AlfaCAD 2.7 Static analysis, (almost) complete solution

This article is a supplement to the :
AlfaCAD 2.6 Supplement (please check it out if you haven't already) and
AlfaCAD 2.6 Supplement 2 (please also check here if this is something new to you)
which were mostly devoted to extending the AlfaCAD program with a module for static analysis of frames and trusses. One of the most important extensions in version 2.7 is the automation of calculations of internal forces, displacements and stresses resulting from the action of load combinations, in accordance with the guidelines of the European Eurocode standards as well as the American ASCE 7-10 and ICC. Due to the multitude of standards, the ACI 318-11 or AISC 2010 standards have been omitted. However, the differences between the standards do not exclude their use thanks to the full flexibility of establishing coefficients that eliminate these differences. Experience will show whether it is necessary to add other standards than those currently developed. The final effect of the calculations is the determination of longitudinal stresses (sigma) on both sides of the element cross-sections and shear stresses (tau).

Load nature

By introducing any type of load, whether a concentrated force or a moment at a node or at any point along the length of the element, or a uniformly or non-uniformly (trapezoidally) distributed load along the entire length or a section of the length of the element, beside the load value or load value range, it is possible to assign its nature, as follows:

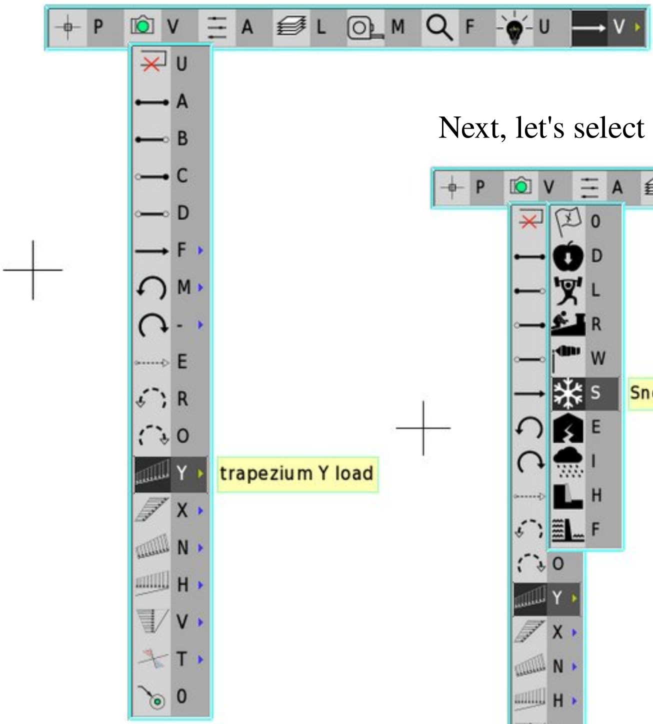
- Dead Load (DL) e.g. the dead weight of the element or the load from other permanent elements of the structure,
- Live Load (LL)
- Load on roof (Lr) understood as the instantaneous operating loads generated by maintenance workers, including their equipment and materials,
- Wind Load (WL),
- Snow Load (SL),
- Earthquake Load (EL) - a seismic load,
- Rain Load (RL) - a rainwater and ice load,
- Hydrostatic Load (HL) - a hydrostatic soil pressure load,
- Fluid Load (FL) - hydrostatic fluid pressure loads,
- Thermal Load (TL).

Each of the load characteristics, visualized by the symbol DL, LL, Lr, WL, etc. may or may not be accompanied by a load variant number. A variant is understood as a load character with different partial load factors and/or different load simultaneity factors in load combinations. If the load character is not declared, the load is treated as a live load (LL). If the load variant is not selected, the default load and load simultaneity factors are assumed.

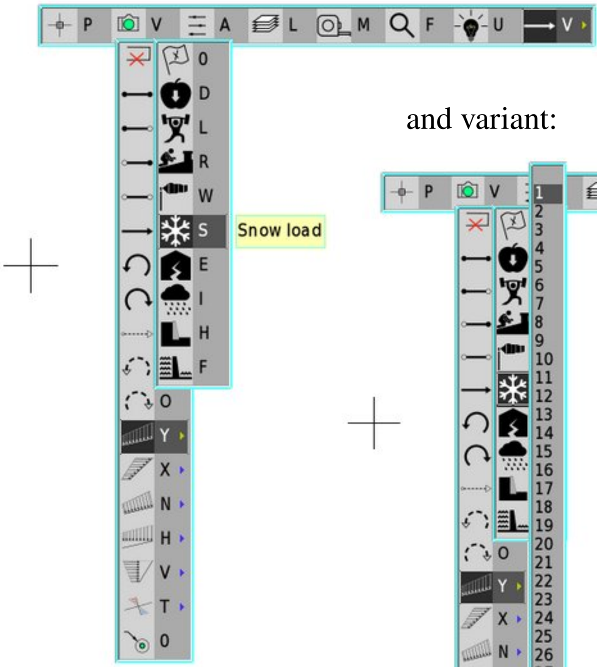
Drawing the load vector

In the Draw - Vector function, the last selected vector type is initialized, and if it has not yet been drawn in the program session, the default vector is a rigid-rigid element. In the auxiliary menu (as a reminder, the menu is opened with the spacebar or the Ins key or the middle mouse button), select the appropriate load type, whether concentrated or distributed. As a reminder: in the case of distributed load, at this stage there is no distinction between uniformly distributed or linearly variable loads, because this is determined by the initial and final values. If both values are identical (set by a single numerical value in the edit line, e.g. 50, or in the form of two identical values, e.g. 50;50 ,both values - as in all operations in AlfaCAD - are separated by a semicolon), and additionally the entire element is covered by the load range, such element load is treated by the program as uniformly distributed. If both values are different or the load covers only a part of the element, such load is treated as a trapezoidal load, even if both initial and final value are identical.

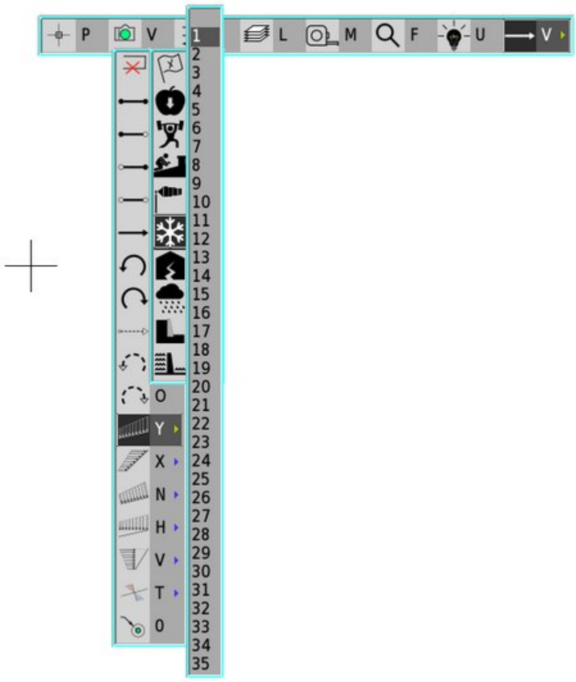
Due to the fact that the Frame 3DD module accepts a single uniformly distributed load for each element, these loads are summed for each load combination and as a single value enter the equation system. Trapezoidal loads can appear as multiple load components, each of them can include the part of the element that it actually loads. This is important when analyzing the results based on the generated printout, if necessary, because all results are presented graphically in the drawing, as a complete result of the solution taking into account the load combinations.
Let's choose a trapezoidal load as an example:



Next, let's select the load nature:



and variant:

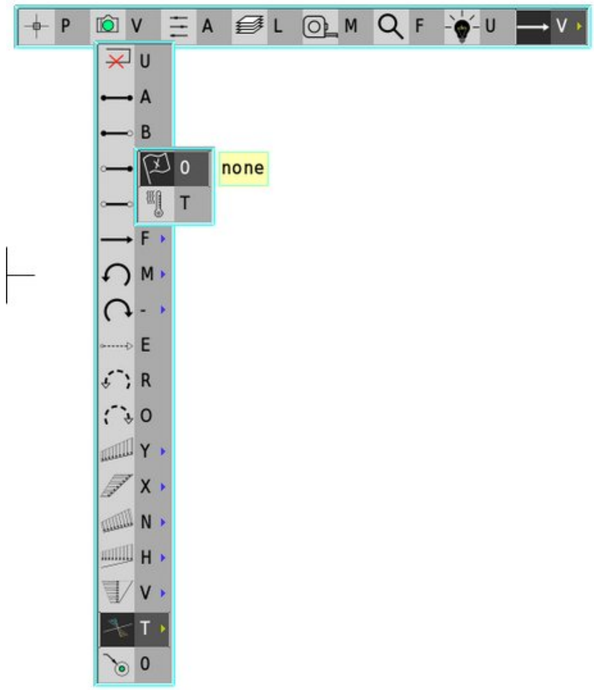


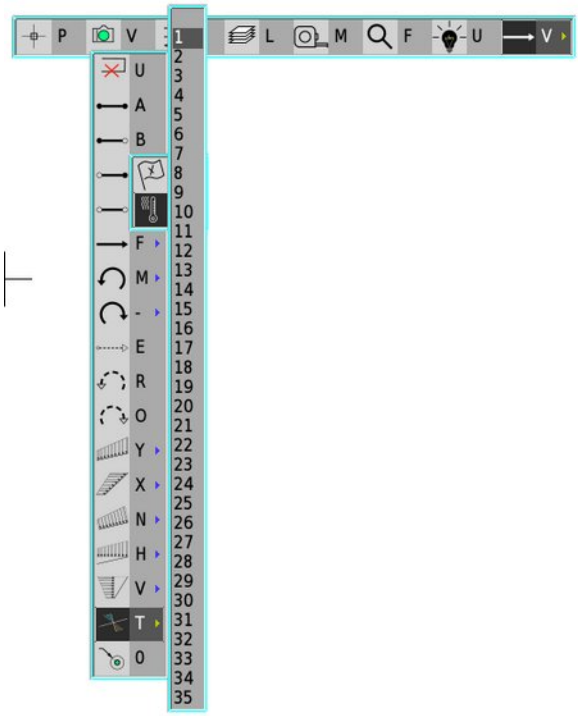
Selecting the load nature is not mandatory, although if not set the load will be treated as a live load.

Therefore, selecting the first icon means accepting the default nature, and the remaining icons suggest the nature of the load, successively - permanent load (Newton's apple related to gravity), live load (lifted weight), roof service load or other temporary technological load (person on the roof), wind load (sleeve - wind indicator), snow load (snowflake), seismic load (cracked house), rainwater and ice load (storm cloud), hydrostatic soil load (retaining wall) and liquid load (dam). The thermal load icon does not appear at this stage, because there is a separate type of vector describing this type of load, selected a step earlier.

When selecting a thermal load, the choice of the load character is basically determined by the type of vector, and is only intended to select a variant if the default values of the coefficients in the combinations should be replaced by other values.

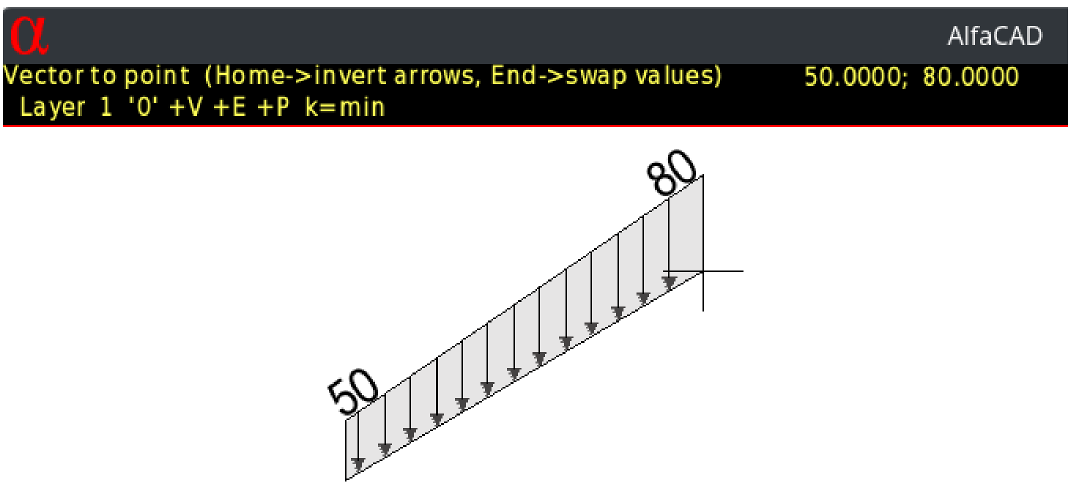
The character can be ignored, the load will always be treated as a thermal load due to the vector type, but if you need to change the default coefficients for this load, select the thermometer icon and then a variant in the form of a number from 1 to 35 (theoretically, this is how many variants of coefficients can be set for each of the load characters)



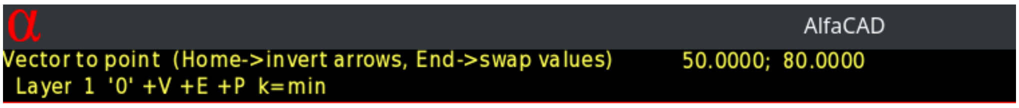


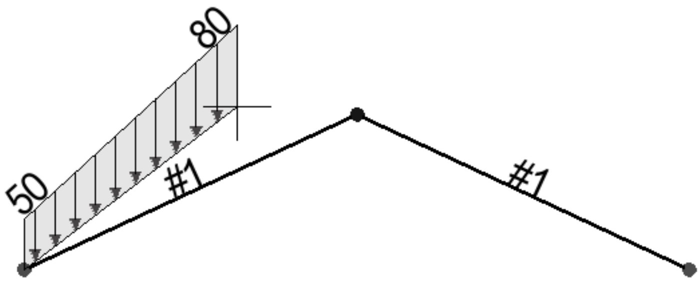
Load vector. Drawing method

The load vector, in the case of a distributed load is drawn in such a way that it "touches" the loaded element. The functions of locating the point relative to the object are helpful, such as attaching it to the ends of the element, or if the load does not cover the entire element, locating it relative to the point closest to the indicated element. In each case, this is sufficient for the load to be associated with a given element. A load vector not associated in this way with any of the elements, but marked in the load combination, will be identified and treated as an error requiring correction, i.e. adjustment of the ends of the load vector to any points lying along the length of the elements. Below is an image of the load vector in the phase of determining the load range points, for demonstration purposes not associated with any of the elements. Initially, the initial and final values of the load are zero. Pressing the number keys automatically switches to edit mode, allowing you to set a value, in this example 50;80.



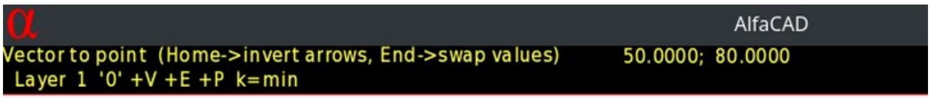
Below is the same vector attached to the ends of the roof rafter element.

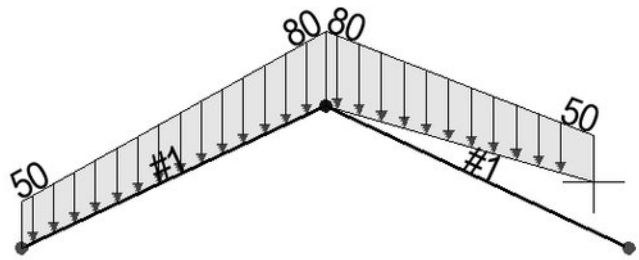




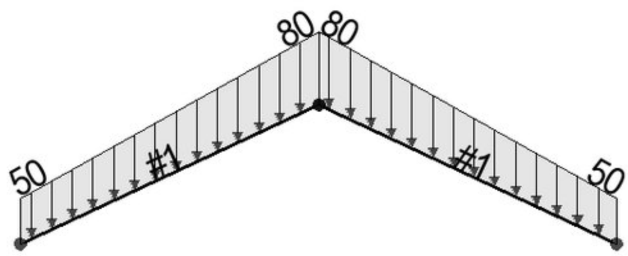
Additional operations for declaring distributed load values

When a load vector is drawn, in this case uniformly or trapezoidally distributed, the initial values are 0;0. Editing the values directly from the keyboard allows the user to declare an alternative constant value, e.g. 50 or 50;50 or a uniformly varying value, e.g. 50;80 (kN/m or kip/in). Once the vector is entered (usually attached to the ends of a frame or truss member) it is possible to continue drawing another load vector, although in the case of trapezoidal loading the initial and final values are swapped, allowing the load to vary uniformly by default, although this is not always the designer's intention.

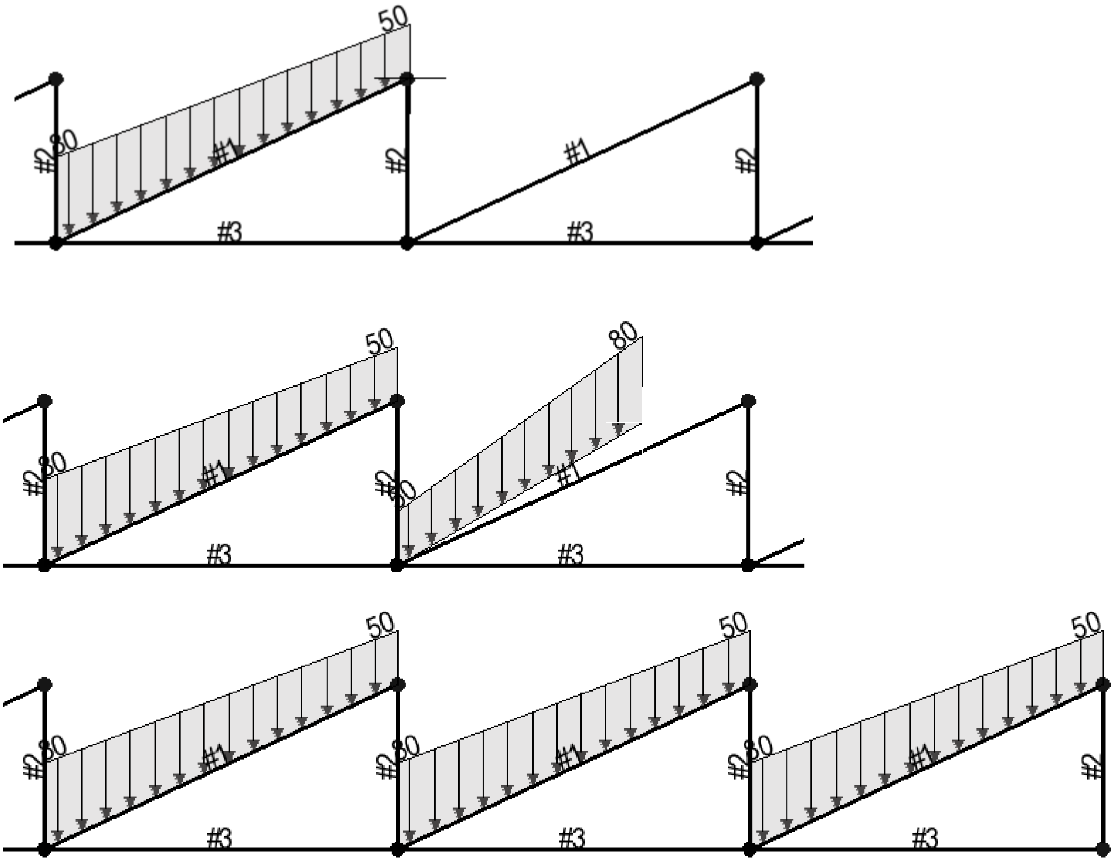




And finally:



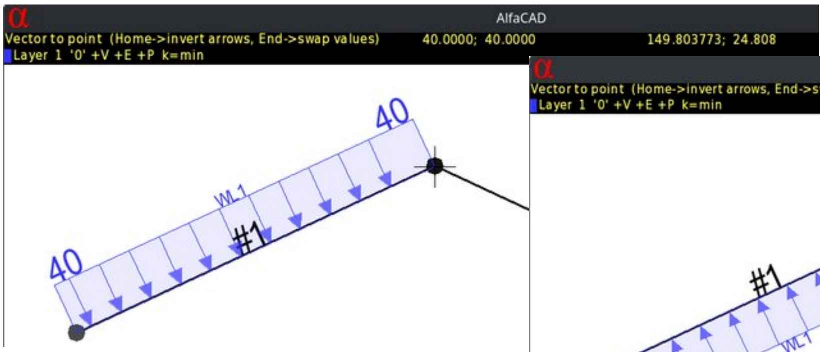
If the values of the next vector should be different, you should simply enter them from the keyboard again, and if the previously entered values are still valid but should not be changed, just press the End key to swap them. This can happen, for example, in the case of snow load on the sawtooth roof segments, as in the drawing below.



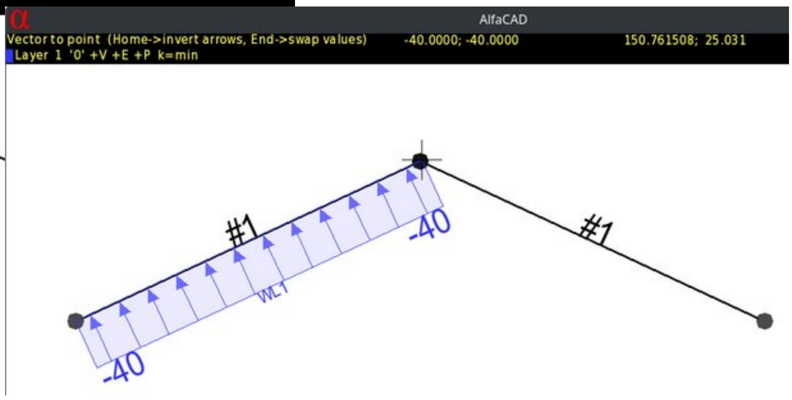
Load inversion

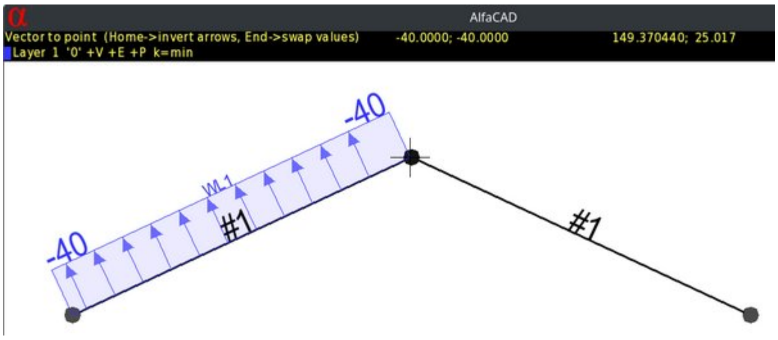
In AlfaCAD, the principle is that a positive load perpendicular to the element axis, e.g. a distributed load, is directed opposite to the local Y axis. This means that if the starting point of the element is to the left of the end point, the load in the direction of gravity forces (and therefore natural for both DL and most of LL loads) has a positive sign, because it acts opposite to the local Y axis.
This principle allows in most cases to avoid a significant number of "-" signs in the declaration of distributed forces and loads, and also naturally orders the setting of loads from left to right, although this does not exclude setting in the opposite direction, in which case the sign of the value should be changed. In each case, the direction of the arrowheads, regardless of the sign of the force or load values, clearly indicates the direction of the forces. In some situations, e.g. in the case of wind load, when the direction of the wind forces is opposite to the direction of the gravity vector, assigning negative values or creating a load vector of the element whose starting point is to the right of the end point, in order to obtain the correct, opposite to the gravity force direction of the force, may be graphically problematic, because the force diagram would be located on the inside of the roof slope, which could cause the recipient of the drawing to misunderstand the intentions.
In such a situation, it would be recommended to use the Home key, which results in the inversion of the diagram, o the arrows are not directed in the direction of the element axis, but rather in the opposite direction, which improves the readability of the drawing. The Home key for inverting the load diagram, just like the End key for changing the trapezoidal load value, works alternately.

Positive load (wind WL)



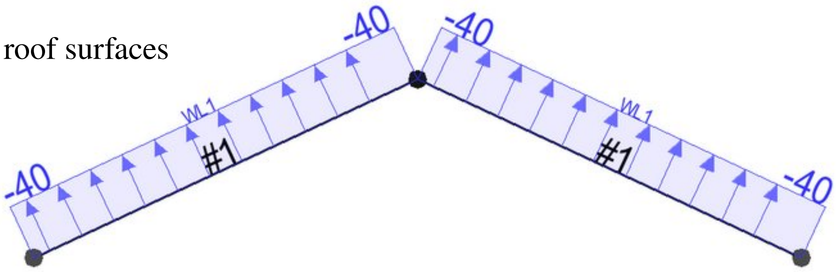
Negative load





Negative load inverted

For both roof surfaces

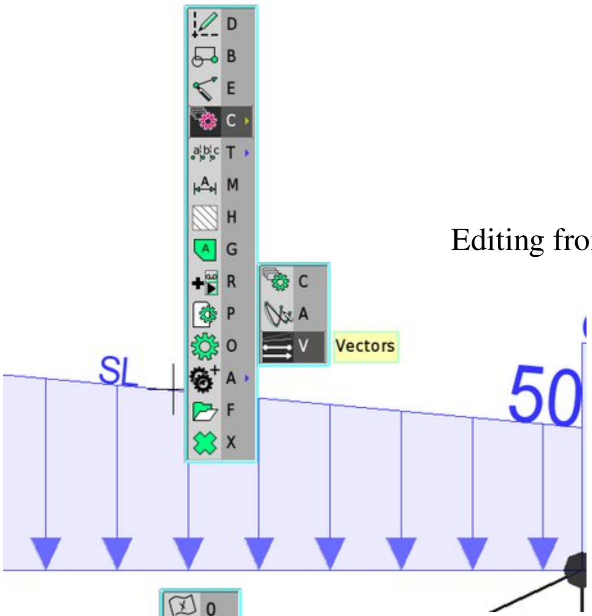


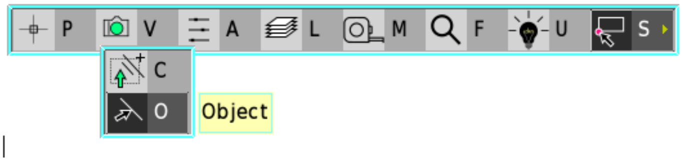
It is worth recalling what was mentioned in the Supplement to version 2.6 and Supplement 2, in each situation it is possible to change the value of the load, both concentrated and uniformly or trapezoidally distributed, in the Edit - Stretch function with the Object selection option or in the newer version 2.7 also or primarily in the Change - Vector function, after indicating either the text defining the initial or final load, or alternatively the appropriate end of the load diagram line. In version 2.7 at the same time, the option of inverting the graphical representation of the load is active via the Home key, operating in alternating mode.

Change of nature or variant of the load

In the Change/Vector or Edit/Stretch function with the Object option, pointing the cursor at the character symbol (or the place where it should appear if it were declared, i.e. at the geometric center of the load on the value side) allows you to edit the character and variant. This operation can also be performed in the Info function of the auxiliary menu.

Editing from the Edit - Stretch function

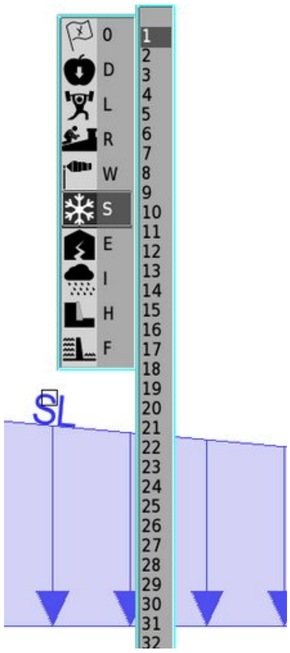


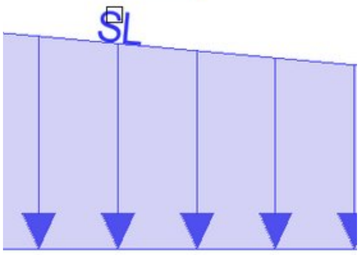


Editing from the Change - Vector function

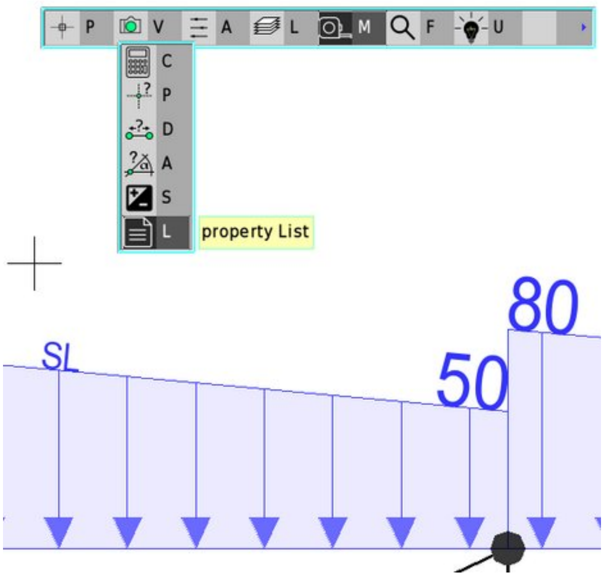
Re-editing load nature

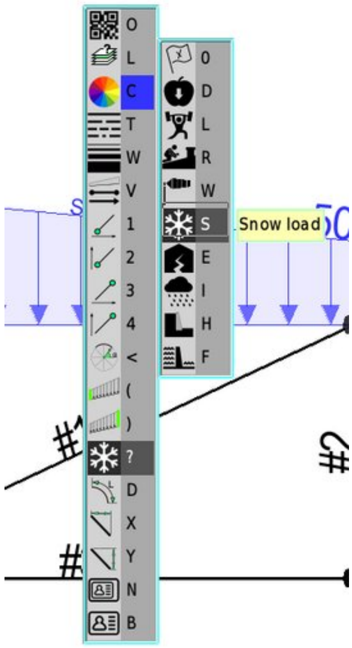
Re-editing the variant number

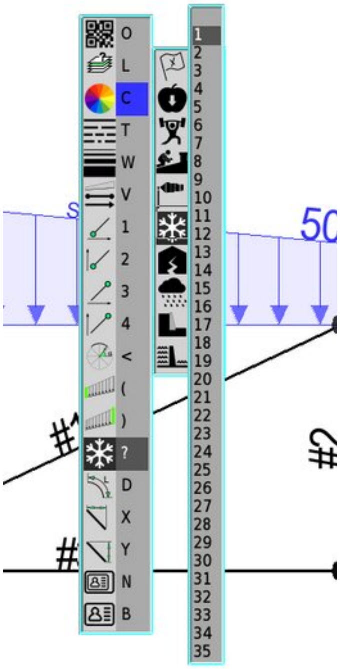


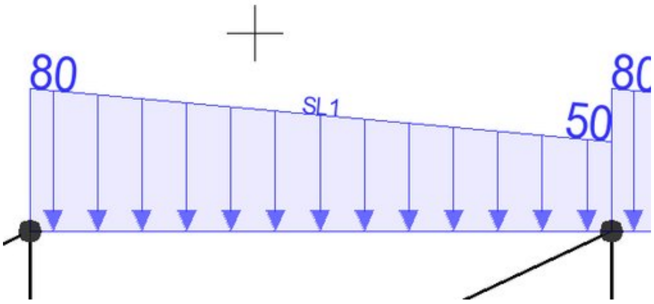


It is also possible to change the nature of the load and the variant, as well as the load value in the "Property List" function available from the auxiliary menu, which displays all the parameters of the indicated object, some of which are editable. In the case of load vectors, the nature of the load and the variant are editable.









Load cases

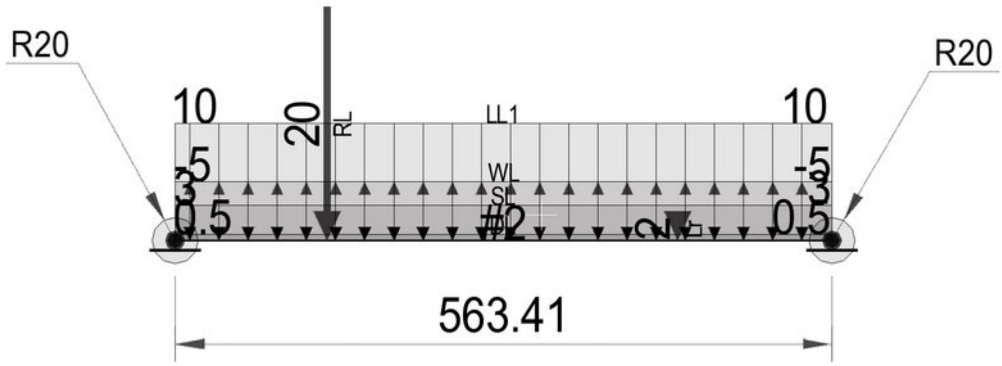
It is possible to declare multiple load cases, along with a declaration of their nature and variant. Loads can be drawn on multiple layers if it makes it easier to work on their correct declaration, because placing all loads on one layer can cause too much nesting of load vectors, which makes the drawing less readable. However, when it comes to selecting loads for static analysis, the WYSIWYG principle applies - "What You See Is What You Get". When starting a static analysis, you should select all structure elements, texts describing properties, text describing the frame/truss identifier, and additional texts describing non-standard load factors and simultaneity factors, if they are needed. Select the value of the gravitational acceleration too, if the self-weight load is taken into account in the calculations, together with any dead load factors and simultaneity factors, and together with all loads taking part in the load combination. This means that everything that is visible (i.e. visible layers) and that is selected, using the same functions used for other operations such as moving, rotating, etc., will be taken into account in the static analysis. As in the editing functions, such as moving, rotating, changing the scale, etc., selecting a block in window / cross mode, and adding objects to the block or eliminating them after pressing the F10 key, all of this has the same application. Just WYSIWYG.

When the operation of indicating the elements of the block for analysis is completed with the Esc key or the right mouse button, the program asks whether to perform the analysis. This is the time when the operation can be abandoned to make some modifications, such as turning on or off the layers on which additional loads are placed. If the user considers the block to be complete, the static analysis initiation dialog allows the selection of the standard: Eurocode, ASCE or ICC, which results in the selection of load factors, load simultaneity and load combinations appropriate for the standard, as well as allowing the consideration of geometric stiffness (**Δ** P). The additional parameter is the combination number (default 1) because it can be assumed that for various reasons more than one set of loads can be taken into account. It should be noted that this is a set of loads, not individual load, so they, once selected, will be taken into account in the load combination in order to determine the maximum stresses in the structure elements, as well as representative displacements of nodes and deformations of elements under the influence of the combination of characteristic loads, as well as permanent deflections under the influence of the quasi-permanent part of the characteristic loads in the case of the Eurocode standard.

Example of a steel beam supported by rigid supports

To demonstrate the details of the analysis with load combinations, let's use a very academic example of a beam loaded with several types of loads:
1. self-weight of the beam (g)
2. dead load of elements resting on the beam with a characteristic value of 0.5 kN/m (DL)
3. live load with a constant value of 10 kN/m (LL1)
4. concentrated force at a certain distance from the support, with a value of 20 kN from the load of rainwater and ice (RL)
5. Temporary (maintenance) load with a value of 2 kN (Lr)
6. Snow load with a constant value of 3 kN/m
7. Wind load with a value of -5 kN/m

Of course, such a set of beam loads can only refer to a purely academic task, but it is a test example in order to take into account many different load characteristics in the case of the simplest possible structure. Intentionally, rigid, non-sliding supports were used on both sides of the beam to demonstrate the limitations of the program. Rigid nodes with a radius of 20 cm (200 mm) were also assigned on both sides of the beam.



Static analysis requires a few more descriptions in the form of texts appended to the analyzed block, these texts may have the hidden text attribute, so they do not necessarily have to be printed or saved in a graphic file or PDF format. Here they are:

```
%LL1 γ=1.5 Ψ0=0.7
%DL1 γ=1.35 ξ=0.85
%FRAME: A4 linear static analysis of a 2D truss (kN, kN/m, mm, Celsius)
#2 h=180 b=91 A=23.95 Asy=12.24 Asz=8.74 Iy=1317 Iz=100.9 Wy=146.3 Wz=22.16 E=210 G=81 r=0 d=7850 a=1.2e-05 γ=1.25 ξ=0.85 IPE 180
%g=9.81 γ=1.2
```

%FRAME defines the name of the frame, and the initial text up to the first space will be treated as an identifier, used as part of the name of the layers containing the results. It is recommended to keep this part of the text as short as possible (in this case it is simply "A4", and this designation uniquely identifies the frame, many of which can be placed in one drawing, although the number of layers for generating results limits this number to 7 in practice).

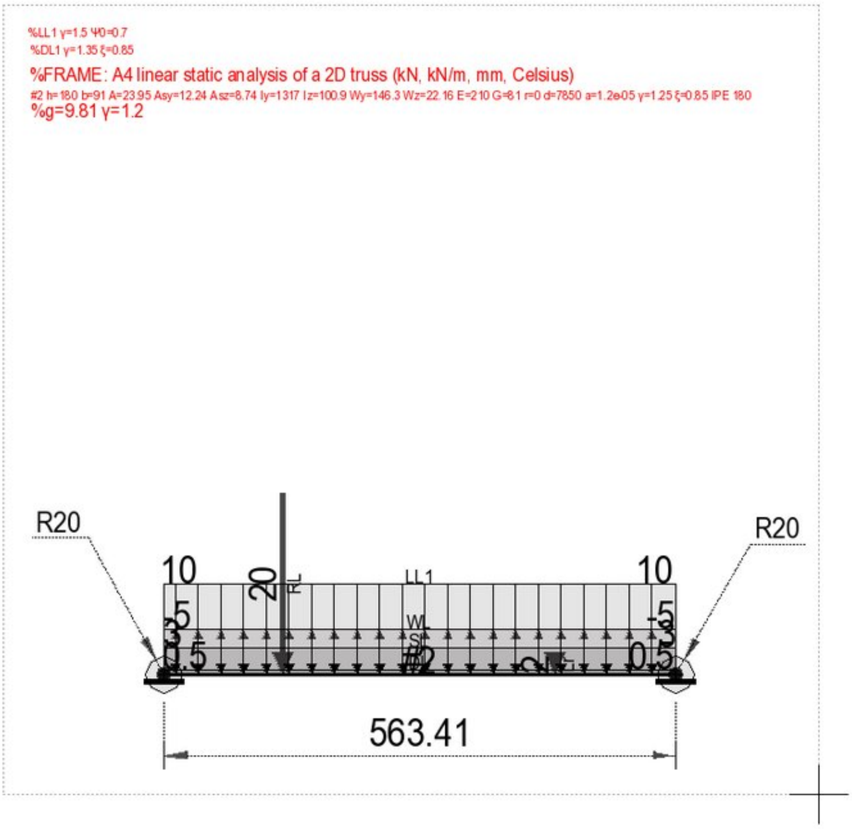
#2 defines properties #2, and such a property has a beam of the structure. Description in Supplement 2.
%g defines the value of the acceleration of gravity. It is commonly known, but it is introduced for identification so that the program calculates the self-weight of the structure, and takes it into account in the load combination, with default factors, or given factors, in this case the partial load factor was given with the value γ=1.2

%LL1 re-defines the partial load factor γ=1.5 and the load simultaneity factor in combinations Ψ0=0.7

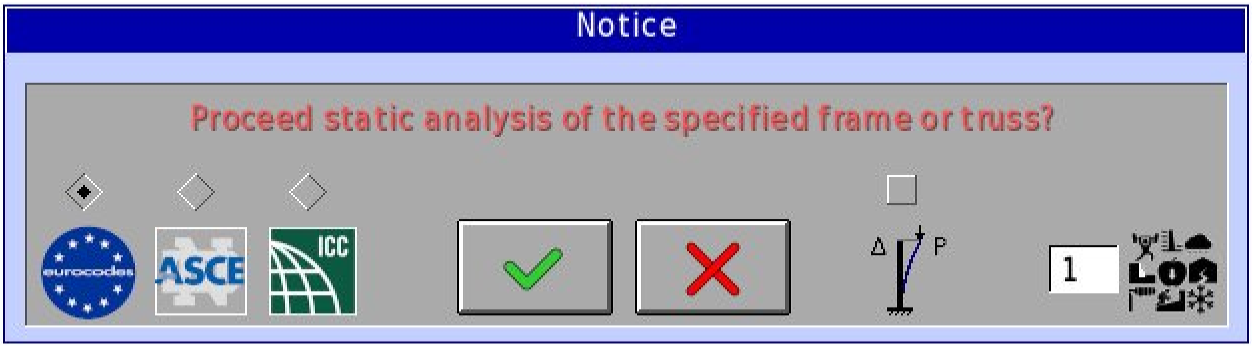
As it results from the load diagram, the beam is acted on by the load declared as LL1, so the change of factors will be taken into account. The principle of establishing the factors is very simple: each of the load characteristics gets the default values of the factors, and then the ones specified for the given variant, in this case variant 1 of the LL load character are changed.
%DL1 redefines the coefficients γ=1.35 and ξ=0.85 (these coefficients will be described later when discussing combinations for each standard).
However, this does not apply to the beam being solved, because there are no loads marked as DL1, rather DL which will get default coefficients.
Including descriptions in the calculation block that do not apply to existing loads is not treated as an error, because it may apply to other load combinations, e.g. invisible among the selected layers.

It is worth recalling that in order to use Greek letters, which are symbols of coefficients, as well as other numerical value symbols presented in technical drawings, in the case of AlfaCAD it is not necessary to install a Greek keyboard layout. When editing text, it is enough to use the Latin "equivalent" in combination with the left Alt key, and possibly Shift or Caps Lock for capital letters. For example, to obtain γ (gamma) you should select Alt-g, for Ψ (psi) Alt-U (Alt-Shift-u), "u", because "p" is reserved for π (pi).

Before starting the static analysis, let's make sure that all elements and loads are visible in the drawing, so all the necessary layers are turned on. In the Applications - Static Analysis function, let's select the analyzed block, i.e. all the structure elements, loads and descriptions.

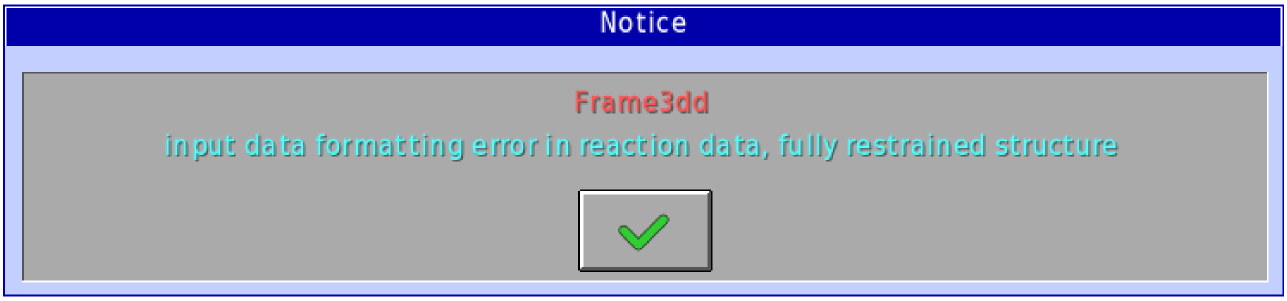


When everything necessary is specified by a block or added to a block, the Esc key that ends the selection leads to the display of the static analysis dialog box:



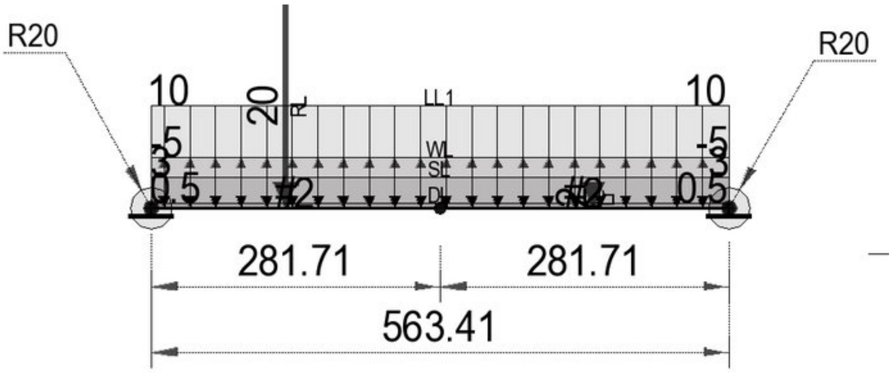
Let's assume the Eurocode standard, load combination number 1 and ignore **ΔP** for now.

Clicking the OK button starts the static calculations of the given structure with the given loads. But here's an "expected surprise":

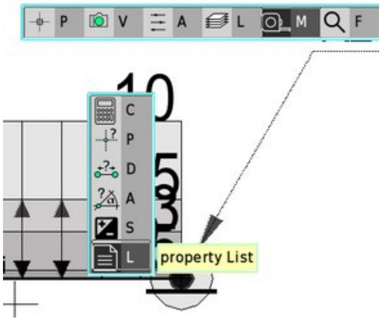


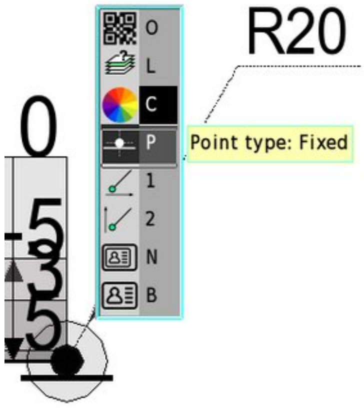

```
"input data formatting error in reaction data, fully restrained
structure"
```

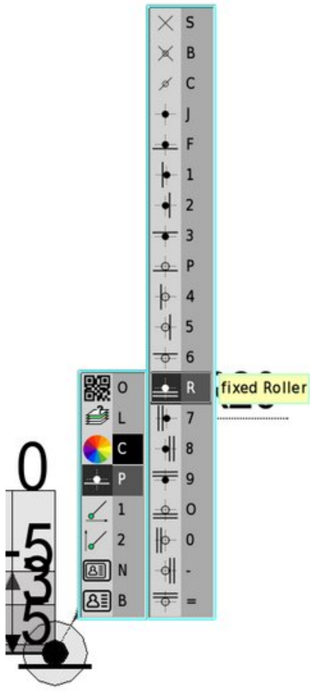
This is the expected behavior of the program, because the single element has been fixed on both sides with rigid non-sliding supports, so the program cannot create a stiffness matrix without an intermediate point on the beam's length. In such a case, the beam should be constructed as a composition of two elements with an intermediate node, e.g.:



Before we solve this composite beam, let's use a simpler solution and free one degree of freedom on one of the supports, replacing the fixed rigid support with a sliding rigid support. To do this, we can use the Property List function in the auxiliary menu:





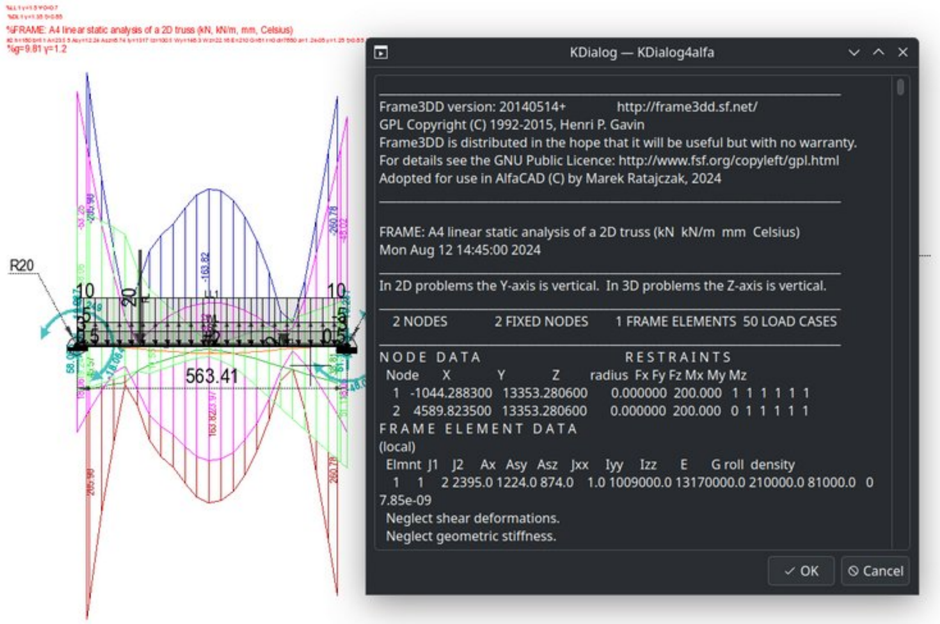


In the dialog box preceding the analysis, there is another switch concerning the inclusion of geometric stiffness in the calculations, also known as the second-order effect, or "Δ P".
Including the second-order effect excludes the use of the principle of superposition, i.e. summing the effects of each individual load to obtain the final results as an effect of the load combinations.

Hence, regardless of the choice of this option, in order to unify the entire algorithm, each combination is solved independently, which may result in the need to perform up to 51 calculations in the case of Eurocode and a similar number in the case of ASCE and ICC codes, i.e. obtaining up to 51 results, from which the maximum internal forces, stresses or deflections, as well as nodal displacements and reactions in support nodes are searched for each element.
The effects are not therefore summed up, but sorted in order to find the maximum effect of any combination in each cross-section of the structure elements, divided into critical states, both ULS ("Ultimate Limit State") for obtaining maximum forces and stresses, and SLS ("Serviceability Limit State") and QPSLS ("Quasi Permanent Serviceability Limit State"). This does not mean that the static analysis program (module) must be engaged 51 times to solve the task. This is done once, but for different load cases, because in each case the structure stiffness matrix remains unchanged, only the loads directly on the nodes or trough elements transferred to the structure nodes are changing.

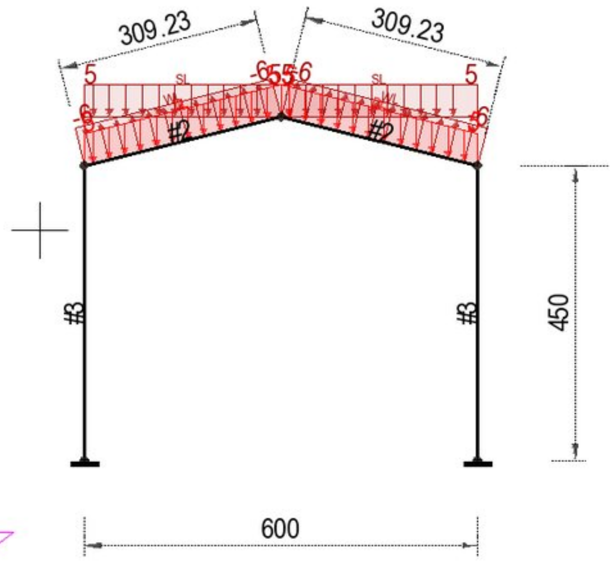
After changing the support method, we can call the functions again. This time everything is OK and we get the results, both in graphical form and in a form of the text report.

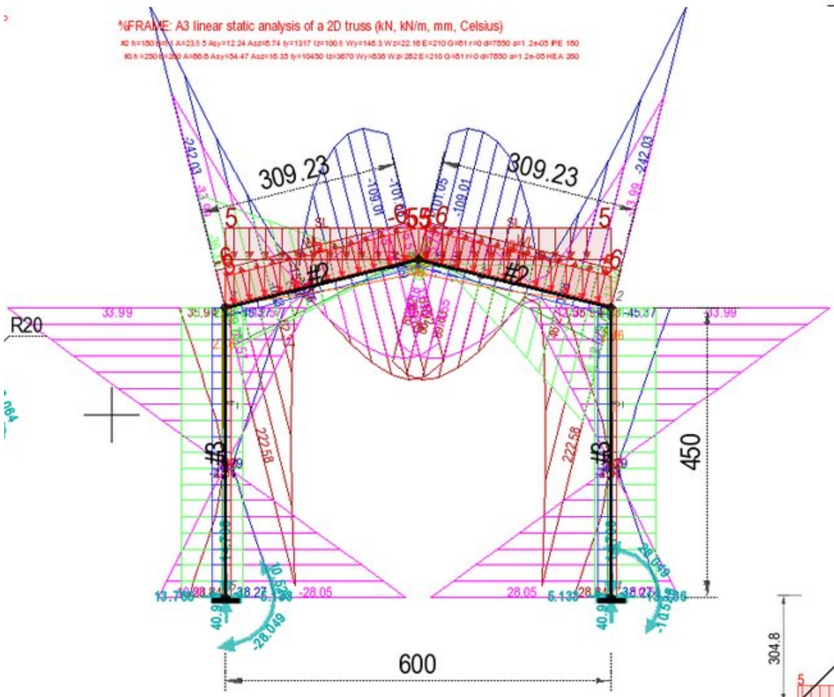
The results are saved in a default file, so if you want to save the report for later, you should save a copy of it to an archive file (Windows) or press the OK button and then select the name of the archive file in which a copy of the report will be made (Linux).



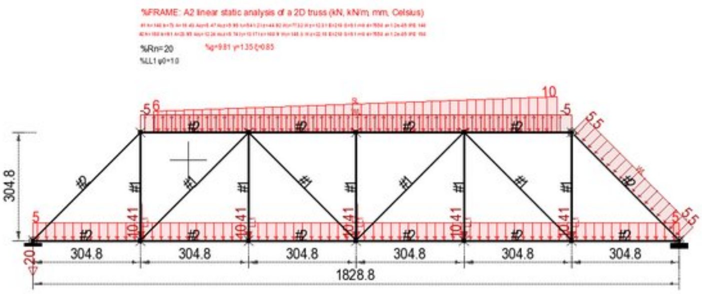
Pressing the Cancel key closes the window without asking for the archive file name, although the resulting file named alfacad.out remains in the AlfaCAD main directory, which always allows you to make a copy of it.
In the drawing we also have a simple frame marked as FRAME: A3

Proceeding in the same way we obtain the results for the frame. In this case the radius of the nodes was ignored. Here are the results:

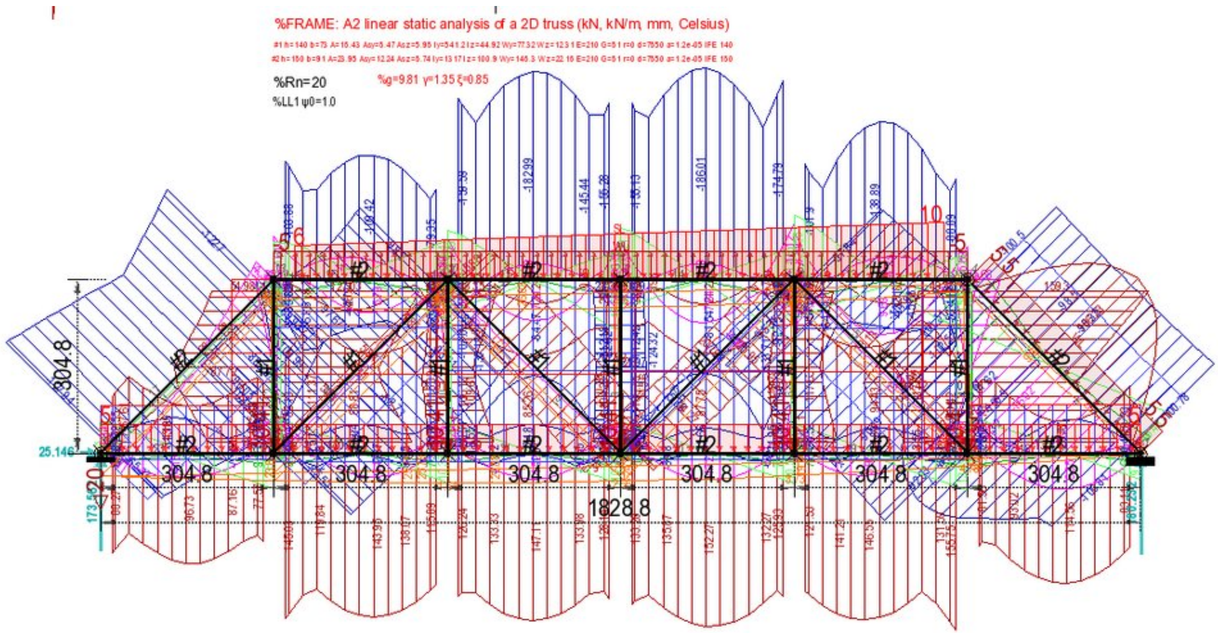




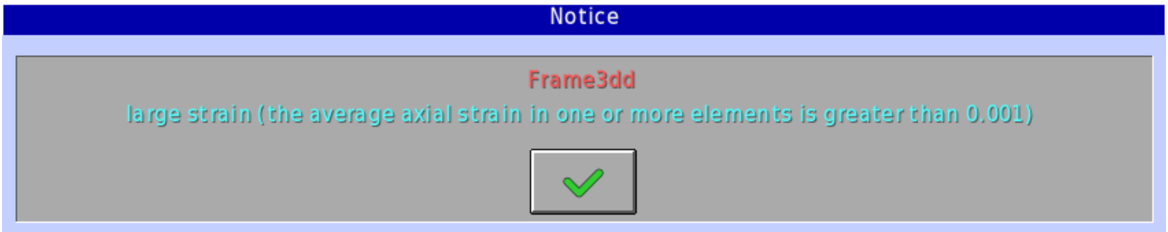
Let's do the same with truss A2:



Here are the results:



It should be mentioned that for the A2 truss, the static analysis program sent the following message:




```
large strain (average axial strain in one or more elements is
greater than 0.001)
```

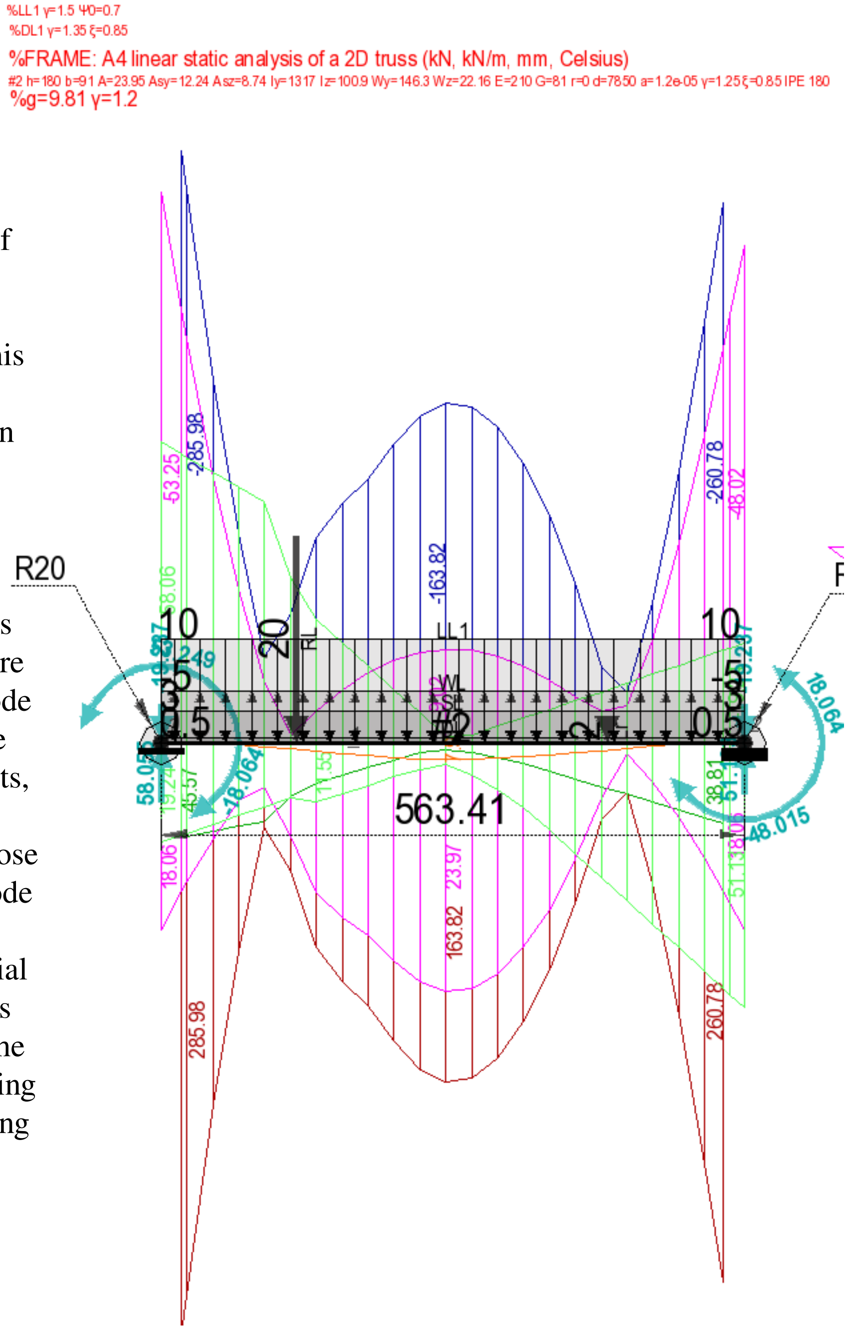
This is not a critical error, so the program has performed the calculations, although the message suggests that the axial deformation is relatively large, which the designer should pay attention to. The fact is that the truss was subjected to a random load, far from the actual cases, hence the somewhat random calculation results. The truss was also subjected to a 20 mm forced displacement of the support, although it is a hinged support, which does not generally lead to the creation of internal forces in a situation where the second support is also hinged, but generally leads to large displacements of the nodes. It should be noted that the displacements of the supports are not treated as loads (although they are often the cause of the creation of additional forces and therefore stresses), but in the combination of loads they are always taken into account, without load factors and without simultaneity factors. It is assumed that the given displacement of the support will be 20 mm as designed and not 20 mm x 1.35 or so.

The results presented in the form of force diagrams seem initially extremely complex and give the impression of an excess of information contained in them. This is just an impression, because it is very easy to control the readability of the diagrams. The overall report of calculation results is presented in the form of blocks, where each block contains different information, what is more, the blocks are placed on layers appropriate for a given state, such as ULS, sls mentioned earlier (for the sls designation, lower case letters are intentionally used, because in this state smaller loads are taken into account compared to the ULS state), qpsls, and additionally ULSLC (this is the ULS state but in a load combination, hence LC - "Load Combination"), slsLC and qpslsLC. The presented diagrams contain only single blocks, because only internal forces, stresses and reactions calculated as the maximum effect of the load combination in the ULS state (i.e. the ULSLC state) have practical design significance, and for deflections and displacements the effect of the load combination in the SLS state, i.e. slsLC, is crucial. The remaining blocks of internal force diagrams, as a simple summary effect of the load action in the ULS state or the effect of displacements in the SLS state, are not reliable, because they do not take into account the fact that some of the loads may act opposite to the action of the other loads, which is the case, for example, in the case of wind load, but also other loads in the case of complex structures. In order for the designer to compare the results of the calculations of all loads with the results of the combination action, these results are saved, but left either on invisible layers, or the diagram blocks remain invisible, until their attributes are changed, when necessary.

Interpretation of the results

Let's go back to the simple beam A4, because it is easier to demonstrate the effect of a given node radius using that example.

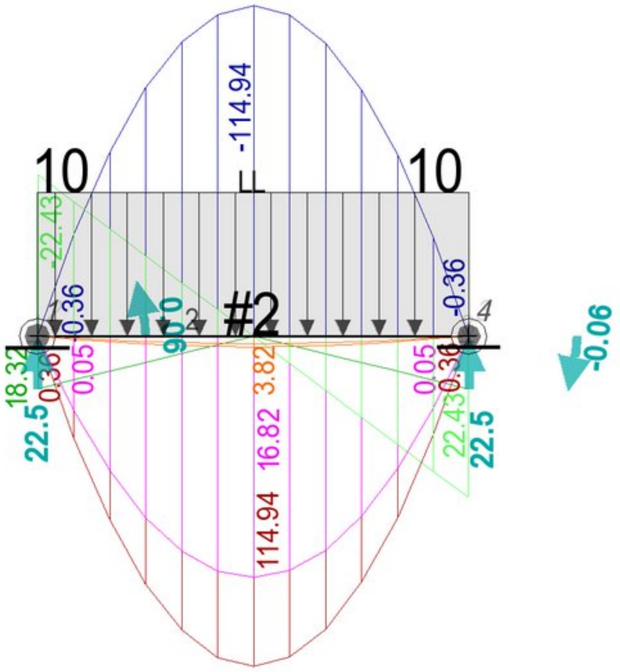
Once again, the internal force diagrams:



The beam is loaded with a set of loads including wind load, directed opposite to the action of other loads. The bending moment diagram shows the occurrence of load combinations in which the bending moment in the beam span and on the supports changes sign to the opposite. This is the effect of the WL load. The stress diagram on both sides of the cross-section is identical, because there are no axial forces from the given loads in the beam, hence there are no forces causing differences in stress on both sides of the cross-section. The stresses on the sections of the beam covered by the node radius are not given, because the structure of the node is unknown, what the cross-section of the gusset plates is and its geometric moments, but certainly greater stresses in the beam cross-section itself will not occur than those noted in the cross-sections outside the node radius. For design purposes of the node itself, all internal forces are given, i.e. axial forces, shear forces and bending moments to the geometric center of the node, i.e. the point where the theoretical node connecting the elements lies, or in this case connecting the element with the supports.

It is the designer's responsibility to assess whether the obtained stresses expressed in MPa or ksi are not exceeded (1 MPa = 0.145038 ksi, although the program automatically assumes units depending on the adopted units of measurement, i.e. mm, cm, m, km for the metric system, and inches, feet, yards and miles for the imperial system).

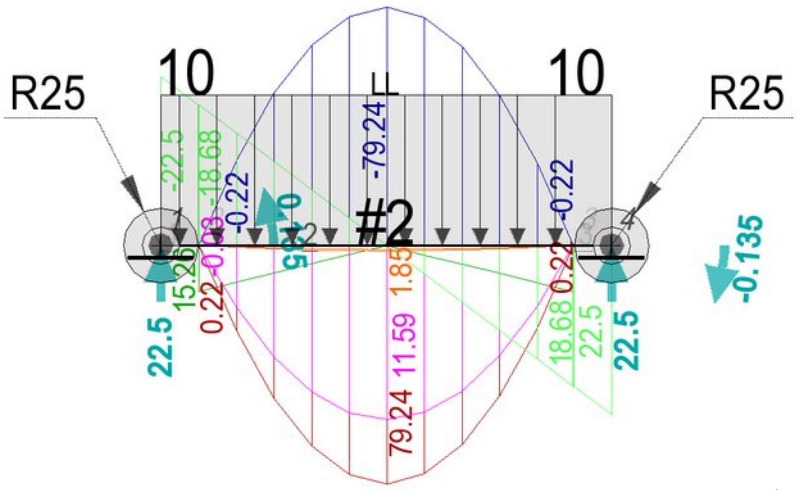
In the context of such a simple example of a beam, it is worth mentioning the subject of pinned connections and node radii once again. Here is a simple beam as a hinged-hinged element, supported by rigid non-sliding supports, loaded with a single uniformly distributed load. As we know from previous descriptions (Supplement and Supplement 2), the connections of all elements entering the stiffness matrix are rigid. To obtain the effect of a hinged connection between elements, a short section is generated with virtually zero length (the length cannot be zero due to the consistency of the system of equations) and with virtually zero bending stiffness (however, it cannot be zero, so it obtains small values). Let's solve this beam and discuss the results. The presented results come from the ULSLC load combination, hence the load factor γ=1.5 is taken into account, i.e. the sum of the loads 10 kN/m x 3 m x 1.5 = 45 kN accompanied by the design support reactions 2 x 22.5 kN = 45 kN, so fair enough.



As can be easily recognized by the support symbols, the node disk touches the support line, which identifies it as a rigid support, when the element line touches the circles symbolizing the pinned node at both ends. So we are dealing with rigid supports but pinned beam connections. This is a test example for the analysis of results, hence this solution was adopted for comparison with another beam scheme. As we know, 4 nodes and 3 elements were generated. The two outermost elements are virtual elements with virtually zero bending stiffness. The middle element, with a length slightly reduced by twice the length of the virtual element, has a given stiffness corresponding to property #2.

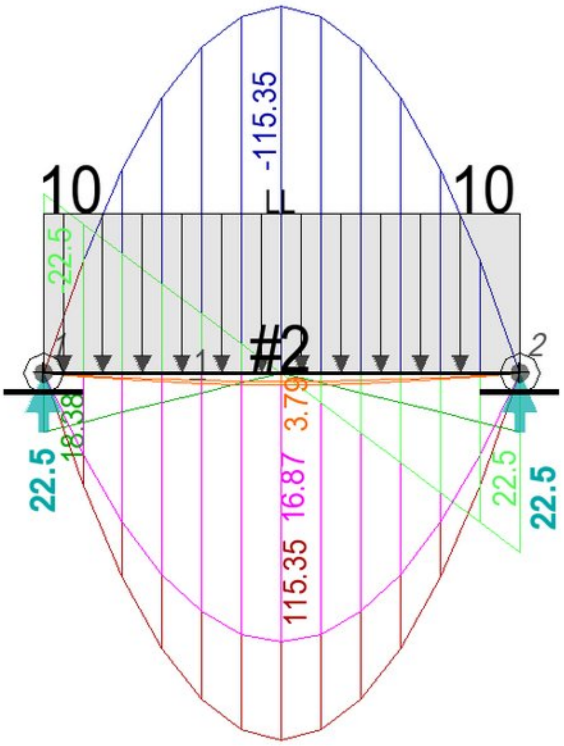
The calculation results show virtually zero, but in fact non-zero support moments, which results from virtually zero stiffness, in fact non-zero stiffness of the edge elements with virtually zero, in fact non-zero length. These moments can be ignored.
Let us now introduce a non-zero node radius value of 25 cm (250 mm), which for a beam span of 300 cm represents an almost 17% reduction in the beam span between node boundaries.

When non-zero radii of adjacent nodes are given for a hinged-hinged element, these virtual elements take on a length equal to the radius of the node increased by a virtually zero value, in reality non-zero in order to obtain the most error-free results of internal forces. These virtual elements enter the stiffness matrix in which the nodes are given infinite stiffness, hence the effective length of an element with virtually zero stiffness is virtually zero length, but lies outside the radius of the node.



As a result, the bending moments at the node radius boundary reach zero values.
This connection can therefore be interpreted as a hinge at the node radius boundary. The remaining internal forces remain non-zero, and for such a node, the axial and shear force diagrams also include the node radius, allowing its design. The stresses within the node radius are not given, because they are not applicable, due to the unknown cross-section of the node. As can be seen in the figure, a virtually zero, although numerically non-zero support moment appears in the calculation results, the reason for which is the virtually zero, although numerically non-zero value of the stiffness of the virtual element. For testing purposes at this stage of development of the static analysis module, AlfaCAD does not filter the results to reject this support moment as negligible, leaving such a decision to the designer. In some situations, due to checking the balance of moments and forces, knowledge of all values may be advisable. The program only ignores support reactions whose value is below the given accuracy, hence if in the first example the reaction accuracy value was given as 0.1, reactions with values of 0.06 and -0.06 kN would be ignored.

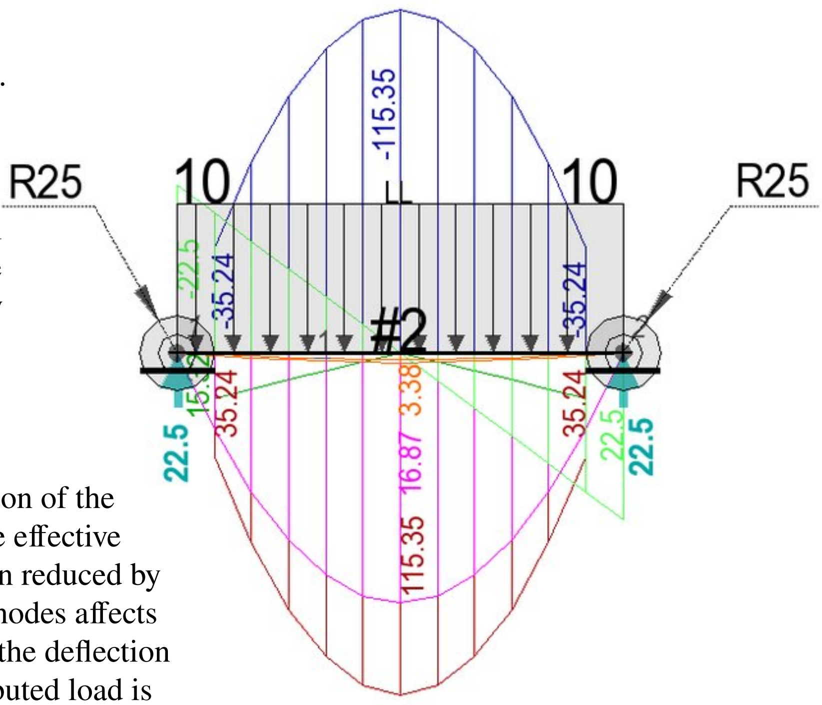
Of course, in the case of a single-span beam, a much better solution is to adopt a rigid-rigid element supported by two pinned supports, which greatly simplifies the interpretation of results. Here is the improved beam diagram with the results:



In the diagram, the beam line is stretched between the centers of the disks (filled circles) symbolizing rigid ends, when the circles of the support symbols touch the lines of the support symbols. This clearly indicates that the element is rigid-rigid and the supports are articulated. Both supports are immovable, but due to the release of at least one degree of freedom of the support system (in reality two), i.e. the degree of freedom of rotation around the Z axis (perpendicular to the drawing, i.e. rotation in the XY plane), the task is solvable, in contrast to the situation if the element was rigid-rigid and both supports were rigid, which was discussed earlier.
The calculation results are slightly different from the results for the hinged-hinged element, but in the case of the maximum moment in the span it does not exceed 0.3% and results from the lack of virtual elements simulating the hinged connection with the support. The support itself is a hinge. Support moments do not occur.

Let's modify the diagram by introducing node radii of 25 cm (250 mm).

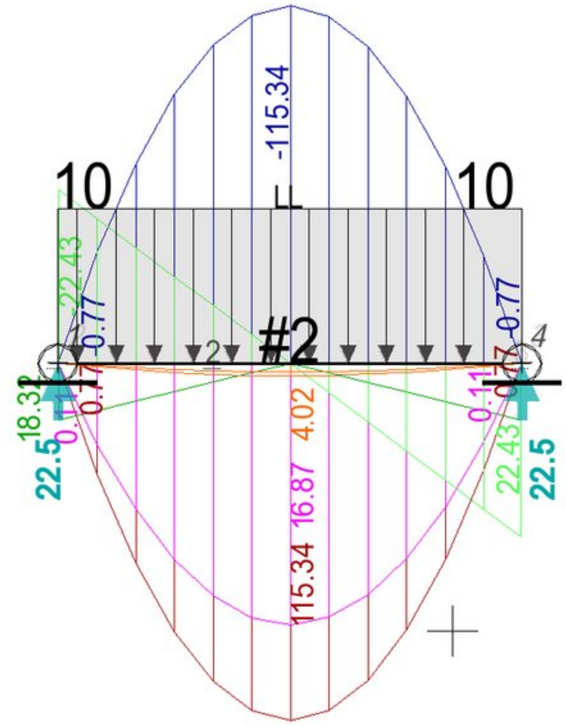
The lack of change in the value of internal forces and therefore stresses is not surprising. The ends of a rigid-rigid beam rotate on supports regardless of the size of the node radius, and therefore the span moments remain unchanged. The same applies to axial and transverse forces. Due to the fact that the cross-section of the nodes is not known, only their radius, stresses are not given in the length of the beam within the radius of the node, which is absolutely justified.



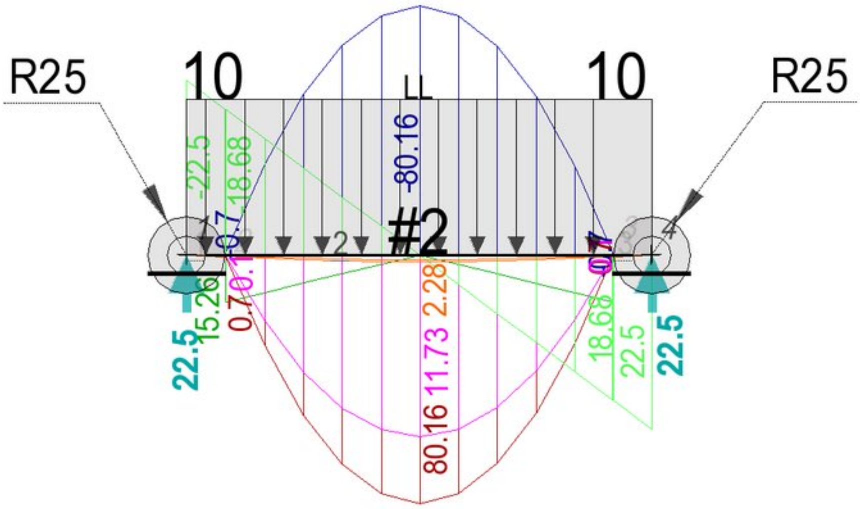
The only change is a reduction in the deflection of the element by about 10% due to the fact that the effective length of the element between nodes has been reduced by about 17%, and although the rotation of the nodes affects the displacement of the beam's center point, the deflection of the beam under the influence of the distributed load is smaller.

Non-zero values of the node radii do not only affect the stiffness matrix and thus the displacements of the nodes under the influence of the load, but also the deformations of the element along the length between the nodes, if the element is loaded.

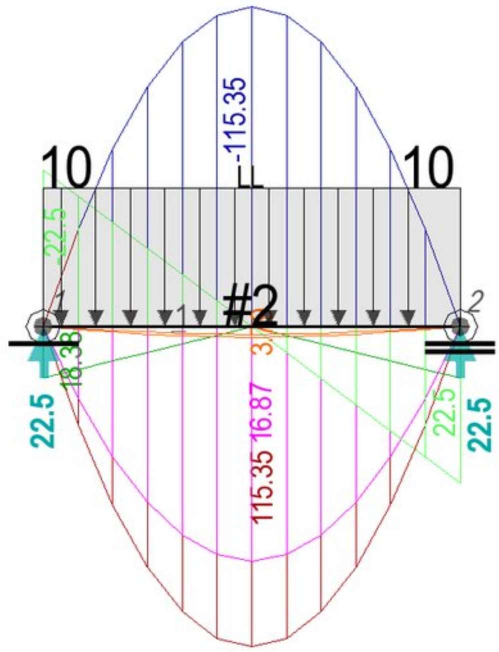
For a complete analysis of the case, let’s consider the situation where the beam is a hinged-hinged element and is supported by pinned supports (so like two pins in each node).



The results of the maximum values of internal forces are very similar to the results of the calculations of a rigid-rigid beam on hinged supports or a hinged-hinged beam on rigid supports, but not identical. Due to the release of degrees of freedom for the rotation of the support nodes, and additionally the creation of additional edge elements with virtually zero length and virtually zero stiffness, it causes the greatest deflection of the beam's center point, without practically changing the value of internal forces, with a negligible difference in maximum stresses. What is important, however, is the fact that such a system is still geometrically invariable and can be calculated. To complete the task, let us consider the hinged-hinged variant with hinged supports and declared node radii greater than zero:



As in the previous case, the force values are similar, although this time to the case with a hinged-hinged beam on rigid supports with given node radii of the same value (Fig. 2 of this chapter), with the difference that the possible rotation of the rigid nodes within their radii on the hinged supports increases the maximum displacement of the beam's center point by about 23%. However, in both cases described above, the structure remains geometrically invariable.

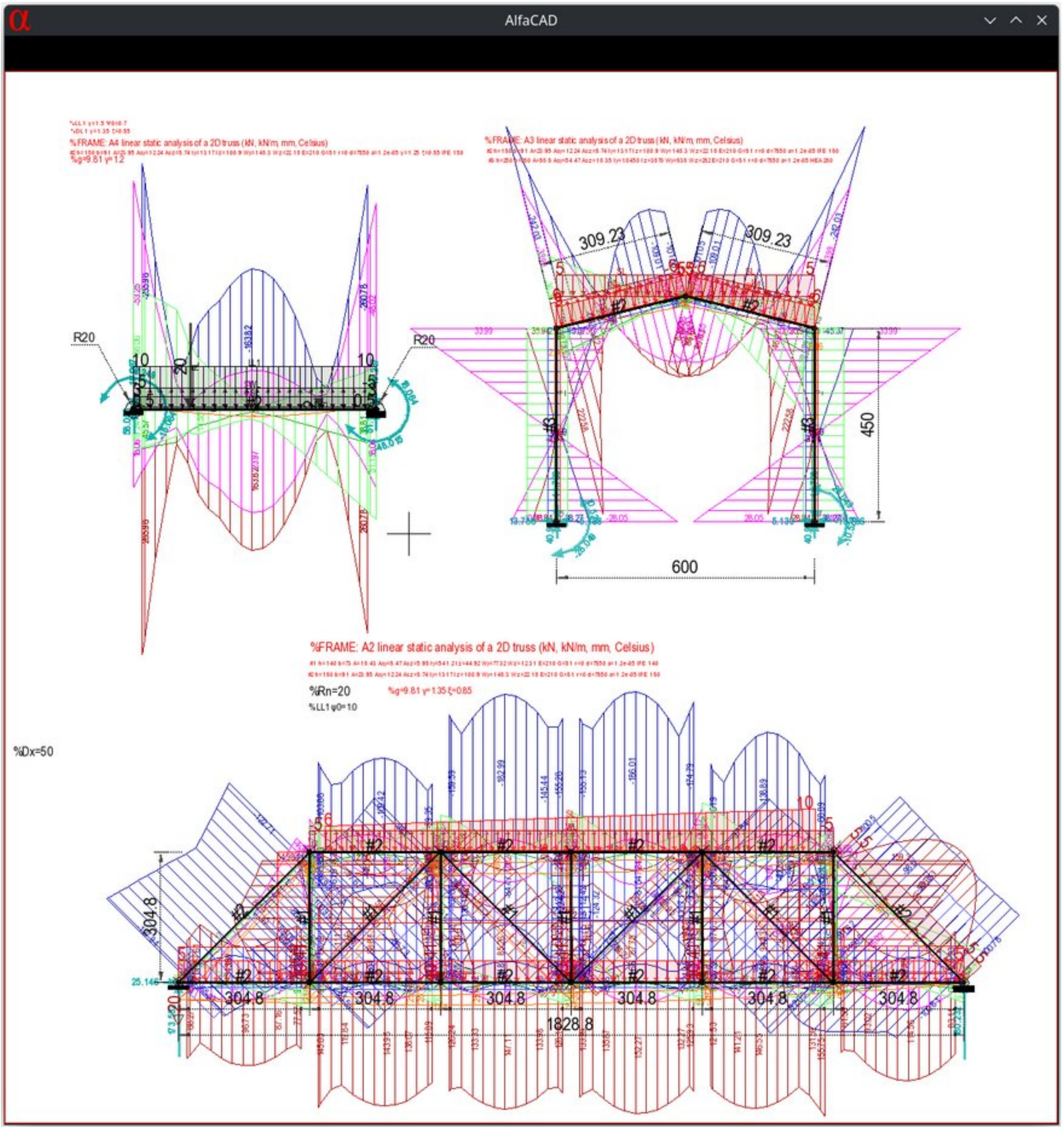


The last example shows a rigid-rigid beam on pinned supports, where one of the supports has a degree of freedom released for translation along the X axis. Compared to a similar case but without the possibility of translation (Fig. 3 of this chapter), the release of the degree of freedom does not affect the internal forces in any way.

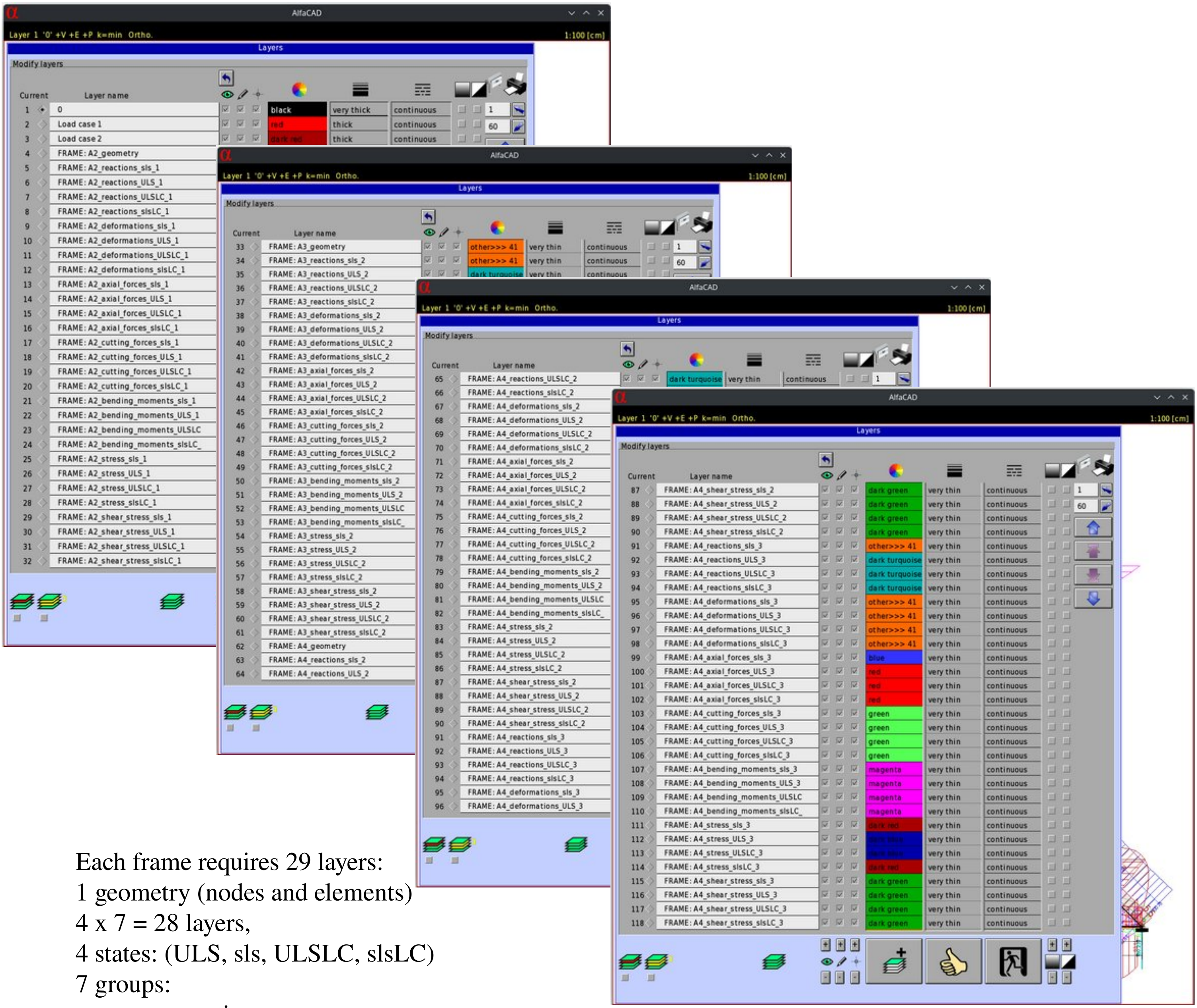
Layers and blocks in version 2.7

Version 2.7 introduced two major enhancements:
1) layers can be removed in their entirety
2) blocks can be given a visibility attribute, and some blocks created during static analysis get the flags of this attribute turned off, thus remaining invisible.

Here is the complete result of the calculations of the three frames



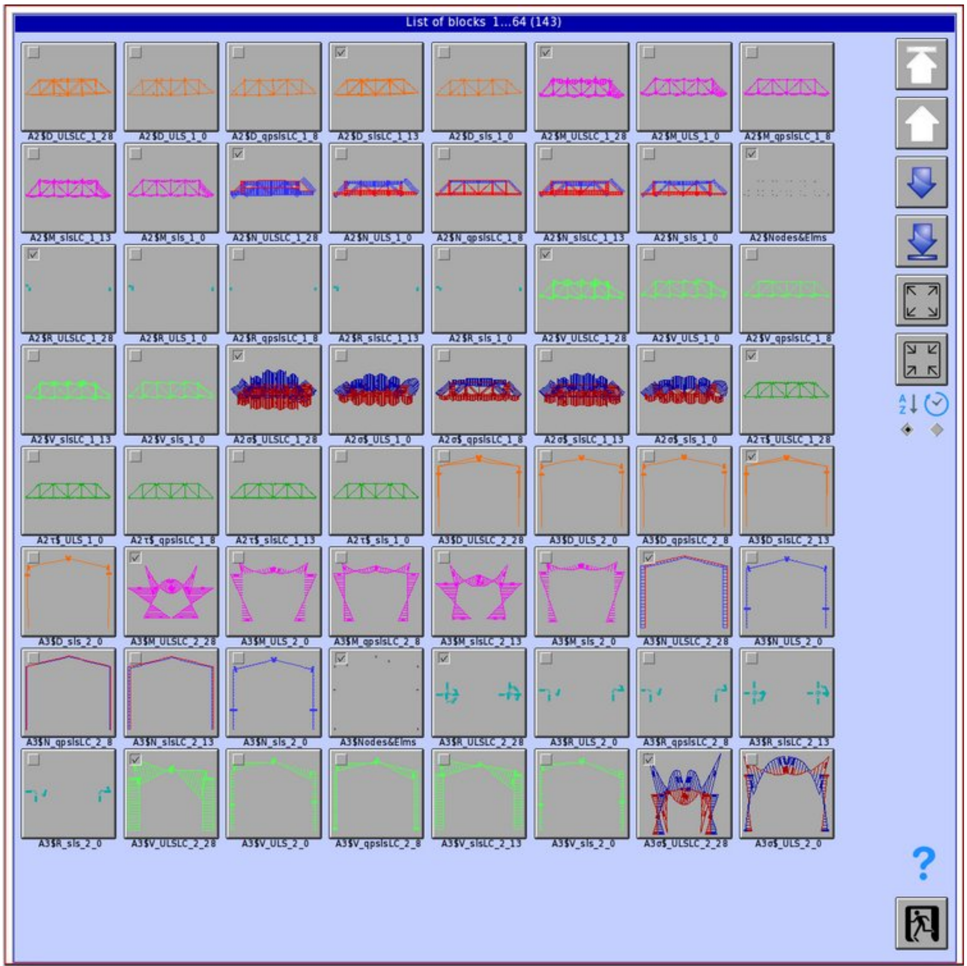
In the A2 truss, the right-most diagonal element was declared as a hinged-hinged element, hence the lack of negative bending moments in the nodes for this element. Additionally, despite the lack of vectors defining the node radii, in the frame description, or rather in one of the texts included in the analyzed block, there is a global description of the radii of all nodes in the form of text: %Rn=20 which defines the radii of all nodes for which the node radius was not declared individually as 20 cm (200 mm). If any of the nodes were described by the node radius vector, it would take the declared value. If not, it takes the global value. In the entire truss, the stress diagrams concern that part of the elements that lie outside the boundaries defined by the node radii. Here are the layers that were created for the calculation results:



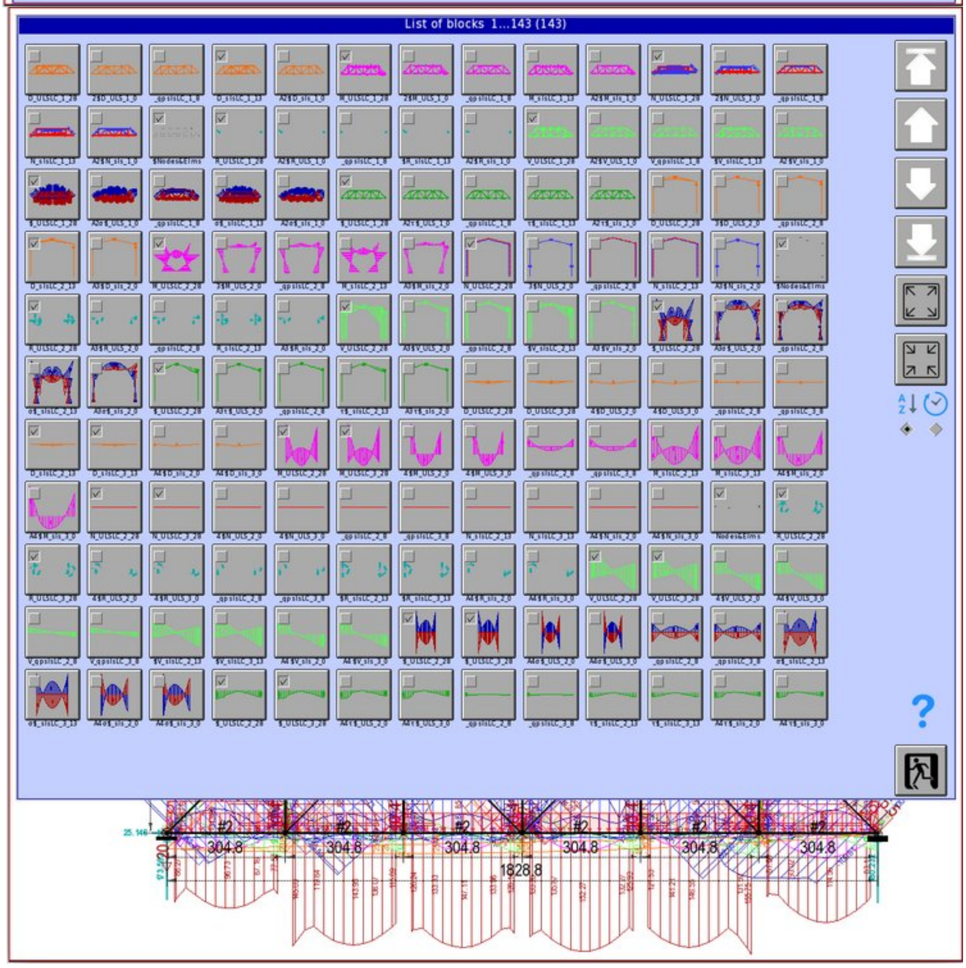
Each frame requires 29 layers:
1 geometry (nodes and elements)
4 x 7 = 28 layers,
4 states: (ULS, sls, ULSLC, slsLC)
7 groups:
- support reactions,
- deformations and displacements
- axial forces
- transverse forces
- moments
- stresses σ
- stresses τ

In the demonstrated example, an additional 115 layers were created, because the A4 frame was analyzed twice, once with the declared combination number 2, and the second time as combination 3, and the aim was to compare the results obtained in the Eurocode and ASCE standards, hence 4 x 29+3-1=118 (the geometry of the A4 frame is common to combinations 2 and 3, so it occupies 1 layer).

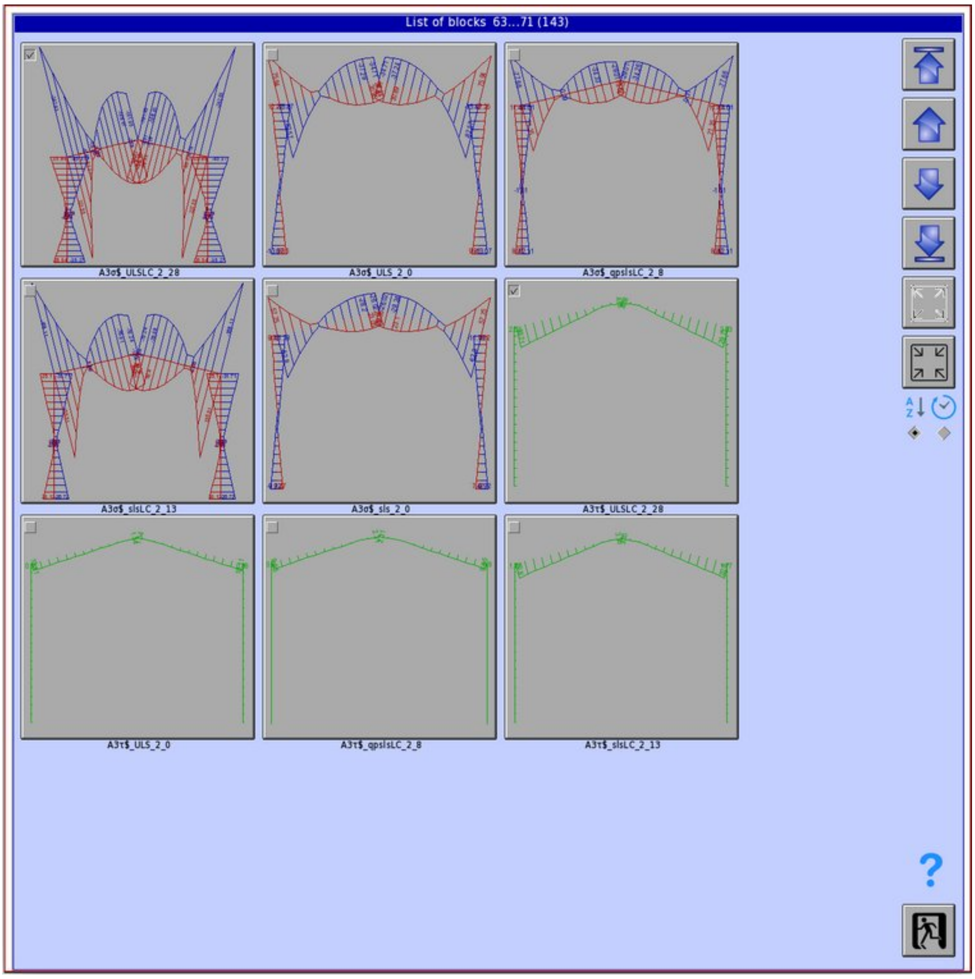
All layers are visible, although not everything on them remains visible. The secret lies in the attributes of the generated blocks. In the Block - List function you can display all the blocks that are in the drawing. If the block names are repeated and the option to group blocks with identical names is selected, they are displayed as a package. Clicking on the package opens the group with access to each of the blocks. In the case of result blocks of static analysis, the block names are not repeated:



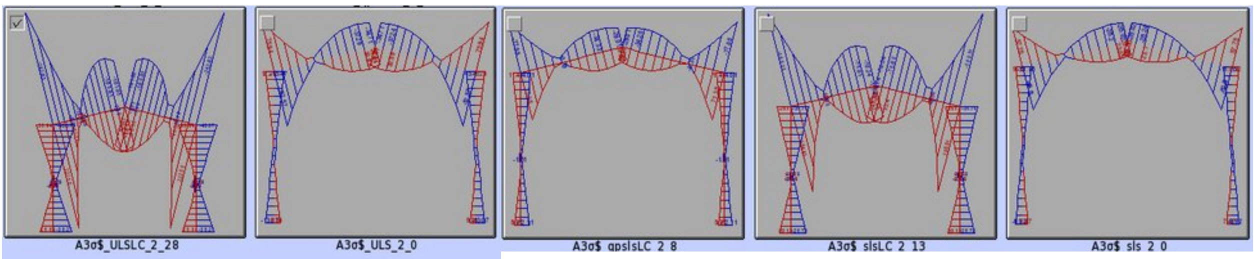
the size of the presented blocks can be reduced:



and significantly enlarged:

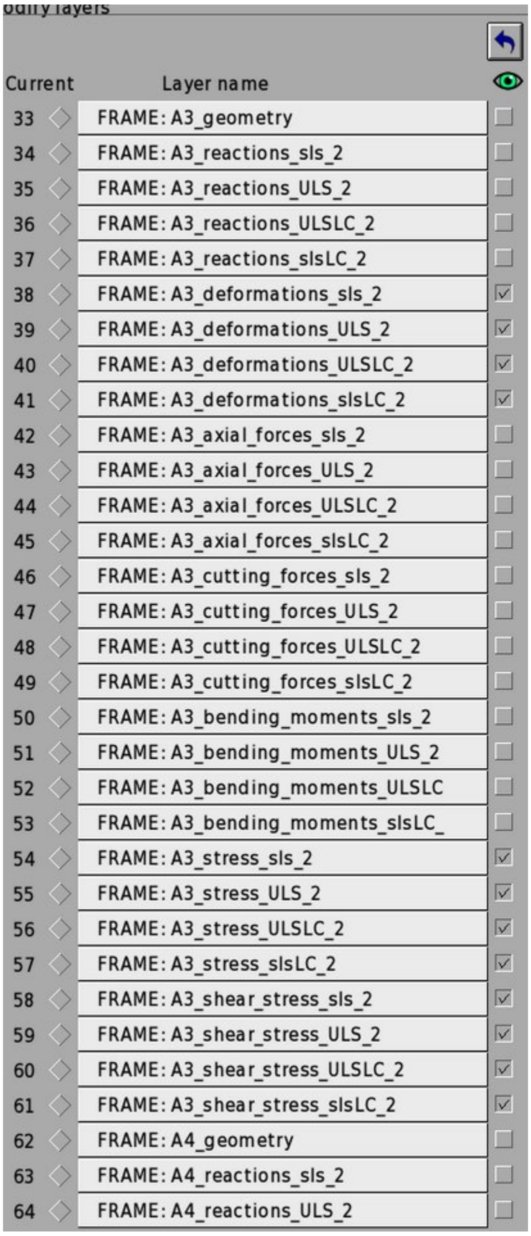


Let's pay attention to the blocks containing the frame designation A3 and the symbol σ, i.e. longitudinal stresses:

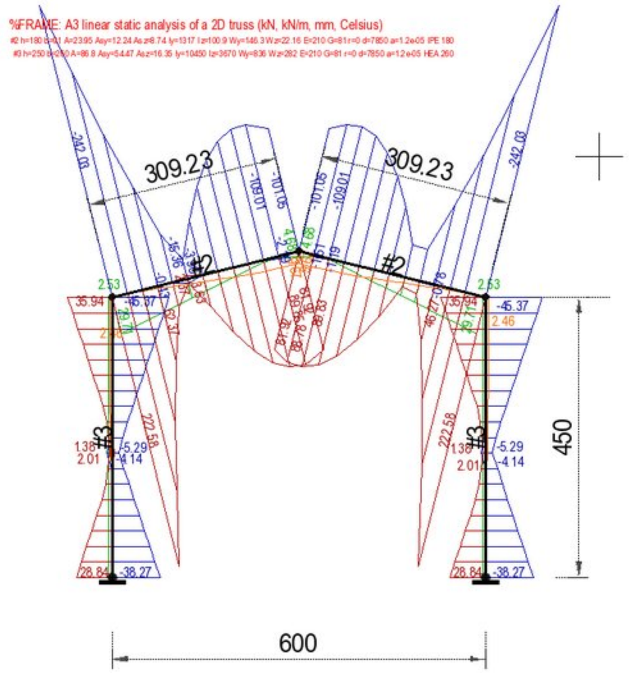


In the upper left corner of each block, the visibility attribute button is visible. Only the block named A3σ$_ULSLC_2_28 is visible. Deciphering the block name, we know that this is an A3 frame, longitudinal stresses (σ), ULS state in load combination (LC), given case 2, as a result out of 28 combinations. As you can see, the stresses in this block are the highest compared to other blocks (the presentation scale is dynamically selected to fit the entire block in a field of the same size, but it is enough to refer the size of the graphs to the frame geometry to see that the stresses in this block are the highest, which was rather expected, as a result of the most unfavorable load combination).

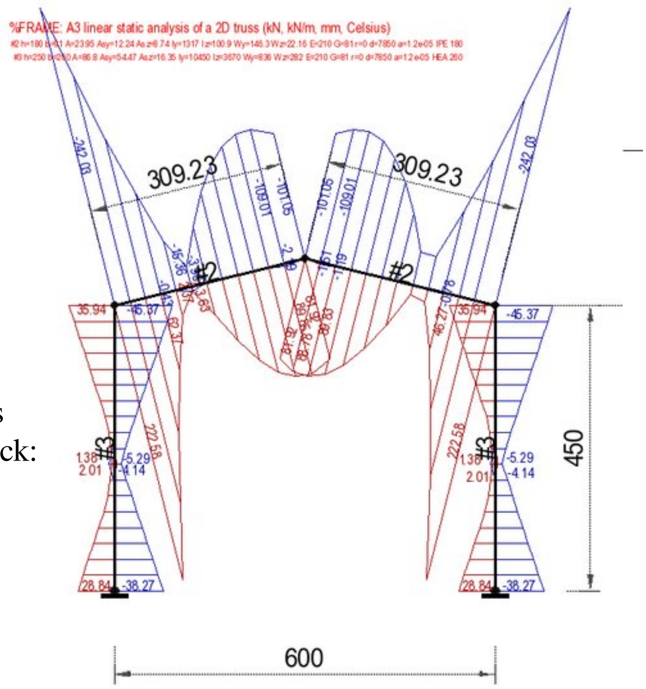
For the analysis of the results, let’s turn off all layers except layer 1 containing the A3 frame geometry, and the layers containing the deformation, sigma and tau stress blocks for the ULS, sls, ULSLC and slsLC states:



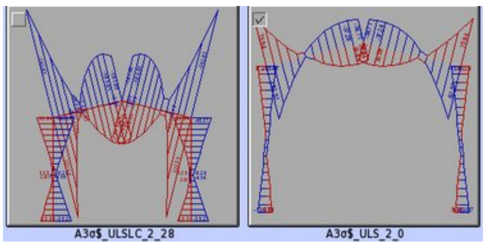
By excluding the remaining layers, no axial or shear forces, moments or support reactions are visible:



Let's go further and turn off the layers containing displacements and shear stresses (tau), leaving only the longitudinal stresses:

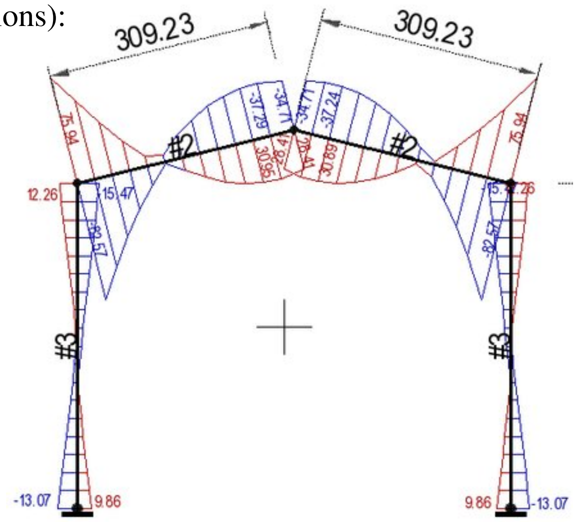


We only see stresses from the ULSLC state because the combination of loads in the ULS state is the basis for designing or checking cross-sections. In the block list, let's now swap the visibility of the ULSLC and ULS sigma block:



The previously visible block will remain invisible, in its place there will appear a block derived from the simple sum of ULS loads, the presence of which serves only the purposes of comparative analysis. Due to the fact that the entire procedure of calculating load combinations requires time to verify its correctness, on many examples, trivial and very complex, for analytical purposes the program was left to generate these blocks not only for the only practical states ULSLC and slsLC, but also for other states not taking into account load combinations. In subsequent versions there will be a possibility to abandon these blocks as well as entire layers.

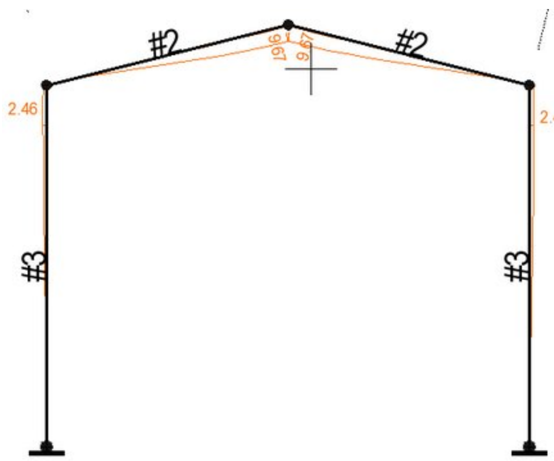
Here is the graph for the ULS state (without combinations):



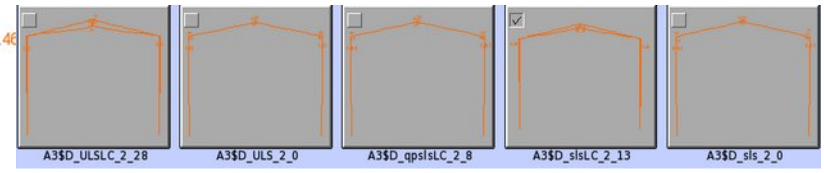
There is a significant reduction in the maximum stress values, which is understandable and expected, because a simple combination of loads with a wind load acting in the opposite direction to the other loads does not make sense, because it causes a reduction of internal forces and such a set of loads should not be solved at all. The result is given only for presentation purposes, indicating the need to take into account the combination of loads to avoid the need to calculate different load cases separately.

This is particularly important when taking into account the geometric stiffness in the calculations, for which the superposition of forces simply does not work.

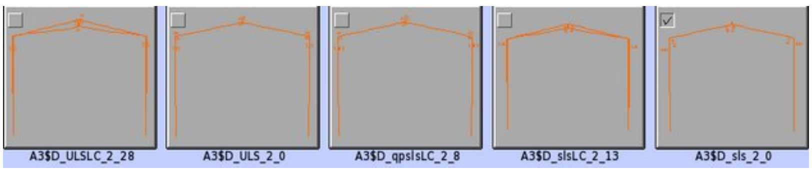
The diagrams of node displacements and element deformations shown in the diagrams are appropriate for the slsLC state layer, i.e. SLS in the load combination:



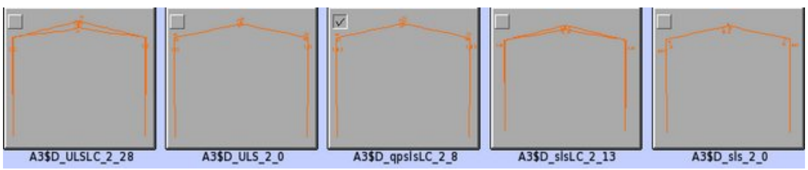
Here are the displacement graph blocks created for different states:



Only the block appropriate for the slsLC state is visible, the others are not. If instead of the slsLC block we enable the visibility of the block for the sls state"

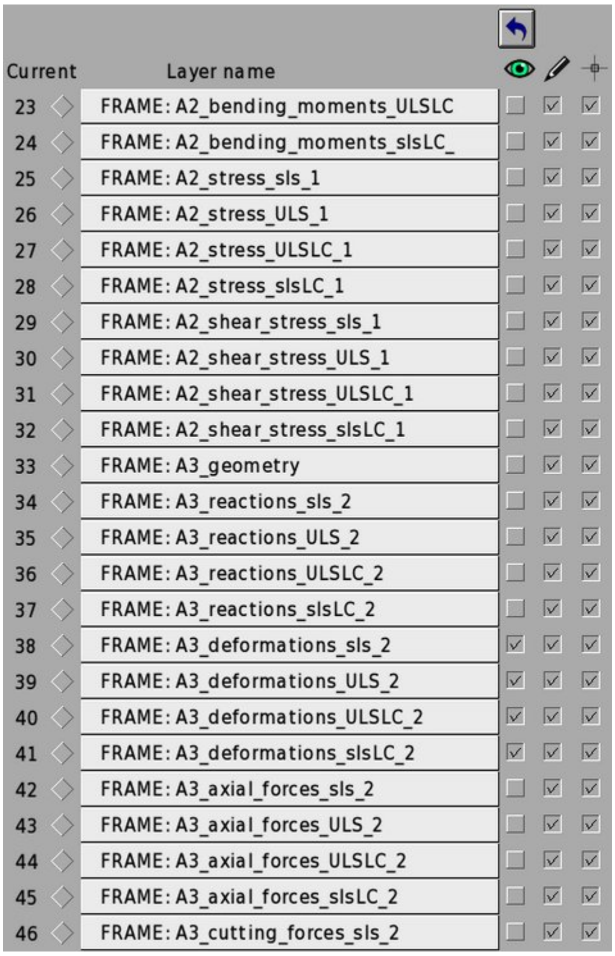


we will see much smaller displacements: They are not representative, because they do not take into account the fact that the wind is acting in a favorable direction, the effect of which is neglected in many load combinations. For the sake of demonstration, instead of sls, I will include the quasi-permanent load combination qpslsLC:

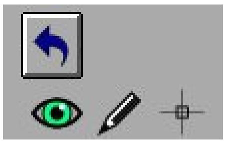


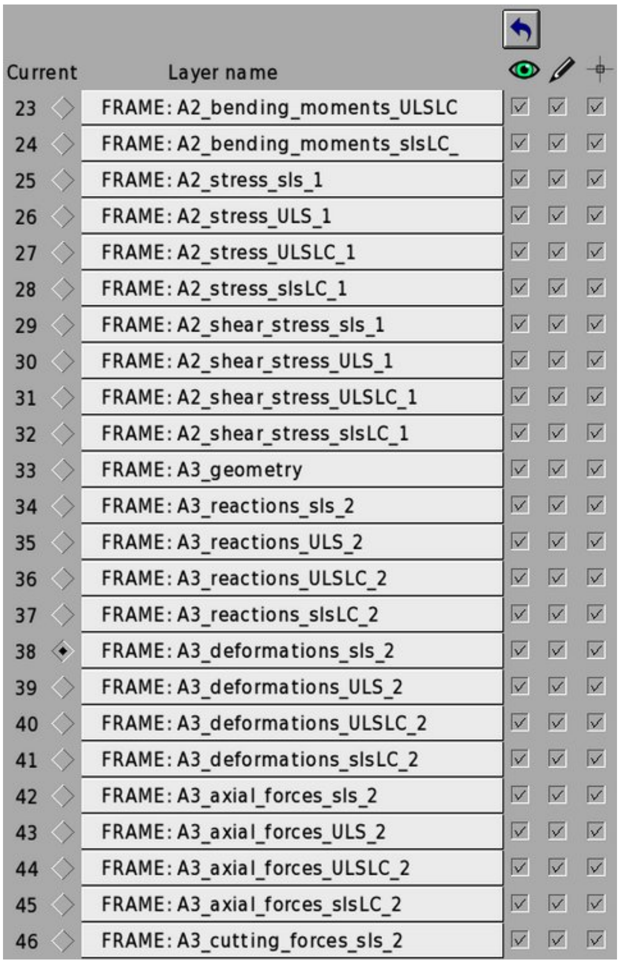
This combination could be taken into account for the assessment of permanent deformations, especially in the case of wooden structures.

Layer Operations. Removing Layers

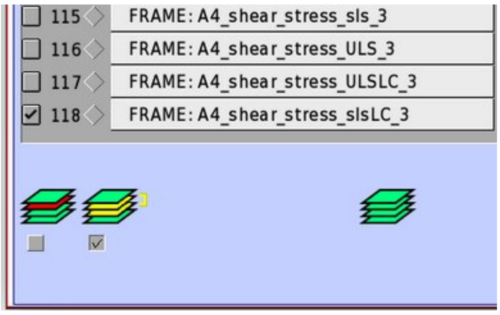


To show selected layers, we turned off all of them, then turned on only the selected ones. The previous state was remembered, so if after analyzing what we were looking for we want to go back to the previous state of layer settings, just use the "Back" key



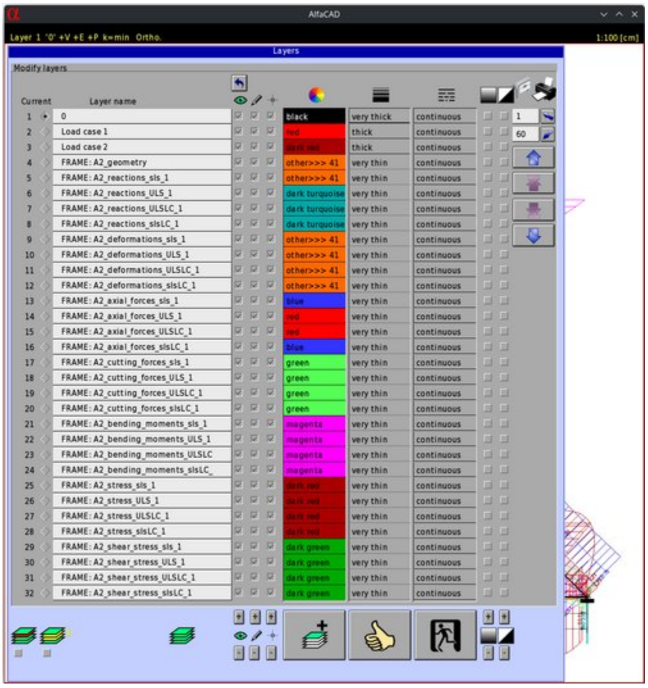


In version 2.7, the ability to delete layers was introduced, which is directly related to the automatic creation of layers by the static analysis module for generating blocks of forces, displacement and stress diagrams, as well as structure geometry with the numbering of nodes and elements that is done automatically. Additional selectors have appeared in the lower left corner of the layers dialog. The left one, Delete layer, allows you to activate the buttons for deleting a single layer, the right one - for marking a range of layers prepared for deletion.

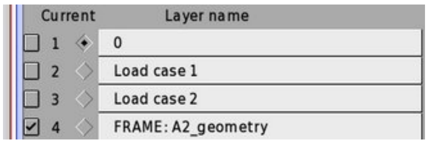


If the layer range selector is active, buttons appear next to the layer numbers to mark the beginning and end of the layer block to be deleted. The order of selection is not important, but only 2 markings can be made. Another one invalidates the selected block and starts marking another range. This range can include layers placed on different pages of the dialog box, and the arrow buttons are used to move between pages as usual.

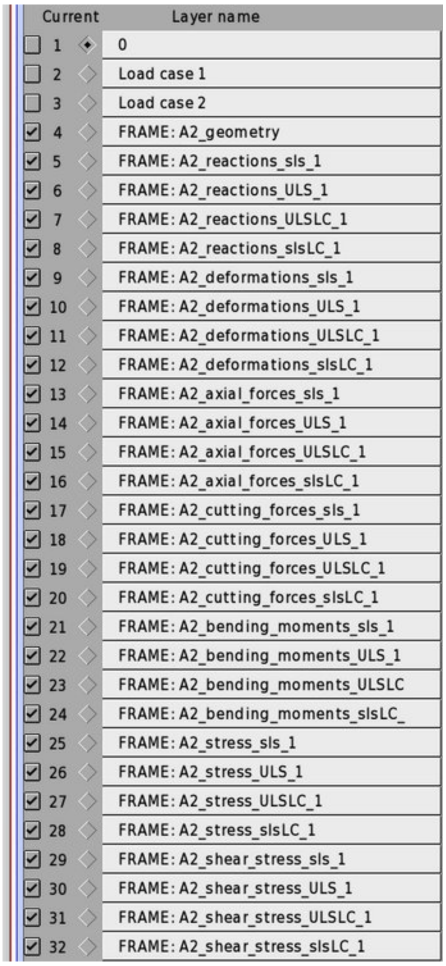
For example, if we want to remove all layers created by the static analysis module, let's mark the last layer as the start of the block, as shown in the figure above.



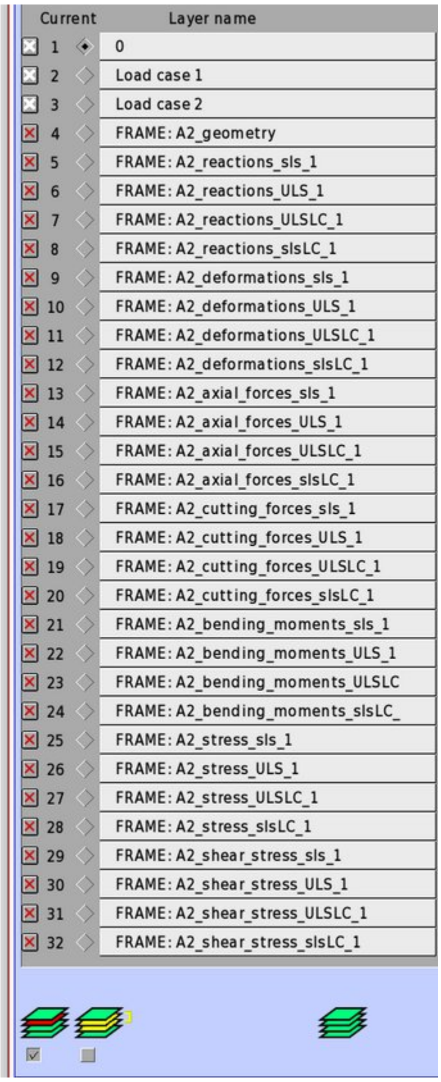
Let's go to the initial page and mark the fourth layer as the end of the block, leave the first 3 layers on which we drew the structure and loads.



We know, as you can see when scrolling through the pages, that all layers between the selected ones were generated by the program. When you select the end of the block, all layers between the beginning and the end will also be selected

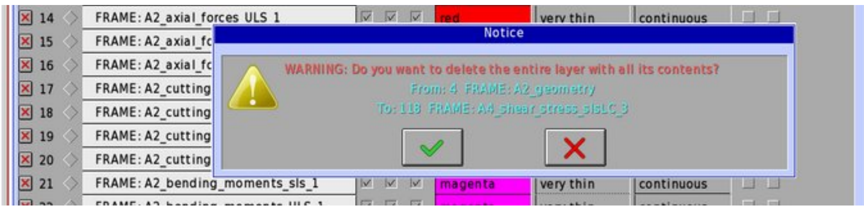


Now let's select the Delete Layers selector. All previously selected layers will get delete buttons:



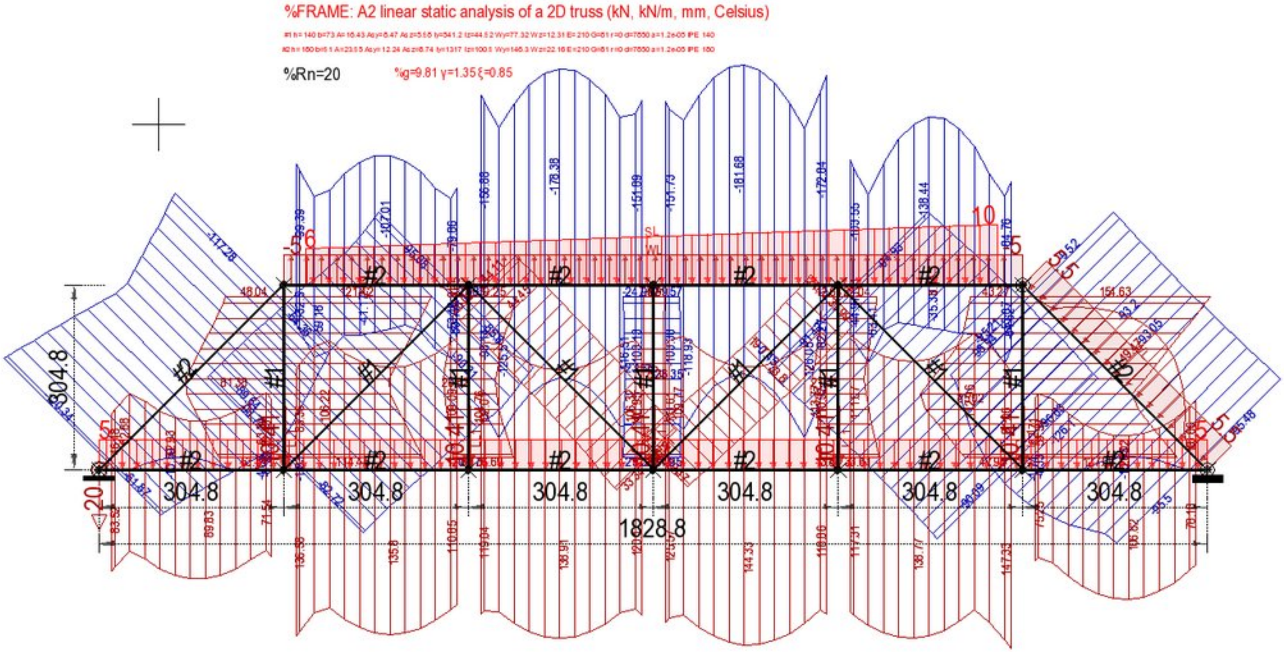
Note: deleting layers involves renumbering the remaining layers if their numbers were higher than the numbers of the deleted layers. This usually does not cause any additional consequences, but it should be remembered, for example, when creating software plug-ins. There is no need to delete layers in the case of recalculation of the same frame, e.g. after changes in the load set or modification of geometry. The program does not delete layers, only the blocks contained in it, leaving the layers in their places, filling them with new blocks.
Deleted layers will be re-created by program by next static analysis.

Pressing any of the active delete buttons allows you to delete all selected layers, which requires confirmation as everything on the layers will be permanently deleted.

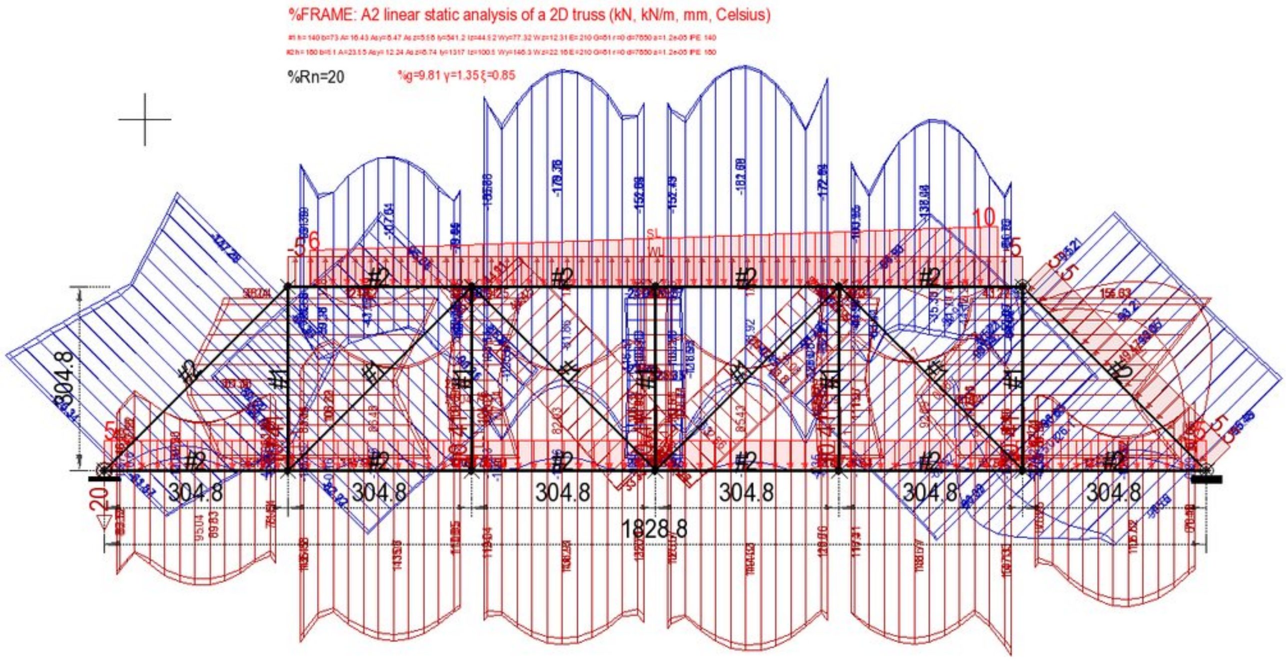


Comparative analysis of standards

For testing purpose, calculations were carried out using load combinations according to Eurocode, ASCE and ICC standards on the example of A2 truss. Here are the results of maximum longitudinal stresses on both sides of the cross-sections (only graphs on USLLC layer are visible), and a different combination number was assumed for each standard to create independent calculation results for each of them.
ASCE and ICC standards presented together:

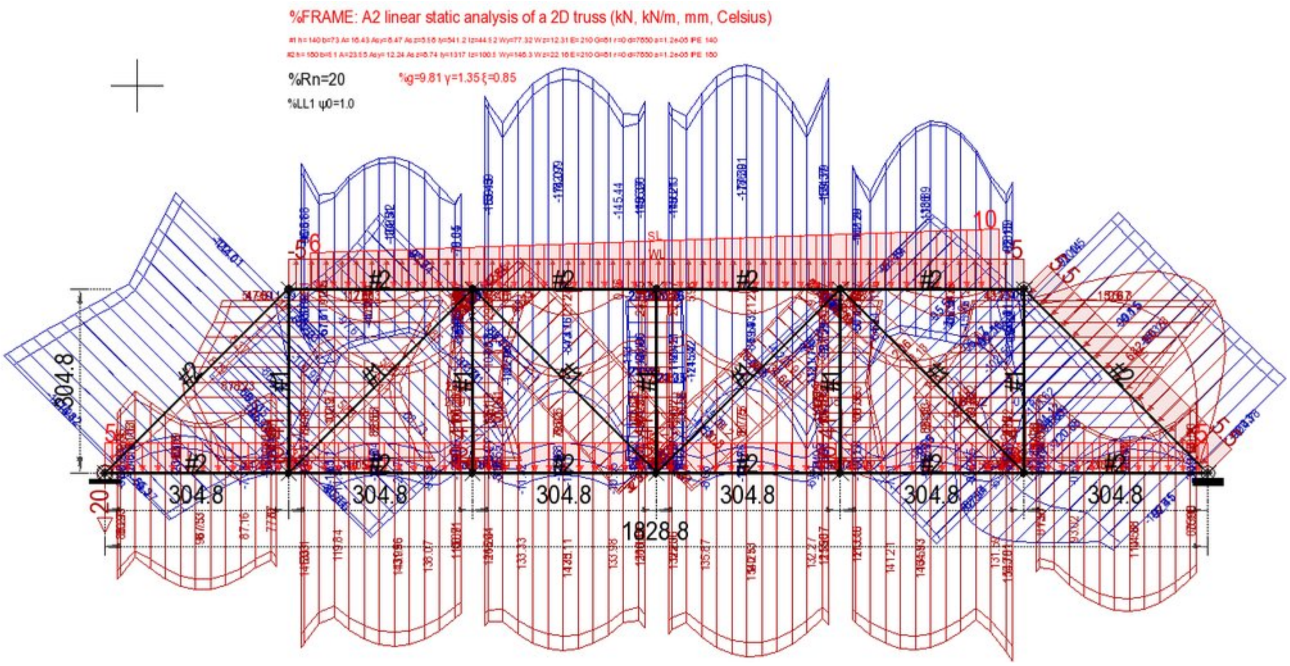


It can be seen that both the ASCE and ICC graphs are the same, which means that at least for loads such as DL, LL, WL and SL, the combinations of loads in both standards give the same results.
Here are the same stress graphs with the results according to Eurocode:



The results are similar, although for Eurocode the maximum stresses are slightly higher. This is due to the difference in the default partial load factors and the default load simultaneity factors.

A small experiment will allow the analysis of the effect of the load simultaneity factor for the variable load LL1, established for concentrated forces loading the nodes of the lower chord of the truss, as additional text has been introduced: %LL1 ψ0=1.0



Although the graphs according to the ASCE and ICC standards still match, the change in the load simultaneity factor for the LL1 characteristic caused not only higher stresses according to the Eurocode standard (which was expected), but also deepened the difference between the standards. This is due, firstly, to the fact that although the ASCE and ICC standards are consistent in terms of the principle of determining load combinations, with minor exceptions mainly concerning loads of the HL, FL and EL nature, they differ in terms of the principle of determining combinations with the Eurocode standard.

Load Combination Standards

This chapter is hellishly boring and for those who do not want to delve into the details of standards and their numerical solution I recommend skipping all in blue to read the next chapter.

The following presentation aims to familiarize with the applied unification that improves the automation of calculations, regardless of the chosen standard, to encourage discussion on this topic. In AlfaCAD, in order to standardize the algorithm, the symbols specific to the Eurocode standard, i.e. the coefficients:

γ Partial coefficient for permanent/variable loads from table A1.2(B) of the EN 1990 standard
γ.inf Partial coefficient for permanent loads (lower value) from table A1.2(B) of the EN 1990
Ψ0 Coefficient of the combination value of live load/snow/wind from table A1.1 of the EN 1990
Ψ1 Coefficient of the combination value of the frequently occurring live load/snow/wind from table A1.1 of the EN 1990
Ψ2 Coefficient of the combination value of the "quasi-permanent" live load/snow/wind from table A1.1 of the EN 1990 standard

have been adopted for the remaining standards, although their meaning may be inconsistent with the Eurocode standard, which will be demonstrated below to ensure which coefficient applies to which combinations.
Despite the lack of full compliance, such unification allows for a quick comparison of calculation results, because the meanings of the coefficients are similar.

Eurocode Standard

For Eurocode, the following combinations are considered among the ULSLC combinations:

ULSLC1=g*γ
ULSLC2=g*γ + q*γ
ULSLC3=g*γ + q*γ + s*γ*Ψ0
ULSLC4=g*γ + q*γ*Ψ0 + s*γ
ULSLC5=g*γ + q*γ*Ψ0 + r*γ
ULSLC6=g*γ + q*γ*Ψ0 + l*γ
ULSLC7=g*γ + q*γ + s*γ*Ψ0 + w*γ*Ψ0
ULSLC8=g*γ + q*γ + r*γ*Ψ0 + w*γ*Ψ0
ULSLC9=g*γ + q*γ + l*γ*Ψ0 + w*γ*Ψ0
ULSLC10=g*γ + q*γ*Ψ0 + s*γ + w*γ*Ψ0
ULSLC11=g*γ + q*γ*Ψ0 + r*γ + w*γ*Ψ0
ULSLC12=g*γ + q*γ*Ψ0 + l*γ + w*γ*Ψ0
ULSLC13=g*γ + q*γ*Ψ0 + s*γ*Ψ0 + w*γ
ULSLC14=g*γ + q*γ*Ψ0 + r*γ*Ψ0 + w*γ
ULSLC15=g*γ + q*γ*Ψ0 + l*γ*Ψ0 + w*γ
ULSLC16=g*γ + s*γ
ULSLC17=g*γ + r*γ
ULSLC18=g*γ + l*γ
ULSLC19=g*γ + w*γ
ULSLC20=g*γ + s*γ + w*γ*Ψ0
ULSLC21=g*γ + r*γ + w*γ*Ψ0
ULSLC22=g*γ + l*γ + w*γ*Ψ0
ULSLC23=g*γ + w*γ + s*γ*Ψ0
ULSLC24=g*γ + w*γ + r*γ*Ψ0
ULSLC25=g*γ + w*γ + l*γ*Ψ0
ULSLC26=g*γ + q*γ + w*γ*Ψ0
ULSLC27=g*γ + q*γ*Ψ0 + w*γ
ULSLC28=g*γ.inf + w*γ

where, due to the fact that the Eurocode does not specify loads, e.g. type HL or FL, the following was assumed:
g=DL+FL
q=LL+EL+HL+TL
s=SL
r=RL
w=WL
l=Lr

If we gave each of the load characters a number of which only the next bit is on, we would obtain the following set of "flags" for each of the loads:
DL=1
LL=2
Lr=4
WL=8
SL=16
EL=32 // equivalent to the value 2 in Eurocode due to the fact that there is no specification of EL

RL=64
HL=128 // equivalent to value 2
FL=256 // equivalent to value 1
TL=512 // equivalent to value 2
whereby due to the lack of EL, RL, HL, FL and TL specifications, it can be assumed that:
flag 1 includes flags 1+256 (load FL is treated as DL)
flag 2 includes flags 2+32+128+512 (loads EL, RL, HL, FL and TL are treated as LL), unless these characteristics obtain individually different load or simultaneity factors, although they will always be considered in combination as variable loads, except for FL which is a permanent load.
It should be noted that in specific cases, e.g. technological loads, when the load acts favorably for the structure's effort, such cases should be analyzed as separate load combinations, because the program does not check whether a given LL-type load acts favorably or not. If the LL load is taken into account in the load combination, then all components of this load are taken into account simultaneously.
This principle results not only from the fact that the final results of the combinations are not a superposition of the results for each of the loads, because this would sometimes lead to an extremely large number of partial calculations, but also from the fact that when taking into account the geometric stiffness (ΔP), the superposition principle does not apply.
Here is therefore a set of load combinations for the ULS condition, which is the basis for the design of cross-sections, given in the form of the flag table described above:

```
COMBINATION EUROCODE_ULSLC[]={
{1},{1+2},{1+2+16},{1+2+16},{1+2+64},{1+2+4},{1+2+16+8},{1+2+64+8},{1+2+4+8},
{1+2+16+8},{1+2+64+8},{1+2+4+8},{1+2+16+8},{1+2+64+8},{1+2+4+8},{1+16},{1+64},
{1+4},{1+8},{1+16+8},{1+64+8},{1+4+8},{1+8+16},{1+8+64},{1+8+4},{1+2+8},{1+2+8},
{1+8}};  //28 combinations
```

where each group contains a mask of combination load flags.

Here is the coefficient table for the Eurocode standard:

```
ST_LOAD_FACTORS st_load_factors_EU_0[]={
//load,variant,  γ,      Ψ0,      Ψ1,     Ψ2,         ξ,      γ.inf
{/*0*/ 0,0,      1.5,     0.7,     0.5,    0.3,        0,       0},
{/*DL*/1,0,      1.35,    1.0,     1.0,    1.0,        0.85,    0.9},
{/*LL*/2,0,      1.5,     0.7,     0.5,    0.3,        0,       0},
{/*Lr*/3,0,      1.5,     0.7,     0.5,    0.3,        0,       0},
{/*WL*/4,0,      1.5,     0.6,     0.2,    0,          0,       0},
{/*SL*/5,0,      1.5,     0.5,     0.2,    0,          0,       0},
{/*EL*/6,0,      1.5,     0.7,     0.5,    0.3,        0,       0},
{/*RL*/7,0,      1.5,     0.7,     0.5,    0.3,        0,       0},
{/*HL*/8,0,      1.5,     0.7,     0.5,    0.3,        0,       0},
{/*FL*/9,0,      1.35,    1.0,     1.0,    1.0,        0.85,    0.9},
{/*TL*/10,0,     1.5,     0.6,     0.5,    0.0,        0,       0}};
```

The first line should be ignored, the "variant" parameter is initialized to 0 and used as a working variable.
Each of these coefficients can be changed for each defined load variant.

Here is a similar set for Eurocode, but covering the loads in the SLS combination that is the basis for determining displacements and deformations:

```
COMBINATION EUROCODE_SLSLC[]={
{1},{1+2},{1+2+16},{1+2+8},{1+2+16+8},{1+2+16+8},{1+2+16+8},{1+16},{1+8},
{1+16+8},{1+8+16},{1+2+16},{1+2+8}};  //13 combinations
```

and the QPSLS combination which is the basis for determining permanent deflections

```
COMBINATION EUROCODE_QPSLSLC[]={{1},{1+2},{1+2+16},{1+2+8},{1+2+16+8},{1+16},
{1+8},{1+16+8}};  //8 combinations
```

Standard ASCE7-10

it is assumed that:
D=DL
L=LL
l=Lr
S=SL
R=RL
W=WL
E=EL
l=Lr

The FLC (factored loads combination) for analogy named after Eurocode as ULS to unify the designations, which is basically based on the same assumptions for taking into account load factors, although they will take a slightly different form.

for analogy to Eurocode:
ASCEFLC1=D*γg
ASCEFLC2=D*ξ + L*γq + Ψ0*l
ASCEFLC3=D*ξ + L*γq + Ψ0*S
ASCEFLC4=D*ξ + L*γq + Ψ0*R
ASCEFLC5=D*ξ + l*γq + L
ASCEFLC6=D*ξ + S*γq + L
ASCEFLC7=D*ξ + R*γq + L
ASCEFLC8=D*ξ + l*γq + Ψ0*W
ASCEFLC9=D*ξ + S*γq + Ψ0*W
ASCEFLC10=D*ξ + R*γq + Ψ0*W
ASCEFLC11=D*ξ + W + L + Ψ0*l
ASCEFLC12=D*ξ + W + L + Ψ0*S
ASCEFLC13=D*ξ + W + L + Ψ0*R
ASCEFLC14=D*ξ + E + L + Ψ1*S
ASCEFLC15=D*Ψ0 + W
ASCEFLC16=D*Ψ0 + E

NL (nominal loads combination) for analogy named after Eurocode as SLS

ASCENLC1=D
ASCENLC2=D + L
ASCENLC3=D + l
ASCENLC4=D + S
ASCENLC5=D + R

ASCENLC6=D + L*Ψ2 + l*Ψ2
ASCENLC7=D + L*Ψ2 + S*Ψ2
ASCENLC8=D + L*Ψ2 + R*Ψ2
ASCENLC9=D + W*Ψ2
ASCENLC10=D + E*Ψ2
ASCENLC11=D + L*Ψ2 + W*Ψ1*Ψ2 + l*Ψ2for analogy named after Eurocode as SLS
ASCENLC12=D + L*Ψ2 + W*Ψ1*Ψ2 + S*Ψ2
ASCENLC13=D + L*Ψ2 + W*Ψ1*Ψ2 + R*Ψ2
ASCENLC14=D + L*Ψ2 + E*Ψ1*Ψ2 + S*Ψ2
ASCENLC15=D*Ψ1 + W*Ψ2
ASCENLC16=D*Ψ1 + E*Ψ2

and additionally:
WDLC (Wind Drift Load Combinations) (rather imperfectly due to unification) named after Eurocode as QPSLS

ASCEWDLC1=D + L*Ψ1 + W*Ψ1

Here are the default coefficients for ASCE:

```
ST_LOAD_FACTORS st_load_factors_ASCE_0[]={
//load,variant, γ,       Ψ0,      Ψ1,     Ψ2,      ξ,      γ.inf
{/*0*/ 0,0,     0,       0,       0,       0,       0,       0},
{/*DL*/1,0,     1.4,     0.9,     0.6,     0,       1.2,     0},
{/*LL*/2,0,     1.6,     0,       0,       0.75,    0,       0},
{/*Lr*/3,0,     1.6,     0.5,     0,       0.75,    0,       0},
{/*WL*/4,0,     1.0,     0.5,     0.75,    0.6,     0,       0},
{/*SL*/5,0,     1.6,     0.5,     0.2,     0.75,    0,       0},
{/*EL*/6,0,     1.0,     0,       0.75,    0.7,     0,       0},
{/*RL*/7,0,     1.6,     0.5,     0,       0.75,    0,       0},
{/*HL*/8,0,     1.6,     0,       0,       0.75,    0,       0},
{/*FL*/9,0,     1.4,     0.9,     0.6,     0,       1.2,     0},
{/*TL*/10,0,    1.0,     0,       0,       0,       0,       0}};

COMBINATION ASCE_ULSLC[]={
{1,0},{1+2+4,0},{1+2+16,0},{1+2+64,0},{1+2+4,0},{1+2+16,0},{1+2+64,0},{1+4+8,0},
{1+8+16,0},{1+8+64,0},{1+2+4+8,0},{1+2+8+16,0},{1+2+8+64,0},{1+2+16+32,0},
{1+8,0},{1+32,0}};  //16 combinations

COMBINATION ASCE_SLSLC[]={{1,0},{1+2,0},{1+4,0},{1+16,0},{1+64,0},{1+2+4,0},
{1+2+16,0},{1+2+64,0},{1+8,0},{1+32,0},{1+2+4+8,0},{1+2+8+16,0},{1+2+8+64,0},
{1+2+16+32,0},{1+8,0},{1+32,0}};  //16 combinations

COMBINATION ASCE_QPSLSLC[]={{1+2+8,0}};  //1 combination
```

ICC Standard

load definitions:
D=DL
L=LL

l=Lr
S=SL
R=RL
W=WL
E=EL
H=HL
F=FL
l=Lr

FLC (factored loads combination) similar to ASCE for Eurocode as ULS

IBCFLC1=D*γg + F*γq
IBCFLC2=D*ξ + F*ξ + L*yq + H*yq + l*Ψ0 //(equation 2)
IBCFLC3=D*ξ + F*ξ + L*yq + H*yq + S*Ψ0
IBCFLC4=D*ξ + F*ξ + L*yq + H*yq + R*Ψ0
IBCFLC5=D*ξ + F*ξ + l*yq + H*yq + L*Ψ0 //(equation 3)
IBCFLC6=D*ξ + F*ξ + S*yq + H*yq + L*Ψ0
IBCFLC7=D*ξ + F*ξ + R*yq + H*yq + L*Ψ0
IBCFLC8=D*ξ + F*ξ + l*yq + H*yq + W*Ψ0
IBCFLC9=D*ξ + F*ξ + S*yq + H*yq + W*Ψ0
IBCFLC10=D*ξ + F*ξ + R*yq + H*yq + W*Ψ0
IBCFLC11=D*ξ + F*ξ + W + L*Ψ0 + H*yq + l*Ψ0 //(equation 4)
IBCFLC12=D*ξ + F*ξ + W + L*Ψ0 + H*yq + S*Ψ0
IBCFLC13=D*ξ + F*ξ + W + L*Ψ0 + H*yq + R*Ψ0
IBCFLC14=D*ξ + F*ξ + E + L*Ψ0 + H*yq + S*Ψ1 //(equation 5)
IBCFLC15=D*Ψ0 + W + H*yq //(equation 6)
IBCFLC16=D*Ψ0 + F*Ψ0 + E + H*yq //(equation 7)

where in the equations it is assumed:
(2) ξ=1.2 for DL, FL, γq=1.6 for LL, HL, SL, RL, γq=1.6 for L, l, H, R, S, γq=1.0 for W Ψ0=0.5 for l, S, R, W
(3) Ψ0=0.5 or 1.0 for LL, assumed 0.5 as for "other live loads”
(5) Ψ1=0.7 or 0.2 for S assumed 0.2 as for "other roof configurations"
(6) Ψ0=0.9 or D i F
(7) Ψ0=0.6 or F

where:
ξ=1.2 for DL, FL,
Ψ0=0.9 for D i F
γq=1.6 for LL, HL, SL, RL
γq=1.6 for L, l, H, R, S
γq=1.0 for W
Ψ0=0.5 for l, S, R, W
Ψ0 dla L = 1 for public assembly areas, live load exceeding 100 pounds per square foot (4,79 kN/m2) and garages
Ψ0 dla L= 0,5 for other live loads
Ψ1 dla S= 0,7 for shed roof configurations that do not shed snow from the structure
Ψ1 dla S= 0,2 for other roof combinations
BLC (Basic load combinations) for analogy named after Eurocode as SLSLC

IBCBLC1=D + F
IBCBLC2=D + H + F + L
IBCBLC3=D + H + F + l
IBCBLC4=D + H + F + S
IBCBLC5=D + H + F + R
IBCBLC6=D + H + F + L*Ψ2 + l*Ψ2
IBCBLC7=D + H + F + L*Ψ2 + S*Ψ2
IBCBLC8=D + H + F + L*Ψ2 + R*Ψ2
IBCBLC9=D + H + F + W*Ψ2
IBCBLC10=D + H + F + E*Ψ2
IBCBLC11=D + H + F + W*Ψ1*Ψ2 + L*Ψ2 + l*Ψ2
IBCBLC12=D + H + F + W*Ψ1*Ψ2 + L*Ψ2 + S*Ψ2
IBCBLC13=D + H + F + W*Ψ1*Ψ2 + L*Ψ2 + R*Ψ2
IBCBLC14=D + H + F + E*Ψ1*Ψ2 + L*Ψ2 + S*Ψ2
IBCBLC15=D*Ψ2 + W*Ψ2 + H
IBCBLC16=D*Ψ2 + F*Ψ2 + E*Ψ2 + H

where :

Ψ0=0.5 for l, S, R, W
Ψ1=0.75 for W, E
Ψ2=0.75 for L,l,S,R
Ψ2=0.7 for E
Ψ2=0.6 for D, W, F

ABLC (Alternative basic load combination) for analogy named after Eurocode as QPSLSLC

IBCABLC1=D + L + l
IBCABLC2=D + L + S
IBCABLC3=D + L + R
IBCABLC4=D + L + 0.6*ω*W
IBCABLC5=D + L + 0.6*ω*W + Ψ0*S
IBCABLC6=D + L + S + Ψ0*Ψ2*ω*W
IBCABLC7=D + L + S + Ψ2*E //in fact Ψ2=0.7143
IBCABLC8=0.9*D + + Ψ2*E //in fact Ψ2 =0.7143

This leads to the establishment of the following table of coefficients:

```
ST_LOAD_FACTORS st_load_factors_ICC_0[]={
//load,variant, γ,       Ψ0,     Ψ1,    Ψ2,      ξ,     γ.inf
{/*0*/ 0,0,     0,       0,      0,      0,      0,      0},
{/*DL*/1,0,     1.4,     0.9,    1.0,    0.6,    1.2,    0},
{/*LL*/2,0,     1.6,     0.5,    0,      0.75,   0,      0},
{/*Lr*/3,0,     1.6,     0.5,    0,      0.75,   0,      0},
{/*WL*/4,0,     1.0,     0.5,    0,      0.6,    0,      0},
{/*SL*/5,0,     1.6,     0.5,    0.2,    0.75,   0,      0},
{/*EL*/6,0,     1.0,     1.0,    0,      0.7,    0,      0},
{/*RL*/7,0,     1.6,     0.5,    0,      0.75,   0,      0},
{/*HL*/8,0,     1.6,       0,    0,      0,      0,      0},
{/*FL*/9,0,     1.4,     0.9,    0,      0.6,    1.2,    0},
{/*TL*/10,0,    1.0,       0,    0,      0,      0,      0}};
```

and combination masks:

```
COMBINATION ICC_ULSLC[]={{1+256},{1+2+4+128+256},{1+2+16+128+256},
{1+2+64+128+256},{1+2+4+128+256},{1+2+16+128+256},{1+2+64+128+256},
{1+4+8+128+256},{1+8+16+128+256},{1+8+64+128+256},{1+2+4+8+128+256},
{1+2+8+16+128+256},{1+2+8+64+128+256},{1+2+16+32+128+256},{1+8+128},
{1+32+128+256}};  //16 combinations

COMBINATION ICC_SLSLC[]={{1+256},{1+2+128+256},{1+4+128+256},{1+16+128+256},
{1+64+128+256},{1+2+4+128+256},{1+2+16+128+256},{1+2+64+128+256},{1+8+128+256},
{1+32+128+256},{1+2+4+8+128+256},{1+2+8+16+128+256},{1+2+8+64+128+256},
{1+2+16+32+128+256},{1+8+128},{1+32+128+256}};  //16 combinations

COMBINATION ICC_QPSLSLC[]={{1+2+4},{1+2+16},{1+2+64},{1+2+8},{1+2+8+16},
{1+2+8+16},{1+2+16+32},{1+32}};  //8 combinations
```

Although the significance of the coefficients for individual standards results from different rules for determining combinations and they differ in value, the ASCE and ICC standards largely overlap. However, when using individual coefficients for individual load characteristics and variants, different standards should not be used interchangeably, unless a set of texts describing the load characteristics are also selected individually depending on the choice of standard.
The reason why the same combination designations are used for different standards is to unify the names of the layers created by the program. The fact is that the name of the layer can be freely changed by the program, while leaving its number, and therefore without the need to delete the layer, which results in the need to change the layer numbers of all other objects. If it turns out that such unification causes misunderstanding, this problem will be solved in a better way in subsequent releases of the program. From the point of view of structural safety, the first two combinations are of key importance, ULSLC for dimensioning cross-sections and supports, and the SLSLC combination for determining maximum deflections and displacements. As a European, I have some difficulty interpreting the purpose of, for example, alternative combinations in the ICC standard, and whether their calculation is necessary, and in the case of the ASEC standard, it seems that the WDLC (Wind Drift Load Combinations) combination is wrongly marked in the program as QPSLS, and whose task is to find, for example, negative support forces, i.e. lifting supports or opposing internal forces under the influence of the nominal dead load, reduced payload, but also reduced wind load, so could this combination not be included in the ULSLC combination, which would save the need to create separate layers and blocks of force diagrams for this combination. I count on the cooperation of users in this matter.

AlfaCAD home page while the program is loading

AlfaCAD in version 2.7 has been equipped with tools that allow for automatic loading of resources from the server where AlfaCAD installation packages are deposited, including loading of alternative home pages containing nice photos obtained under the "Public domain" license and displayed while the program loads the necessary resources into memory. This moment of delay not only allows you to view the photos, but also, alternatively, to familiarize yourself with the content of the advertisement that could appear there, if there are those who would like to support the development of AlfaCAD in return for the opportunity to encourage users of the program to their products and services. Of course, advertisements from the technical sector are mostly welcome.

🇬🇧 🇺🇸 Your ad could be here.
Think about it, and when you feel the time is right, contact me.

🇵🇱 Twoja reklama może się tu znaleźć.
Pomyśl o tym i kiedy uznasz że nadszedł właściwy czas, skontaktuj się ze mną.

🇺🇦 Ваша реклама може бути тут.
Подумайте про це, і коли відчуєте, що настав час, зв’яжіться зі мною.

🇪🇸 Tu anuncio podría estar aquí.
Piénsalo y cuando sientas que es el momento, contáctame.

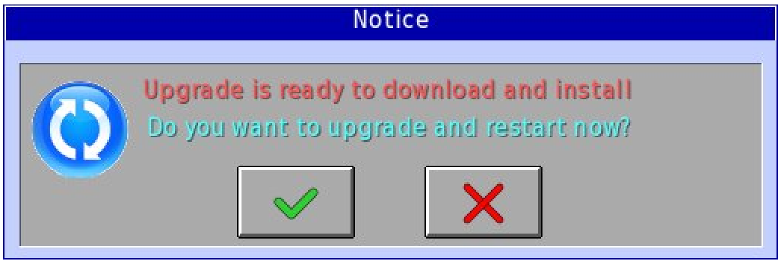
author@alfacad.net

Automatic program update

Since the tools that allow for automatic loading of resources from the server where AlfaCAD installation packages are deposited are already active, immediately after the program is started, but also at intervals of no less than 7 days from the last check, the program checks whether a newer version of the program has appeared on the server, and this is only an update package, not a full installation package, which is deposited on the server when many of its elements have undergone real changes, about which the program will also inform. Usually, the update package does not exceed several MB, when the complete package takes up from 400 to 500 MB depending on the system platform. Usually, the update package contains the executable file of the main program, so in the case of the Linux platform, after decompressing the file in ZIP format, because in this format the update file is sent from the server, the program changes the file attributes so that the file becomes an executable file. Windows executable file is executable ever.

Before checking for updates on the server, the program checks whether the system has the right tools to update. If not, the appropriate message about the need to install packages appears on the screen. This applies to curl, unzip and sed packages on the Linux platform, and curl and tar on the Windows platform. The sed program for Windows is included in the update to version 2.7.

A recognized update ready for installation requires acceptance by the user in order to download, install, and automatically restart the program.



Credits

for developing and sharing the code of Frame3DD:
Henri P. Gavin, Ph.D., P.E. Department of Civil and Environmental Engineering, Duke University, Durham, NC and **John Pye**, Dept of Engineering, Australian National University
https://frame3dd.sourceforge.net/

for motivating me to take the first step to include load combinations in the static analysis module
Laurin Ernst
https://www.structuralbasics.com/
https://www.structuralbasics.com/load-combinations/

for Clion IDE, which suits me like no other
JetBrains
https://www.jetbrains.com/clion/

for a system that still makes me think that software development is good for the health of the programmer himself

Arch Linux

Enjoy AlfaCAD!

🇬🇧 🇺🇸 alfacad.net
🇵🇱 pl.alfacad.net
🇺🇦 ua.alfacad.net
🇪🇸 alfacad.net/es/ *(sólo descargar)*

author

Marek Ratajczak

August 8, 2024