

Introduction



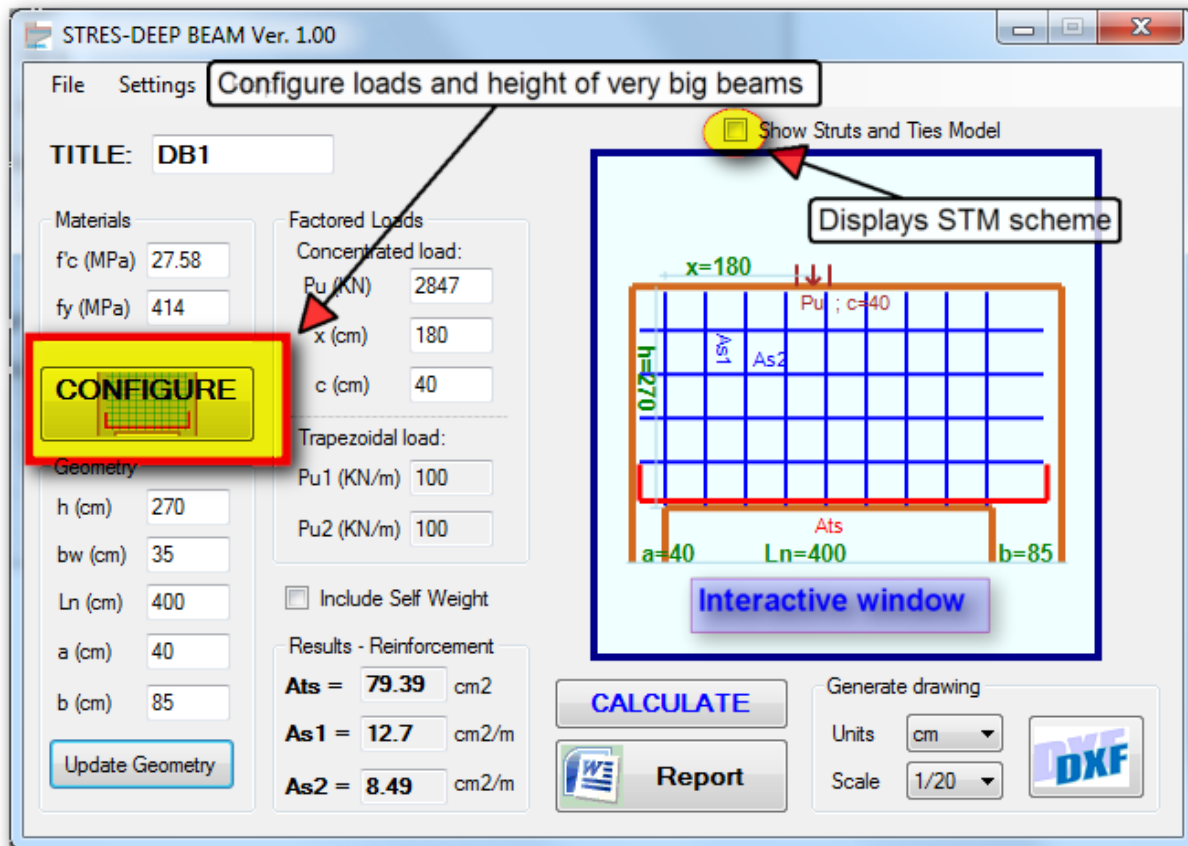
The **DEEP BEAM** software designs reinforced concrete deep beams as per the recommendations of Chapter 23 of ACI318-19 metric version. It is a module of **STRES** structural design suite which offers complete solutions for buildings problems.

STRES-DEEP BEAM has a very simple interface but a sophisticated calculation engine which allows the user to do much more than the classic limitations of the Code. Design of deep beams is not clarified enough in ACI and most of engineers rely on Strut-and-Tie Models (STM) as described in Chapter 23. However, this chapter does not provide complete answers for most of encountered problems, and the examples actually present in the literature are usually stereotyped and simple.

The present software designs one-span deep beams with the following special features:

- Loads can be a concentrated force or linear trapezoidal.
- Strut-and-tie model is generated considering the calculated widths of struts and ties through an iterative process. Positions of nodes are therefore imposed by calculations and not assumed in an approximate way. The software automatically chooses the best STM configuration to suit the problem, respecting the struts angle limitations of the Code.
- Left and right supports can have different lengths. This has an influence on the STM and modifies inclination of struts and positions of nodes.
- The program considers the case where deep beams have a very large depth (bearing wall with transparency at the ground floor for instance). The software calculates the effective height to be used for design and correctly transposes the applied loads.
- Any kind of nodal zones are treated, not just hydrostatic ones which are a particular case. Even if struts are not perpendicular to nodal zone face, **DEEP BEAM** correctly calculates the exact dimensions of struts, ties and nodal zones, and designs them accordingly.
- Detailed design note with figures showing STM and all nodal zones.
- Generation of quality DXF drawings with customization of bars diameters.

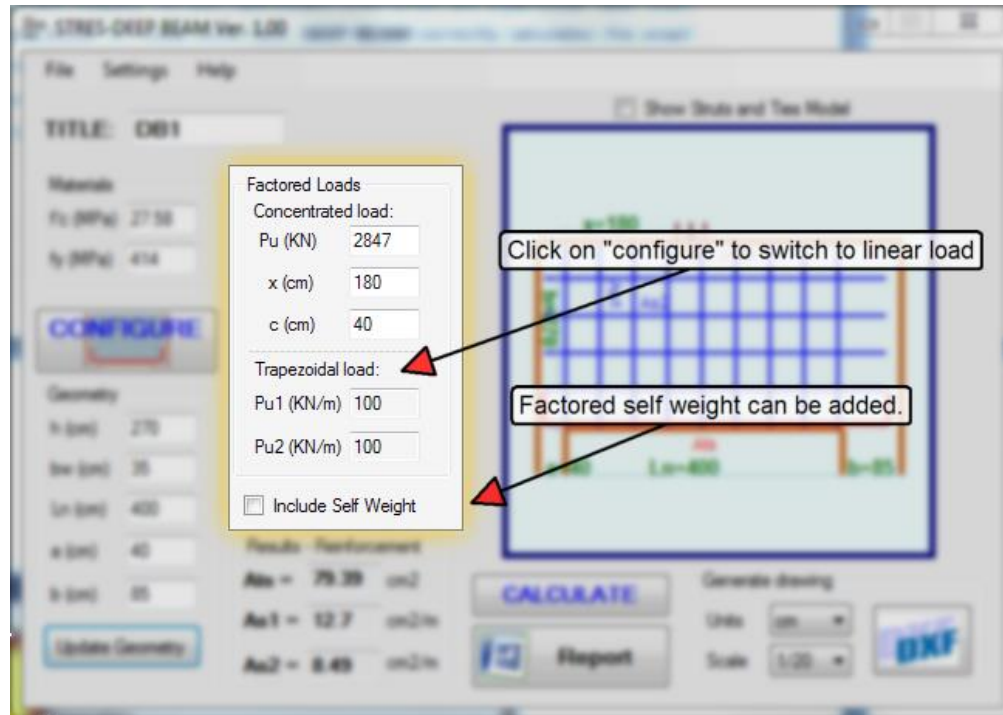
Main window



The interface of **STRES-DEEP BEAM** is clear and efficient. Main tools are directly accessible to the user.

- The “Update Geometry” button refreshes the graphic window. “b_w” is the width of the deep beam (out of the plane of the screen). “c” is the width bearing of the applied force P_u.
- The “Configure” button allows the user to switch between concentrated and linear loads. Furthermore, in case the beam has a very big height, the program fixes an effective depth for design purpose.
- The “Calculate” button verifies the introduced dimensions in function of the applied forces and generates the reinforcement.
- The “Report” button generates a complete Word report which includes equations, Code references and many figures showing the STM scheme and all nodal zones.
- The “DXF” button creates a design drawing compatible with AutoCAD. The drawing includes an elevation and a section both at scale with full annotations. Each component of the drawing (formwork, reinforcement, text...) is included in a separate layer. The layers feature allows the user once the drawing opened in the CAD software, to eventually change the font of the text or lines colors. This will help in customizing the drawing and integrating it within the user’s main details.

Applying loads



- Only factored concentrated force (number =1) and factored trapezoidal loads can be introduced
- It is possible to include self-weight of the deep beam (factored by 1.2, 1.35 or 1.4) which will be added to P_u or to P_{u1} and P_{u2} whether concentrated or linear loads are introduced. The self-weight is hence considered as a concentrated or linear load applied at the top of the beam.
- “c” is the width of the concentrated force P_u (can be that of the implanted column, for instance) required to design the nodal zone at this location.
- It is not possible to introduce concentrated and linear load simultaneously. User should click the “configure” button to choose.

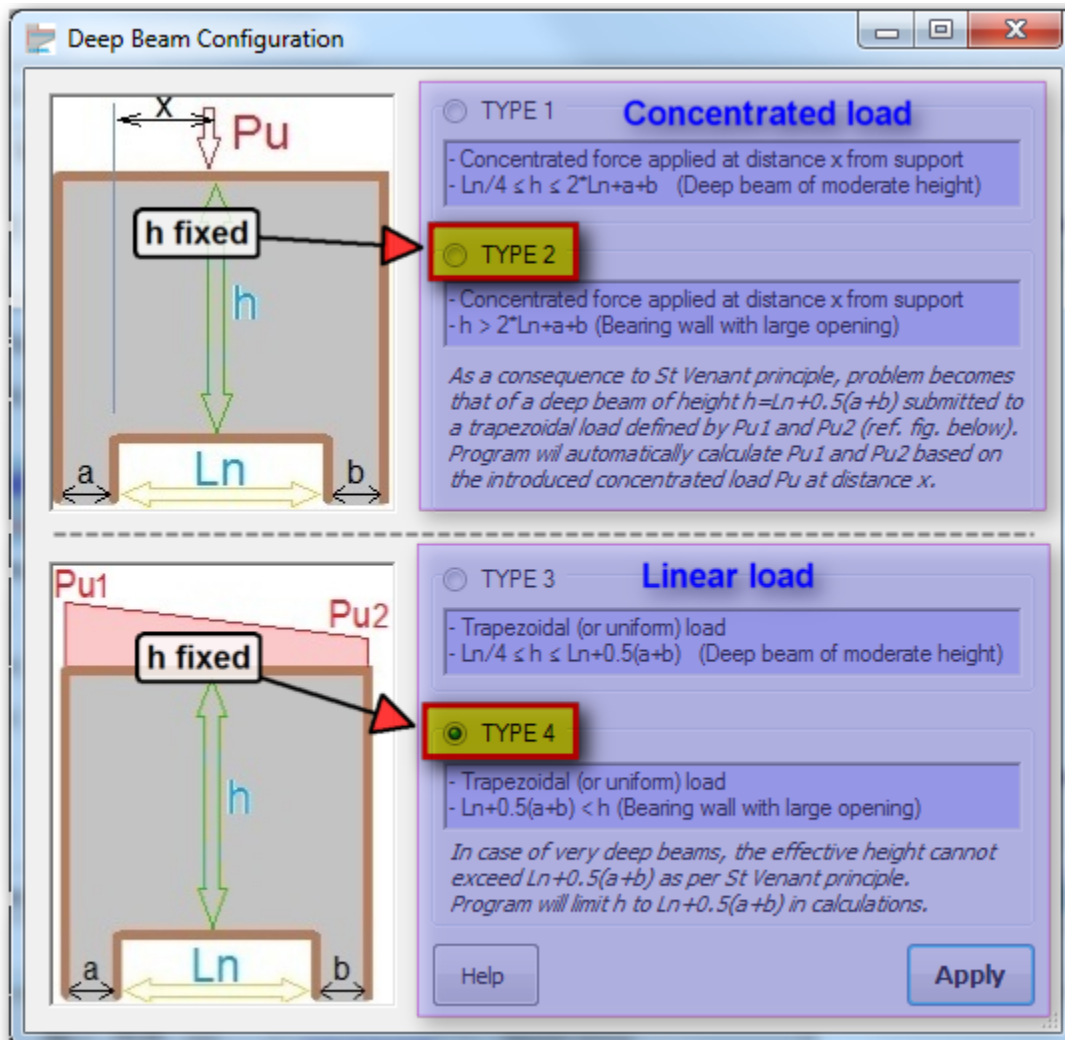
The “Configure” button

There are four main “types” of deep beams available in the software. In all cases the height of the deep beam should be bigger than its clear span divided by four.

1. **Type 1:** Deep beam with height less than two times its axial span, submitted to a concentrated force.
2. **Type 2:** Deep beam with height more than two times its axial span, submitted to a concentrated load. This configuration is similar to that of a bearing wall plain in the upper floors and having a large opening at the level of the ground floor. In this case, as per St-Venant principle, only the

part of the wall comprised within a height comparable to the axial span is a D-region (non-uniform deformation field). Since the top of the wall is submitted to a concentrated force, there too we have a disrupted region of height comparable to the axial span. Consequently, if the height of the beam is equal to two times its axial span, its section at mid-height becomes outside the D-region and remains plane after deformation (plane section assumption of flexure theory). **Therefore, it is possible to replace the concentrated force by an equivalent linear trapezoidal load defined by:**

- $P_{u1} = 2P_u \frac{L_n + b - x}{(L_n + a + b)^2}$
- $P_{u2} = 2P_u \frac{a + x}{(L_n + a + b)^2}$



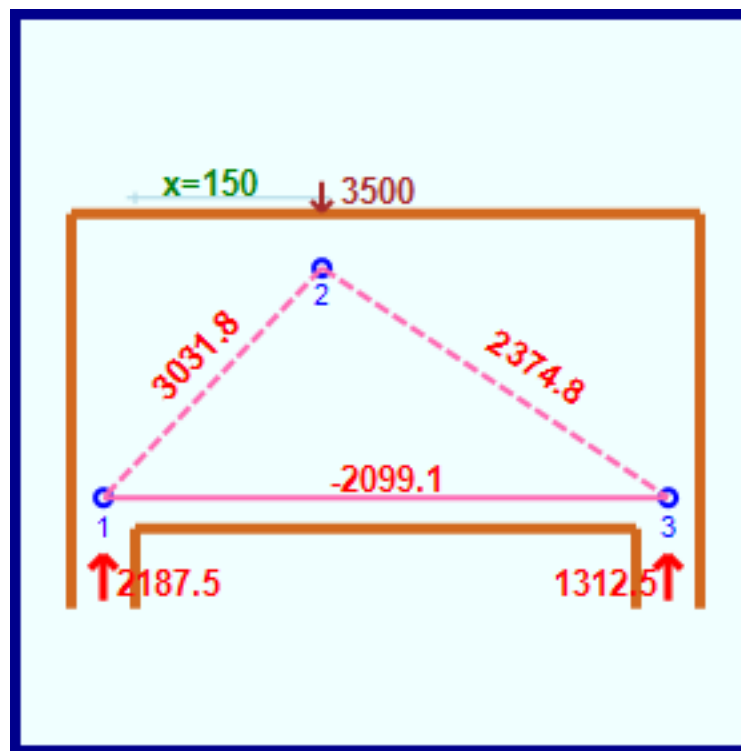
It is important to note that in case of type 2, the program will fix the height of the beam to $L_n + 0.5(a+b)$ which is equal to its axial span. The height field “h” becomes frozen. As per St Venant principle, the effective part of the beam conveying stresses to supports is located within a height close to its span. The

upper part is just transmitting a uniform normal stress field, undisrupted, and thus not participating in the resistance. When the beam is submitted to linear load and its height is more than its axial span, the effective height used for design cannot exceed $L_n+0.5(a+b)$.

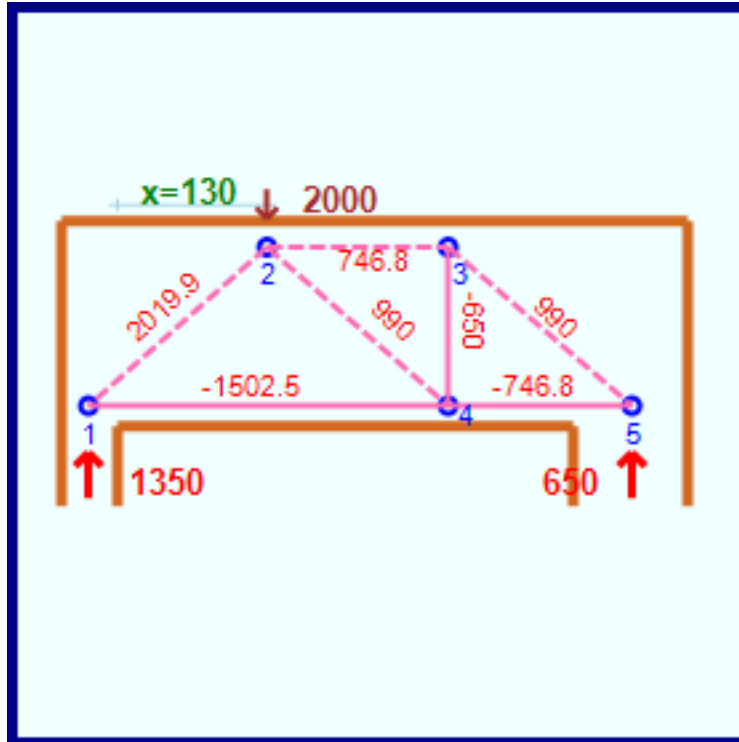
3. **Type 3:** Deep beam with height less than its axial span, submitted to linear load.
4. **Type 4:** Deep beam with height more than its axial span, submitted to linear load. The effective height for design is set equal to $L_n+0.5(a+b)$ for reasons explained above. **The height field "h" becomes frozen**

The Strut-and-Tie Model (STM)

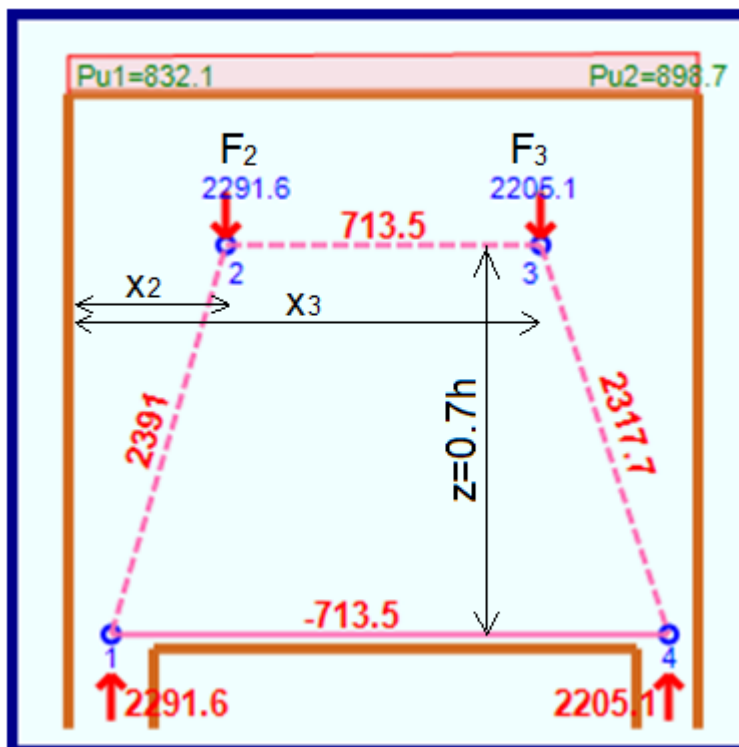
DEEP BEAM can generate several strut-and-tie schemes according to the nature of loading and to the limitations on struts inclination. In case of a **concentrated load**, the figure below shows a simple STM. The smallest allowable angle between a strut and a tie is 25° . Forces are in KN, positive for struts and negative for ties.



In case the angle between the strut and tie becomes less than 25° , then the program adopts a new scheme with additional struts and ties, respecting in all cases the angle limitation (Fig. below). Locations of nodes and mainly their distance from upper and lower faces of the beam are determined by the program in function of the sizes of the nodal zones.



In case of **linear loads** (or concentrated forces on a deep beam of big height, as per type 2) are applied, the adopted STM is shown in the figure below.



Here the program makes a conservative assumption by fixing the distance between strut 2-3 and tie 1-4 (lever arm) to 0.7h. All remaining dimensions (widths of struts and ties, sizes of nodal zones) are calculated.

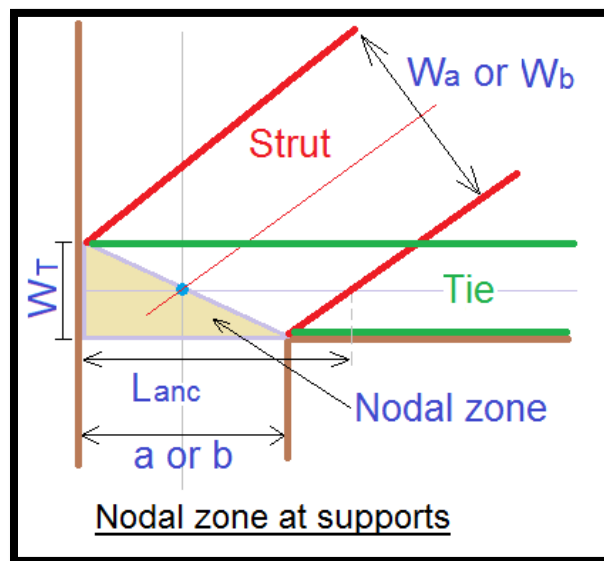
Linear load defined by P_{u1} and P_{u2} is transformed into concentrated loads F_2 and F_3 applied on nodes 2 and 3.

$$F_2 = \frac{1}{2} \left(\frac{L_n}{2} + a \right) \left(\frac{P_{u1} + P_{u2}}{2} + P_{u1} \right) \quad x_2 = \left(\frac{L_n + a + b}{3} \right) \left(\frac{2P_{u1} + P_{u2}}{3P_{u1} + P_{u2}} \right)$$

$$F_3 = \frac{1}{2} \left(\frac{L_n}{2} + b \right) \left(\frac{P_{u1} + P_{u2}}{2} + P_{u2} \right) \quad x_3 = \left(\frac{L_n + a + b}{2} \right) \left(1 + \frac{1}{3} \frac{5P_{u2} + P_{u1}}{3P_{u2} + P_{u1}} \right)$$

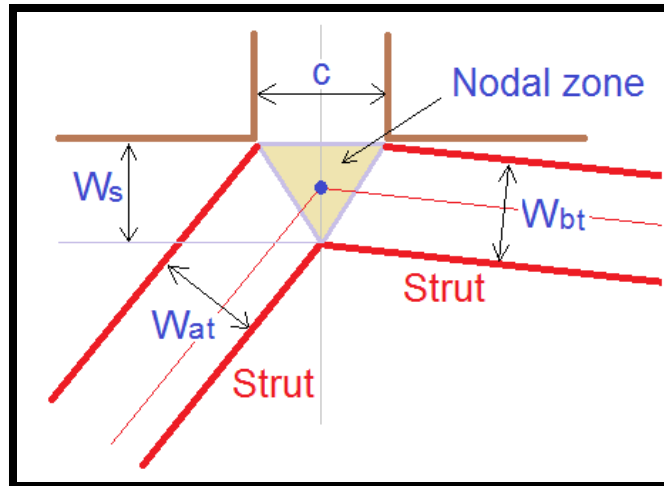
For all strut-and-tie models, nodes are located at the intersection of struts and ties and nodal zones have a triangular shape.

- Nodal zones at supports extend all over the support area. (Fig. below)



The software specifies the location of the critical section from which the development length of the main tension reinforcement is calculated. This section is located at the intersection of the tie centerline with the limit of the strut. The strength of the nodal zone is calculated as per §23.9.4. A cross-section is considered within the nodal zone, perpendicular to the strut.

- Nodal zone at the location of the concentrated load P_u has the same width than that “c” of the bearing surface of P_u (column’s size in case of implanted column). (Fig. below)



The height W_s of the nodal zone determines the location of the node and therefore the geometry of the whole STM. Thus, **the geometry of the STM depends on the intensity of the applied loads, for any STM configuration adopted by the software.**

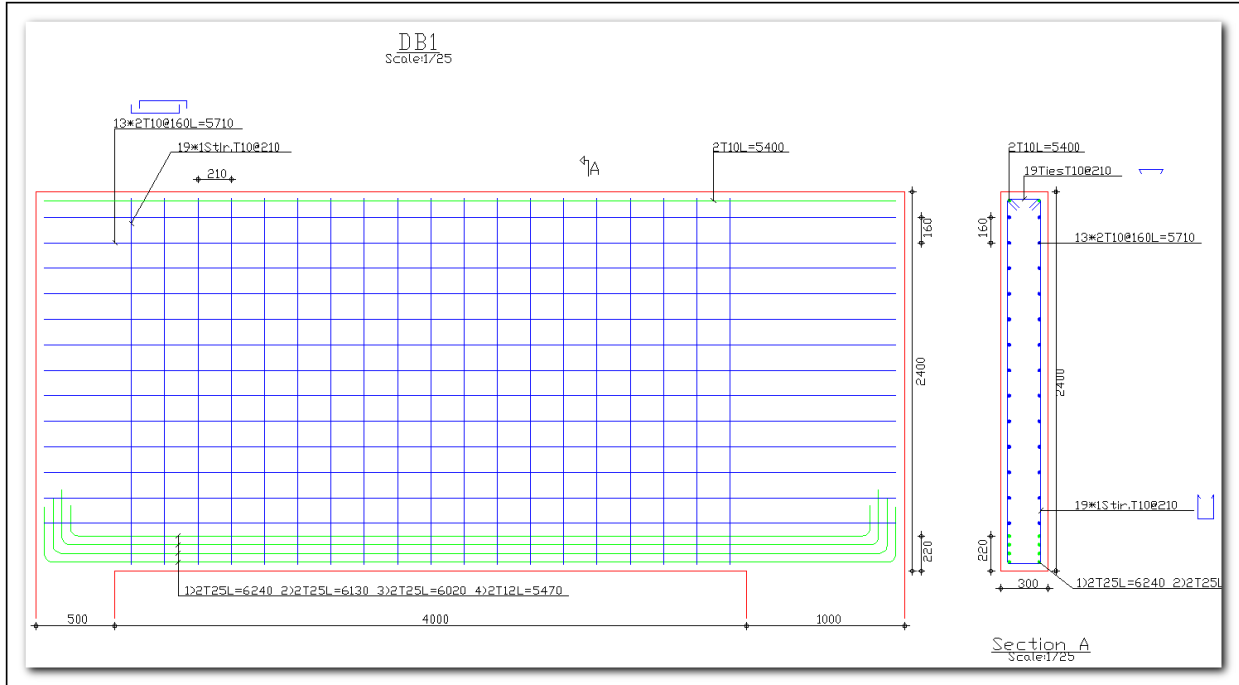
In the case of STM for beams submitted to linear loads, nodal zones 2 and 3 are not restricted by a particular dimension and the software optimizes their sizes in function of the forces in struts.

Generation of reinforcement

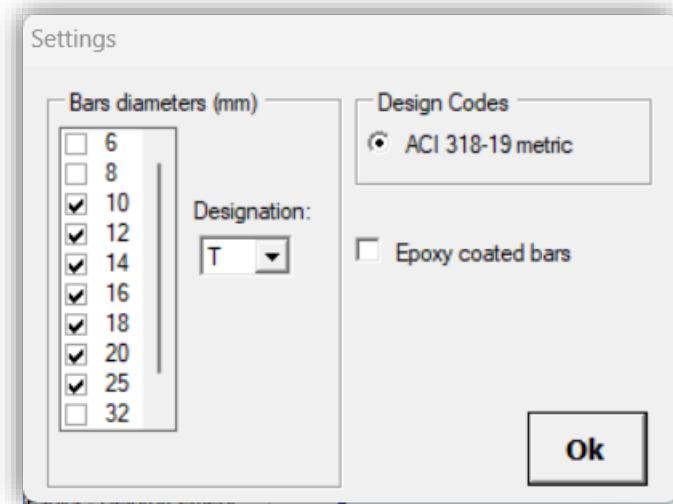
The program calculates three types of reinforcement.

- A_{ts} the main bottom reinforcement.
- A_{s1} and A_{s2} which are respectively the vertical stirrups and horizontal bars required to resist the splitting stresses inside the inclined struts in order to improve their axial strength.

The software always provides such reinforcement as per §23.4.3 of ACI318-19 and thus assumes that $\beta_s=0.75$ for inclined struts.



The **settings menu** allows customizing the diameters of bars to be used for drawing generation. Furthermore, this tool allows the user to control in an indirect way the number of bars **columns** and the number of bars **layers** as well regarding A_{ts} , the main bottom reinforcement.



Indeed, by preventing by example the software from using diameter 25mm, the remaining choices will be to use smaller diameters with additional layers, or to increase the number of bars columns within the width “ b_w ” of the section. These two alternatives are studied by the steel optimization engine of the program in order to provide at last the most economical reinforcement disposition.

The steel optimization engine investigates several solutions by varying the following parameters:

- The diameters to be used as per the list above.
- The number of bars to be disposed within the width of the beam without however transgressing the minimal clear distance between two bars as allowed by the Code.
- The number of layers.

The bottom main reinforcement always extends all over the supports since the width of the nodal zone is that of the supports. The program decides, based on the calculation of the available anchorage length, whether standards hooks are required or not.

Regarding the distributed reinforcement A_{S1} and A_{S2} , the software calculates the optimal spacing between two bars in order to match the theoretical reinforcement value obtained in calculations.