# Introduction

**RCBEAM** software designs reinforced concrete continuous beams according to the recommendations of **ACI 318-19 metric version** (imperial units not supported). It is a module of **STRES** structural design suite which offers complete solutions for buildings.

**RCBEAM** is conceived for Engineers who need a simple and efficient tool to generate complete calculations, design notes and drawings for continuous beams, in the most effortless way. **RCBEAM** is a must-have for any structural engineer, whether designing everyday beams or complicated projects.

The main features of **RCBEAM** are:

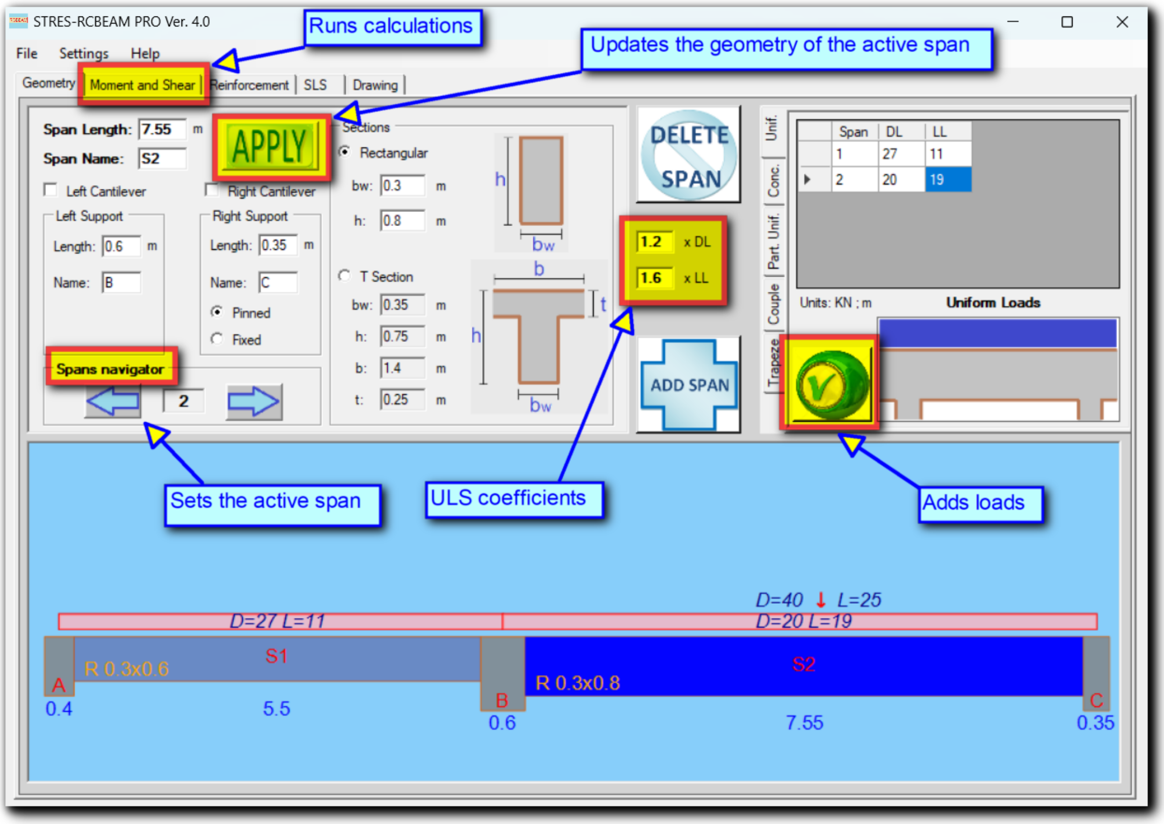
- Intuitive and compact graphical interface where ***beam geometry and loadings*** can be directly introduced in the same window and ***both graphically displayed***. This helps the user to introduce data's in few minutes without any risk of mistakes.
- All assumptions regarding materials, calculations options, etc… are grouped within ***one menu*** called "Settings". It is a kind of panel board controlling all design features.
- Advanced features like moment's redistribution and seismic dispositions are available.
- ***Strut-And-Tie method*** (Chapter 23 of ACI 318-19) is implemented to solve the issue that is often even not tackled by many other commercial programs regarding ***bars anchoring at supports***: the steel area required, anchoring length of the tension bars, compression stresses in struts…
- ***Confining reinforcement for hooked bars*** is designed as per section 25.4.3.2 of ACI 318-19 and reflected in both the calculation note and the drawings.
- ***Seismic dispositions*** for flexural and shear reinforcement as per section 18.4 of ACI 318-19.
- Determination of the flexural reinforcement as SLS ***considering cracks width limitations*** as per ACI 224.1R-07.
- Design of beams poured in two phases, with ***horizontal construction joint***, as per section 16.4 of ACI 318-19.
- ***Numerous warnings and errors pre-implemented messages*** ensure that from the geometry input procedure till the design note generation, the user is continuously notified about the Code limitations.
- Automatic display of all results (moments, reinforcement, deflection...) by the mean of clearly annotated and colored diagrams.

- ***Graphical visualization of the complete RC design of the beam*** (bars curtailment, diameters and lengths; stirrups diameters and spacing's…) allowing the user to control and modify the design before generating the design note or the CAD-compatible drawing.
- Calculation of the long-term cracked deflection with the possibility to modify the reinforcement in order to improve the deflection.
- Calculation of the stresses in concrete and tension reinforcement under service loads.
- Generation of high quality design note where results are displayed by the mean of ***diagrams and tables***.
- Generation of an excellent AutoCAD-compatible drawing (extension DXF) where each component of the beam has its own layer.

**Advanced options are included:**

- **2D frame analysis:** Possibility to consider in the analysis of the beam the stiffness of columns below and above.
- **Rigid links at columns**. Since ACI considers axial spans, this classical approach may lead to unrealistic moments and deflections in case the dimension of column is more than the tenth of the span. The rigid link feature considers the part of the beam located within the column zone as very rigid, and thus improves noticeably deflection results.
- Possibility to introduce **settlements of supports** by the mean of vertical displacements. This feature is useful when one or more supports are subject to settlement (columns shortening or beam deflecting is case the concerned support is a beam).
- Generation of **bars bending schedule tables** according to ACI detailing manual. The BBS tables are placed below the elevation of the beam and allow complete steel procurement.

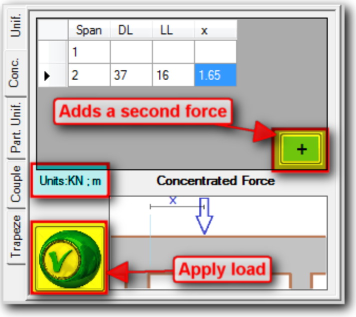
## Geometry tab



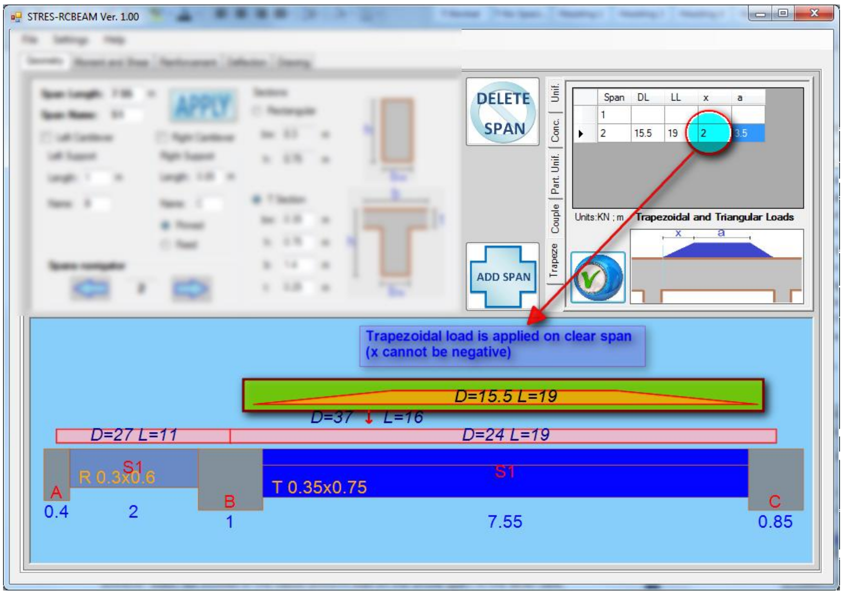

The Geometry tab is divided into three zones.

1. The graphical view at the bottom of the screen displays the elevation of the beam with information regarding its span's length, sections, the applied loads…
2. The upper left part is used to introduce the geometry of the beam. Rectangular and T sections are available. Ends supports of the beam could be pinned or fixed. Intermediate supports are pinned in the sense that they provide just a vertical support to the beam ($U_Z$ blocked). The treated problem is that of a ***continuous beam simply supported*** where only extreme supports could be either hinged or fully fixed.
3. It is possible to introduce sections of different heights "h" and widths "bw" for each span. In case of sections of different widths, a constant number of reinforcement bars will be used for all the sections. The software cannot put let's say 4 bars within the width of section span 1 and 7 bars within that of section 2. It is recommended to impose the number of bars columns in the "Settings" menu.
4. The right part of the window is intended for loads input.

## Applying loads




- Only dead and live loads can be introduced. Self-weight of beam can be considered by clicking on the "Settings" menu. Units are always KN and meters and cannot be changed.
- It is possible to input 2 concentrated forces only per span. Regarding concentrated couples, only 1 couple can be applied per span. "x" is the distance from ***face of support*** to the concentrated force or couple. In case of concentrated couple, if x is negative (within the length of left support) or more than the span length (within the length of right support) the software automatically shifts the location of the couple to the axis of the support.
- Partial uniform loads can be applied on a part of the span or on the whole clear span of the beam if x=0 and a=span length. ***In case of supports of big length***, it is more accurate to use the "Partial uniform" loads tab instead of the classic uniform load on the whole span. In the latter case, loads are applied on the whole span including the supports zone, which leads to moments and deflections greater than if loads are applied on the clear span only. In reality, loads located within the support zone are directly transmitted to the column below and do not induce bending or deflection in the beam.
- Trapezoidal loads are as well applied on clear span (x>0 and a<span length). If a=0 the load becomes triangular.
- Only uniform loads (first loads tab, called "Unif.") are applied on the whole axial span.



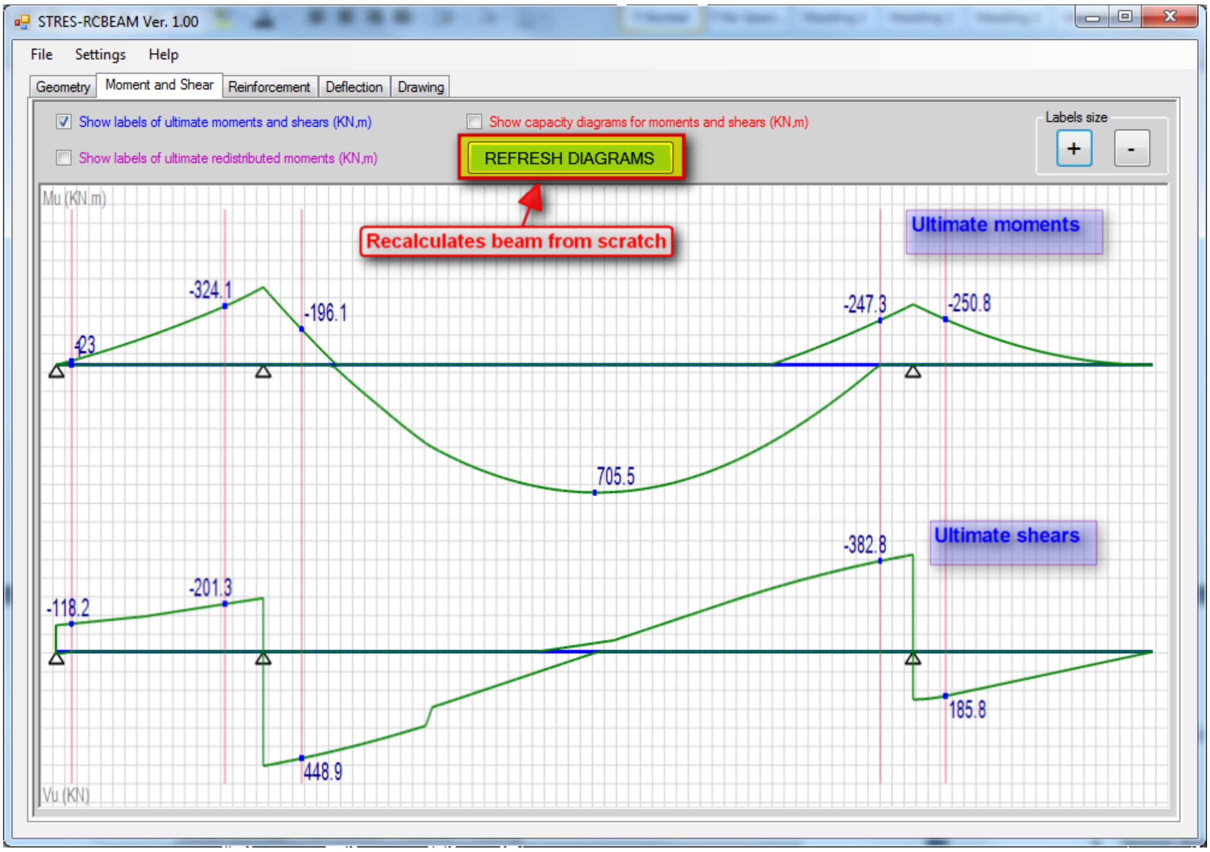

*Trapezoidal load can extend over the whole clear length of the span*

## Calculations and design


**RCBEAM** does not require a "Calculate" button. By navigating through tabs, each relevant calculation is performed, and warning/errors messages are displayed. However, it is important to note the following:

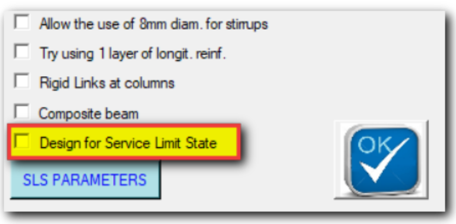
- If the user is in the "Geometry" tab for the first time or performing a modification (changing any parameter related to the geometry or to the loads) on an already designed beam, the ***"Moment and Shear"*** tab should be selected first. Moments and shears should always be calculated prior to reinforcement, after any change made to the beam.
- If the active tab is "Geometry" and the "Moment and Shear" tab is clicked, the beam is calculated whether modifications were done or not. For precaution reasons, the software always performs a calculation from scratch to the beam if the passage from "Geometry" to "Moment and Shear" tab is made.
- If the user is in any other tab than "Geometry" and changes are made through the "Settings" menu, the ***software automatically recalculates the beam*** without having to go to the "Moment and Shear" tab. When the "Settings" menu is called and changes are done, recalculation is directly performed.
- It is recommended to go to the "Deflection" tab only after calculating the real reinforcement of the beam in the "Reinforcement" tab. Deflection calculation considers the effective reinforcement generated by the software.
- It is possible to navigate from a tab to another, in any direction. Just keep in mind that any change performed in "Geometry" tab requires passing by the "Moment and Shear" one.

## Moments and Shears tab

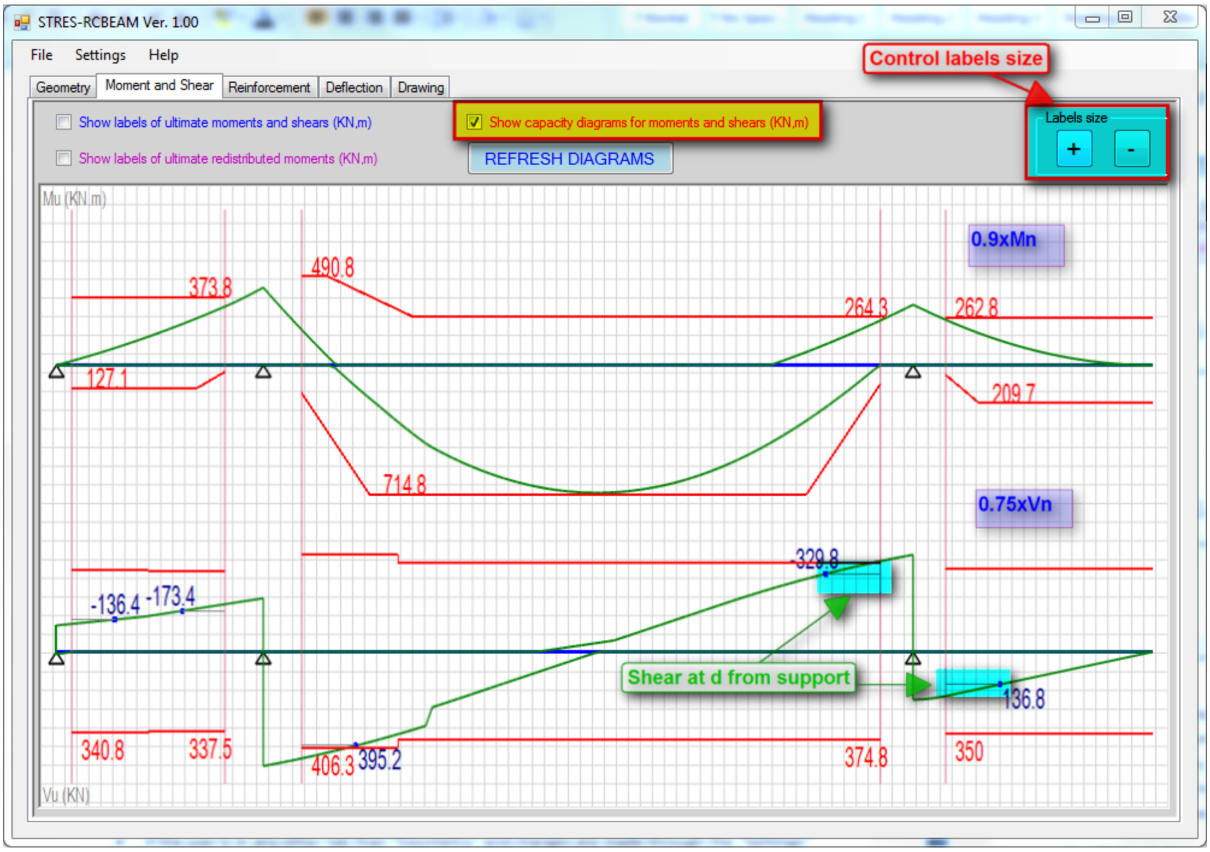



Once this tab is clicked, ultimate moments and shears are calculated and ***envelope diagrams*** are displayed. Dead and lives loads are factored by 1.2 and 1.6 respectively or the factors that were introduced in the Geometry tab. Dead loads alone are factored as well by 1.4. Live loads patterning is performed in order to obtain the maximal forces at supports and spans. Moments and shear values are always displayed at faces of supports and at their maximal values within the span.

The ***"Refresh Diagrams"*** button is clicked when capacity diagrams or shear and moments overlap or when part of them is outside the limits of the graphical window. A whole recalculation of the beam is then done, like if the "Moment and Shear" tab was selected again.




***If the design for Service Limit State is selected***, the program calculates the service moments and shears. Only the capacity diagram for moments is displayed, since design for shear is performed only at ULS, not SLS.




When ***"Show capacity diagrams for moments and shears"*** checkbox is selected, the nominal diagrams for moments and shears are plotted, factored respectively by 0.9 and 0.75. Ultimate diagrams (in green lines) should be comprised within capacity ones. If the "Design for Service Limit State" is selected, the capacity diagram for moments corresponds to the imposed crack cracks width value that is introduced in the "SLS parameters" window.

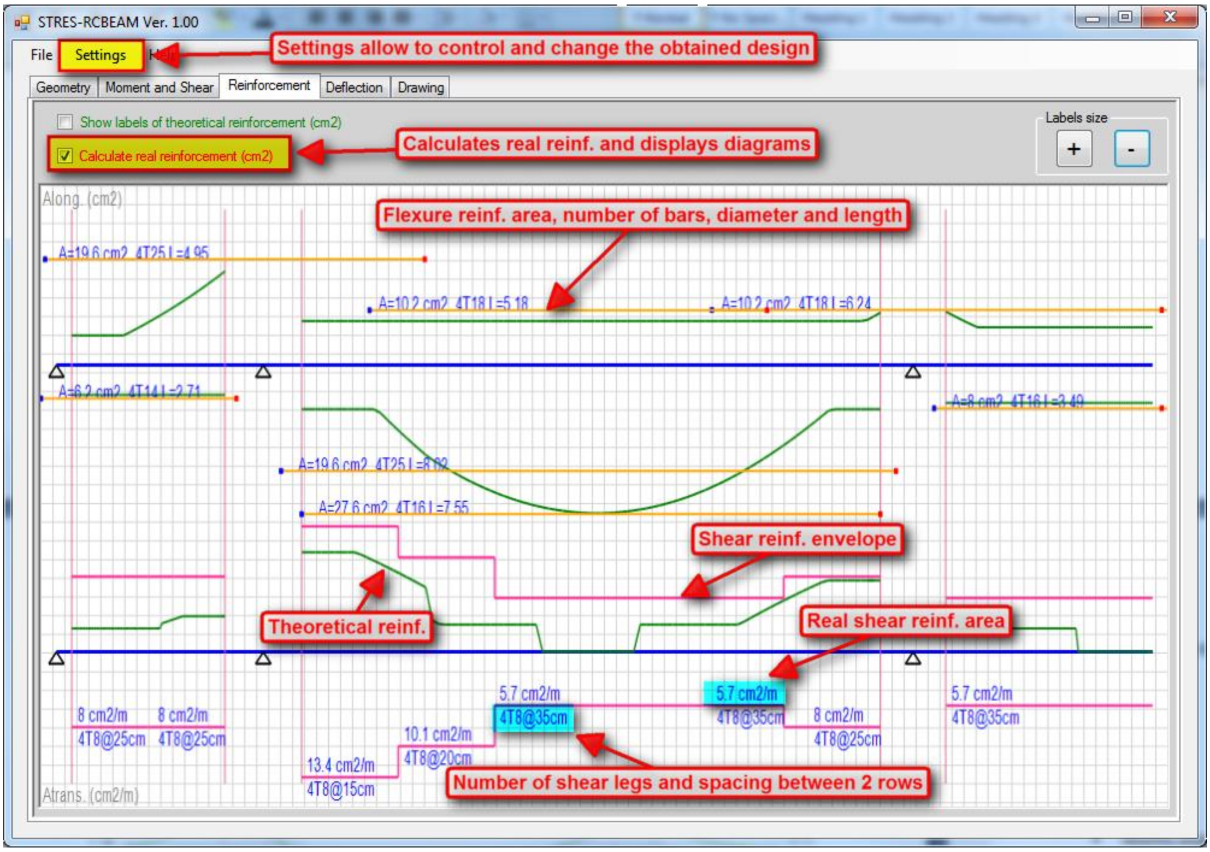
Capacity diagrams are obtained after the software generates the real reinforcement of the beam for flexure and shear. Thus, by modifying some features in the "Settings" menu, the whole design of the beam may be affected, and capacity diagrams may change.

A powerful reinforcement optimization engine explores the most economical solutions for reinforcement, varying several parameters like the number of layers, the diameters to choose and the number of bars to put within the width of the section.

For shear design near supports, ***values at distance "d" from faces are considered***. ("d" is the distance between the extreme compression fiber and the center of tension reinforcement). One should note that ***the program does not make any assumption regarding the lever arm*** and calculates it accurately starting from the concrete cover value and the layers grouping option located in the "Settings" menu. ***Thus, any results that depend on the lever arm (maximal and minimal reinforcement areas for flexure, nominal shears and moments, etc…) are all calculated in function of the value of "d" at the considered section.***

**NB:** If a calculation note is generated by clicking ***"Word design note"*** from the "File" menu, the diagrams and labels shown inside the picture box will be inserted in the design note as an image. Labels sizes may be tuned in order to attain visible figures in the design note. The diagram's picture is inserted in the report "as shown" in the corresponding tab of the program.

## Reinforcement tab



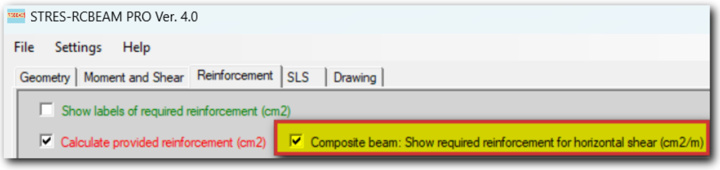

This tab displays ***theoretical (required) and real (provided) reinforcement*** diagrams.

Real (provided) longitudinal reinforcement is displayed by the means of bars layers, showing their lengths, diameters and area. When several layers exist, the reinforcement area shown is cumulative and not proper to each layer. By example, in the figure above, there are two layers of bottom reinforcement. ***The second layer 4T16 shows A=27.6cm² which is the sum of 4T25+4T16.***

Anchoring of bars at support is calculated according to Chapter 23 of ACI 318-19 (***Strut-And-Tie Method***). The whole supports area is verified regarding the compression strength of struts, the required steel area of tie (bottom reinforcement) and the required anchoring length of the tie. ***The program considers shear force and the algebraic value of the moment at face of support to calculate forces in strut and tie.***

- It is possible to modify the minimal anchoring length allowed by the Code (15cm) in the "Settings" menu.

- The development length of hooked bars in tension can be reduced, as per section 25.4.3.2 of ACI 318-19, if confining reinforcement is provided.
- If "Structural Integrity" is checked in the Settings menu, then the instructions of ACI 318-19 section 9.7.7 are implemented. At least the fourth of the mid-span bottom reinforcement will be continuous at intermediate supports. To be noted that at end supports the required anchored reinforcement should be full developed.



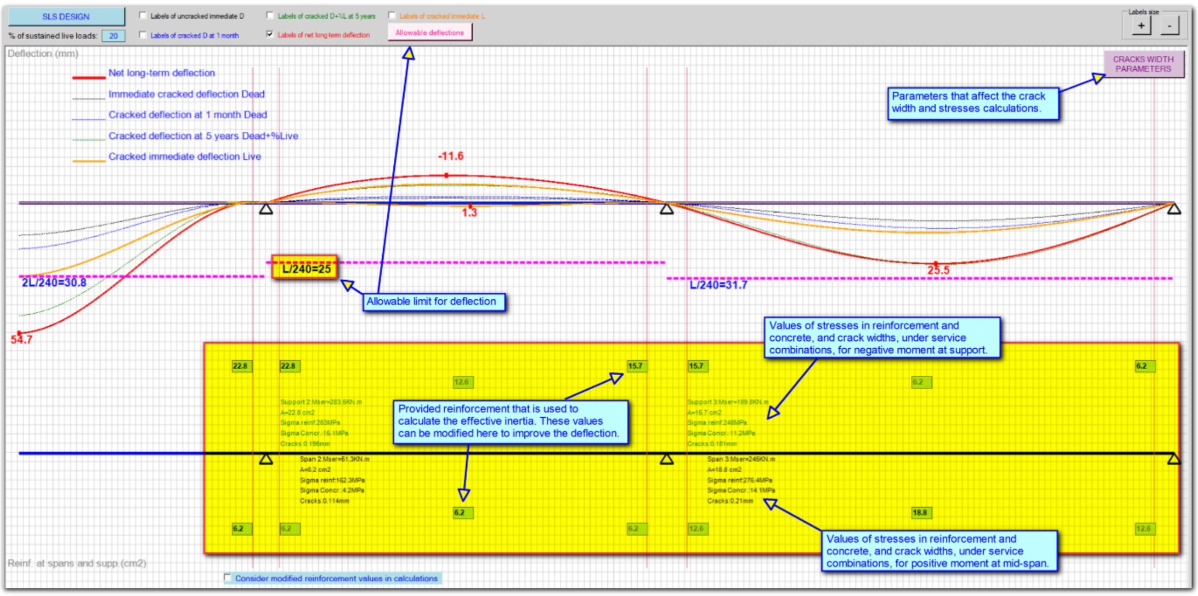

If "Composite beam" option is considered, then the transversal reinforcement that is required to ensure full transfer of shear forces along the construction joint is calculated, as per section 16.4 of ACI 318-19.

Shear reinforcement zones are accurately determined to fit the demand diagram. It is possible to ask the software to try using 8mm diameters for shear reinforcement in the "Settings" menu. ***The design is instantaneously updated each time the "Settings" window is closed***.

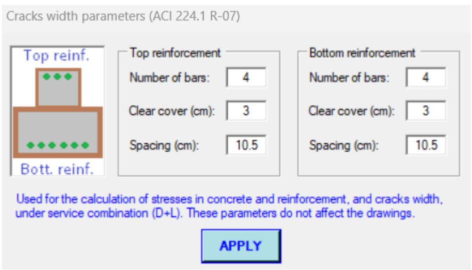
***One should note that the program always considers that every longitudinal bars is tied to the upper one with either a tie or a stirrup.*** This is why ***the number of shear legs is always equal to that of bars present in a layer***. For instance, in the example above there are 4 longitudinal bars per layer; there will be 4 shear legs as well. The program optimizes the diameter of the shear links and the spacing between two rows. In some situations, if a span is heavily loaded, it may have its shear legs diameter different from that of other spans. When a drawing is generated, the program correctly implements stirrups and ties for shear legs.

In case ***seismic dispositions*** are set, both flexural and shear design are affected in order the match the full requirements of section 18.4 of ACI 318-19 for intermediate moment resisting frames. Nominal shear strength of the section considers then the additional shear resulting from the nominal moments strength "Mn" of each span.

## Deflection tab




Beside the calculation of the deflection, this tab provides the stresses in concrete and tension reinforcement, together with the crack width, under service moments, at mid-spans and supports.




It is possible to control the parameters required for the calculation of the crack width by clicking on the “Cracks width parameters” button at the top right of the screen. The number of bars, their spacing and clear cover are the main variables required by ACI 224.1 R-07. In some cases of precast beams, the number of top bars might be different from the bottom ones. This difference concerns only the crack width design.

Deflection is calculated based on the ***envelope curve of elastic deflections*** under all service combinations. For each span, service moments at span and supports are used to calculate the effective inertias at these sections.

Long-term deflection considering shrinkage and creep is calculated by using the factor $\lambda_\Delta$ in section 24.2.4.1.1 of ACI 318-19. This factor multiplies the immediate cracked deflection of a specific combination, as a consideration of creep and shrinkage effects, to give the additional time-dependent deflection.

After introducing the amount of live loads (%L) to be considered as sustained, the following deflections are calculated:

- **Immediate uncracked deflection under all dead loads**. It is provided for information only.
- **Cracked deflection at 1 month.** This deflection is calculated using the effective inertia of the beam, one month after removal of the scaffolding. It supposes that during this month all dead loads are applied (including partitions) which are just completed. The time-dependent factor for sustained loads is $\xi$=0.5. Any deflection that will occur starting from this date is considered as affecting the installed brittle partitions.

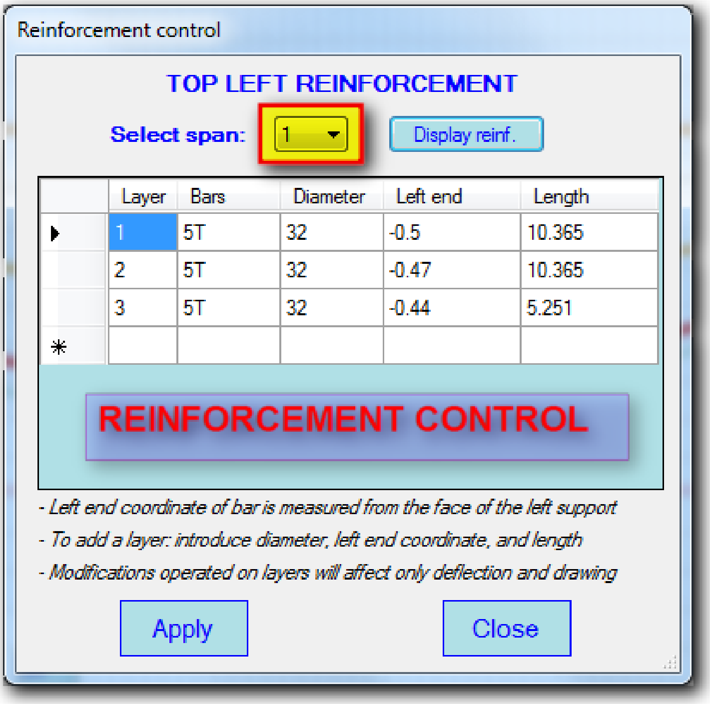
- **Cracked deflection after 5 years**. It includes all dead loads plus sustained live loads. The time-dependent factor for sustained (%L) loads is $\xi$=2. The deflection reached its maximal value regarding shrinkage and creep effects.
- **Net long-term deflection**. Net long-term deflection is the subtraction of the one-month deflection from the 5 years deflection plus that of not sustained (immediate) live loads (1-%L). It accounts for any deflection that occurred after completion of partition works.
- **Live loads cracked immediate deflection**. It is obtained by subtracting instantaneous deflection under dead loads calculated with effective inertia, from that under dead + 100% live loads calculated as well by considering effective inertia. This deflection is used for parking floors. Creep is not considered in calculations.

The lower part of the deflection window shows the provided reinforcement. These values are considered to calculate the effective inertia of each span. In case the deflection exceeds the allowable limit, ***it is possible to reduce it by increasing the values of the real reinforcement*** shown in the text boxes.

In order to take advantage of the features of the **"Reinforcement control"** window, it is advised to follow these steps. To be noted that it is possible to directly go to the "Drawing" tab to generate the DXF drawing and bypass these steps.

1. Once the deflection is calculated, reinforcement textboxes will display in a table the corresponding layers of bars with their diameters and location. Clicking inside the textboxes will open the corresponding table.
2. It is possible to manipulate reinforcement (extend a layer, remove it, modify its diameter…) not just to improve the deflection but mainly to personalize a bit more the drawing to be generated, if needed.
3. Finally, it is possible to either recalculate the deflection (the checkbox at the bottom of the window should be checked so the modified reinforcement areas are considered) or to directly regenerate the drawing again.

The **"Reinforcement control"** window is an additional tool that helps the user to fine-tune the drawing, if needed. It is used as well to increase the reinforcement in order to improve the deflection.

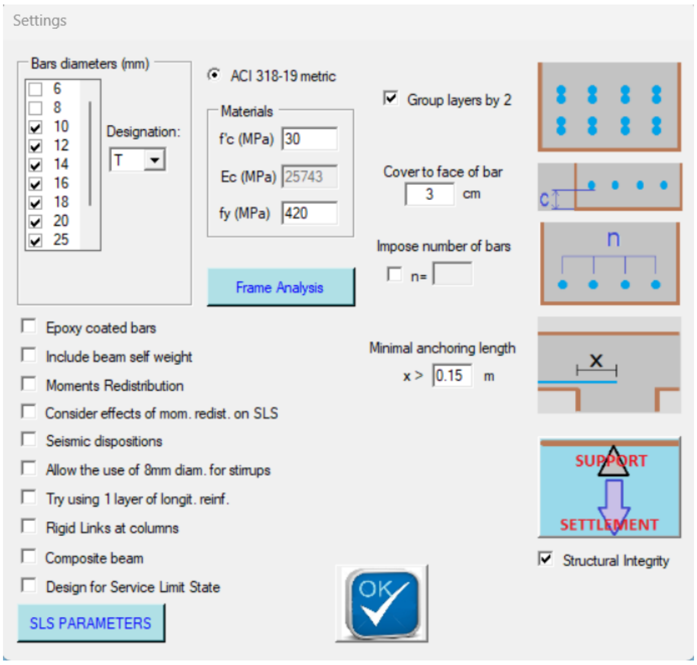



*Diameters, positions and lengths of bars can be controlled before generating the drawing*

All bars are positioned with their left end coordinate toward the face of the left support of the actual span. Negative value means the layer is anchored within the support.

**For first and last supports of the beam**, if the bar end coordinate (left, right, or both) is beyond the limits of the beam, this means that it will be affected with a 90 or 180 degrees hook. If the user modifies the ends of such bars –by example decreasing bar length in order to fit the right end of the layer within the width of the extreme right support- the software may not anchor the bar once the drawing is generated.

## Settings menu




This window contains all parameters used to control design and calculations.

It can be called from any tab (reinforcement, moment and shear…) and will directly update the design of the beam once the “Ok” button is clicked. The program decides whether just a reinforcement design should be redone or the whole recalculation, starting from the moments and shears, depending on the modified items in this window.

For instance, if ***“Include beam self weight” or “Moments Redistribution”*** are modified, then a complete re-calculation is required which is directly done. Other options –like seismic dispositions- if modified require just a RC redesign because they do not affect the obtained forces.

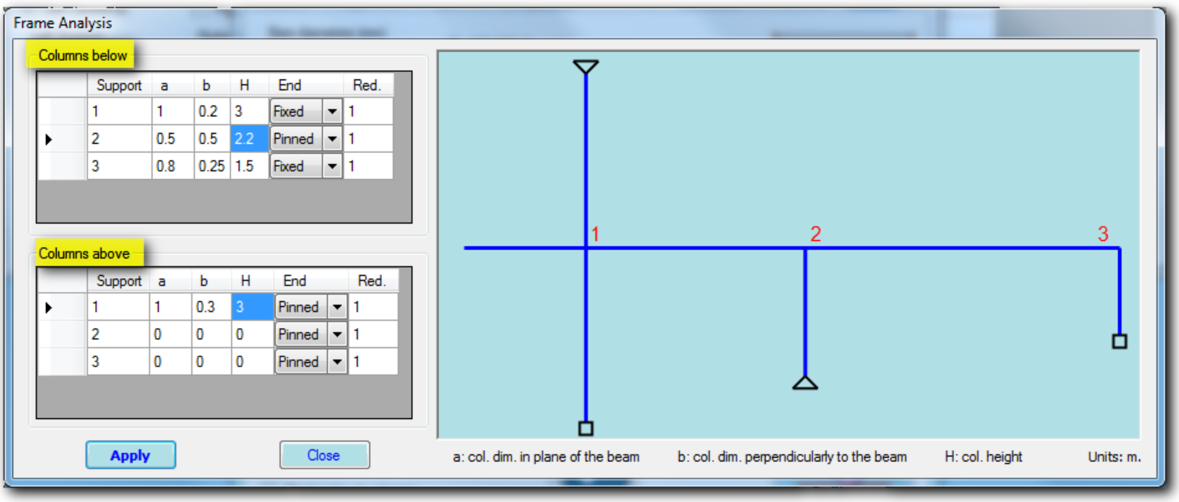
The “group layers by 2” option allows to reduce the lever arm and thus to decrease reinforcement area by grouping layers by packets of 2. The clear distance between two packets of two layers should be at least 2.5cm, as per the Code requirements. This option, and that fixing the amount of cover allow the program to accurately calculate the lever arm at any section.

While being in any tab –especially in the reinforcement one- it is possible to open the “Settings” window and to optimize the design by ***imposing the number of bars within the width of the beam***, ***imposing the diameters to be used***, ***allowing the use of 8mm diameters for shear links***, or pushing the program to ***use just one layer of flexural reinforcement*** where possible. This last option will prevent the program from performing an accurate steel optimization by preferring to use one layer instead of 2, which is helpful in case of shallow beams. ***Each time a feature is modified the reinforcement diagram –and those of moments and shears too- shown in the “Reinforcement” tab are directly updated***.

The minimal anchoring length of a bar within the support is 15cm as per the Code. This length is not always adopted since it is a minimal limit. Anchoring length depends –beside other parameters- on the tension force in the layer and on the diameter of the bar. Tension force is not generated only by a moment, but by inclined struts due to shear forces. These struts induce a horizontal tension component that should be equilibrated by the anchored layers of longitudinal reinforcement. ***The Strut-And-Tie Method is implemented in the program*** and verifies the required tension steel area for tie, its anchoring length and the compressive stress in the struts.

## 2D Frame analysis

It is possible to consider the flexural stiffness of columns belonging to the story below -and/or story above-. This will induce rotational constraints on the concerned supports. The reduction of the gross inertias of columns can be done by introducing a number less than 1 in the last column of the tables below ("Red."). The stiffness reduction factor cannot be zero in case height of column is not null.

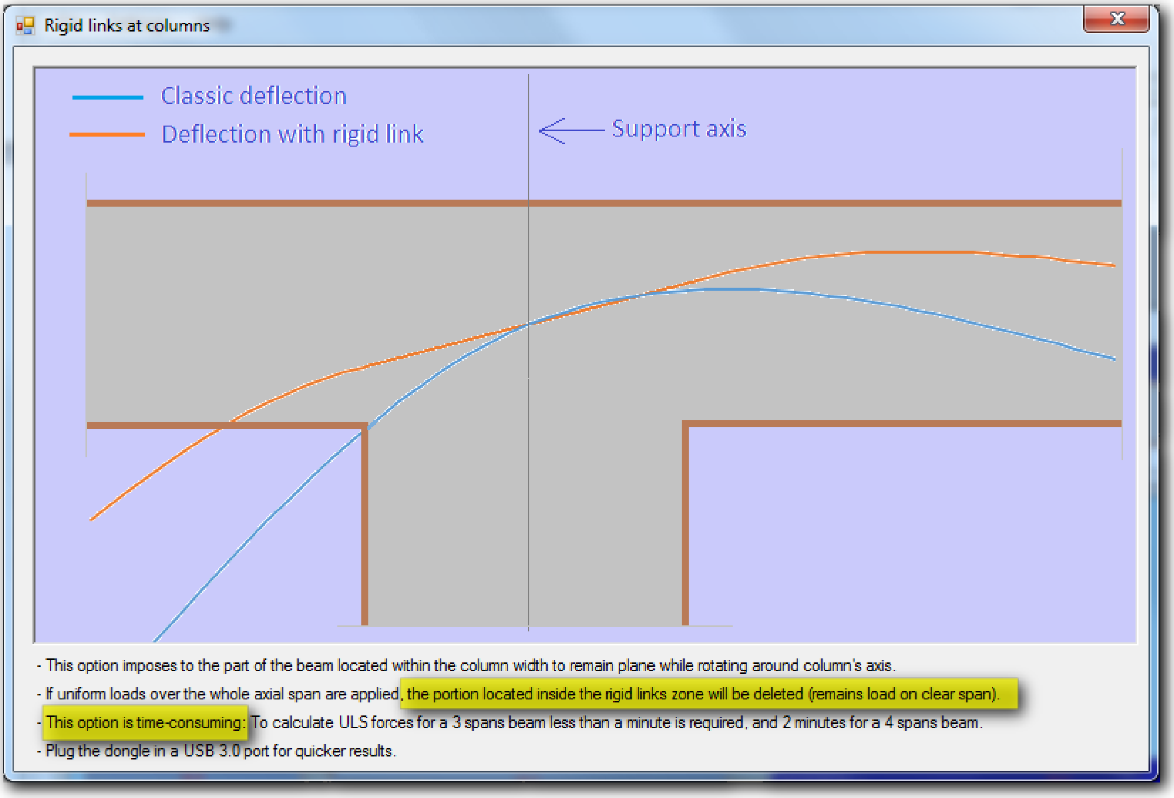



If the user wishes to disregard the stiffness of a column, a zero value should be introduced for its height. In this case a simple support will be considered and no moment will be transferred to the column. A column with **zero height** is equivalent to a **pinned support**. End condition for a column can be either pinned or fixed.

NB: The software will not display for columns results like diagrams of shears and moments. Only results for beams are displayed.

## Rigid links at columns

ACI calculates the continuous beam considering its axial spans, whatever the dimensions of the columns are. This approach is acceptable as long as the dimension of the column in the plane of the beam is negligible toward the span. But in many cases this hypothesis is not fulfilled. Intermediate columns could have a substantial length (more than the tenth of the span) and extreme supports could be walls or big columns as well.




The figure above shows the deflection shape of a beam at intermediate support. In analysis programs, a support is physically located at the axis of the column. It can be seen that the curvature of the beam starts immediately from each side of the support axis, if no rigid links are used (blue curve).

The rigid link option affects to the part of the beam located within the column zone a very big inertia intended to model the effect of the real dimension of the column. Indeed, it is not sufficient to represent a column by a nodal support on its axis; its physical dimension should be considered. The part of the beam common to the column cannot deflect freely due to the presence of the column. In this area, the beam is supposed to rotate around the column axis without deflecting, but while remaining plane (orange curve). Deflection will start at faces of the column, to the direction of the span.

This option resolves a commonly encountered problem by providing:

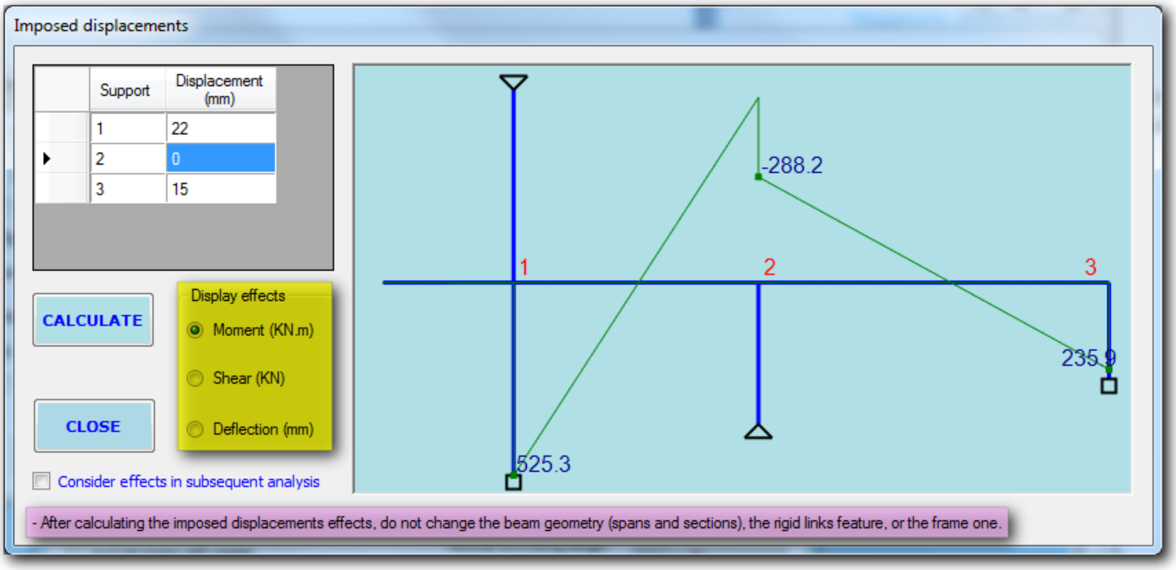
- A more correct moment and shear diagrams
- A realistic deflection that could be much overestimated if this option is not used.

## Supports settlements

In continuous beams, the settlement of one or more supports can remarkably increase negative moments and even could lead to the creation of positive moments at supports (see figure below). The whole moments envelope is affected beside of course shears and deflections.

Practically, the reason of supports settlement can be due to:

- ➢ Elastic and inelastic shortening of columns under compression forces
- ➢ Settlement of foundations
- ➢ One of the supports is a beam, which is more flexible than columns.




**RCBEAM** allows to:

- Introduce for one or more supports the imposed displacements.
- Displays the consequents **intrinsic** effect by the mean of moments, shears and deflection diagrams.
- In case of more than one support are concerned, choose whether the introduced displacements should be considered as acting **simultaneously** or as an **envelope** of individual cases.
- Specify the proportion of imposed displacement that is occurring after 1 month (the remaining will be applied at long term, i.e. after 5 years). This option is very important for the net deflection calculation since this latter involves those obtained after 1 month and 5 years.

After introducing the displacements, their effects should be calculated by clicking the “calculate” button. One can visualize the consequent moments, shears, and displacements by checking the corresponding radiobuttons (refer to the highlighted area in the figure above). **It is essential to note that these results are based on the span geometry, sections, columns stiffness and rigid links options specified before clicking the calculate button! In other words, user has to fix all the mentioned parameters before calculating effects of imposed displacements.**

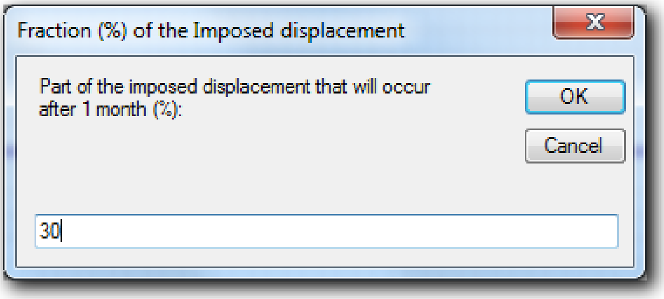
If after closing the imposed displacements window, the spans or sections or rigid link option or 2D frame feature are modified, the calculated effects of settlement will not correspond to the new geometry of the beams and they will have to be recalculated.

However, all remaining parameters like moments redistribution, seismic dispositions and even loads modification (removing or adding of any kind of loads) do not require a recalculation of the imposed displacements effects.

The obtained forces effects (moments and shears) will be superposed to the ULS envelope without being factored because the imposed displacement occurs always at service stage. (Imposed displacement values are associated with the service behavior of the structure).

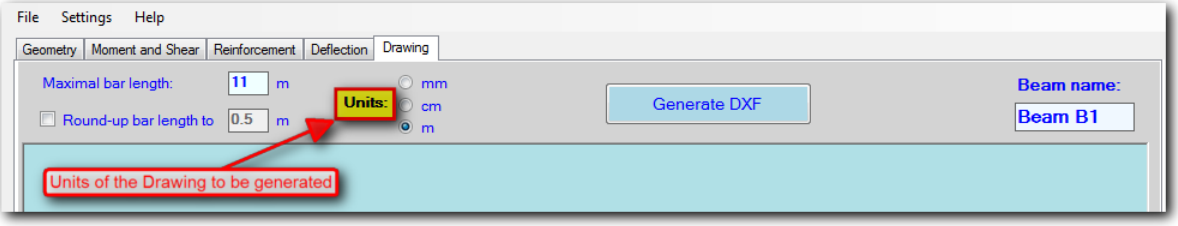
**Effect on deflections:**

The obtained displacement effect will be superposed to the different stages of deflection of the beam according to the amount specified by the user of imposed displacement that will occur one month after removing the shoring of the slab.




As per the figure above, 30% of the total settlement will be superposed to the deflection at 1 month, and the remaining will be added to that after 5 years. While calculating the cracked inertias, the software considers the contribution of the moments due to the imposed settlement at each stage, providing thus a **complete incorporation** of the effect in question.

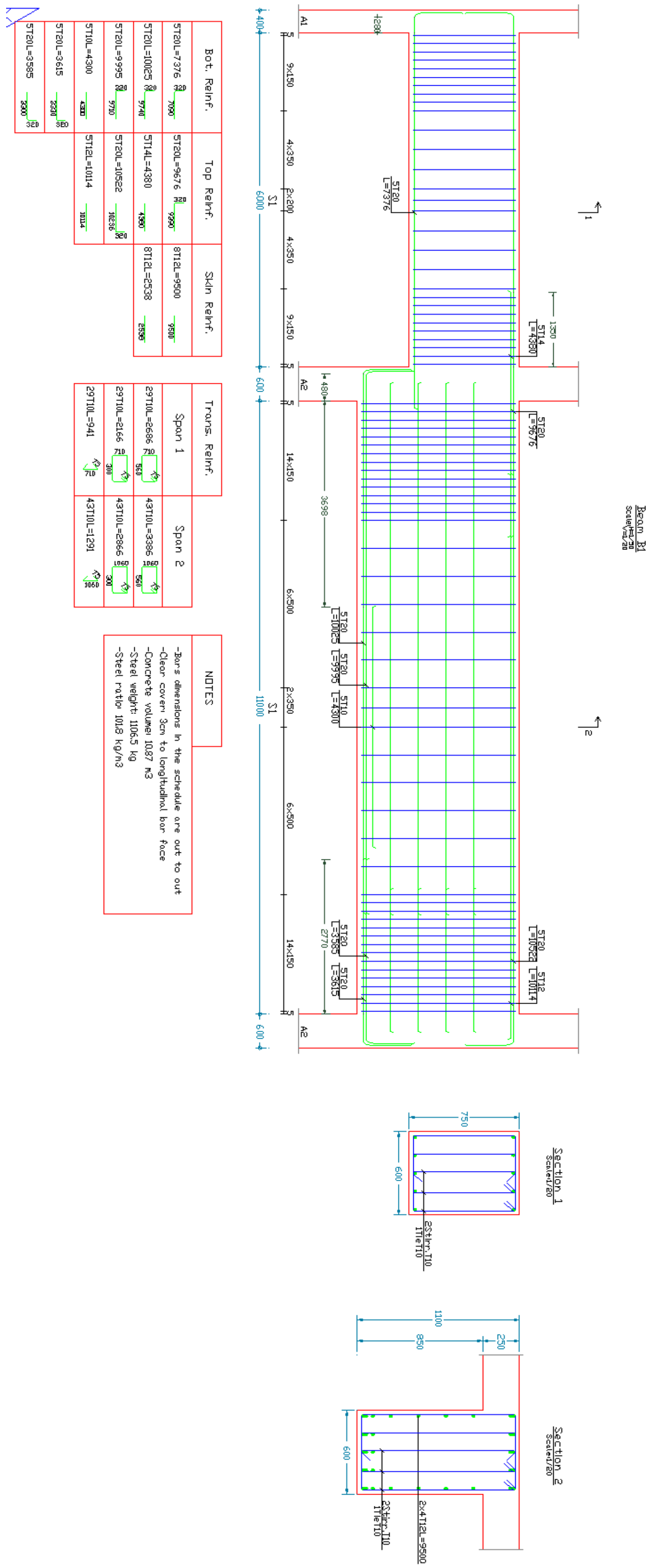
# Drawing tab




**RCBEAM** is able to generate excellent AutoCAD-compatible drawings that can be considered as **shop drawings ready to be executed by the Contractor**.

The generated drawing reproduces all the components of the beam with its smallest details.

- 90° and 180° hooks are generated automatically when required. Length of hooks respects ACI formulas regarding the diameter of the bent and the length of the straight part.
- Each bar has its total length indicated. In the case of hooks at its ends, ***the total length correctly comprises that of hooks***.
- Concrete cover and distances between layers are respected.
- In case the bar requires a hook at support and the anchoring length $l_d$ is not assured, the program displays a warning message. This verification is made for each anchored layer since the anchoring space becomes smaller when the number of layers increases. An “insufficient anchoring length” message will trigger.
- Each component of the drawing has its individual layer.
- ***Maximal bar length can be set***.
- ***Rounding-up bars can be done*** in order to obtain lengths multiple of 0.5m by example.
- ***Seismic dispositions*** for ties and stirrups are automatically adopted if seismic dispositions are required. Seismic hook is used for ties and stirrups.
- ***Complete Bars bending schedule tables*** conform to the latest ACI detailing manual with quantities survey and reinforcement ratio.


Beam B1
Bot. Reinf.
Top Reinf.
Skin Reinf.
Trans. Reinf.
Span 1
Span 2
NOTES
-Bars dimensions in the schedule are out to out
-Clear cover: 3cm to longitudinal bar face
-Concrete volume: 10.87 m3
-Steel weight: 1106.5 kg
-Steel ratio: 101.8 kg/m3
Section 1
Section 2

## Sample of a design note

### Beam B1

**Assumptions**

Concrete strength f'c=30MPa.    Steel yield limit fy=420MPa

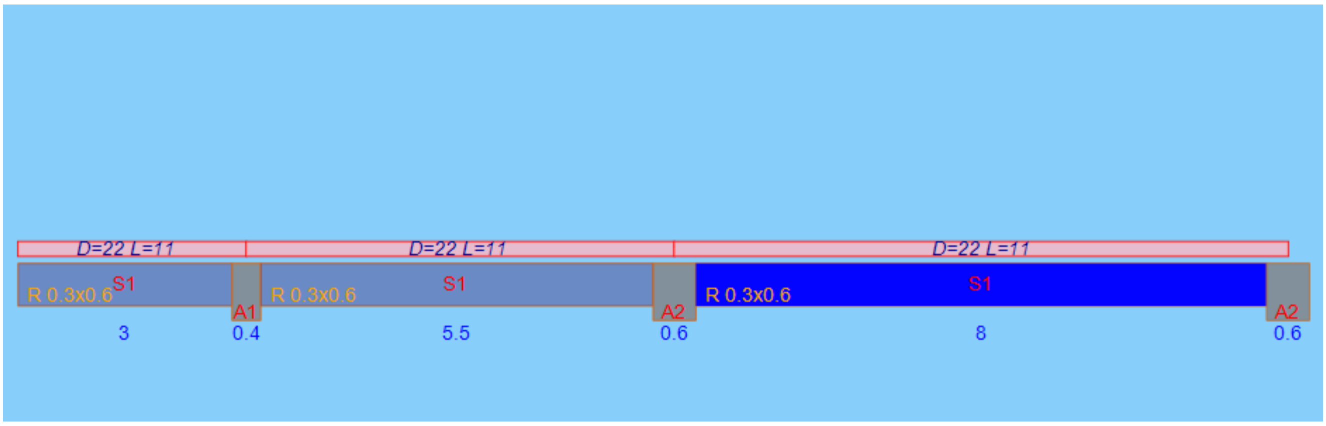
Seismic dispositions: None

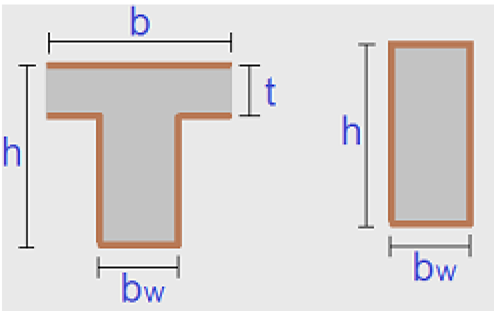
Rigid links implemented at zones of intersection between supports and spans

Beam self-weight not considered

*Design According to ACI318-11 metric*

**Beam Geometry (m):**



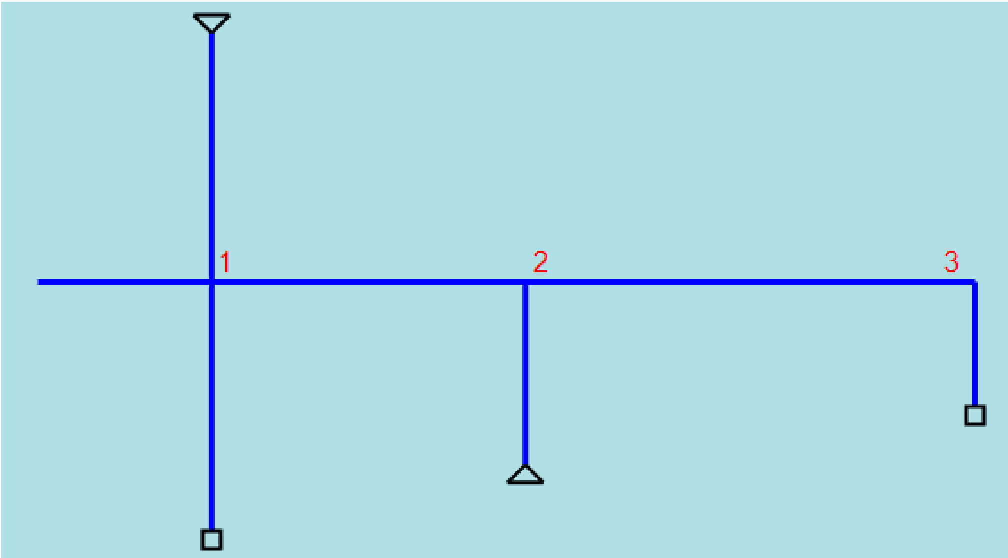

| ***Span#*** | ***Clear span length*** | ***Left supp. length*** | ***Right supp. length*** | ***Section dimensions*** |
|---|---|---|---|---|
| 1 | 3 | Cantilever | 0.4 | bw=0.3 h=0.6 |
| 2 | 5.5 | 0.4 | 0.6 | bw=0.3 h=0.6 |
| 3 | 8 | 0.6 | 0.6 | bw=0.3 h=0.6 |

**Loads tables:**

Uniform Loads (KN/m):

| *Span#* | *Dead Loads* | *Live Loads* |
|---|---|---|
| 1 | 22 | 11 |
| 2 | 22 | 11 |
| 3 | 22 | 11 |

**2D Frame Analysis option enabled:**




| *Columns below:* | *a (m)* | *b (m)* | *H (m)* | *End connection* | *Inertia reduction* |
|---|---|---|---|---|---|
| 1 | 1 | 0.2 | 3 | Fixed | 1 |
| 2 | 0.5 | 0.5 | 2.2 | Pinned | 1 |
| 3 | 0.8 | 0.25 | 1.5 | Fixed | 1 |

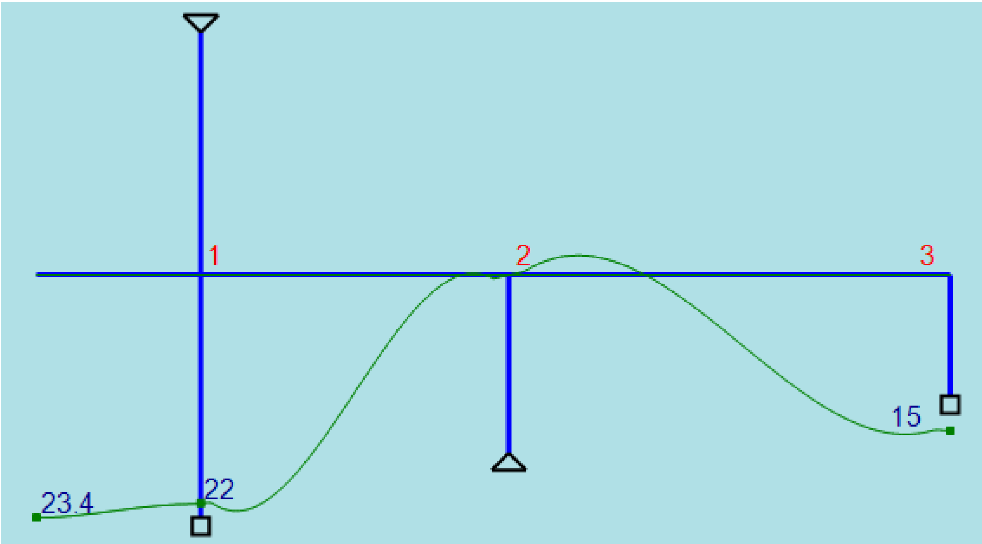
| *Columns above:* | *a (m)* | *b (m)* | *H (m)* | *End connection* | *Inertia reduction* |
|---|---|---|---|---|---|
| 1 | 1 | 0.3 | 3 | Pinned | 1 |
| 2 | 0 | 0 | 0 | Pinned | 1 |
| 3 | 0 | 0 | 0 | Pinned | 1 |

NB: a: col. dim. in plane of the beam.   b: col. dim. perp. to the plane.   H: col. height.

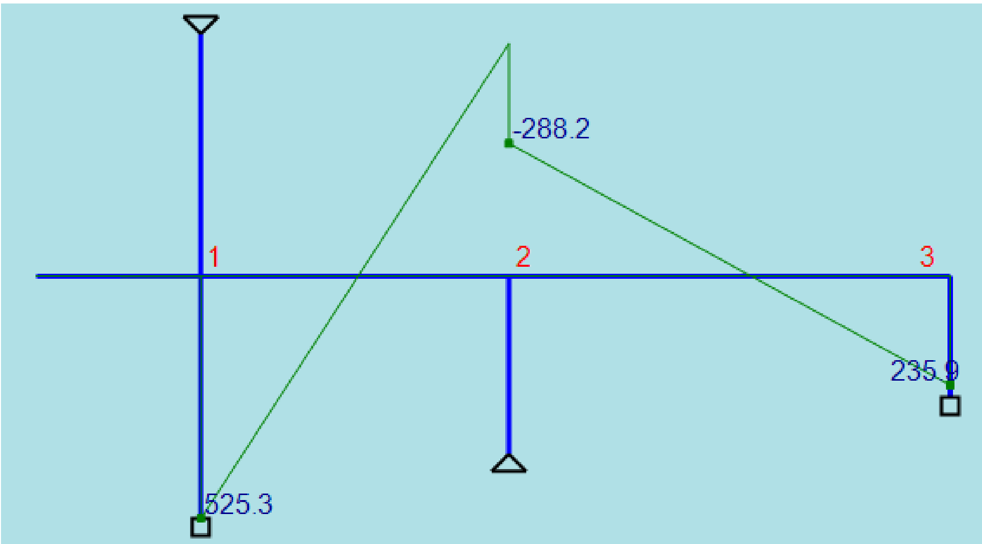
**Imposed displacements at supports:**

| ***Support#*** | ***Imposed disp. (mm)*** |
|---|---|
| 1 | 22 |
| 2 | 0 |
| 3 | 15 |

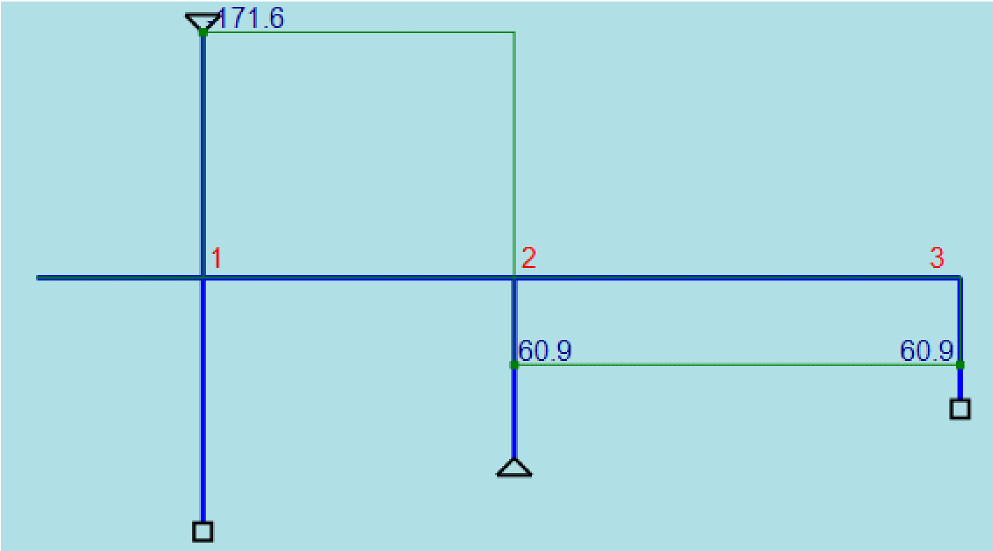
Displacements effect (mm)



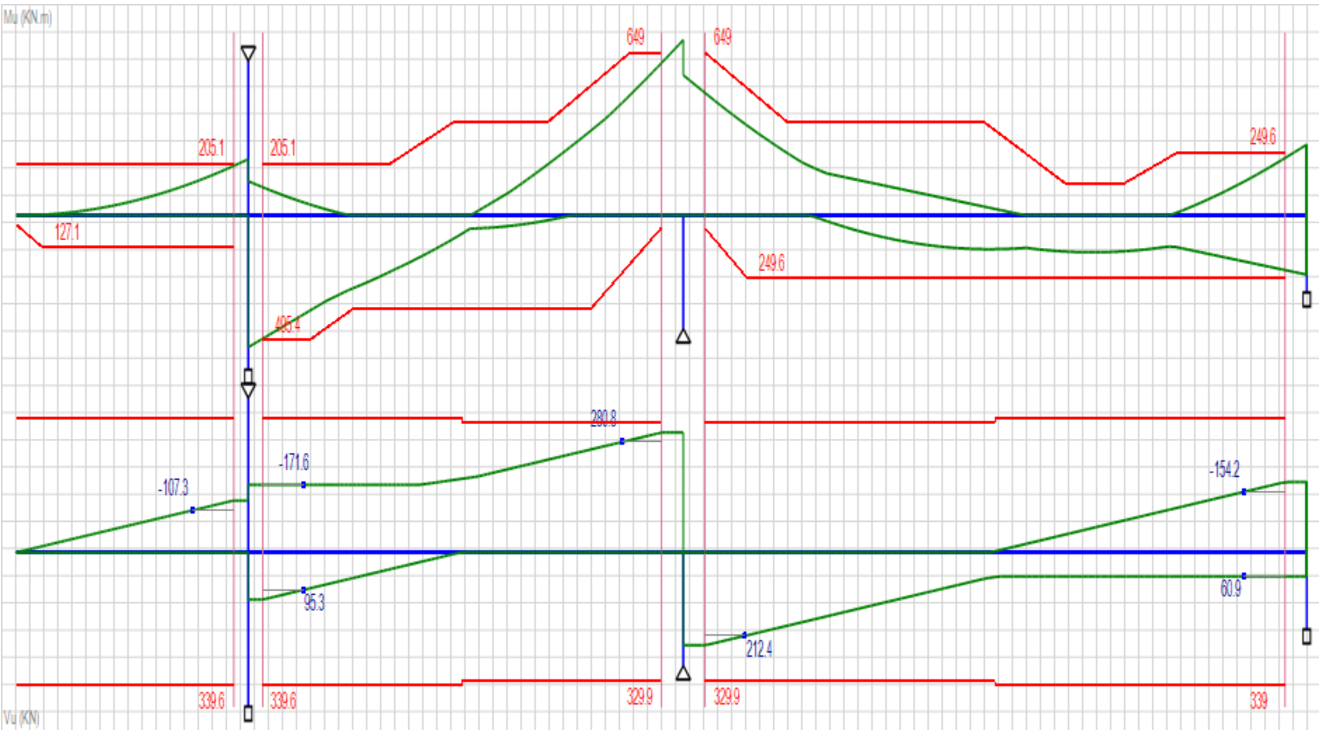

Moments effect (KN.m)



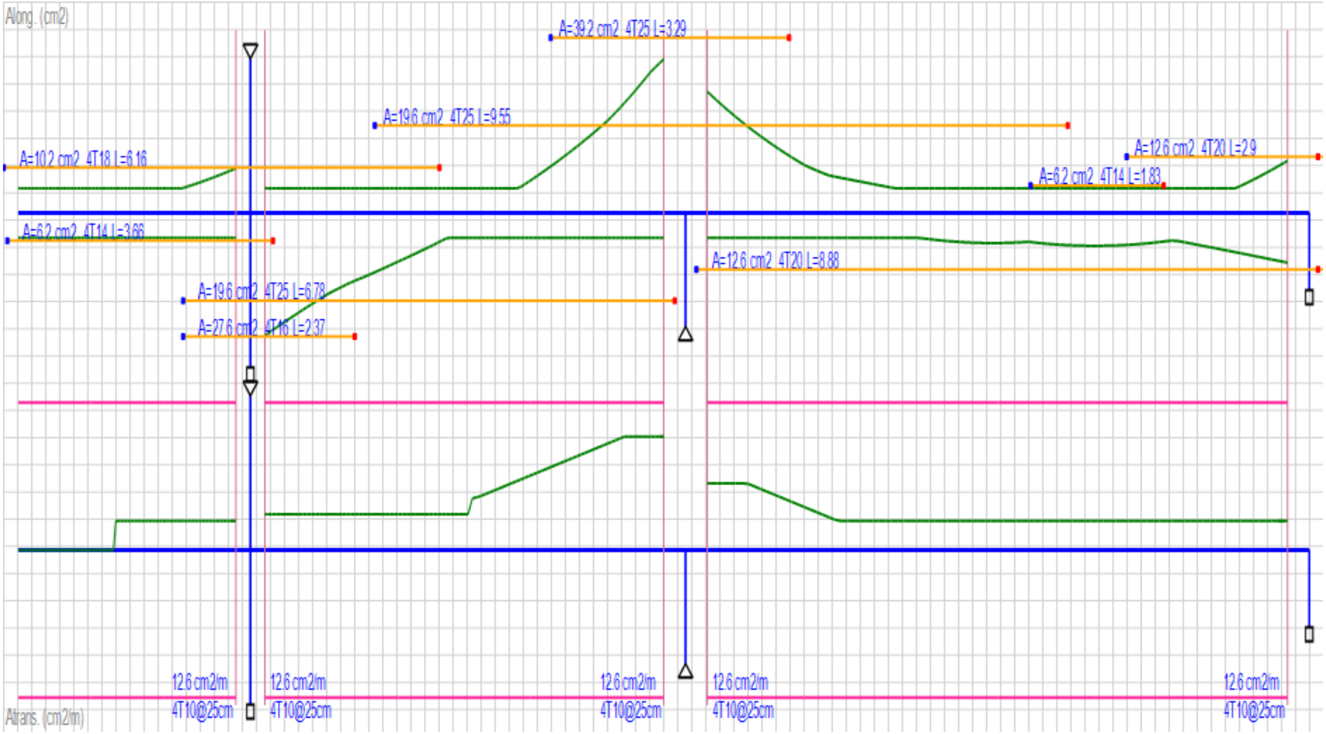
Shears effect (KN)



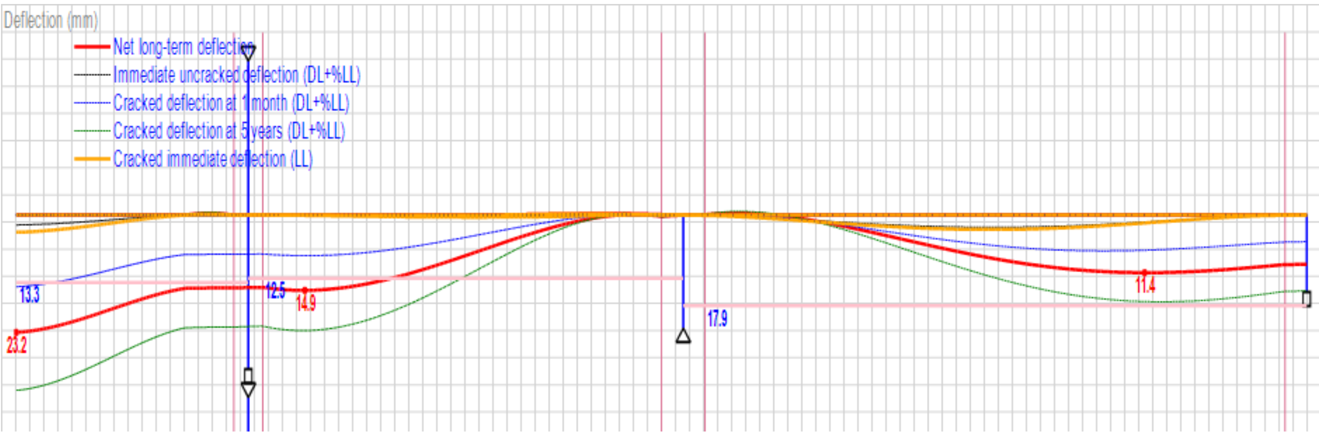

**Ultimate moments and shears (Mu & Vu) - Capacity curves (0.9Mn & 0.75Vn) - (KN.m; KN)**



## Theoretical and provided reinforcement (cm²)




## Deflections (mm)




20% of live loads are sustained.

Net long-term deflection (red curve) is the subtraction of the one month deflection (blue curve) from the 5 years deflection (green curve), and adding that of not sustained live loads (80% of LL)

35% of imposed displacements are applied 1 month after removal of scaffolding

**Shear results at supports (KN)**

| *Span#* | *Left supp. Vn* | *Left supp. Vc* | *Left supp. Vs* | *Right supp. Vn* | *Right supp. Vc* | *Right supp. Vs* |
|---|---|---|---|---|---|---|
| 1 | 0 | 0 | 0 | 452.8 | 156.71 | 296.08 |
| 2 | 452.8 | 156.71 | 296.08 | 439.9 | 152.24 | 287.64 |
| 3 | 439.9 | 152.24 | 287.64 | 452 | 156.43 | 295.55 |

**Struts and ties verifications at left supports (Appendix A of ACI318-11):**

| *Span#* | *Atie (cm²)* | *Strut length (m)* | *Strut strength (0.75xFn,KN)* | *Force on strut (Fu,KN)* |
|---|---|---|---|---|
| 1 | 0 | 0 | 0 | 0.03 |
| 2 | 34.66 | 0.57 | 1521.88 | 242.74 |
| 3 | 0 | 0.15 | 415.21 | 333.01 |

**Struts and ties verifications at right supports (Appendix A of ACI318-11):**

| *Span#* | *Atie (cm²)* | *Strut length (m)* | *Strut strength (0.75xFn,KN)* | *Force on strut (Fu,KN)* |
|---|---|---|---|---|
| 1 | 0 | 0.51 | 1279.74 | 186.7 |
| 2 | 0 | 0.15 | 427.38 | 431.05 |
| 3 | 0 | 0.57 | 1437.43 | 253.03 |

*Atie is the required area of reinforcement due to the tension forces on the anchored bars. Atie=0 if the STM (Strut-And-Tie Model) does not lead to tension on the anchored bars, due to the compensation resulting from negative moments.*