What's New
# What's New in GRAITEC Advance Design 2020

---

## Table of contents

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Welcome to Advance Design 2020</strong></td>
<td>5</td>
</tr>
<tr>
<td><strong>New Options &amp; Improvements - Calculations</strong></td>
<td>6</td>
</tr>
<tr>
<td>Modal Analysis with the Load-dependent RITZ vectors</td>
<td>6</td>
</tr>
<tr>
<td>Ground motion analysis</td>
<td>8</td>
</tr>
<tr>
<td><strong>New Options &amp; Improvements - Loads</strong></td>
<td>12</td>
</tr>
<tr>
<td>Improved load visualization</td>
<td>12</td>
</tr>
<tr>
<td>Wind on open structures according to ASCE 7-10</td>
<td>13</td>
</tr>
<tr>
<td>Choice of method</td>
<td>13</td>
</tr>
<tr>
<td>Choice of elements to be loaded</td>
<td>14</td>
</tr>
<tr>
<td>Load generation</td>
<td>14</td>
</tr>
<tr>
<td>Improvements to the Wind generator (Eurocode), according to the new CNC2M recommendations</td>
<td>15</td>
</tr>
<tr>
<td>Improvements to defining Load areas</td>
<td>23</td>
</tr>
<tr>
<td><strong>New Options &amp; Improvements - Results</strong></td>
<td>26</td>
</tr>
<tr>
<td>Deformation with full body of elements</td>
<td>26</td>
</tr>
<tr>
<td>Improvements to the Results ribbon tab</td>
<td>27</td>
</tr>
<tr>
<td>Improvements in displaying support reactions</td>
<td>28</td>
</tr>
<tr>
<td>New axes convention for Resultant forces on the individual wall</td>
<td>30</td>
</tr>
<tr>
<td>Improvements to color scales</td>
<td>30</td>
</tr>
<tr>
<td>New parameters for easy verification of design status</td>
<td>32</td>
</tr>
<tr>
<td>Possibility for selection according new criteria: Design status and Work ratio</td>
<td>33</td>
</tr>
<tr>
<td>Additional information on deflection for timber elements</td>
<td>35</td>
</tr>
<tr>
<td>Animation for time history analysis</td>
<td>35</td>
</tr>
<tr>
<td><strong>New Options &amp; Improvements – Steel Design</strong></td>
<td>37</td>
</tr>
<tr>
<td>Improvements in intermediate restraints’ definitions (for lateral-torsional buckling)</td>
<td>37</td>
</tr>
<tr>
<td>Advanced stability on Graitec profiles</td>
<td>44</td>
</tr>
<tr>
<td>Steel optimization: Options to manage / sort sections used for searching</td>
<td>46</td>
</tr>
<tr>
<td>Tension resistance of angle sections with net area (acc. EC3)</td>
<td>48</td>
</tr>
<tr>
<td>New report table - Details about section and classification</td>
<td>52</td>
</tr>
<tr>
<td>Interactive optimization table</td>
<td>54</td>
</tr>
<tr>
<td>New HSS bracing joints</td>
<td>55</td>
</tr>
<tr>
<td>Improvements to BIM Designers Steel Connection module</td>
<td>57</td>
</tr>
<tr>
<td><strong>New Options &amp; Improvements – Reinforced Concrete Design</strong></td>
<td>59</td>
</tr>
<tr>
<td>Improvements for BIM Designers report generation on Advance Design</td>
<td>59</td>
</tr>
<tr>
<td>Export the theoretical reinforcement area for planar elements</td>
<td>60</td>
</tr>
<tr>
<td>BIM Designers Wall on AD</td>
<td>61</td>
</tr>
<tr>
<td>Workflow</td>
<td>62</td>
</tr>
</tbody>
</table>
Structural usage type .................................................................................................................................... 65
Load transfer ................................................................................................................................................. 66
Calculations and results ................................................................................................................................ 67
Main improvements to BIM Designers - Reinforced concrete modules ......................................................... 70

NEW OPTIONS & IMPROVEMENTS - MISCELLANEOUS ............................................................................. 73

Export data from Advance Design to CS-STATIK.......................................................................................... 73
Detailed graphical representation of releases on linear elements ................................................................. 74
Reduction coefficients for stiffness for linear elements ................................................................................. 77
Improvements on Grid axes ............................................................................................................................. 78
Update to the timber material database ......................................................................................................... 80
Changes in naming of selected options ......................................................................................................... 81
Easy display of element length ......................................................................................................................... 82
Possibility to display footing dimensions on a model ...................................................................................... 83
Possibility for selecting Load areas by their types ......................................................................................... 84
New options to the unit’s settings ................................................................................................................... 85
The drift-check for seismic SLS load combinations according to the Romanian P100-2013 code................. 86

MISCELLANEOUS IMPROVEMENTS & CORRECTIONS .............................................................................. 87
Welcome to Advance Design 2020

GRAITEC is very pleased to present the latest version of our leading structural analysis software - Advance Design 2020, part of the Graitec Advance suite.

GRAITEC has continuously strived to provide first-rate advancements for innovative software solutions to our valued customers, and the recent launch of our new and upgraded product range for 2020 is no exception, proving we are still top of the game in terms of providing state-of-the-art AEC and Building Design software solutions worldwide.

Version 2020 of Advance Design is enhanced with plentiful of new functionalities, bringing high benefits for the end user, and is articulated around a few main areas of interest:

- **New FEM calculation capabilities** for modal and dynamic temporal analyses;
- **New possibilities for automatic wind generation**, including wind on open structures, according to the American code ASCE and new building geometries according to the Eurocode 1-4 (CNC2M provisions);
- **New possibilities for steel design**, including improvements in the definition of restraints for the lateral-torsional buckling, possibility of analyzing a new large range of profiles for advanced stability analysis, improvements to steel optimization, possibility for modeling new bracing connection types, etc.;
- **New capabilities for reinforced concrete design**, including design and detailing of reinforced concrete walls (following the introduction of our brand new BIM Designers Wall module);
- **New possibilities for cooperation with other software**, including direct link to CS-STATIK, export of results for walls and slabs to Revit®.

The 2020 version of Advance Design also comes with a big number of improvements and adjustments, following the feedback received from thousands of users, among which we note:

- Improvements in presenting support reactions;
- Possibility for easy verification and selection of elements according to their design status;
- Possibility for better control over stiffness of linear elements;
- Detailed method for displaying releases;
- Full control over naming of grid axes;
- …and many others.

Advance Design 2020 is the invaluable tool for all your projects!
What's New in GRAITEC Advance Design 2020

New options & Improvements - Calculations

The below subchapters deploy a list of the main new options and improvements related to the general calculation possibilities.

Modal Analysis with the Load-dependent RITZ vectors

Main features & benefits:
- New solver for Seismic and Spectral analysis
- Automatic determination of required number of modes
- Reducing the time spent for the seismic analysis

Advance Design 2020 introduces a new solver method available for modal analysis: Load-dependent RITZ vectors.

Ritz vector analysis is better adapted for Seismic and Spectral analysis, since it manages to compute only the vibration modes that are significantly excited by the loading (discarding modes with low participation factor).

In addition, Ritz vectors allow for automatic determination of number of modes, according to a user target participation ratio. During the analysis, the solver will keep searching for modes, until the required modal mass is reached.

Advantages of the Ritz vectors method over Eigen vectors:
- Calculation runs faster (less modes required);
- Same level of accuracy;
- Ignores the small / localized modes (where only a small part of the building is oscillating);
- Only seeks modes excited by a loading;
- Guarantees that the modes will produce displacement in a given direction.

The Method can be changed from the Property list of modal analysis, under a new ‘Type’ parameter. There are two selections available:
- Eigen Vectors (default);
- Load Dependent Ritz Vectors.

Advance Design 2020 also introduces a new option related to the model mass matrix.
This option is available for both Eigen and Ritz analyses.

Users can choose either type of mass matrix:

- **Lumped** (default): masses are assigned to nodes. The matrix contains only diagonal, translational components (no rotational ones)

  \[
  \begin{bmatrix}
  m_1 & 0 & \cdots & 0 \\
  0 & m_2 & \cdots & 0 \\
  \vdots & \vdots & \ddots & \vdots \\
  0 & 0 & \cdots & m_n
  \end{bmatrix}
  \]

- **Consistent**: masses are distributed along the elements using the shape functions. This allows for a better distribution of masses along the elements. The matrix contains off-diagonal translational components, as well as rotations

  \[
  \begin{bmatrix}
  m_1 & m_{12} & \cdots & m_{1n} \\
  m_{21} & m_2 & \cdots & m_{2n} \\
  \vdots & \vdots & \ddots & \vdots \\
  m_{n1} & m_{n2} & \cdots & m_n
  \end{bmatrix}
  \]

The choice of the type of mass matrix affects the calculation time and accuracy: selecting a matrix as Lumped, results in shorter calculation time at the expense of accuracy.

In the case of choosing the new Ritz vector method, stopping of the analysis can be caused by two newly introduced types of conditions (**Target** property):

- Number of modes (default) -> allows for imposing the number of modes for Modal Analysis;
- Participation ratio -> allows for defining a minimum acceptable Dynamic Participation Ratio (from all directions). This value is by default set to 95%.

Thus, the analysis is stopped:

- When the desired number of modes is achieved:

<table>
<thead>
<tr>
<th>Modes</th>
</tr>
</thead>
<tbody>
<tr>
<td>Target</td>
</tr>
<tr>
<td>Number</td>
</tr>
<tr>
<td>Participation ratio</td>
</tr>
</tbody>
</table>
When the desired participation ratio is reached:

![Modes Table](image)

Below is a comparison of calculation time for a sample building, using both methods of analysis:

<table>
<thead>
<tr>
<th>Method</th>
<th>Eigen Vectors</th>
<th>LDRV</th>
</tr>
</thead>
<tbody>
<tr>
<td>Modal mass participation</td>
<td>90%</td>
<td>98%</td>
</tr>
<tr>
<td>Number of modes</td>
<td>50 modes</td>
<td>28 modes</td>
</tr>
<tr>
<td>Calculation time</td>
<td>10 min</td>
<td>5 min</td>
</tr>
</tbody>
</table>

It should also be noted that targeting a participation ratio ensures that the required modal mass will be excited at the end of the very first calculation.

As a result, the user no longer needs to iterate the number of modes, thus performing repeated analyses with an increased number of modes every time the percentage of oscillating mass is too low.

This significantly reduces the time spent for the seismic analysis.

### Ground motion analysis

- Predicting building behavior on a given location and for various ground motion intensities
- Study the response to a user-defined earthquake event
- Large availability of natural earthquake records

Advance Design 2020 introduces a new tool to assess the seismic performance of buildings, by inputting ground motion records and performing a Time-history analysis.

Although this new addition to the software might demand a higher computational power, the advances in software and numerical models have made this approach a common practice among design professionals.

After creating a Time-history analysis, users can select an Accelerogram as solicitation type.
Ground motions can subsequently be imported or even manually defined.

Advance Design 2020 provides a few samples (.acc files), consisting of acceleration history samples recorded from actual events.

Moreover, research center websites offer a wide array of natural earthquake records from all over the world.
The .acc files mentioned above can be opened with any text editor.

They consist of two sets of values:

- Time, measured in seconds – 1st column
- Acceleration, measured in m/s² – 2nd column
Please be aware that some of the .acc files are scaled, while some others are not (are normalized to $g^1$), which requires for the users to appropriately set the Scale factor parameter:

In the post-processing phase, the users can analyze the building’s response to the input ground motion, in terms of story drift, shear force, deformation etc.

1 $G$ is the acceleration of gravity 9.8 (m/s$^2$) or the strength of the gravitational field (N/kg) (which it turns out is equivalent) – acc. https://earthquake.usgs.gov.
New options & Improvements - Loads

The below subchapters deploy a list of the main new options and improvements related to Loads definition.

**Improved load visualization**

**Main features & benefits:**
- Better load visualization within the model
- Easier load selection

Linear and surface loads have received new display possibilities:
- displaying with filling;
- displaying with arrows.

These settings greatly improve load presentation, facilitating both the work itself and the creation of better-looking views, for documentation purposes. In addition, they facilitate their selection in a graphical way, because it is enough to point at any internal arrow or any filling point.

These new settings are available in the Display Settings window (ALT+X), separately for linear and planar loads. The new parameters can be set independently of each other.

The “Arrow density” slider can be used, in order to change the arrow density from the default setting.
Wind on open structures according to ASCE 7-10

Main features & benefits:
- Automatic generation of wind loads on lattice open structures according to the American code
- High flexibility in parameters handling

Starting with Advance Design 2020, it is possible to automatically generate wind loads on lattice open structures, according to the corresponding provisions from the American code ASCE 7-10 (Minimum Design Loads for Buildings and Other Structures, ASCE/SEI 7-10).

This method can be used for any kind of open structures, such as pipe racks, industrial / petrochemical structures or trusses. It can also be used for lattice masts, such as transmission towers or lattice telecom towers.

Choice of method

The selection of a new wind generation method is available for Wind load cases family (according to ASCE 7-10) by using the list under the “Wind Design Method” parameter. Two possible methods are displayed on this list:

- **Envelope Procedure for Low-Rise Buildings**
  - this is an already existing Advance Design method, for automatic wind generation for low-rise buildings, in accordance with Chapter 28 of ASCE 7-10.

- **Directional Procedure for Open structures (Lattice Framework & Trussed Towers)**
  - this is a **newly implemented method** for automatic wind generation for open structures, in accordance with Chapter 29 (Wind loads on other structures and buildings appurtenances—MWFRS) of ASCE 7-10.
Most of the parameters available in the Wind load cases family are the same for both methods, except for the new parameter: “Gust Effect Factor (G)”, which is by default equal to 0.85 (acc. ASC 7-10, 26.9.1).

Choice of elements to be loaded

Unlike a typical wind load generation for buildings, where loads are generated on the load areas, this method generates loads directly on the linear elements covered by load areas. In order to control which elements will be loaded, the appropriate parameters must be defined: load areas and linear elements.

First, the load areas that transfer the wind load to the linear elements must be set to “Lattice structure or scaffolding”, in the Type field:

![Lattice structure or scaffolding](image)

Similarly, the linear elements that are directly loaded should have the option “Lattice structure or scaffolding” enabled. This option is available on the Property list of linear elements, in the Load Area Transfer section:

![Load Area Transfer](image)

The same group of parameters provide the possibility to impose a user value for the Force factor (Cf). In addition, this value can be modified separately for each wind direction. This allows full freedom in determining the parameters, depending on the types of structure and loaded elements.

Load generation

Loads are defined automatically by activating the “Automatic generation” command, available on the Wind Load Case Family context menu (in the Project Browser window):

![Automatic generation](image)

Linear loads are defined directly on the linear elements on 8 newly created load cases: 4 for wind, in main directions (X+, X-, Y+, Y-) and 4 for wind, in diagonal directions (X+Y+, X-Y+, X+Y-, X-Y-).
Loads are calculated separately for each linear element, according to its vertical location (z coordinate of the middle point of the element) and to the dimensions of its section, as projected to the direction of the acting wind.

Improvements to the Wind generator (Eurocode), according to the new CNC2M recommendations

Main features & benefits:
- Automatic wind force generation on complex building shapes not covered by Eurocode 1
- More favorable loading schemes on awnings and canopies
- Automatic generation remains fast and accurate regardless of the geometry complexity

The EN1991-1-4 norm and its national appendices define the rules for determining wind action on buildings.

It should however be noted that this norm only covers basic building shapes, leaving aside any information regarding more complex geometries, such as buildings with setbacks, although these are very common in European – and worldwide - building practice.
Wind pressure on awnings is not covered in EN1991-1-4 either, with chapter §7.2.1 addressing protruding roofs.

Unfortunately, considering an awning to a protruding roof has proven to be an unnecessary conservative practice.

EN1991-1-4 has been widely criticized for the way it applies wind pressure on canopies, the use of a single concentrated load being considered by many as highly unrealistic as well as pessimistic.

In order to fill this existing gap left in Eurocode 1, the CNC2M College of Experts established a set of rules to complement EN1991-1-4.

These rules from the CNC2M committee were published in July 2017:

CNC2M rules from July 2017
The wind generator of Advance Design 2020 has been updated accordingly.

These new wind rules can easily be applied when the “**CNC2M wind rules**” option is checked in the Properties window:

- Buildings with **horizontal setbacks** can now be covered by the climatic generator.

Wind effects on a setback are complex, especially when the setback is upwind. Therefore, taking only the main 4 wind direction (X+, X-, Y+, Y-) into consideration is not enough.
Some wind directions will require that two outcomes be taken into consideration:

- With a (0°; 45°) incidence about the main direction

- With a (-45°; 0°) incidence about the main direction:

In Advance Design 2020, case names now mention whether the Y+ wind is leaning towards X+ or X-.
Indeed, for the Y+ wind leaning towards X+, the two faces highlighted in the picture below are considered as upwind (zone D, with positive $C_{pe}$) even though one side is clearly parallel to the wind.

On the other hand, for the Y+ wind leaning towards X-, the C wind zone (with negative $C_{pe}$) is extended on the face perpendicular to the wind.

This is due to the longer building block acting as a protection, thus avoiding any positive external pressure inside the setback.
One may notice that the previous images only focus on vertical walls, yet the same concepts are obviously applied on roofs with their respective F, G and H zones.

In Advance Design 2020, wind forces for all these possible outcomes will be instantly generated by simply clicking the “Automatic generation” option from the Project Browser context menu.
Wind actions on awnings are now covered by the CNC2M wind rules:

No matter the wind direction, awnings are considered as A and B wind zones.

The corresponding pressure coefficients $C_{p,\text{net}}$ depend on the position (altitude) of the awning.

<table>
<thead>
<tr>
<th>Rapport des hauteurs $h_1/h$</th>
<th>Zone A</th>
<th></th>
<th>Zone B</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Charge descendante</td>
<td>Charge ascendante $h_2/d_1 \leq 1.0$</td>
<td>Charge descendante</td>
</tr>
<tr>
<td>0.1</td>
<td>1.1</td>
<td>-0.9</td>
<td>0.9</td>
</tr>
<tr>
<td>0.2</td>
<td>0.8</td>
<td>-0.9</td>
<td>1.4</td>
</tr>
<tr>
<td>0.3</td>
<td>0.7</td>
<td>-0.9</td>
<td>1.4</td>
</tr>
<tr>
<td>0.4</td>
<td>0.7</td>
<td>-1.0</td>
<td>1.5</td>
</tr>
<tr>
<td>0.5</td>
<td>0.7</td>
<td>-1.0</td>
<td>1.5</td>
</tr>
<tr>
<td>0.6</td>
<td>0.7</td>
<td>-1.1</td>
<td>1.6</td>
</tr>
<tr>
<td>0.7</td>
<td>0.7</td>
<td>-1.2</td>
<td>1.7</td>
</tr>
<tr>
<td>0.8</td>
<td>0.7</td>
<td>-1.4</td>
<td>1.9</td>
</tr>
<tr>
<td>0.9</td>
<td>0.7</td>
<td>-1.7</td>
<td>2.2</td>
</tr>
</tbody>
</table>

$C_{p,\text{net}}$ on awnings
Wind loads on mono-pitch canopies are more realistic in the CNC2M wind rules than in Eurocode 1.

The new CNC2M wind rules consider wind forces on mono-pitch canopies as a combination of a thin, uniform load (governed by a C1 pressure coefficient) and a variable load (governed by C2 and C3 pressure coefficients).

As a reminder, EN1991-1-4 is particularly conservative in this matter and only recommends a single concentrated force.
Improvements to defining Load areas

Main features & benefits:
- New, more effective scenarios for defining Load areas on selection
- Creation of Load Areas during extrusions

Advance Design 2020 introduces a set of improvements to the way Load Areas are defined, including:

1. Possibility to use a line object for selection;
2. Possibility to define selection after calling the command;
3. Possibility to define Load areas in repetitive mode (in a chain);
4. Possibility to use more than 2 objects on selection (envelope mode);
5. Creation by Extrusion.

Improvements 1 to 4 focus on the definition of **Load Areas on Selection**, which, in previous versions, allowed for only 2 linear members being selected.

With version 2020, the “**Load area on Selection**” command allows users to select previously defined lines / polylines / arcs. The selection can also be mixed, (users can select a linear element and an arc for instance):
When “Load area on Selection” command is applied without an existing selection, users can choose the first and the second element afterwards.

<table>
<thead>
<tr>
<th>Command Line</th>
<th>Command Line</th>
</tr>
</thead>
<tbody>
<tr>
<td>&gt; Select first linear element &gt;</td>
<td>&gt; Select second linear element &gt;</td>
</tr>
</tbody>
</table>

**Note:** These commands work in automatic repetition mode, therefore, after selecting a second object and creating a Load area, users can immediately point to another pair of elements.

The next improvement to the Load area on Selection command is the possibility to select multiple elements. The command supports the selection of members and/or lines / polylines that are defined on the same plane. The load area is created based on an envelope of end points of all selected elements.

This method allows for a very quick load area definition, simply by graphically selecting objects in 3D view (as is the case with whole walls).

There is also a new method of defining Load areas when using the Extrusion of an Element command (available on the Home ribbon, on the CAD Functions panel):
Regarding this, a new option is available in the **Extrude** dialog:

![Extrude dialog with options](image1)

This allows for a highly effective definition of Load areas:

![Load area example](image2)
New options & Improvements - Results

The subchapters below detail the main new options and improvements related to the displaying of calculation results.

Deformation with full body of elements

Main features & benefits:
- More readable and better-looking graphical results on the deformed model
- Easier verification of the structure

Version 2020 allows results in the case of displacements to be displayed as colored maps, on the full shape of the elements.

The settings can be changed in the Results Settings window (<ALT+Z>), Options tab.

This type of display not only improves the readability of results, but also helps to better understand the behavior of the elements within the structure.
Improvements to the Results ribbon tab

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

In order to increase the ease of access to typical graphical results and to facilitate their configuration, we have brought a set of improvements to the Results ribbon tab.

The first change concerns a new list from which users can select the results for the supports, directly from the above-mentioned ribbon tab, on the FEM Results panel:

This makes it very easy to activate the display of support reactions.

Another significant change is the addition of new icons / buttons in front of the results lists. They have two purposes:
- to make it easier for users to identify these lists (thanks to the icons corresponding to elements category)
- to make it easier for users to modify related settings (by directly opening the Result Settings dialog)
By pressing a selected button, the Result Settings dialog opens on the appropriate tab, according to the selected parameters. For example, by pressing the Results for Linear Elements button from the Steel Design Results panel, the configuration window will open directly on the Steel tab, displaying at the same time, the selected type of results.

Improvements in displaying support reactions

Main features & benefits:
- More support reactions visible at the same time
- Easier identification of support reaction values

Two improvements have been made to the graphical representation of support reactions:
- the possibility to display multiple support reactions at once;
- the possibility to display prefixes with names of forces.
Users can now select more than one reaction to be displayed in the Results Settings window for Point Supports. Thus, up to 3 forces or up to 3 moments can be selected at once:

In the Results menu, Options tab, users can select whether the values of support reactions visible in the view should be preceded by appropriate identifiers (i.e., prefixes)
What's New in GRAITEC Advance Design 2020

New axes convention for Resultant forces on the individual wall

Main features & benefits:
- Easier verification and comparison of results for walls
- More consistent results for walls

In Advance Design 2020, a new (simplified) internal axes convention is implemented for calculating resultant forces on walls.

The orientation of section cuts created automatically for the values’ numerical integration is now related to the new internal local axes convention – the \( x \) axis is always vertical (along the wall height), the \( z \) axis is perpendicular to the wall with the same orientation (sign) as the local \( z \) axis of the wall in question, and the \( y \) axis is obtained using the right-hand rule.

Due to the sign of resultant forces (and, consequently, also their graphical representation), these forces do not depend on the local axis definition of walls anymore.

Improvements to color scales

Main features & benefits:
- Better control over the ranges of presented results
- Easier identification of work ratio ranges for the designed element

The color scale used in the graphical presentation of the results has been improved by three novelties:
- Possibility to reset the scale;
- New color scale type;
- Possibility to set any scale limits.

Access to these improvements is possible from the Color Map Configuration window.

In the lower part of this window, there is a new Reset button, which resets the current scale setting to the default presetting.
In the upper part of the Color Map Configuration window on the other hand, a new button for selecting the desired color scale type is now available:

This new scale type has two main capabilities:

- It allows users to define a scale beyond the range of current results
- It allows an easy adjustment for Work Ratio results (with the purpose of distinguishing values above and below 100%).

The first feature allows users to set new Upper and Lower limits for any value.

In the example shown in the picture below, although the range of available values is from -10.33 to 16.29, the limits have been set from -20 to +20.
This makes it possible, for instance, to set a typical scale of results in percentages (range 0-100%), as such a scale range is typical for Work ratio results. For this reason, a special template was done to facilitate its definition. This template is activated after selecting a new scale type, by pressing the **Reset** button.

This scale template has the lower limit = 0% and the top limit = 100%, it also offers the possibility to select intermediate values: 20%, 40%, 60%, 80% and 95%. Above the 100%, there is only one color available (one range 100% to +INF). All ranges can obviously be freely adjusted. Such a scale makes it easier for users to visually verify which elements are outside the required Working Ratio range.

### New parameters for easy verification of design status.

<table>
<thead>
<tr>
<th>Main features &amp; benefits:</th>
</tr>
</thead>
<tbody>
<tr>
<td>Easy verification of design calculation status</td>
</tr>
</tbody>
</table>

New parameters were added to the list of elements properties: **Design Results**, **BIM Designers Results** and **Work Ratio**. The main purpose of these parameters is to simplify and speed up checking the status of the design calculations.

The new parameters are visible in the **Properties** window, in the **Design** section (Steel / Concrete / Timber Design). However, a different range of these parameters is available depending on the type of element.

Description of new parameters:

- **Design Results**
  - Shows the status of design results ("Available" / "Not available");
  - Is available for steel/ timber/ concrete linear elements and concrete planar elements.
What's New in GRAITEC Advance Design 2020

- **BIM Designers Results**
  - Shows the status of design results from the BIM Designers modules (“Available” / “Not available”);
  - Is available for concrete linear and planar elements as well as for steel connections.

- **Work Ratio**
  - Shows the maximum work ratio value (in percentage) for a given element from all performed design verifications;
  - Is available for steel and timber linear elements.

---

### Possibility for selection according new criteria: Design status and Work ratio

<table>
<thead>
<tr>
<th>Main features &amp; benefits:</th>
</tr>
</thead>
<tbody>
<tr>
<td>- Quick selection of elements, according to their design status</td>
</tr>
<tr>
<td>- Quick selection of elements for the selected range of Work ratios</td>
</tr>
</tbody>
</table>

Two new tabs have been added to the **Elements Selection** window (Alt + S), in order to increase the possibility of selecting elements according to different criteria which were not taken into account until now: **Design Parameters** tab and **Work Ratio** tab. This makes it much easier to manage and check the status of structural components, due to their dimensional status.

- **Design Parameters**

  The **Design Parameters** tab allows for selected elements, using three design parameters:

  - **To calculate**
    - for selecting elements according to the activation of the “To calculate” parameter;
    - there are two possible statuses: “Enabled” and “Disabled”;
    - this criterion is available separately for steel / timber / concrete linear elements, as well as for concrete planar elements.

  - **Design Results**
    - for selecting elements according to the availability of design results;
    - there are two possible statuses: “Available” and “Not available”;
    - this criterion is available separately for steel / timber / concrete linear elements, as well as for concrete planar elements.
What's New in GRAITEC Advance Design 2020

- **BIM Designers Results**
  - for selecting elements according to the availability of design results, if calculations are performed using built-in BIM Designers modules;
  - there are two possible statuses: “Available” and “Not available”;
  - this criterion is available separately for steel connections, as well as for linear and planar concrete elements.

- **Work ratio**

The Work Ratio tab allows for steel / timber linear elements and steel connections to be selected within defined work ratio limits. The selection is possible for those elements for which the design analysis was performed.

There are three possible selection ranges depending on the maximum work ratio (WR) value:

- Over top limit – undersized elements having a WR greater than the top limit;
- Between limits – elements having a WR between top and bottom limits;
- Below bottom limit – oversized elements having a WR lower than the bottom limit.

On the right side of the tab, there is the possibility to define Top and Bottom limits for the maximum Work ratio. Please note that the adjusted limit values are taken into consideration only after pressing the Apply filter button.
Additional information on deflection for timber elements

Main features & benefits:
- Better control over the deflection results of timber elements

In order to make it easier to for users to check the deflection values of timber elements resulting from codes verification, additional results are now provided in displacement units. Such values are visible in both simplified and detailed shape sheets.

In addition, in the element’s detailed shape sheet – Deflections section – formulas for Wfin and Wfin,net have been added:

<table>
<thead>
<tr>
<th>Section</th>
<th>Deflections</th>
<th>Strength (61%)</th>
<th>Stability (66%)</th>
<th>Fire resistance 0</th>
</tr>
</thead>
<tbody>
<tr>
<td>W inf 1</td>
<td>n’113 = -76[Z G]</td>
<td>L / 444 &lt; L / 300</td>
<td>(15.76 mm &lt; 23.33 mm)</td>
<td>68 %</td>
</tr>
<tr>
<td>W inf 2</td>
<td>n’113 = -76[Z G]</td>
<td>L / 422</td>
<td>(16.53 mm)</td>
<td>-</td>
</tr>
<tr>
<td>W creep 1</td>
<td>n’121 = 331 [Z G] = 0.331 [101 COMB]</td>
<td>L / 219</td>
<td>(2.30 mm)</td>
<td>-</td>
</tr>
<tr>
<td>W inf 3</td>
<td>-</td>
<td>L / 353 &lt; L / 125</td>
<td>(19.84 mm &lt; 56.00 mm)</td>
<td>35 %</td>
</tr>
<tr>
<td>W inf 4</td>
<td>-</td>
<td>L / 353 &lt; L / 200</td>
<td>(19.84 mm &lt; 35.00 mm)</td>
<td>57 %</td>
</tr>
</tbody>
</table>

Animation for time history analysis

Main features & benefits:
- Visualization of the building’s behavior during an earthquake

One form of presenting displacement results in Advance Design is the animation. Until now, the animation was possible for dynamic analysis displacements, but only for a selected step (animation was then obtained in the same way as in the case of results from static analysis).

Now it is also possible to generate animations for displacements coming from dynamic time analysis for the full range of its duration. This is especially useful for generating animations for results coming from time analysis where the solicitation is an earthquake accelerogram.

In order to define the time frames for which the animation will show the displacements, it is necessary to set the desired time interval in the Properties of dynamic load cases so that the results are saved, then displayed in the Animation.

Note: When selecting the results, there is no need to select a specific time step, instead “MAX D (s)” should be chosen from the respective drop-down list on the FEM Results panel of the Results ribbon tab.
The animation is generated by selecting the **Animation**... command available on the **Post Processing** panel of the above-mentioned ribbon tab. The received animation will show the movement of the structure within the full range of recorded results.
**New Options & Improvements – Steel Design**

The list below details the main new options and improvements related to Steel design.

### Improvements in intermediate restraints’ definitions (for lateral-torsional buckling)

<table>
<thead>
<tr>
<th>Main features &amp; benefits:</th>
</tr>
</thead>
<tbody>
<tr>
<td>▪ Significant acceleration of definitions and greater control</td>
</tr>
<tr>
<td>▪ Faster definition methods</td>
</tr>
<tr>
<td>▪ Effective visualization</td>
</tr>
</tbody>
</table>

Advance Design 2020 introduces a set of improvements which significantly facilitate the definition of intermediate restraints for steel linear elements, (used for reducing lateral-torsional buckling, according to Eurocode 3).

These improvements focus on the following areas:

- Possibility for displaying symbols of restraints on a model;
- A set of new methods for graphical restraints’ definition on a model;
- Improvements to the “Lateral-torsional Buckling” dialog, which allows new methods of restraints definition, graphical preview, etc.
- Possibility to impose the Critical moment (Mcr)

#### Displaying symbols of restraints on a model

To increase the control on the restraints definition, a new graphical representation of these is now available. The activation of the newly introduced symbols is available on the **Display Settings** dialog (<ALT+X>), by selecting “Restraints” from the drop-down list of symbols for linear elements:
The following symbols are used:

- Pinned restraints on the bottom flange
- Pinned restraints on the top flange
- Fixed restraints on the bottom flange
- Fixed restraints on the top flange
- Continuous restraints on the bottom flange
- Continuous restraints on the top flange

**Graphical definition of restraints**

The definition of restraints is now possible graphically as well. To do this, users need to select one or more linear steel elements, right-click it in order to access the corresponding context menu, then select the desired commands shown here below:

There are several commands available, applicable for different scenarios. The first two commands are used to define pinned or fixed restraints on selected elements respectively. The third command, **Continuous restraints on selection**, is used for quickly defining continuous restraints on a top or bottom flange. The last one, **Remove restraints**, removes all intermediate restraints which were previously defined on all selected linear elements.

There are five methods available for defining pinned and fixed restraints on selections (first two commands):

- **On points**

Restraints are defined at all points indicated by the mouse cursor on the axis of the selected element. Before points are indicated, a prompt is visible on the **Command line**, in order for users to select the flange where restraints will be defined. Pressing one of three key letters (U, L or B) automatically selects the upper, lower or both flanges.
Please note that the above key letters are universal for all language versions.

In the case of a selection of multiple linear elements, restraints are defined only on the elements that were pointed out.

- **On intersecting members**

When using this command, restraints are defined automatically at all points where other linear elements intersect certain considered elements. Thanks to this method, it is possible to quickly define restraints for a large group of elements. For example, for defining restraints on all rafters at once, it is enough to:
What’s New in GRAITEC Advance Design 2020

- Select rafters
- Call the command
- Select all the purlins that intersect the rafters in question

As an effect, restraints will be automatically defined on all rafters, at once:

- By number

If this option is selected, restraints are defined at equal intervals, based on the entered number. It is further possible to select whether the restraints are to be defined on the top, bottom or both flanges.
### By abscissa

In this case, restraints are defined according to inputted values of abscissas along the selected elements.

```
Command Line
Restraints > Please input a list of abscissa along the element(s) (m) > 1.2 2.5
```

![Diagram showing restraints by abscissa](image)

### By ratio

This option allows for restraints to be defined according to imputed values of relative abscissas along the selected elements:

```
Command Line
Restraints > Please input a list of ratio (e.g. 0.25 0.5) * L > 0.33 0.65
```

![Diagram showing restraints by ratio](image)
Improvements to the “Lateral-torsional Buckling” dialog

The dialog that allows defining lateral-torsional buckling parameters, accessed from the list of properties of a linear element, has been modified to make it easier for users to define the location of restraints and also to better visualize them.

On the top-right part of the window, a special viewer is available. It displays the preview of the real geometry of the selected linear elements, with the current location of defined restraints. The same type of symbols as on a 3D model are used. Using the mouse, it is possible to rotate, zoom or pan the element on the preview. Also, the button located on the top-right corner of the viewer resets the view to the starting position.
Above and below the table located on the top-left part of the window, new buttons are now available:

- The **Graphical definition** button temporarily hides the dialog and allows for graphical selection, on the 3D model, of the points where the restraints are located.
- The **Auto detect** button finds and adds to the table below it all the points where other linear elements intersect the element under consideration.
- The **Copy to other flange** button copies the table with the restraints definition to the other (top or bottom) flange tab.
- The **Add**, **Equal spacing**, **Remove**, **Remove all** buttons are used for editing the content of the table above them.

Below the table, there have also been introduced new options for a quick definition of the restraints' location, by using three methods: by number, by abscissas and by ratio. Depending on the selected mode, different data can be entered on the text field right below them:

- **Number** – number of restraints;
- **Abscissas** – list of abscissas along the element; a space or a semicolon could be used as a separator of next values;
- **Ratio** - list of locations using relative length of the element (e.g. 0.25 0.5); a space or a semicolon could be used as a separator of next values.

Adding new restraints to the table can be made by using the **Define** button. The type of added restraints is selected from the drop-down list located to the left of this button.
Imposing the Critical moment

To allow verification of profiles in cases of solutions for which the user already knows the value of the critical moment, there is a new selection available in order to impose this value manually. By default, the value of critical moment is calculated automatically, according to a set of parameters defined in the **Lateral-torsional Buckling** dialog. When the value of the critical moment is imposed by the user, all the other parameters in the dialog are ignored and disabled with regards to editing.

Advanced stability on Graitec profiles

**Main features & benefits:**
- Enables the Advance stability method on any cross-section shape
- Allows the design of geometries not covered by the existing norms

Advance Design 2020 now offers the possibility to use section shapes available in the Graitec Profile library (AstorProfiles) in Steel Design, with the **Advanced Stability** solver.

**Note:** Some sections from Graitec Profile database (mainly panels and steel decks) are not compliant with Advance Stability analysis. If defining an element with one of these sections, an error message will be displayed in the Command Bar.

As a reminder, the Advance stability solver was introduced in the 2019 version of Advance Design.

The solver ensures a thorough analysis of individual members by performing a 2nd order analysis with lateral-torsional buckling taking into account imperfections and warping effects.

- Imperfections are introduced by a scaled eigenmode acting as initial deformation;
- Torsional warping effects are considered as a 7th degree-of-freedom;
- The eccentricity of the loads is considered by choosing the application point of the loads on the cross-section;
- The boundary conditions and lateral restraints are introduced as centric or eccentric nodal and continuous springs along the member;
- The effects of the deformed geometry are considered by performing a second-order analysis.
What's New in GRAITEC Advance Design 2020

By allowing the Advanced stability method on sections from the Graitec Profiles library, Advance Design 2020 enables the verification of cross-section shapes, even mono symmetric or asymmetric shapes, which are widely used in practice but not covered by design rules.

Note: The cross-section class calculation does not work on certain catalogues labeled as “User-defined”.
Steel optimization: Options to manage / sort sections used for searching

Main features & benefits:

- Optimization of profiles, according to new criteria (e.g. according to weight)
- Greater control over the range of profiles to be searched.

Advance Design 2020 introduces several improvements in the search process for optimal steel profiles:

- Possibility to set limits for profiles width and height;
- Possibility of sorting profiles, by using cross-sectional parameters as searching criteria (e.g. height, inertia, etc.);
- Possibility to automatically determine the searching limit (i.e., the number of iterations per section).

Limits for profiles

On the Sort Profiles tab of the Steel Design Calculation Settings dialog, new options for determining geometric limits for searched profiles can be found. Four parameters can be set independently of each other: minimum and maximum profile height and minimum and maximum profile width.

Note: When searching for the optimal cross-section, all profiles not meeting the defined geometric criteria are ignored.

Search type

On the same Sort profiles tab of the Steel Design Calculation Settings dialog, there is a set of new options for selecting the desired searching method. The selection of the method determines the way profiles are sorted during the searching process. One of five methods can be selected:

- By height ➔ profiles are sorted by cross-section height
- By width ➔ profiles are sorted by cross-section width
- By Iy inertia ➔ profiles are sorted by the Iy moment of inertia of the cross-section
- By Iz inertia ➔ profiles are sorted by the Iz moment of inertia of the cross-section
- By weight ➔ profiles are sorted by cross-section area
For many types of steel profiles (for example, for hot-rolled I-sections such as IPE), all types of sorting trigger the same result, as the all cross-sectional parameters increase accordingly, along with the height.

<table>
<thead>
<tr>
<th>Designation</th>
<th>Area (cm²)</th>
<th>Iy (cm⁴)</th>
<th>Iz (cm⁴)</th>
<th>Iyz (cm⁴)</th>
<th>it (cm)</th>
<th>lw (cm)</th>
<th>Wely (cm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>PE100</td>
<td>10.32</td>
<td>171.00</td>
<td>15.92</td>
<td>0.00</td>
<td>1.20</td>
<td>350.00</td>
<td>34.00</td>
</tr>
<tr>
<td>PE120</td>
<td>13.21</td>
<td>317.80</td>
<td>27.67</td>
<td>0.00</td>
<td>1.74</td>
<td>890.00</td>
<td>52.00</td>
</tr>
<tr>
<td>PE140</td>
<td>16.43</td>
<td>541.20</td>
<td>44.92</td>
<td>0.00</td>
<td>2.45</td>
<td>1880.00</td>
<td>77.00</td>
</tr>
<tr>
<td>PE160</td>
<td>20.09</td>
<td>869.30</td>
<td>68.31</td>
<td>0.00</td>
<td>3.63</td>
<td>3960.00</td>
<td>108.00</td>
</tr>
<tr>
<td>PE180</td>
<td>23.95</td>
<td>1317.00</td>
<td>100.90</td>
<td>0.00</td>
<td>4.79</td>
<td>7430.00</td>
<td>146.00</td>
</tr>
<tr>
<td>PE200</td>
<td>28.48</td>
<td>1943.00</td>
<td>142.40</td>
<td>0.00</td>
<td>6.98</td>
<td>12990.00</td>
<td>194.00</td>
</tr>
<tr>
<td>PE220</td>
<td>33.37</td>
<td>2772.00</td>
<td>204.90</td>
<td>0.00</td>
<td>9.07</td>
<td>22870.00</td>
<td>252.00</td>
</tr>
<tr>
<td>PE240</td>
<td>39.12</td>
<td>3892.00</td>
<td>283.60</td>
<td>0.00</td>
<td>12.88</td>
<td>37300.00</td>
<td>324.00</td>
</tr>
<tr>
<td>PE270</td>
<td>45.95</td>
<td>5790.00</td>
<td>419.90</td>
<td>0.00</td>
<td>15.94</td>
<td>70580.00</td>
<td>428.00</td>
</tr>
<tr>
<td>PE300</td>
<td>53.24</td>
<td>8355.00</td>
<td>603.00</td>
<td>0.00</td>
<td>22.43</td>
<td>125800.00</td>
<td>557.00</td>
</tr>
</tbody>
</table>

However, there is a whole range of profiles for which this order varies. Such is the case, for instance, for round pipes (where both the diameter and the wall thickness of the profile change) or W sections:

<table>
<thead>
<tr>
<th>Designation</th>
<th>Area (cm²)</th>
<th>Iy (cm⁴)</th>
<th>Iz (cm⁴)</th>
<th>Iyz (cm⁴)</th>
<th>it (cm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>W200x165x35.9</td>
<td>45.75</td>
<td>3433.00</td>
<td>764.30</td>
<td>0.00</td>
<td>14.56</td>
</tr>
<tr>
<td>W200x100x19.3</td>
<td>24.91</td>
<td>1662.00</td>
<td>115.40</td>
<td>0.00</td>
<td>3.99</td>
</tr>
<tr>
<td>W200x135x21</td>
<td>27.08</td>
<td>1989.00</td>
<td>251.30</td>
<td>0.00</td>
<td>3.74</td>
</tr>
<tr>
<td>W200x200x46.1</td>
<td>58.62</td>
<td>4545.00</td>
<td>1535.00</td>
<td>0.00</td>
<td>22.27</td>
</tr>
<tr>
<td>W200x165x41.7</td>
<td>53.17</td>
<td>4088.00</td>
<td>900.50</td>
<td>0.00</td>
<td>22.39</td>
</tr>
<tr>
<td>W200x100x22.5</td>
<td>28.69</td>
<td>2004.00</td>
<td>142.00</td>
<td>0.00</td>
<td>5.97</td>
</tr>
<tr>
<td>W200x200x52</td>
<td>66.50</td>
<td>5268.00</td>
<td>1764.00</td>
<td>0.00</td>
<td>32.41</td>
</tr>
<tr>
<td>W200x135x28.6</td>
<td>34.00</td>
<td>2587.00</td>
<td>329.80</td>
<td>0.00</td>
<td>7.35</td>
</tr>
<tr>
<td>W200x135x31.3</td>
<td>39.92</td>
<td>3139.00</td>
<td>409.60</td>
<td>0.00</td>
<td>12.04</td>
</tr>
<tr>
<td>W200x200x59</td>
<td>75.68</td>
<td>6111.00</td>
<td>2040.00</td>
<td>0.00</td>
<td>45.86</td>
</tr>
</tbody>
</table>

For these profile types, only the appropriate sorting can ensure that the optimum profile be found both quickly and accurately.

**Search range**

On the Optimisation tab, the available parameters on how to determine the number of profiles to be checked during optimization have been changed. Instead of only one, there are now two options available for setting a search range:

- Automatically determined;
- Automatically determined with a maximum limit.

The “Automatically determined” option causes the search to start from the initial profile and then continue with checking subsequent profiles (according to their sorting) until the first profile meeting the criteria (optimal) is found.

The “Automatically determined with a maximum limit” option functions almost identically, but, in this case, only the profiles within the set limit are checked.

*Note:* This second method was available in the previous version of Advance Design and setting limit value was done by entering the appropriate number in the text box next to the ‘Maximum number of iterations per cross section’.
What's New in GRAITEC Advance Design 2020

Note: An additional small change in the Steel Design Settings dialog: namely the “Number of complete calculations” option previously located in the Optimization tab has been moved to the “Calculation Sequence” tab, next to “Chained optimization”, as it concerns the iteration number for the chained optimization process.

Tension resistance of angle sections with net area (acc. EC3)

Main features & benefits:
- Verification of tension for angles with net section area
- Automatic determination of bolt hole diameter

Advance Design 2020 introduces the possibility of verifying the tension resistance (according to Eurocode 3) of angle cross sections considering a net section area.

According to the EN 1993-1-1 (§ 6.2.2.2), the net area is smaller than the gross area, because it is from the gross area that holes and other openings are deducted. Therefore, instead of designing angles based on the tension resistance of their gross cross-section (Npl,Rd), it is the resistance of the net cross-section (Nu,Rd) that should be checked.
EN 1993-1-1 (§6.2.3) provides a simple method (one single formula) for computing the tension resistance of the net area (\(N_{u,Rd}\)). However, we have implemented a more precise method in the 2020 version of Advance Design as provided in the EN 1993-1-8 (§3.10.3), where there are 3 different formulas for ultimate tension resistance depending on the number of bolts.

- For one bolt: 
  \[
  N_{u,Rd} = \frac{2 \times (e_1 - 0.5d_0) \times t \times f_y}{\gamma_{M2}} 
  \]  
  (3.11)

- For two bolts: 
  \[
  N_{u,Rd} = \frac{\beta_2 \times A_{net} \times f_y}{\gamma_{M2}} 
  \]  
  (3.12)

- For 3 or more bolts: 
  \[
  N_{u,Rd} = \frac{\beta_3 \times A_{net} \times f_y}{\gamma_{M2}} 
  \]  
  (3.13)

Where:
- \(t\) – thickness of the angle section
- \(d_0\) – bolt hole diameter
- \(\beta_2, \beta_3\) – reduction factors depending on the pitch between bolts.
- \(\gamma_{M2}\) – partial safety factor for bolts
- \(e_1, e_2\) – bolts edge distances
- \(p_1\) – bolts spacing
- \(A_{net}\) – the net area of the angle

The EN 1993-1-8 method applies to joints with a single row of bolts in one leg, which is the usual way of connecting angle or double-angle sections.

The new tension verification with \(A_{net}\), available on Advance Design 2020, also supports 2L sections (double angle cross-section), but only when these are defined from the library. 2L sections defined as a compound cross-section are not supported.

The global parameters used in the method are available on the Steel Design Calculation Settings dialog, in the newly introduced Angle verification tab:
Please note that entered values for spacing end and edge distances cannot be smaller than the limit values (according to 1993-1-8, tab 3.3):

- $e_1$, $e_2$ should not be less than $1.2 \times d_0$
- $p_1$ should not be less than $2.2 \times d_0$

The activation of the analysis with Anet, as well as that of the remaining bolt parameters (that might be highly depending on the individual element) are defined on the Property list of linear elements. In this respect, a set of new parameters is also available in the new **Angle verification** subgroup, located within the **Steelwork Design** set of parameters:

![Diagram of Angle verification subgroup](image)

**Note:** The **Angle verification** group is available for editing only for steel angle sections.

The **Net area** parameter activates a verification of the tension resistance considering Anet. It also edits all following parameters.

The **Bolt holes on** parameter is for selecting an angle section leg where bolts are located. If it is used for an unequal-leg angle connected by its shorter leg, Anet is taken as equal to the net area of an equivalent equal-leg angle with a size equal to that of the shorter leg. For example, for a 50x30x5 unequal-leg angle, the Anet area is considered as the same as for a 30x30x5 equal-leg angle.

The **Number of bolts** parameter, located at the end of the list, is for defining the number of bolts at one end (2 by default). Bolts are considered in one row.

The **Bolt nominal diameter** and the **Bolt hole diameter** properties are used for selection if $d$ and $d_0$ values are imposed or can be selected automatically.
Automatically selecting diameters if bolts and bolt hole are available on a dialog opened with a button located on the **Bolt / Bolt hole** property:

On the top, the editable table contains a definition of how the bolt diameter is selected in relation to the leg of the angle section. On the bottom, the rule for automatic determination of a diameter of bolt hole is displayed.

When the Nu,Rd check is the most unfavorable, then the relevant results (and Anet value) are visible on the steel shape sheet:
What's New in GRAITEC Advance Design 2020

A new report table for steel analysis results is available: **EC3 Tension verification for angles (§3.10.3(2) EN1993-1-8)**.

It contains the description of main parameters and detailed results for each verified linear element with angle section.

### Tension verification for angles

- **A_{net}**: Net area (§6.2.2.2 - EN1993-1-1)
- **\(N_{ed,td}\)**: Tension resistance of the net cross-section (§3.10.3(2) - EN1993-1-8)
- **d**: Bolt diameter
- **d0**: Bolt hole diameter

\[
\gamma_{M2} = 1.25 \quad e_1 = 2.00 \times d_0 \quad e_2 = 2.00 \times d_0 \quad P_1 = 3.00 \times d_0
\]

<table>
<thead>
<tr>
<th>No.</th>
<th>Name</th>
<th>Cross section</th>
<th>Case</th>
<th>(N_{ed}) (kN)</th>
<th>d (mm)</th>
<th>d0 (mm)</th>
<th>Bolt quantity (at each end)</th>
<th>Anet (cm²)</th>
<th>(N_r) (kN)</th>
<th>Work ratio (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>8</td>
<td>Linear</td>
<td>CE707</td>
<td>102</td>
<td>105.03</td>
<td>18</td>
<td>20</td>
<td>2</td>
<td>8.00</td>
<td>105.98</td>
<td>99</td>
</tr>
</tbody>
</table>

**New report table - Details about section and classification**

- **Better control when verifying profiles with variable cross-section**

Starting with this version of Advance Design, a new report table for steel analysis results is available: **Details about section and classification**. It contains the cross-section properties and the cross-section class for each location at which the checks are made.
This is a supplementary table for the standard Shape Sheet (which displays the cross-section properties and cross-section class only for the most detrimental cross-section of the element). However, these properties can vary along the length of the element if:

- the element has a variable cross-section;
- variable stresses along the element also cause variable cross-section class.

The new table has a similar structure to the Shape Sheet, having the 3 main categories: **Cross sections strength, Elements Stability** and **Fire resistance**, with the corresponding subcategories. For each subcategory, the following information are included:

- Case number for which the verification is made
- Location at which the verification is made – Mesh number and node
- Cross-section class and classification
- Cross-section effective characteristics (for Class 4 cross-sections)
- Cross-section properties for elements having variable sections
## Interactive optimization table

### Main features & benefits:
- Quick verification of details for steel design calculation

A small, but very helpful improvement has been introduced into the **Suggested Shapes** window, which contains a summary list with main design results of steel elements, together with information about suggested shapes.

This window has become more interactive, due to the possibility of clicking on cells with results, which results in opening a shape sheet for a given member.

The shape sheet is opened on the tab according to the column which was clicked on.
New HSS bracing joints

Main features & benefits:
- Modelling new types of connections
- Design for new types of connections

Advance Design 2020 allows definition and export for designing three new types of steel connections, using BIM Designers Steel Connection module:
- Tube Connection with Gusset Plate with one secondary element
- Tube Connection with Gusset Plate with two secondary elements
- Tube Connection with Gusset Plate with three secondary elements

Commands for creating new gusset plate connections with one, two or three additional elements (diagonals of round or rectangular tubes) can be found in the Connection panel, on the Objects ribbon.
The parameters of the defined connection can be found in the Properties window.

After the static calculation, new connections can be designed using the built-in BIM Designers module.
Improvements to BIM Designers Steel Connection module

Main features & benefits:
- Code verification (Eurocode 3) for bracing joints with hollow sections
- Increased modeling capabilities for column bases
- Easier definition of connection parameters

Detailed information about the changes to the BIM Designers Steel Connection module (embedded in Advance Design) can be found in a separate document about the novelties in BIM Designers Steel modules. Below you will find only brief information about selected main changes.

- **New connection type – gusset plate joint for bracings from hollow sections**

The new connection type allows for code verification of gusset plate connections with one, two or three diagonals of round or rectangular tubes.

- **Reduced Base Plate – reduced plate for pinned base plate columns**

The connection for column base has been extended to define a reduced plate (for pinned type connection).
- **Improvements to GUI**

Some parts of the dialog windows are redesigned in order to simplify the defining of connections. One of the changes is the introduction of a tree-view navigation that displays a hierarchical list of components that makes it much easier to navigate and modify parameters.
New options & Improvements – Reinforced Concrete Design

The main new options and improvements are related to the Reinforced Concrete Design.

Improvements for BIM Designers report generation on Advance Design

Main features & benefits:
- Better control over the generation of reports from BD modules
- Quicker and easier generation of reports

Advance Design 2020 has introduced additional options for accessing reports with design results from the BIM Designers modules. These improvements allow better control over the generation of reports, as well as quicker and easier report generation.

- Setting parameters for reports directly from the ribbon

The first change is a new Report Setting option added to the Design ribbon in the Result group. It opens a dialog, which is used to define the settings for the default generated report. The settings include the type of the report (e.g. if it is to be a Detailed report), the selection of chapters, the output file format and options for saving a file.
Generating different kind of reports directly from the ribbon

The list of options available after the expansion of the **Generate report** command has also been changed. There is a new **Generate Report** command to directly generate the default report, a command **Errors** to generate a report with warnings and errors and a command **Combinations** to quickly generate a report containing a list of load combinations.

Export the theoretical reinforcement area for planar elements

**Main features & benefits:**
- Possibility to export the theoretical reinforcement area for planar elements to Revit®
- Allows for reinforcement generation on slabs in Revit® using the BIM Designers Slab module

With Advance Design 2020, the export possibilities are increased by the export of results with theoretical reinforcement area for planar elements by Graitec BIM.

Export of such results is possible after activating of the **Export theoretical reinforcement** option available on the Graitec BIM tab of the **Options - Application** dialog.
These results can then be imported into Autodesk Revit®, so that it can be used to generate real reinforcement by using the BIM Designers Slab module.

**BIM Designers Wall on AD**

Main features & benefits:
- Analysis and reinforcement generation for concrete walls
- Analysis bearing walls and shear walls
- Analysis of individual walls and groups of walls
- Detailed design results, reports and drawings

Advance Design 2020 enables detailed analysis and generation of reinforcement for concrete walls thanks to a new BIM Designers module for Walls.

Using the new BIM Designers module, it is possible to design reinforced walls acting as Bearing walls or as Shear walls. Walls can be analyzed as individually or as groups.
Workflow

The workflow is the same as for other elements that can be designed using BIM Designers (beams/ columns/ footings/ steel connections): walls can either be opened directly, using the Design tab or can be exported to the BIM Designers standalone module, using a command from the right-click menu.

A wall modeled in Advance Design must meet several conditions in order to be exported / opened in the BIM Designers Wall module.
Required conditions:

- Element type: planar element, defined as Shell
- Material: concrete
- Geometry: vertical, rectangular

Walls that are analyzed as a group must belong to the same system, having activated the **Walls group** option:

In order to open walls in a BIM Designers Wall module as a group, several conditions must be met:

- Only 90-degree angles between two intersected walls are allowed
- Walls must be placed on the same level
- Wall must have the same Structural usage type (Bearing walls or Shear walls)
- Walls must be divided at intersection line - this means that, for example, the T-shaped group must be modelled with 3 walls instead of 2.
In the Design tab, a group system is displayed with all component walls:

If the group system is double-clicked, then the wall group is opened. If a component wall is double-clicked, then the individual wall is opened. It is also possible to use right-click menu on a group system; it depends on the selected command if either a wall group is opened (Open/Walls) or a wall group and all existing elements on its' system are opened.

If a wall is selected in the CAD view on the Analysis tab, and from the right-click menu Open with BIM Designers or Export to BIM Designers command is used, then:

- if this wall doesn’t belong to any group – this individual wall is opened/exported;
- if this wall belongs to a wall group - the entire wall group will be opened/exported.
Structural usage type

As different types of verification and reinforcement are required depending on a structural usage of exported wall (for example, if it is used to carry mainly vertical loads or acts as a shear wall), it is necessary to maintain consistent data when data is transferred.

For this purpose, in the property list of a concrete planar element there is a new parameter available: **Structural usage**.

It could have 3 possible values:

- **Auto (default)**
  - wall is exported as **Shear Wall**, if the model includes **Seismic load cases**
  - wall is exported as **Bearing Wall**, if the model doesn’t include **Seismic load cases**

- **Shear Wall**
  - wall is exported as **Shear Wall**

- **Bearing Wall**
  - wall is exported as **Bearing Wall**, if the model doesn’t include **Seismic load cases**
  - an error message (no export), if the model includes **Seismic load cases**
Load transfer

Sending a wall as a Bearing wall or Shear wall influences the transferred loads. For both cases, the source of load data is a resultant force calculated on the bottom of the wall.

For Shear walls, the resultant force is transferred and it is directly used for design;

For Bearing walls, the resultant force is transformed into an equivalent trapezoidal load.

For a group of walls, in addition to the resultant forces for each wall, a resultant force for a group is exported.
Resultant forces are transferred between Advance Design and BD Wall module with appropriate mapping:

For individual walls:

- $N \rightarrow N$ (vertical axial force)
- $M_z \rightarrow M_z$ (bending moment in wall plane)
- $M_f \rightarrow M_x$ (bending moment out of wall plane)
- $T_{xy} \rightarrow V_x$ (shear force in wall plane)
- no mapping for $T_{yz}$ (shear force perpendicular to wall plane)
- signs of resultant forces may be the same or reversed, depending on the local axis definition in wall.

For group of walls:

- $N_{Z/Group} \rightarrow N$ (vertical axial force)
- $M_{X/Group} \rightarrow M_X$ (bending moment about global X axis)
- $M_{Y/Group} \rightarrow M_Y$ (bending moment about global Y axis)
- no mapping for $T_{X/Group}$ and $T_{Y/Group}$ (shear forces in X and Y directions)
- signs are the same in Advance Design and in the BD Wall module, as the global axis system from AD corresponds with the global axis system in BD.

Calculations and results

The BD module supports rebar calculation according to Eurocode 2 and 8 (with national appendixes) and American ACI and Canadian CSA codes.

The outputs of the module depend on the wall type and if walls are analyzed individually or as a group:

- **3D rebar cage** – available for both wall types, for individual walls and group of walls.
What's New in GRAITEC Advance Design 2020

- **Reports** – available for both wall types, for individual walls and group of walls.

- **Diagrams** – available for Bearing walls.
- Interaction curves – available for Shear Walls.

- Drawings – available for individual walls

**Note:** More details about the BIM Designers Wall module's capabilities can be found in a separate What’s New document for BIM Designers 2020.
Main improvements to BIM Designers - Reinforced concrete modules

Main features & benefits:
- New precast section types for beams
- Automatic splitting of bars for beams
- Design of Corbels
- Improvements to report tables

Detailed information about the changes to the BIM Designers Concrete modules can be found in a separate document about the novelties in BIM Designers 2020. Below you will find only brief information about selected main changes, especially the ones relevant when working in Advance Design environment.

- Beam - New Precast sections

A set of new section types for Precast beams is available: Asymmetric, Rebate, Cantilever and Precast Slabs. All new section types are fully supported in terms of geometry definition, reinforcement generation and documentation:
- **Beam – Automatic bar splitting**

The new version offers the possibility to verify the maximum length of bars and the possibility for automatic splitting bars that are too long.

There are also three methods available for joining divided bars: by welding, by mechanical couplers and by lapping of bars.

- **Beam – Longitudinal bars across the entire multispan beam**

The new version offers the possibility to define Longitudinal and Anti-crack bars continuously across the entire multi-span beam.

- **Beam - New geometry type: Corbel**

New beam geometry type is available: Corbel. It allows for designing a corbel (short cantilever) having constant or variable height according to EN 1992-1-1 or ACI/CSA codes.
Beam – improvements to reports

The Deflection verification table is now extended and arranged into two tables and contains more information and intermediate results.

<table>
<thead>
<tr>
<th>Intermediate values</th>
</tr>
</thead>
<tbody>
<tr>
<td>Span</td>
</tr>
<tr>
<td>1</td>
</tr>
<tr>
<td>2</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Deflection verification</th>
</tr>
</thead>
<tbody>
<tr>
<td>Span</td>
</tr>
<tr>
<td>1</td>
</tr>
<tr>
<td>2</td>
</tr>
</tbody>
</table>

The limit for L/d is calculated according to EN 1992-1-1, 7.4.2, (7.16.a or 7.16.b).

Summary tables were also revised to show results for decisive sections only, so that are shorter now and clearer.
New options & Improvements - Miscellaneous

Below is a list of new options and improvements from other domains, especially those that address modelling and cooperation with other software.

Export data from Advance Design to CS-STATIK

Main features & benefits:
- Possibility of using the Advance Design model as a source for CS-STATIK
- Possibility of cooperation at the different modelling levels (by using a descriptive or a calculation model)

To enable a more complete cooperation between Advance Design and CS-Statik modules, a direct link between the two systems has been introduced in version 2020. It is possible to export data from Advance Design from the descriptive and the analytical model. For both cases, a single CSX file is imported into the Project center of CS-Statik.

Note: It is required that CS-Statik version 2020 is installed on the computer to be able to export data.

In the case of an export data from the descriptive model, all information on geometry, cross-sections, materials, as well as configurations for timber connections are sent.

In the case of the analytical model, the export additionally covers internal loads or forces:
- for exported footings the support forces of the column;
- for exported beams and columns equivalent loads that result the same deformation and internal forces (include the external loads as well as loads being generated from internal forces, moments and support forces);
- for timber connections the internal forces and moments of the connected beams or their support forces.

The export is run with using a new Export CSX command, located on the BIM ribbon:

Note, that this export button is only visible if the localization is set to Germany.
When pressed, the selected elements (and even the entire structure) are exported. Next, the CS-Statik application is launched.

The objects exported are:

- Foundations (punctual, linear);
- Columns (concrete, steel and timber);
- Beams (concrete, steel and timber);
- Walls (concrete);
- Timber connections;
- Concrete slabs (only CAD definition is exported)

Detailed graphical representation of releases on linear elements

Main features & benefits:
- Easy graphical identification for defined releases
- More precise symbols and descriptions

Advance Design 2020 provides a new graphical representation method to be able to easy distinguish releases for each direction.

The selection of the type of symbols for releases is made on the Display Settings dialog (ALT+X), on the list of symbols for linear elements:
There are two selections available:

- **Releases (simplified)**
  
  This is the previous symbols type; regardless of which directions are released, the same symbol is displayed (2D circle); it provides information on the release’s existence and not for its exact definition.

- **Releases (detailed)**
  
  This is a new symbols type; displays a combination of the 6 different symbols for each end, according to the defined releases; it provides detailed information on the releases in a graphical form.
The detailed symbols for releases:

- **Rx**
  - 2D circle in the YZ plane

- **Ry**
  - 2D circle in the XZ plane

- **Rz**
  - 2D circle in the XY plane

- **Tx**
  - Two lines parallel to X direction

- **Ty**
  - Two lines parallel to Y direction

- **Tz**
  - Two lines parallel to Z direction

When defining multiple releases, the component symbols are displayed simultaneously, at the end of the bar. To make it easier to distinguish directions, when selecting an element, the colors of the symbols correspond to the colors of the local axis:

Additional improvements are also done to annotations of releases. Descriptions are now displayed separately for each end of linear elements, just above a defined release. In addition, in the case of releases defined on more than one local direction, the text is reduced to not repeat the R or T character, for example Rxyz instead of Rx Ry Rz.
Reduction coefficients for stiffness for linear elements

Main features & benefits:
- Full control over the structure
- Analyzing using cracked sections of the concrete elements

Earlier versions of the program allowed manual or automatic reduction of stiffness for a concrete linear element, by modifying the modulus of stiffness $E$, resulting in a modification of stiffness in all directions.

Advance Design 2020 allows for defining of effective stiffness coefficients per element, separate for 4 stiffness (axial, torsional, bending in the strong and in the weak axis).

The manual reduction of the stiffness of the element in a given direction may be useful for modelling the different behaviors of the element, and the reduction of the section's moments of inertia allows for considering section cracking of the concrete elements in static calculations independently in the chosen direction.

The definition of the reduction coefficients is available on the property list of the concrete linear elements, when the Concrete inertia type property is set to **Imposed value**. For Cracked section inertia a button is available to open a new dialog:

![Dialog for defining reduction coefficients](image)

In this window, users can define effective stiffness, by entering the reduction coefficients separately for:

- the bending stiffness on the strong axis ($y$);
- the bending stiffness on the weak axis ($z$);
- the torsional stiffness;
- to the axial stiffness.


All provided coefficients must be > 0.0, and by default are equal to 1.0 (no reduction).

The deflection of cantilevers under the same load, but when the vertical stiffens of the right one is reduced by half.

**Improvements on Grid axes**

Main features & benefits:
- Possibility to define names of Grid axes

The tool for defining Grid axes has been improved regarding names of axes.

One of the novelties is the possibility for using prefixes with automatic naming of axis (1, 2, 3, ... or A, B, C, ...), separately for each direction. Prefixes are defined on the Grid’s property list:
Another novelty is the possibility for renaming any individual axis.

In a first step, the Text type property should be set to Defined. Then, on the Text definition property, it is possible to use a button for opening a dialog for renaming axes:

On this dialog, we can define names for each axis separately, using any text:
A novelty is the possibility to hide a frame around an axis, separate for each direction:

**Update to the timber material database**

Main features & benefits:
- Design of timber structures using material parameters, compliant with the latest standards
- More flexibility in the definition and use of custom wood for calculation

In the new version of the Advance Design, a set of changes has been made concerning timber materials.

- **Update of timber material library to current versions of EN 338 and EN 14080 codes**

The list of available timber classes as well as their strength parameters have been modified to comply with the latest version of the EN 338 standard (*EN 338:2016 Structural timber – Strength classes*) and EN 14080 standard (*EN14080: 2013 Timber structures. Glued laminated timber and glued solid timber*).

The changes concerned three libraries:

**EN 338 – 2016 (Softwood)**, that now contains:
- Classes C14 to C50 (acc. Table 1 - Strength classes for softwood based on edgewise bending tests)
- Classes T8 to T30 (acc. Table 2 - Strength classes for softwood based on tension tests)

**EN 338 – 2016 (Hardwood)**, that now contains:
- Classes D18 to D50 (acc. Table 3 - Strength classes for hardwoods based on edgewise bending tests)

**EN 14080 – 2013 (Glued woods)**, that now contains:
- Classes GL20h to GL36h (Strength classes for glued laminated timber having homogeneous layers)
- Classes GL20c to GL36c (Strength classes for glued laminated timber having combined layers)
**Modification of the determination of selected timber modulus**

When designing timber elements, both $E_{0.05}$ and $G_{0.05}$ parameters (5% value of modulus of elasticity and shear modulus) are no longer determined by the calculation engine but are taken from the material database. This is particularly useful for calculations using custom timber types.

For that purpose, new columns for $E_{0.05}$ and $G_{0.05}$ have been introduced in the material database.

For timber materials that comply with EN338:2016 and EN4080:2013 standards, both values are according to the data from the relevant code tables. For other timber materials existing in libraries that are not covered by EN14080 or EN338, the $E_{0.05}$ value was computed as $2/3 \times E_{0.05}$ (mean value of modulus of elasticity) and $G_{0.05}$ as $2/3 \times G_{0.05}$ (mean value of shear modulus).

**Changes in naming of selected options**

<table>
<thead>
<tr>
<th>Main features &amp; benefits:</th>
</tr>
</thead>
<tbody>
<tr>
<td>- More unified terminology</td>
</tr>
</tbody>
</table>

To unify and better understand the used terminology, some terms and names of options used in Advance Design have been modified.

These changes concern mainly selected English terms, but may also be visible in all other language versions of the program.
Examples of changes:

- Pilot -> Project Browser
- Windwall -> Load Area
- Torsor -> Resultant Force(s)

Easy display of element length

Main features & benefits:
- Quick way to check the length of the element

To the list of available parameters that can be presented in a tooltip, a length for linear elements has been added. This small but useful improvement lets users check the length of the modeled linear elements, quickly and easily.
To configure the type of information displayed in the tooltip (visible when hovering the cursor over an element), use the tooltip configuration window, opened by using a small icon located on the bottom status bar.

In this window, you can easily decide what the content of Tooltip should be.

### Possibility to display footing dimensions on a model

**Main features & benefits:**
- Possibility of presenting more comprehensive information about the model in the view

To increase the possibility of presenting information about modelled elements, new annotation types for displaying footing dimensions are now available.

The dimensions of the foundation are the parameters of the support and can be defined manually or can be updated automatically (thanks to the possibility of sizing the foundation, using the BIM Designers Footing module). Thanks to the possibility for displaying annotations with footing dimensions, it is very easy to see the current foundation’s geometry data.
To display dimensions of supports in a view, in the Display Settings window (ALT+X) add a new annotation for supports, and then select one of the two new annotation types from the list.

There are two annotation types available:

- Footing dimensions (a x b x h) → displays all 3 footing dimensions for point supports
- Footing dimensions (a x b) → displays both footing widths for point supports or a width and a height for linear supports

Possibility for selecting Load areas by their types

Main features & benefits:

- Easier verification of data for automatic wind generation

On the Elements selection dialog (ALT+S) - Types tab, there is a new possibility for selecting Load areas by Type parameter. The type of load areas is used for an automatic wind generation.
New options to the unit’s settings

Main features & benefits:
- Simpler presentation of results for specific cases

Two new options are added to the Working Units Definition dialog:
- Hide zeros after the decimal point
- Scientific notation for values < than a limit
If the option for hiding zeros after the decimal point is checked, then all values without a decimal part are presented without zeros after the decimal sign (for example: 25.00 → 25). This option affects only the results displayed on viewers.

If the option for displaying scientific notation for values smaller than defined limit is checked, then it is possible to select the limit value from the list. Then those values that are less than the limit are presented in the scientific notation (or example: 0.000025 -> 2.5e-005).

### The drift-check for seismic SLS load combinations according to the Romanian P100-2013 code

**Main features & benefits:**
- More complete seismic verification for Romania

Advance Design 2020 allows now to perform the level drift checks according to the Romanian code P100-2013 also for SLS states with seismic actions.

For this purpose, a new type has been added to the list of available load combination types: ASLS.

As the Romanian combinations code CRO-2012 does not offer a possibility to create SLS combinations with Seismic component, such combinations need to be defined manually. These combinations can now be declared as the ASLS type, allowing them to be used by level drift checks.
Miscellaneous Improvements & Corrections

Advance Design 2020 brings many improvements and corrections. Below you can find a brief description of the most noticeable corrections we’ve selected:

Selected corrections:

- Commands added to the ‘fast start’ are no longer lost after restarting the program. (#16723)
- Annotations related to releases for planar elements on selection are now displayed without any issue (#18125);
- Values of min and max limits for the strength/stability work ratios as displayed on the Steel Design Settings dialog - Optimization tab, are now validated in order to avoid entering a bigger value for min limit than the one entered for max limit. (#18741)
- For tapered beams, the \( k_{cr} \) for timber design is now calculated along the entire span of the tapered beam, not only once for a given section as before. (#18807)
- The picture on the Buckling dialog for Timber design is modified to avoid misleading of axis convention. (#18611)
- Improvements have been brought to the issue related to the influence of local coordinate systems on walls regarding the values for resultant forces (torsor) for a group of walls. (#18492);
- On reports, the manual size adjustment is available for the window in Masses > Combinations (Select cases button) from the Modal Load Case properties list. (#18644)
- An information that the linear element (RC beam or column) is set as a ‘main seismic element’ can now be exported to the Graitec BIM Designers modules. (#18148)
- Steel Design with 2nd order effects (local analysis with warping) is now also possible for selected AISC/CSA steel design codes.