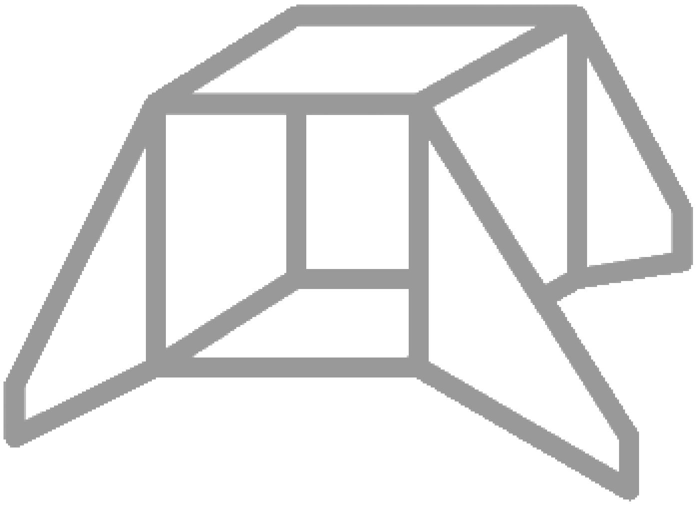
Software for Architecture,
Engineering and Construction

StruBIM Box Culverts

User's manual

Design and checking of reinforced concrete box culverts (rectangular or trapezoidal, single or multi cell) for underpasses under roads and drainage works. Free polygonal outline on plan and section.



Contents

1 Introduction

StruBIM Box Culverts is a tool for designing and checking reinforced concrete box culverts used for underpasses under roads and drainage works. These box culverts -or frames- can be rectangular, trapezoidal, or have a freely-designed polygonal layout both in floor plan and elevation drawings, and they can be single-cell or multi-cell.

The program can be used for designing frames for prefabrication by defining joints or for in-situ assembly. The section allows users to configure multi-cell box culverts.

The tool includes several assistants to help users enter data for standard cases: straight, skewed, and general box culverts.

The design model used is a 3D thick-shell triangular finite element type, which considers shear deformation. These elements consist of six nodes, at the vertices and midpoints of the sides, each with six degrees of freedom.

Truck loads may be entered in any position, as well as strip loads and loads on slabs.

2 Calculations

2.1 Preliminary concepts

StruBIM Box Culverts consists of:

- **Modules**

  Lintel slabs and base slabs embedded between two lateral walls. The module's interior cell can be single or multi-cell, where additional central walls are added to these two side walls, into which the lintel and base slab or foundation slab are also embedded. All these slabs have a constant thickness.

- **Wingwalls**

  Wingwalls are independent elements of the main structure. They consist of a reinforced concrete wall working as an overhang with constant thickness, embedded into the continuous footing, with variable front and rear overhangs, depending on the wall's height.

2.2 Applied regulations

For checking and designing reinforcements in the sections, the following options may be selected:

- **EHE (Spain)**
- **REBAP (Portugal)**
- **BAEL 91 (France)**

Eligible materials will comply with these standards.

Additionally, the **EC-2 (Eurocode)** provisions, **CYPE** criteria and various authors' criteria have been applied.

Regarding the generation of actions, the following have been considered:

- **AP-98, Instructions for actions to be considered in road bridge projects** *"Instrucción sobre las acciones a considerar en el Proyecto de Puentes de Carreteras"* **(Spain)**
- **RSA (Portugal)**
- **CFC, Fascicule 61 (France)**

ROM 0.2-90, Actions in maritime and port work projects *"Acciones en el proyecto de obras marítimas y portuarias"* (Spain), has also been considered for areas that are not covered in the aforementioned standards.

2.3 Loads

In addition to self-weight loads, the program generates the following loads:

2.3.1 Earth pressures

This includes all the actions caused by the fill on the structural elements in contact with it. Earth pressure has two components: the weight on horizontal elements (frame lintel and base of the wingwalls) and the thrust on vertical elements (sidewalls and wingwalls).

- The soil's weight on the horizontal elements is determined by applying the specific weight of the poured and compacted fill to the volume of soil that gravitates over the surface of the horizontal element.
- Earth pressure is estimated via a two-dimensional calculation, inserting sections that are perpendicular to the module's walls and wingwalls.

The module's lateral walls are considered rigid enough to assume a loadcase of at-rest earth pressure. The at-rest pressure coefficient is estimated through the use of Jaky's formula. If there is a certain slope angle, the additional formulation from the Corps of Engineers, 1961, is applied.

For wingwalls, the pressure on the walls is calculated using the Coulomb method (active pressure coefficient method), considering a slope angle equal to the internal friction coefficient. The slope ends at the level of the top road surface; from this point onward, the soil is considered to be horizontal. The model can consider the cohesion effects, and the friction angle between the soil and the wall.

For combinations, the earth pressure is considered in accordance with the provisions of IAP-98, as a permanent load with non-constant value.

2.3.2 *Loads applied on the top road surface*

The program defines the variable service and/or exploitation loads applied on the road surface:

Top surcharge: Infinite-extension superficial dead load. Different dead loads can be defined for the modules and each of the wingwalls. Combinations are considered as incompatible with other loads applied on the road surface.

Strip loads: Used to simulate corresponding traffic surcharges on the roadway. Their effect on the wingwalls is not considered.

Truck loads: The available truck loads are described in the Spanish IAP-98, Portuguese RSA, Brazilian NBR, French CBC, Eurocode, and ROM 0.2-90 standards. Additionally, users can define their own truck loads. Their effect on the wingwalls is not considered.

All of these loads are considered to be applied in the Z global direction and only affect the module.

These loads may act on the structure in different ways:

- Effect on the lintel: The loads transfer their action onto the lintel through the fill, which behaves as a distribution layer that projects the load enclosure as a pyramid. The transmission angle can be configured by the user. The surface value of the projected load is determined with the condition that the point value is the same as in the original load. If no fill is defined or the thickness of the layer is null, the load will be considered as directly applied to the lintel.

- Effect on the lateral walls: The loads transfer their action onto the module's lateral walls through the fill, so that the earth generates additional pressure on them. The loadcase of an at-rest earth pressure state is still valid.

  If the load is uniform, the effect of an earth pressure increase is equal to the one caused by the additional height of the earth, of value $\frac{q}{\gamma \cos(\beta)}$, where **β** is the slope angle, **q** load value, and γ the specific weight of the soil.

  The effect of point load rigid walls, with prevented displacement, is determined by using elasticity theory. The problem is solved using the Boussinesq approximation, replacing the condition of no wall deformation, with an additional virtual load symmetrical to the real one, with respect to the wall's backfill.

  If the applied action is a strip load or truck load, they are calculated by point load superposition.

2.3.3 *Loads applied on the bottom slab*

Bottom uniform load: Superficial load applied exclusively on the bottom slab of the module.

2.3.4 *Hydraulic surcharge*

Load caused by a water flow inside the frame. This is evaluated based on a specific weight of water equal to 9.8 kN/m^3. The water pressure is hydrostatic. As for combinations, they are considered incompatible with the bottom uniform load.

> ***Note:*** *The effects produced on box culverts by thermal loads, breaking loads, and seismic loads are not considered.*

2.4 Completed discretisation

The design model used is a 3D thick-shell triangular finite element type, which considers shear deformation. These elements are formed by six nodes located at the vertices and midpoints of the sides with six degrees of freedom each. A box culvert mesh is generated based on its dimensions (thickness, spans and applied loads).

The module's foundation is assumed as a slab supported on an elastic soil with springs at the nodes (ballast coefficient method), according to Winker's model, based on a proportionality constant between forces and displacements, whose value is the ballast coefficient. This loadcase is valid for homogeneous soils. The ballast modulus is a parameter to be entered into the program. This is determined through empirical methods with a load plate test. Typically, a geotechnical study should provide the exact value of this modulus for the dimensions of the foundation slab.

The wingwall is designed as a cantilevered element. However, if the length of the wingwall is large, the reinforcement is divided into several spans (the minimum length for each span is defined in the design options) and each span will have a different footing size. In the case of variable wall height, each span is discretised into 2-metre-long vertical strips (measured in the floor plan) for the reinforcement. The reinforcement design is based on the worst-case scenario for the largest strip, and this design is applied to the entire wall corresponding to the same segment. For the stability analysis, to stop the wingwalls from overturning and sliding, a single result is obtained for the entire elevation, and it is checked against the entire footing, not the individual strips.

Continuous footing is designed under the wingwall, which can have various configurations: with both toe and heel, only toe, or only heel.

For the joints, the provisions on road instructions of the Spanish Ministry of Public Works (**MOPU**) *“Road crossings. Collection of small crossings”* (*“Obras de paso de carreteras. Colección de pequeñas obras de paso 4.2. I.C.”*) have been applied. This states the following: *"The main bodies of rigid structures are generally divided by a set of shrinkage and watertight joints”*. Therefore, if joints are present, the program will consider the modules as independent for all purposes. The instruction also states that *“The wing is an independent element of the same body”*, therefore; the wingwalls are also considered independent modules.

2.5 Analysis method

To obtain the loads, the principles of **Rational mechanics** have been considered along with the classical theories of **Material resistance and elasticity**.

The applied analysis method follows **Limit State** principles. The aim is to ensure that the effect of external loads, weighted by certain coefficients, does not exceed the structure’s response, accounting for reduced material strengths (following **EHE**, **RSA**, **REBAP**, and **BAEL standards**).

For **Ultimate Limit States (ULS)**, checks are made for equilibrium, fatigue, or rupture.

In the **States of Serviceability (SLS)**, checks are made for deformations (deflections), stresses on the soil and slab separations.

Once the load states have been defined according to their origin, the possible combinations are analysed with the corresponding increase and decrease coefficients, based on the safety factors and basic loadcases defined in the standards.

To determine stresses under different loadcases, analyses are performed using first-order linear analysis, assuming proportionality between stresses and deformations, the principle of action superposition, and linear geometric behaviour of materials and structure.

In order to obtain the determining stresses in the element design, the envelopes are obtained for each stress.

2.6 Results

2.6.1 *Modules*

In each module, eight stresses are obtained through elastic and linear analysis to verify and design the concrete section and reinforcement. Displacements are used to check deflections, stresses on the soil and the detachment of the foundation slab, among others.

The checked states are:

- **Minimum geometric reinforcement ratio**

  In order to control the cracking due to deformations caused by temperature and shrinkage, minimum reinforcements ratios are applied, which vary according to each standard.

- **Minimum mechanical reinforcement ratio**

  Minimum mechanical reinforcement ratios are required to prevent brittle failure when the section cracks due to bending moment stresses.

- **Bending moment check**

  The structural check of the cross-section is carried out using the simplified parabola-rectangle stress-strain diagram as the constitutive law of concrete, which is suitable for delimiting the bending moment rupture stress zone from the non-rupture zone of a reinforced concrete cross-section. The bending moment verification is implemented for all standards supported by the program, with specific considerations for stress integration in the cross-section and the pivots that define the maximum allowable deformations for the materials constituting the cross-section (steel and concrete).

  For the bending moment check, the reinforcement must be anchored in order to be considered effective in the bending moment analysis.

  Furthermore, as bending moment stresses act simultaneously with shear stresses, there is an interaction between them. This phenomenon is considered by shifting the bending moment law by a certain distance in the most unfavourable direction.

- **Shear check**

  This ultimate limit state is checked in the same way as the bending moment. Since there is no transverse reinforcement in the cross-section, only the contribution of concrete to shear resistance is considered. The contribution of concrete to shear stress is evaluated using the experimental term V_{cu}. This term is usually included in the tensile fatigue shear check of the cross-section's web. Various expressions for evaluating the V_{cu} component have been considered based on the chosen standard.

- **Maximum displacement and relative deflection**

  The displacements and deflection for structural elements are limited according to the applicable standard.

- **Angular distortion**

  The angular distortion generated in the top slab, bottom slab, lateral walls, and dividing walls, is considered as a serviceability limit state (SLS).

- **Slenderness ratio**

  The maximum slenderness of compressed elements, such as lateral walls and module slabs, is limited.

- **Minimum anchorage hook length**

  Calculations are carried out in compliance with the various implemented standards.

- **Minimum reinforcement spacing**

  The minimum spacing between reinforcement bars according to standards is required in order to allow the concrete to be poured correctly.

- **Maximum reinforcement spacing**

  This limit is set to ensure that there are no areas without reinforcement. This is a minimal condition for distinguishing between "reinforced concrete" and "mass concrete".

- **Uplift**

  A check is carried out to verify that there is no upward vertical displacement at any node in the foundation slab, as this would invalidate the analysis (the soil cannot pull the slab). If this occurs, the structure should be checked, by stiffening the slab if possible.

- **Allowable bearing pressure**

  The maximum bearing pressure that can be applied to the soil is limited to the value specified by the user.

2.6.2 *Wingwalls*

The states to be checked are as follows:

- **Check for joint shear at base of wall/span**

  The calculated shear at the joint between the wingwall elevation and the footing is checked to ensure that it is less than what the section can withstand at that point, considering the concrete cross-section and the steel provided.

- **Minimum thickness**

  The minimum thickness is restricted according to the standard's requirements.

- **Minimum geometric ratio**

  Minimum reinforcement ratios are imposed to control cracking due to deformation caused by temperature and shrinkage and vary according to each standard.

- **Minimum mechanical ratio**

  For vertical reinforcements, minimum mechanical ratios are required to prevent brittle failures when the section cracks due to bending momentum stresses.

- **Maximum geometric ratio**

  A maximum ratio is imposed on the total vertical reinforcement.

- **Minimum reinforcement spacing**

  The minimum spacing between reinforcement bars according to standards is required in order to allow the concrete to be poured correctly.

- **Maximum reinforcement spacing**

  This limit is established to ensure that there are no areas without reinforcement. This is a minimal condition for distinguishing between "reinforced concrete" and "mass concrete".

- **Bending moment check**

  The structural check of the cross-section is carried out using the simplified parabola-rectangle stress-strain diagram as the constitutive law of concrete, which is suitable for delimiting the bending moment rupture stress zone from the non-rupture zone of a reinforced concrete cross-section. The bending moment verification is implemented for all standards supported by the program, with specific considerations for stress integration in the cross-section and the pivots that define the maximum allowable deformations for the materials constituting the cross-section (steel and concrete).

For the bending moment check, the reinforcement must be anchored in order to be considered effective in the bending moment analysis.

Furthermore, as bending moment stresses act simultaneously with shear stresses, there is an interaction between them. This phenomenon is considered by shifting the bending moment law by a certain distance in the most unfavourable direction.

- **Shear check**

  The ultimate limit state is checked in the same way as the bending moment. As there is no transverse reinforcement in the section, only the contribution of concrete to shear resistance is considered. The value of the concrete contribution to shear stress is evaluated using the experimental term V_{cu}. This term is usually included in the tensile fatigue shear verification of the section's web. Various expressions for evaluating the V_{cu} component are considered based on the chosen standard.

- **Cracking check**

  The cracking limit state is a Serviceability Limit State (SLS) that is checked to control the appearance of cracks in the concrete structures. Crack control is crucial for walls as it occurs primarily on the backfill face. This is an area where the reinforcement corrosion is likely to proliferate and where it cannot usually be observed. Deterioration of the structure can occur without the negative effects on the wall being clearly visible. The cracks caused by actions directly acting on the wall (soil, water table, surcharges, etc.) should be controlled, rather than the cracks caused by shrinkage and temperature, which are already taken into account when considering geometric minimums.

  A simplified process in simple bending has been followed to calculate the limit crack opening, which provides safer results than those that can be obtained by applying bending moment methods.

  The general crack opening method is followed for the different standards applied in the program. The results obtained are compared with the limits imposed by each standard, according to the type of exposure or environment in which the structure is located.

  Unlike the ultimate limit states of bending moments and shear, in which the combination of actions related to the ultimate limit states are used, in the case of cracking, the combinations of actions related to the characteristic actions are applied. The program works by calculating the characteristic crack opening for all the loadcases.

  The calculation is repeated at different screen heights in the same way as for the bending moment and shear checks. The most unfavourable value is extracted and compared with the limit crack opening indicated by each standard. This way, it is possible to determine whether or not the Serviceability Limit State (SLS) is met.

- **Lap splice length check**

  The lap splice length check analysis is carried out according to the different implemented standards.

- **Base reinforcement crown anchor check**

  The anchor length analysis is carried out according to the different implemented standards.

2.6.3 *Wingwall footings*

The load on a wall is converted into a load distribution along the wall in a discrete way. This is like transforming a resultant force into a stress distribution applied along the base of the wall, discretised into steps that the program calculates internally according to the dimensions.

The states to be checked are the following:

- **Overturning/Sliding stability check**

  By applying the corresponding limit state combinations, the resultant is checked to ensure that it lies within the footing, and the stability coefficient for overturning and sliding is calculated.

- **Soil bearing pressures**

  A plane strain deformation assumption is made for the footing, so that, depending on the stresses, trapezoidal-shaped soil bearing pressure will be obtained. Traction is not allowed, therefore, if the resultant force falls out of the central core, unstressed areas will appear. The resultant must lie within the footing; otherwise, there would be no equilibrium. The self-weight of the footing is considered. The average stress is checked to ensure that it does not exceed that of the soil and that the maximum edge stress does not exceed the average stress by more than 1%.

- **Minimum depth**

  The minimum depth specified by the corresponding standard is checked.

- **Anchor length**

  The anchors at the ends of the reinforcements are checked, placing the corresponding hooks as necessary and according to their position.

- **Minimum bar diameter**

  The diameter is checked to ensure that it does not exceed the minimum specified in the standard.

- **Maximum bar spacing**

  This limit is established to ensure that there are no areas without reinforcement. This is a minimal condition for distinguishing between “reinforced concrete” and “mass concrete”.

- **Minimum bar spacing**

  Minimum spacing between reinforcements specified by the standard are checked.

- **Footing bending**

  The footing bending is checked with the reference section located at 0.15 times of the wall dimension towards the interior. Bending design is needed to provide adequate depths to avoid the need for compression reinforcement. If tractions appear in the top face of the footing, upper reinforcements will be inserted.

- **Shear**

  The reference section is placed at a useful depth of the wall edges. Shear design makes it necessary to provide adequate depths to avoid the need for transverse reinforcement.

- **Geometric and mechanic reinforcement ratio**

  Compliance with the minimum mechanical and geometric reinforcement ratios specified in the standard is checked.

3 Program description

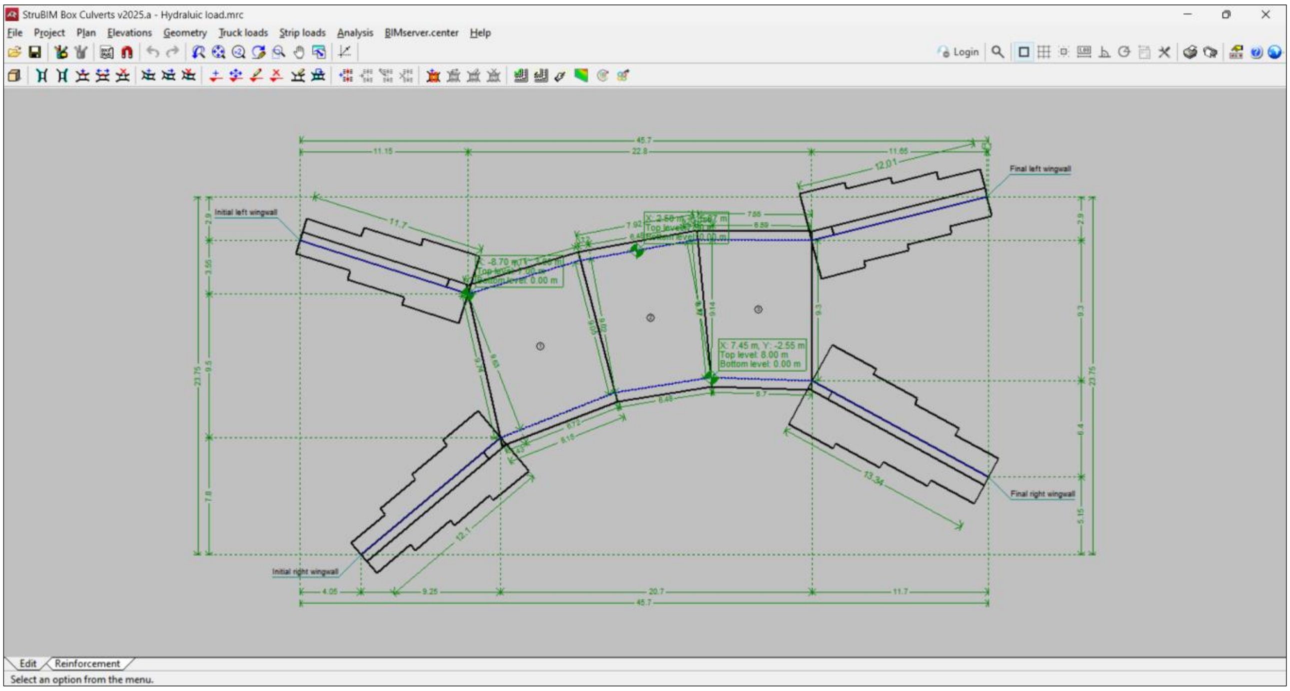


Figure 1

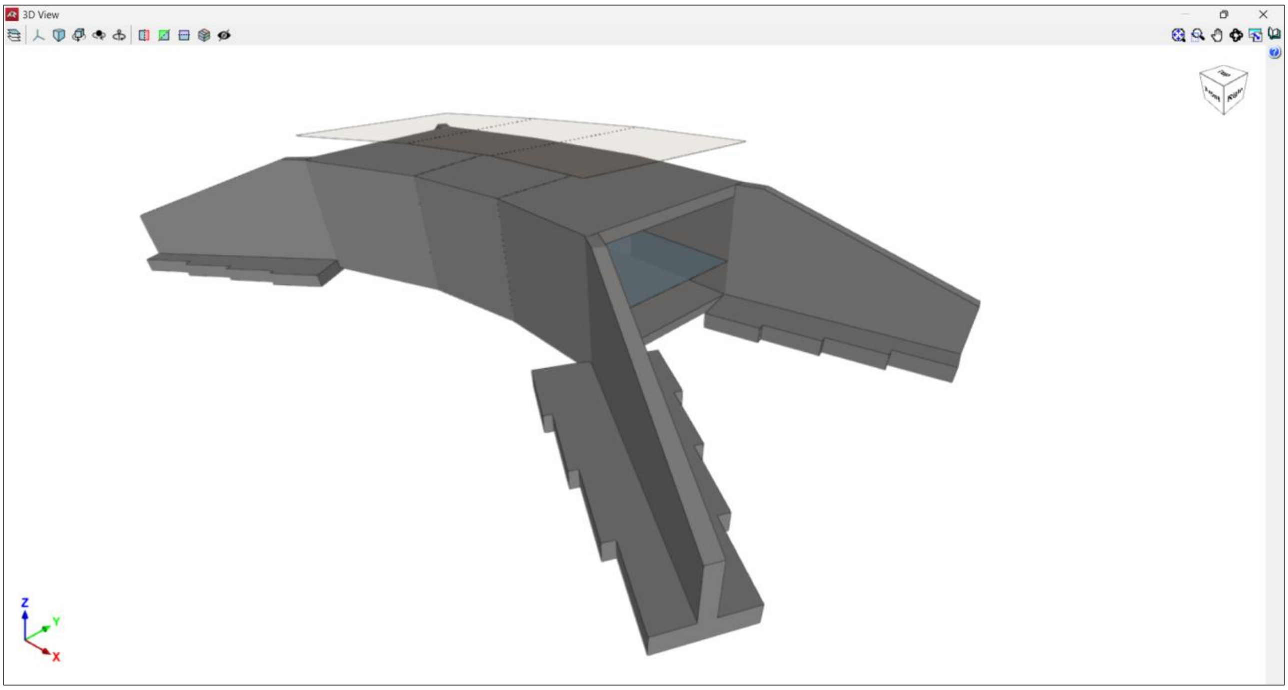


Figure 2

3.1 Creating a new job

To create a new job, the **New** button, under File shall be clicked in the start-up menu, as seen in Figure 3.

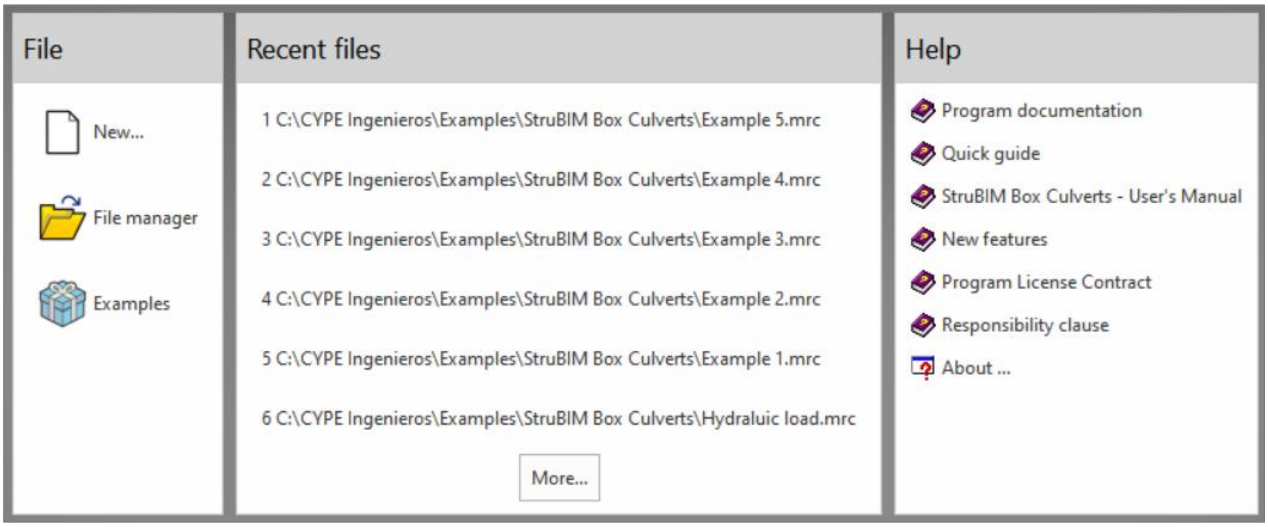


Figure 3

Then, the *New job* window will open, where a **File name** and **Description** may be entered if desired. By default, the name *"new"* will be assigned.

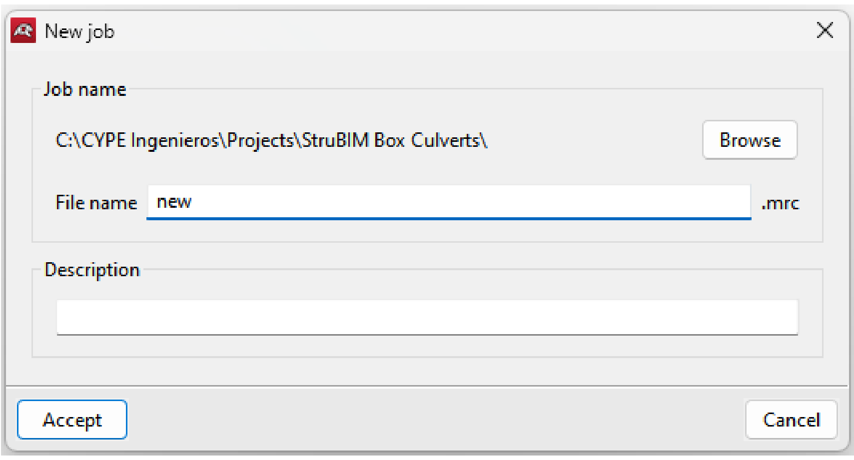


Figure 4

After clicking on **Accept**, the *Assistant Manager* will pop-up. For more details on Assistants, refer to section 3.2.

If no Assistant is selected, the *BIMserver.center Project selection* window will open. There, the linking of the new job with an existing project is possible. If an Assistant is selected, this menu will appear after each of their corresponding minimum parameters is entered.

Otherwise, after selecting the preferred option on BIMserver.center project linking, the **Materials** options will appear where Concrete, Steel Grade, and Type of environment configurations may be set (Figure 5).

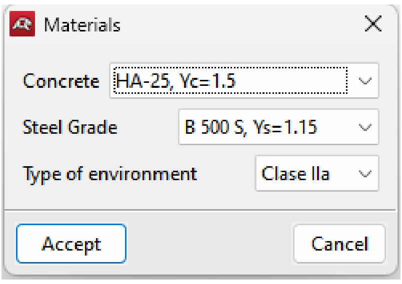


Figure 5

After configuring the materials settings, and clicking on **Accept**, the *General data* window will open, where information regarding Ground, Geometry, Loads, and Reinforced concrete parameters may be entered (Figure 6).

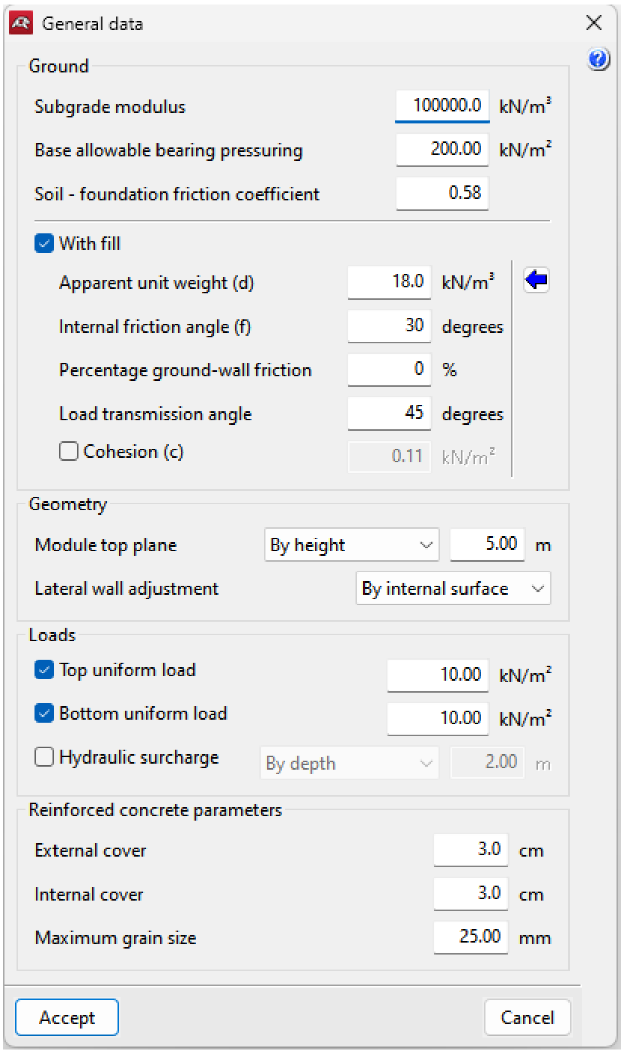


Figure 6

Please note that both **Materials** and **General data** options may be accessed at any moment in the design process under the *Project* menu in the toolbar. Additionally, tools such as **Options**, **Reinforcement tables**, **Views**, and **3D View**, may be also accessed at any point under the same menu.

3.2 Assistants

When creating a new job, the *Assistant selection* dialog will be displayed.

If the new job is created with an assistant, the program will generate the necessary data to describe it (depending on the selected assistant type) from a small number of sequentially entered parameters. This includes geometry and soil loads generation, truck loads, uniform strip loads, and uniform loads on slabs.

The slab thicknesses are dependent on the clear span between lateral walls. The lateral wall thicknesses are dependent on the internal height.

The desired assistant shall be selected:

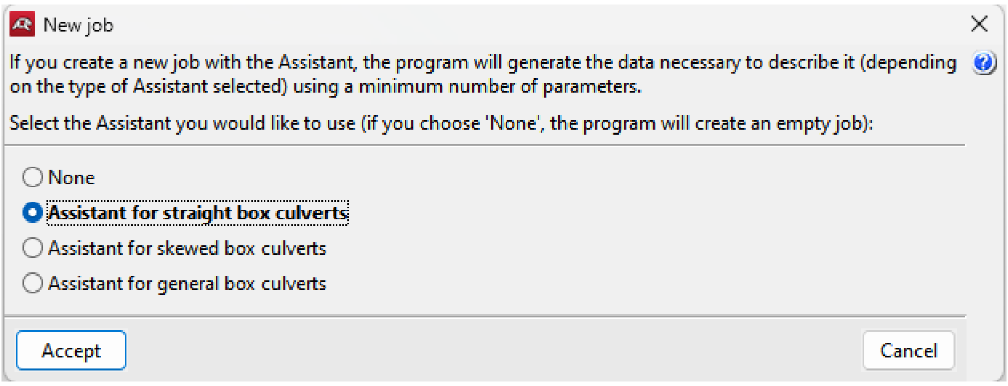


Figure 7

- **None**

  The program will create an empty job. Refer to section 3.1 for further information.

- **Assistant for straight box culverts**

  A box culvert with joints and parallel lateral walls can be defined, in other words, a rectangular floor plan layout. The sequence of data entry is shown as an example. For other types of box culverts, the requested data varies slightly.

  Here, the internal height (G), clear span (Li), and length axis (Le) of the modules are requested.

The elevation of the top roadway surface is not requested. This top surface will be determined by the top slab thickness, which is defined by the clear span (refer to the table shown later).

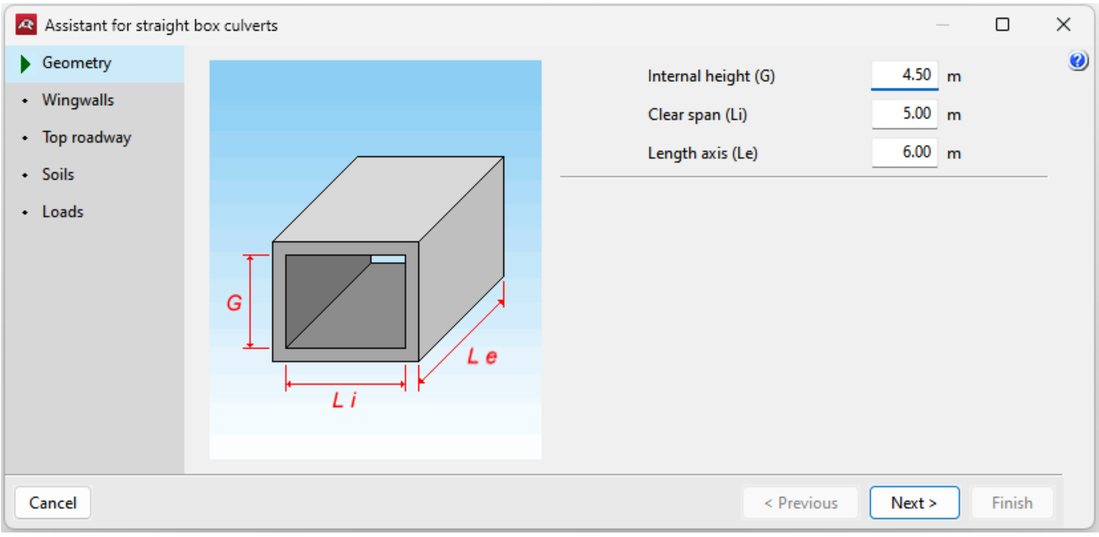


Figure 8

Then, the existence of wingwalls and their angles can be activated or deactivated. The program generates the load on the backfill of the wingwalls based on the side of the angle. For example, assuming a surcharge of 1 t/m^2, a 0-degree angle generates 0 t/m^2 of surcharge, a 45-degree angle generates 0.7 t/m^2, and a 90-degree angle generates 1 t/m^2.

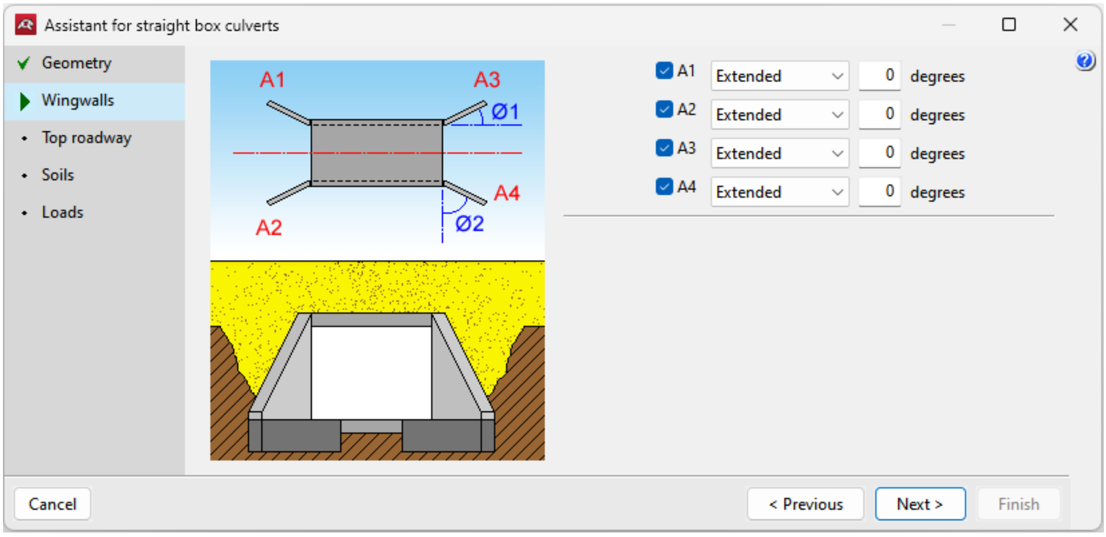


Figure 9

In the next step, the top roadway surface is defined. Also, the strip loads are placed, and their position and rotation angles are defined.

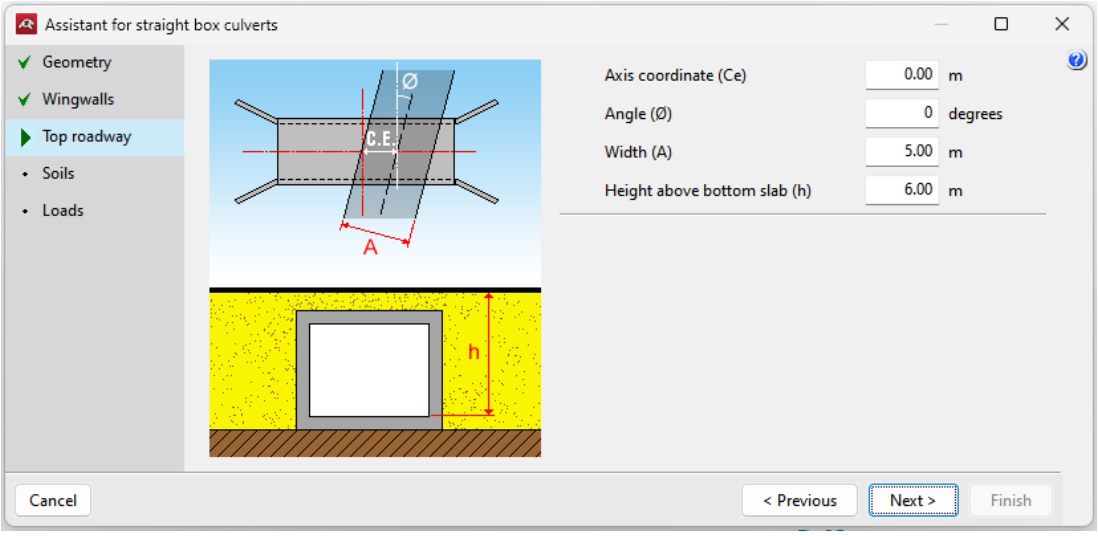


Figure 10

Afterwards, the base and infill soil types are defined, with the corresponding slope angle.

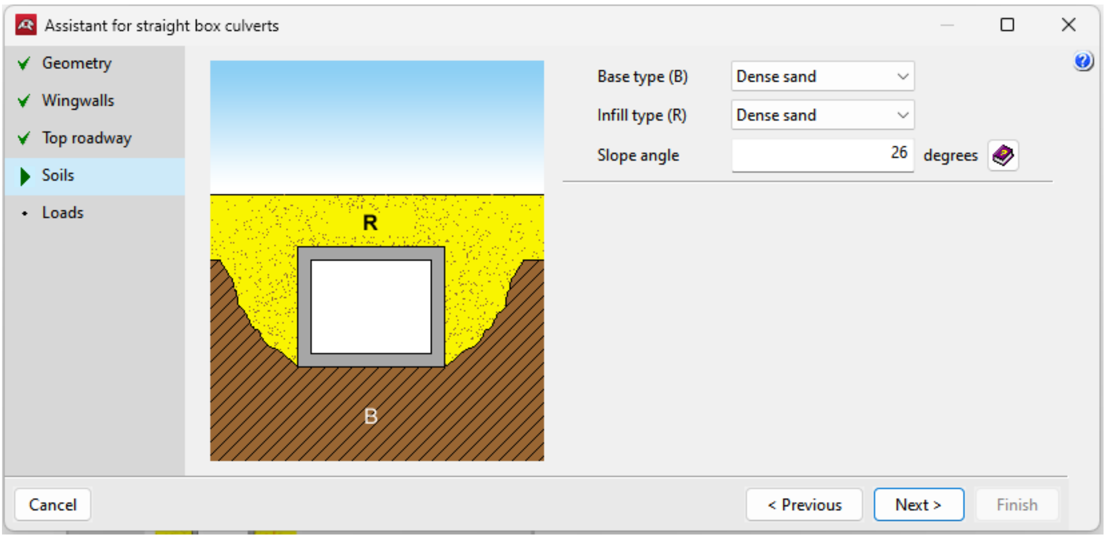


Figure 11

Then, the program will request for the type of truck load generation and the surcharge on the bottom slab.

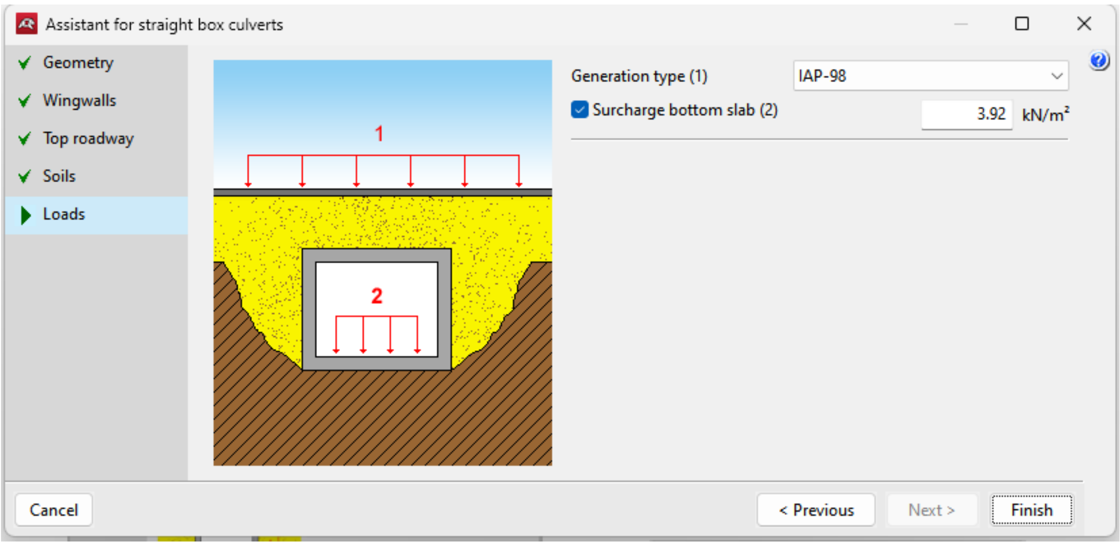


Figure 12

Finally, a summary of the box culvert generation parameters is displayed.

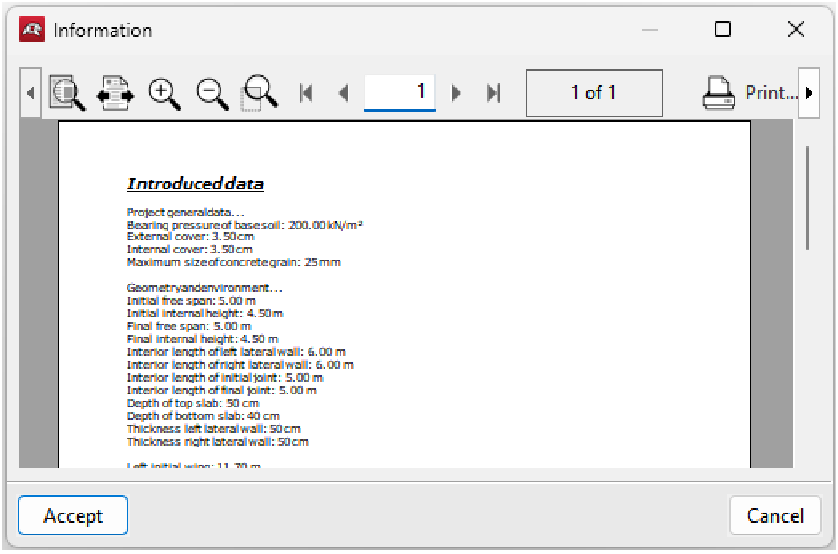


Figure 13

Once the generation is complete, users can modify any parameter as desired.

- **Assistant for skewed box culverts:**

  Joint box culverts with any angle and parallel wingwalls can be defined.

- **Assistant for general box culverts**

  Joint box culverts and wingwalls with any angle can be defined.

For any of the assistants provided in the program, the following criterion is used for the thickness design:

Span	Top slab thickness	Bottom slab thickness
$L \leq 5$	$0.02 \cdot L^2 + 0.10$	$0.02 \cdot L^2 + 0.10$
$5 < L < 7$	0.5	0.4
$L \geq 7$	$0.00835 \cdot L^2 + 0.10$	$0.00816 \cdot L^2 + 0.10$

Interior height	Lateral wall thickness
$G \leq 4.5$	$0.02 \cdot G^2 + 0.10$
$4.5 < G < 7$	0.5
$G \geq 7$	$0.007126 \cdot G$

The generated road surfaces are horizontal in all the assistants.

3.3 Inserting the geometry in plan

This is carried out via the options in the *Plan* menu. First of all, the adjustment lines of the lateral walls must be entered (in **blue**). Please note that, by default, the lateral walls' inner faces are aligned with the adjustment lines, so when entering the adjustment lines, this must be done considering they are the lateral walls' inner faces, and therefore, the separation between the right and left lateral walls is the clear span.

This menu also allows users to indicate where the joints are.

Although it is not strictly necessary, it is much quicker to enter the box culvert's geometry if there is a DXF or a DWG file available where the lateral walls are inserted in plan (preferably the inner face line).

Using a DXF or a DWF as a template for entering the lateral walls is convenient compared to the coordinate entry, and may be the preferred method when the use of assistants is not desired.

If using a DXF, before exporting from the CAD program, the numbers must be set to three decimal places.

Also, when importing DXF or DWG files, the unit of measurement must be set to metres beforehand.

To import DWF or DWG files into the program's native format, these steps shall be followed:

1. Select the **Edit template** icon from the toolbar. The *DXF-DWG Template views manager* window will open (Figure 14).

2. Click the **Add** icon. The **Selection of templates to read** window will open (Figure 15) and the selection of a DXF or DWG file will be requested. Search for the file, select it, and click on **Open**.

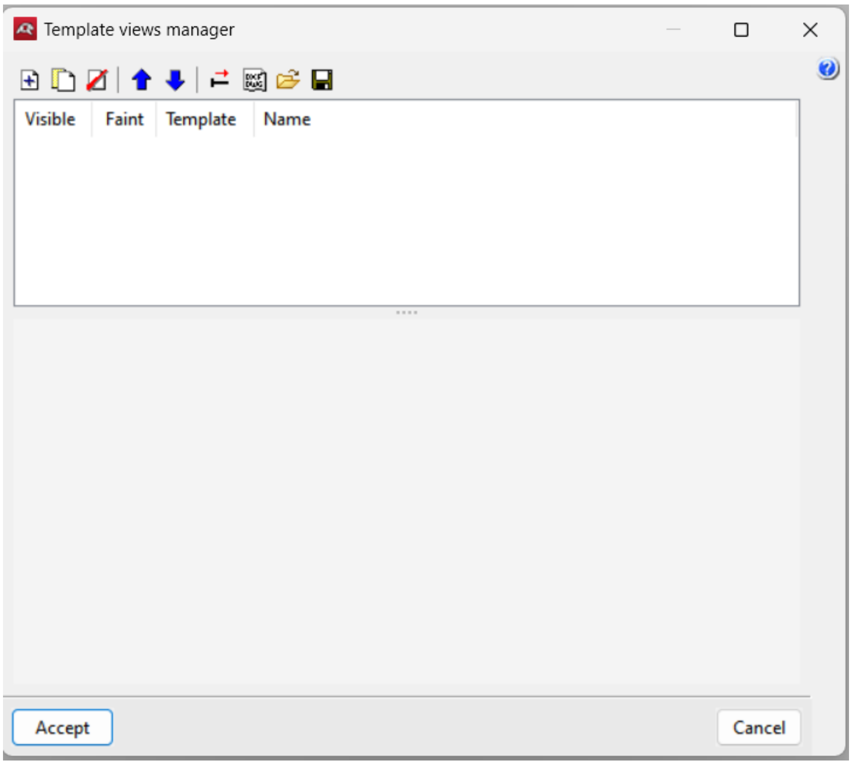


Figure 14

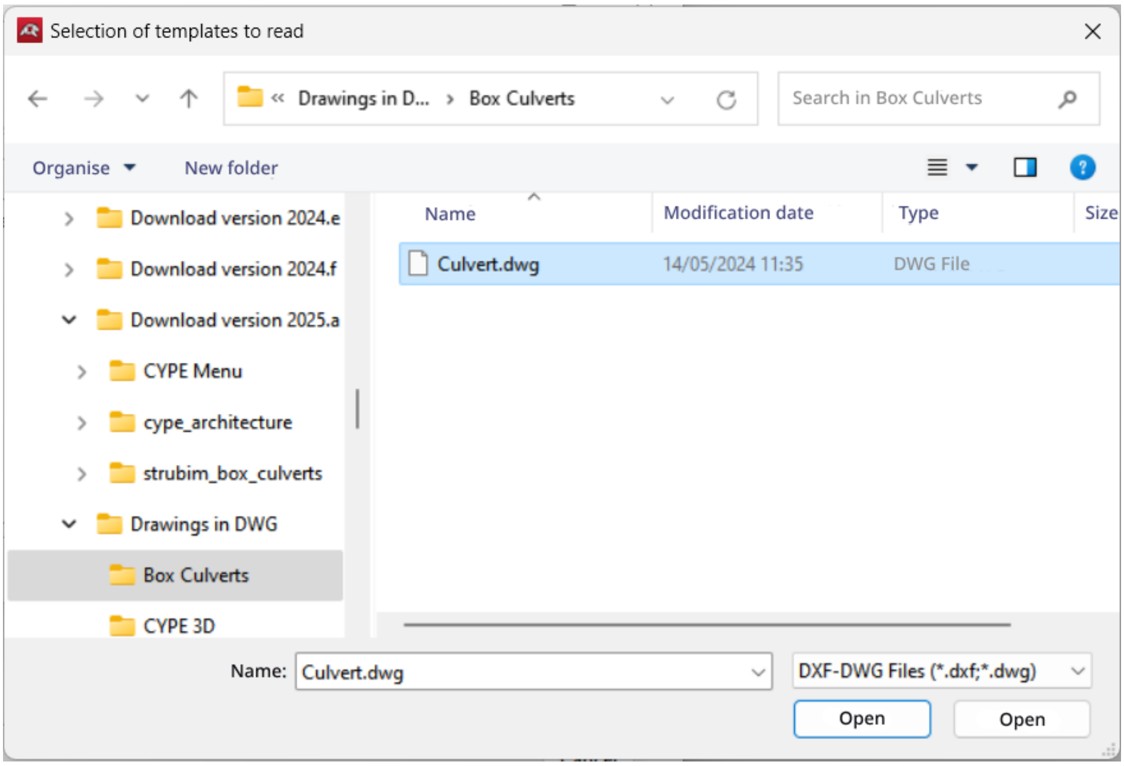


Figure 15

3. Click **Accept** on the *Available Files* windows to return to the *Template views manager* window and click on **Accept** again to view it on screen.

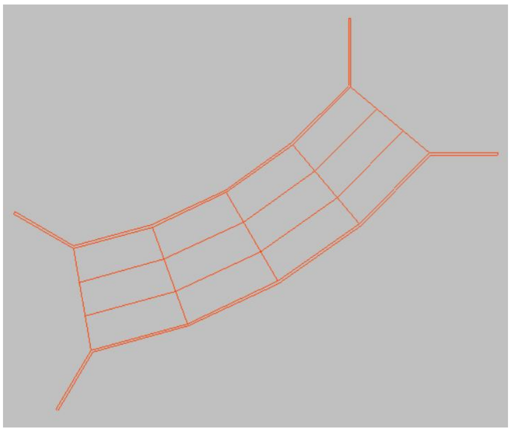


Figure 16

For using the Snaps, click on **Template Object Snaps** in the tool bar and activate **Endpoint** and **Intersection,** for instance.

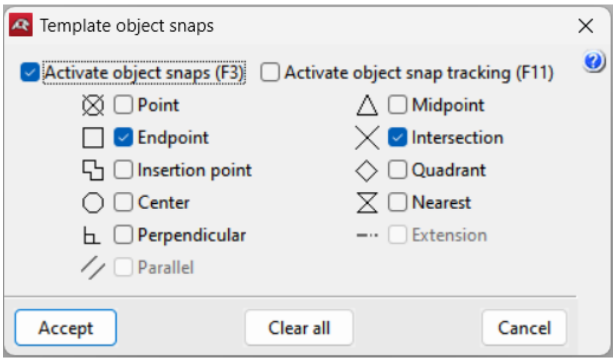


Figure 17

Note: *If the box culvert plan is generated with the DWG or DXF in the background, this will appear in the floor plan.*

With the **New left lateral wall point** option, the points that constitute the left lateral walls can be entered. Each subsequent point will create a new lateral wall in relation to the previous point. Therefore, the entry should start with the entry of the end of the initial left wingwall, continue with the left lateral walls of the modules, and finish with the end of the final left wingwall.

In the following figure (Figure 18), the template can be seen in the background in **orange**, and, in **blue**, a first line is shown representing the interior face of the initial left wingwall, followed by four more lines representing the interior face of the left lateral walls of the modules, and finally, a line representing the interior face of the final left wingwall.

Figure 18

To insert the right wingwall, the **New right lateral wall point** option shall be used.

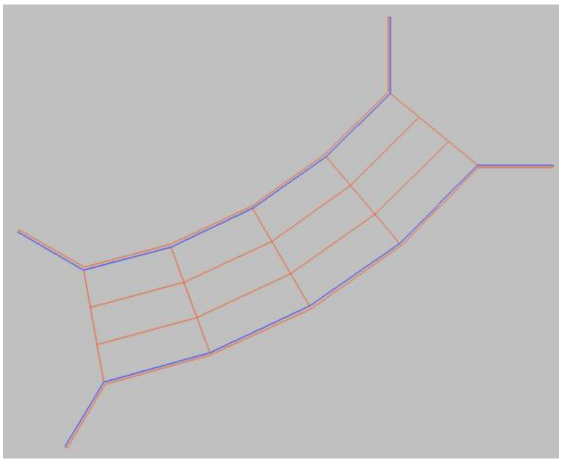


Figure 19

The **New joint/free edge** option is used for indicating where the module's transverse faces are. To do so, click on a point of a lateral wall, followed by the opposite point on the other lateral wall.

After entering two joints, the corresponding module is generated. Therefore, the joints should only be entered once the left and right lateral walls are defined.

Next, the input of the first joint is shown in Figure 20.

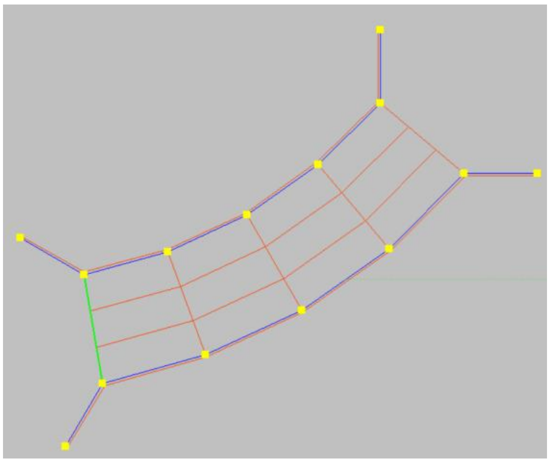


Figure 20

After entering the second joint, the program then creates the first module (where the thicknesses of the lateral walls can be seen) and the wingwalls (Figure 21).

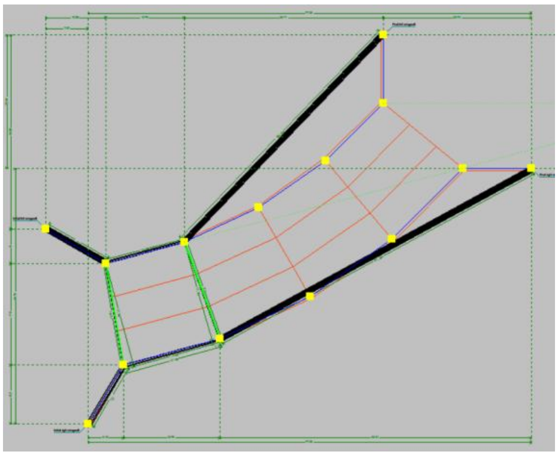


Figure 21

After entering the third joint, the second module is generated (Figure 22).

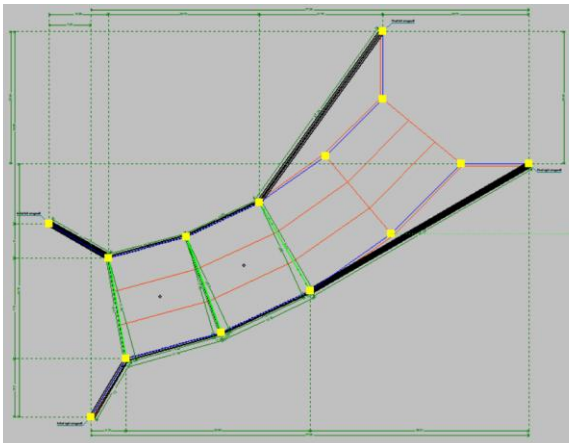


Figure 22

The process is repeated for the third module (Figure 23).

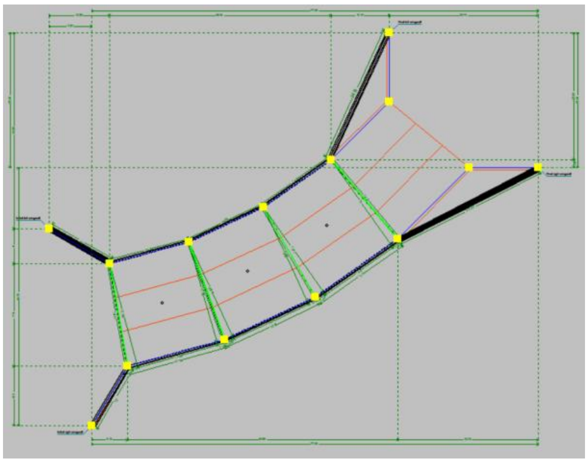


Figure 23

Finally, the insertion of the last joint completes the in-plan insertion (Figure 24).

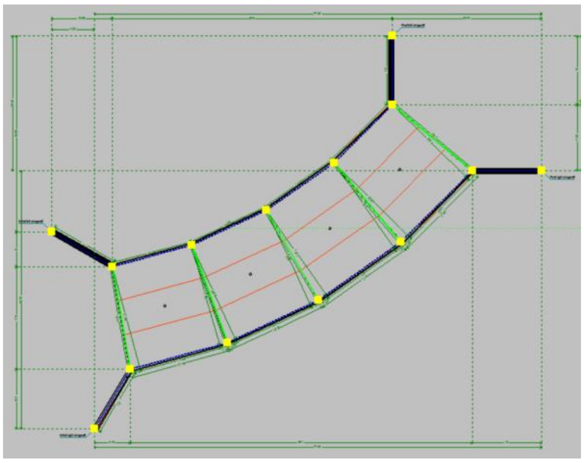


Figure 24

***Note:** If a lateral wall has been entered in one direction, and the opposite one in the other direction, the joints or geometry cannot be generated.*

3.4 Elevations

The **Elevations** options in the toolbar allow users to manage the vertical elevation profiles of the box culvert design. These tools allow adding, moving, editing, and deleting elevation points providing control over the box culvert's vertical alignment.

To add elevation points, select the **New** tool and then click on the desired point in the box culvert. The *New level point* window will open (Figure 25), where a Reference Name can be entered, as well as Top and Bottom level elevation points. To change a value, select the **Introduced** option.

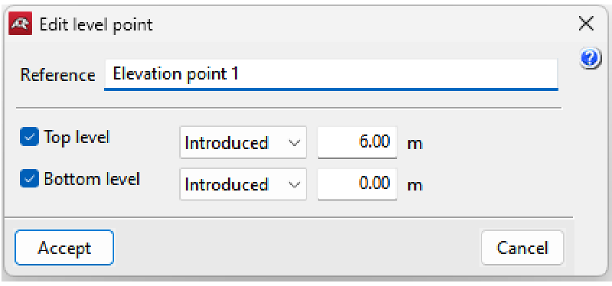


Figure 25

If different elevation point values are assigned in the modules (Figure 26), a slope in the top surface will be automatically generated, as seen in Figure 27.

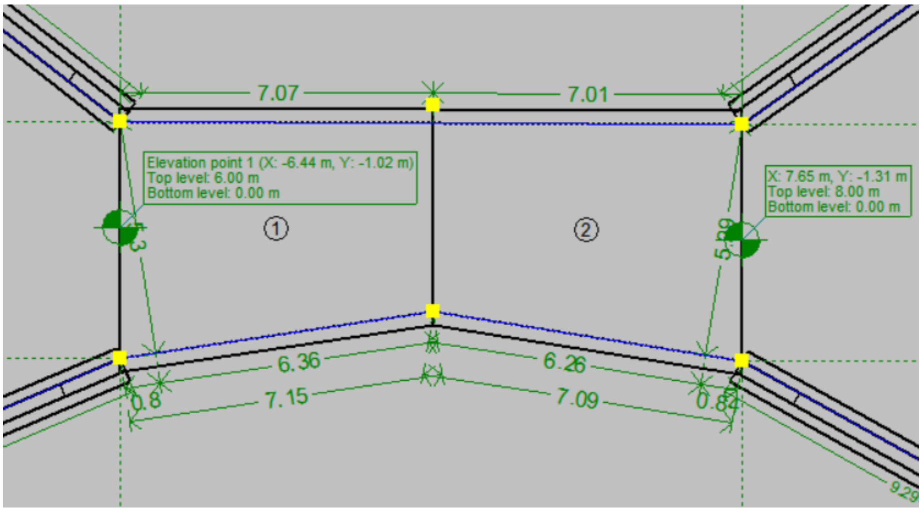


Figure 26

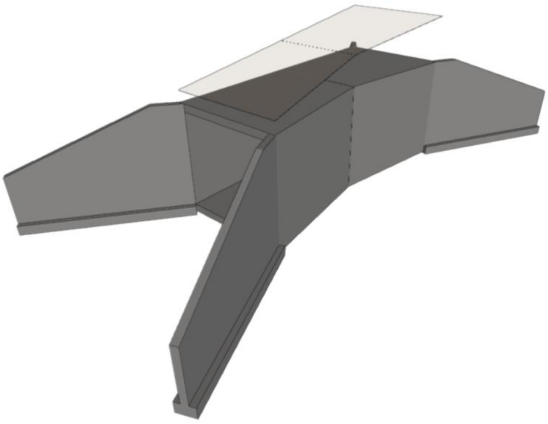


Figure 27

CAUTION: *When entering elevation values for the points, compatibility between the top plane and the bottom plane needs to be ensured to avoid issues with the geometry generation. To check this, the 3D View should be checked right after the modification has been made.*

3.5 Geometry

If needed, the geometry of modules and wingwalls may be manually and independently modified by clicking the **Edit** tool under the **Geometry** option in the toolbar.

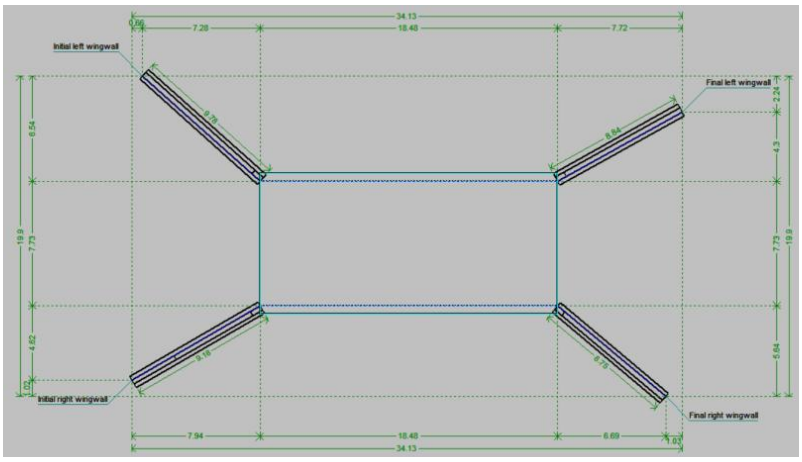


Figure 28

3.5.1 *Modules*

If a module is selected (Figure 28), its composing elements' thicknesses, such as left and right lateral walls and tops and bottom slabs, may be set under the *Thicknesses* section (Figure 29).

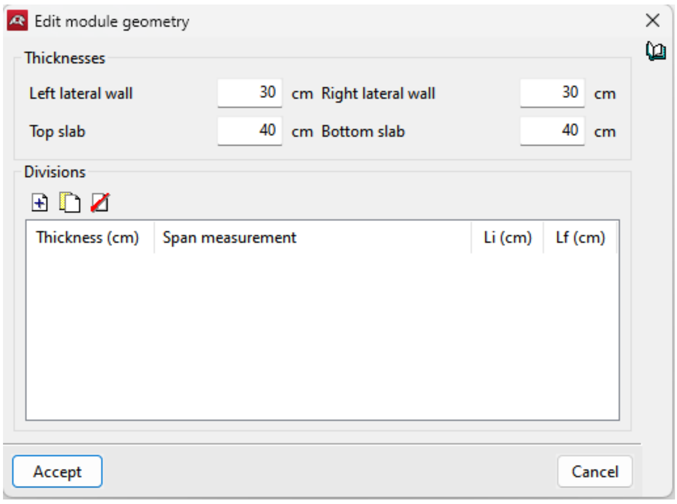


Figure 29

Additionally, divisions in the module may be added if needed, under the *Divisions* section, by specifying their thickness, Span measurement (Perpendicular to the left wall or Over the joint), and Initial (Li) and Final (Lf) distances (Figure 30).

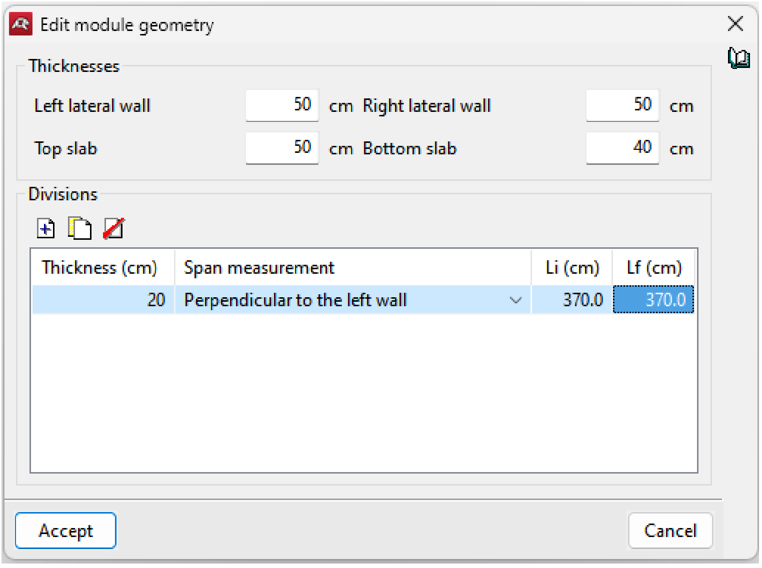


Figure 30

The inserted divisions may be configured for each module independently and may be seen both in the *Floor Plan View* represented with dotted lines (Figure 31) and in the *3D View* (Figure 32).

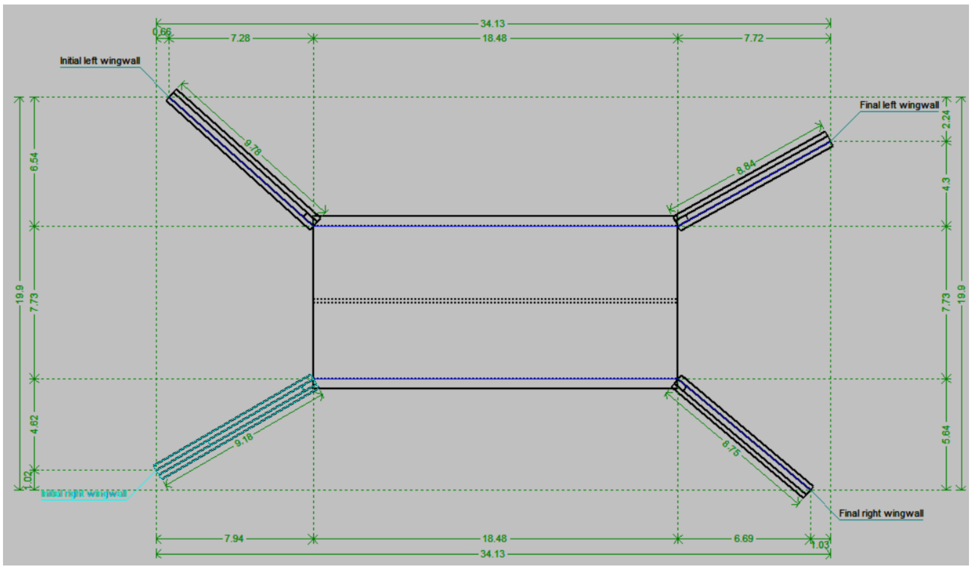


Figure 31

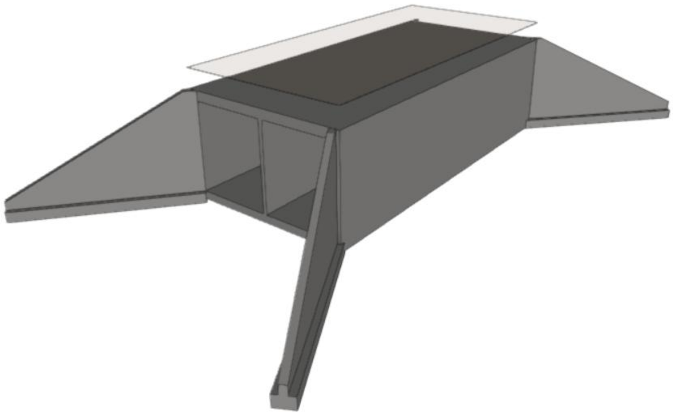


Figure 32

3.5.2 *Wingwalls*

When editing geometry, if a wingwall is selected, the *Edit wingwall geometry* window will open. There, the walls' Length (l), Top length (ls), Depth at end (c), Height (h), End extension length (a), and Thickness may be set. If the checkboxes are unselected –when applicable- their default values will be automatically assigned and greyed out (Figure 33). Also, the Ground surcharge on the backfill may be configured.

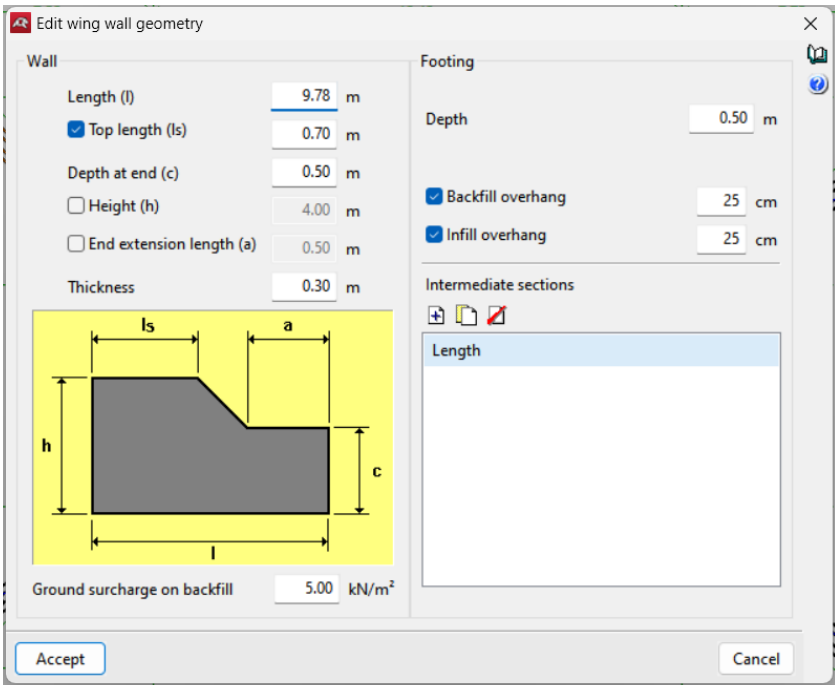


Figure 33

Additionally, the wingwalls' **Footing** may be also configured regarding its Depth, Backfill overhang, and Infill overhang. Furthermore, intermediate sections of the footing may be configured manually, to adjust and optimise their structural behaviour.

The desired number of intermediate sections may be added, and configured regarding their lengths, and backfill and infill overhangs (Figure 34).

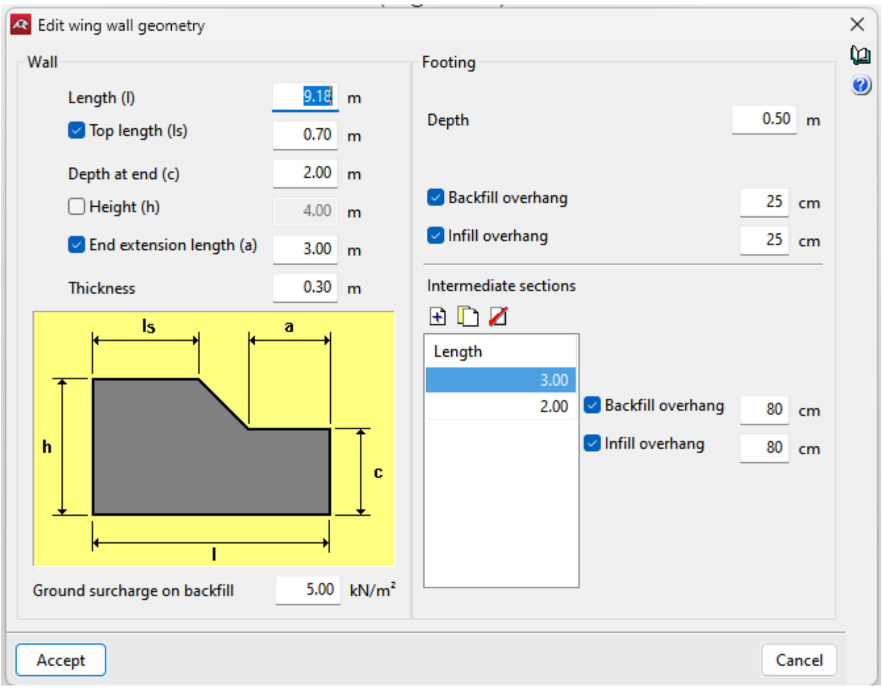


Figure 34

Finally, the *Footing's intermediate* section may be seen both in *Floor Plan* (Figure 35) and in *3D View* (Figure 36).

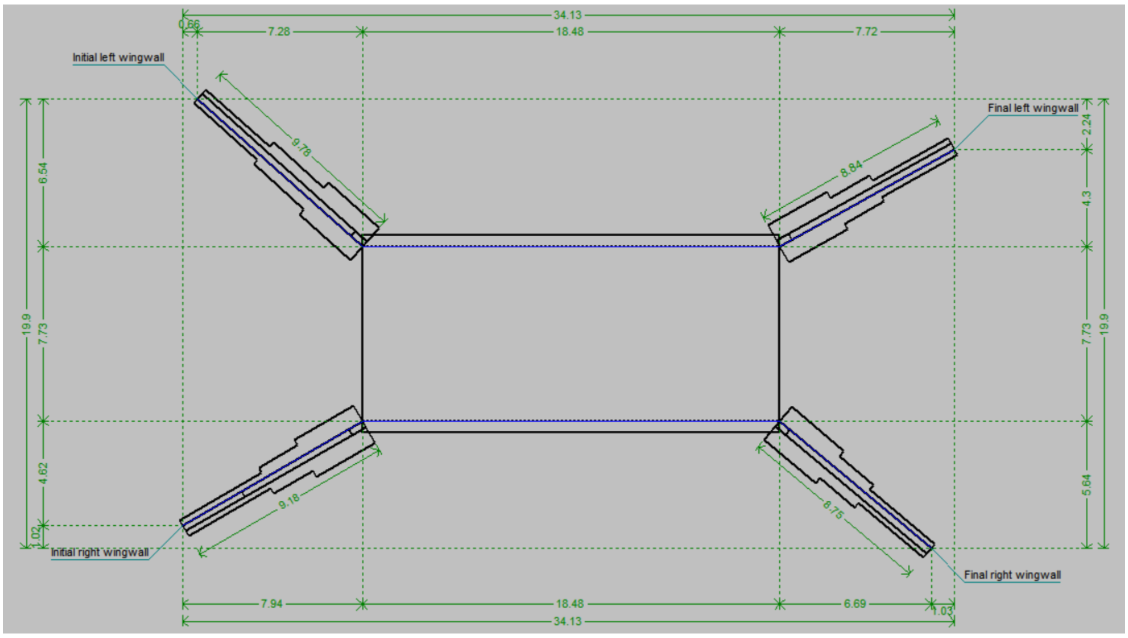


Figure 35

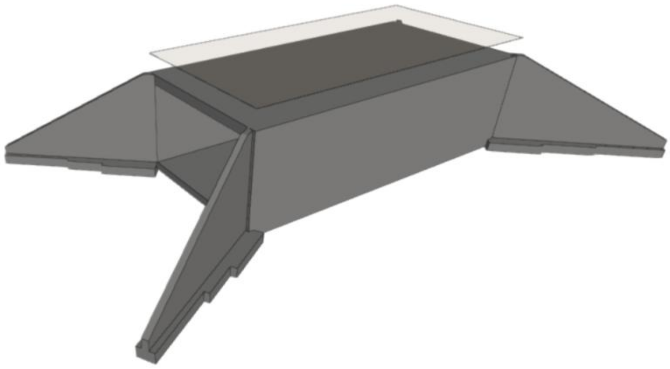


Figure 36

3.6 Loads

Loads are managed manually and independently and may be added as truck or strip loads. They can be also edited, moved, or deleted in their corresponding menus.

3.6.1 *Truck loads*

To create truck loads, under the *Truck loads* drop-down menu, select the **New** option. Then, enter the direction of the load by clicking a start point and an end point. After that, the *New load trucks* window will open (Figure 37).

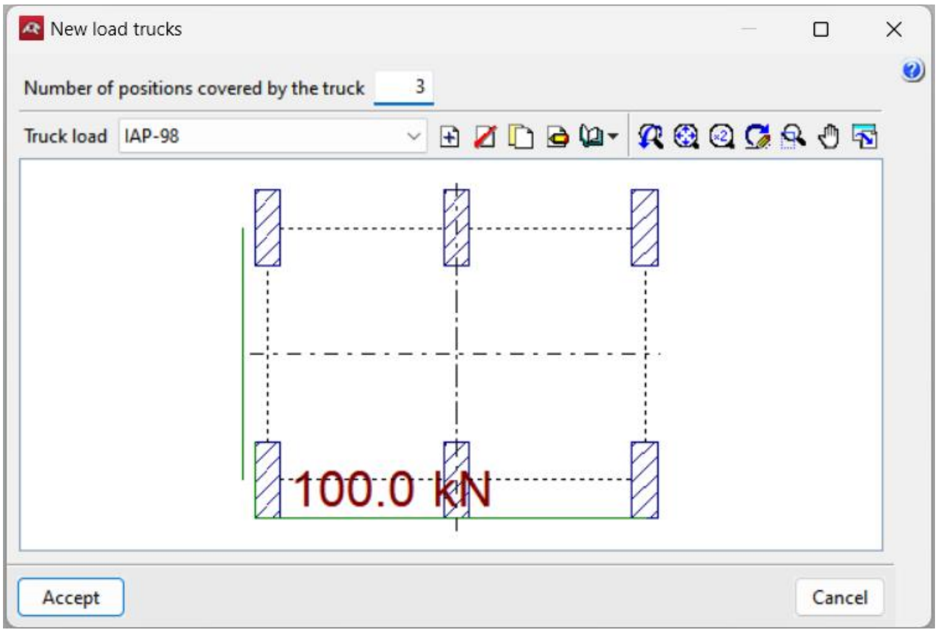
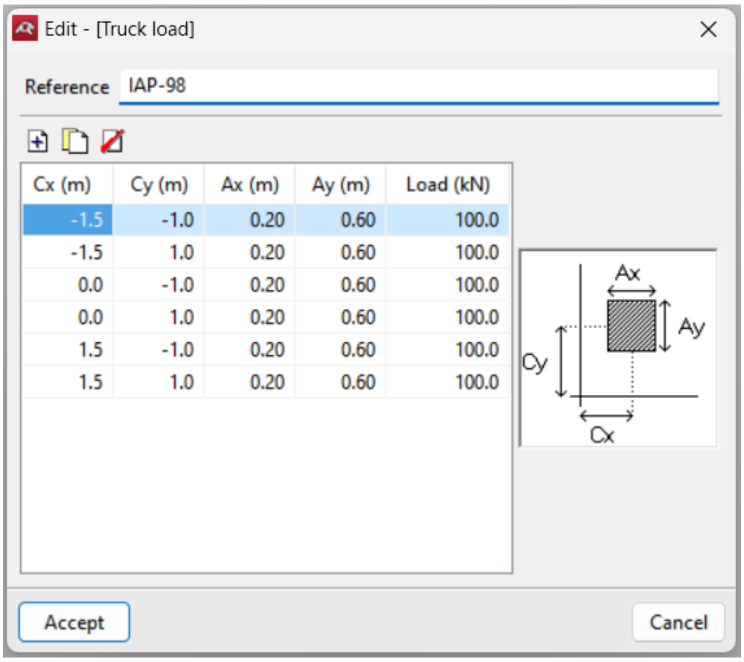


Figure 37

There, settings such as the number of positions covered by the truck and the truck load itself may be configured.

The type of truckload can be selected from the drop-down menu. By default, the IAP-98 is selected. Nevertheless, existing truck loads may be deleted, copied, and edited. Additionally, new ones can be added by clicking on the **Create** button.

In the *Create – [Truck load]* window, the coordinates (Cx and Cy) relative to the insertion position, the dimensions (Ax and Ay), and the loads shall be entered. Loads can also be copied and deleted, and then configured separately (Figure 38).



Cx (m)	Cy (m)	Ax (m)	Ay (m)	Load (kN)
-1.5	-1.0	0.20	0.60	100.0
-1.5	1.0	0.20	0.60	100.0
0.0	-1.0	0.20	0.60	100.0
0.0	1.0	0.20	0.60	100.0
1.5	-1.0	0.20	0.60	100.0
1.5	1.0	0.20	0.60	100.0

Figure 38

After clicking on **Accept** twice, the designed truck load will be displayed in Floor Plan, using the initial start and end point as a reference line (with its corresponding coordinates displayed), and the number of positions covered along that line.

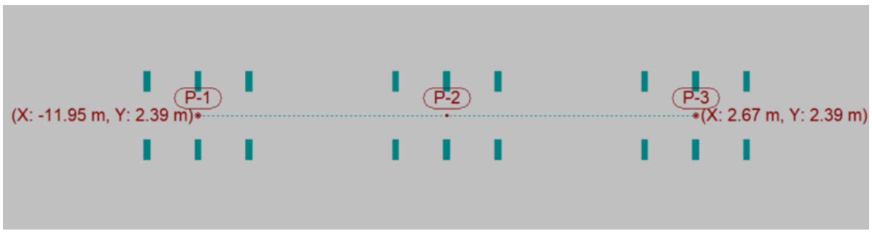


Figure 39

3.6.2 *Strip loads*

Strip loads can be created by selecting the **New** option under Strip loads in the tool bar. Similarly to truck loads, the direction of the load shall be entered first by selecting a start point and an end point.

After doing so, the *New strip loads* window will open, where the Width and the Load are to be entered, as seen in Figure 40.

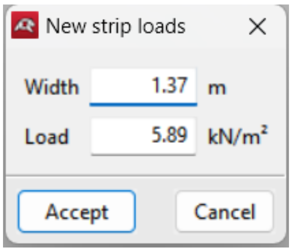


Figure 40

After clicking on **Accept**, the configured strip load will be visible in the Floor Plan, where the start and end point coordinates can be seen (Figure 41).

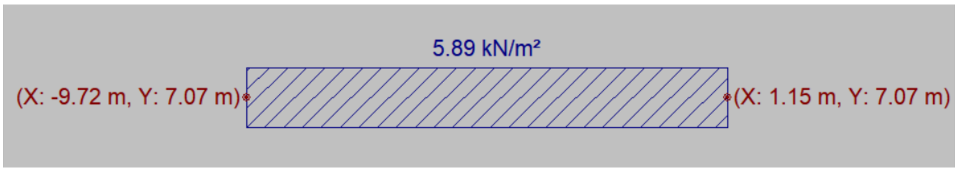


Figure 41

Additionally, existing Strip loads may be moved, edited and deleted with the corresponding options under the **Strip Load** options in the tool bar.

3.7 Analysis

The *Analysis* menu in the program includes several tools for evaluating and designing box culverts:

- The **Check** tool performs an analysis of displacements, forces, and reinforcement layout. Any failed elements are displayed in red in the Floor Plan allowing users to view carried out checks (Figure 42). Also, a report is generated showing the checks carried out (Figure 43).

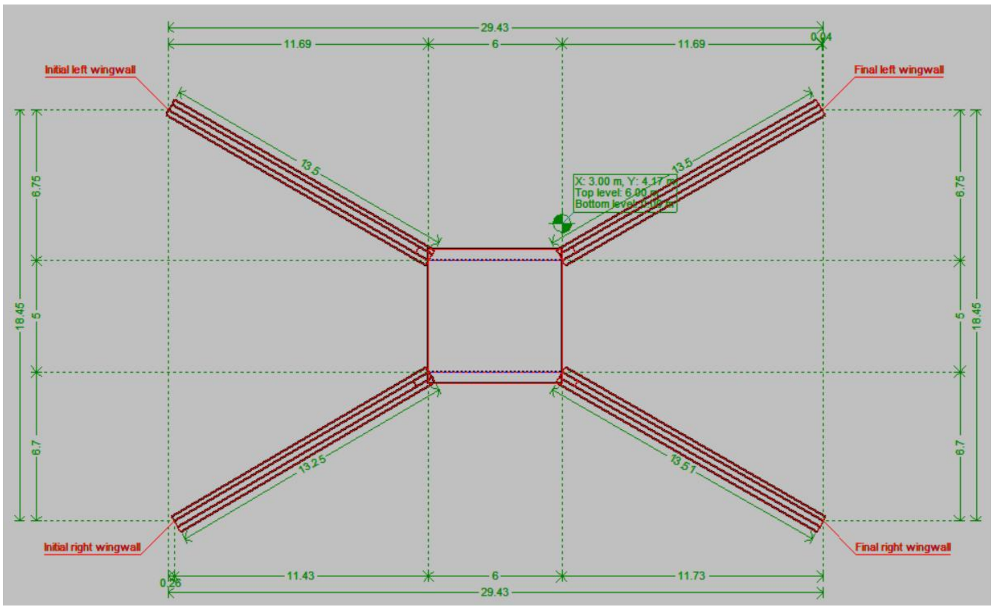
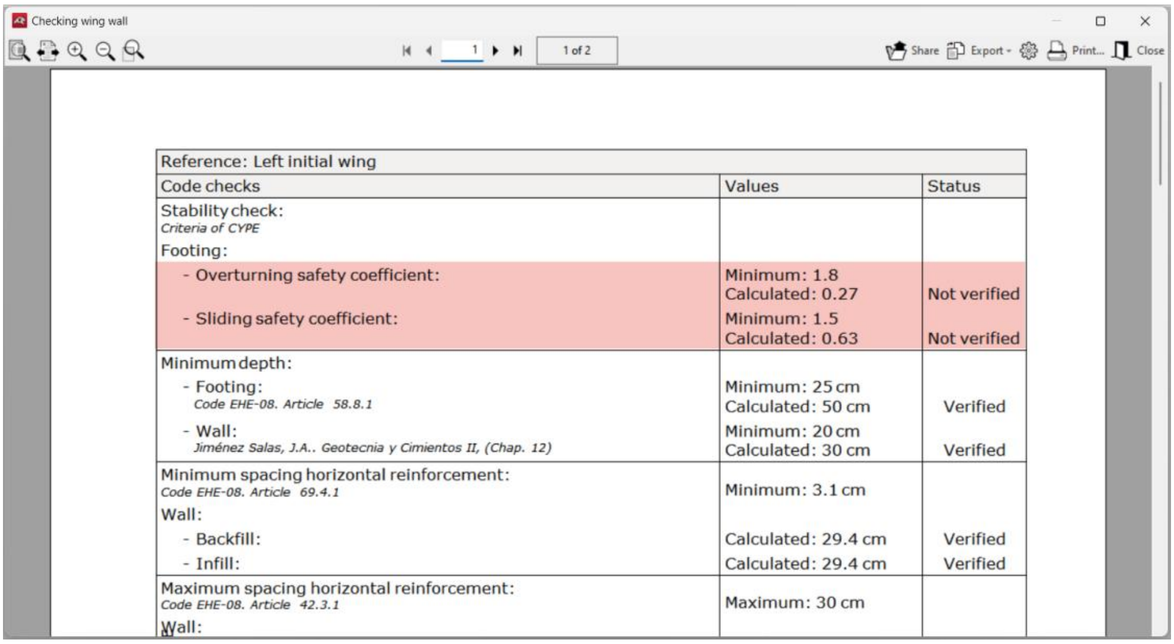


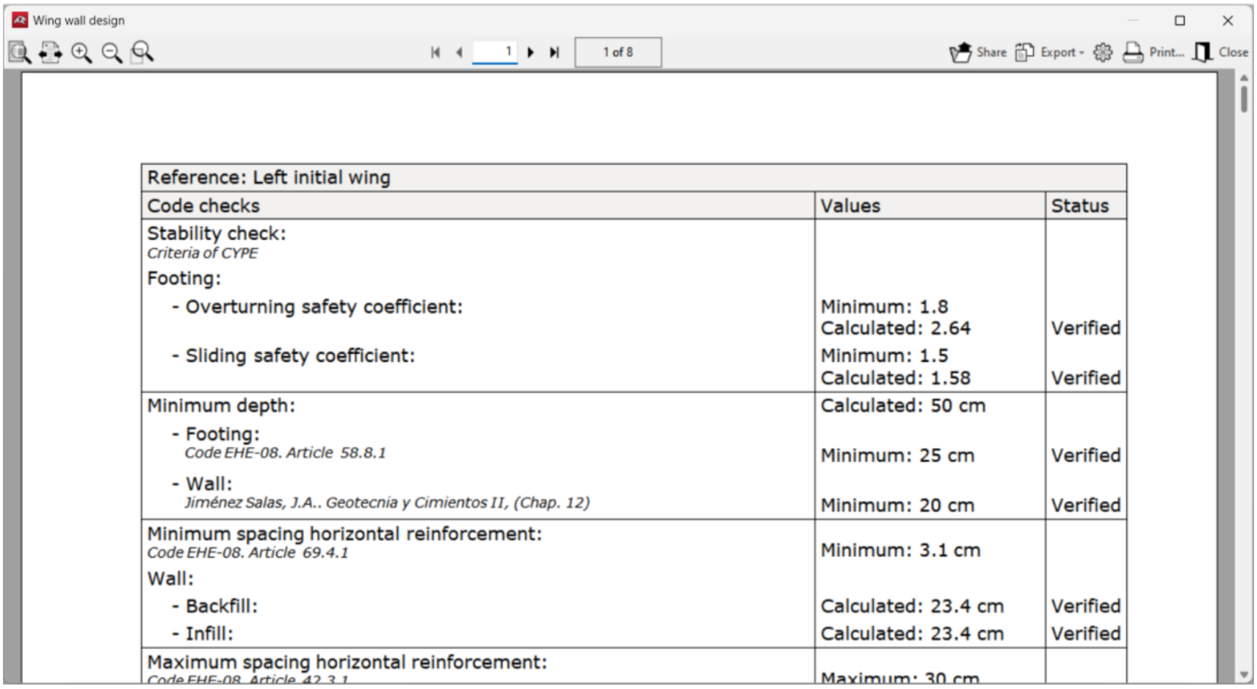
Figure 42



Reference: Left initial wing		
Code checks	Values	Status
Stability check: *Criteria of CYPE* Footing:		
- Overturning safety coefficient:	Minimum: 1.8 Calculated: 0.27	Not verified
- Sliding safety coefficient:	Minimum: 1.5 Calculated: 0.63	Not verified
Minimum depth:		
- Footing: *Code EHE-08. Article 58.8.1*	Minimum: 25 cm Calculated: 50 cm	Verified
- Wall: *Jiménez Salas, J.A.. Geotecnia y Cimientos II, (Chap. 12)*	Minimum: 20 cm Calculated: 30 cm	Verified
Minimum spacing horizontal reinforcement: *Code EHE-08. Article 69.4.1*	Minimum: 3.1 cm	
Wall:		
- Backfill:	Calculated: 29.4 cm	Verified
- Infill:	Calculated: 29.4 cm	Verified
Maximum spacing horizontal reinforcement: *Code EHE-08. Article 42.3.1*	Maximum: 30 cm	
Wall:		

Figure 43

- The **Design** tool allows users to design selected elements, such as modules or wingwalls, analysing displacements and forces and designing reinforcements. A report is generated, where failed elements are shown in red (Figure 44). Additionally, while module thicknesses remain unchanged, the thicknesses and foundation sizes of wingwalls are designed (Figure 45).



Reference: Left initial wing		
Code checks	Values	Status
Stability check: *Criteria of CYPE*		
Footing:		
- Overturning safety coefficient:	Minimum: 1.8 Calculated: 2.64	Verified
- Sliding safety coefficient:	Minimum: 1.5 Calculated: 1.58	Verified
Minimum depth:	Calculated: 50 cm	
- Footing: *Code EHE-08. Article 58.8.1*	Minimum: 25 cm	Verified
- Wall: *Jiménez Salas, J.A.. Geotecnia y Cimientos II, (Chap. 12)*	Minimum: 20 cm	Verified
Minimum spacing horizontal reinforcement: *Code EHE-08. Article 69.4.1*	Minimum: 3.1 cm	
Wall:		
- Backfill:	Calculated: 23.4 cm	Verified
- Infill:	Calculated: 23.4 cm	Verified
Maximum spacing horizontal reinforcement: *Code EHE-08. Article 42.3.1*	Maximum: 30 cm	

Figure 44

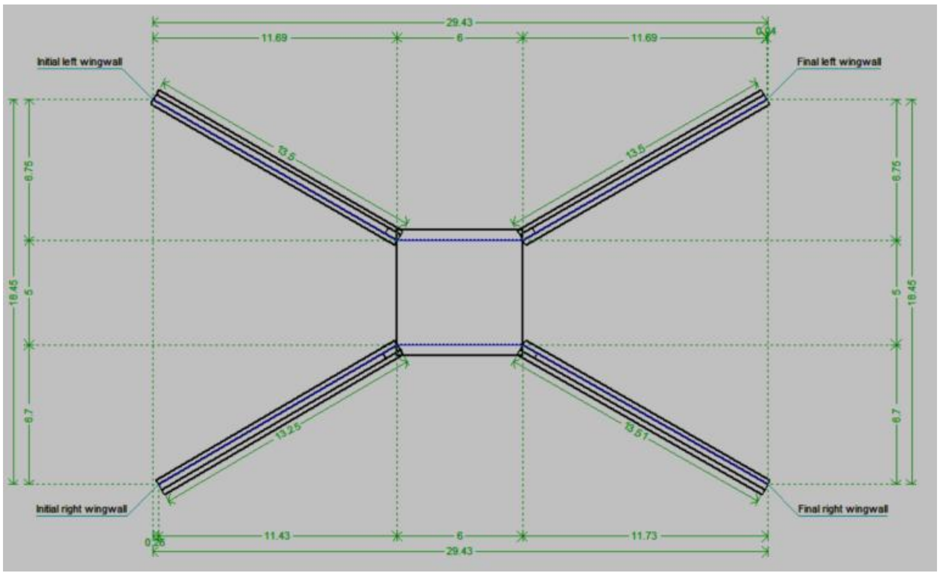
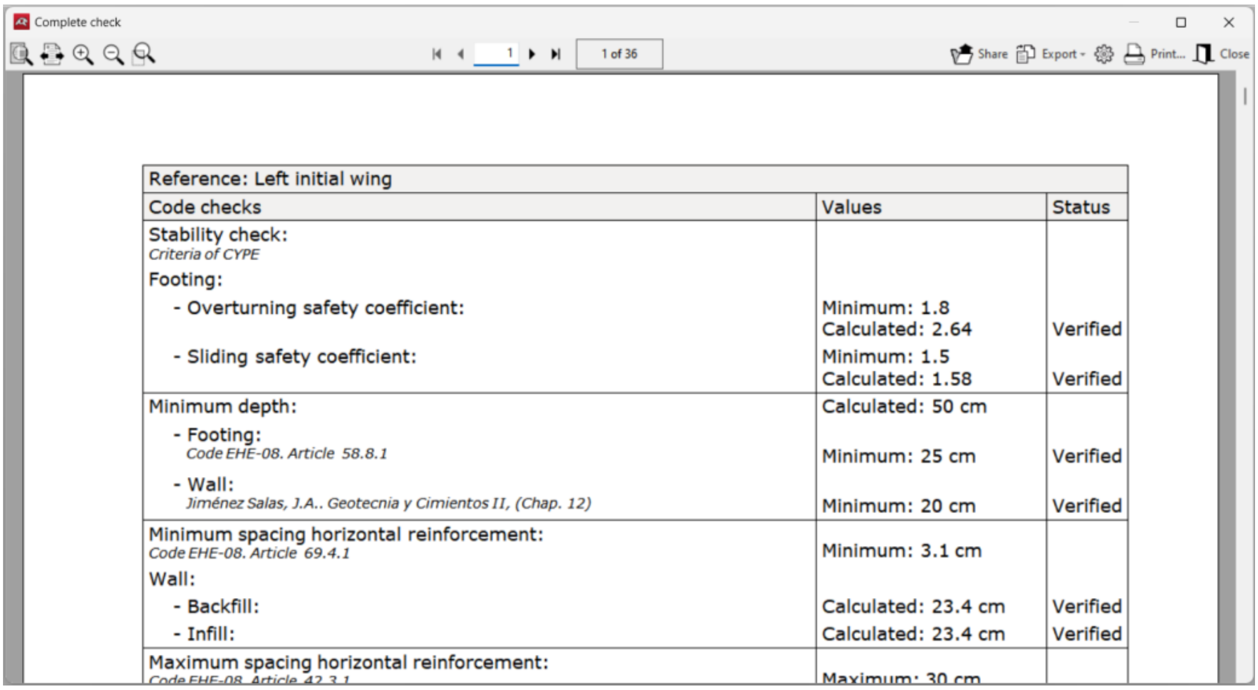


Figure 45

- The **Design all** option carries out the design and checks for all elements of the box culvert. A complete report with checks on all the elements of the box culvert is generated (Figure 46). Additionally, the complete geometry of the box culvert's composing elements is automatically redefined until the standards are met (Figure 47).


Complete check

1 of 36

Share Export Print... Close

Reference: Left initial wing		
Code checks	**Values**	**Status**
Stability check: *Criteria of CYPE* Footing: - Overturning safety coefficient: - Sliding safety coefficient:	 Minimum: 1.8 Calculated: 2.64 Minimum: 1.5 Calculated: 1.58	 Verified Verified
Minimum depth: - Footing: *Code EHE-08. Article 58.8.1* - Wall: *Jiménez Salas, J.A.. Geotecnia y Cimientos II, (Chap. 12)*	Calculated: 50 cm Minimum: 25 cm Minimum: 20 cm	 Verified Verified
Minimum spacing horizontal reinforcement: *Code EHE-08. Article 69.4.1* Wall: - Backfill: - Infill:	Minimum: 3.1 cm Calculated: 23.4 cm Calculated: 23.4 cm	 Verified Verified
Maximum spacing horizontal reinforcement: *Code EHE-08. Article 42.3.1*	Maximum: 30 cm	

Figure 46

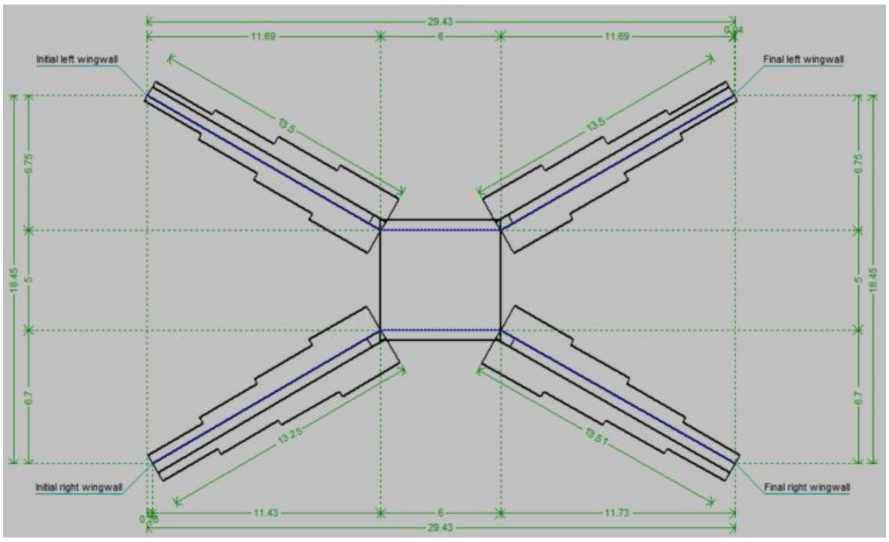


Figure 47

The main advantage of the generated reports is that they identify which standards are not met by specific elements of the box culverts. This allows users to modify the structure based on the exact criteria needed for compliance.

***Note:** After making modifications to elements, they should be redesigned using the **Design** or **Design all** option before selecting the **Check** option. This ensures that the changes are properly analysed and incorporated into the design, allowing for an accurate compliance check of the specific elements.*

- The **Forces** option shows a schematic representation of the module with contour maps indicating displacements. This section offers various display options, including a view of the structure, deformed shapes, or contour plots. Users can choose to visualise displacements (XYZ, total and rotation) and forces (axial, moment and shear). Additional settings allow displaying the shell analysis mesh, axes of the shells, and setting minimum and maximum values for the selected loadcase (Figure 48).

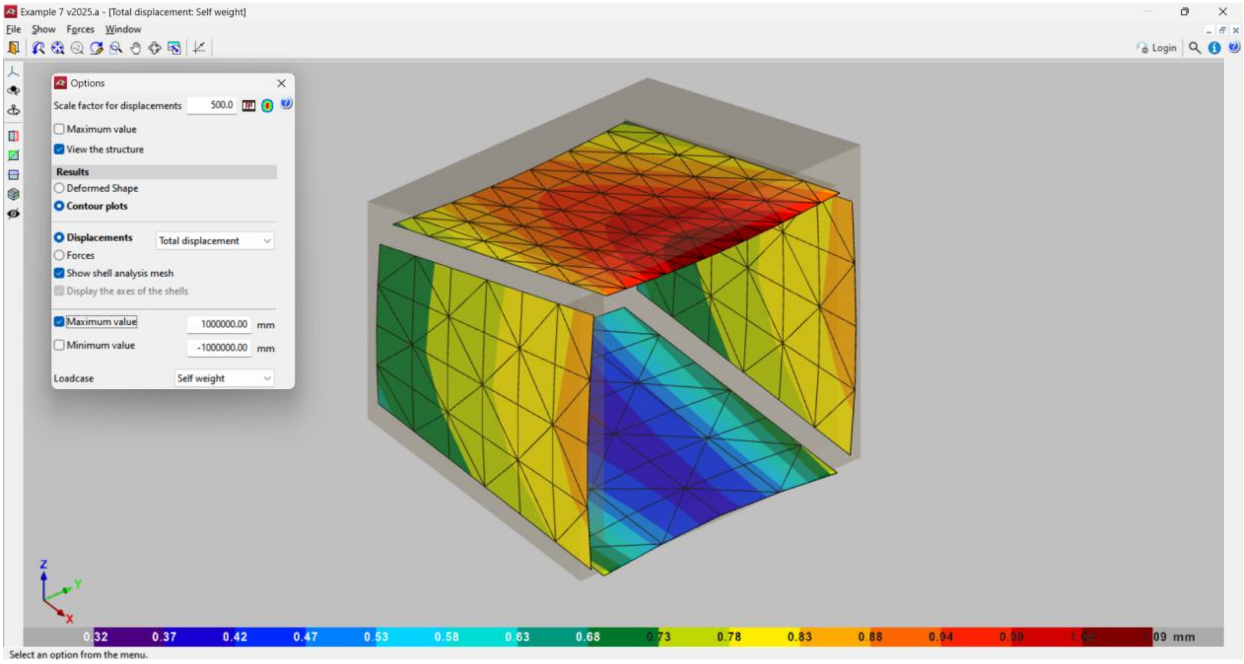


Figure 48

3.8 Reinforcements

In the Reinforcements tab (bottom left corner of the interface), the detailed schematics of the box culvert's reinforcement layout are displayed. This section is organised into categories regarding its parts (modules and several wingwalls). Each category includes subsections for different parts of the box culvert, showing the reinforcement details for each section. The schematic provides precise reinforcement specifications, including bar diameters, spacing and positions, allowing users to review and modify the reinforcement design as needed (Figure 49).

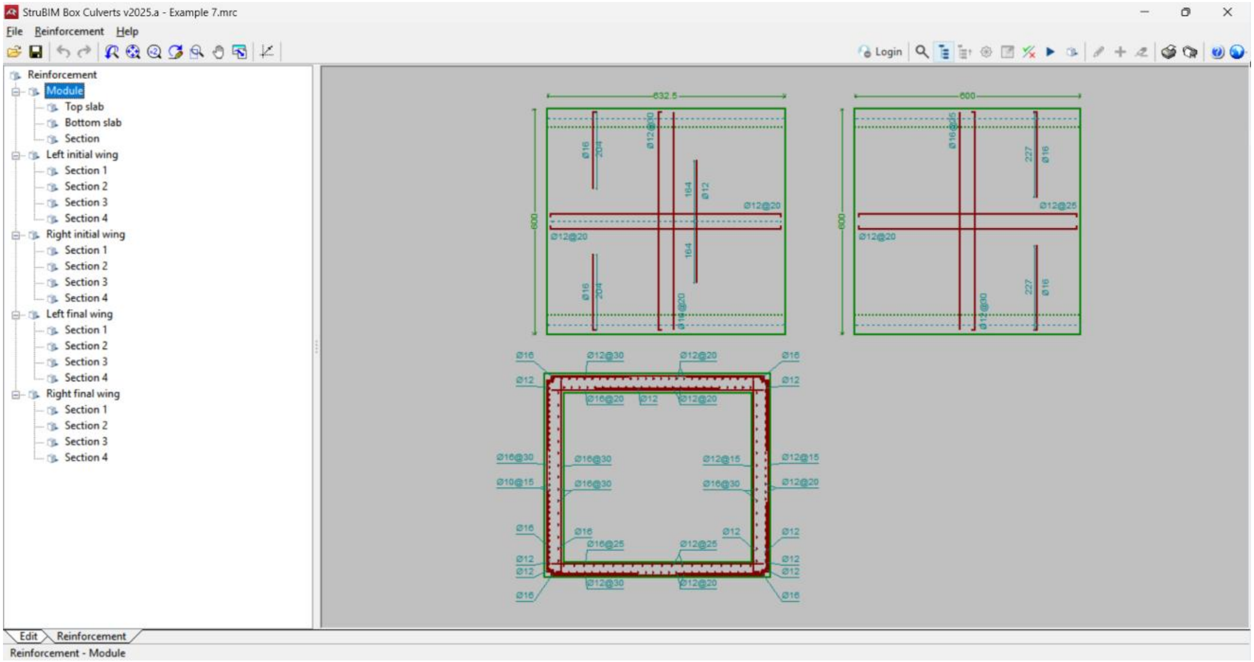


Figure 49

To modify reinforcements, de **Edit** option should be selected, under the *Reinforcements* menu in the toolbar. Then, the desired bar shall be selected and the corresponding properties will pop-up for modifications to be made (Figure 50).

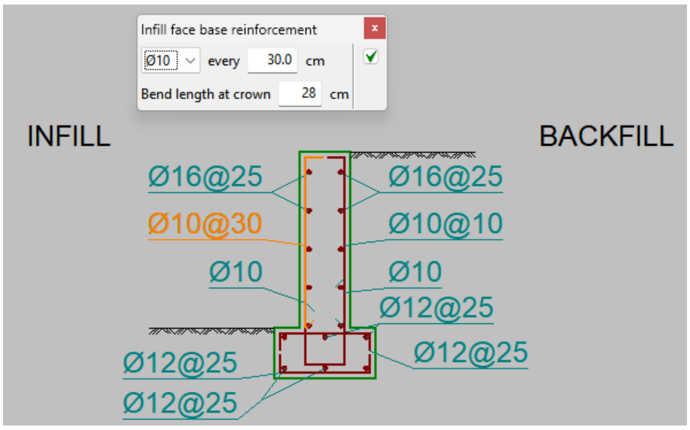


Figure 50

Finally, when clicking on the elements of the general categories of a layout, this will redirect users to the specific reinforcement layout or section.

3.9 Reports

To generate Reports, select the **Report** option under File, in the toolbar. This opens the *Report selection* window where Chapters can be included or excluded by checking their corresponding checkboxes (Figure 51).

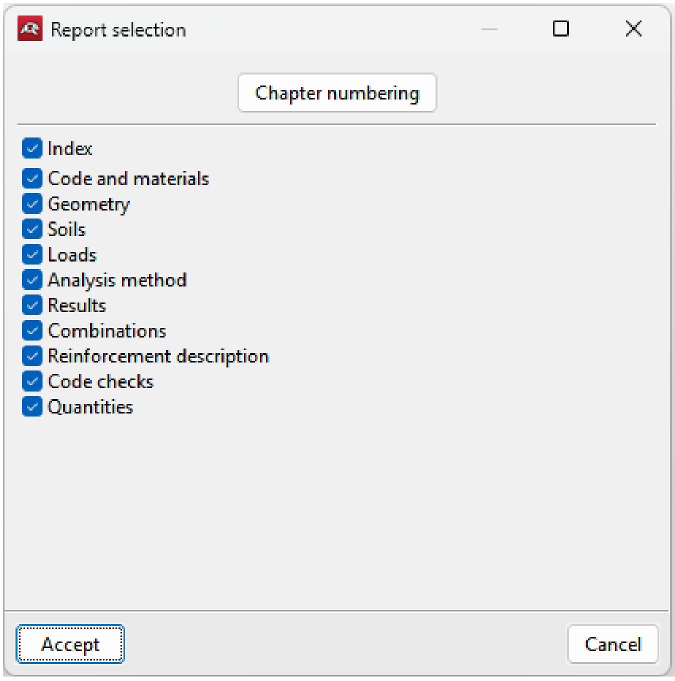


Figure 51

Additionally, the chapters' numbering can be configured by clicking on the **Chapter numbering** option and selecting the levels to be shown in the index and manually editing their numbers.

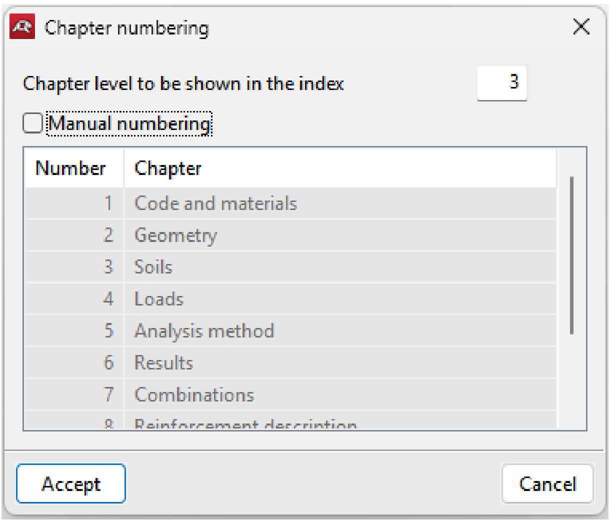


Figure 52

Reports can also be sent to the printer (with optional print preview, page setup, etc.) in HTML, PDF, RTF and TXT formats, after clicking on **Accept** on the *Report Selection* window (Figure 53).

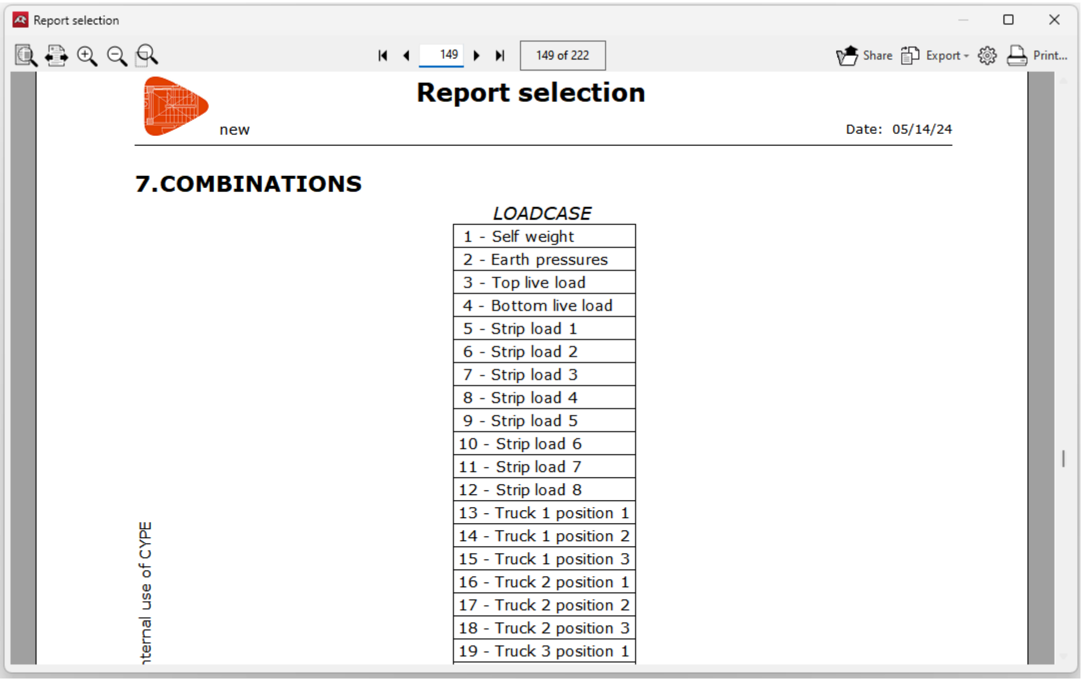


Figure 53

3.10 Drawings

To create drawings, select the **Drawings** option under File in the toolbar. The *Drawing selection* window will open (Figure 54).

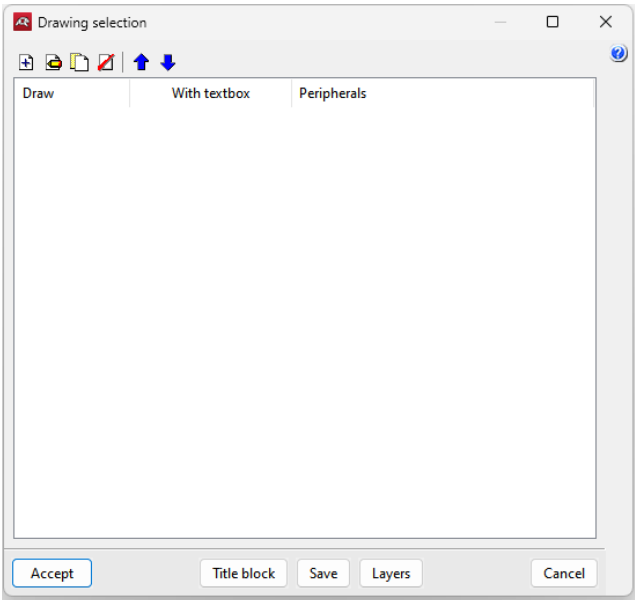


Figure 54

After clicking the **Add** button, the *Drawing editor (Box culverts)* menu will pop-up (Figure 55).

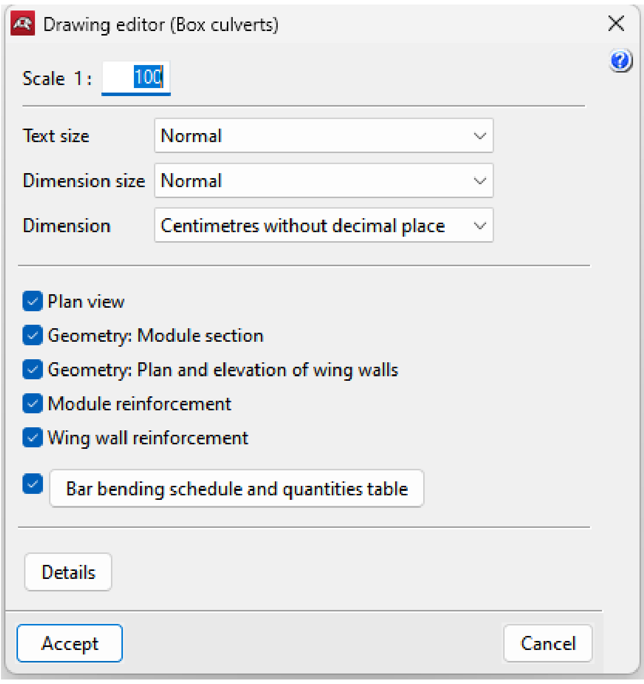


Figure 55

There, the following operations can be carried out for the generation of drawings:

- Changing the scale of the drawing.
- Graphical annotations, such as Text and, Dimension size, and Dimensions unit format.
- Adding one or several drawings to be printed simultaneously with that configuration, by checking and unchecking the desired ones.
- Including previously imported details and manage them accordingly.

After configuring the desired drawings to be included in the layouts, and clicking on **Accept**, other options in the *Drawing selection* window may be set, such as including or excluding textboxes and peripherals (printing machine) (Figure 56).

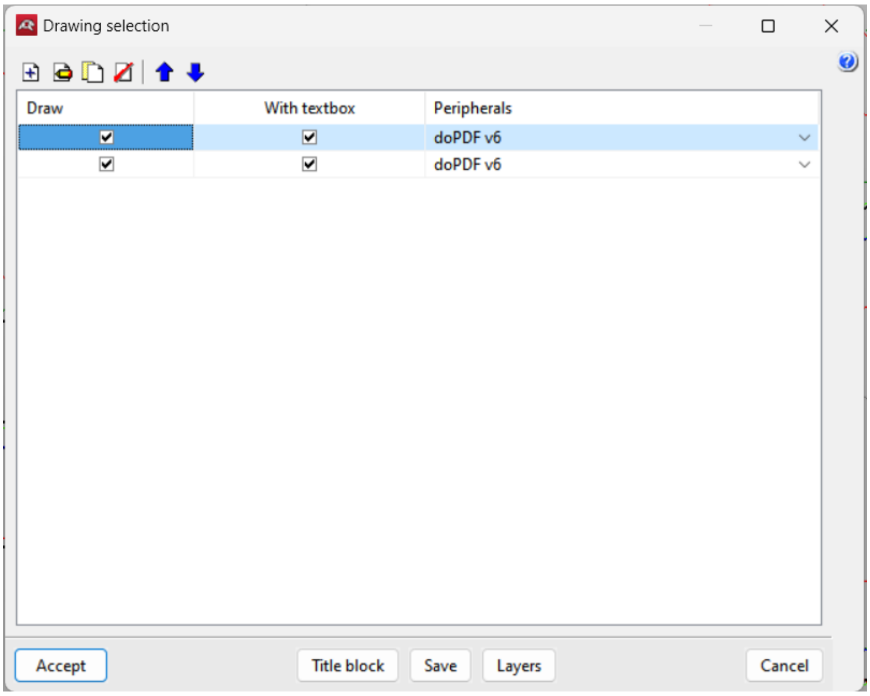


Figure 56

Title blocks can also be managed in this menu, by clicking on the **Title block** option and configuring them in the *Title block selection* window (Figure 57). Additionally, layer definition may also be carried out under Layers.

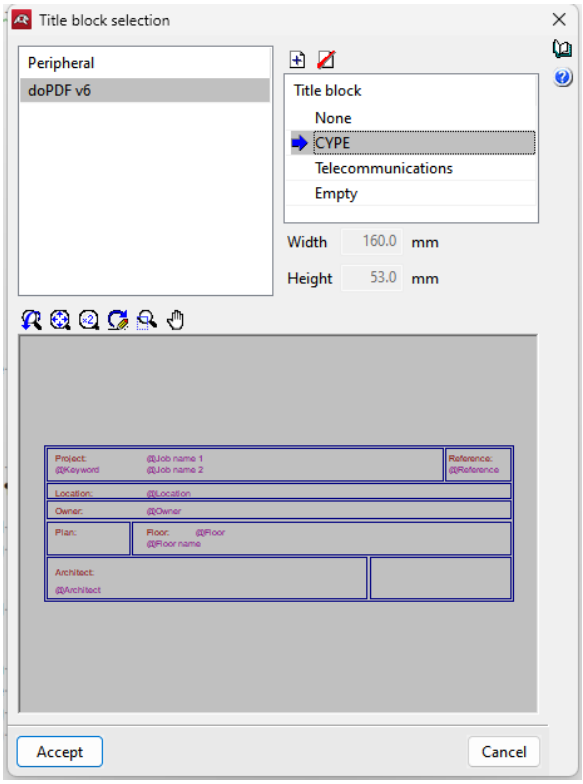


Figure 57

When clicking on **Accept** in the *Drawing selection* window, the *Drawing layouts* window will open. There, all the generated layouts can be managed with the following options:

- **New drawing**: Creates a new a blank layout to move objects into.
- **Delete empty spaces**: Deletes drawings with no objects
- **Centre all drawings**: Restores the original position of all objects, in all drawings.
- **Centre selected drawings**: Restores the original position of the objects moved by the user, after selecting them.
- **Edit object**: Allows users to move the texts on a drawing.
- **Move object**: Allows users to move objects on a drawing or move them to other drawings.
- **Drawing detail**: Allows viewing the real layout of an object.
- **Detail of all the drawings**: Allows viewing the real layout of all the objects.
- **Print all** and **Print selection**: Allows users to print all the layouts or only the desired ones.

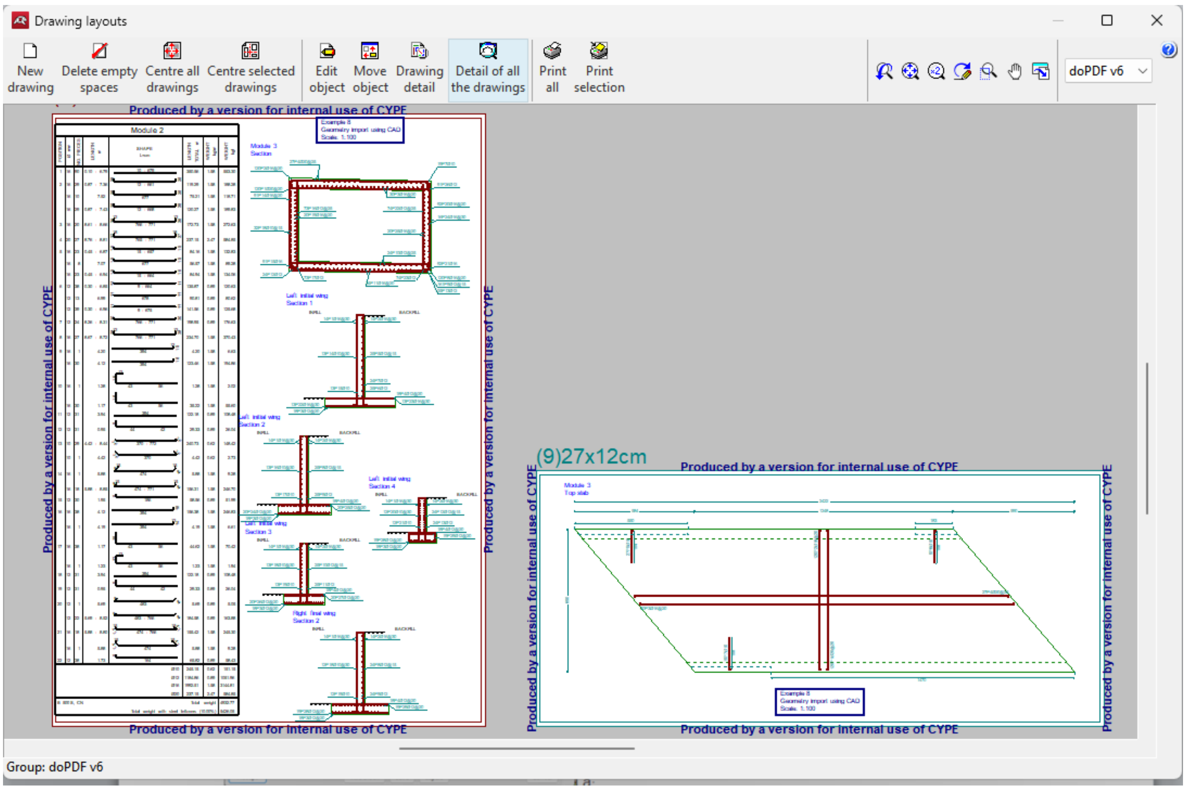


Figure 58