is activated for SLS Quasi-permanent only. When more than one SLS combination type is selected, stresses are calculated for each selected type and check against the corresponding limit.

Note that when converting old models, all SLS types are activated, to keep the same condition from previous versions of Advance Design, where all SLS combination types were used for the check of stresses.

Wmax check

A new selection has been added to the property list of concrete elements (for Eurocode): SLS type for Wmax having two available selections: SLS Quasi-permanent (default) and SLS Frequent.

A new property is used for the verification of the maximum value of the crack width - by default the SLS Quasi-permanent load combinations are used during this check. Depending on the National Annex and structure type, using SLS Frequent load combinations could be recommended. Now the user can decide which SLS load combination type he wants for the Wmax check.

Report for concrete – New column on the Concrete deflection table

Main features & benefits:
- More detailed documentation

A new column has been added to the report table with deflections of reinforcement linear elements with the cracked section inertia coefficient (I_interpolated/Ic).

<table>
<thead>
<tr>
<th>Element No</th>
<th>Name</th>
<th>Interpolate d/lc</th>
<th>Wtotal (cm)</th>
<th>Wcracked (cm)</th>
<th>Wnon-crack ed (cm)</th>
<th>Wadm (1/xxxx)</th>
<th>Wadm (cm)</th>
<th>Ratio (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
<td>Linear</td>
<td>0.54</td>
<td>-0.62</td>
<td>-0.85</td>
<td>-0.26</td>
<td>1250</td>
<td>-1.60</td>
<td>39</td>
</tr>
<tr>
<td>6</td>
<td>Linear</td>
<td>0.73</td>
<td>-3.44</td>
<td>-3.70</td>
<td>-1.91</td>
<td>1250</td>
<td>-2.08</td>
<td>123</td>
</tr>
<tr>
<td>7</td>
<td>Linear</td>
<td>1.21</td>
<td>0.13</td>
<td>0.43</td>
<td>0.13</td>
<td>1250</td>
<td>1.60</td>
<td>8</td>
</tr>
</tbody>
</table>

Update of the design ribbon

Main features & benefits:
- Easier access to new options in design modules
- Possibility to generate diagrams for wall reinforcement mesh

For Advance design 2021, the Design ribbon has been updated by adding new commands and changes on the Results panel.

Two new commands have been added to the Calculation group: Constructive Dispositions and Verify.
Both are available for concrete elements and are used, respectively, to generate the minimum reinforcement without calculation and to verify an existing (e.g. manually modified) reinforcement. (More details about these functions can be found further in this document).

The **Reinforcement** command has been extended to enable the selection of dialogs for modifying the generated reinforcement. Availability of individual commands depends on the type of element being currently analyzed.

In addition, a new **Fabrics** command has been added to the same section, which is available for reinforced concrete walls and is used to display the fabrics schemas with the cutting info. Schemas are presented in a separate window and can be saved as images to graphic files.
Main improvements to the Concrete design modules

In the latest version of Advance Design a large number of novelties and improvements have been introduced to the Concrete design modules. Detailed information about the changes can be found in a separate document related only to the novelties in Advance BIM Designers 2021. Below you will find only brief information about selected main changes.

Main features & benefits:
- Easier data entry thanks to a new format of dialog windows
- Easy and full control over the content of the drawing
- Easy verification of existing reinforcement
- Quick generation of reinforcement without calculations
- Possibility of choosing the method of placing transversal bars for uneven bar distribution on beams
- New method of analysis for multilayer soils for foundations
- Automatic generation of reinforcement schemas on drawings for walls

New format of dialog windows

The dialogs for defining the geometry, for defining the design assumptions and for editing the reinforcement have been completely modified. All windows have a uniform appearance, size, layout and data input. Thanks to the menu list on the left side of dialogs and its tree structure, the user can quickly add/remove or modify certain components (for example openings). It is especially useful on windows for reinforcement editing, where modifying one of the features (e.g. bar diameter) for the parent position in a tree changes the value also in all elements in a given tree branch.

Interactive drawings

The mechanism of creating reinforcement drawings has been completely modified. Thanks to the new layout, and especially a tree with all drawing components, the user has control over the composition of all drawing elements.
The biggest changes concern the drawing area, which is now interactive. This means that you can select and graphically modify drawing elements. Views can be moved freely within the sheet and easily rescaled. It is also possible to move or delete most of the components of the view, such as dimensional lines, view descriptions, level symbols, bending details and more. All this makes it very quick and easy to adapt the drawing to your needs.

Verification of existing reinforcement

In order to perform the verification of the element with modified / manually created reinforcement the Verify command has been added. After starting it performs all verifications that are carried out during calculations, considering all settings and set loads as well as modeled reinforcement. After verification, all diagrams, drawings and reports are available. If any conditions are not met, appropriate warnings appear on the Errors and warnings tab.
What's New in GRAITEC Advance Design 2021

Generation of the reinforcement for minimal constructive dispositions

In order to enable quick generation of the full 3d reinforcement cage without full design calculations but relying only on minimal constructive dispositions, the Constructive Dispositions command has been added. After its use, a full 3d reinforcement cage is generated in the element which is compliant with the current program reinforcement settings and meets all minimum requirements.

Transversal reinforcement spacing compensations for beams

The latest version of the beam module offers the possibility to choose the method of an automatic transversal bars spacing compensation. It allows choosing the method of placing transversal bars for uneven bar distribution.

New methods of analysis for multilayer soils for footings

The method of determining the load-bearing capacity of the foundation with using average parameters of multilayer soil layers has been extended and is available for all locations/design standards. Now there are two main ways for the homogenization of layered subsoil by determining the mean soil parameters:

- according to DIN 4017, which has been expanded with new options;
- by averaging soil parameter to a given depth, using various types of averaging.

Both methods are universal and can be used to verify the resistance according to all standards available in the application.
Bending details on drawings for walls

With the latest version, the possibility of generating rebar schemes on wall reinforcement drawings was introduced. As with the other modules, there are several settings that allow you to choose how to measure the length of bar sections and how to describe the schemes. Bending details can be generated on drawings of single bearing walls or shear walls.
What's New in GRAITEC Advance Design 2021

New Options & Improvements - Miscellaneous

This chapter contains a list of the main new options and improvements from other / miscellaneous domains.

New start page

Main features & benefits:

- New Start page with direct access to CREATE & LEARN information
- Easy opening of recent files
- Direct access to training materials and online information

When starting Advance Design, a new Start page is displayed.

On the left side of the start window there is a part that includes basic tools such as creating a new or opening an existing project, localization settings and links to social media.

The remaining part of the start window can be displayed in two modes - Create or Learn. Switching between modes is done by using the switch in the lower part of the window.

In the Create (default) mode, users can see a list of recent files, with a preview, and a list of sample projects.

In the Learn mode online materials are presented in three groups:

- Learning resources - videos available on dedicated learning resources;
- Graitec Information - links to the content available on the Graitec Advantage website:
  - News - the latest news from the www.graitec.com website;
  - Documentation - the latest technical documents available on the Graitec Advantage website;
  - Knowledge Base - the latest FAQs published on the Graitec Advantage website.
- Social World - displays links to our social channels;
- Graitec Social HUB - The latest posts from Graitec Social resources.
Update of the ASTM steel material database (US/CAN)

Main features & benefits:
- New types of steel materials for the North American market

With the upgrade of the steel design standard to AISC 360-16, new materials were added to the list of available steel types in the ASTM library.

New materials are according to ASTM reference standards:

- ASTM A1065/A1065M — new HSS standard
  - ASTM A1065-GR 50
  - ASTM A1065-GR 50W
- ASTM A1066/A1066M — new plate standard
  - ASTM A1066-GR 50
  - ASTM A1066-GR 60
  - ASTM A1066-GR 65
  - ASTM A1066-GR 70
  - ASTM A1066-GR 80
- ASTM A1085/A1085M — new HSS standard
  - ASTM A1085
What's New in GRAITEC Advance Design 2021

Update of the AISC section database (US/CAN)

Main features & benefits:
- Update of the AISC section database to the latest version
- New HSS steel sections according to ASTM A1085

The Graitec steel sections database has been updated by adding new profiles to AISC libraries.

Three new cross-section libraries with hollow structural sections according to ASTM A1085 have been added:
- AISC A1085 HSS Pipe
- AISC A1085 HSS Rectangular
- AISC A1085 HSS Square

The existing AISC section libraries have been updated with new profiles according to AISC Shapes database v15.0.

<table>
<thead>
<tr>
<th>Library</th>
<th>New Sections</th>
</tr>
</thead>
<tbody>
<tr>
<td>AISC HP</td>
<td>HP12X89</td>
</tr>
<tr>
<td>AISC WT</td>
<td>W120X327.5, W18X462.5, W18X426.5, W18X401, W18X361.5, W10.5X137.5, W10.5X124, W10.5X111.5, W7X436.5, W7X404</td>
</tr>
<tr>
<td>AISC HSS square</td>
<td>HSS24X12X3/4, HSS24X12X5/8, HSS24X12X1/2, HSS22X22X7/8, HSS22X22X3/4, HSS20X20X7/8, HSS20X20X3/4, HSS20X20X5/8, HSS20X20X1/2, HSS20X12X3/4, HSS18X18X7/8, HSS18X18X3/4, HSS18X18X5/8, HSS18X18X1/2, HSS16X16X7/8, HSS16X16X3/4, HSS14X14X7/8, HSS14X14X3/4, HSS12X12X3/4, HSS10X10X3/4</td>
</tr>
<tr>
<td>AISC HSS rectangular</td>
<td>HSS24X12X3/4, HSS24X12X5/8, HSS24X12X1/2, HSS22X22X7/8, HSS22X22X3/4, HSS20X20X7/8, HSS20X20X3/4, HSS20X20X5/8, HSS20X20X1/2, HSS20X12X3/4, HSS18X18X7/8, HSS18X18X3/4, HSS18X18X5/8, HSS18X18X1/2, HSS16X16X7/8, HSS16X16X3/4, HSS14X14X7/8, HSS14X14X3/4, HSS12X12X3/4, HSS10X10X3/4</td>
</tr>
<tr>
<td>AISC Pipe</td>
<td>Pipe26STD, Pipe24STD, Pipe20STD, Pipe18STD, Pipe16STD, Pipe14STD, Pipe26XS, Pipe24XS, Pipe20XS, Pipe18XS, Pipe16XS, Pipe14XS, Pipe12XS, Pipe10XS</td>
</tr>
</tbody>
</table>

In addition, the existing libraries of compound angles according to AISC (AISC Double Angled) have been corrected to be displayed on the list of available sections.
Improvement of icons for enabling the snap mode and tooltips

Main features & benefits:
- More intuitive usage thanks to clear icons

The buttons for enabling snapping mode and for enabling displaying tooltips are now different for on and off mode. This small improvement will make it easier to distinguish the current status of the options.

Faster moving between cells on the Load Combination dialog

Main features & benefits:
- Faster edition of combination tables

A slight improvement has been made to the table in the Combinations definition window - now you can move between table cells using the TAB key and the Up, Down, Left, Light arrow keys when editing table values.

Additional formats for imperial units (US/CAN)

Main features & benefits:
- Separation of length units into single (foot, inch, yard) and compound (foot-inch)
- Displaying lengths with fraction part

The Imperial Unit System in Advance Design has been extended to perform well on the North American market. The changes concern the separation of units into single and compound, the addition of yards and displaying lengths with fraction part.

The units for imperial lengths can be displayed and formatted separately for single and compound. For single units, feet, inches and yards (new) are available.

```
16.40'    196.85'    5.47 Yd
```

The compound unit (foot-inch) can be displayed with decimal precision or with fractional precision.

```
16'-4.85"    16'-4 7/8"
```

For fractional units it is possible to set precision as: 1/2, 1/4, 1/8, 1/16, 1/32, 1/64 1/128 and 1/256.
New About window with the network license status

Main features & benefits:
- New, standardized view and content of the About dialog
- Easier access and improvements in license management

In the 2021 version of the Graitec programs the About window has been modified. One of the changes is a special window for managing network license - it’s opened by the network status button available on the License tab.

With this window the user can assign a network license available on the company’s server(s) but can also check which users are currently using the seats available for a given server license. The dialog presents only licenses and list of users that are related to the current application. This will make it much easier to control network licenses.
Miscellaneous improvements & corrections

Advance Design 2021 brings many small improvements and corrections. Below you can find a brief description of selected corrections:

Selected corrections:

- On the Suggested shapes dialog for steel sections, the Accept all and Reject all buttons are now working also when choosing the by name optimization type.
- For the selected Polish national appendix to the Eurocode 2, the DC.dur.st value (for minimal cover reduction while using stainless steel) is now available on the property list of concrete elements.
- Improvement of design templates for Reinforced Concrete elements with missing properties related to the correction of cracking.
- Title block is no longer overlapping the Color map when printing views with results.
- The definition of the height of the T section of Reinforced Concrete beam, used for the reinforcement calculation when the rib design option is activated, has been restored as the height of the beam (instead the slab + beam).
- When optimizing steel elements, the condition for the fire strength is now also considered.