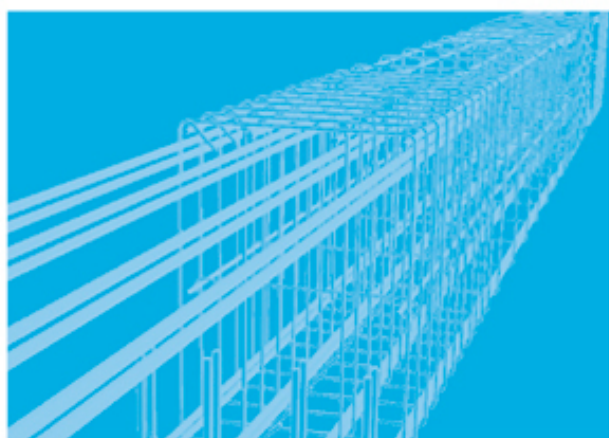




GRAITEC
G ADVANCE

TUTORIAL

G GRAITEC

www.graitec.com

Advance Concrete

Tutorial



Table of contents

About this tutorial	9
How to use this guide	10
Lesson 1: Creating a building grid.....	11
Step 1: Create a default building grid	11
Step 2: Set the distances between the axes using the Smartbar	12
Step 3: Modify the grid numbering properties	12
Step 4: Renumbering the grid axes	13
Lesson 2: Setting the building properties	15
Step 1: Set the building properties	15
Step 2: Set the level properties	15
Step 3: Activate the Layer Manager	16
Step 4: Define working units	17
Lesson 3: Creating building columns	18
Step 1: Create columns with L sections	18
Step 2: Create columns with T sections	21
Step 3: Create columns with rectangular sections	25
Lesson 4: Creating walls	27
Step 1: Create straight walls with 2 points	27
Step 2: Create continuous walls	28
Step 3: Create inner walls with 25 cm width	29
Step 4: Create inner walls with 30 cm width	30
Step 5: Create full brick walls	31
Lesson 5: Creating beams	34
Step 1: Create beams with rectangular sections	34
Step 2: Create continuous beams with rectangular sections	36
Step 3: Modify the beam properties	38
Lesson 6: Creating slabs	40
Step 1: Create slabs using automatic detection	40
Lesson 7: Creating ramps	42
Step 1: Define the ramp slope	42
Step 2: Create a ramp	45
Step 3: Adjust the slope position relative to the lower level	46
Step 4: Bind the ramp to the structural elements	47
Step 5: Adjust the ramp's auxiliary element position	48
Step 6: Create the second ramp surface	51
Step 7: Hide the control surface	52
Lesson 8: Creating openings	54

Step 1: Complete the model	55
Step 2: Create wall openings.....	60
Step 3: Create window openings	62
Step 4: Create the door openings.....	65
Step 5: Create the slab openings	67
Step 6: Add the openings required for the wiring	68
Step 7: Create beam opening lines	70
Step 8: Create a polygonal slab opening	75
Step 9: Complete the model by adding all remaining openings	76
Lesson 9: Creating an elevator pit	77
Step 1: Activate and view the levels	78
Step 2: Create the foundation slab	79
Step 3: Create the elevator pit.....	80
Step 4: Create upper depressions in the foundation slab	81
Lesson 10: Creating top and underground levels	84
Step 1: Create a new level above the building	84
Step 2: Modify the elevator walls.....	86
Step 3: Create the intermediate level	87
Step 4: Create the Ground Level.....	90
Step 5: Copy elevator elements to the Ground Level.....	91
Step 6: Modify the elevator wall heights	94
Step 7: Adjust the Ground Level connection elements	96
Lesson 11: Creating drawings	102
Step 1: Create section cuts	103
Step 2: Create a cut with parallel broken plane.....	104
Step 3: Create an elevation	105
Step 4: Create a top view	106
Step 5: Create a local top/bottom view.....	109
Step 6: Create an isometric view.....	112
Step 7: Modify and update drawings	114
Step 8: Group and ungroup the section cuts.....	116
Step 9: Create a reinforcement view	118
Step 10: Create a new empty drawing	121
Step 11: Create a plan view	122
Lesson 12: Creating dimensions.....	126
Step 1: Create a plan view	127
Step 2: Create intersection dimensions.....	128
Step 3: Display decimals and millimeters in the exponent	133
Step 4: Create an associative dimension	135
Step 5: Create a dimension by points	139

Step 6: Create a symbol.....	140
Step 7: Create multiple symbols.....	144
Lesson 13: Reinforcing a column	147
Step 1: Create the reinforcement drawing.....	148
Step 2: Create the necessary dimensions.....	151
Step 3: Create a rectangular frame on the top view.....	154
Step 4: Distribute the rectangular frame.....	158
Step 5: Create a list on the layout	163
Step 6: Add the longitudinal reinforcement	166
Step 7: Distribute the longitudinal reinforcement	169
Step 8: Place a reference mark on the distribution bars	170
Step 9: Create a one by one distribution of a straight bar	171
Step 10: Create a bar bending detail on the right bar of the Face elevation 1 cut.....	173
Step 11: Create a list.....	179
Step 12: View the drawing in 3D	181
Step 13: Group the 3D Reinforcement view with the Reinforcement column drawing	183
Lesson 14: Reinforcing a beam.....	184
Step 1: Activate the level	185
Step 2: Create a custom view arrangement.....	185
Step 3: Create the reinforcement drawing.....	187
Step 4: Create all the necessary dimensions	194
Step 5: Create the bar bending details.....	205
Step 6: Create a rectangular frame on Left cut A-A	208
Step 7: Create a linear distribution on Face elevation 1	216
Step 8: Add visual elements	218
Step 9: Renumber the bars	224
Step 10: Create a list on the layout	224
Step 11: View the reinforcement in 3D.....	225
Lesson 15: Creating a layout.....	228

About this tutorial

GRAITEC provides a multimedia tutorial to help you learn how to use various Advance tools.

The tutorial contains step-by-step instructions for creating Advance elements, drawings, layouts and understanding Advance functions.

Since Advance works with AutoCAD® and Autodesk Architectural Desktop® basic tools, the tutorial involves training of the methodologies used in these systems.

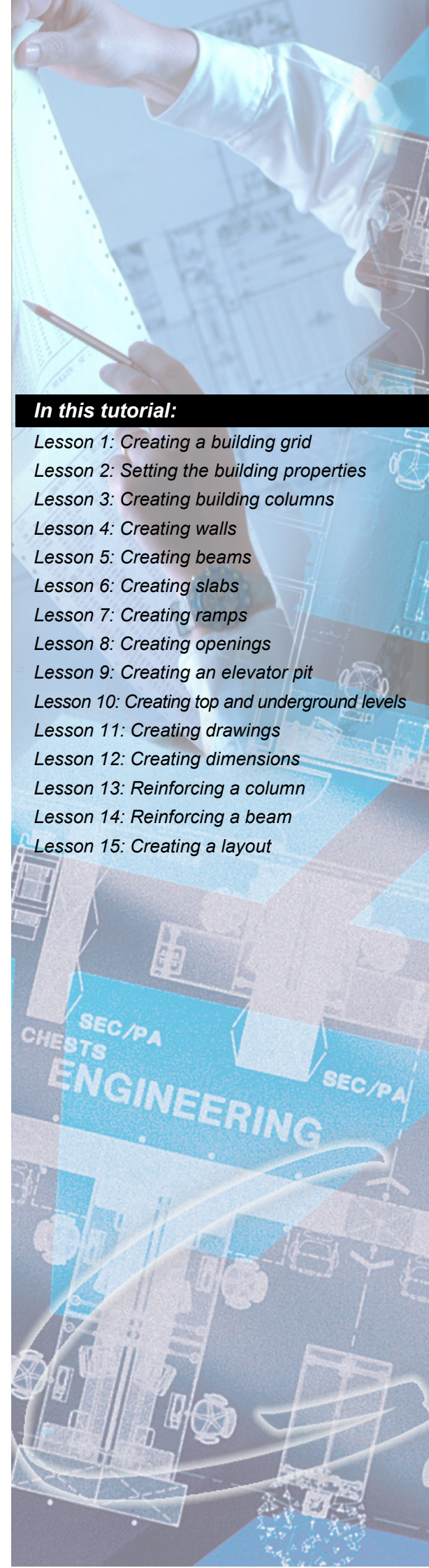
During the first 10 lessons, you will model the foundation of a building with two elevators.

In the following 2 lessons, the building model is used to create a set of structural and reinforcement drawings. The drawings are completed with all necessary visual elements (e.g., dimensions, symbols, annotations).

Two lessons are dedicated to the reinforcement module. You will learn to reinforce Advance basic elements: a column and a beam.

In this tutorial:

- Lesson 1: Creating a building grid*
- Lesson 2: Setting the building properties*
- Lesson 3: Creating building columns*
- Lesson 4: Creating walls*
- Lesson 5: Creating beams*
- Lesson 6: Creating slabs*
- Lesson 7: Creating ramps*
- Lesson 8: Creating openings*
- Lesson 9: Creating an elevator pit*
- Lesson 10: Creating top and underground levels*
- Lesson 11: Creating drawings*
- Lesson 12: Creating dimensions*
- Lesson 13: Reinforcing a column*
- Lesson 14: Reinforcing a beam*
- Lesson 15: Creating a layout*



How to use this guide

You can go through the tutorial and create the model from scratch or use the drawings on your documentation CD, or you can start any lesson by selecting the .dwg with the corresponding number.

Each lesson is presented in an .avi file.

 **Note:** *All drawings used in this tutorial are available in the Tutorial-DWG folder. If you want to save your work, backup the drawings.*

This tutorial does not explain all Advance features and commands. For more details, see the online help.

Lesson 1: Creating a building grid

In this lesson, you will create a building grid that is used in the following lessons of this tutorial.

You will learn how to:

- Create a grid.
- Set the number of axes and the distance between them using the Smartbar.
- Modify the grid representation.
- Modify the grid numbering.

The following examples describe how to create a building grid with axes in the X and Y directions, with the distances in Figure 1.

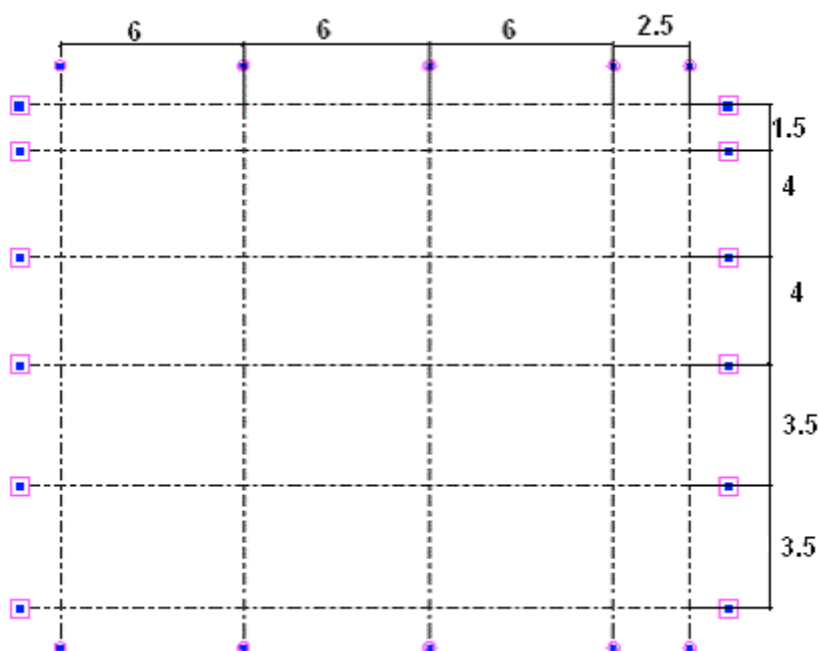


Figure 1: Building grid

Step 1: Create a default building grid

A building grid consists of axes in the **-X** and **-Y** directions and helps in placing the building elements on the layout. A building grid is created by specifying an origin point and a rotation angle.

1. On the **Model** toolbar, **Grid** flyout, click .
2. On the command line, enter the grid origin coordinates **0,0,0** and press **Enter**.
3. On the command line, enter a rotation angle value of **0°** and press **Enter**.

A grid with a default number of axes in the **X** and **Y** directions is created.

Step 2: Set the distances between the axes using the Smartbar

1. Click the grid to select it.
2. On the Smartbar, modify the number of axes and the distances in the -X and -Y directions by setting the number of spaces between the construction lines and the distance between two grid lines.
 - In the **Along X** field, enter **3x6 1x2.5** (an axes group consisting of 3 spaces of **6 m** and one space of **2.5 m** on X).
 - In the **Along Y** field, enter **2x4 2x3.5 1x1.5** (an axes group consisting of 2 spaces of **4 m**, 2 spaces of **3.5 m** and one space of **1.5 m** on Y).

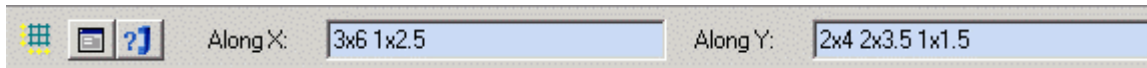


Figure 2: Setting the distances on the Smartbar

3. Press **Enter** to finish.

Note: After every edit on the **Smartbar**, press **Enter**.

Step 3: Modify the grid numbering properties

The axes of a grid are labeled using letters or numbers, starting from the desired letter or number. An **angle** for bubbles and **texts** are set for the extremities of the grid.

In this example, the axes in the X direction are labeled with numbers with the square type.

1. Click the grid to select it.
2. On the Smartbar, click to access the properties dialog box.
3. In the properties dialog box, select the **Bubbles** tab and make the following settings.
 - From the **Axis choice** drop-down list, select **X-axis** to number the axes in X direction.
 - From the **Bubble type** drop-down list select **Square**.
 - From **Angle** drop-down list select a **0°** bubble rotation angle.
 - From the **Position** drop-down list select **On the left and on the right** for the bubble positioning.

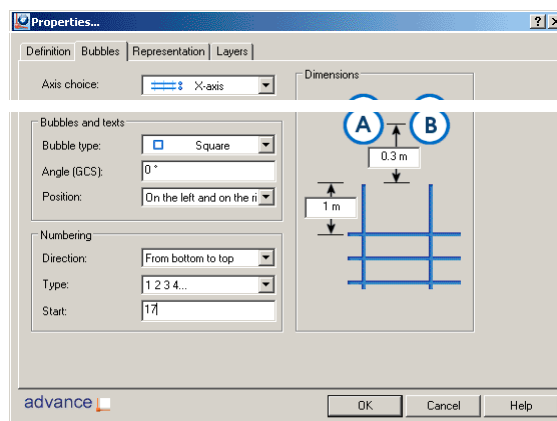


Figure 3: "Properties" dialog box – **Bubbles** tab

Note: The effect of the modifications from the dialog box is visible only after **OK** is clicked.

Select the direction of the numbering, the type of numbering and the starting value of the numbering.

In the **Numbering** area, set the numbering options:

- From the **Direction** drop-down list select **From bottom to top** to start the numbering from the bottom axis.
- From the **Type** drop-down list select **1 2 3 4 ...** to label the axes with numbers.
- In the **Start** field enter a start value of **17**.

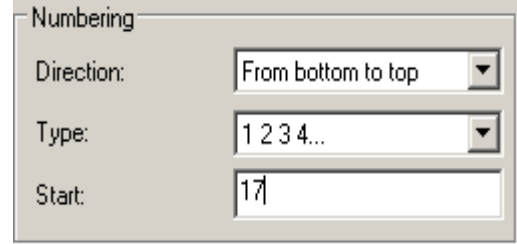
A screenshot of the 'Numbering' dialog box. It has three fields: 'Direction' with a dropdown menu showing 'From bottom to top', 'Type' with a dropdown menu showing '1 2 3 4...', and 'Start' with a text field containing '17'.

Figure 4: Numbering option for the **X** axis

Use the same process for axes in the **Y** direction:

- From the **Axis** choice drop-down list, select **Y-axis** to number the axes in the **Y** direction.

In the **Numbering** area, set the numbering options:

- From the **Type** drop-down list, select **A B C D...** to label the axes with letters.
- In the **Start** field, enter a start letter of **C**.

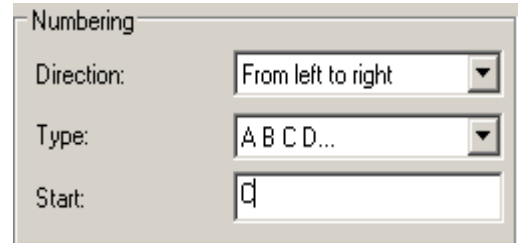
A screenshot of the 'Numbering' dialog box. It has three fields: 'Direction' with a dropdown menu showing 'From left to right', 'Type' with a dropdown menu showing 'A B C D...', and 'Start' with a text field containing 'C'.

Figure 5: Numbering option for the **Y** axis

2. Click **OK** to confirm.

The axes in the **Y** direction are labeled with letters from **C** to **G** and the axes in the **X** direction are labeled at both ends, with numbers from **17** to **22**, in square bubbles.

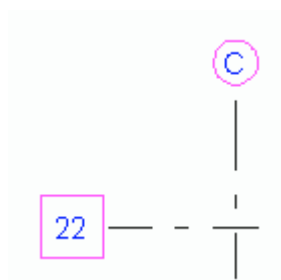


Figure 6: Labeled axis

Step 4: Renumbering the grid axes

You can renumber or rename the created axes with similar names or numbers. In our example, change the label of the **G** and **22** axes.

1. On the **Model** toolbar, **Grid** flyout, click .
2. Click the **G** axis. The **G** axis is highlighted in red and a context menu is displayed.

3. Select **F'** from the options in the context menu.

The selected axis is labeled with **F'**.

Using the same process, rename the **22** axis with **21'**.

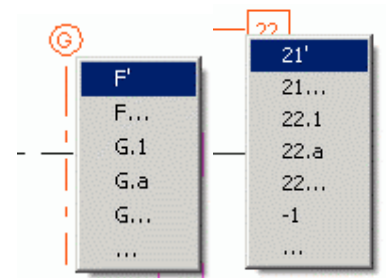


Figure 7: Renumbering the G and 22 axes

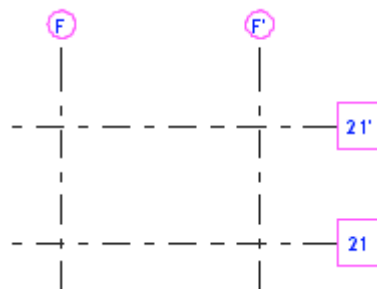


Figure 8: Renumbered grid axes

Lesson 2: Setting the building properties

During this lesson, you will name the building and set the default level height that is applied to every new created level of the building. In addition, the two levels created by default are renamed.

You will learn how to set:

- The building properties.
- The level properties.
- The project preferences.
- The working units.

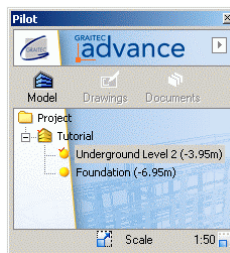


Figure 9: Building levels

Step 1: Set the building properties

This step describes how to set the building properties by setting a name, a default level height and the building altitude.

1. In the Pilot, click  to enter the **Model** mode. Under the project name, a tree structure shows the building and its levels.
2. Right click **Building** and select **Properties** from the context menu.
3. In the properties dialog box, make the following changes:
 - Set the building name to **Tutorial**.
 - Set the default level height to **3 m**.
 - Set the building altitude to **-6.950 m**.

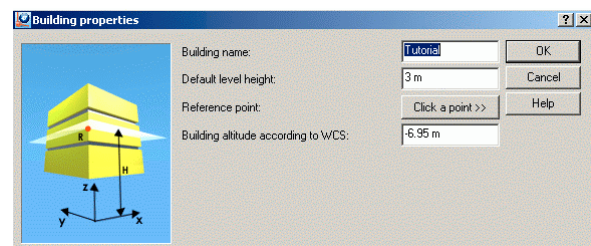


Figure 10: Building properties

4. Click **OK**.

Step 2: Set the level properties

In this step, the two levels of the building are renamed.

1. In the Pilot, click  to enter the **Model** mode.
2. Right click **Level 1** and select **Properties** from the context menu.

- In the properties dialog box, in the **Level name** field, enter the new level name: **Underground Level 2 (-3.95m)**.

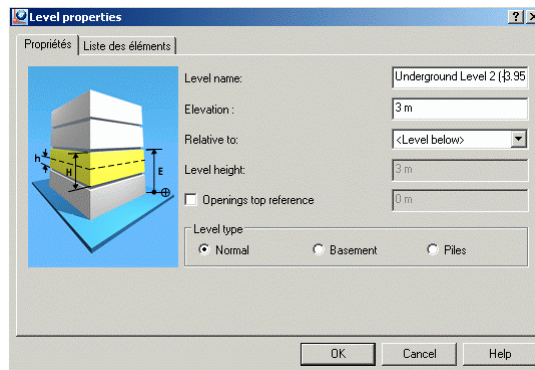


Figure 11: Renaming Level 1

- Set the building elevation to **3 m**.

Using the same process, rename the **Basement** level to **Foundation**.

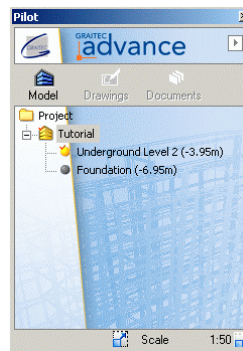


Figure 12: The Pilot displays the renamed levels

Step 3: Activate the Layer Manager

To define the general options for the project perform the following steps:

- In the Pilot, in **Model** mode, right click **Project** and select **Project settings** from the context menu.
- On the **Project Settings** toolbar, click to access the "Project preferences" dialog box.
- Select the **Layer naming** tab.
- Select the **Automatic layer name management** option.
- Click **OK**.

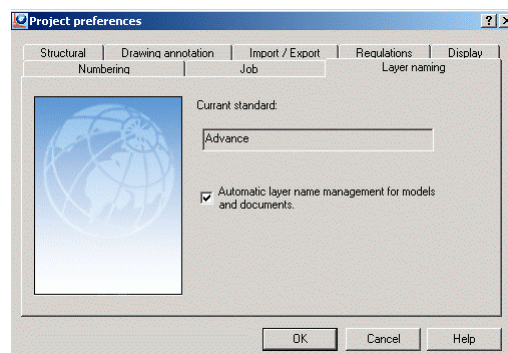


Figure 13: "Project preferences" dialog box - Layer naming tab

Step 4: Define working units

This step describes how to set the working units. The units appear in all the dialog boxes and apply to the element characteristics. The values selected are specified with decimal precision.

1. On the **Project Options** toolbar, click  to display the working definition dialog box. Define the following settings:
 - Click lengths "Type" and select **Meter**. In the "Precision" column select the **0.000** format. All the values entered on the properties bar and all the length values for structural objects are in meters.
 - Click section dimensions "Type" and select **Centimeter**. Set the precision to **0**.
 - Click openings dimensions "Type" and select **Centimeter**. Set the precision also to **0**.
 - For **Schedule Lengths**, set the precision to **0.00**.
 - For **Fabric Length**, set the lengths to **Centimeter** and the precision to **0**.
2. Click **Apply**.

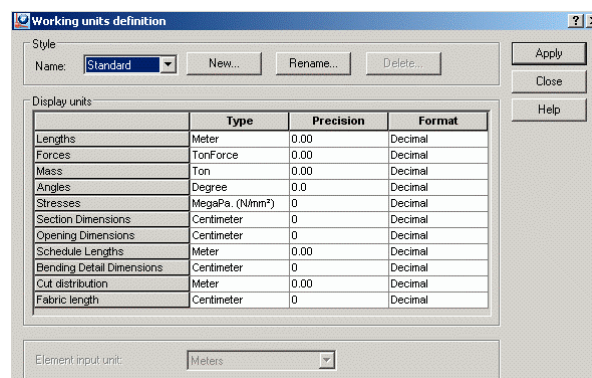


Figure 14: "Working units definition" dialog box

Lesson 3: Creating building columns

In this lesson, you will create columns with rectangular, T, and L sections in the **Underground** level.

You will learn how to:

- Create columns.
- Modify the column section.
- Use AutoCAD® tools to adjust and copy columns.
- Use the AutoCAD® tools to view the model in 3D.

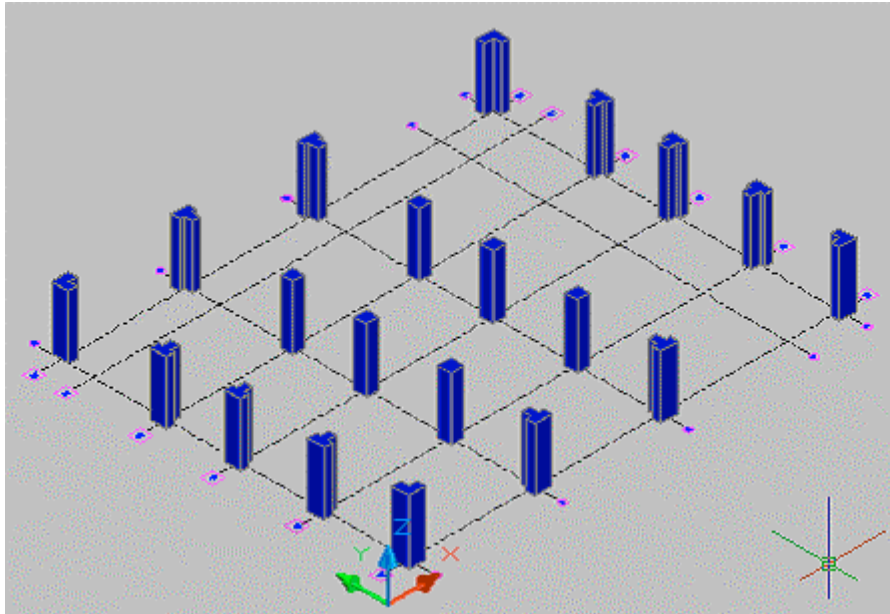


Figure 15: Columns in the model



*The columns are created at the axis intersection, therefore make sure that the AutoCAD® **Snap** mode is active.*

Step 1: Create columns with L sections

In this step, you will create columns with an L section in the **C17, F'17, F'21', C21'** axis intersections (i.e., the corners).

1. On the **Model** toolbar, **Structural** flyout, click .
2. On the Smartbar, click  to select another section.

3. In the section properties dialog box, modify the section size (e.g., height, width, etc.).
 - From the **Material** drop-down list, select **Precast concrete**.
 - From the **Type** drop-down list, select the **PF16 Section** (L shape) section type.
 - In the **Description** and **Value** columns, enter the values shown in the Figure 16.

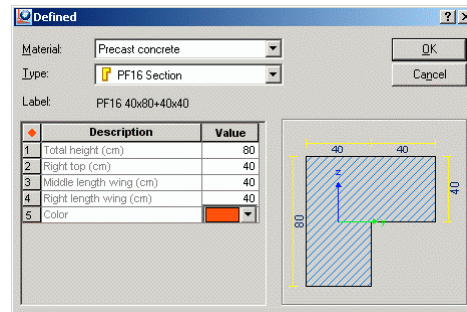


Figure 16: Measurements for the L section columns

4. Click **OK**.
5. Place the first column at the **C21'** intersection.
6. Click to indicate the column orientation angle (**90°**).
7. Press **Enter**.

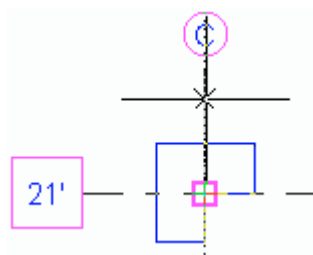


Figure 17: Creation of the first column

8. Using the same process, create columns at the **F' 21'** (0°), **F' 17** (270°), and **C 17** (180°) axis intersections.

Rotation angle	0°	90°	180°	270°
Result				

Figure 18: Columns – Rotation angle

9. Press **Esc** to finish.

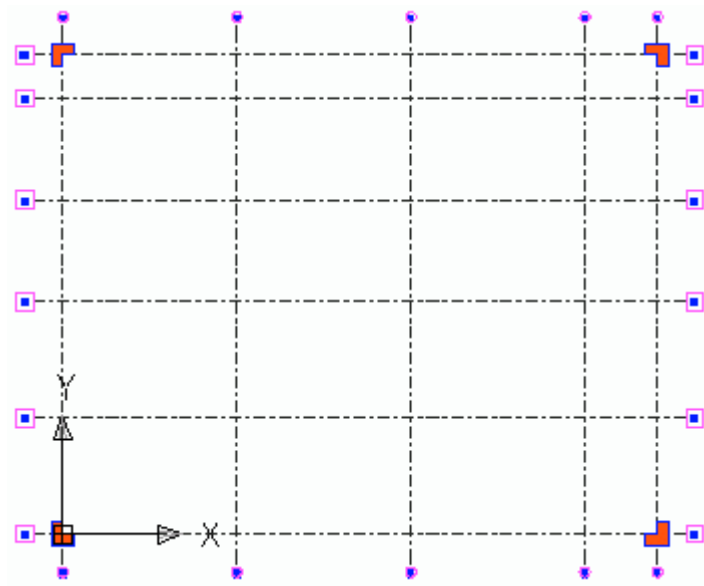


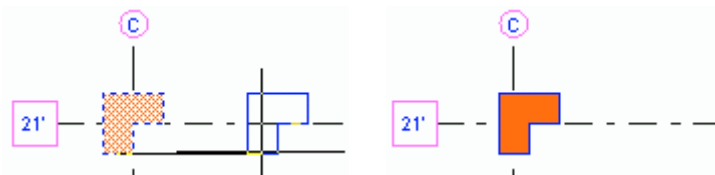
Figure 19: L section columns



Adjusting column positions

Use the AutoCAD® **Move** command to adjust the position of the columns **20 cm** along the **X** and **Y** directions so that the grid axes are over the section axes.

1. Select the **C21'** and **C17** columns.
2. On the command line, type **m** (move) and press **Enter**.
3. Specify a base point.
4. To move the columns, move the cursor in the X direction and enter a distance of **0.20**.

Figure 20: Moving the **C21'** column

5. Press **Enter**.

Repeat the command to move the columns **0.20 m** in the X and Y directions to place them as in the Figure 21.

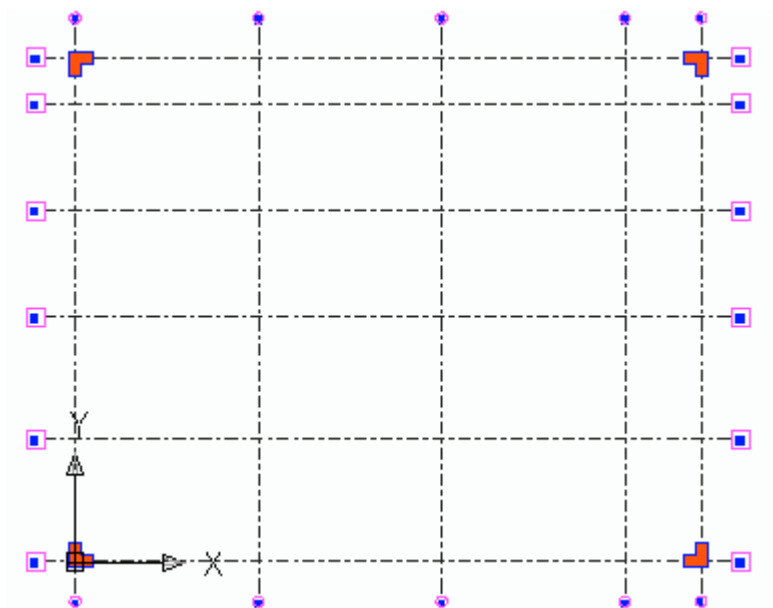


Figure 21: L columns – adjusted position

Step 2: Create columns with T sections

This step describes how to create columns with T sections with different rotation angles and how to modify their properties.

1. On the **Model** toolbar, **Structural** flyout, click .
2. On the Smartbar, click  to select another section.
3. In the section properties dialog box, select a section type and modify the size.

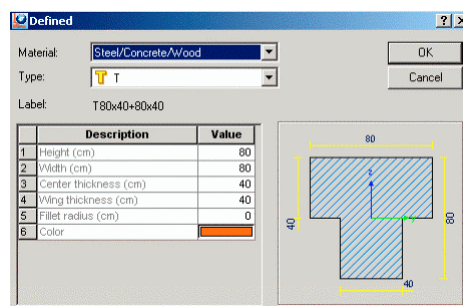


Figure 22: Measurements for the T section columns

- From the **Material** drop-down list, select the **Steel/Wood/Concrete**.
 - From the **Type** drop-down list, select the **T** section.
 - Modify the section size: Set the **Height** and **Width** to **80**, and the **Center** and **Wing** to **40**.
4. Click **OK**.

5. Place the first column at the **D21'** intersection.
6. Set the rotation angle to **0°**.

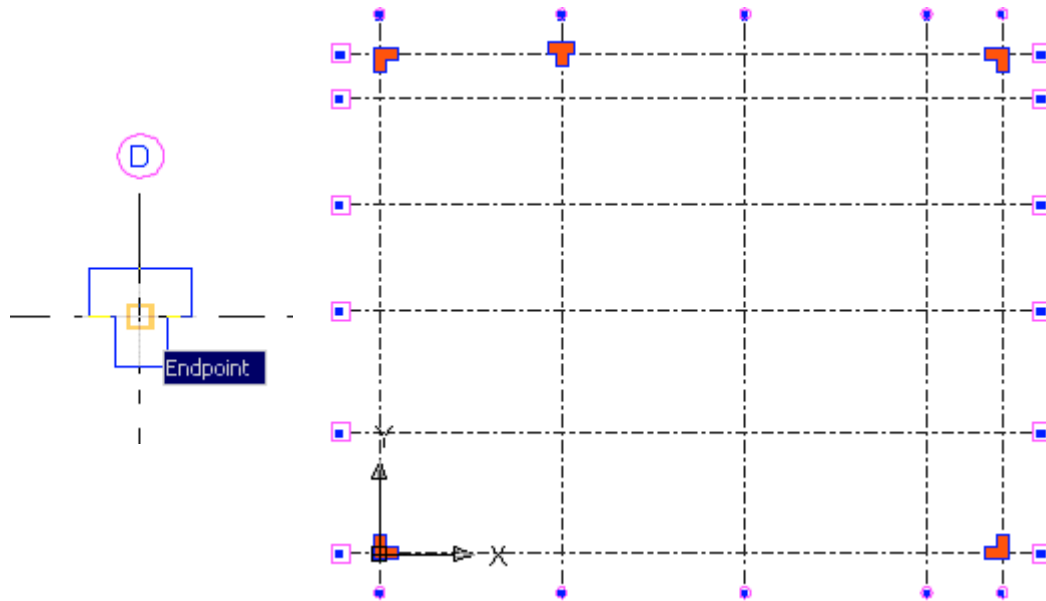
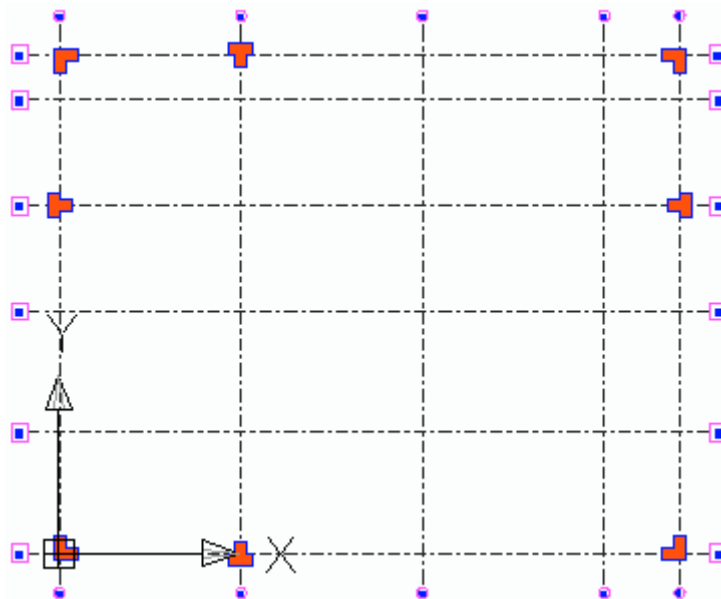


Figure 23: Creating the first T column

7. Place columns at the following intersections: **C 20** (90°), **F'20** (270°) and **D17** (180°).

Figure 24: Creation of the **C20**, **F'20** and **D17** columns

Adjusting column positions

As in the previous step, use the AutoCAD® **Move** command to adjust the column position **20 cm** in the **X** or **Y** direction.

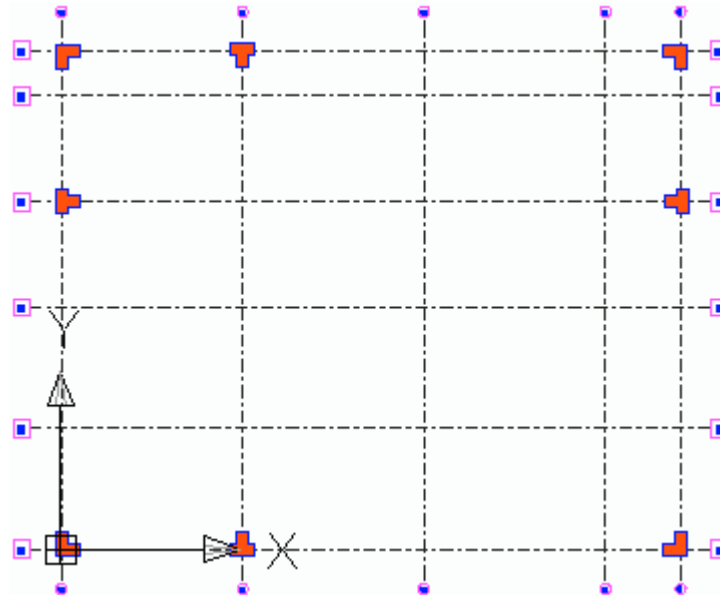


Figure 25: Adjusted T section columns

Using the AutoCAD® Copy tool

Use the AutoCAD® **Copy** tool to create new columns with T sections.

1. Include the **C20** column in a window selection.
2. On the command line, type **cp** (Copy) and press **Enter**.
3. Specify the base point for the selected column at the axis intersection.
4. Click the **C19** and **C18** intersections to create the columns.

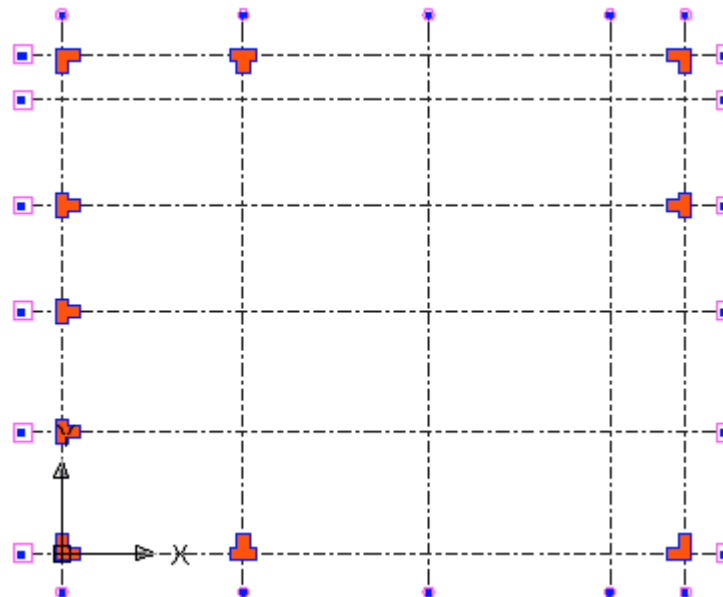
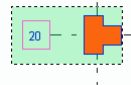


Figure 26: Creation of the **C18** and **C19** columns

Repeat the AutoCAD® **Copy** command and copy the **F'20** column at **F'19** and **F'18** intersections.

Next, copy the **D21'** and **D17** columns at the **E21'** and **E17** intersections.

1. Select the **D21'** and **D17** columns.

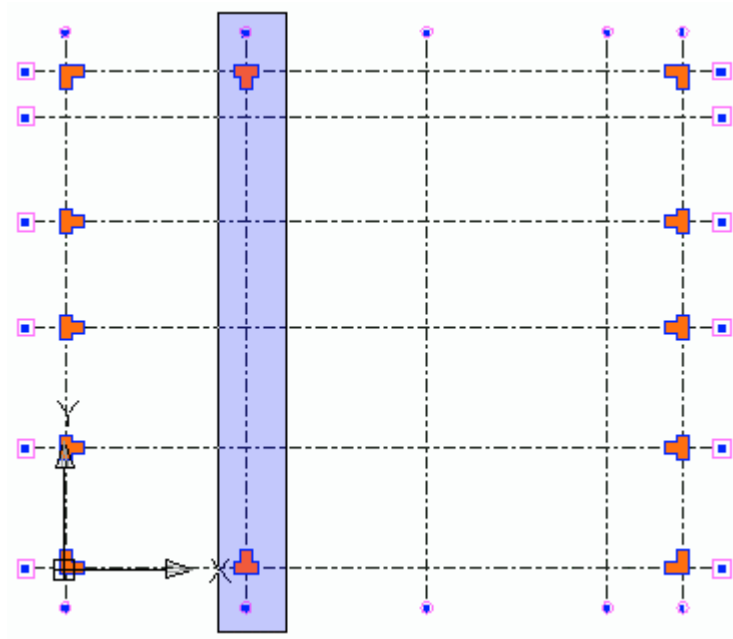


Figure 27: Selecting the **D21'** and **D17** columns

2. On the command line, type **cp** (Copy) and press **Enter**.
3. Specify the base point for the selected columns at the **D21'** axis intersection.
4. Click the **E21'** and **E17** intersections to create the columns.

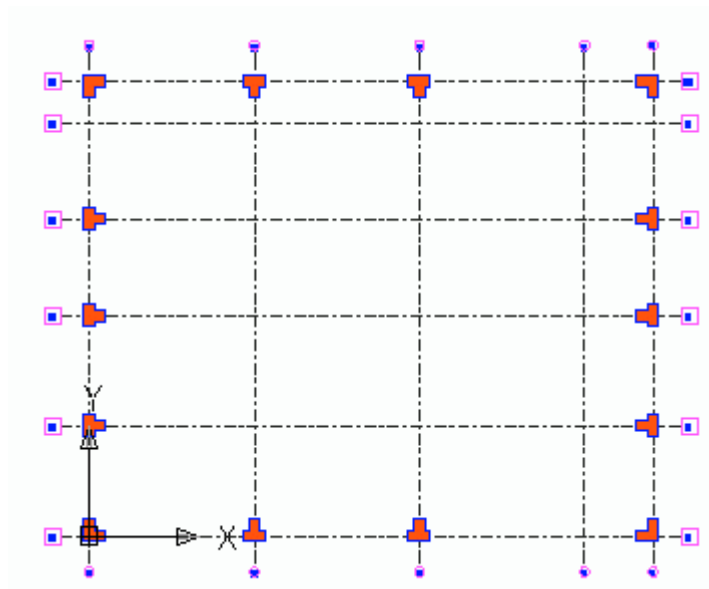


Figure 28: Creation of the **E21'** and **E17** columns

Step 3: Create columns with rectangular sections

Next, create columns with rectangular sections.

1. On the **Model** toolbar, **Structural** flyout, click .
2. On the Smartbar, click  to select another section.
3. In the section properties dialog box, modify the column size.

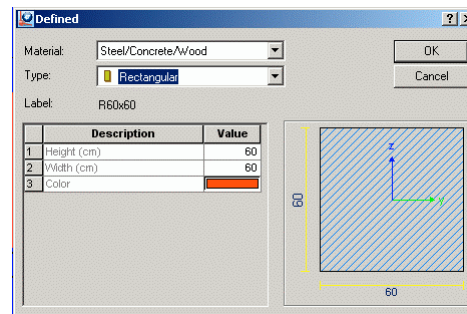
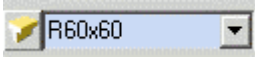


Figure 29: Measurements for the rectangular column sections

- From the **Material** drop-down list, select a material.
 - From the **Type** drop-down list, select the **Rectangular** type.
4. Click **OK**.
 5. On the Smartbar, set the size of the column to **R60x60**. 
 6. Place the columns at the **D18**, **D19** and **D20** axis intersections, with a **0°** rotation angle.
 7. Place the other columns at the **E18**, **E19** and **E20** intersections.

The following figure shows the result.

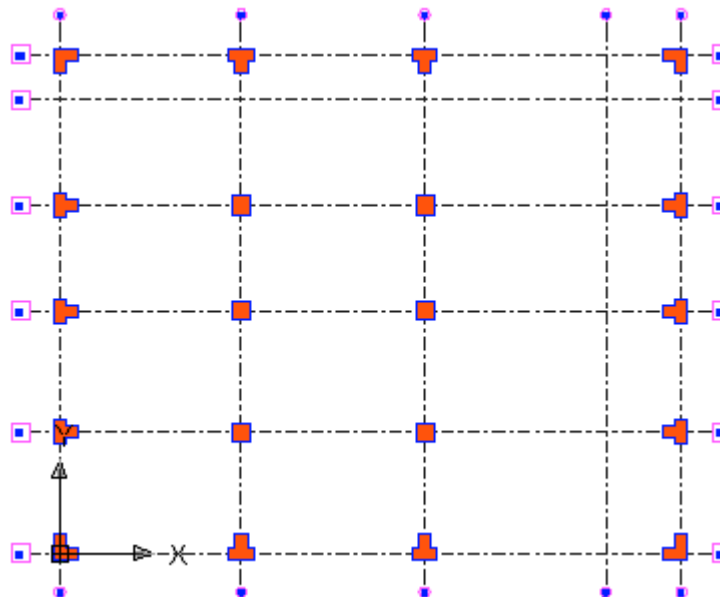


Figure 30: Creation of the **D18**, **D19**, **D20** and **E18**, **E19**, **E20** columns

Using AutoCAD® tools to view the building

To change the view angle, use the buttons on the AutoCAD® **View** toolbar.



Figure 31: AutoCAD® **View** toolbar

For a more realistic presentation of the model, use the AutoCAD® **Visual Styles**.



Figure 32 AutoCAD® **Visual Styles** toolbar

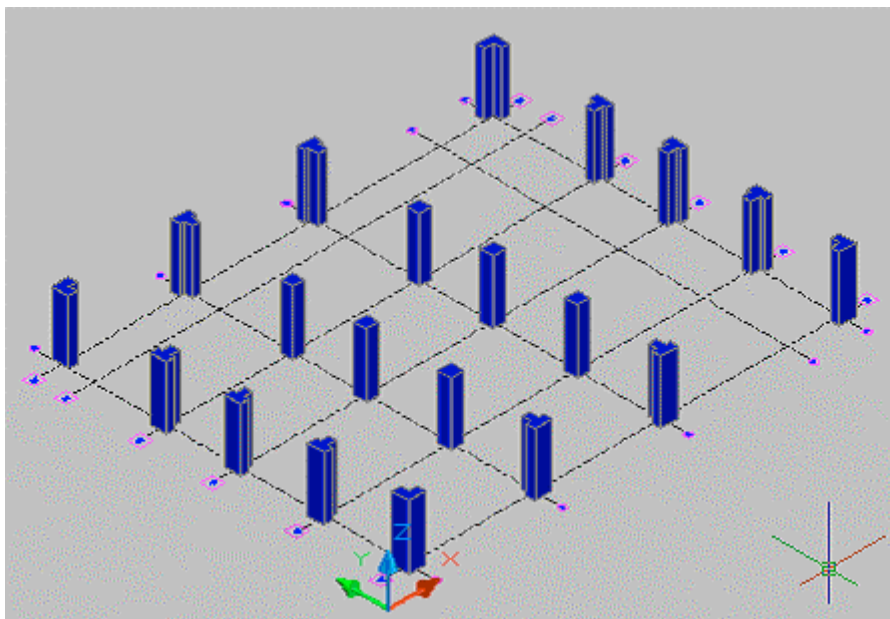


Figure 33: Columns in 3D view

Lesson 4: Creating walls

In this lesson, you will create walls in the **Underground level**. The outer walls are made of concrete, while the inner walls are thicker and made of bricks.

You will learn how to:

- Create straight walls with two points.
- Create continuous walls.
- Adjust the position of a wall.
- Modify the wall properties (e.g., material, thickness, etc.).
- Use the AutoCAD® **View** toolbar for a 3D view of the model.

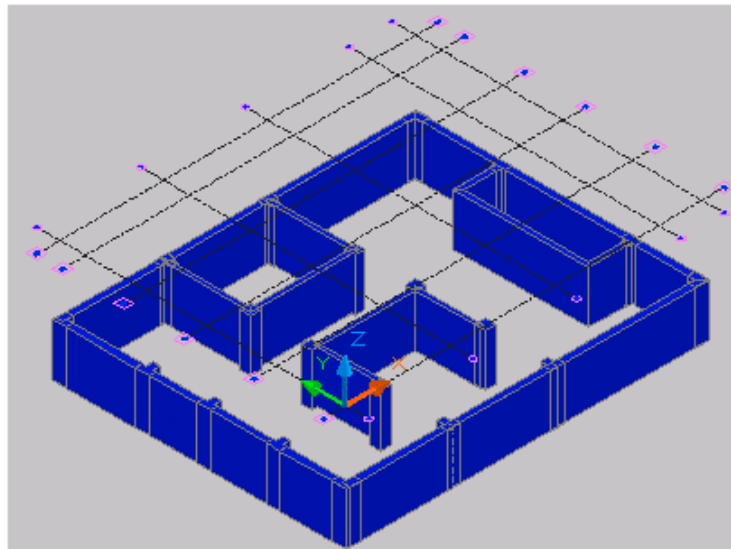


Figure 34: Walls of the Model

Step 1: Create straight walls with 2 points

In this step, 4 straight walls are created along the **C** axis, between the **C21'**, **C20**, **C19**, **C18**, **C17** columns.

1. On the **Model** toolbar, **Structural** flyout, click .
2. On the Smartbar, set the wall thickness to **40 cm**  and press **Enter**.
3. Define the wall start point (**C21'** column section – the middle of the bottom edge).

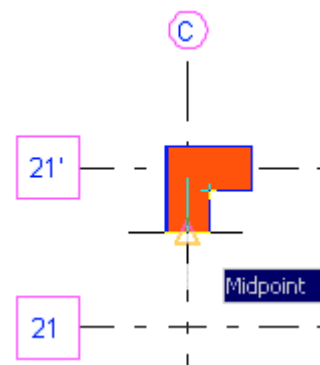


Figure 35: First point of the first straight wall

- Define the end point (**C20** column section – the middle of the top edge).

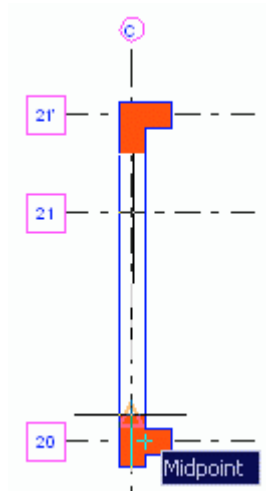


Figure 36: Setting the end point of the first straight wall

- Press **Esc** to finish.

Using the same process, draw the other walls along the C axis: **C20 – C19**, **C19 – C18**, and **C18 – C17**.



Use the AutoCAD® **Repeat** command, to repeat the last executed command – in this case, **grtccolumn**.

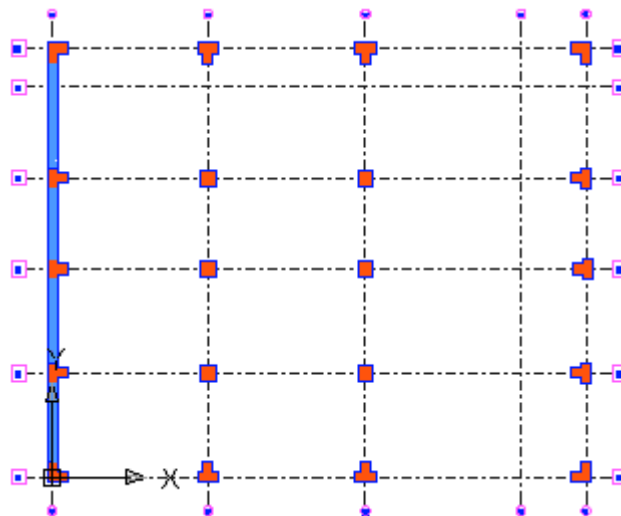


Figure 37: Walls along the C axis

Step 2: Create continuous walls

In this step, create a wall made up of several segments between the **C17**, **D17**, **E17**, **F'17**, **F'21'** and **C21'** columns.

- On the **Model** toolbar, **Structural** flyout, click .
- Click the **C17** axis intersection to define the start point of the wall.

- Click **D17** axis intersection to set the end point.

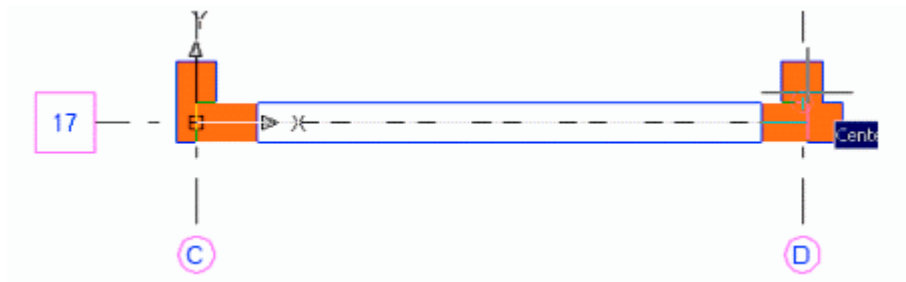


Figure 38: Creation of continuous wall between the **C17** and **D17** axis intersections

- Draw the walls by clicking the following intersections: **E17**, **F'17**, **F'21'** and **C21'**.
- Press **Esc**.

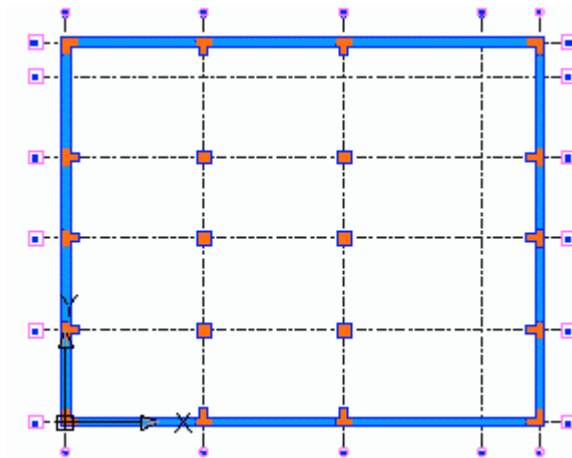
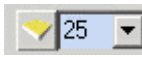


Figure 39: **Underground level** – columns and outer walls

Step 3: Create inner walls with 25 cm width

Next, create full brick walls with 25 cm width between the **D21'**, **D20**, **E20** and **E21** axis intersections.

- On the **Model** toolbar, **Structural** flyout, click .
- On the Smartbar, enter **25** to draw a wall with a **25 cm** width .
- Press **Enter**.
- Click the start point at the **D21'** axis intersection.

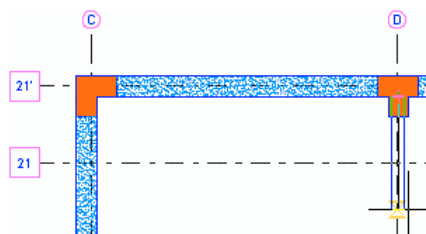


Figure 40: Defining the start point at the **D21'** intersection

Continue to draw walls by clicking the **D20**, **E20** and **E21'** axis intersections.

5. Press **Esc**.

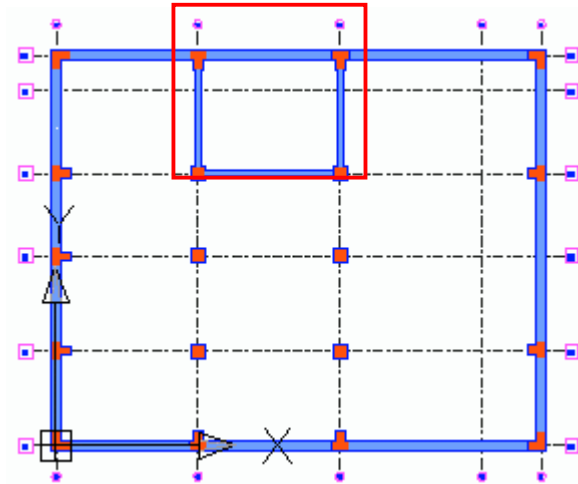


Figure 41: Inner walls

Step 4: Create inner walls with 30 cm width

1. On the **Model** toolbar, **Structural** flyout, click .
2. On the Smartbar, from the **Width** drop-down list, select **30 cm** for the wall width.
3. Click the start point of the first wall at the **F'20** axis intersection.

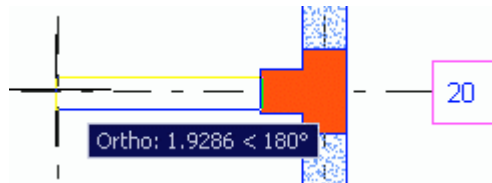


Figure 42: Creation of the first inner wall

4. Continue to draw walls by clicking the **F20**, **F18** and **F'18** axis intersections.
5. Press **Esc**.

Note: Delete the T section column at the **F'19** axis intersection.

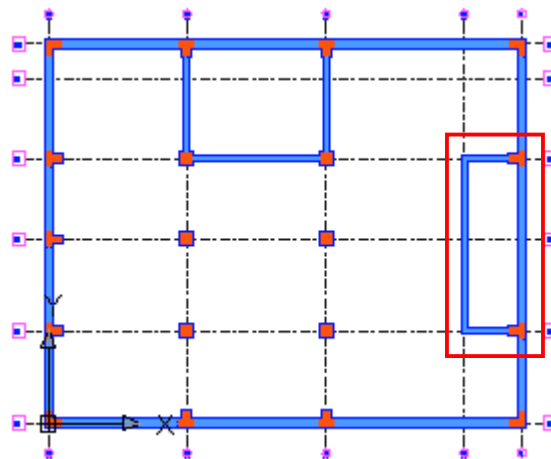


Figure 43: Underground level –outer and inner walls

Step 5: Create full brick walls

This step describes how to create full brick walls between the **D18**, **D19**, **E18** and **E19** axis intersections.

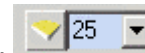
Creating a wall

1. On the **Model** toolbar, **Structural** flyout, click .
2. On the Smartbar, define the following settings:

- From the **Material choice** drop-down list, select **Full Bricks**.



- From the **Width** drop-down list, select **25 cm** for the wall width.



3. Press **Enter**.

To draw the first wall:

- Click the **D19** axis intersection to define the wall start point.
- Click the **E19** axis intersection to define the end point and press **Enter**.

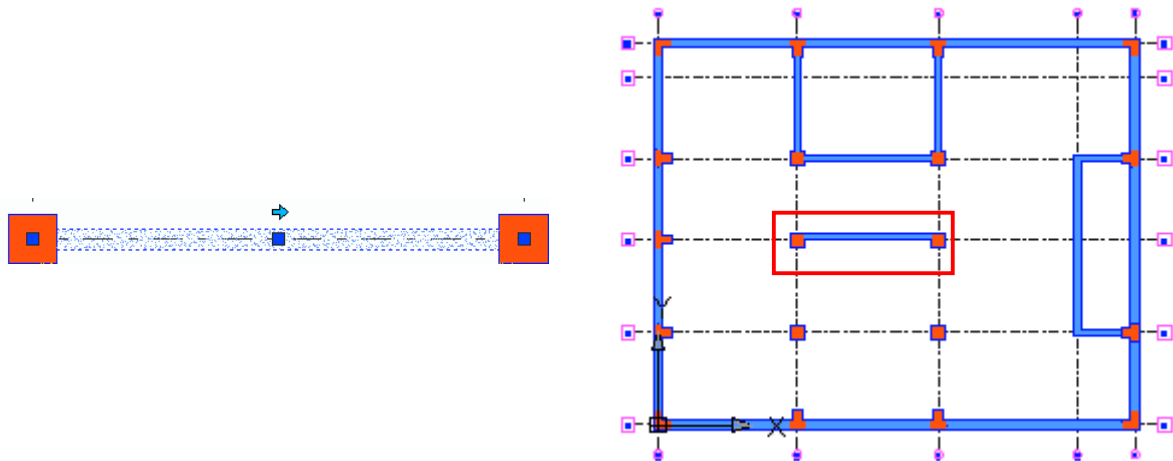


Figure 44: First brick wall: **D19 – E19**

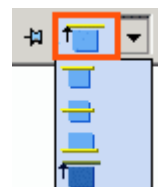
Adjusting the wall position

Next, align the wall to the bottom edge of the column section. The position of a wall is adjusted using the **Justification** parameter.

1. Select the wall between the **D19** and **E19** axis intersections.



2. On the Smartbar, from the **Justification** drop-down list, select the custom justification.



3. Enter **-0.175 m** for the justification value and press **Enter**.

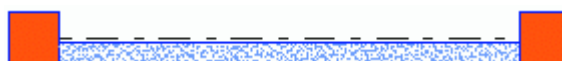


Figure 45: Wall aligned to the bottom edge of the column section

Next, create the other full brick walls.



Use the AutoCAD® **Repeat** command, to repeat the last executed command – in this case, *grtccolumn*.

1. On the **Model** toolbar, **Structural** flyout, click .
2. Click the start point at the bottom right corner of the **D19** column.

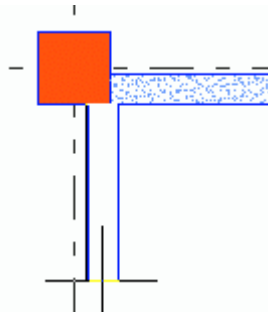


Figure 46: Creation of a wall along the **D** axis

3. Click the end point at the top right corner of the **D18** column.
4. Press **Esc**.

Using the same process, draw another full brick wall between the **E19** and **E18** axis intersections.

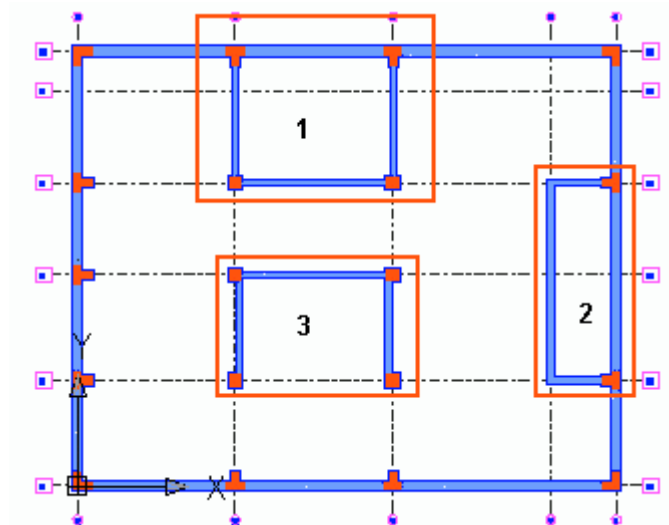


Figure 47: **Underground level** – outer and inner walls

Changing the wall representation style

1. Select the **D19-D18-E18-E19** full brick walls.
2. On the Smartbar, from the **Style** drop-down list, select **Full bricks** to change the wall representation style.
3. Press **Enter**.

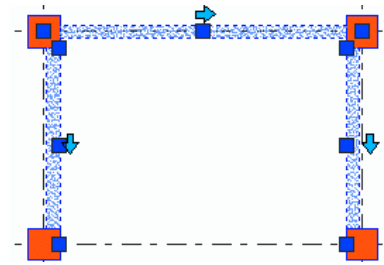
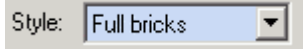


Figure 48: D19-D18-E18-E19 walls

The wall representation is changed.

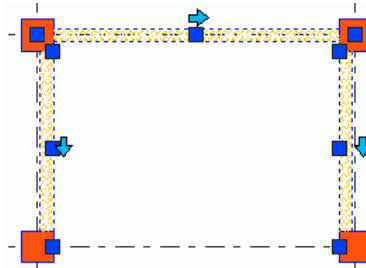


Figure 49: New representation of the walls



Using AutoCAD® tools to view the building

To change the view angle, use the buttons on the AutoCAD® **View** toolbar.



Figure 50: AutoCAD® View toolbar

For a more realistic presentation of the **Model**, use the AutoCAD® **Visual styles** toolbar.



Figure 51: AutoCAD® Visual Styles toolbar

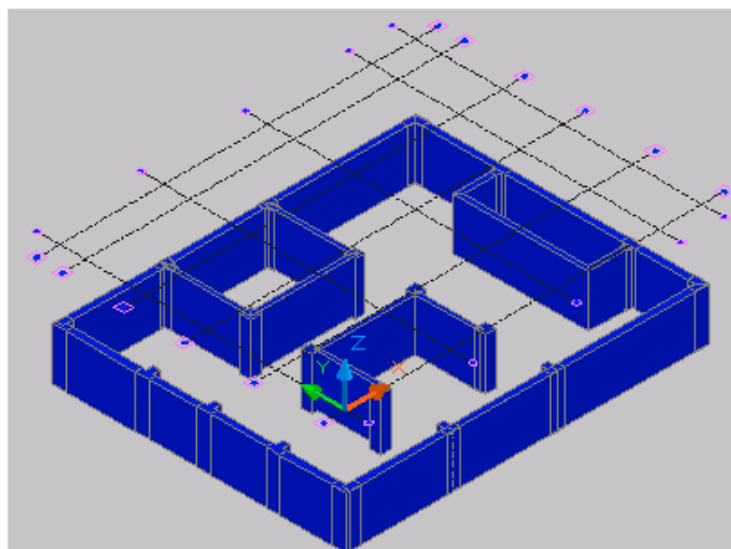


Figure 52: Walls of the model

Lesson 5: Creating beams

In this lesson, you will create beams in the Underground level.

You will learn how to:

- Create straight beams.
- Adjust beam positions.
- Modify beam properties.

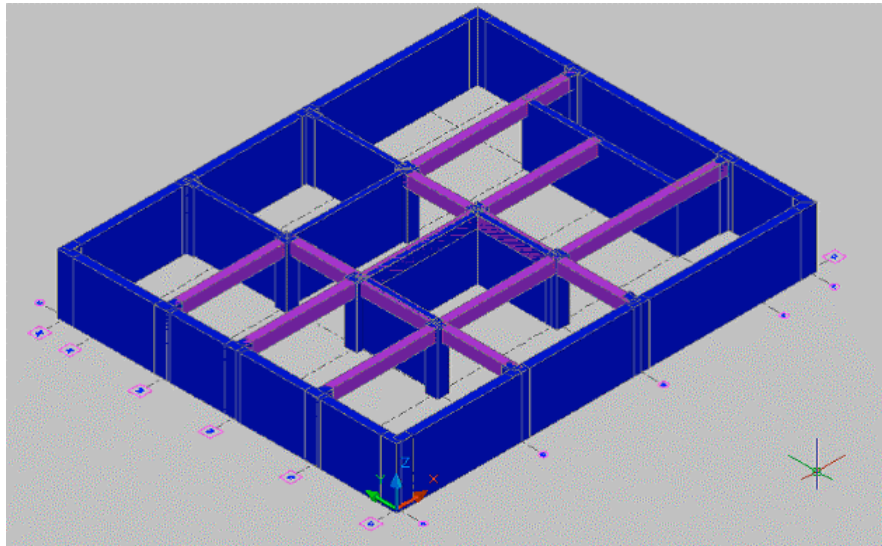
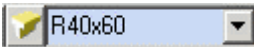


Figure 53: Final Model

 **Note:** Make sure the **Snap** mode is active.

Step 1: Create beams with rectangular sections

This step describes how to create beams and how to modify their properties (e.g., material, type, etc.).

1. On the **Model** toolbar, **Structural** flyout, click .
2. On the Smartbar, set the beam section size to **R40x60**. 
3. Press **Enter**.
4. Draw the first beam from **C20** to the **D20** axis intersection.
5. Press **Esc** to finish.

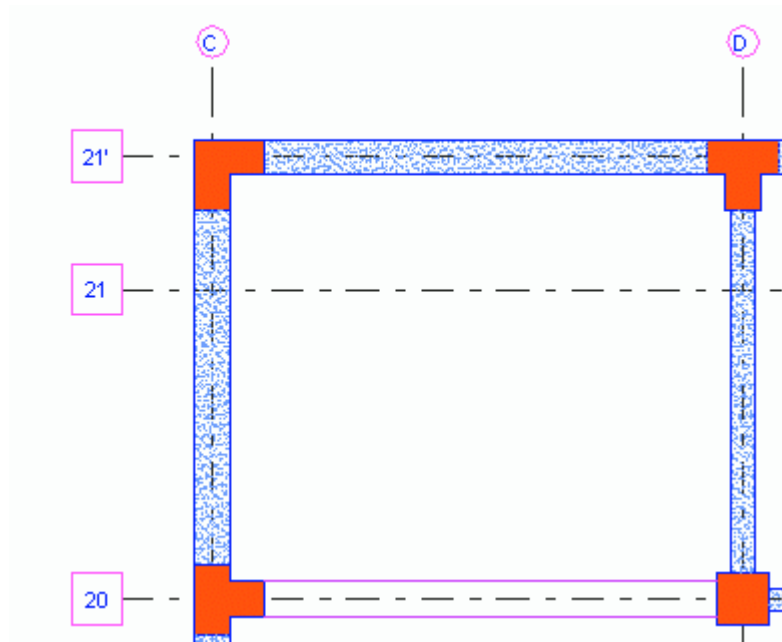


Figure 55: C20 – D20 beam

Using the same process, draw another beam between the E20 and F'20 intersections.



Use the AutoCAD® **Repeat** command, to repeat the last executed command – in this case, *grtcbearm*.

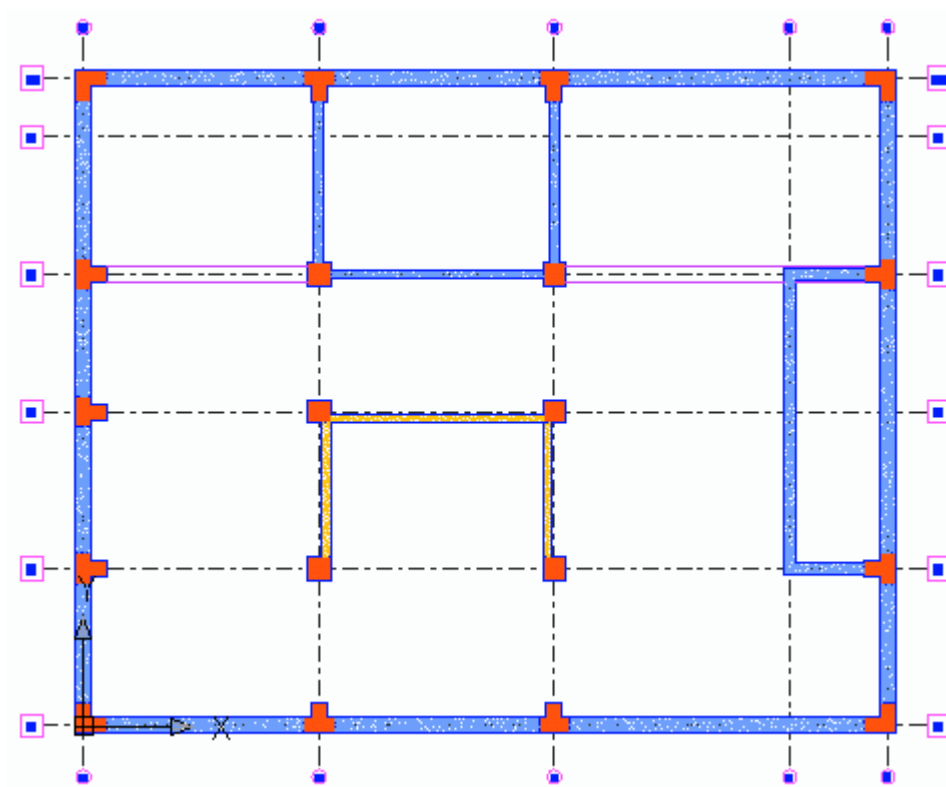


Figure 56: Beams along the 20 axis

Step 2: Create continuous beams with rectangular sections

In this example, you will create beams made up of three segments along the **18** and **19** grid axes.

1. On the **Model** toolbar, **Structural** flyout, click .
2. Click the **C18** axis intersection to define the beam start point.
3. Click the **D18** intersection to define the end point.
4. Draw the beams by clicking the **E18** and **F'18** intersections.

Using the same process, draw the beams along the **19** axis by clicking the **C19**, **D19**, **E19**, and **F19** (the middle point of the **F20 – F18** wall) axis intersections.

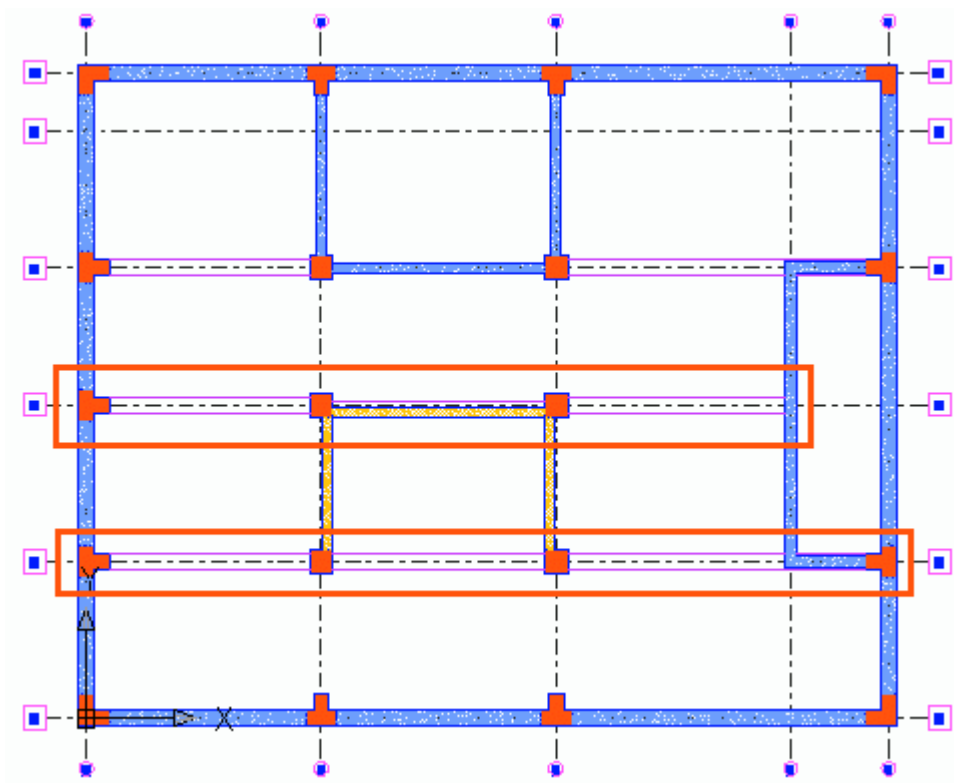


Figure 57: Created beams along the **18** and **19** axes

Create beams along the **D** axis, between the **D17-D18**, **D18-D19** and **D19-D20** columns.

1. On the **Model** toolbar, **Structural** flyout, click .
2. Place the start point at the **D17** axis intersection and create beams at the **D18**, **D19** and **D20** intersections.
3. Press **Esc**.

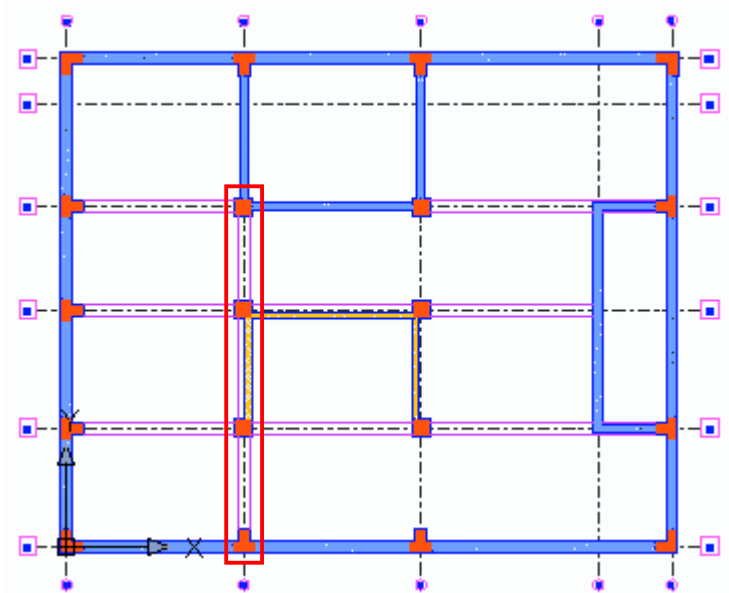


Figure 58: Creation of beams on the **D** axis

Creating beams on the **E** axis, between columns at **E17-E18**, **E18-E19** and **E19-E20**.

1. On the **Model** toolbar, **Structural** flyout, click .
2. Select the start point at the **E17** axis intersection and continue creating beams at the **E18**, **E19** and **E20** intersections.
3. Press **Esc** to finish.

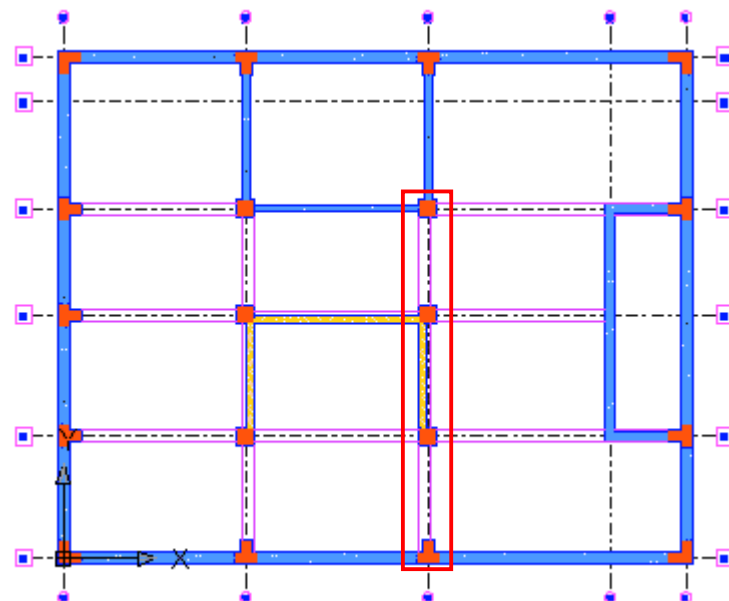


Figure 59: Creating beams on the **E** axis

Step 3: Modify the beam properties

This step describes how to adjust the beam position and change the representation style.

Modifying the position of the beams

1. Select the **D18-D19** and **D19-E19** beams.

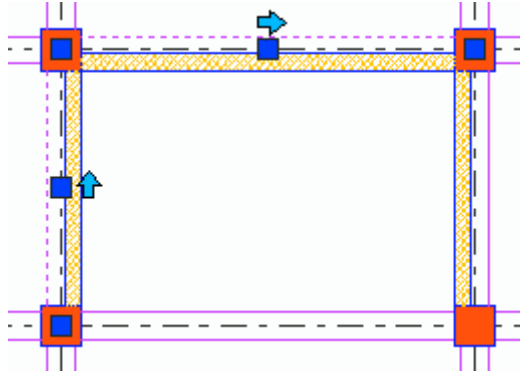
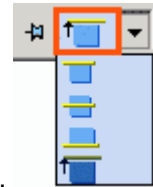
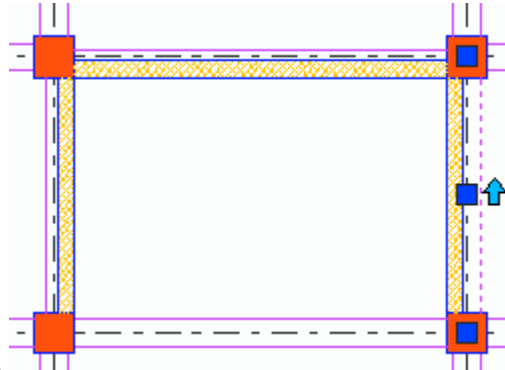


Figure 60: Selection of the **D18-D19** and **D19-E19** beams

2. On the Smartbar, from the **Justification** drop-down list select custom justification.



3. Enter **-0.10 m** for the justification value  and press **Enter**.



4. Select the beam located on the **E18-E19** columns.



5. On the Smartbar, from the **Justification** drop-down list, select custom justification.

- Enter **0.10 m** for the justification value  and press **Enter**.

Using AutoCAD® tools to view the building

To change the view angle, use the buttons on the AutoCAD® **View** toolbar.



Figure 61: AutoCAD® **View** toolbar

For a more realistic presentation of the model, use the AutoCAD® **Visual styles** toolbar.



Figure 62: AutoCAD® **Visual Styles** toolbar

Lesson 6: Creating slabs

In this lesson, you will learn how to create slabs by automatic detection.

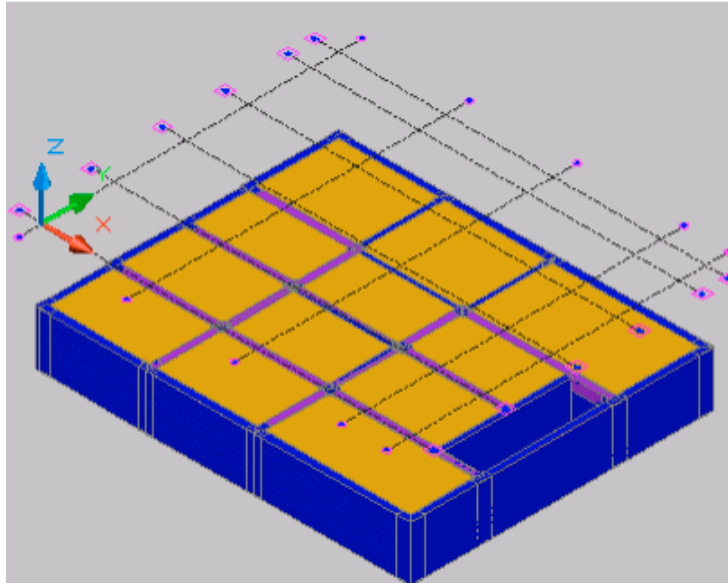


Figure 63: Final model

Step 1: Create slabs using automatic detection

1. On the **Model** toolbar, **Structural** flyout, click .
2. On the command line, type **D** (Detection) and press **Enter**.
3. On the command line, type **a** (to input an interior point between axes) and press **Enter**.
4. On the command line, type **1.00** to set an extension value for the slab and press **Enter**.
5. Click the areas (as shown in the Figure 64) to create the slabs. The slabs are automatically created using boundary detection.

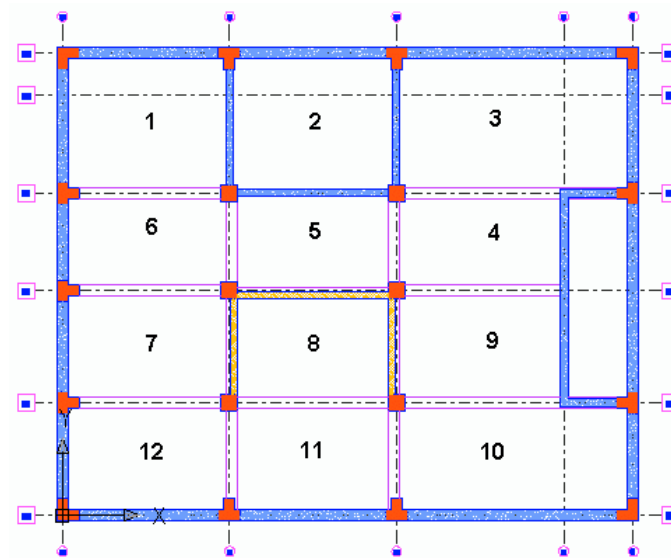


Figure 64: Steps for creating slabs

6. Press **Esc** to finish.

The slabs are numbered from **S1** to **S12**.

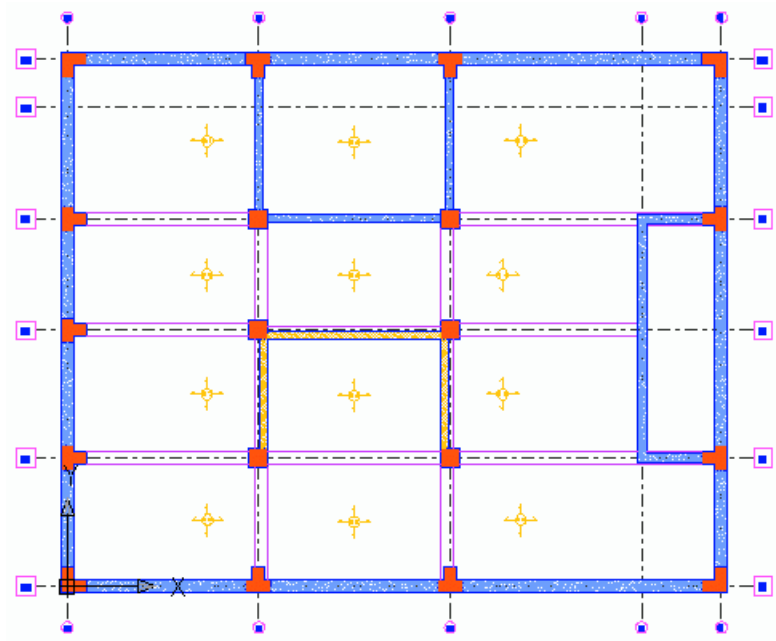


Figure 65: Slabs

Lesson 7: Creating ramps

In this lesson, you will create two straight ramps and an additional element.

You will learn how to:

- Define the ramp slope.
- Create the "surface" using the Advance roof tool.
- Adjust the ramp position.
- Bind the surface to the structure elements to create the ramp.

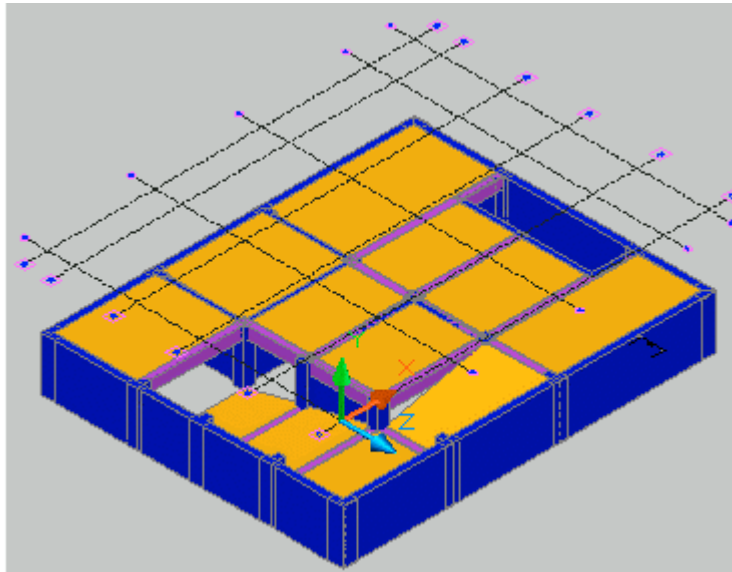


Figure 66 Ramps of the model

Concepts

Advance roof elements and sloped shapes are created from AutoCAD® lines or surfaces.

The roof elements are used to define the lower and upper levels for structural elements (e.g., slabs, beams, columns and walls) to which they are linked. The surface surrounded by the roof element or the sloped shape replaces the upper or lower Advance reference plane.

In our example, the first ramp is made up of **S6** and **S7** slabs. Figure 67 illustrates the ramp profile.

Step 1: Define the ramp slope

This step describes how to define the ramp slope using an AutoCAD® polyline. Define the slope profile as in a front view of the ramp.

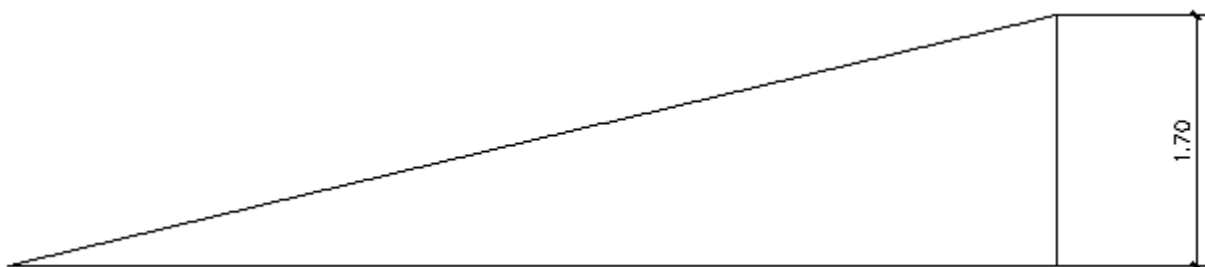


Figure 67 Ramp profile

Creating a polyline defining the ramp slope

To define the ramp profile, draw two auxiliary lines defining the ramp projections (i.e., length and height). As the ramp is made up of **S6** and **S7** slabs, snap to slab corners to define the ramp length.

 **Note:** Make sure that the AutoCAD® OTRACK and OSNAP modes (Intersection) are active.

1. On the AutoCAD® **Draw** toolbar, click .

2. Specify the start point at the bottom right corner of the **C20** column.

3. Move the cursor in the **-Y** direction and define the second point.
4. Move the cursor in the **-X** direction, enter **1.7** on the command line and press **Enter**.

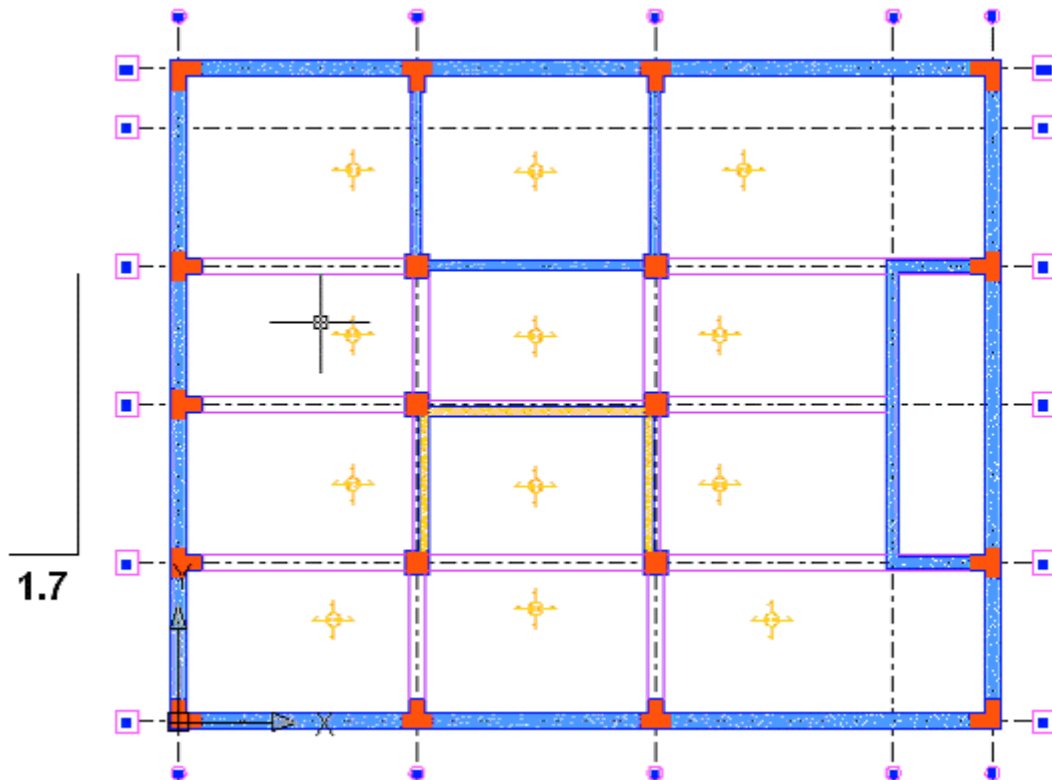


Figure 68: Creation of the auxiliary lines

5. Press **Esc** to finish.

Draw a segment to close the polyline using the **Polyline** tool .

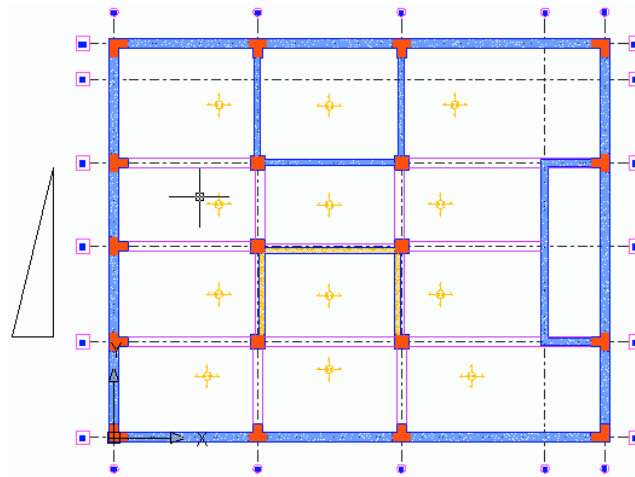


Figure 69: Created polyline

Next, rotate the last segment to get the correct profile (see Figure 71).

1. Select the segment.

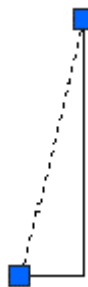


Figure 70: Selection of the polyline

2. Right click the polyline and select **Rotate** from the context menu.
3. Select the base point (anywhere in the drawing).
4. On the command line, set the rotation angle to **90°**.

The segment defining the ramp slope has the correct position.

5. Delete the two auxiliary segments.

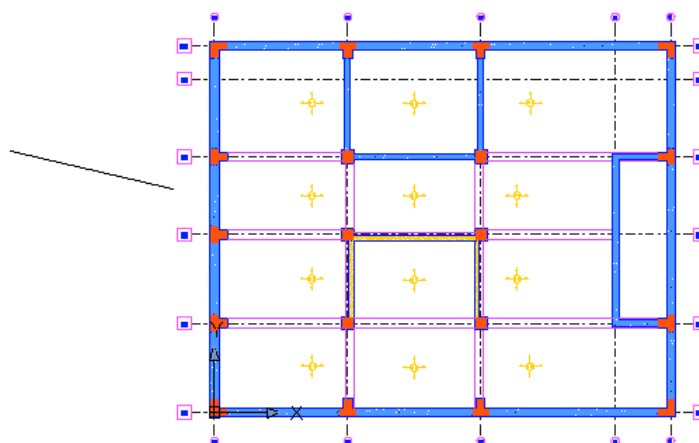


Figure 71: Ramp profile

Step 2: Create a ramp

1. On the **Model** toolbar, **Roof and Ramp** flyout, click .
2. Select the polyline from the previous step (Step 1) to define the ramp profile.
3. Define the first point of the roof at the **D20** intersection as in Figure 72.

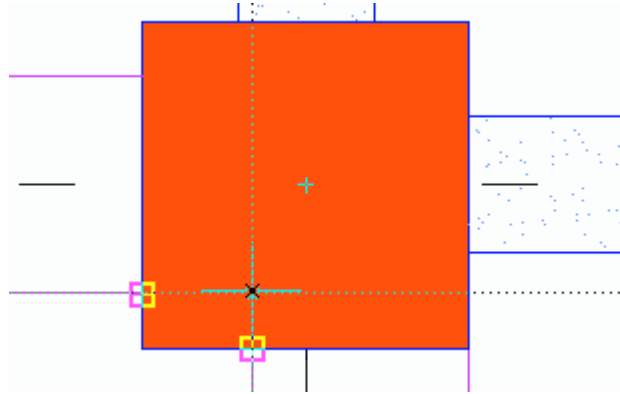


Figure 72: Selecting the first point

 **Note:** Make sure that the AutoCAD® OTRACK and OSNAP modes (Intersection) are active.

4. Define the second point of the roof at the corner of the **C20** column section.

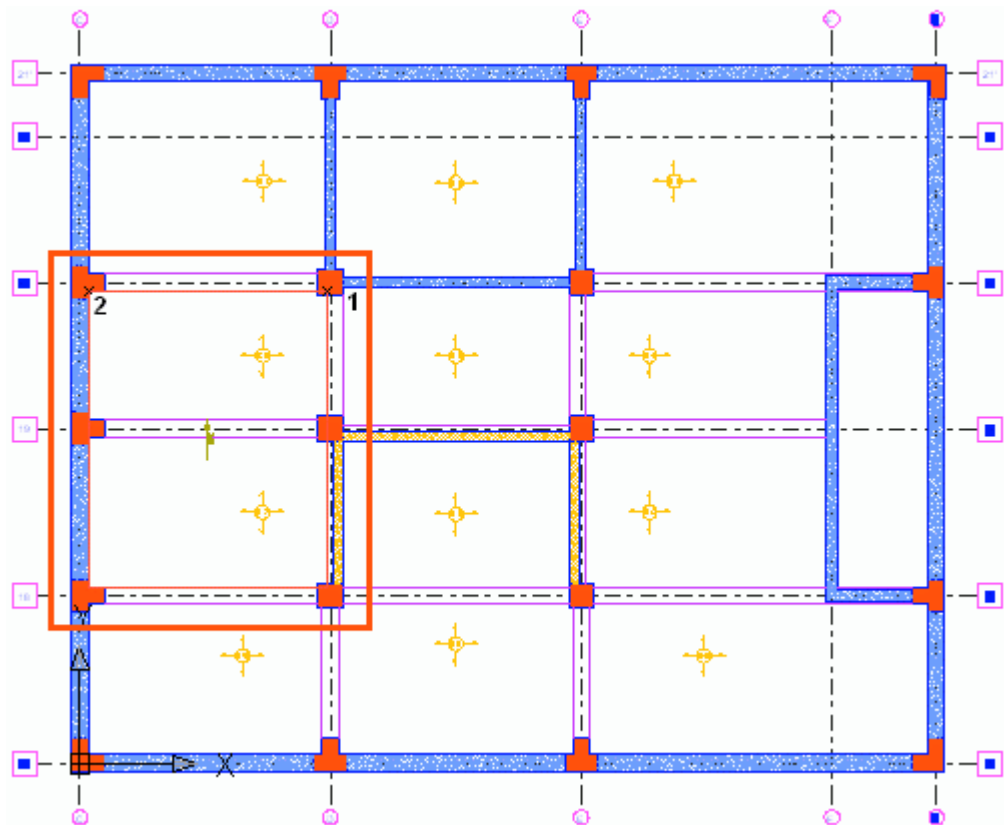


Figure 73: First ramp

5. Press **Esc** to finish.
6. Delete the polyline.

Using AutoCAD® tools to view the ramp

Use the AutoCAD® **View** toolbar to view the ramp in 3D.

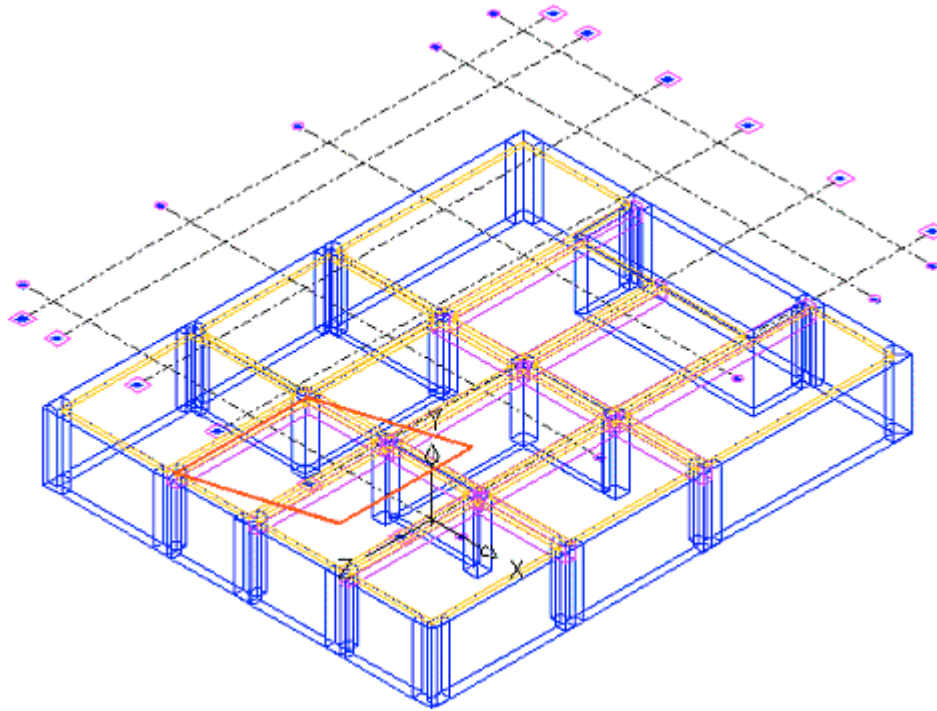


Figure 74: Ramp surface in 3D

Step 3: Adjust the slope position relative to the lower level

By default, the ramp surface position is defined relative to the upper level. To place the ramp correctly, define its position relative to the lower level.

 On the AutoCAD® **View** toolbar, click  to view the ramp from the side.

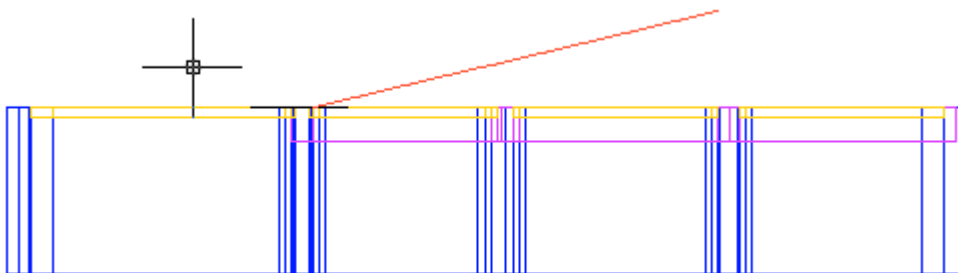


Figure 75: Ramp in side view

1. Select the ramp surface.

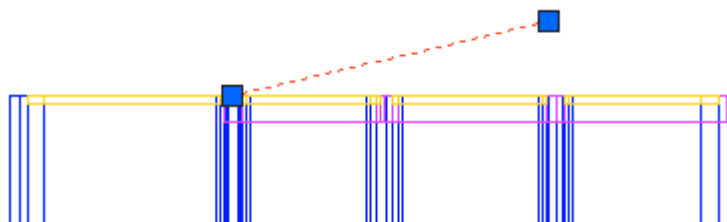
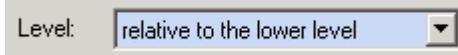


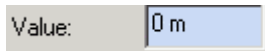
Figure 76: Selection of the first ramp

2. On the Smartbar:

- From the **Level** drop-down list, select **Relative to the lower level**.



- In **Value** field, enter **0 m**.



3. Press **Enter**.

The ramp surface is correctly placed.

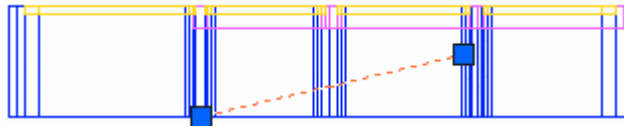


Figure 77: Ramp placed relative to the lower level

Step 4: Bind the ramp to the structural elements

View the model in 3D to see the elements.

- On the **Model** toolbar, click .
- On the command line, type **B** (Bind) and press **Enter**.
- Select the ramp surface and the elements to bind: the **S6** and **S7** slabs and the **C19 – D19** beam.
- Press **Enter** to finish.

The selected elements adjust to the roof surface.



Using AutoCAD® tools to view the ramp

For a realistic view of the model, click  on the AutoCAD® **Visual styles** toolbar.

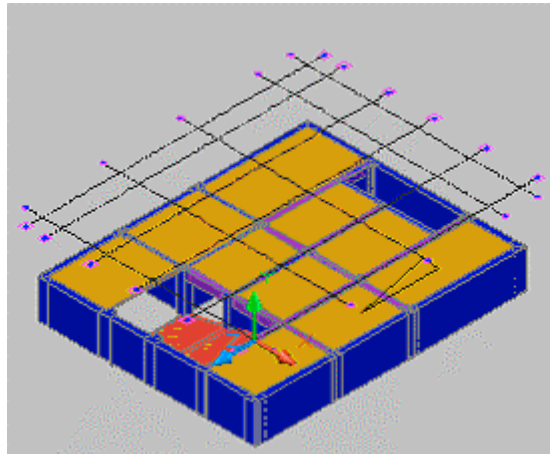


Figure 78: 3D view of the model

Step 5: Adjust the ramp's auxiliary element position

Next, modify the **S12** slab and the **C18 – D18** beam upper level to fit the ramp. As the ramp height is **1.7m**, the auxiliary element must be placed at **1.7m** relative to the lower level.

1. Select the **S12** slab.

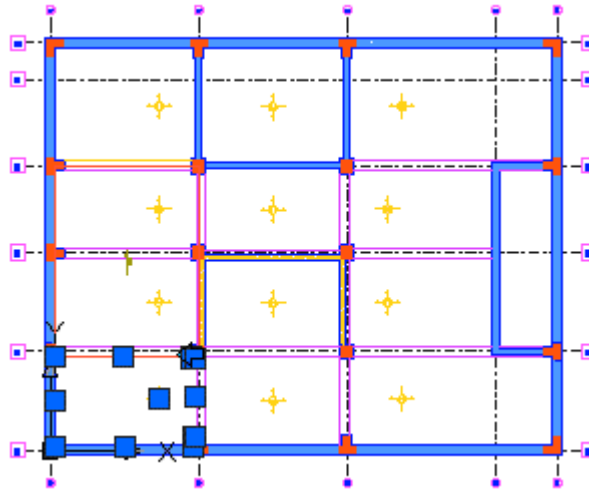


Figure 79: Selection of the **S12** slab.

2. On the Smartbar, click  to access the slab properties dialog box.
 - On the **Definition** tab, from the **Upper level** drop-down list, select **relative to the lower level**.
 - In the **Value** field, enter **1.7 m** (the ramp height).

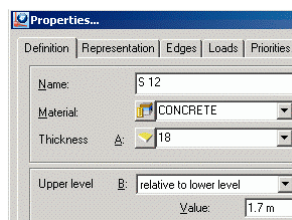


Figure 80: Setting the slab upper level

3. Click **OK**.

Using the same process, modify the height of the **C18 – D18** and **D17- D18** beams relative to the lower level.

1. Select the **C19 – D19** and **D17 – D18** beams.

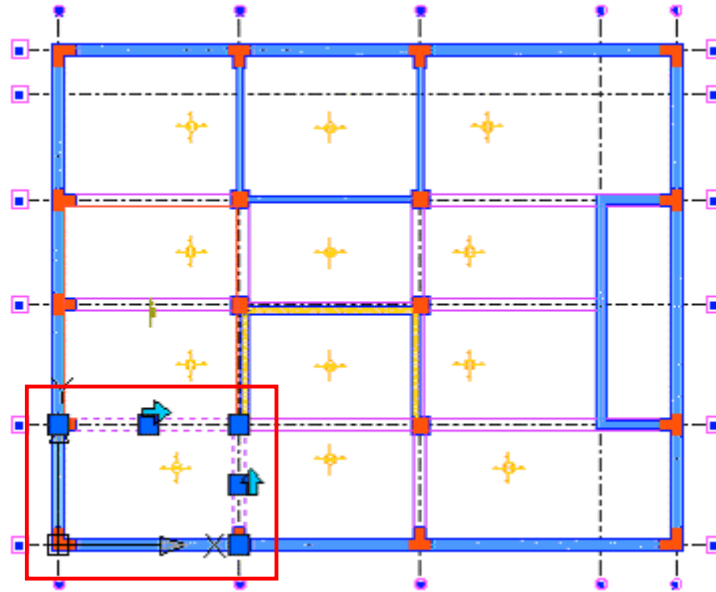


Figure 81: Beams for the ramp auxiliary element

2. On the Smartbar, click  to access the beam properties dialog box.
 - On the **Definition** tab, from the **Upper level** drop-down list, select **relative to the lower level**.
 - In the **Value** field, enter **1.7 m** (the ramp height).

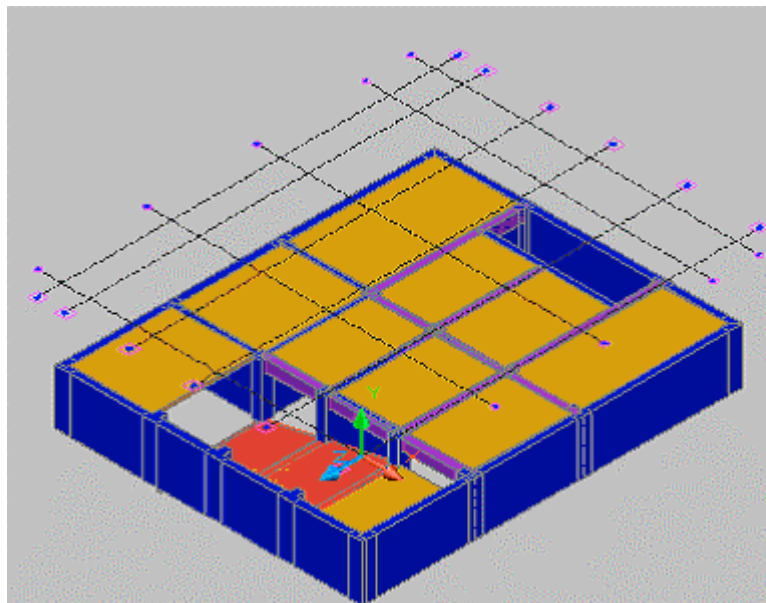


Figure 82: Ramp

Using the same process, create the second flight of the ramp. The slope profile must be defined as in a front view of the ramp.

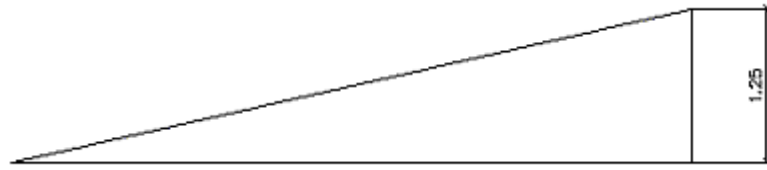


Figure 83: Ramp profile

Creating a polyline defining the ramp slope

To define the ramp profile, draw two auxiliary lines defining the ramp projections (i.e., length and height).

1. On the AutoCAD® **Draw** toolbar, click .
2. As the ramp is made up of the **S11** slab, snap to the slab corners to define the ramp length.

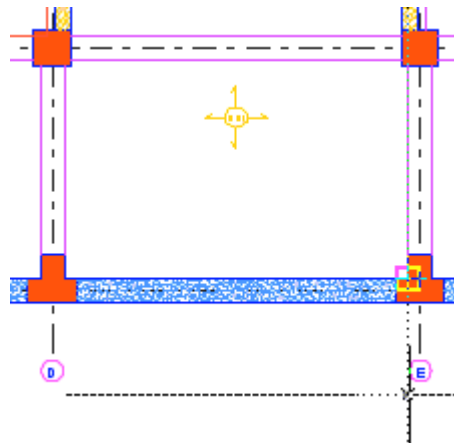


Figure 84: First segment of the polyline

3. Move the cursor in the Y direction, enter **1.25** on the command line and press **Enter**.
4. Press **Esc** to finish.
5. Draw a segment to close the polyline.

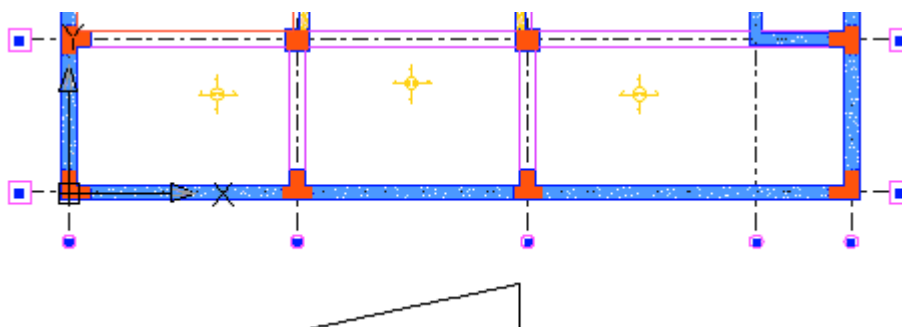


Figure 85: Creation of the second polyline

Step 6: Create the second ramp surface

1. On the **Model** toolbar, **Roof and Ramp** flyout, click .
2. Select the previously drawn polyline.
3. Position the first point of the ramp at the **E17** intersection.
4. Position the second point of the ramp at the **E18** intersection.

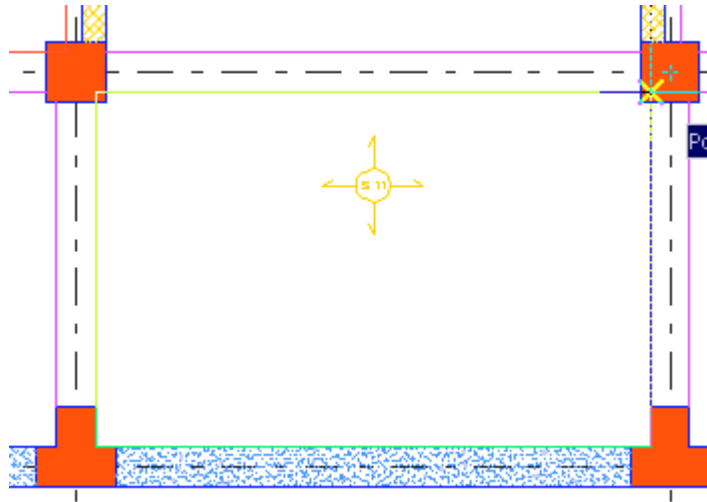


Figure 86: Draw the second ramp

5. Delete the polyline.

Adjusting the position of the second ramp



On the AutoCAD® **View** toolbar, click  to view the ramp from the side.

1. Select the ramp.

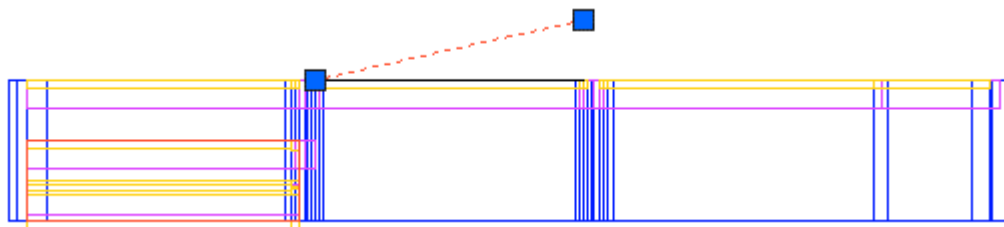


Figure 87: Selection of the second ramp

2. On the Smartbar, in the **Value** field, set the new altitude, relative to the upper level, to **-1.25 m**.



3. Press **Enter** to finish.

The ramp is correctly placed.

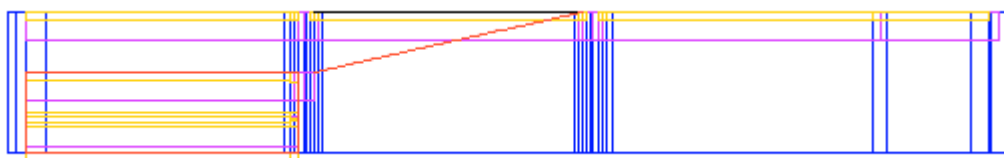


Figure 88: Ramp in side view



Use the AutoCAD® **View** toolbar to view the model in 3D.



Figure 89: AutoCAD® **View** toolbar

Using the previous created ramp surface to create the ramp

1. On the **Model** toolbar, click .
2. On the command line, type **B** (Bind) and press **Enter**.
3. Select the elements to bind: the ramp surface and the **S11** slab.
4. Press **Enter**.



Use the AutoCAD® **Visual Styles** toolbar to view the ramp surface.

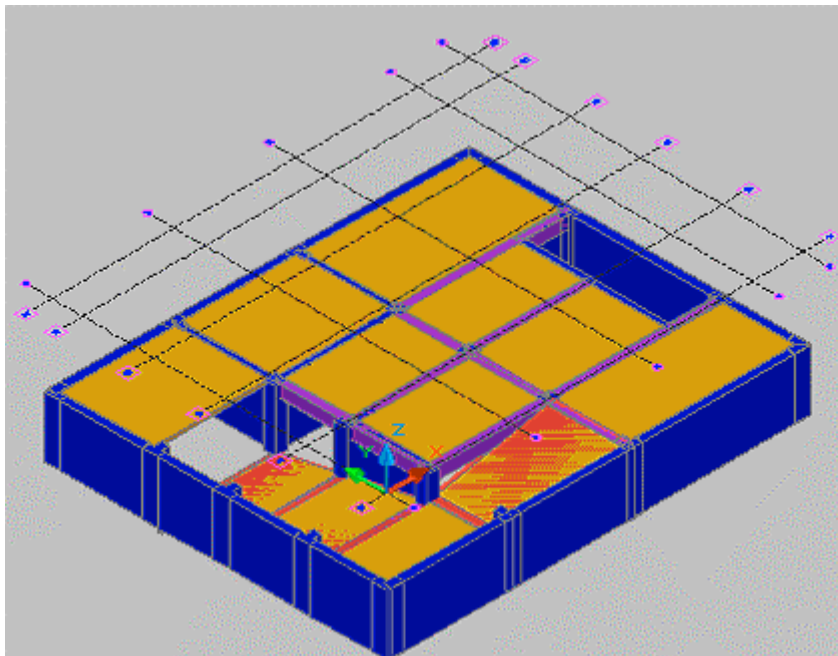


Figure 90: Ramp surface

Step 7: Hide the control surface

1. In the Pilot, click  to display the quick menu.
2. Select **Display by type**.

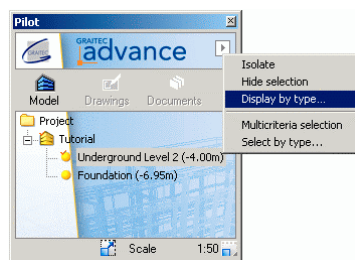


Figure 91: Pilot – Quick menu

3. In the "Element visibility" dialog box, select the elements to hide.

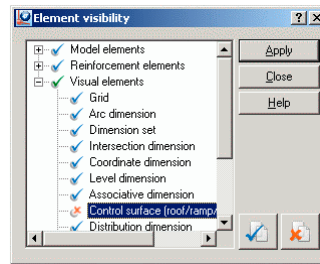


Figure 92: "Element visibility" dialog box

4. Click **Apply**.
5. Click **Close** to close the dialog box.



Use the AutoCAD® **Visual Styles** toolbar for a better visualization of the drawing.

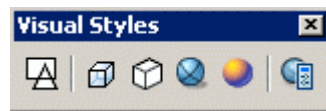


Figure 93: AutoCAD® **Visual Styles** toolbar

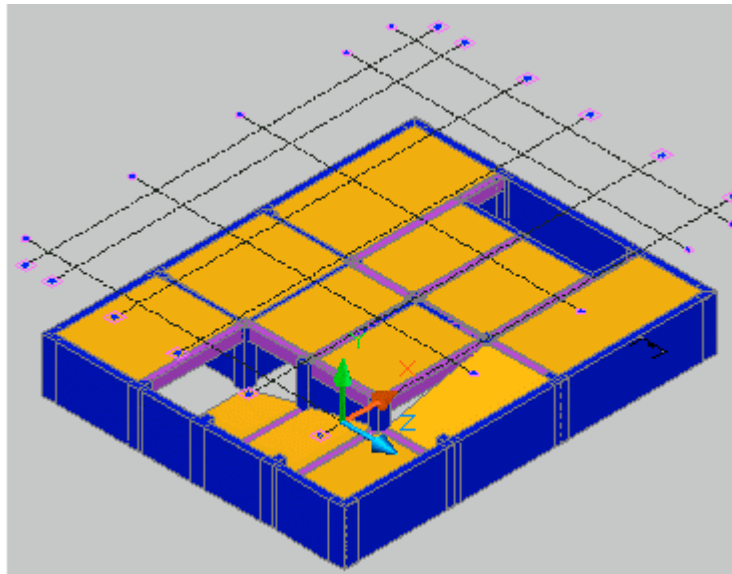


Figure 94: 3D view of the model

Lesson 8: Creating openings

In this lesson you will create elevators and all required openings for wiring, pipes, etc.

You will learn how to create:

- Slab and wall openings.
- Door and window openings.
- Wall depressions.
- Beam opening lines.

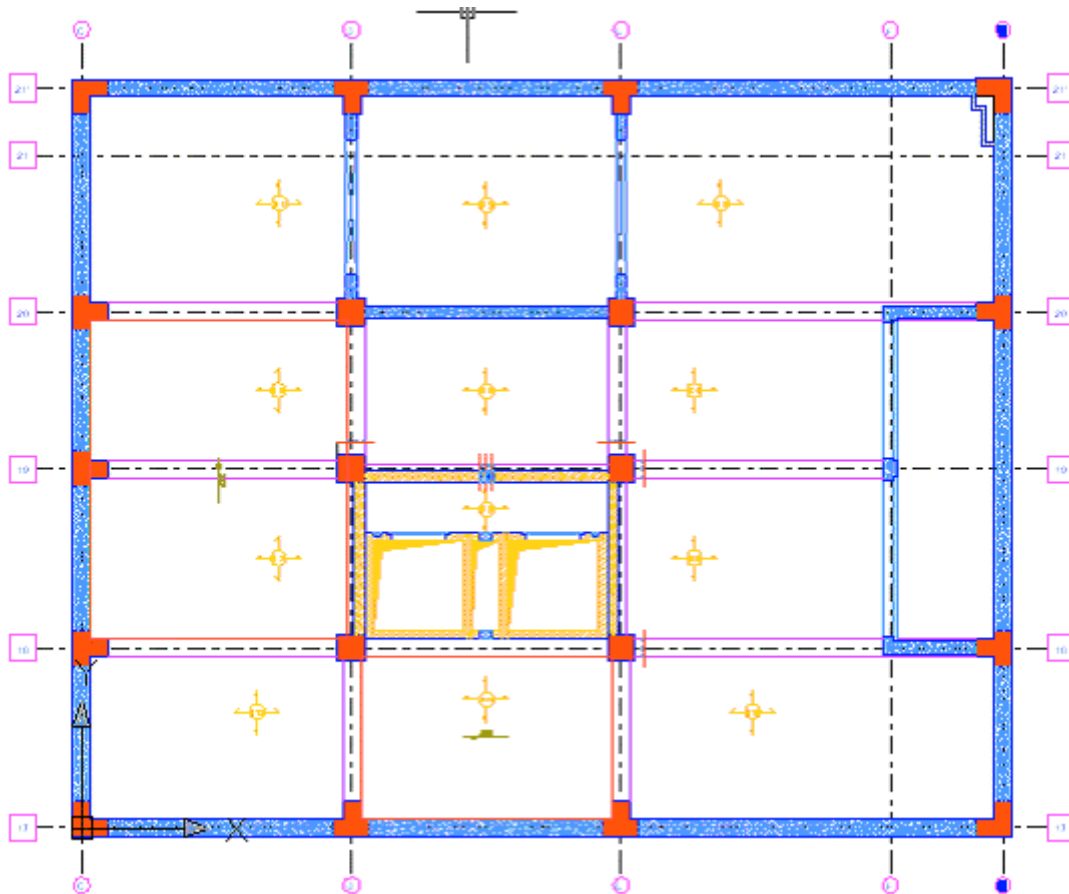


Figure 95: Openings in the model

Step 1: Complete the model

Before creating the openings, create the walls for the two elevators.

Zoom to the model area as in Figure 96.

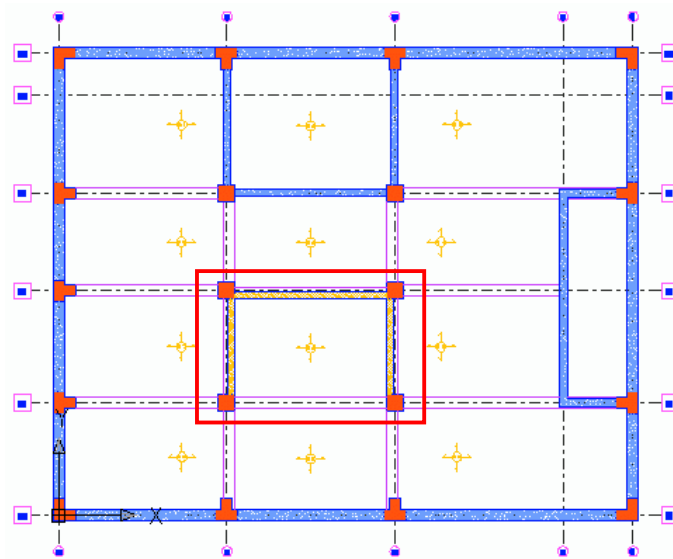
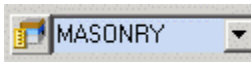
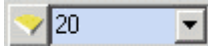


Figure 96: Elevator area

1. On the **Model** toolbar, **Structural** flyout, click .

 **Note:** During creation, you can modify the wall properties using the Smartbar or the properties dialog box.

2. On the Smartbar, set the elevator wall properties:

- From the **Material** drop-down list, select the **Masonry** material. 
- From the **Thickness** drop-down list, select **20 cm** for the wall thickness. 
- From the **Style** drop-down list, select the **Masonry** representation style. 

3. On the Smartbar, click  to access the properties dialog box.

- Select the **Junctions** tab.
- Select the **Extremity 1** and **Extremity 2** options and enter **0 m**.

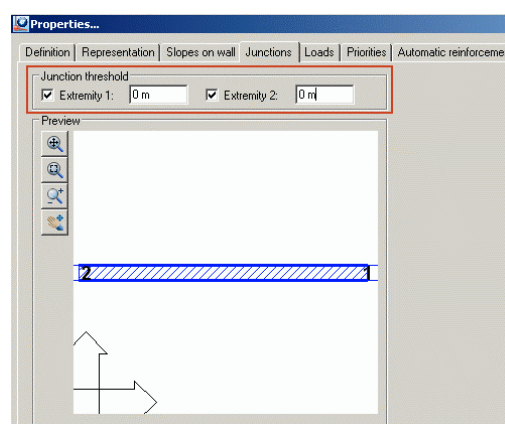


Figure 97: "Properties" dialog box – Modifying the wall junction

 **Note:** Using the **Junctions** tab, a radius value can be set for each extremity of the selected element. If the two circles in the extremities of the selected element cross each other, then the junction works.

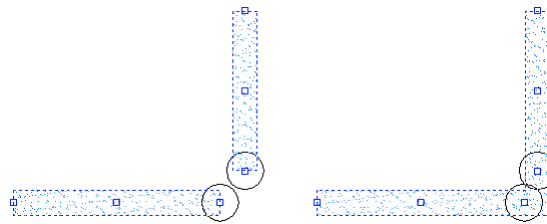
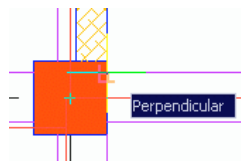


Figure 98: Example of Junction effects

4. Click **OK**.

 **Note:** Activate the **ORTHO** mode to draw the walls.



1. Define the start point at the **D18** axis intersection.
2. Move the cursor in the **X** direction, enter **2.4** on the command line and press **Enter**.
3. Move the cursor in the **Y** direction, enter **2.4** and press **Enter**.
4. Move the cursor in the **-X** direction, enter **2.4** and press **Enter**.
5. Move the cursor in the **-Y** direction, enter **2.4** and press **Enter**.
6. Press **Esc** to finish.

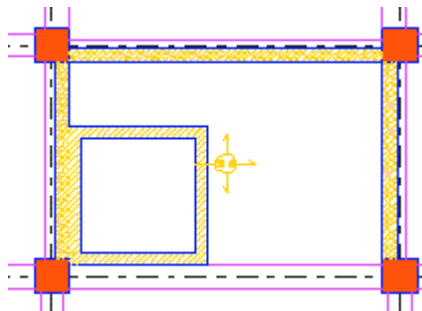


Figure 99: First elevator

 **Note** Notice that an edge is not displayed between the inner wall of the building and the elevator wall. This happens because the two walls have the same priority.

Next, modify the priority of the elevator walls.

1. Select the elevator walls.

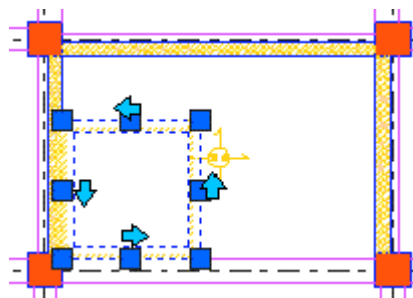


Figure 100: Selecting the elevator walls

2. On the Smartbar, type **41** for the new priority.
3. Press **Enter**.

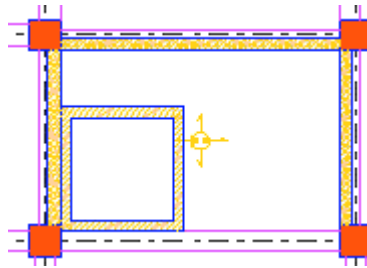
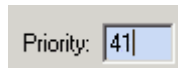
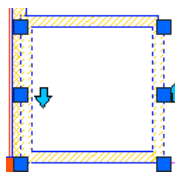


Figure 101: The edge between the elevator walls

Using the AutoCAD Mirror tool

Since the second elevator has the same size as the first one, you can use the AutoCAD® **Mirror** tool to create the second elevator.

1. On the AutoCAD® **Modify** toolbar, click .



2. Select the left and right walls.
3. Press **Enter** to validate the selection.
4. Click the middle point of the **D18 – E18** beam to set the first point of the symmetry axis.
5. Activate the **ORTHO** mode and click a point to set the second point of the symmetry axis.

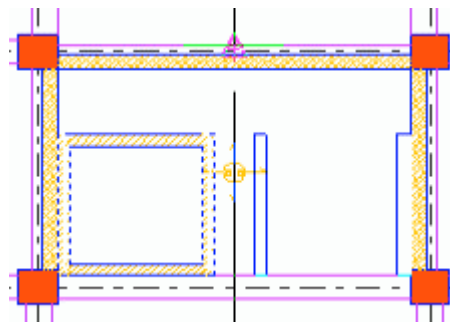


Figure 102: Placing the second point of the axis

6. Next, a question appears on the command line: “Erase source objects? <Y/N>”. Enter **N** (No) to keep the original wall and press **Enter**.

A copy of the selected walls appears.

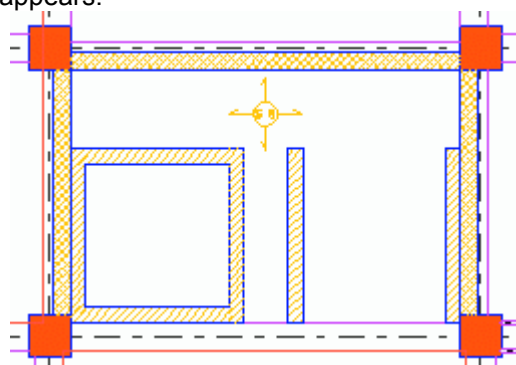


Figure 103: Left and right walls of the second elevator

Extending the walls using the grip points

1. Select the bottom wall of the first elevator.
2. Click the right grip point and extend the wall in the X direction to the right wall of the second elevator.

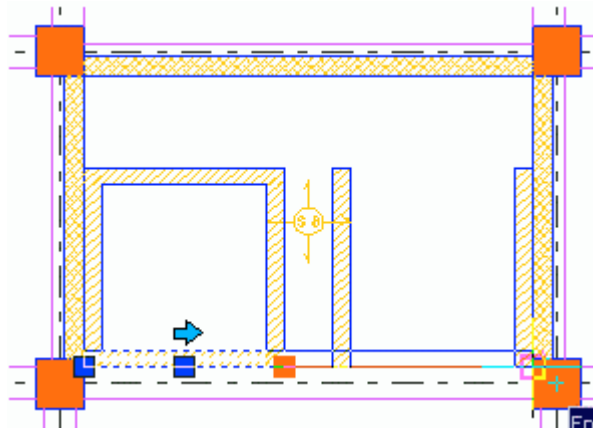


Figure 104: Extending the elevator wall

Use the same process for the upper wall.

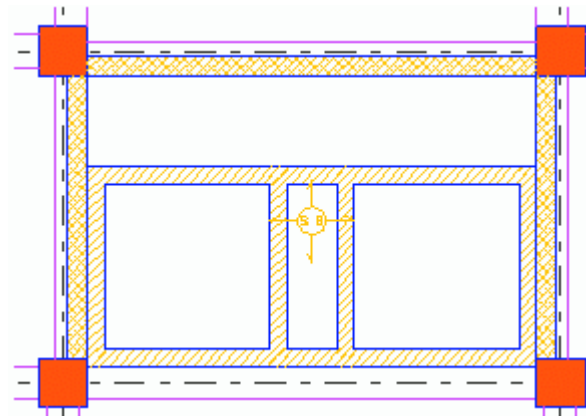


Figure 105: Elevator walls

To get a clear view, move the slab symbol outside the opening.

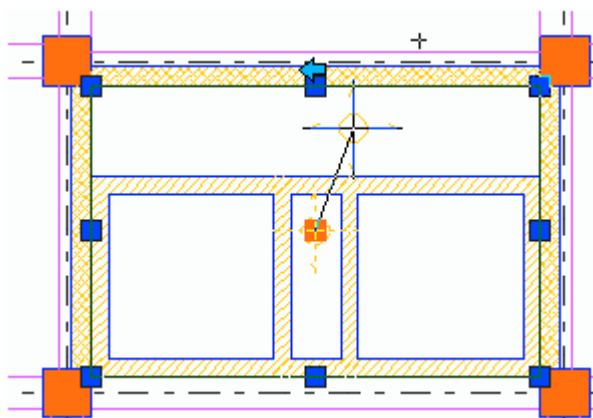
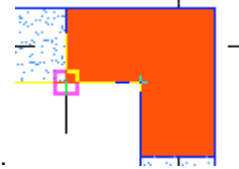


Figure 106: Moving the slab symbol

Drawing the walls

Draw the auxiliary polyline defining the wall contour in the **F'21'** corner.

1. On the AutoCAD® **Draw** toolbar, click .



2. Define the start point at a corner of the **F'21'** column as in the next picture.
3. Move the cursor in the **-Y** direction, enter **.2** on the command line and press **Enter**.
4. Move the cursor in the **X** direction, enter **.2** and press **Enter**.
5. Move the cursor in the **-Y** direction, enter **.6** and press **Enter**.
6. Move the cursor in the **X** direction, enter **.2** and press **Enter**.
7. Press **Esc** to finish.

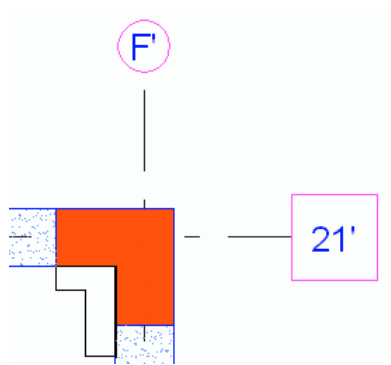
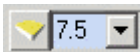


Figure 107: Polyline

Creating walls along the polyline

1. On the **Model** toolbar, **Structural** flyout, click .
2. On the Smartbar:

- Set the wall thickness to **7.5**. 

- Select the top justification. 

3. Define the start point and draw the walls along the previously created polyline.

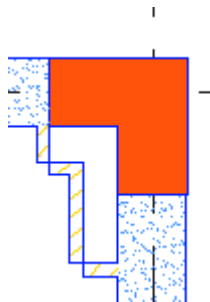
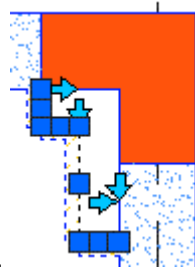


Figure 108: Wall

Modifying the wall junction



1. Select the created walls.
2. On the Smartbar, click  to access the properties dialog box.
 - Select the **Junctions** tab.
 - Select the **Extremity 1** and **Extremity 2** options and enter **0.3 m**.

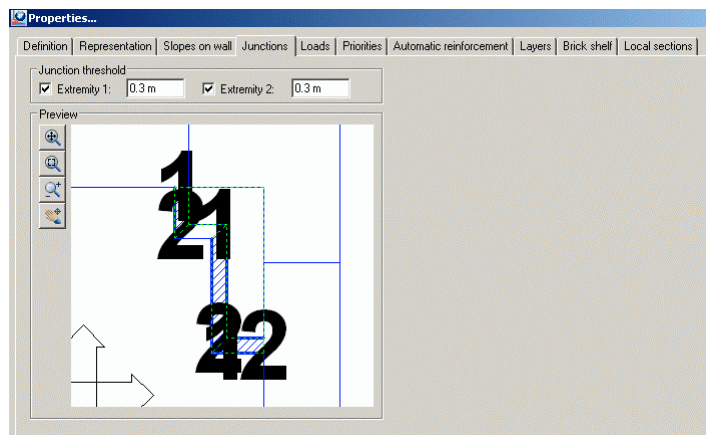


Figure 109: The "Properties" dialog box – **Junctions** tab

3. Click **OK** to validate.

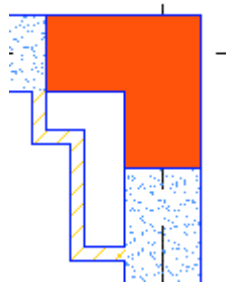


Figure 110: Modified wall

Step 2: Create wall openings

 **Note:** The Opening object depends entirely on the Wall object. Thus, when moving a wall, the opening is moved with the wall. If a wall is deleted, the opening linked to the wall is also deleted.

In this step you will create rectangular openings (**R300x230**) on the **E21' - E20** and **D21' - D20** walls.

1. On the **Model** toolbar, **Openings** flyout, click .
2. Click the **D21'-D20** wall.

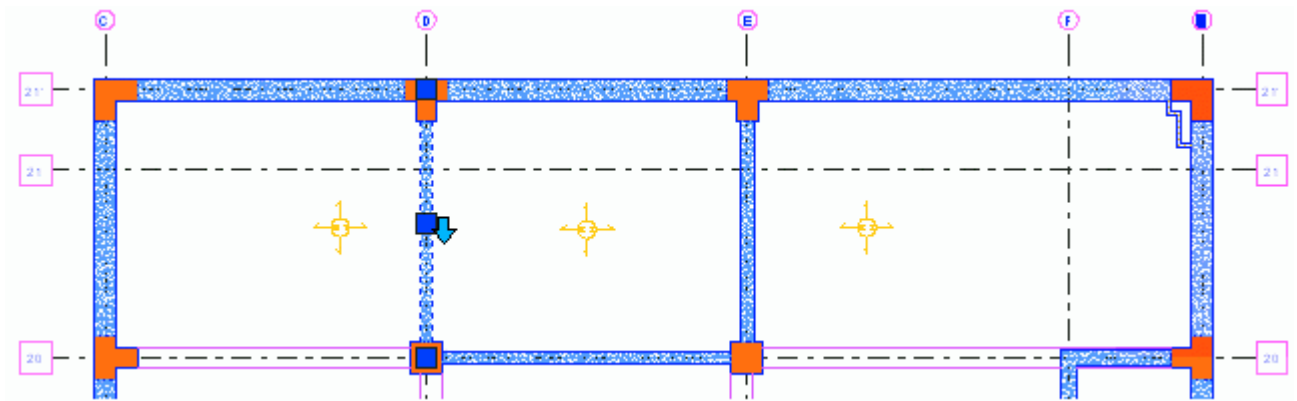
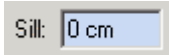


Figure 111: Selection of the **D21'-D20** wall

3. On the Smartbar, modify the opening properties.
 - Set the opening size to **R300x230**. 
 - In the **Sill** field, enter **0 cm**. 
4. Press **Enter**.
5. Click the middle of the **D21' – D20** wall to place the opening.

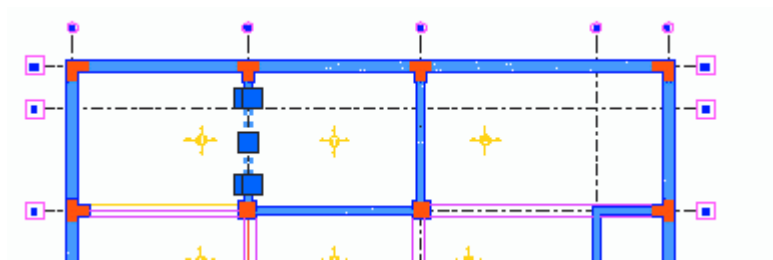


Figure 112: Opening on the **D21'-D20** wall

Create another wall opening (with the same properties) on the **E21' - E20** wall.

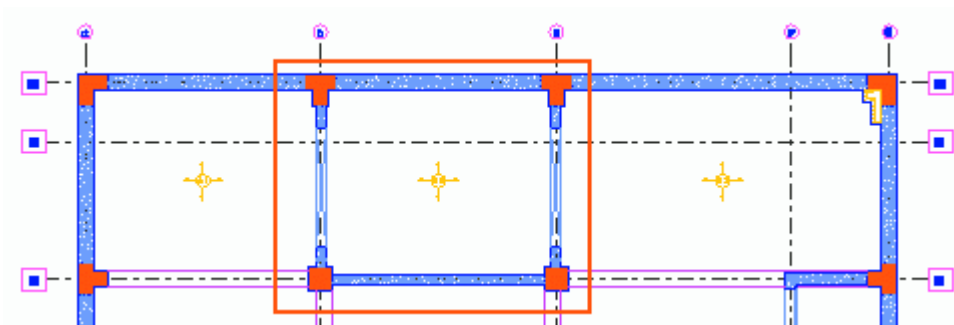


Figure 113: Wall openings

Step 3: Create window openings

1. On the **Model** toolbar, **Openings** flyout, click .
2. Select the wall along the **F** axis.

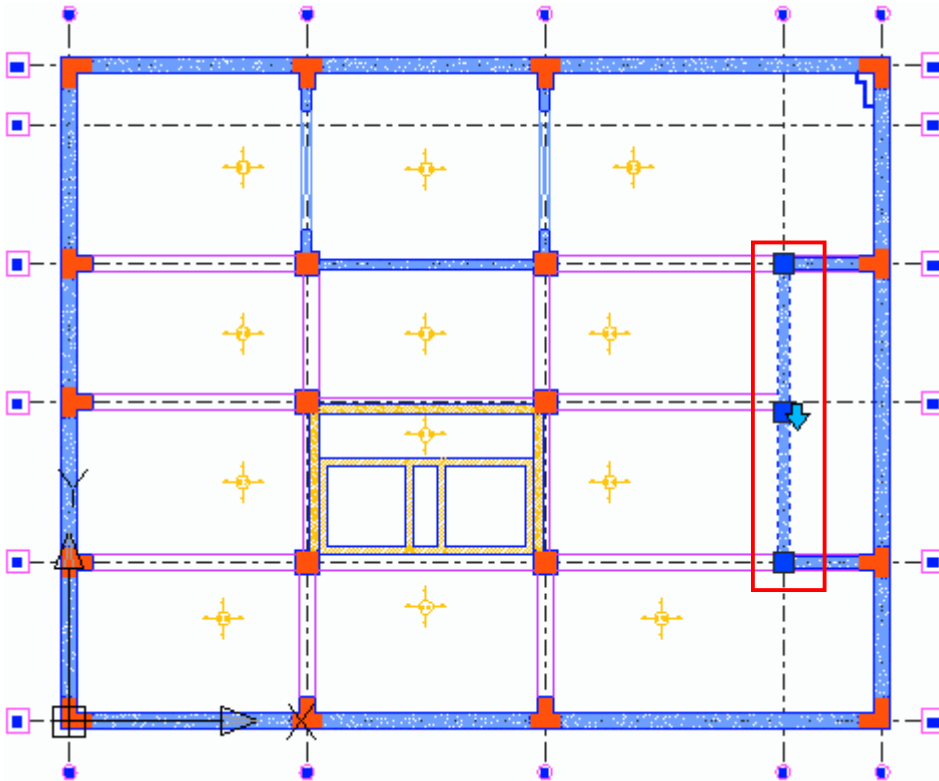


Figure 114: Wall along the **F** axis

3. On the Smartbar, click  to access the window library.
4. Select an opening type from the library.
 - In the dialog box, click **Add**. A second panel appears. The **Libraries** tab lists the openings available in the library.

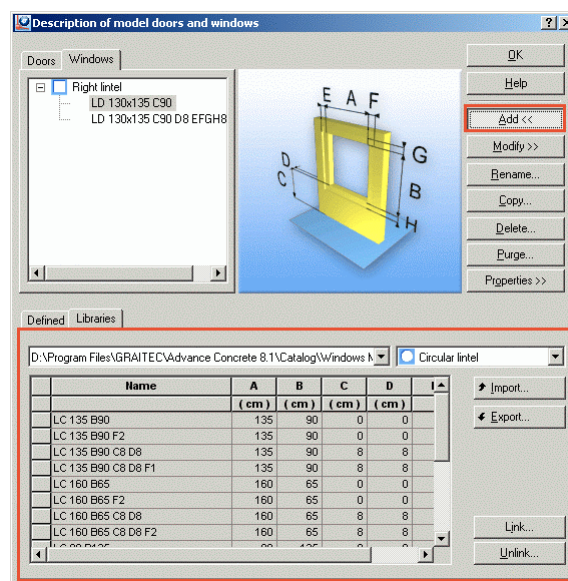


Figure 115: Accessing the **Libraries** tab

- From the drop-down list on the right side of the dialog box, select the **Right lintel** opening shape. The **Libraries** tab lists all windows with a right lintel.
- Select **LD 160X135 C90 D8 EFGH8888**.
- Click **Import**.

The opening is imported into the current project. The openings available in the project are listed in the openings list and on the openings Smartbar.

- Click **OK**.

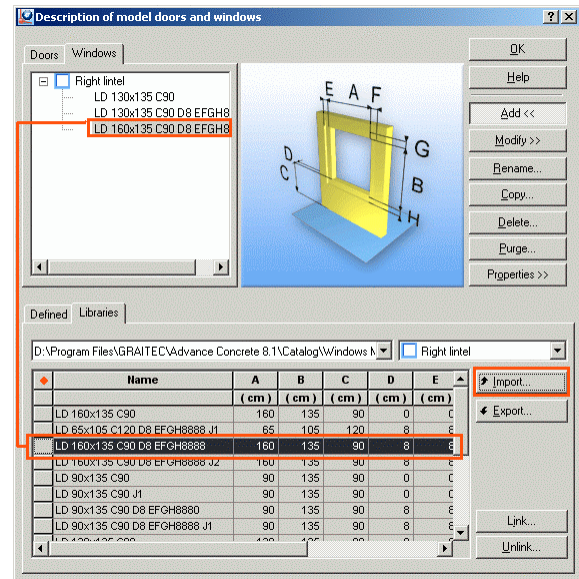
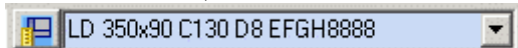


Figure 116: Importing a window opening into the current project

Draw the window:

- On the Smartbar, select **LD 160X135 C90 D8 EFGH8888** and enter the desired size of the window.



- Place the first window on the **F18 - F19** wall.
- Click a point on the right side to define the opening side.

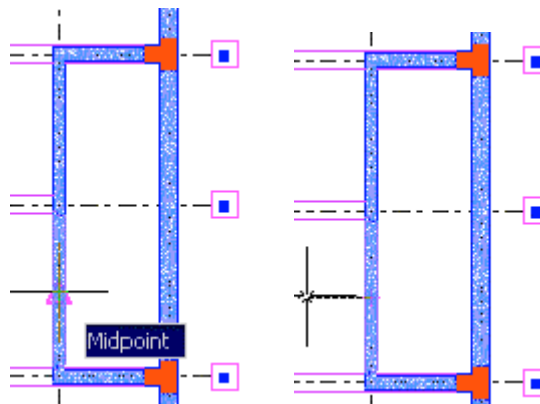


Figure 117: Creating the window opening

Using the same process, create another window opening of the same type on the **F19-F20** wall.

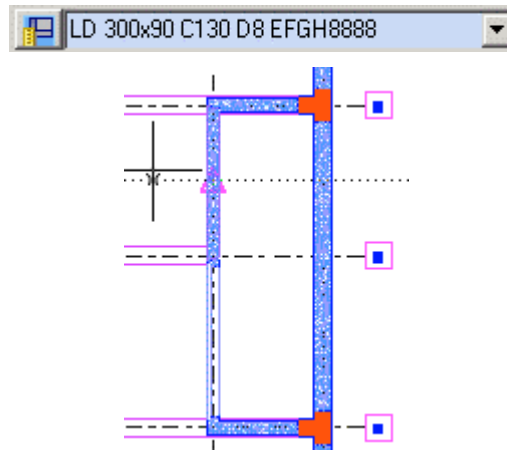


Figure 118: Creation of the second window

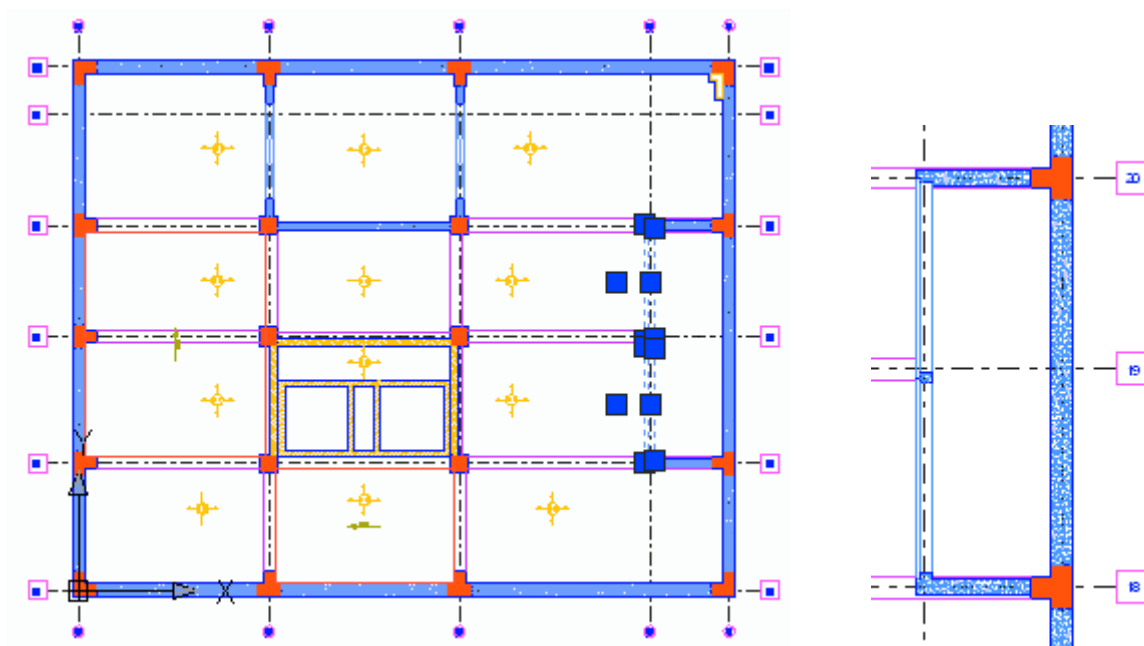


Figure 119: Window openings



Use the AutoCAD® **View** toolbar to view the window openings.



Figure 120: AutoCAD® **View** toolbar

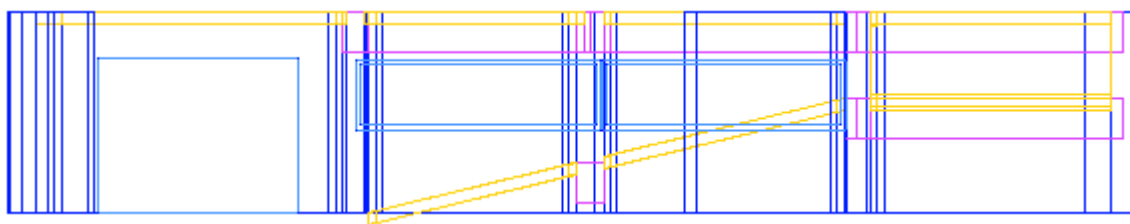


Figure 121: Window openings – side view



On the AutoCAD® **View** toolbar, click  to return to the plan view.

Step 4: Create the door openings

This step describes how to create the elevator door openings.

1. On the **Model** toolbar, **Openings** flyout, click .
2. Select the upper walls of the elevator.

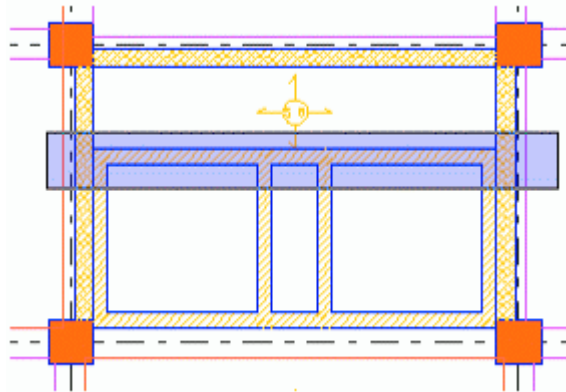


Figure 122: Creating a door – Wall selection

3. On the Smartbar, click  to access the door library.
 - In the dialog box, click **Add**. A second panel appears. The **Libraries** tab lists the door openings available in the library.
 - Select the **Right lintel** opening shape from the drop-down list on the right side of the dialog box. The **Libraries** tab lists all doors with a right lintel.
 - Select **LD 70x215 D8 EFGH8880**.
 - Click **Import**.

The opening is imported into the current project. The openings available in the project are listed in the openings list and on the openings Smartbar.

- Click **OK**.

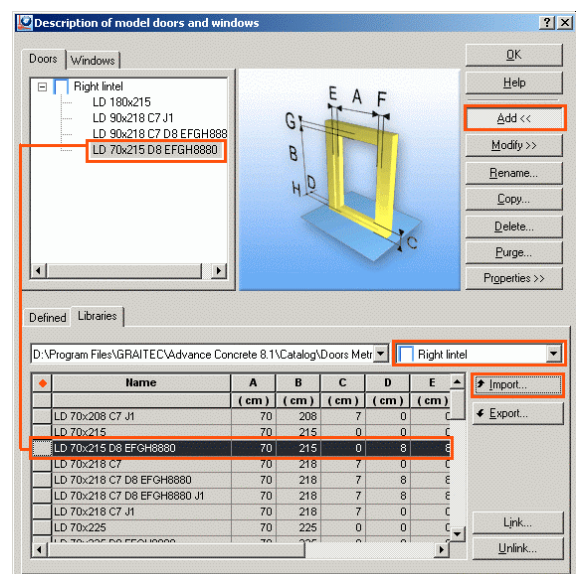


Figure 123: Importing a door type from a library

4. On the Smartbar, select **LD 70x215 D8 EFGH8880** and set the desired size of the window.



5. Place the first door in the middle of the top wall of the second elevator.

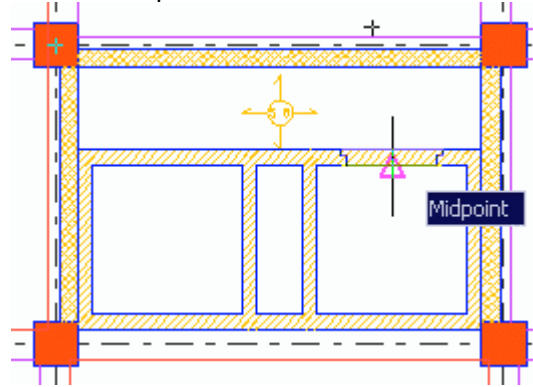


Figure 124: Placing the first door opening

6. With the cursor, select the opening side of the door.

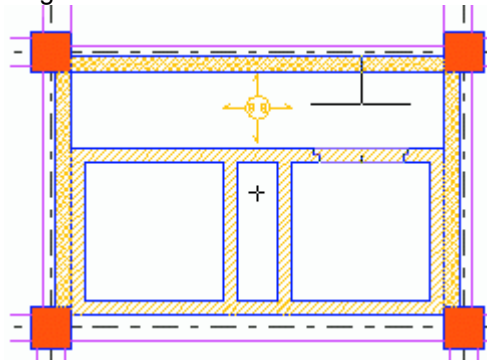


Figure 125: Placing the door on the elevator walls

7. Press **Enter**.

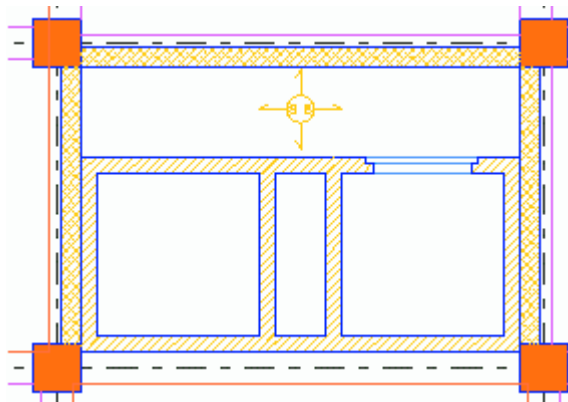


Figure 126: Creation of the first door

Using the same process, draw a door of the same type and size for the other elevator.

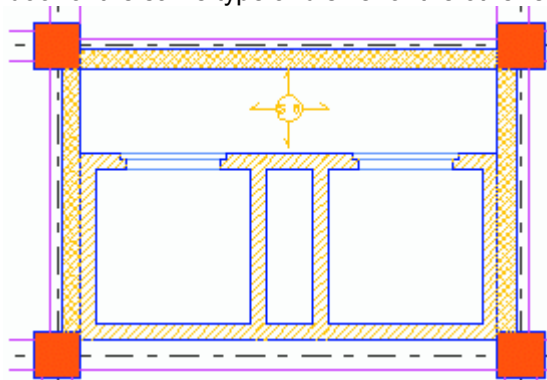


Figure 127: Elevator doors

Step 5: Create the slab openings

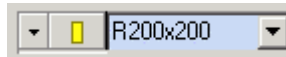
Next, you will create **R200x200** slab openings for the two elevators.

1. On the **Model** toolbar, **Openings** flyout, click .



2. Click the **S8** slab symbol to select the slab.
3. On the Smartbar, set the opening properties.

- Set the opening size to **R200x200**.



- Select the top-right attachment point.



4. Press **Enter**.
5. Click the top-right corner of the first elevator to place the opening.

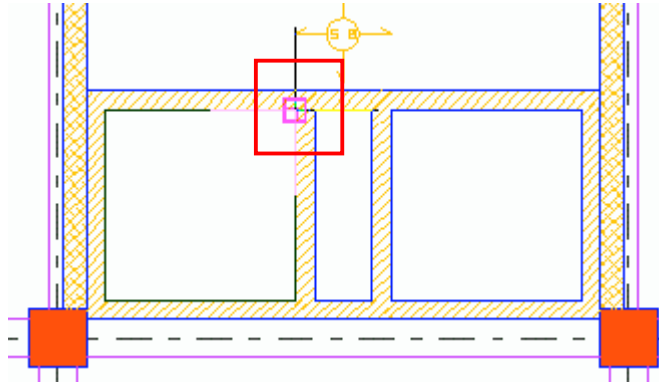


Figure 128: Creation of the first opening

6. Click a point to define the opening position.

Repeat the command to create an opening for the second elevator.

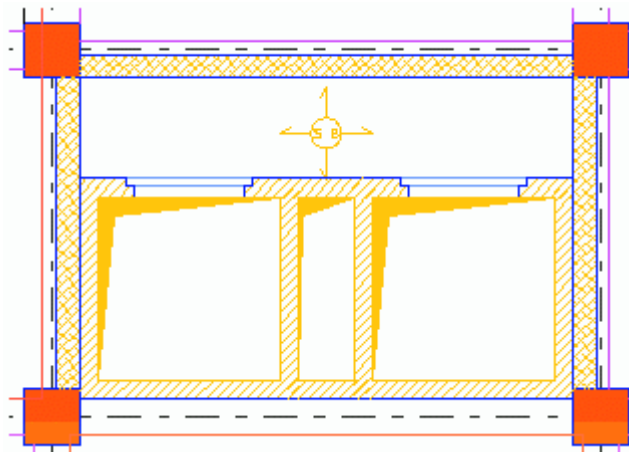
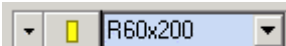


Figure 129: Slab openings

Creating a slab opening for the space between elevators

1. On the **Model** toolbar, **Openings** flyout, click .
2. On the Smartbar, set the opening size to **R60x200**. 
3. Click the top right corner to place the opening.

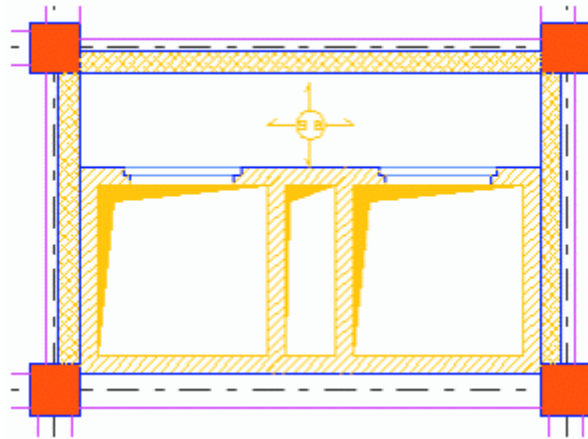


Figure 130: Elevator openings

Step 6: Add the openings required for the wiring

Create a depression near the elevator door for the elevator control panel.

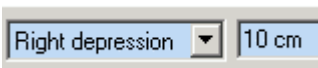
1. On the **Model** toolbar, **Openings** flyout, click .
2. Select the upper wall of the elevator.
3. On the Smartbar set the opening properties.
 - Set the opening size to **R15x30**.
 - Select **Right depression** and set the depth. 
 - Select the justification. 
 - Set the sill to **130 cm**.
4. Place the depression.



Figure 131: Placing the depression

Using the same process, create a depression for the control panel of the other elevator.
Create an opening in the middle of the top wall of the elevator.

1. On the **Model** toolbar, **Openings** flyout, click .
2. Select the upper wall of the elevator wall.

3. On the Smartbar set the opening properties:

- Set the opening size to **R25x25**.
- Select **Crossing**.



- Select the center justification.

4. Click in the middle of the wall to place the opening.

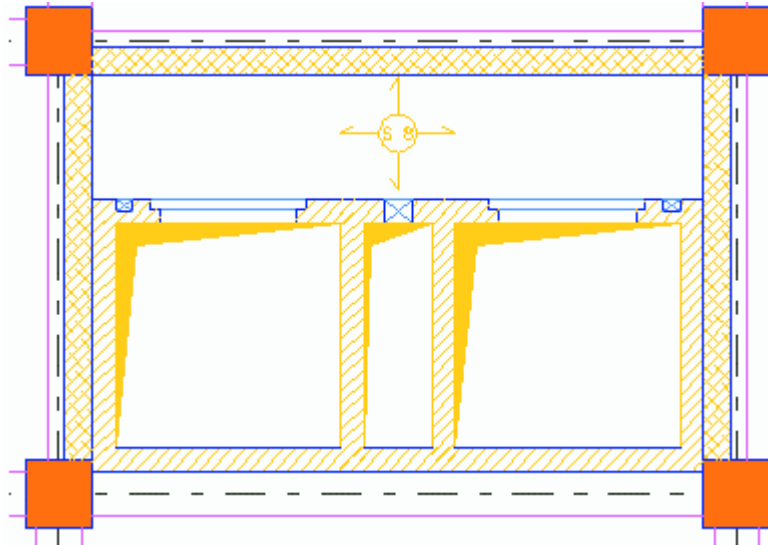


Figure 132: Elevator openings

Using the same process, create an opening of the same type and size in the bottom wall.

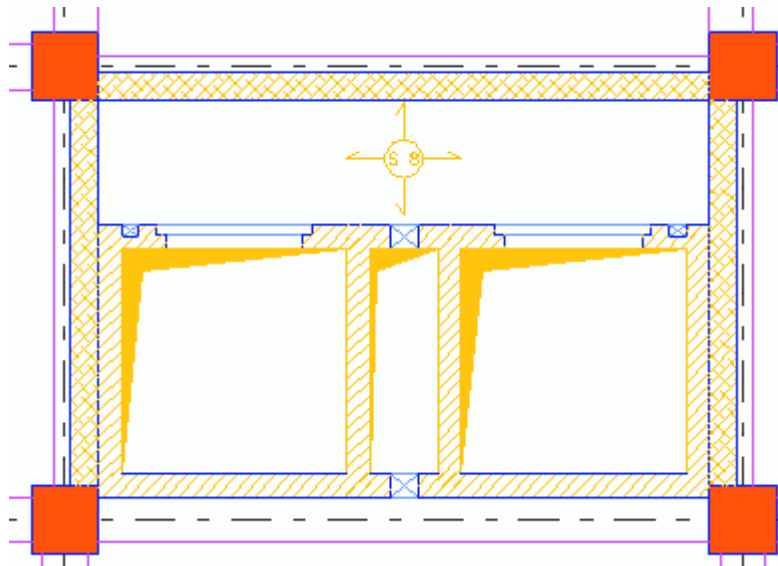
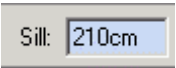


Figure 133: Elevator area – Wall openings

Adjust the opening position

For the opening on the top and bottom wall and middle wall, enter **210 cm**  and press **Enter**.

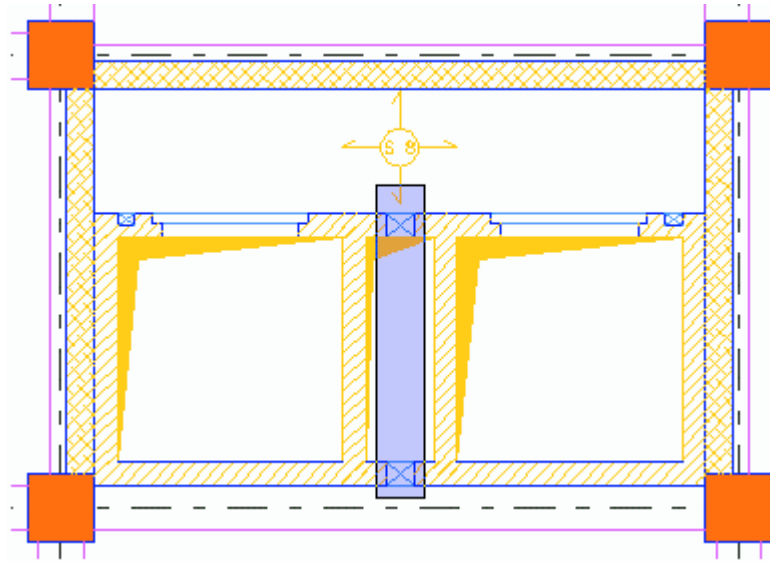


Figure 134: Middle wall opening

 Use the AutoCAD® **View** toolbar to view the openings in the front view.



Figure 135: AutoCAD® **View** toolbar

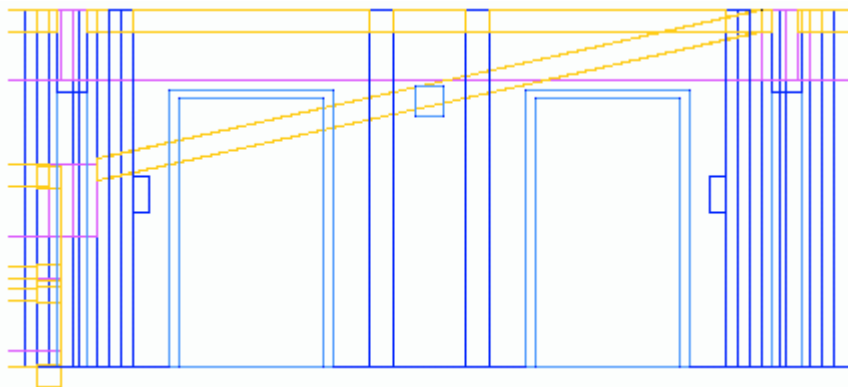


Figure 136: Elevator doors - front view

Step 7: Create beam opening lines

This step describes how to create circular openings in a beam.

Concept

The **opening line** object does not depend on the beam object. Thus, when you move a beam, the openings are not moved with the beam. If you delete a beam, the opening line is not deleted.

Zoom to the **D19 – E19** beam.

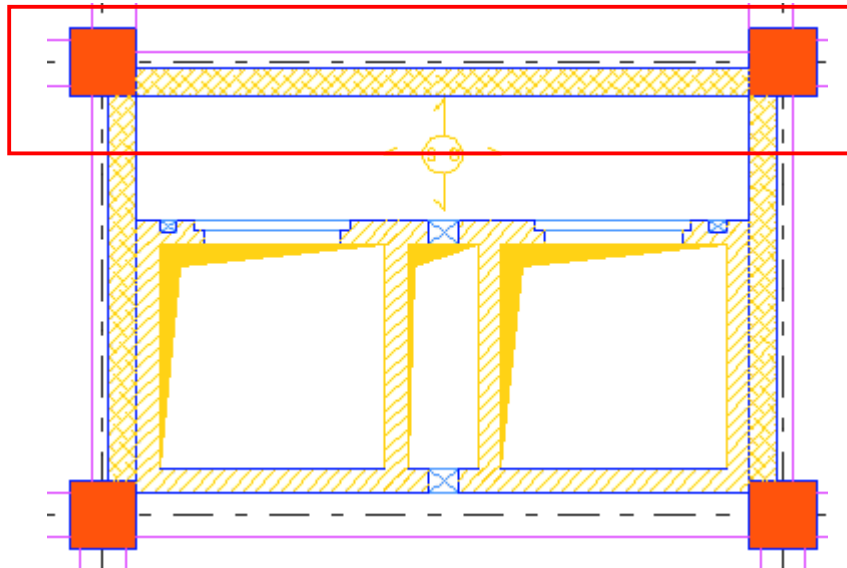


Figure 137: The **D18-D19** beam

1. On the **Model** toolbar, **Openings** flyout, click .
2. Define the line start point in the middle of the **D19-E19** wall.

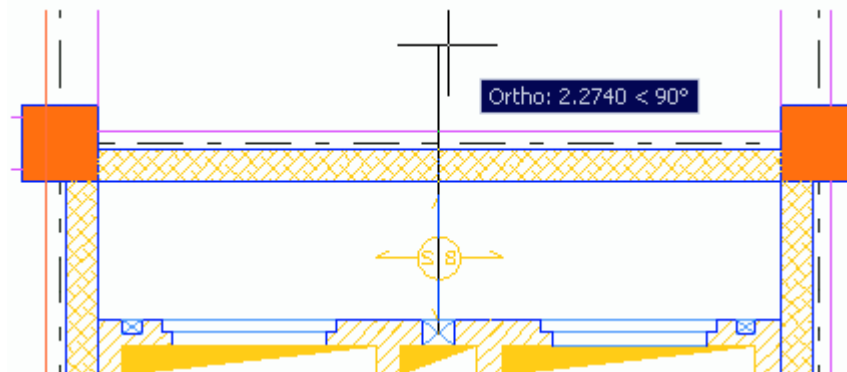
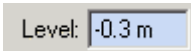


Figure 138: Start point of the opening

3. On the Smartbar, set the opening properties.
 - Set the hole size to **D5**. 
 - Set the height, relative to the upper level, to **-0.3 m**. 
 - Press **Enter**.
4. Press **Esc** to finish.

The first beam opening line is created.

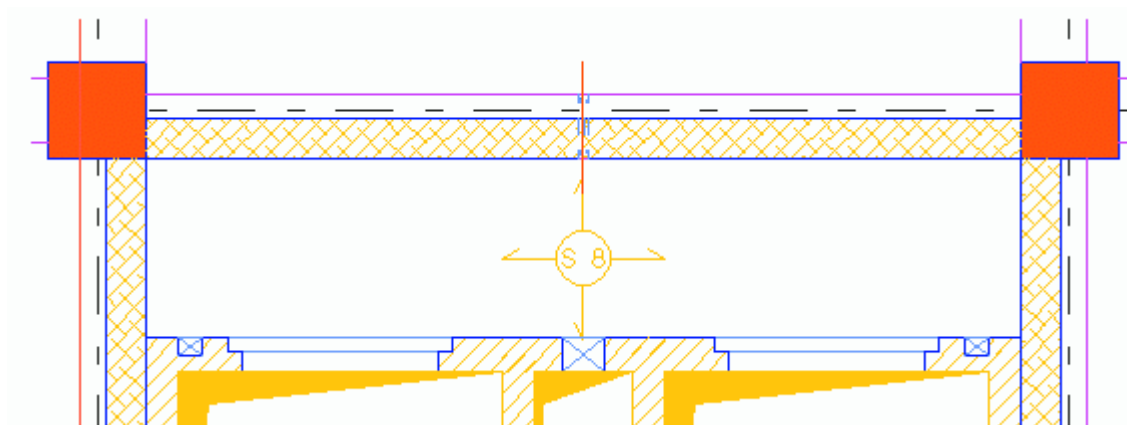


Figure 139: Creation of the first beam opening line

Using the same process, create two additional opening lines.

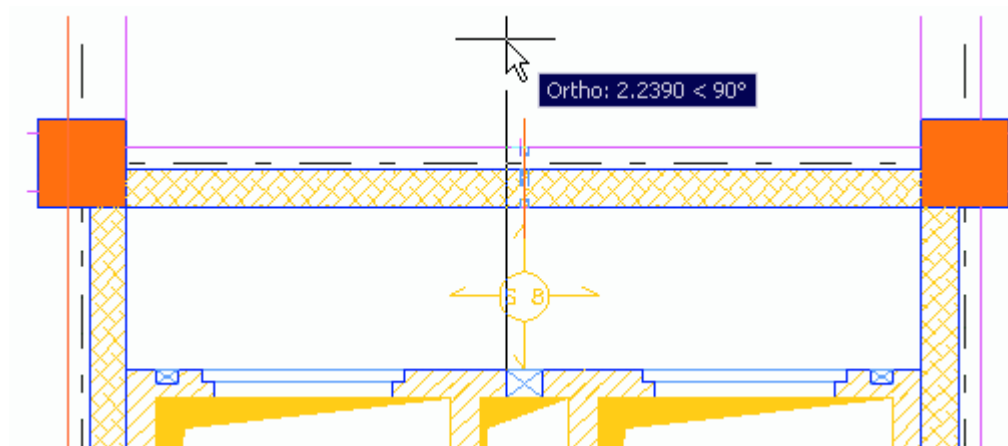


Figure 140: Placing the opening

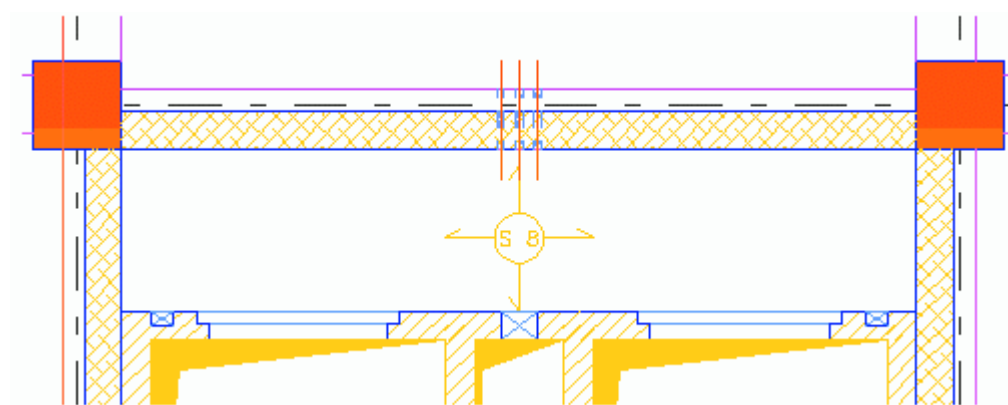


Figure 141: Created opening

On the AutoCAD® **View** toolbar, click  to view the beam openings.

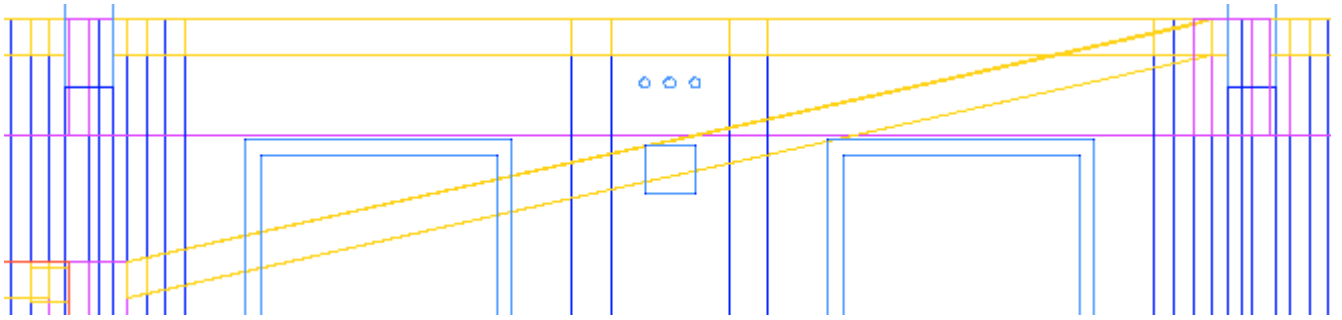


Figure 142: Front view of the model

Create another beam opening line to create openings in the **D19 – D20** and **E19 – E20** beams, at a distance of **30 cm** from the columns.

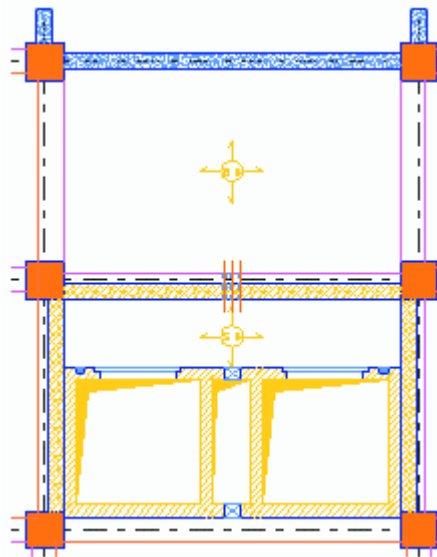
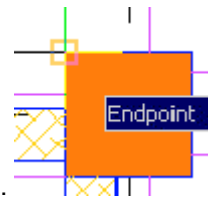
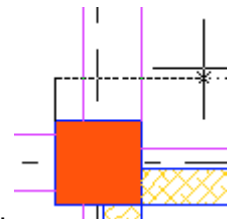


Figure 143: Creating a beam opening line

1. On the **Model** toolbar, **Openings** flyout, click .



2. Select the first point at the top-right corner of the **E18** column.



3. Move the cursor in the **Y** direction, enter **0.3** (the distance) on the command line.

4. Press **Enter**.

5. Move the parallel with the beam and click the second point.

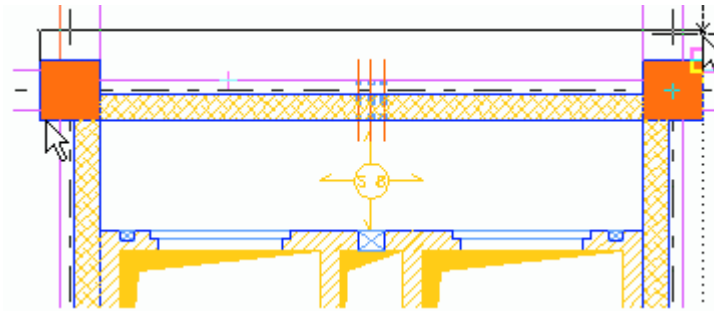


Figure 144: Create the beam opening line

6. Press **Enter** to finish.

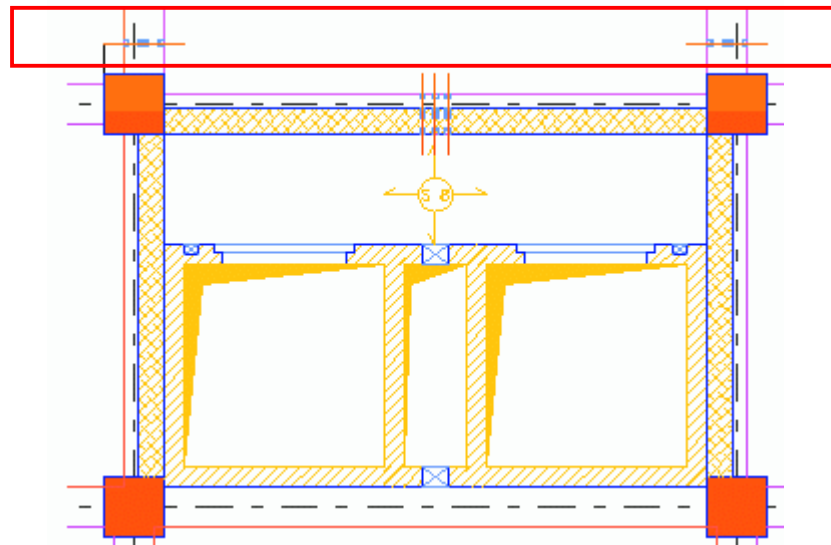


Figure 145: Openings

Using the same process, create another beam opening line parallel with the **E19 – E18** beam. The openings are created in the **E19 - F19** and **E18 – F18** beams, **20cm** from the column.

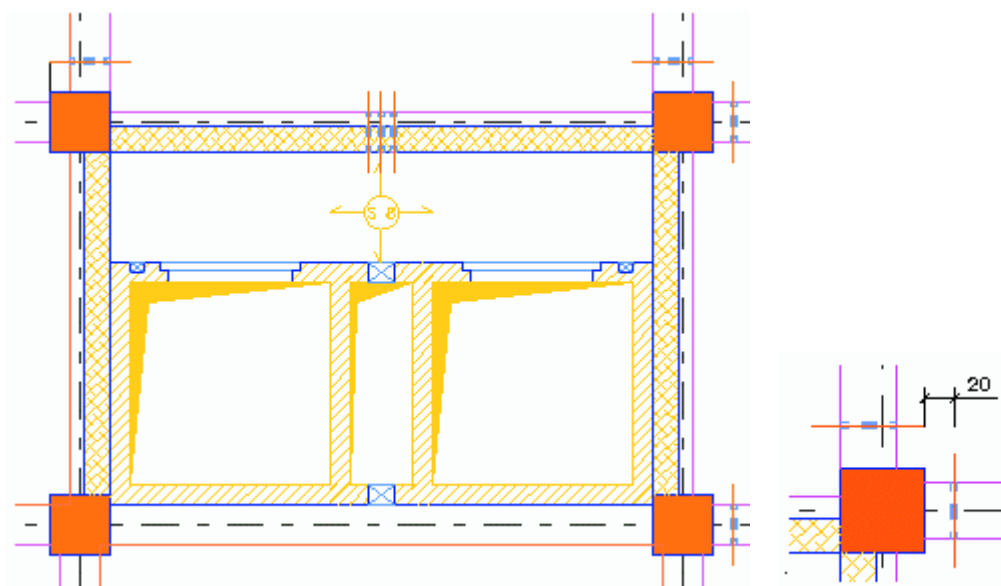


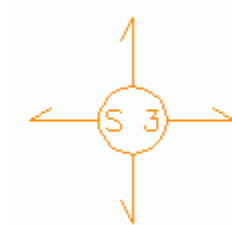
Figure 146: Representation of the openings

Step 8: Create a polygonal slab opening

This step describes how to create a polygonal slab opening by clicking the corner points.

Zoom to the **F'21'** axis intersection.

1. On the **Model** toolbar, **Openings** flyout, click .



2. Click the **S3** slab symbol to select the slab.
3. Create a polygonal opening, as you would draw a polyline in AutoCAD® following the numbering in the Figure 147.

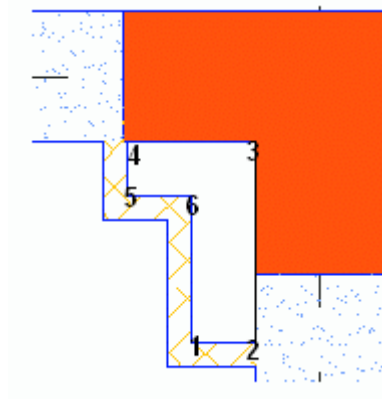


Figure 147: The steps of creating the polygonal opening

4. Press **Enter** to finish.

The polygonal opening is created.



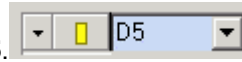
Figure 148: Polygonal opening

Step 9: Complete the model by adding all remaining openings

Create round openings with **5cm** diameters in the **D19 - E19** wall.

1. On the **Model** toolbar, **Openings** flyout, click .
2. Select the upper wall of the elevator area.
3. On the Smartbar define the following settings:

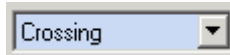
- Set the hole size to **D5**.



- Set the wall sill: **265**.



- Select a crossing wall opening.



4. Create round wall openings in front of the beam openings.

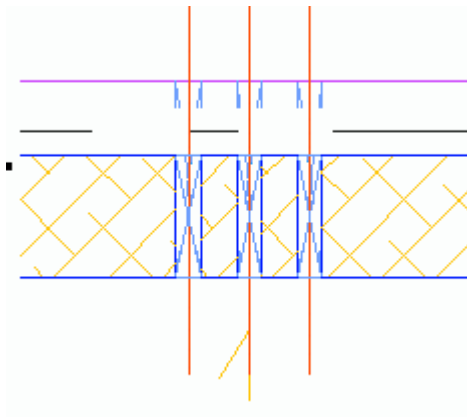


Figure 149: Created openings

 On the AutoCAD® **View** toolbar, click  to view the openings.

Lesson 9: Creating an elevator pit

In this lesson, you will create a foundation with elevator pit. The foundation is created in the **Foundation level** using the Advance slab tool.

You will learn how to:

- Activate and view the building levels.
- Use the "slab" tool to create the building foundation.
- Create an elevator pit.
- Create openings.

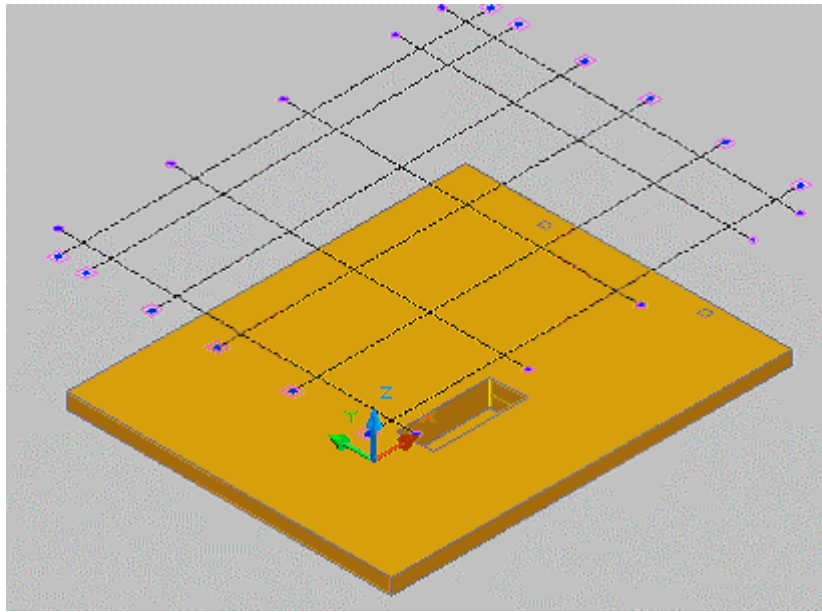


Figure 150: Elevator pit of the Model

During this lesson, a foundation slab is created with a thickness of **80 cm** and an elevator pit with a depth of **100 cm**. Additionally, a slab opening is created next to the **F'20** and **F'18** columns.

Concepts

The **active level** is the level in which the modeling elements are created.

A **visible level** is a level displayed on the screen but entities cannot be created on it.

A level could be visible without being active.

Step 1: Activate and view the levels

Activate the **Foundation** level.

1. In the **Pilot**, in **Model** mode, right click the **Foundation** level and select **Enable** from the context menu.

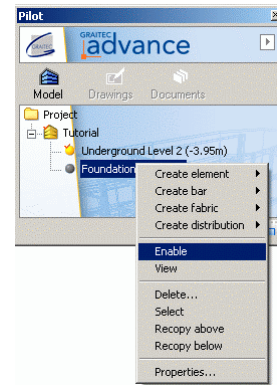


Figure 151: Activating the **Foundation** level

The **Foundation** level is enabled and only the building grid is visible. Since the foundation is created for the entire building, it is necessary to display the elements created in the **Underground** level.

2. Right click **Underground Level 2 (-4.00 m)** and select the **View** option to display the elements.

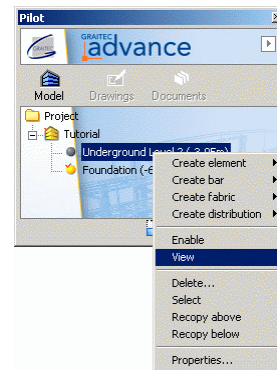


Figure 152: View **Underground Level 2**

The elements created in the **Underground** level are displayed in gray.

You can still work in the **Foundation** level and use the elements from the **Underground** level.

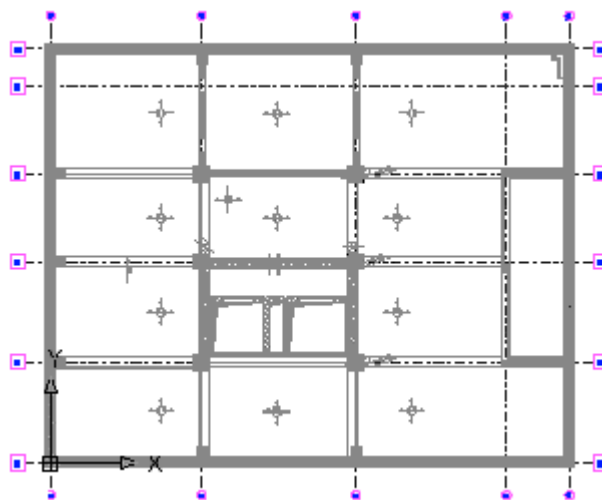
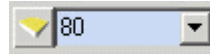


Figure 153: **Underground** level - disabled

Step 2: Create the foundation slab

1. On the **Model** toolbar, **Structural** flyout, click .

2. On the Smartbar, set the slab thickness to **80 cm**.



3. Press **Enter**.

Create a slab by clicking the corner points: the first point (located at the **C21'** intersection on the left corner).

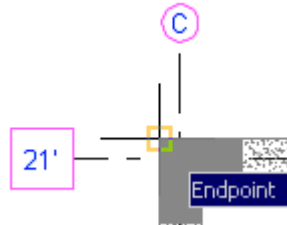


Figure 154: Creation of the slab

4. Click the corner points of the column sections at the **F'21'**, **F'17** and **C17** intersections.

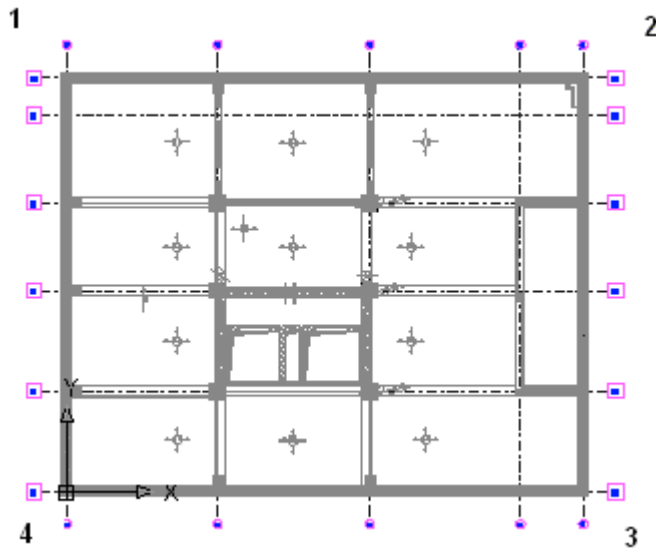


Figure 155: Creation of the slab

5. Press **Enter** to finish.

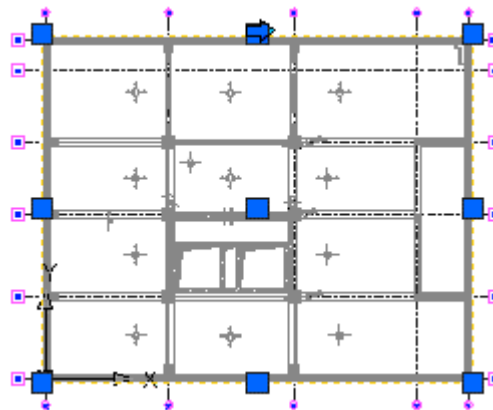


Figure 156: Foundation slab

For a clear view, move the slab symbol:

1. Click the slab symbol to select the slab.
2. Click the symbol grip point and move the symbol to the desired position.

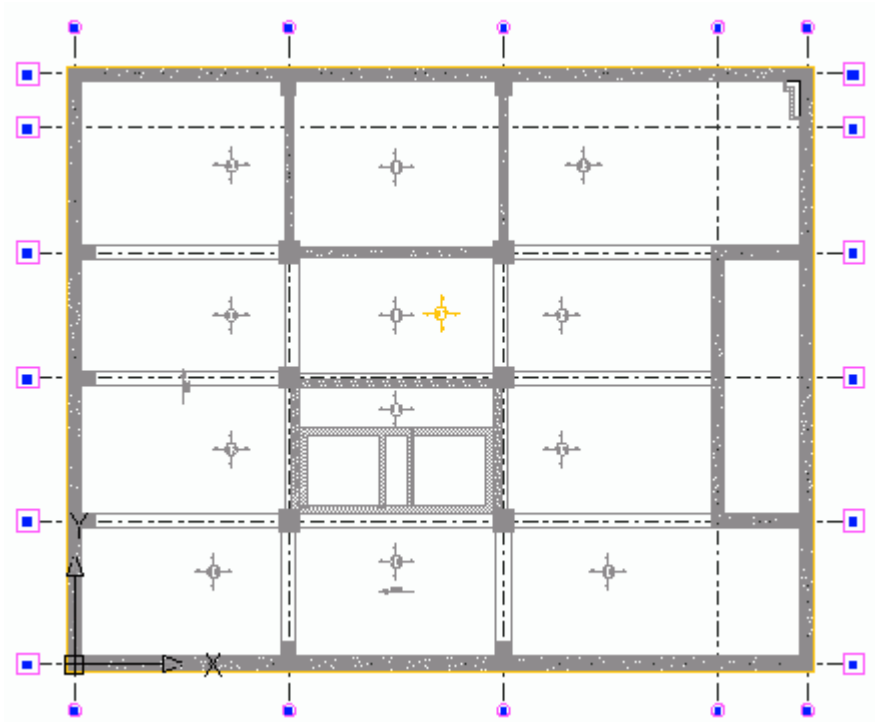
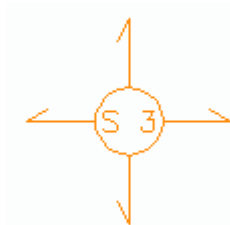
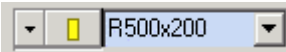


Figure 157: Adjusted slab symbol

Step 3: Create the elevator pit

1. On the **Model** bar, **Openings** flyout, click .



2. Click the slab symbol to select the slab.
3. On the Smartbar, set the opening size to **R500x200**. 
4. On the Smartbar, click  to display the properties dialog box.
 - On the **Definition** tab, from the **Depth A** drop-down list, select **elevator pit**.
 - In the next field, set the elevator pit depth to **100 cm**.

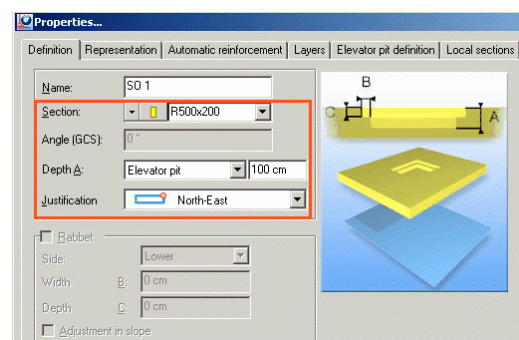


Figure 158: "Properties" dialog box- Definition tab

Other parameters can be modified on the **Elevator pit definition** tab.

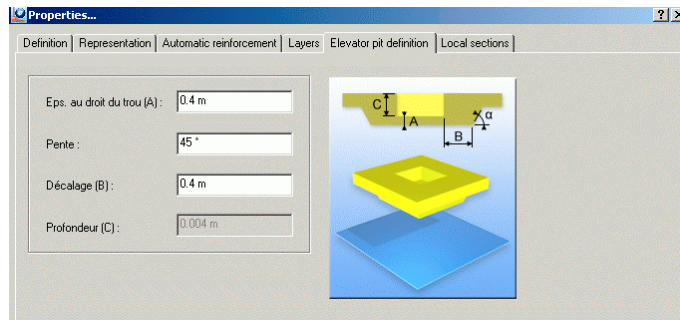


Figure 159: Elevation pit parameters

5. Click **OK**.
6. Click to define the position. Use the elevator walls to place the elevator pit.
7. Click a point to define the rotation angle (0).

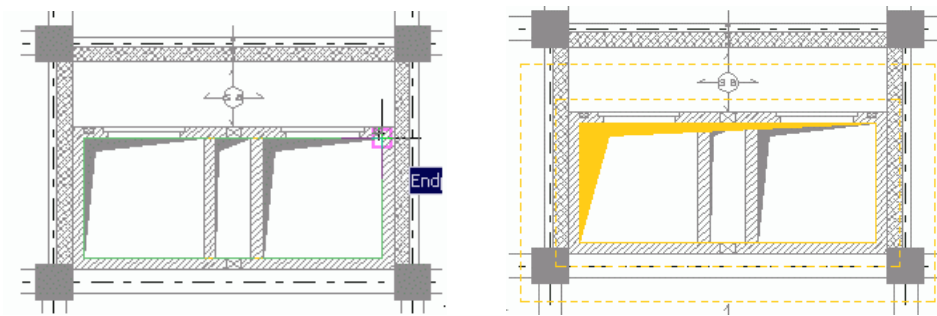


Figure 160: Elevator pit



On the AutoCAD® **View** toolbar, click  to view the elevator pit.

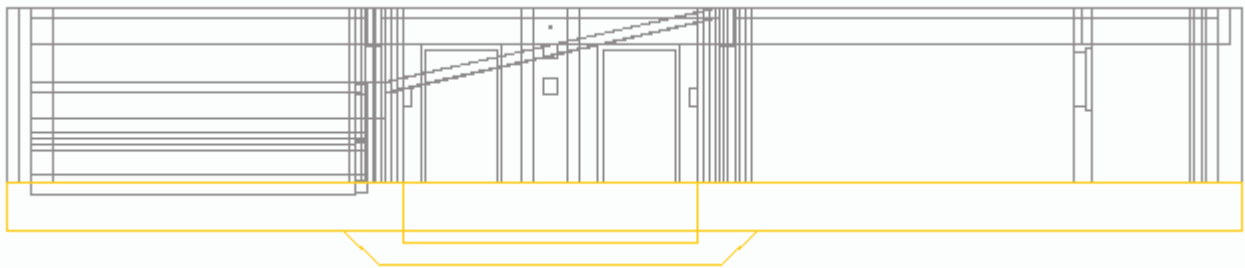


Figure 161: Elevator pit – Front view

Click  to return to the top view.

Step 4: Create upper depressions in the foundation slab

In this step, square depressions with **40 cm** depth are created.

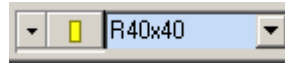
1. On the **Model** bar, **Openings** flyout, click .



2. Click the slab symbol to select the slab.

3. On the Smartbar, define the following settings:

- Select upper depression.



- Set the opening size to **R40x40**.
- Set the depression depth to **40 cm**.

4. Position the first hole next to the **F'20** column as in Figure 162.

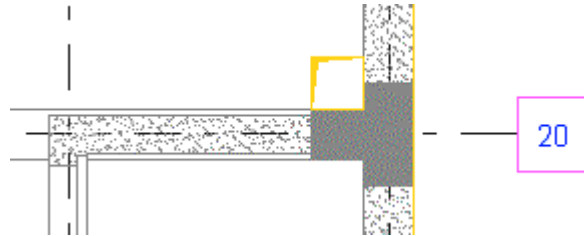


Figure 162: Creation of the first depression

Using the same process, create the second hole next to the **F'18** column.

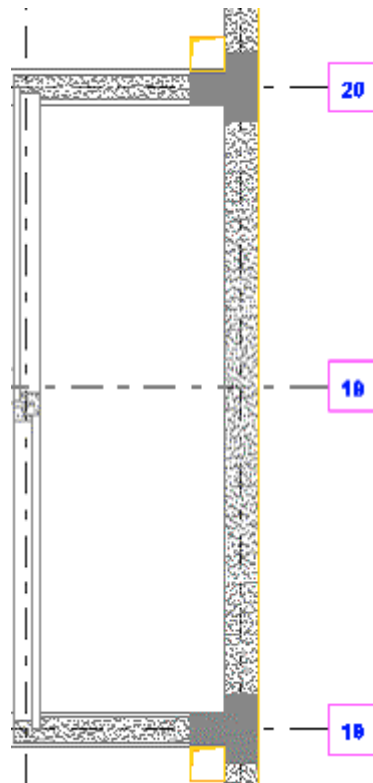


Figure 163: Creation of the second hole



Using AutoCAD® tools to view the result

To view the elevator pit and the slab holes in 3D, use the AutoCAD® **View** toolbar.



Figure 164: AutoCAD® View toolbar

For a more realistic presentation of the **Model**, use the AutoCAD® **Visual Styles**.



Figure 165: AutoCAD® **Visual styles** toolbar

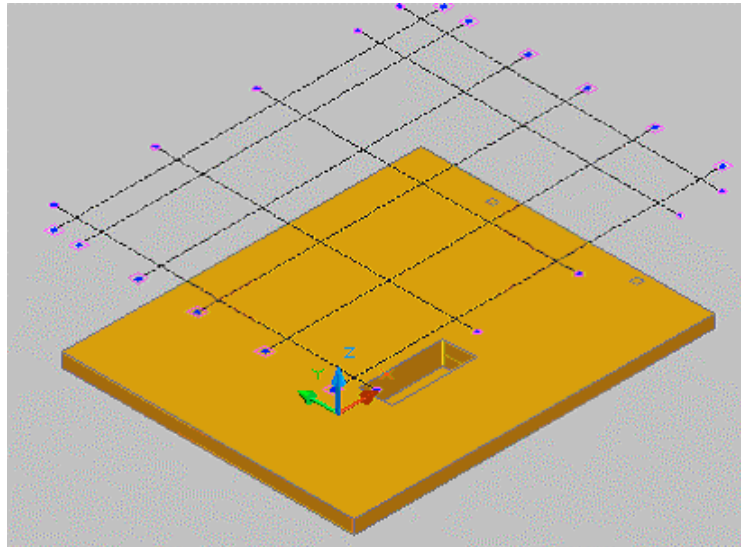


Figure 166: View of the model in 3D

Use the AutoCAD® **Orbit** tool to rotate the drawing in 3D.



Figure 167: AutoCAD® **Orbit** toolbar

In the Pilot, right click **Underground level 2** and select **View** from the context menu to display the **Underground** level.

To view the entire building in **3D**, click  on the AutoCAD® **View** toolbar.

Lesson 10: Creating top and underground levels

In this lesson, you will add new levels.

You will learn how to:

- Copy an existing level.
- Create a new level above the building.
- Copy elements from one level to another.

You will create three levels: **Underground –1st Level**, **Intermediate Level** and **Ground Level** and copy the elevator elements from one level to another.

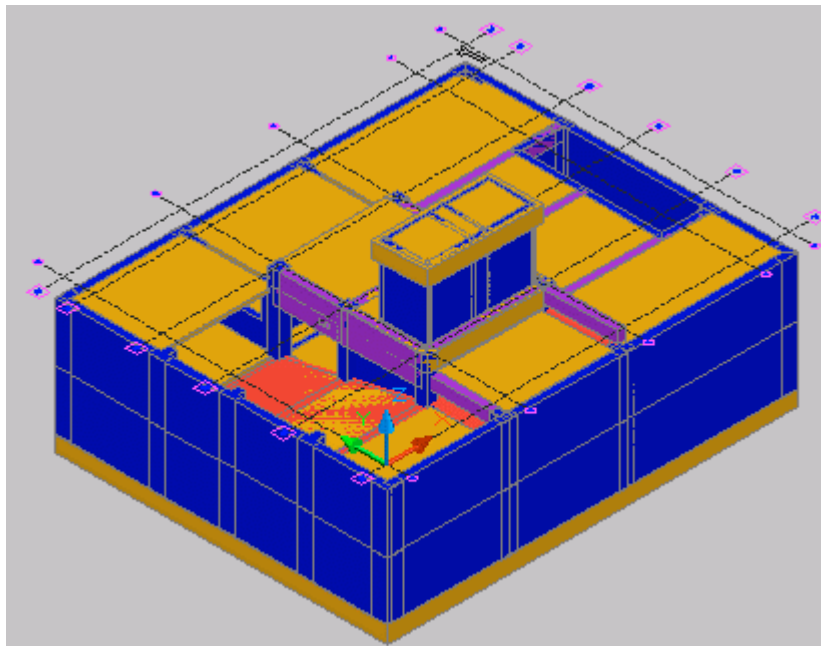


Figure 168: Levels of the Model

Step 1: Create a new level above the building

1. In the Pilot, right click **Underground Level 2** and select **Recopy above** from the context menu.

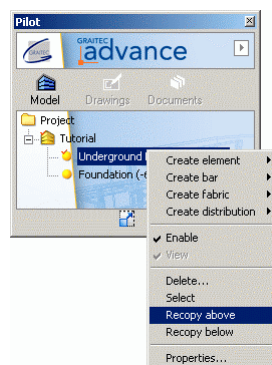


Figure 169: Recopy the **Underground Level 2**

2. A question appears on the command line:

Would you like to copy the visual elements (dimensions, annotations) of the drawing layout? [Yes/No]

Type **N** (No) and press **Enter**.

The level and all its elements are copied.

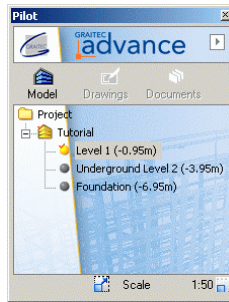


Figure 170: New level displayed in the Pilot

Delete the elements that are not necessary in the new level.

1. Select the **S5** and **S8** slabs and delete them.

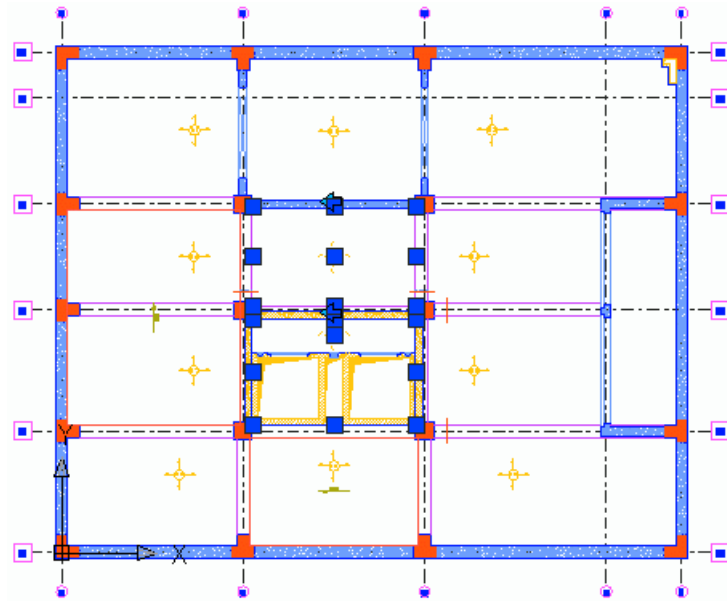


Figure 171: Selection of the **S5** and **S8** slabs

2. Select the **D19 – E19** wall and delete it.

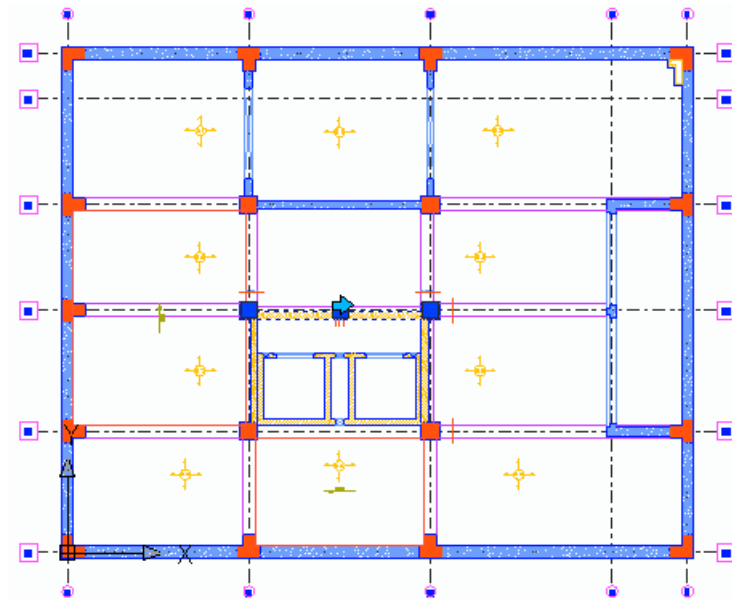


Figure 172: **D19 - E19** walls

Step 2: Modify the elevator walls

Selecting the elevator walls

1. Include the elevator area in a window selection.

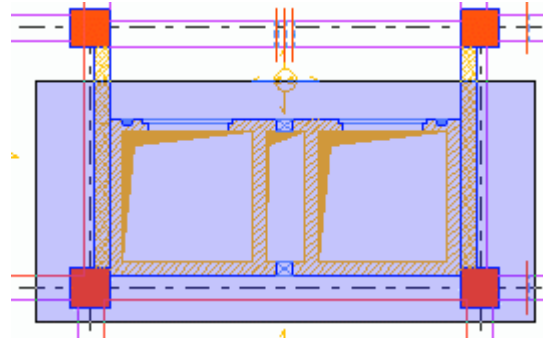


Figure 173: Selecting the elevator area

Use the AutoCAD® **Quick select** tool to select only the elevator walls.

2. Right click the selected elements and select **Quick Select** from the menu.
 - From **Apply to** drop-down list, select **current selection**.
 - From **Object type** drop-down list, select **Wall**.
 - From **Operator** drop-down list, select **Select all**.

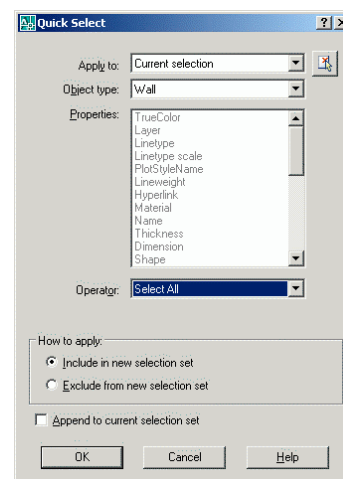
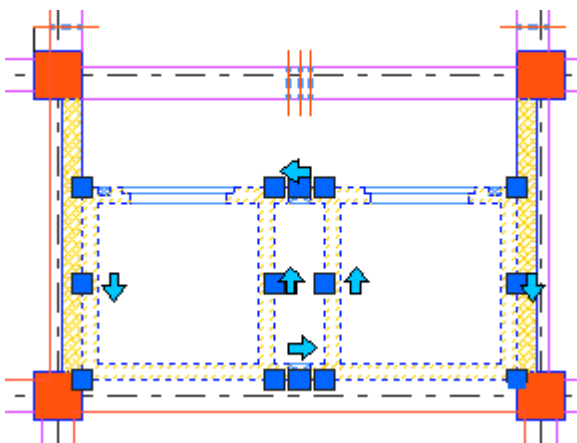


Figure 174: Selecting the walls using quick selection

3. Click **OK** to validate. Only the elevator walls are selected.

Modifying the wall height

1. On the Smartbar, click to display the properties dialog box.
 - On the **Definition** tab, in the **Value** field, set the wall height, relative to upper level, to **0.95 m**.

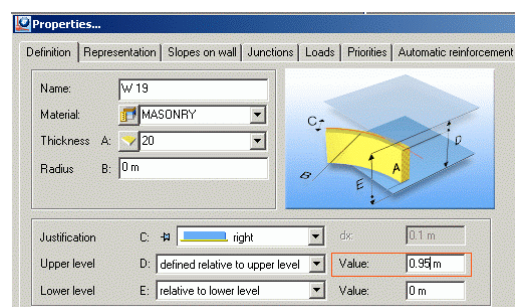


Figure 175: Properties dialog box – Definition tab

2. Click **OK**.

Using AutoCAD® tools to view the elevator walls

On the AutoCAD® **View** toolbar, click  for a front view.

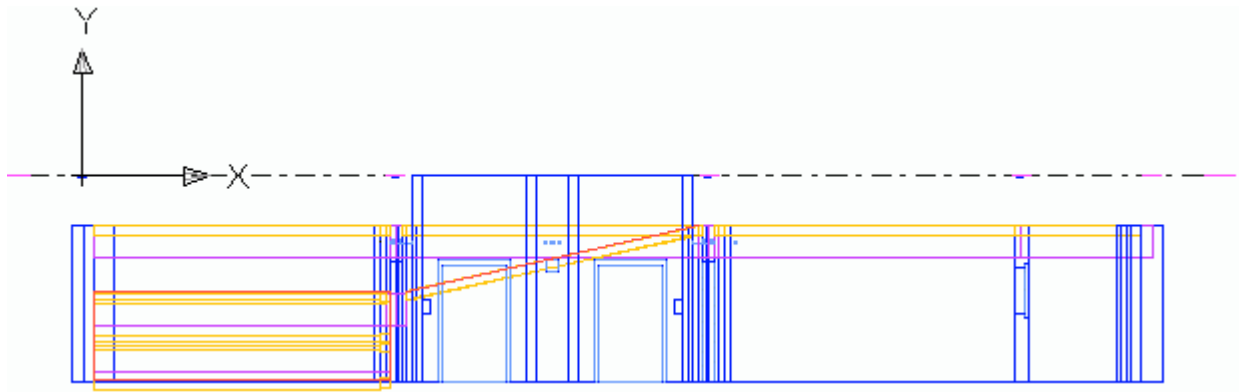


Figure 176: Front view of the model

On the AutoCAD® **View** toolbar, click  to return to the top view.

Step 3: Create the intermediate level

In this step, you will create a new level, containing only a slab.

1. In the Pilot, right click **Tutorial** and select **Create > Top level** from the context menu.

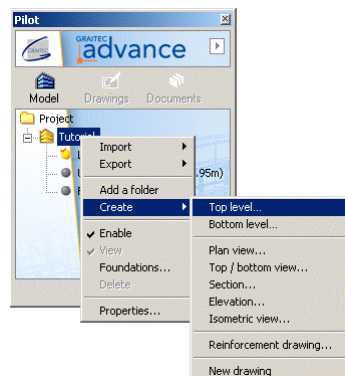


Figure 177: Creating the top level

2. On the **Properties** tab, in the **Elevation** field, set the level height to **0.95 m**.

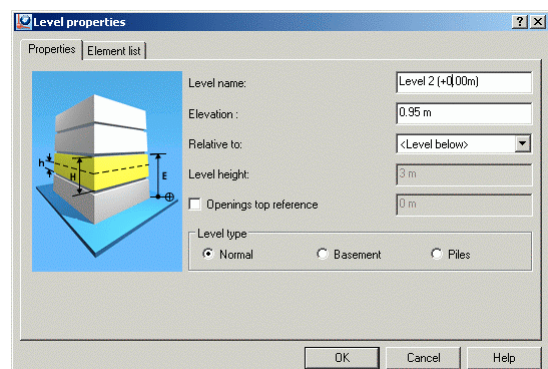


Figure 178: "Level properties" dialog box – **Properties** tab

3. Click **OK**.

The new level is created.

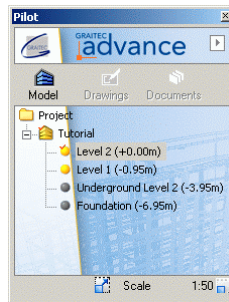


Figure 179: Top level displayed in the Pilot

Rename Level 1

1. In the **Pilot**, right click **Level 1 (-1.00 m)** and select **Properties** from the context menu.

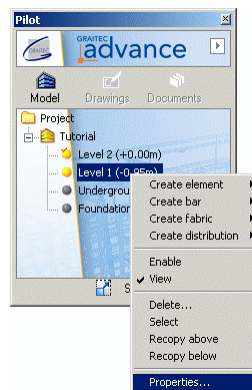


Figure 180: Accessing the Level 1 properties

2. On the **Properties** tab, enter the new name: **Underground Level 1 (-0.95 m)**.
3. Click **OK**.

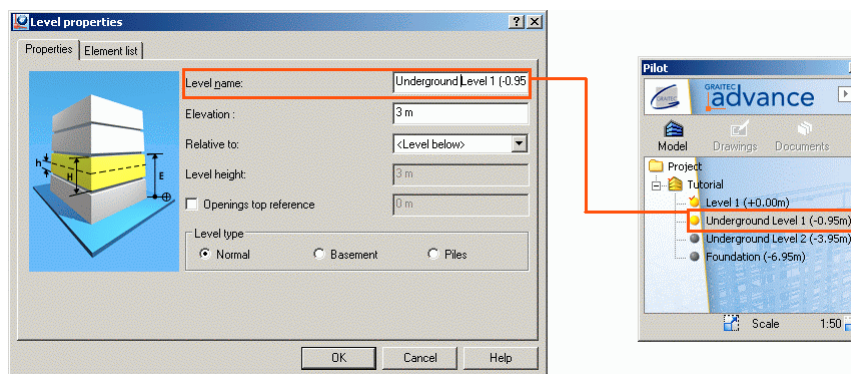


Figure 181: Renaming a level

Creating a slab in the top level on the elevator walls

1. In the Pilot, right click and select **Create element > Slab**.

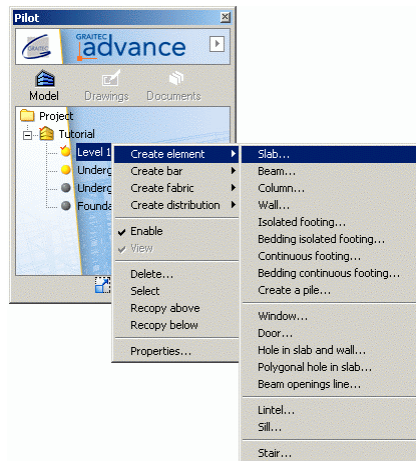


Figure 182: Access the "Slab" tool from the pilot context menu

2. On the Smartbar, set the slab thickness to **18 cm**.
3. Input the slab corners, point by point as in Figure183.

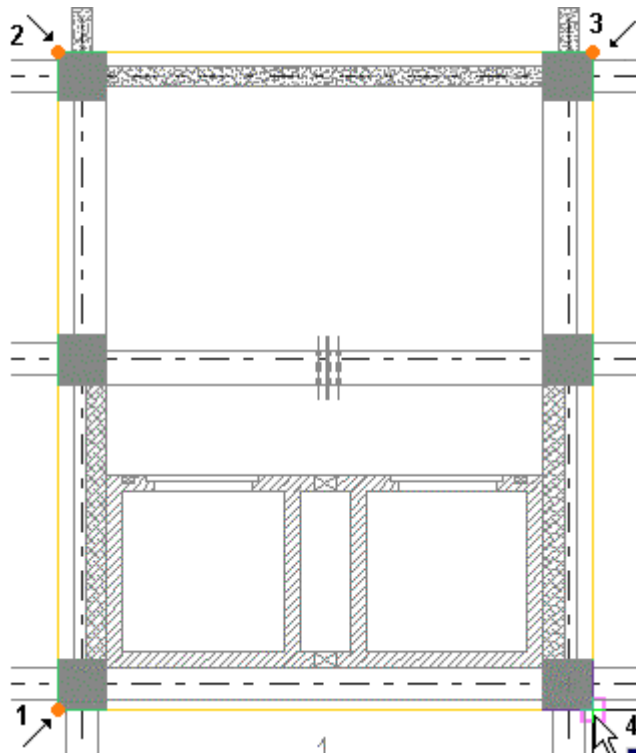


Figure 183: Setting the points of the slab creation

4. Press **Enter** to finish.
The slab is created.

On the AutoCAD® **View** toolbar, click for a front view.

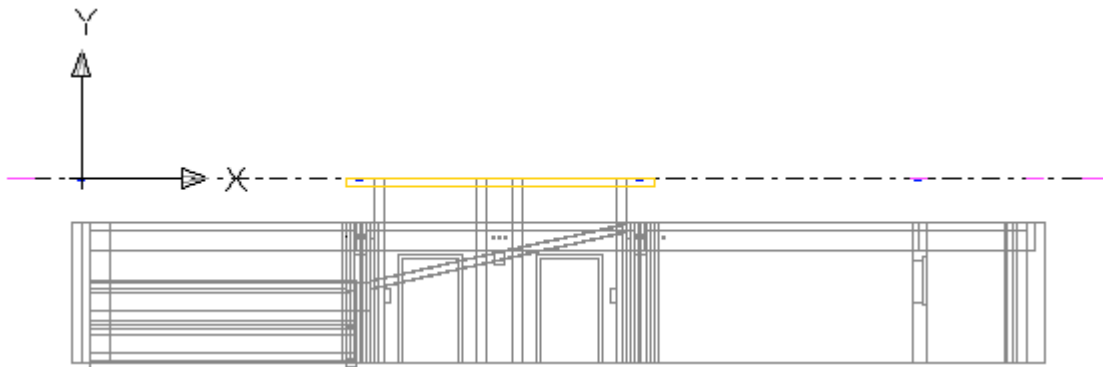


Figure 184: Front view of the model

Step 4: Create the Ground Level

1. In the Pilot, right click **Tutorial** and select **Create > Top level**.

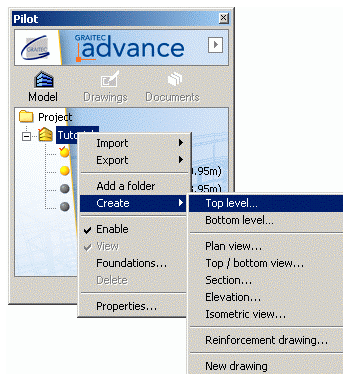


Figure 185: Selecting **Top Level** from the context menu

2. In the "Level properties" dialog box, make the following changes:
 - Set the new name of the level to **Ground Level**.
 - In the **Elevation** field, set the level height to **3.50 m**.
3. Click **OK**.

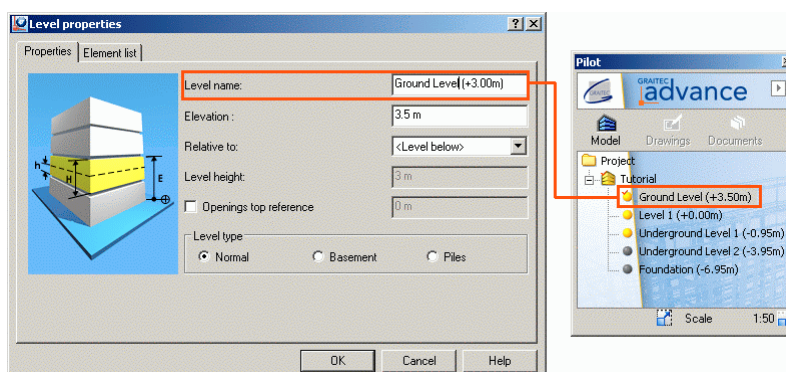


Figure 186: Ground level

Notice that in the Pilot the level elevation is automatically displayed.

Step 5: Copy elevator elements to the Ground Level

1. Select the elevator walls.

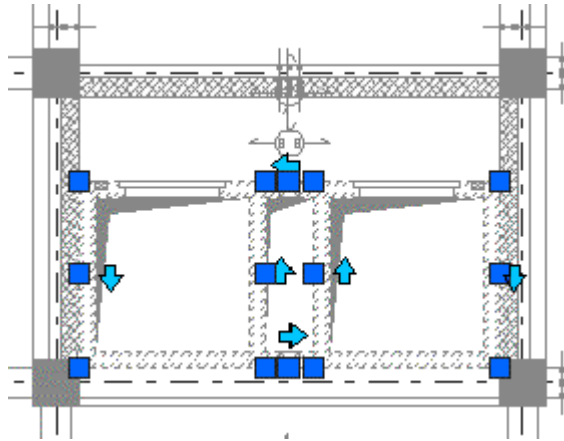


Figure 187: Selecting the elevator walls

2. On **Model** toolbar, **Elements** flyout, click  to copy the elevator elements.
3. In the "Copy/move" dialog box, from the **Target Level** drop-down list, select **Ground Level**.
4. Check the **Copy** option.
5. Click **OK**.

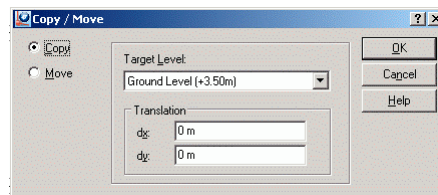


Figure 188: "Copy/Move" dialog box

The elevator elements are copied on the **Ground Level**.

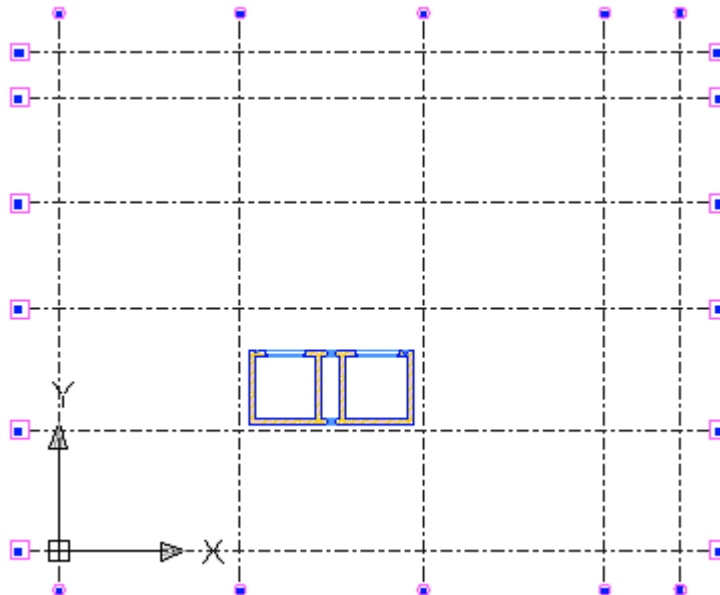


Figure 189: Elevator walls on the **Ground Level**

Create a slab on the Ground Level

1. On the **Model** toolbar, **Structural** flyout, click .
2. Click the slab corners as in the Figure 190.

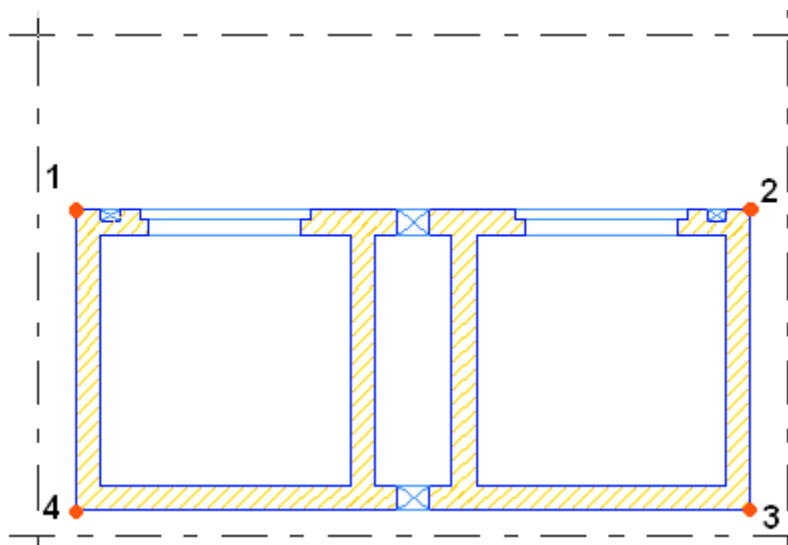


Figure 190: Placing the slab

3. Press **Enter** to finish.

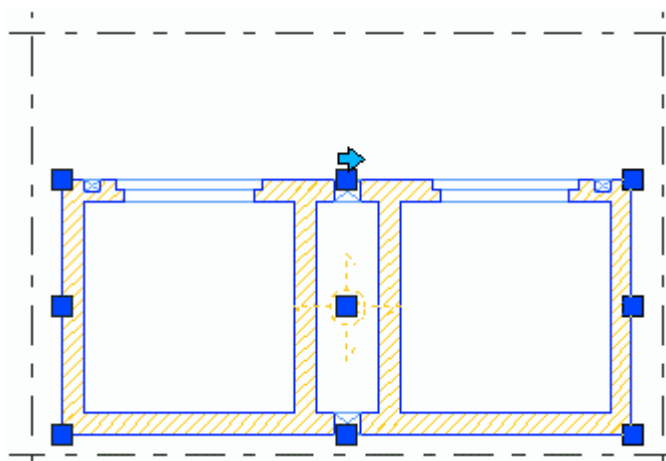


Figure 191: Created slab

Modifying the slab edges

1. Click the slab symbol to select the slab.
2. On the Smartbar, click  to access the slab properties dialog box.
3. On the **Edges** tab set the offsets.

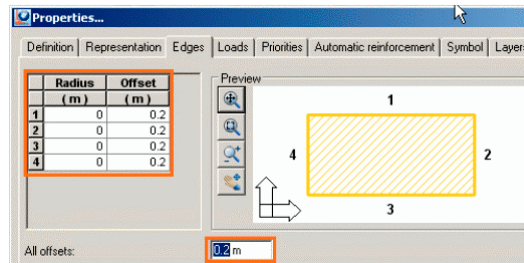


Figure 192: "Properties" dialog box – Defining slab edges

4. Click **OK**.

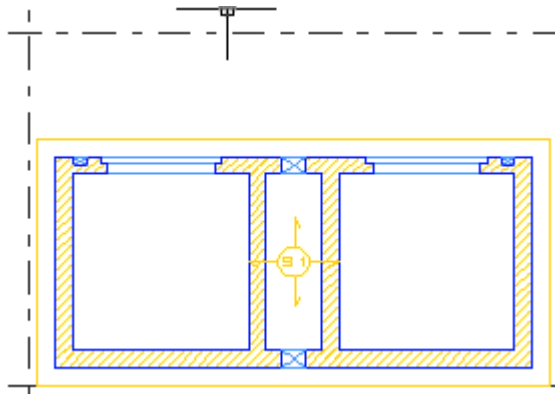


Figure 193: Slab edges



On the AutoCAD® **View** toolbar, click  to select a front view of the model.

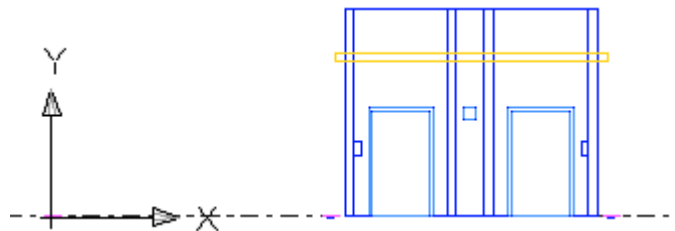


Figure 194: Front view of the drawing

Step 6: Modify the elevator wall heights

1. Select the elevator walls.

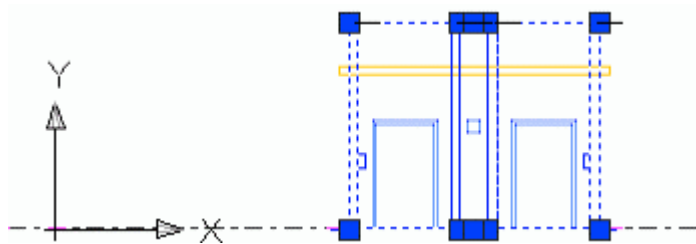


Figure 195: Selected walls

2. On the Smartbar, click  to access the properties dialog box.
3. On the **Definition** tab, change the elevator wall height relative to the upper level. In the **Value** field, type **0 m**.
4. Click **OK**.

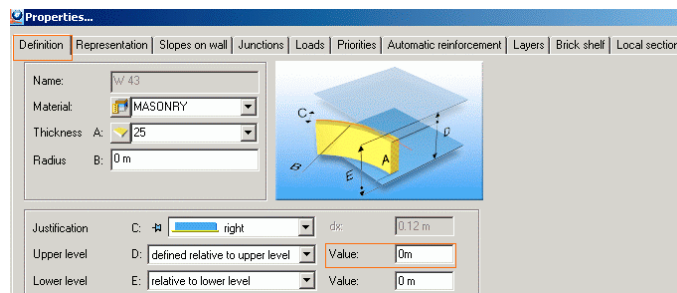


Figure 196: "Properties" dialog box - **Definition** tab

The height of the walls has changed.

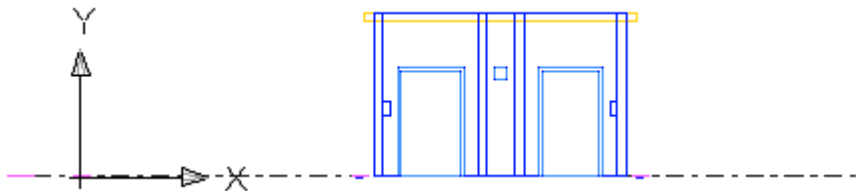


Figure 197: Adjusted walls

Using AutoCAD® tools to view the drawing in 3D



Figure 198: AutoCAD® View toolbar

For a realistic view, click  on the AutoCAD® Visual styles toolbar.

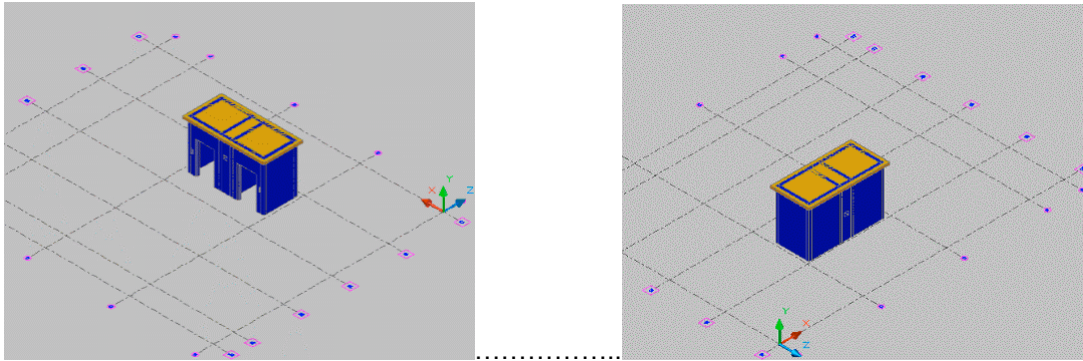


Figure 199: Model in 3D view

In the **Pilot**, right click **Tutorial** and select **Enable** to view the entire building in 3D.

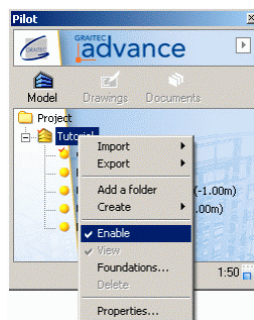


Figure 200: Display all levels

On the AutoCAD® Visual Styles toolbar, click .

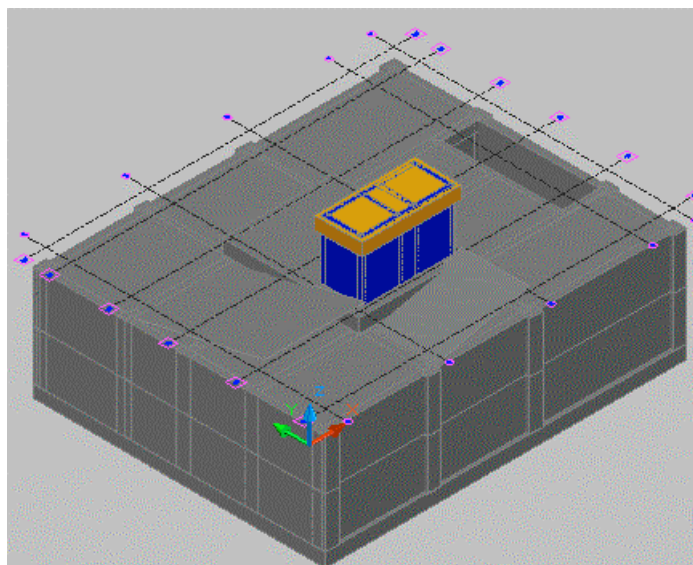


Figure 201: Building in 3D view

Step 7: Adjust the Ground Level connection elements

To complete the model, some adjustments are necessary in **Level 1**. You will modify the beam section size and the level relative to the upper level of the beams and columns near the elevator slab.

1. In the Pilot, right click **Level 1** and select **Enable** from the context menu.

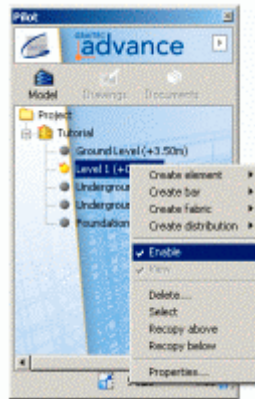


Figure 202: Activating Level 1

2. On the AutoCAD® **View** toolbar, click  for a top view of the level.

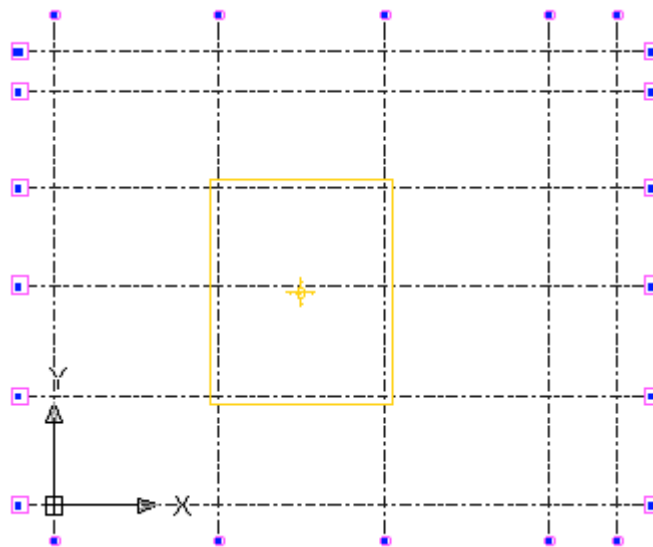


Figure 203: Slab in the Level 1 view

3. In the Pilot, in **Model** mode, right click **Underground 1st Level** and select **View** from the context menu to display the level.

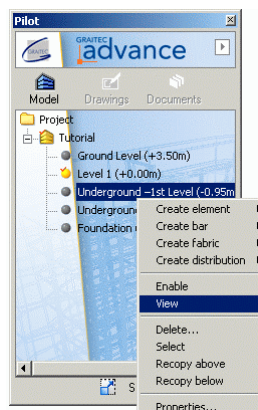


Figure 204: View **Underground – 1st Level**

4. Select the middle beam opening lines and delete them.

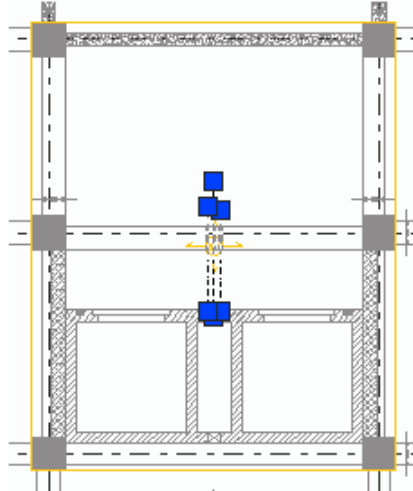


Figure 205: Middle beam opening lines

5. Select the **D19-E19** beam.

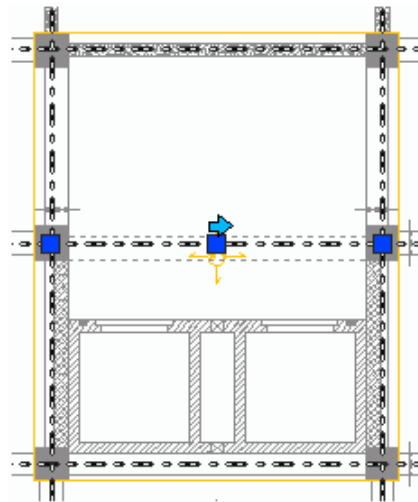


Figure 206: Selecting the **D19-E19** beam

6. On the Smartbar, click  to access the properties dialog box.
- On the **Definition** tab, set the beam upper level, relative to the upper level, to **0.95 m**.

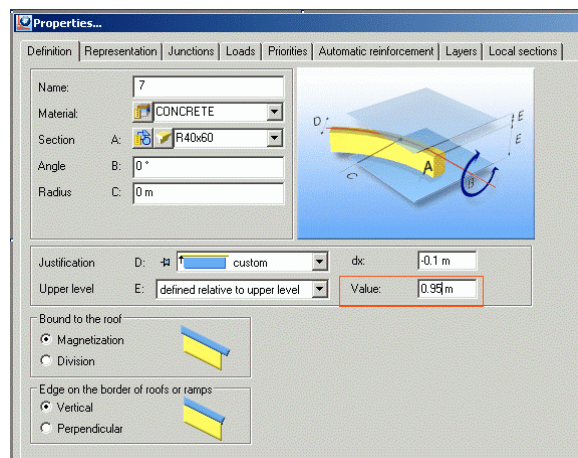


Figure 207: "Properties" dialog box - **Definition** tab

7. Click **OK**.

Modify the properties of the beams along the left and right sides of the slab.

1. Select the beams.

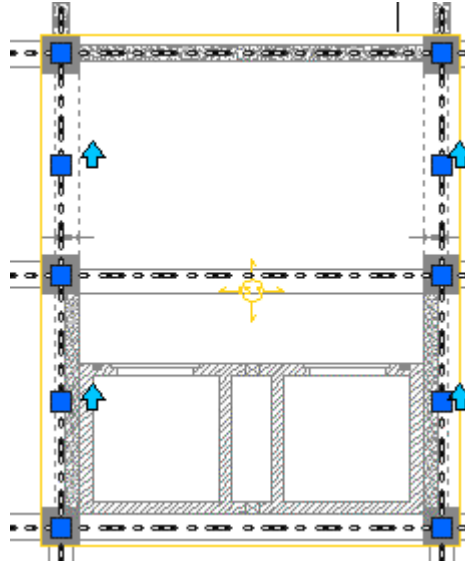


Figure 208: Selecting the **D18-D19-E18-E19** beams

2. On the Smartbar, click  to access the properties dialog box.

On the **Definition** tab:

- In the **Section** drop-down list, change the section of the beams to **R40x155**.
- Set the beam upper level to **0.95 m**.

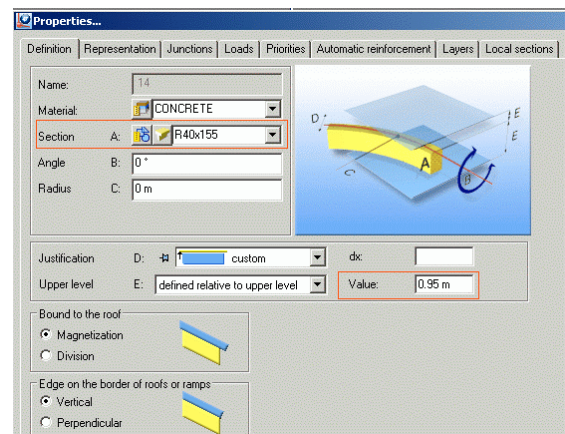


Figure 209: “Properties” dialog box – **Definition** tab

3. Click **OK**.

Use the same process for the **D18-E18** beam and set the upper level to **0.95m**.

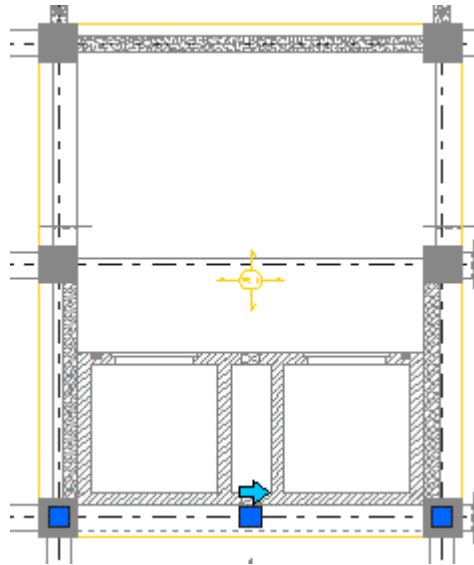


Figure 210: Adjusted beam

Use the same process for the **D20-E20** wall and set the upper level value to **0.95m**.

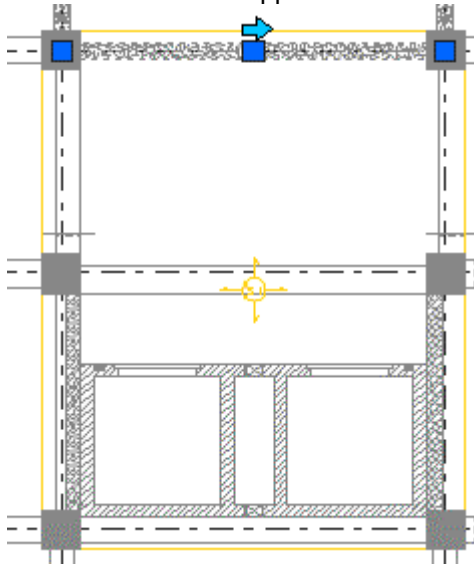


Figure 211: Adjusted wall



View the entire building in 3D

In the Pilot, in **Model** mode, right click **Tutorial** and select **Enable** from the context menu.

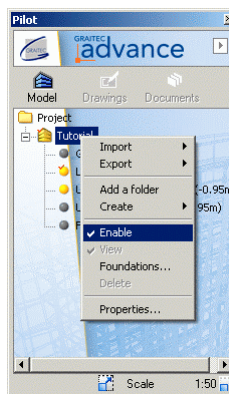


Figure 212: Preparing to view the model in 3D

Use the AutoCAD® **View** toolbar to view the drawing in 3D.



Figure 213: AutoCAD® **View** toolbar

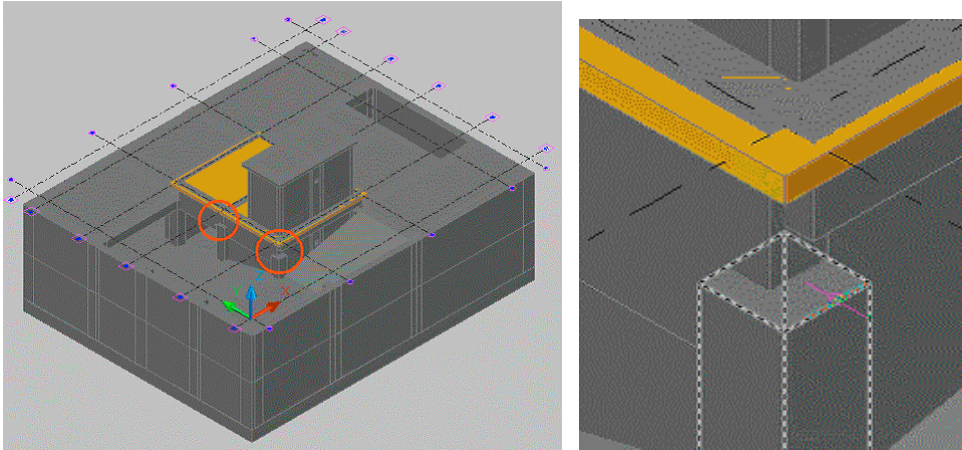


Figure 214: 3D View – the created slab

Notice that the column height must be adjusted.

1. Select the columns on each side of the slab (**D18, D19, D19** and **E18, E19, E20**).

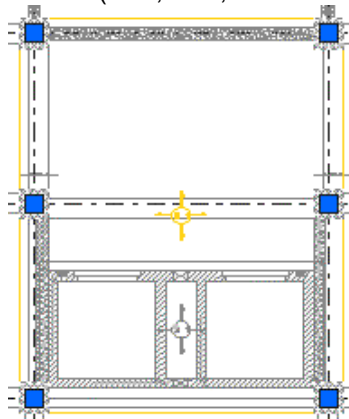


Figure 215: Selection of the columns

2. On the Smartbar, click  to access the properties dialog box.
 - On the **Definition** tab, set the value for the upper level to **0.95 m**.

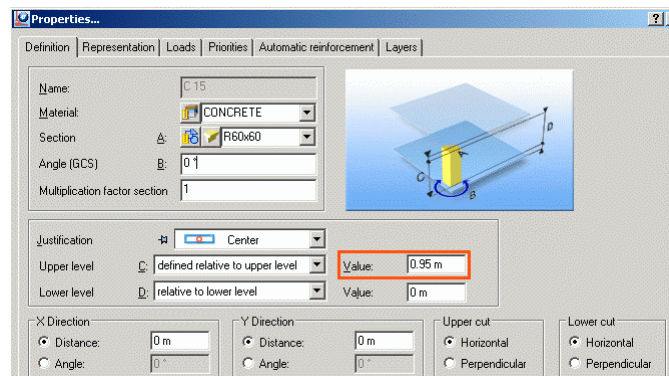


Figure 216: “Properties” dialog box - **Definition** tab

3. Click **OK**.

Viewing the model in 3D

In the Pilot, in **Model** mode, right click **Tutorial** and unselect the **Enable** option.

Use the AutoCAD® **Visual Styles** and **View** toolbars to view the model in 3D.



Figure 217: AutoCAD® **Visual Styles** toolbar



Figure 218: AutoCAD® **View** toolbar

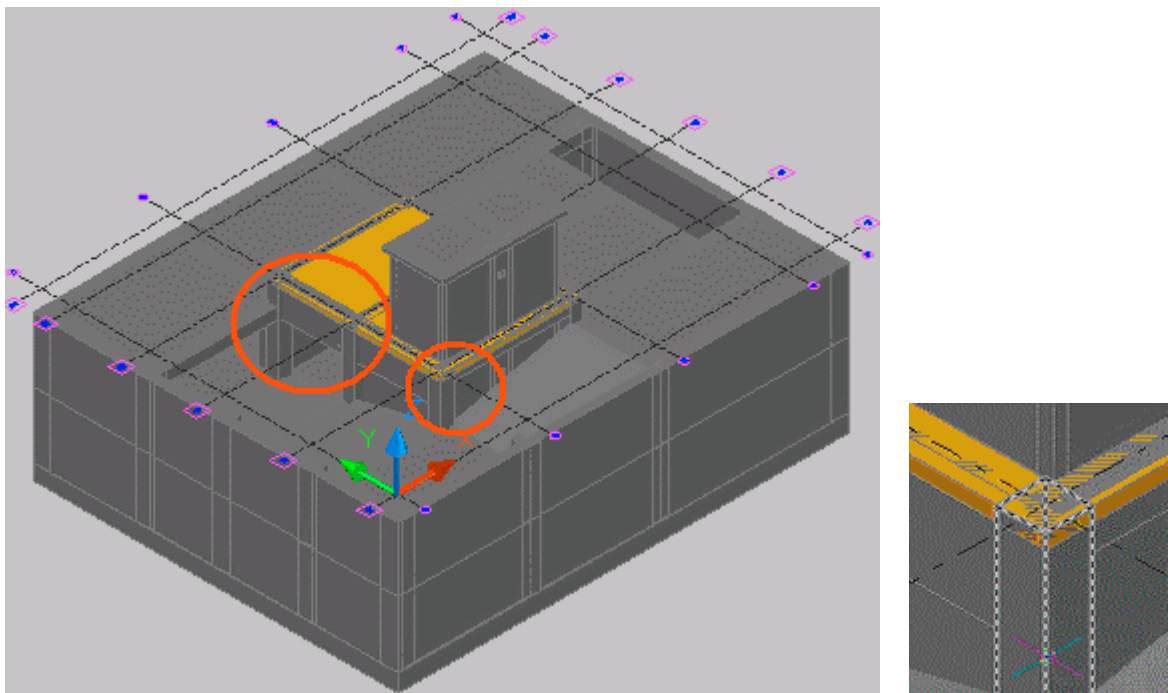


Figure 219: The columns adjusted to the slab

Lesson 11: Creating drawings

Once the model is complete, start the creation of the construction drawings.

You will learn how to:

- Create sections, elevations, isometric views, and total and partial cuts.
- Modify drawing properties.

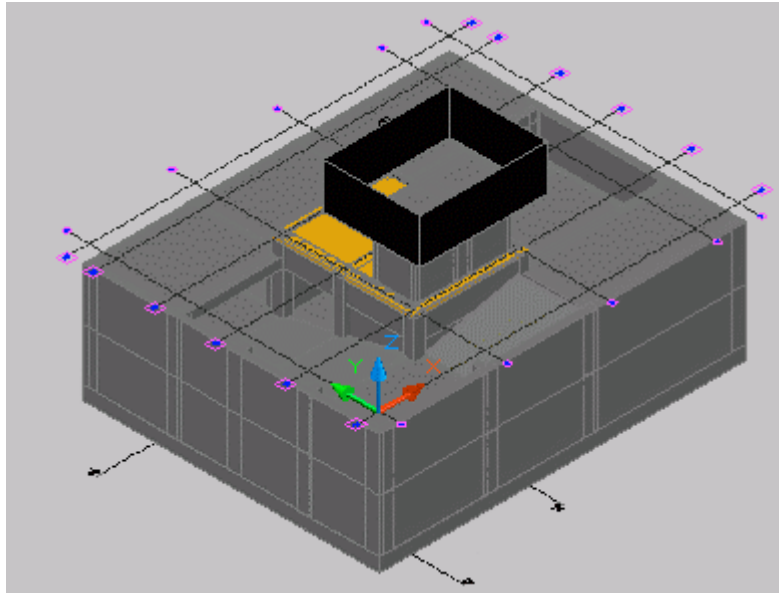


Figure 220: Drawings of the Model

Using the AutoCAD® Visual Styles and View toolbars

On the AutoCAD® **Visual Styles** toolbar, click  to view the drawing in 2D.



Figure 221: AutoCAD® Visual Styles - 2D Wireframe

On the AutoCAD® **View** toolbar, click  for a top view of the model.

Step 1: Create section cuts

1. On the **Model** toolbar, **Drawings** flyout, click .
2. Define the cutting plane by drawing a vertical line across the building.

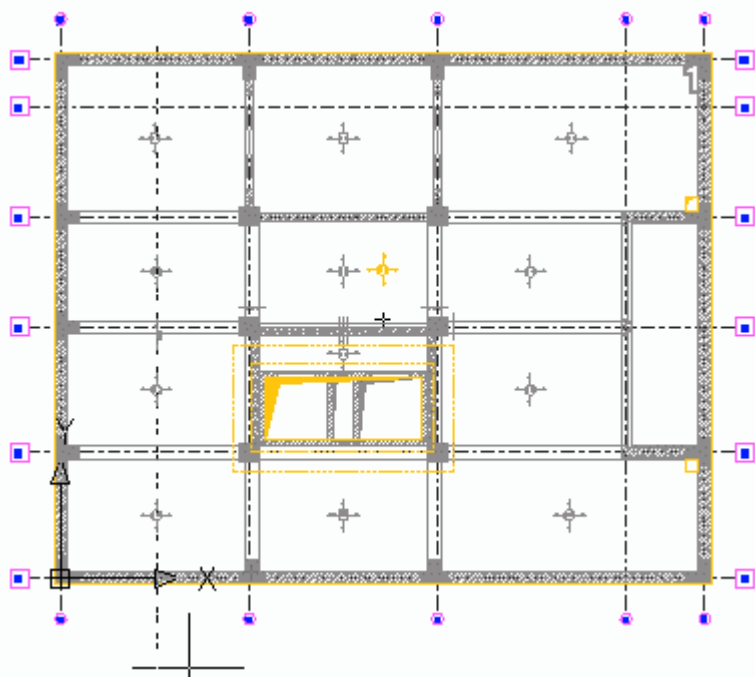


Figure 222: Creating the first cut

3. Press **Enter**.
4. Move the cursor in the **-X** direction to define the cut depth. Include the entire left part of the building as shown in Figure 223 and click a point.

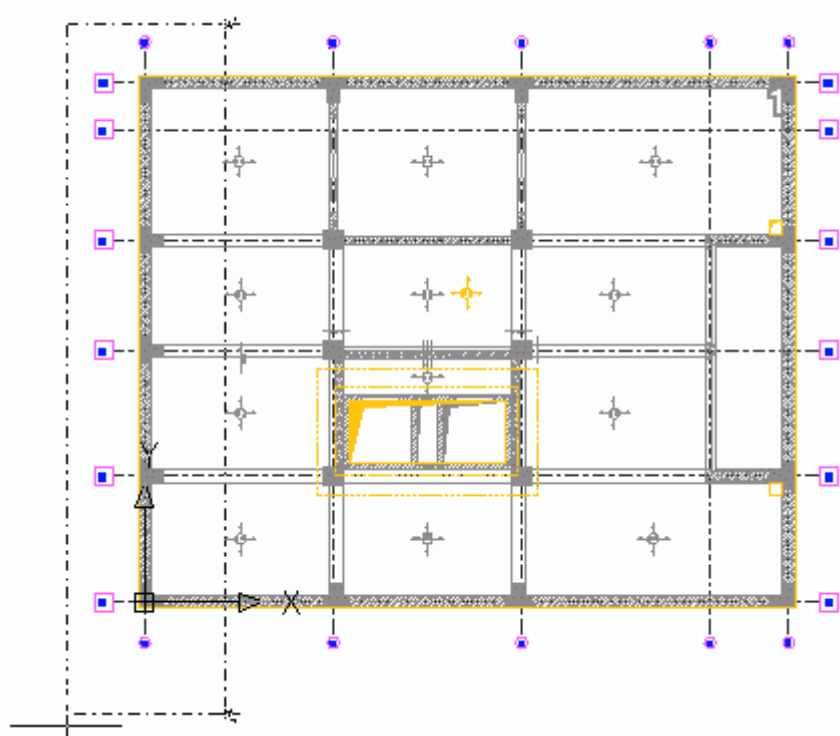


Figure 223: Creating the cut

Using the same process, create the cuts in the following figure.

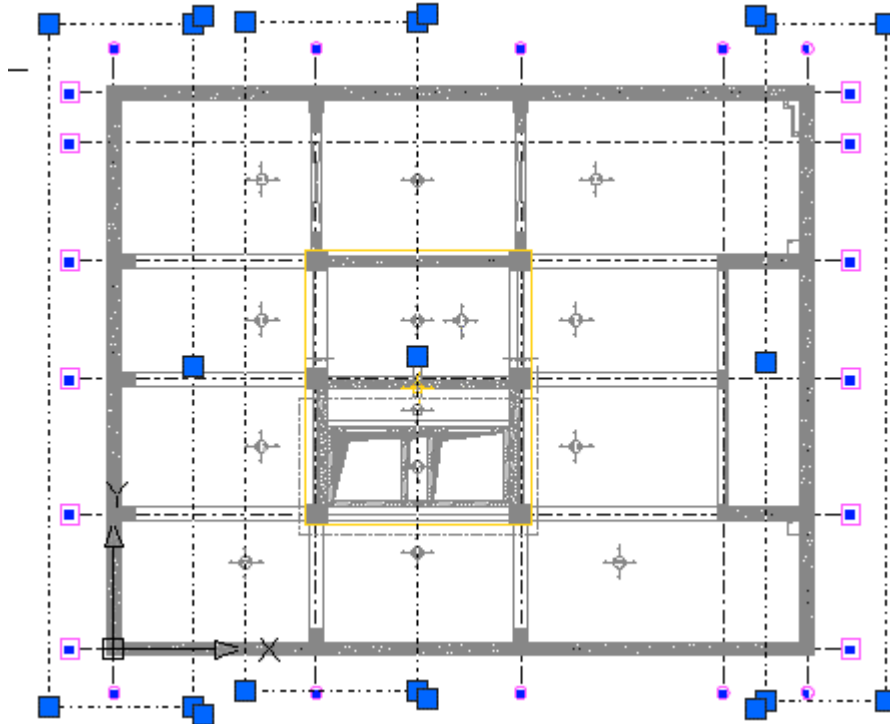


Figure 224: Cuts

 **Note:** The sections are numbered automatically: section A-A, section B-B, section C-C, etc. When a section is deleted, the gaps in numbering are automatically filled by the new drawings.

Step 2: Create a cut with parallel broken plane

1. On the **Model** toolbar, **Drawings** flyout, click .
2. Draw the cutting plane as you draw an AutoCAD® polyline.
3. Press **Enter** to finish.

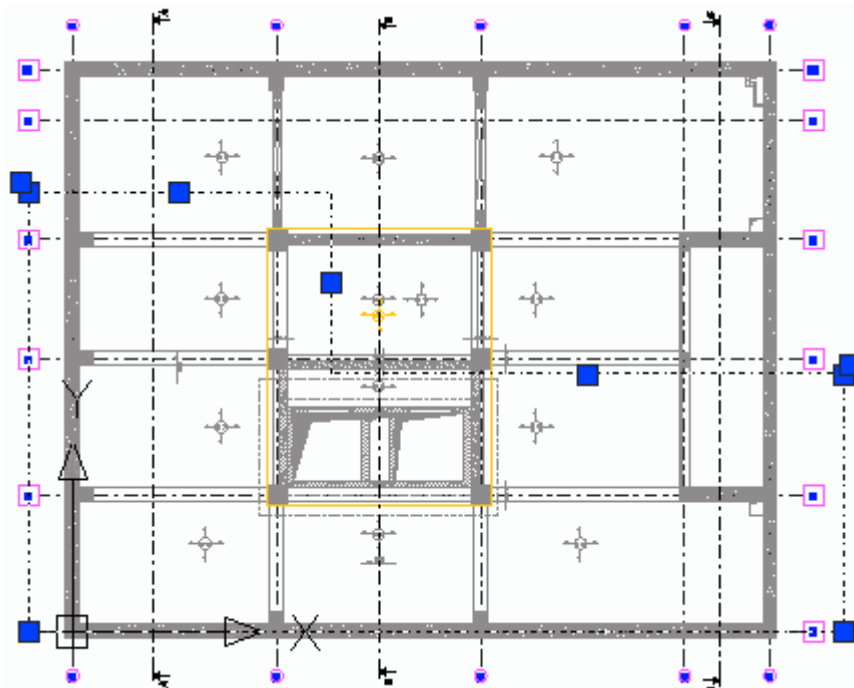


Figure 225: Created cut

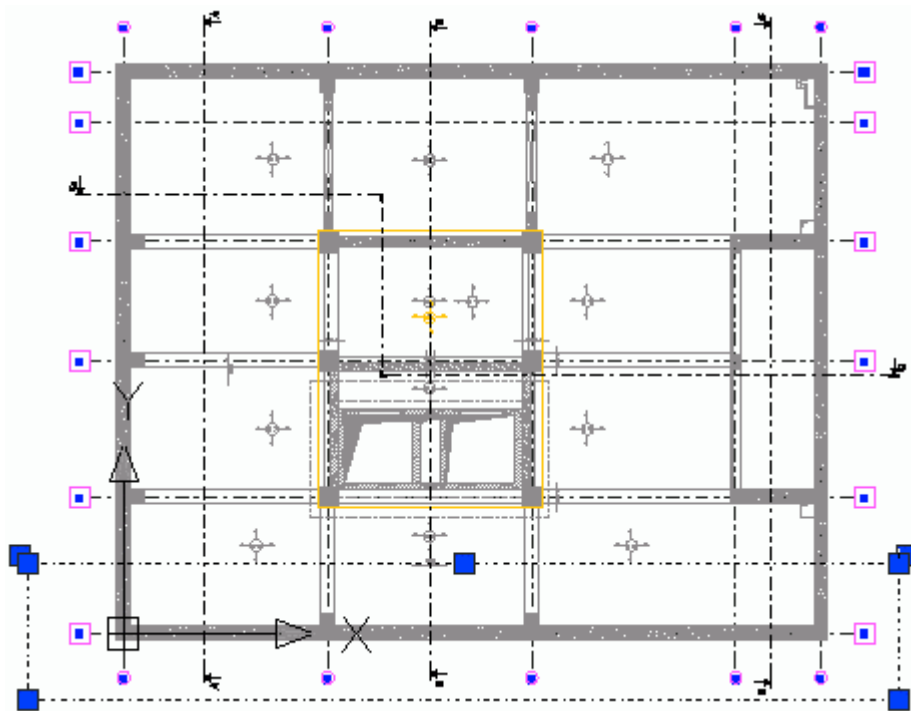


Figure 226: All created cuts

Step 3: Create an elevation

1. On the **Model** toolbar, **Drawings** flyout, click .
2. Input the definition plane, point by point, as you input a polyline in AutoCAD®.
3. Click a point on the bottom side of the building and move the cursor in the X direction.

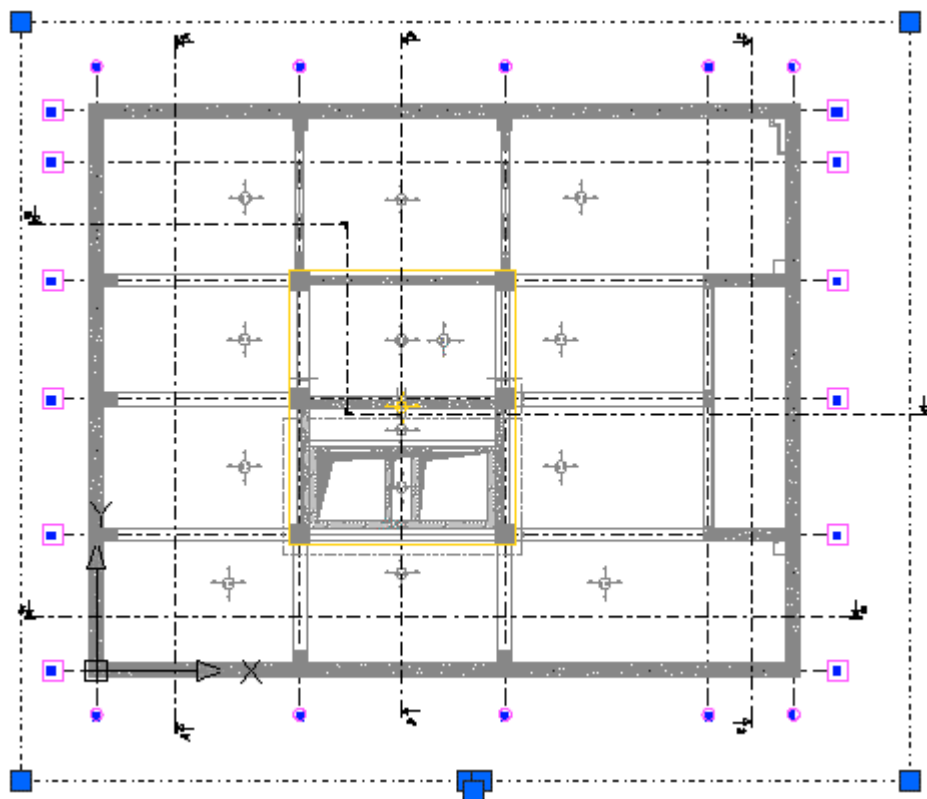


Figure 227: Creating the elevation

4. Press **Enter** to confirm.
 5. Click a point on your drawing to input the back cutoff plane.
- The elevation is created.

Step 4: Create a top view

This step describes how to create a top view using the top/bottom view creation wizard.

1. On the **Model** toolbar, **Drawings** flyout, click . The top/view creation wizard starts.
2. In the first dialog box, set the view definition parameters.
 - In the "View definition" area, select the **Bottom view from top to bottom** view direction.
 - Define the cut plane height relative to the **Underground level 2** level (the level on which the view must be calculated): select the first radio button and select **Underground level 2** from the drop-down list.

Input the upper and lower limits (according to the view direction).

- H2: **1 m**
- H1: **1.2 m**

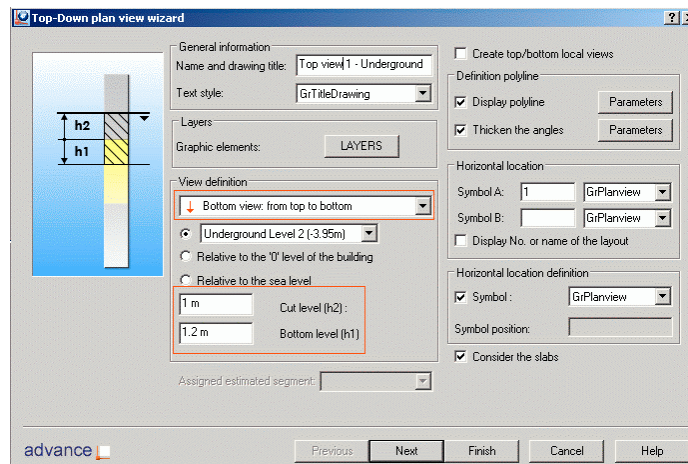


Figure 228: "Top-Down plan view wizard" dialog box – setting the view parameters

3. Click **Next** to set the visible objects representation.
 - In the "Definition of the cut visible edges" area, select the **Uniform edges** option and from the **Line thickness** down list, select a thickness of **0.50 mm**.

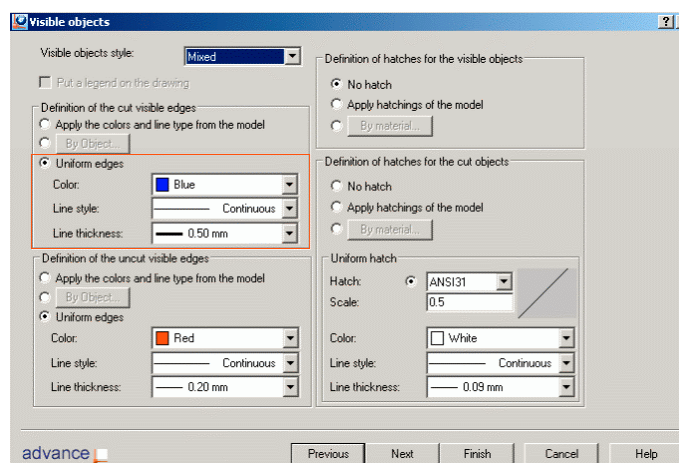


Figure 229: Setting the line thickness for visible edges

- In the "Definition of the uncut visible edges" area, select **Uniform edges** to display all visible edges using the same style. Next, select the color (red), the line style and line thickness.

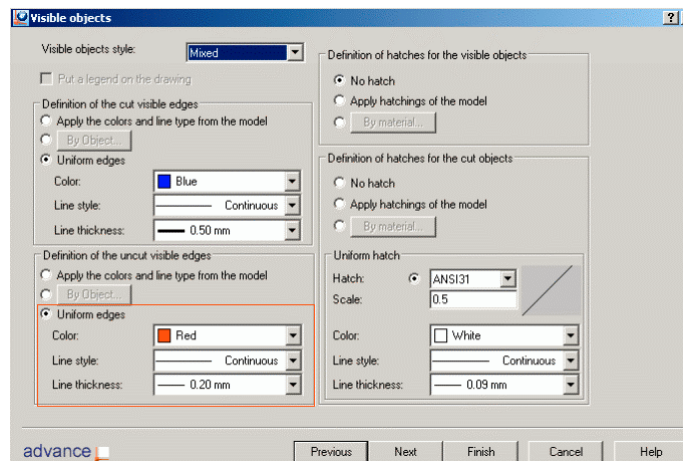


Figure 230: Setting the color for the uncut edges

- In the "Uniform hatch" area, select **Hatch** to hatch the visible objects. Next, set the hatch parameters.
- Scale: **0.35**
- Color: **White**
- Line thickness: **0.13 mm**

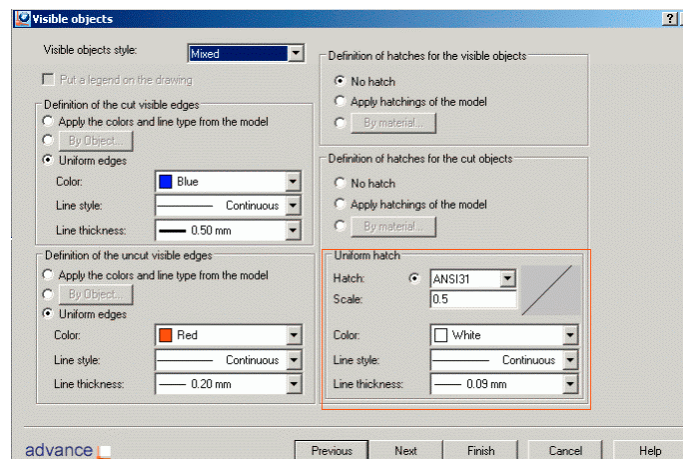


Figure 231: Setting the hatch parameters

4. Click **Next** to set the **Hidden objects** representation.

- In the "Definition of hidden objects" area, select **Uniform edges** and set the parameters as shown in Figure 232.
- In the "Definition of hatches for the hidden objects" area, select **No hatch**.

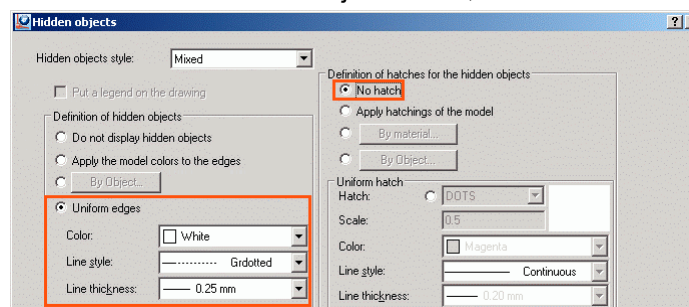


Figure 232: Setting the hatch of hidden objects

5. Click **Next** to modify the finishing representation.

In the "Inside section edges" area:

- From the **Line thickness** drop-down list select a thickness of **0.25 mm**.

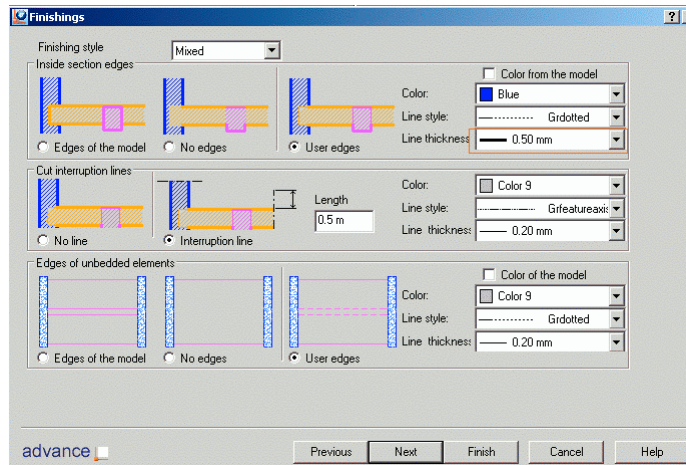


Figure 233: Setting the section edges

6. Click **Next** until you reach the "Visual elements" dialog box to modify the visual elements.

7. In the "Visual elements" dialog box, select **Display formwork symbols** to display symbols on the drawing.

8. Click **Parameters**.

9. In the "Multiple symbols" dialog box, select the set of objects for which to add symbols: select only **Beams** and **Columns**.

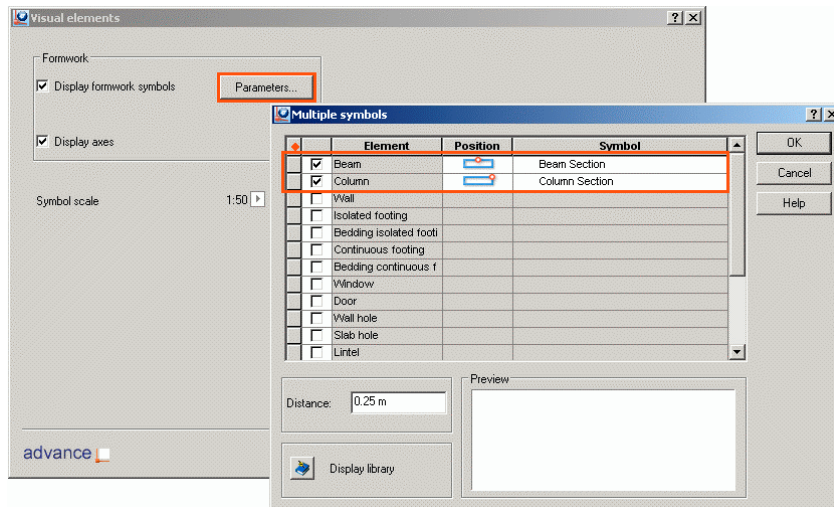


Figure 234: "Multiple symbols" dialog box

10. Set the symbol relative position according to the object for every object type:

- For beams:
- For columns:

The preview area displays the selected attributes: the beam section.

11. Click **OK**.

12. Click **Finish**.

The top view is created. The Pilot changes automatically to **Drawings** mode.

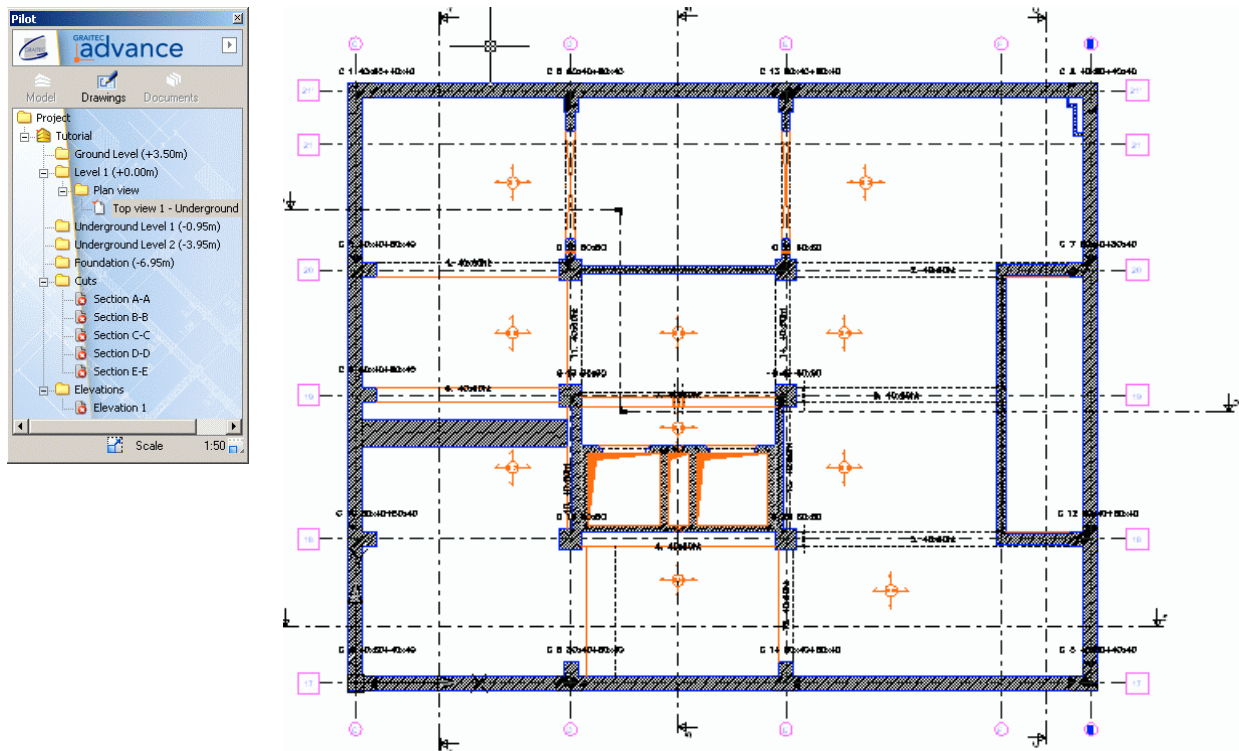


Figure 235: Top view

Step 5: Create a local top/bottom view

In the Pilot, in **Model** mode, right click the levels and enable them.

Display all levels and the model in 3D.

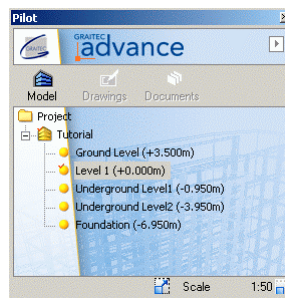


Figure 236: The Pilot – enabled levels

1. On the **Model** toolbar, **Drawings** flyout, click .
2. In the "Top-Down plan view wizard" dialog box, select **Create top/bottom local views**.

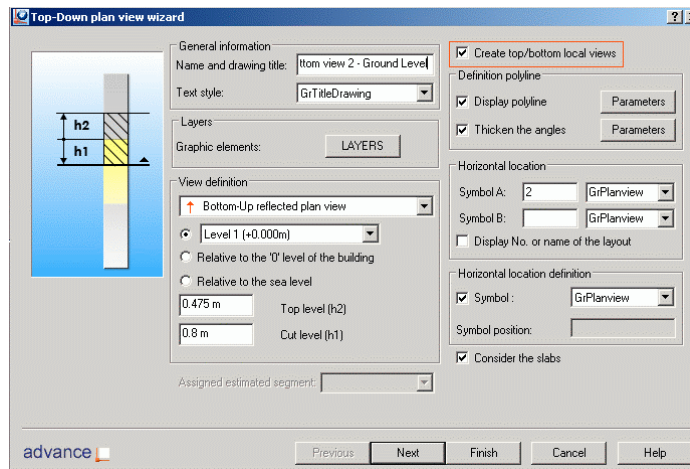


Figure 237: "Top-Down plan view wizard" dialog box – **Create top/bottom view** option

3. Click **Finish**.
4. Draw the local view definition polyline by three points and include the elevator walls as in Figure 238.

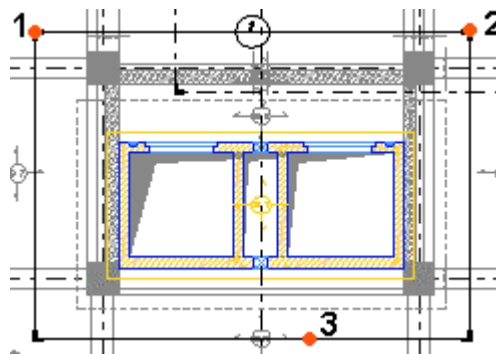


Figure 238: Local view definition

5. Right click the polyline and select **Activate and update structure** from the menu.

The top view is calculated and **Drawings** mode is activated.

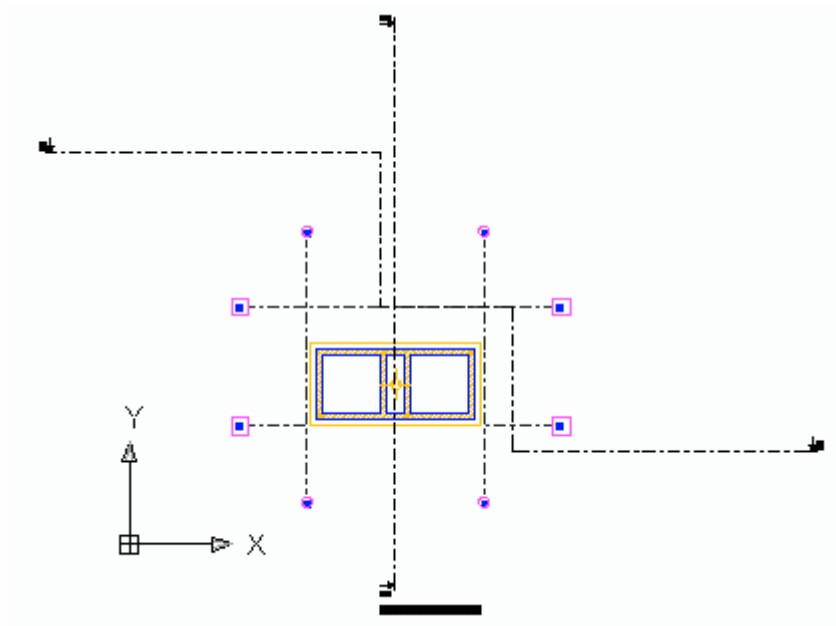


Figure 239: Local top/bottom view

In the Pilot, in **Model** mode, right click **Tutorial** and enable all the levels.



Use the AutoCAD® **View** toolbar to view the model in 3D.

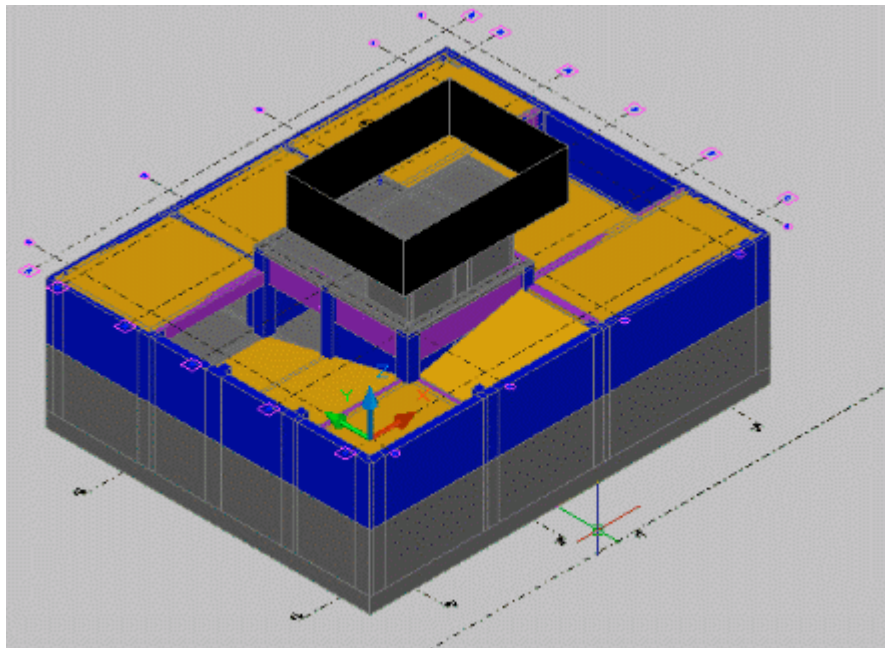


Figure 240: Local top/bottom view

Step 6: Create an isometric view

To create an isometric view, use the **View** toolbar for a 3D view of the model.



Figure 241: AutoCAD® **View** toolbar

1. On the **Model** toolbar, **Drawings** flyout, click .

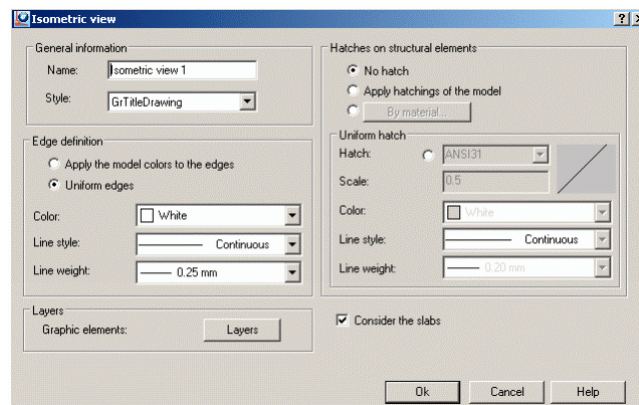


Figure 242: "Isometric view" dialog box

2. Click **OK** to validate.

Advance switches automatically to the **Drawings** mode. The isometric view is displayed.

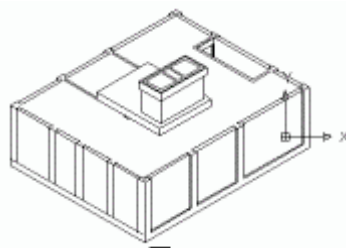


Figure 243: Isometric view of the building

 **Note:** All the created isometric views are grouped, by default, in the **Isometrics** folder.

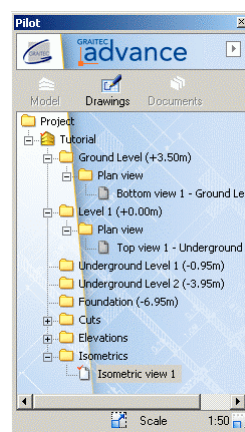


Figure 244: Pilot display of Isometric

Modifying the isometric properties

Next, you will remove the slabs from the isometric view.

1. In the Pilot, in **Drawings** mode, right click **Isometric view 1** and select **Properties** from the context menu.

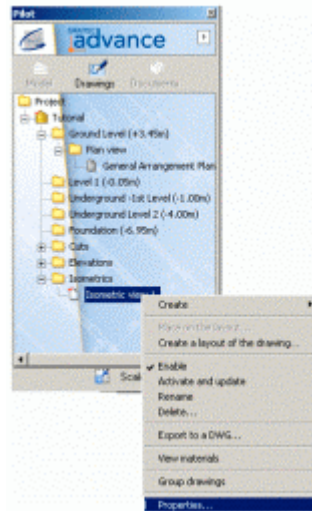


Figure 245: Accessing the Isometric View properties

2. In the properties dialog box, unselect the **Consider the slabs** option to hide the slabs from the isometric view.

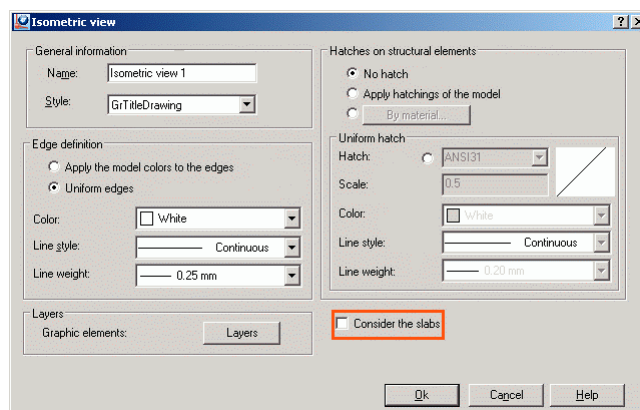


Figure 246: Consider the slabs option

3. Click **OK** to validate.

Note: If you wish to view the result of the modification, recalculate the drawing.

Step 7: Modify and update drawings

Update the isometric view.

In the Pilot, in **Drawings** mode, right click **Isometric view 1** > **Activate and update** to view the result.

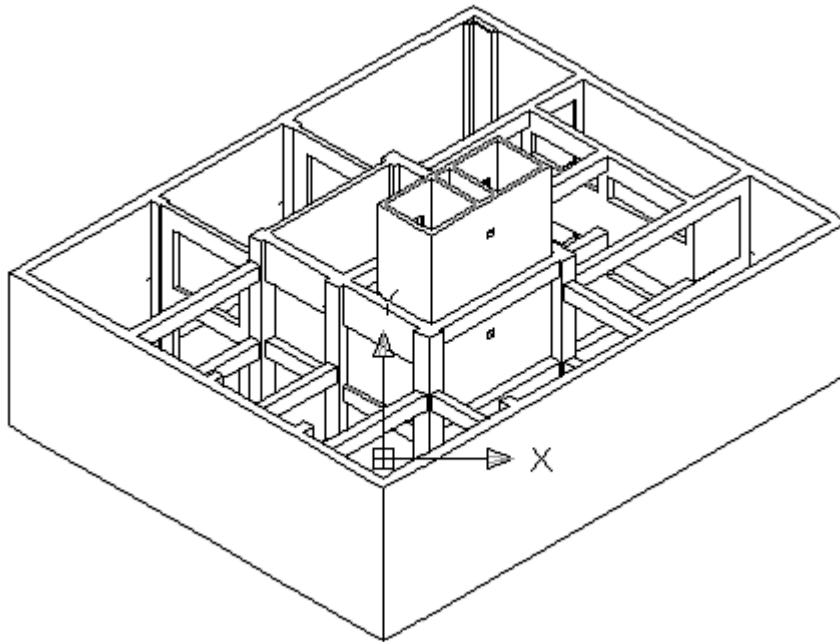


Figure 247: Updated Isometric view – Without slabs

Using the same process, update the cuts.

Example:

In the Pilot, in **Drawings** mode, right click **Section A-A** and select **Activate and update** from the context menu.

The **Section A-A** cut is displayed.

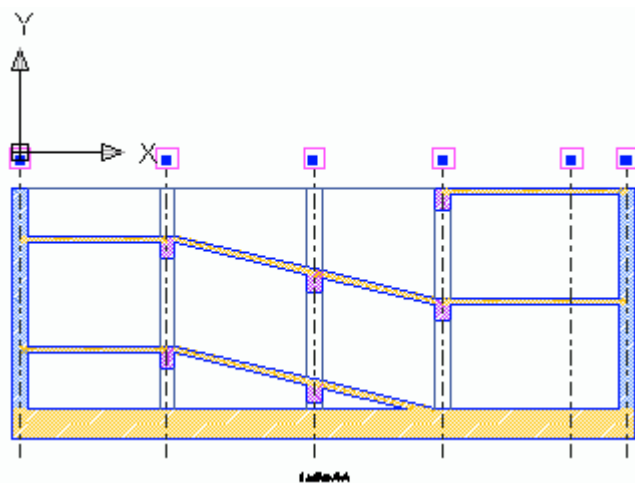


Figure 248: Section A-A cut

Modifying the drawing properties

1. In the Pilot, in **Drawings** mode, right click **Section A-A** and select **Properties** from the context menu.
2. In the properties dialog box, select the **Visible Objects** tab and modify the cut edge line thickness.
 - In the **Definition of the cut visible edges** area, from the line thickness drop-down list select **0.50 mm**.

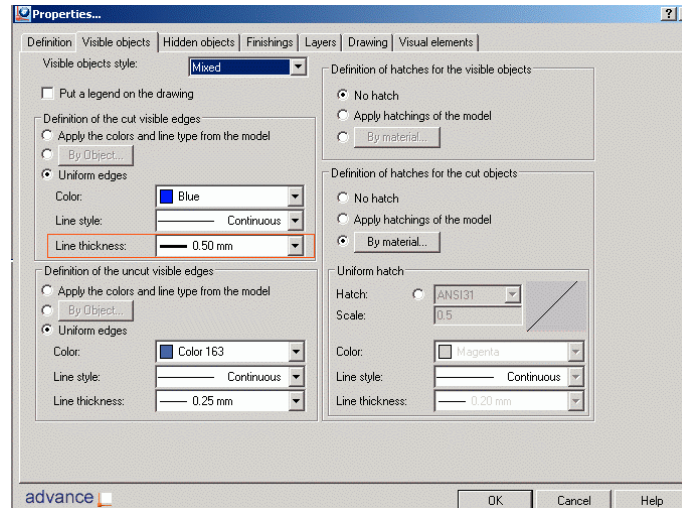


Figure 249: "Properties" dialog box – **Visible objects** tab

3. Click **OK**.

The cuts that have modifications are marked with .

Next, update the drawing to see the result.

In the Pilot, in **Drawings** mode, right click **Section A-A** and select **Activate and update**.

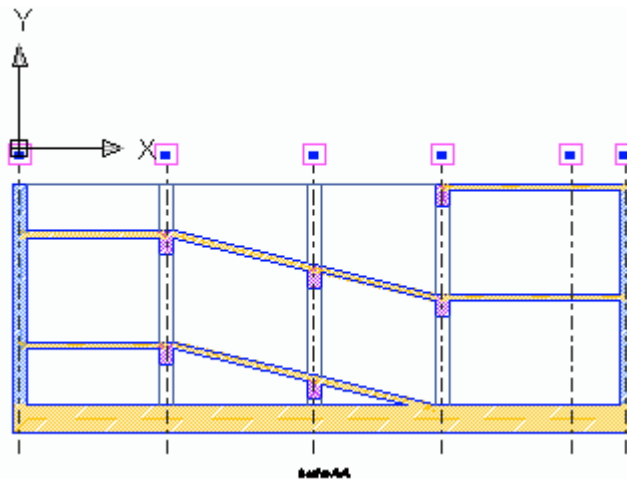


Figure 250: Section A-A – updated

Updating several drawings

1. On the **Model** toolbar, **Drawings** flyout click

A dialog box displays the list of all created drawings.

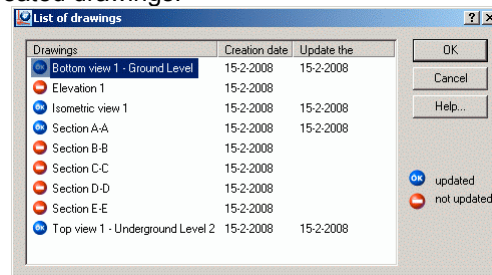


Figure 251: List of drawings table

The updated elements are marked with and the elements that are not up-to-date are marked with .

2. Select all the sections that are not updated.

Note: For a multiple selection, use the **CTRL** key.

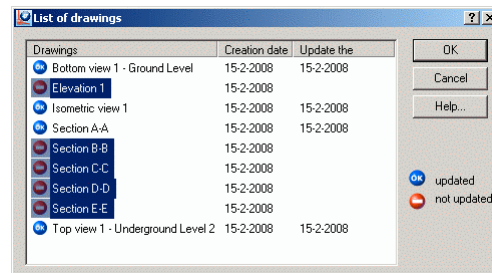


Figure 252: Selection of the cuts

3. Click **OK**.

The sections are updated.

Step 8: Group and ungroup the section cuts

In this step, you will group all created sections.

1. In the Pilot, in **Drawings** mode, right click **Section A-A** and select **Group drawings** from the context menu.
2. Select all the sections and click **OK**. (Use SHIFT and UP or DOWN arrow for a multiple selection.)

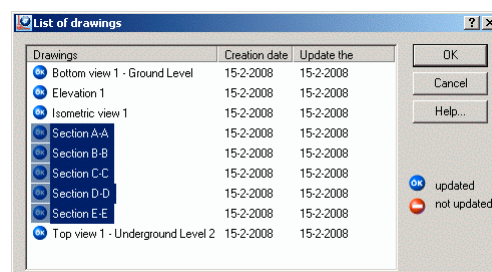


Figure 253: Selection of cuts

3. On the command line, type **r** (rows) and press **Enter** to group the drawings in a row.

The drawing frames are linked to the cursor.

4. Set the alignment point in the origin and press **Enter**.

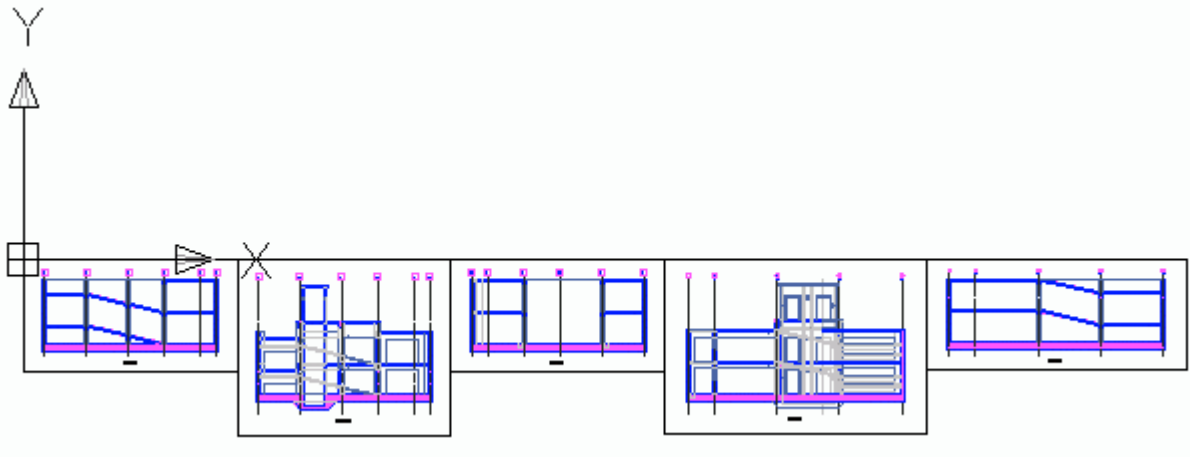


Figure 254: Grouped sections

Ungrouping the drawings

1. In the Pilot, in **Drawings** mode, right click the drawing group and select **Ungroup drawings** from the context menu.

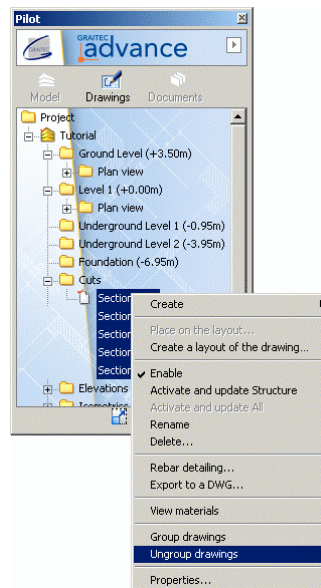


Figure 255: Selection of the cuts

2. In the "List of drawings" dialog box, select **Section E-E** and click **OK**.

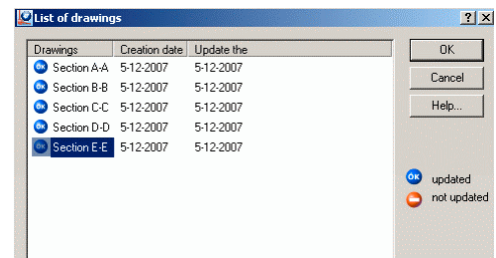


Figure 256: Drawing selection

The **Section E-E** is separated from the drawings group.

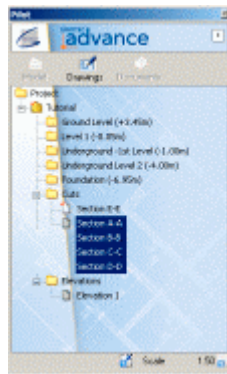


Figure 257: Ungrouped section cut in the Pilot

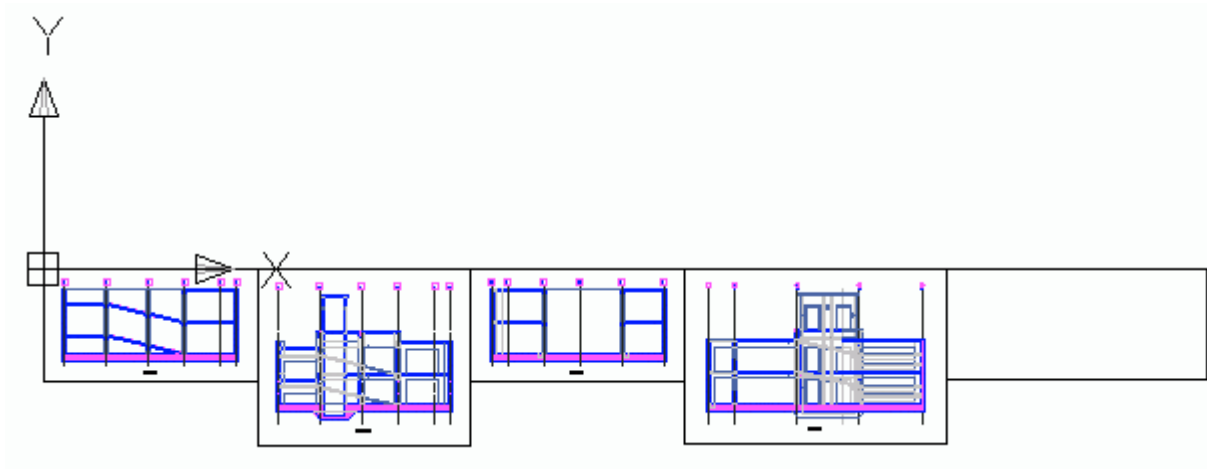


Figure 258: Ungrouped sections

Step 9: Create a reinforcement view

In this step, you will create a reinforcement drawing with 3 views (i.e., a cut, a top view and an elevation) for the wall along the **21'** axis.

 **Note:** This command is available only in **Model** mode.

Switch to **Model** mode:

1. In the Pilot, in **Model** mode, right click **Underground Level 1** and select **Enable** from the context menu.

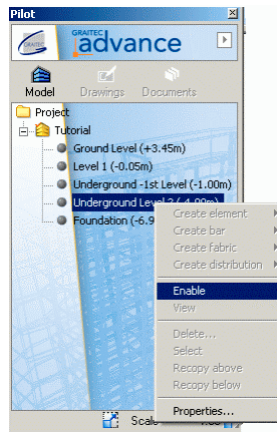


Figure 259: Activating **Underground Level 1**

2. Select the wall along the **21'** axis.

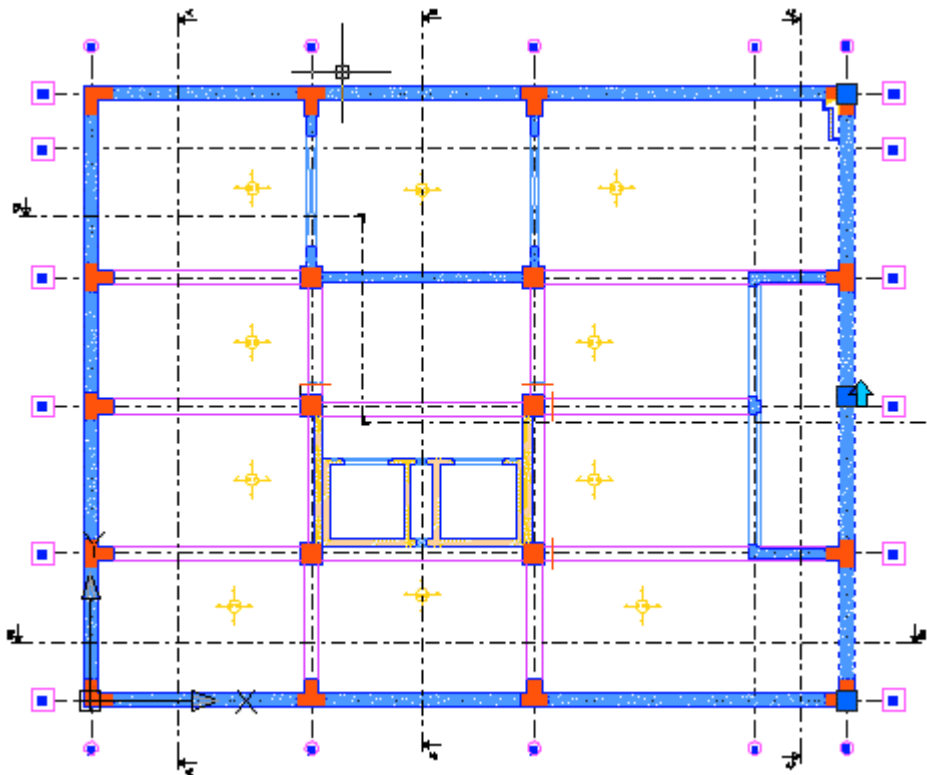


Figure 260: Wall along the **21'** axis

3. On the **Model** toolbar, **Drawings** flyout, click .
4. In the "Creation of a reinforcement drawing" dialog box, from the **3 Views** branch, select the **Cut left, top view and cut face**.

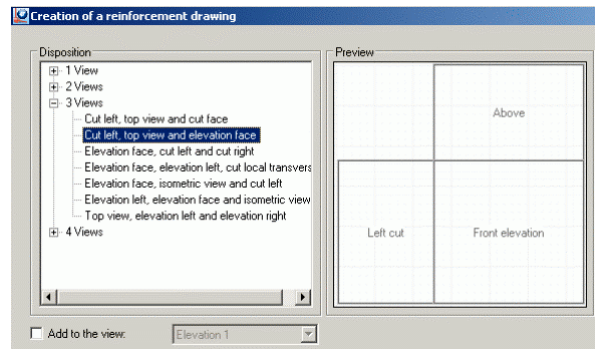


Figure 261: "Creation of reinforcement drawing" dialog box

5. Click **Next**. The "View parameters" dialog box appears.
6. Click **Next**.
7. In the "Visible objects" dialog box, define the edge style for the visible objects.
 - In the "Definition of the cut visible edges" area, select a line thickness of **0.50 mm**.

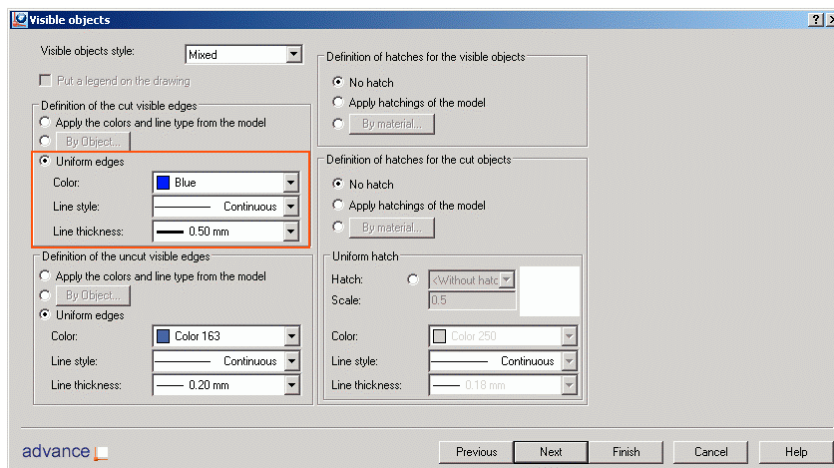


Figure 262: Setting the line thickness

8. Click **Next**.
9. In the "Hidden Objects" dialog box, define the edge style for the hidden objects.
 - In the "Definition of hidden objects" area, set the line thickness to **0.25 mm**.

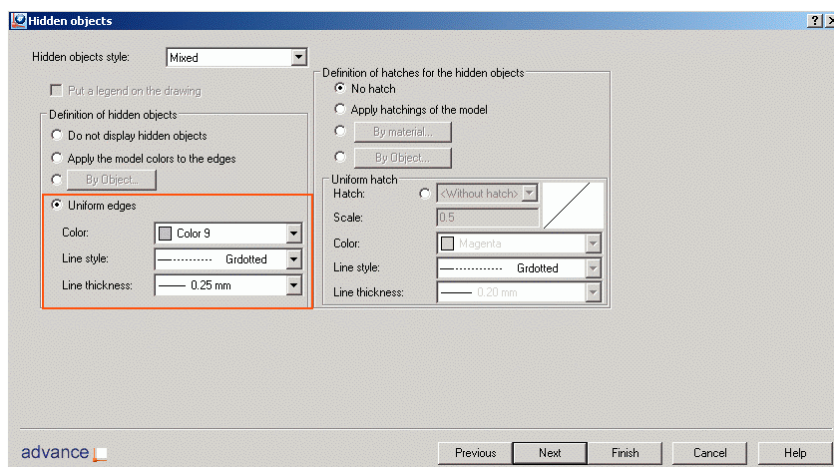


Figure 263: "Hidden Objects" dialog box

9. Click **Next** in every dialog box (i.e., "Finishing", "Drawing", and "Visual elements") until the "Title" dialog box.
10. In the "Title" dialog box, select the information to display above each view of the reinforcement drawing.

- Check **Element names** to display the wall name in the view title.
- Check **Place level name in the title box** to display the level name.
- Uncheck **Quantities**.

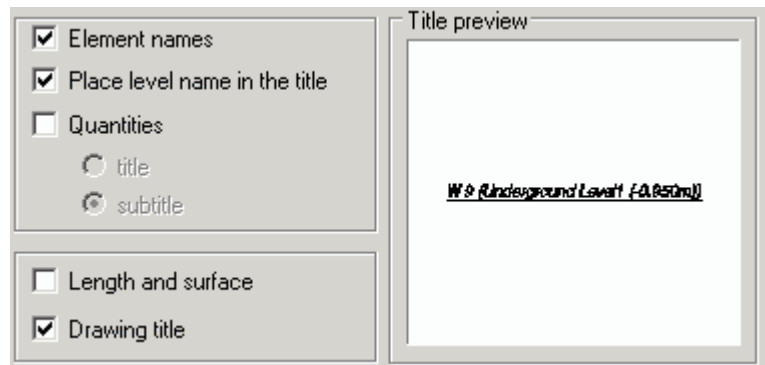


Figure 264: Setting the information to display of the reinforcement drawing

11. Click **Finish**.

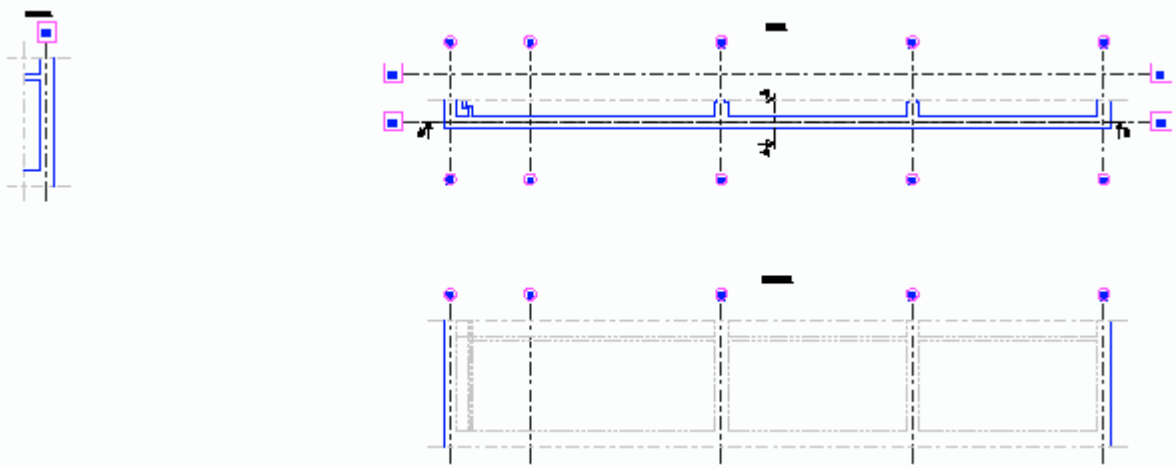


Figure 265: Resulting reinforcement drawing

Step 10: Create a new empty drawing

In the Pilot, in **Model** mode, right click **Tutorial** and activate all the levels.

In the Pilot, in **Model** mode, right click **Tutorial** and select **New drawing** from the context menu.

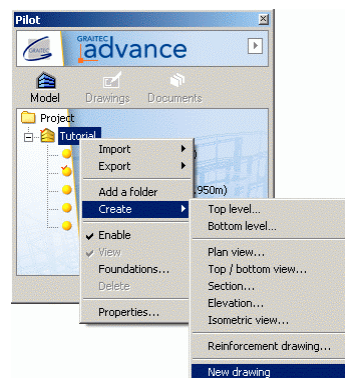


Figure 266: Creating a new drawing

The new drawing is displayed in **Drawings** mode.

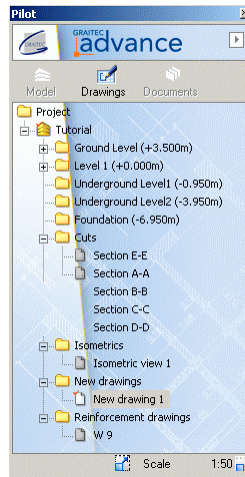


Figure 267: Pilot – New drawing

 **Note:** All created empty drawings are grouped, by default, in the **New drawings** folder.

Access the drawing properties

1. In the Pilot, in **Drawings** mode, right click **New drawing** and select **Properties** from the context menu to access the drawing properties.

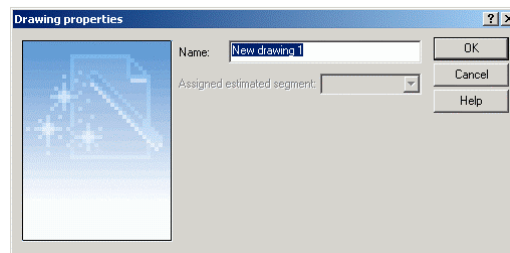


Figure 268: “Drawing properties” dialog box

2. Click **OK**.

Step 11: Create a plan view

In this step, you will create a plan view for **Level 1**.

Concepts

The plan view is a special case of views: it is a model representation and not a generated view. Therefore, the plan view is updated in real time.

1. In the Pilot, in **Model** mode, right click **Underground Level 1** and select **Enable** from the context menu.
2. On the **Model** toolbar, **Drawings** flyout, click .
 - In the “Select a level” dialog box, select **Level 1**.

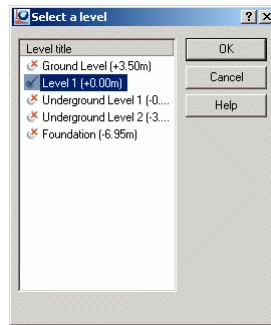


Figure 269: “Select a level” dialog box

3. Click **OK**.

The plan view is automatically created and Advance activates the **Drawings** mode. The plan view is displayed on the screen.

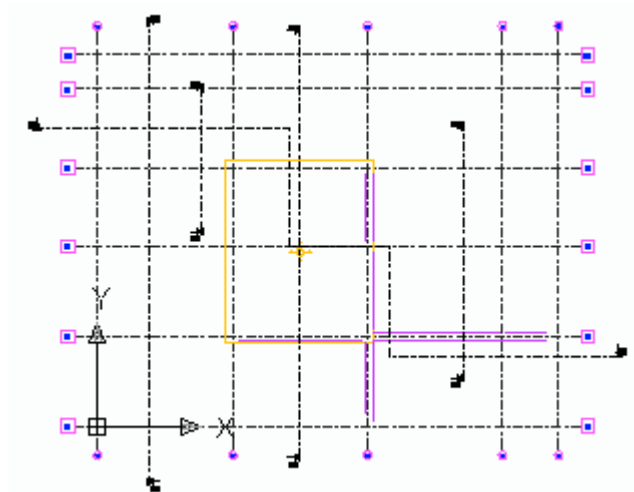


Figure 270: Plan view

Modifying the plan view properties

Next, display the elements that are in contact with the top slab.

1. In the Pilot, in **Drawings** mode, right click **Floor plan** and select **Properties** from the context menu.

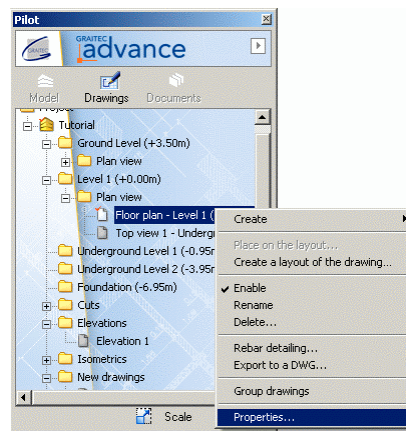


Figure 271: Accessing the Plan view properties

2. In the "Floor plan properties" dialog box, select the **Impact on upper level** option to display the elements from the level immediately above (i.e., the **Ground level**) that are in contact with the top slab.

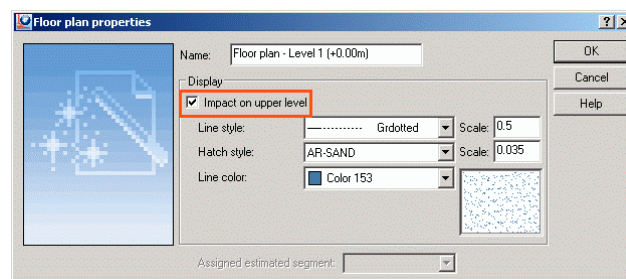


Figure 272: "Floor plan" dialog box

The elevator walls are displayed in the plan view.

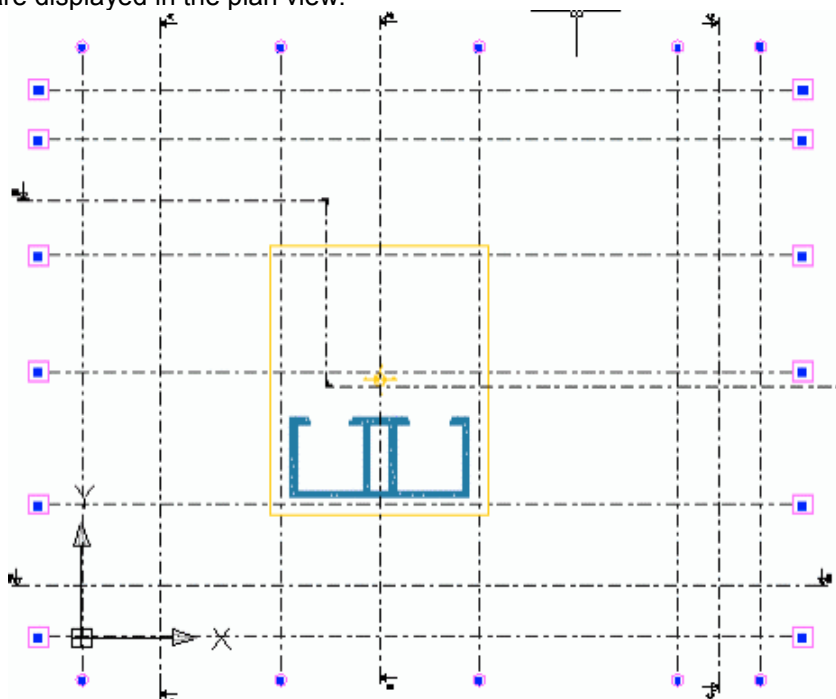


Figure 273: Plan view – Impact on upper level



Use the AutoCAD® **View** toolbar and **Visual Styles** toolbar to view the drawing in 3D.



Figure 274: AutoCAD® **View** toolbar



Figure 275: AutoCAD® **Visual Styles** toolbar

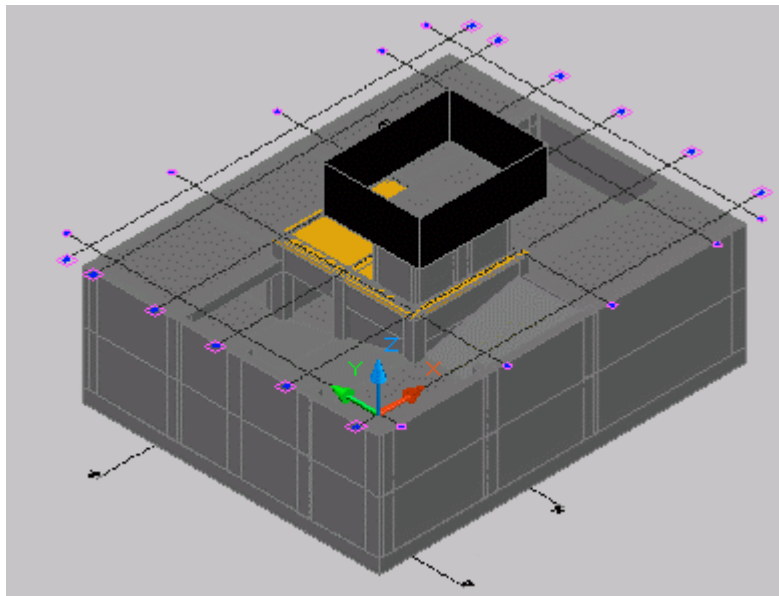


Figure 276: Building in 3D view

Lesson 12: Creating dimensions

In this lesson, you will dimension a plan view of the **Underground level**.

You will learn how to:

- Create an intersection dimension.
- Add or remove intersected elements.
- Create an associative dimension.
- Create a dimension by points.
- Create a symbol.

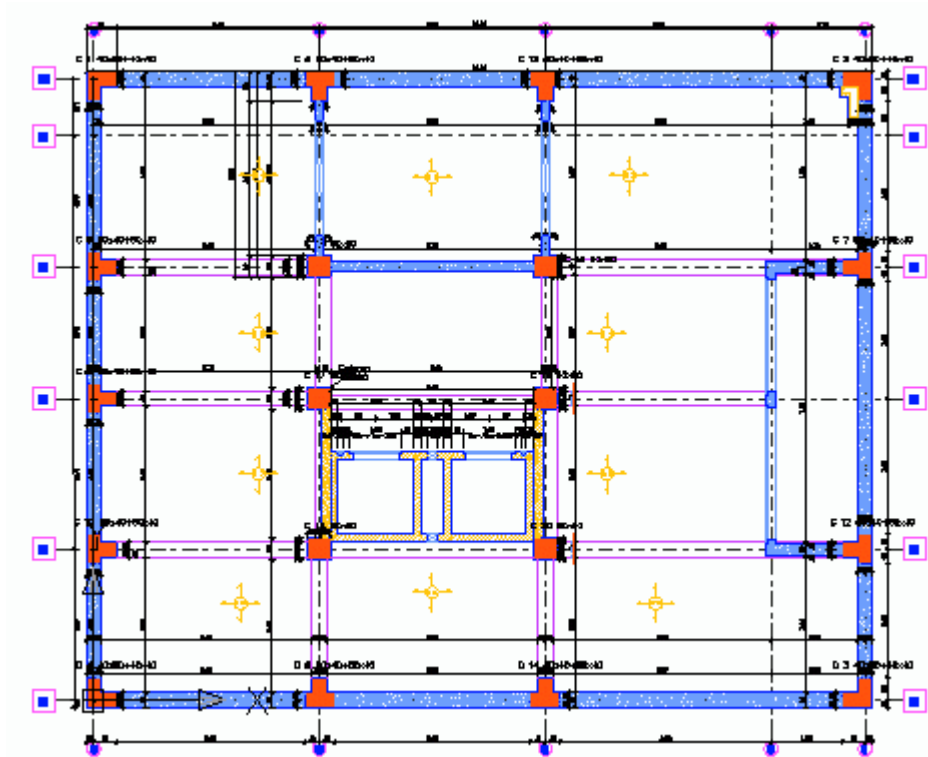


Figure 277: Dimensions on the plan view

Step 1: Create a plan view

In this step, you will create a plan view for the first **Underground level**.

1. In the Pilot, in **Model** mode, right click **Tutorial** and select **Create > Create a plan view** from the context menu.

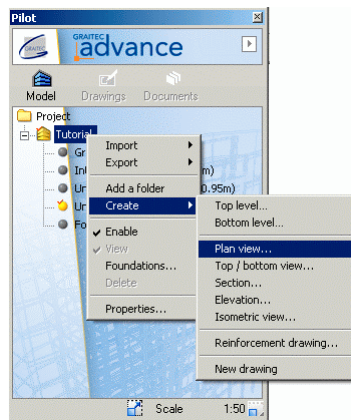


Figure 278: Pilot - Create a plan view

The “Select a level” dialog box lists all the building levels.

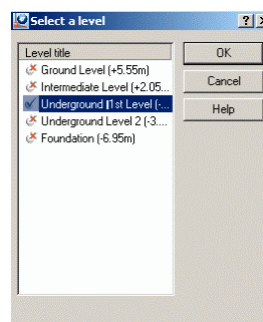


Figure 279: “Select a level” dialog box

2. Select the level for the dimensions, for example, **1st Underground Level**.
3. Click **OK**.

The plan view is created.

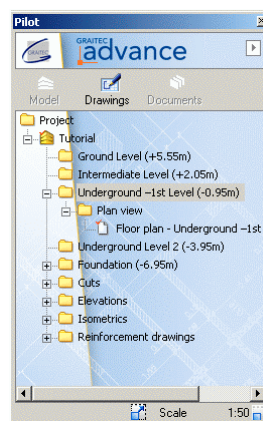


Figure 280: Pilot – Plan view

Step 2: Create intersection dimensions

In this step, you will dimension the elements along the **C** axis (see the red line in Figure 281).

1. On the **Views** toolbar, click .
2. Click two points to define the dimension line as shown in Figure 281.

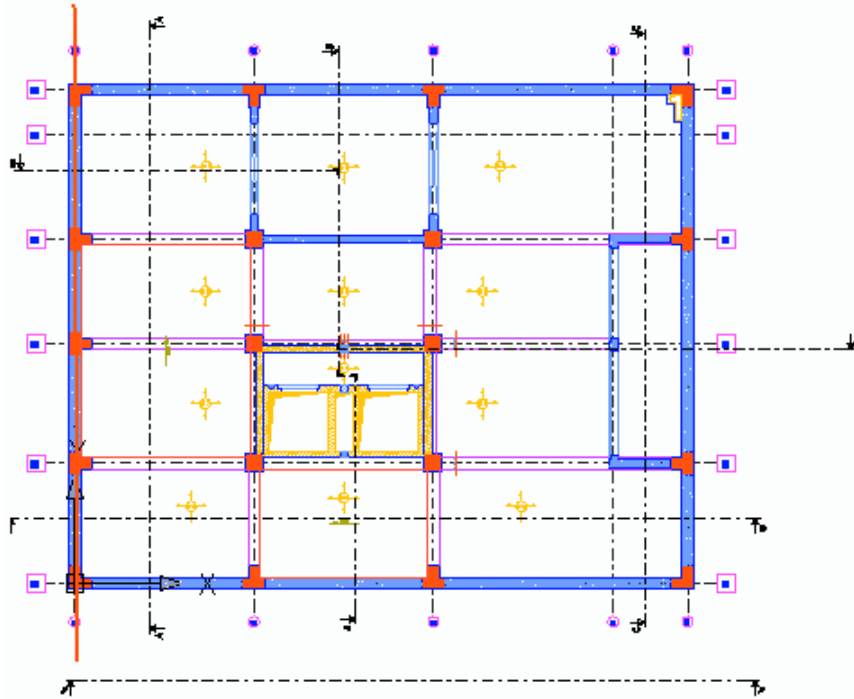


Figure 281: Create the first intersection dimension

3. Press **Enter**.
4. Click a point on the drawing to set the intersection dimension position.

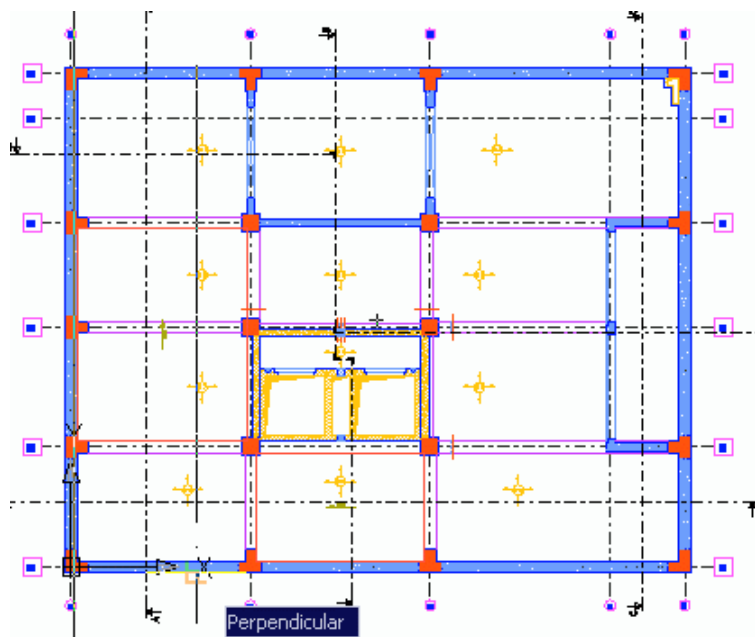


Figure 282: Placing the intersection dimension

The dimension line is automatically created.

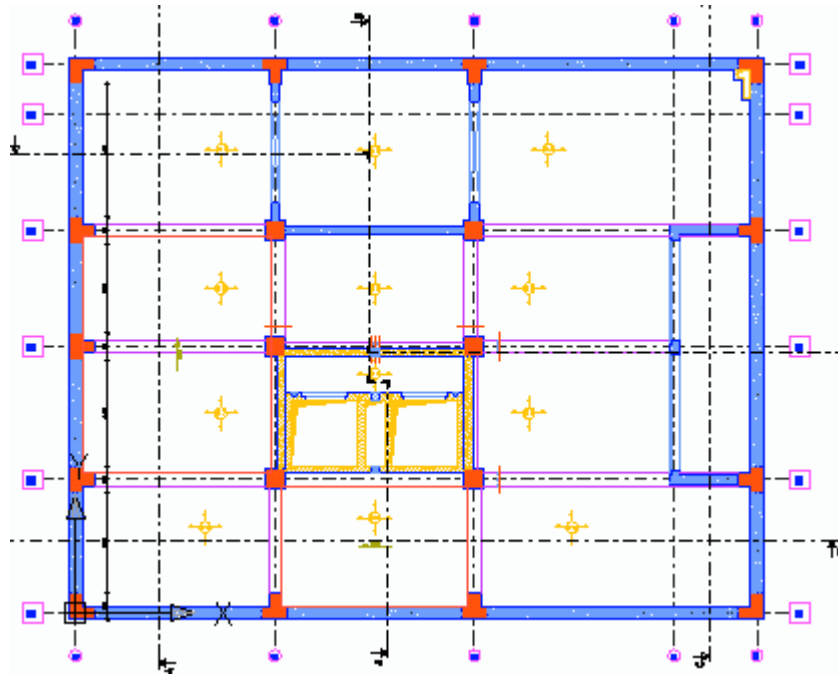


Figure 283: Dimension line

Use the same process to dimension the elements along the **D**, **E**, **F'** axes.

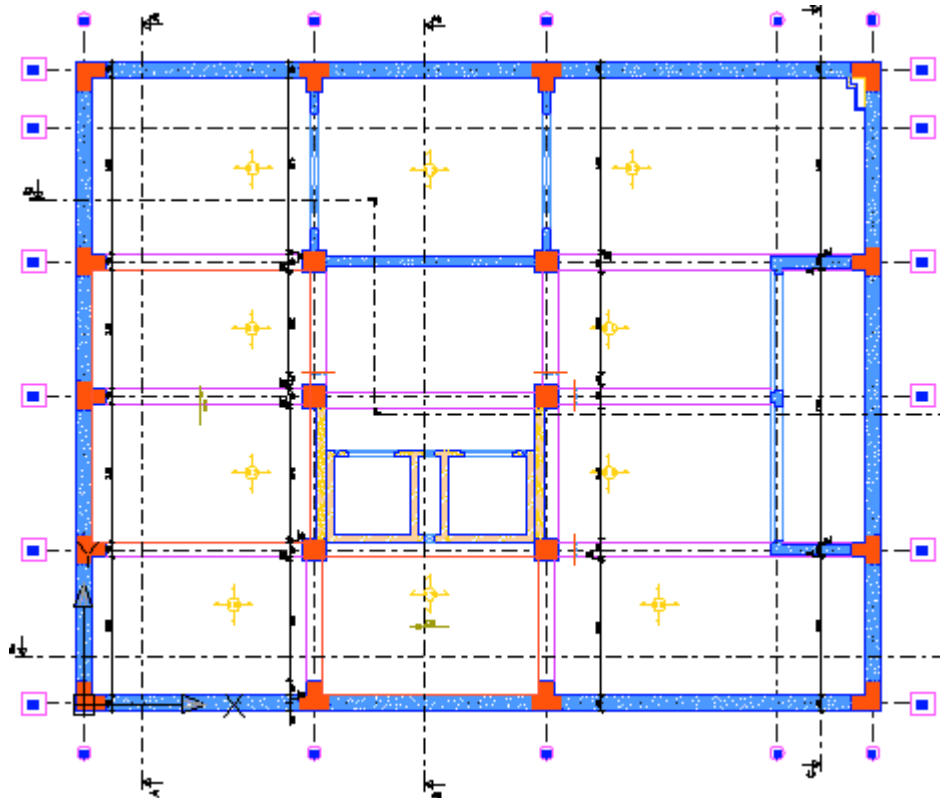


Figure 284: Locations of intersection dimensions

Using the same process, create intersection dimensions as shown in Figure 285.

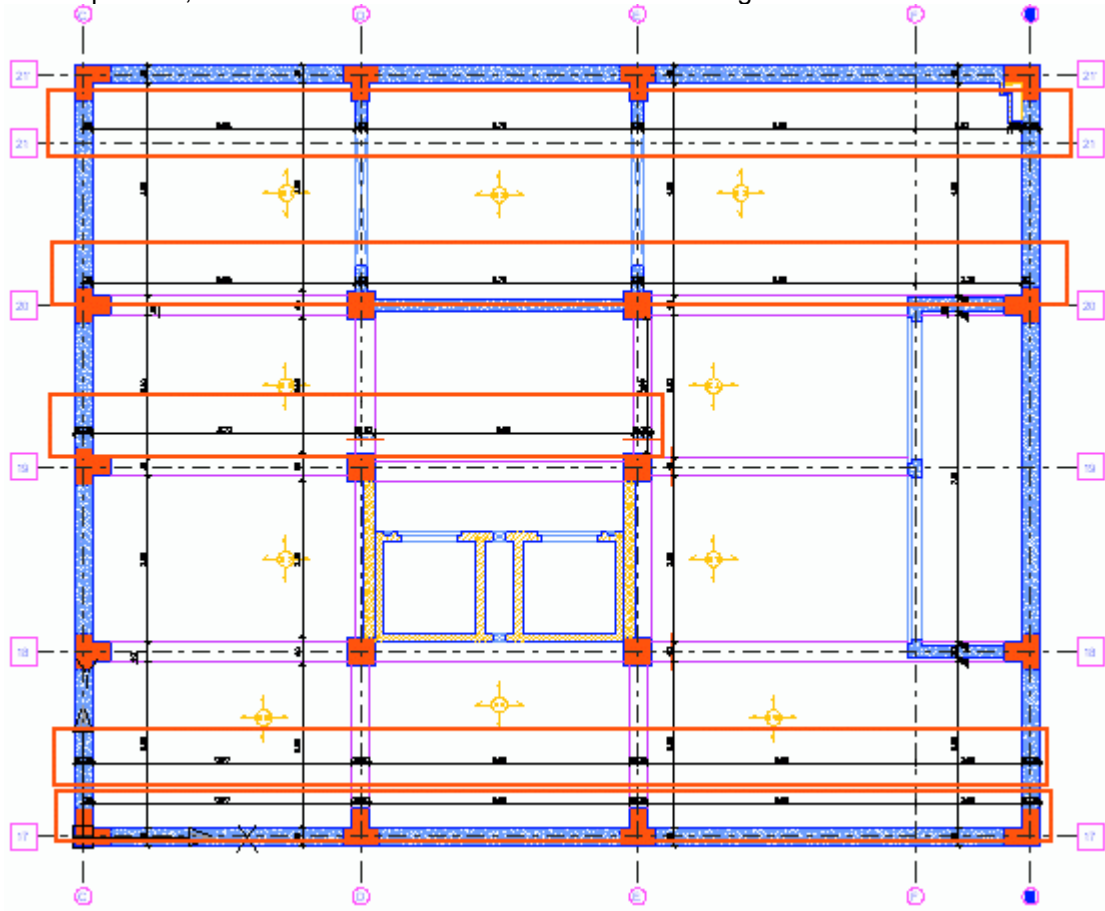


Figure 285: Intersection dimensions

Taking the grid axis into account

Next, dimension the columns on the **C** axis, taking the grid into account. Zoom to the **C20** column.

1. On the **Views** toolbar, click .
2. On the Smartbar, click  to access the properties.
3. In the “Dimension properties” dialog box, select the intersecting elements.
 - Expand the **Model elements** tree branch.
 - Click **Model elements** to unselect all elements.
 - Select **Grid**.

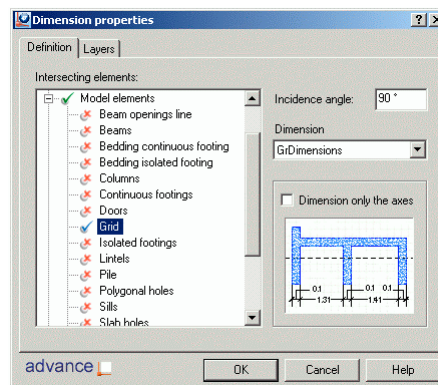


Figure 286: “Dimension properties” dialog box - Selecting the grid

Note: The elements taken into account are preceded by a blue sign ✓ while those not taken into account are preceded by a red [X] ✗.

4. Click **OK** to validate.

Using the same process, create dimension lines next to the **C18**, **C19** and **C20** columns.

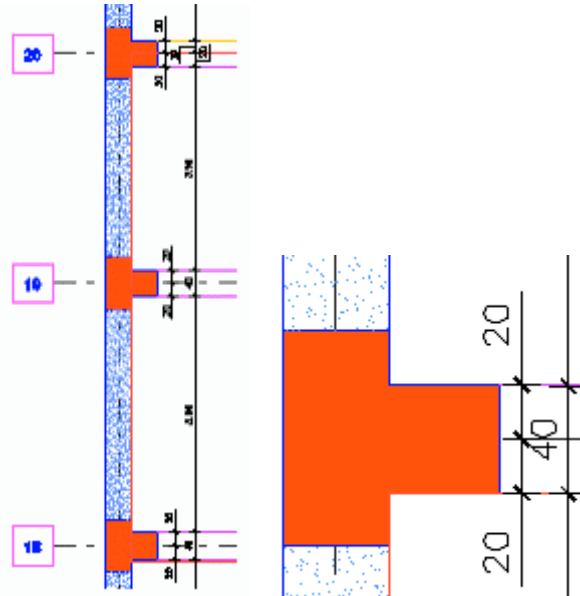


Figure 287: Intersection dimensions for the **C18**, **C19** and **C20** columns

Also, create dimension lines next to the **D18**, **D19** and **D20** columns.

Taking other elements into account

Next, dimension the area around the **D20** column, taking the beams and the columns into account.

1. On the **Views** toolbar, click
2. Position the dimension line next to the **D20** column.
3. On the Smartbar, click to access the “Dimension properties” dialog box.
4. On the **Definition** tab, expand the **Model elements** branch, and enable only the **Beams** and **Columns**.

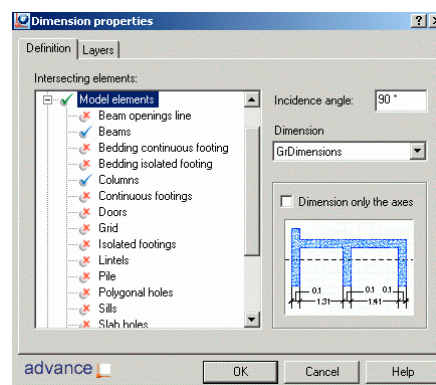


Figure 288: “Dimension properties” dialog box - **Definition** tab

5. Click **OK** to validate and close the dialog box.

6. Create dimension lines next to the **D20** column.

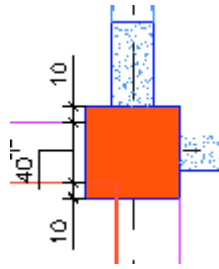


Figure 289: **D20** column – Dimension by intersection taking the beams and columns into account

Using the AutoCAD® dimension settings

Next, use the AutoCAD® dimension settings to always keep text between the exterior lines.

1. On the **Views** toolbar, click .
2. From the **Styles** list, select **GrDimensions**.

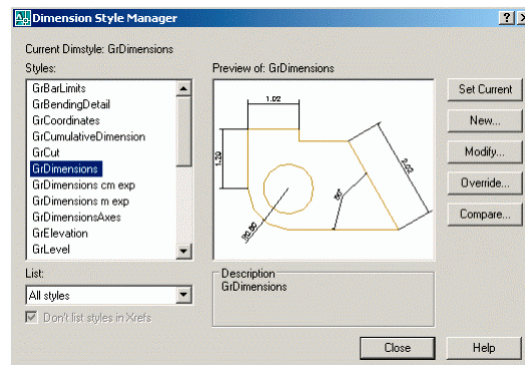


Figure 290: "Dimension Style Manager" dialog box

3. Click **Modify**.
4. In the "Modify dimension style" dialog box, select the **Fit** tab.
 - In the "Fit options" area, unselect the **Suppress lines if they don't fit inside** extension lines.
 - Check the **Always keep text between ext lines** option.

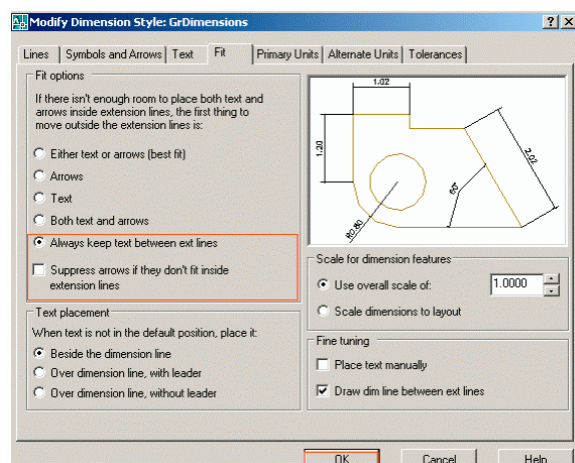


Figure 291: "Modify Dimension Styles" dialog box – **Fit** tab

5. Click **OK** to validate and return to the "Dimension Style Manager".
6. Click **Close** to finish.

Use the same process to dimension the **D19** and **D20** columns:

1. On the **Views** toolbar, click .
2. Create the dimension lines next to the **D19** column.

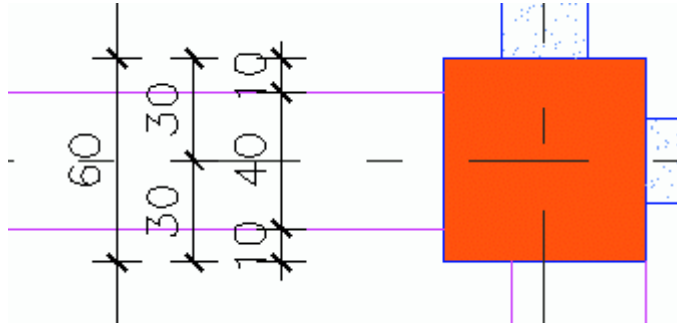


Figure 292: **D19** column – Text displayed between the extension lines

3. Press **Enter** to finish.

Using the same procedure, dimension all the columns in the drawing.

Step 3: Display decimals and millimeters in the exponent

In this step, you will display decimals and millimeters in the exponents.

1. In the **Pilot**, click  to enter the **Drawings** mode.
2. Right click **Project** and select **Project settings** to display the **Project Settings** toolbar.

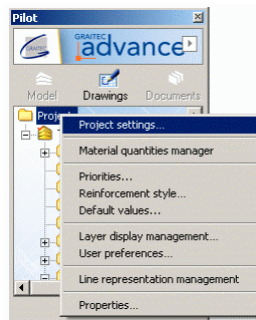


Figure 293: Pilot – accessing the **Project settings** toolbar

3. On the **Project Settings** toolbar, click  to access the project preferences.

4. Select the **Drawing and annotations** tab and make the following settings:

- In the “Dimensions” area, in the **number of decimals** field, enter **1** to set the number of decimals of the dimension.
- Check the **suppress trailing zeroes** option to not display the trailing zeros.
- Check the **mm in exponent** option to display millimeters in the exponent.

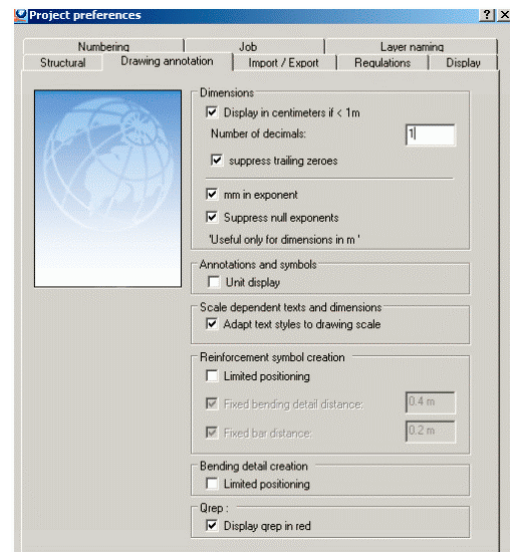


Figure 294: Drawing dimensions settings

5. Click **OK**.

Note: When the distances are higher than 1 m, they are expressed in meters and if they are lower than 1 meter, they are expressed in centimeters.

For example, create an intersection dimension at the bottom of the **E21'** column.

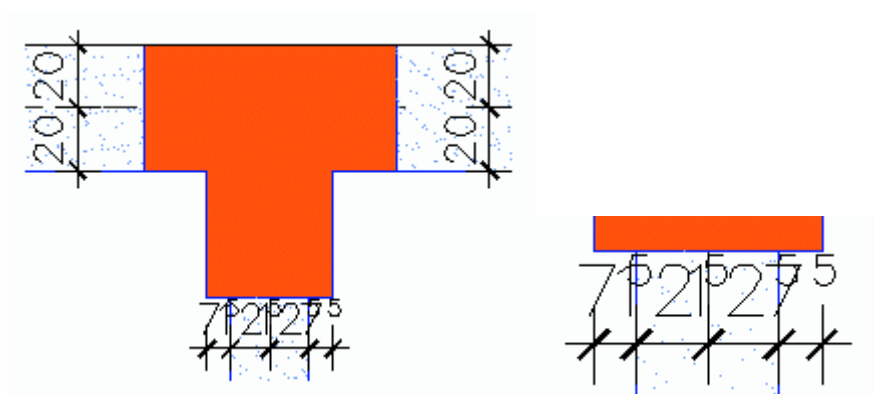


Figure 295: Dimensions with millimeters in the exponent

Step 4: Create an associative dimension

In this step, you will use the associative dimension to dimension the elevator walls

 **Note:** The creation of an associative dimension is possible only in Drawings mode.

1. On the **Views** toolbar, click .
2. Select the middle, left and right elevator walls.

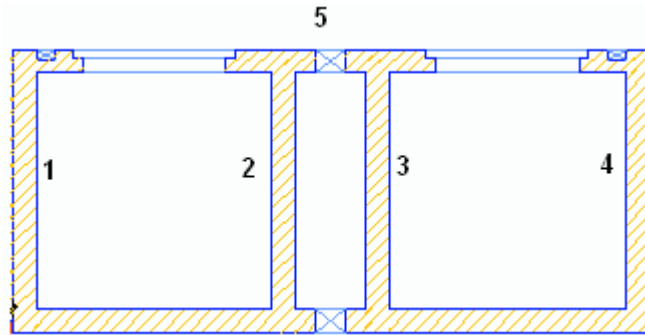


Figure 296: Selecting the elevator walls to create the associative dimension

3. Press **Enter**.
4. Click a point on the drawing to position the associative dimension.

The associative dimension is automatically created.

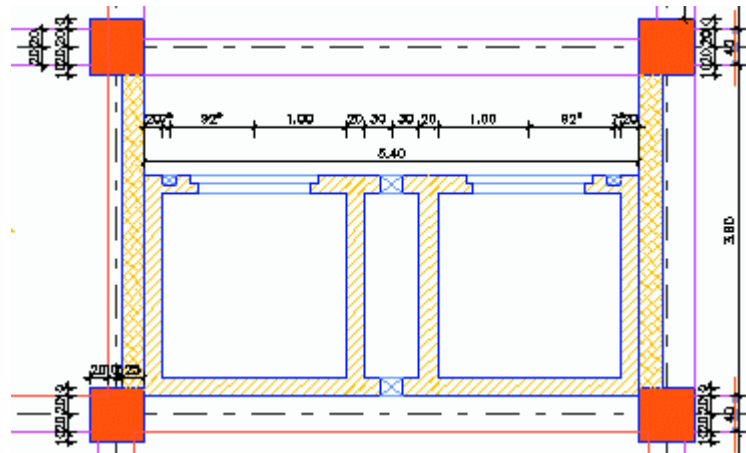


Figure 297: Associative dimension

Selecting the dimensions to display

1. Select the associative dimension.
2. On the Smartbar, click  to access the properties dialog box.

3. Select the **Definition** tab to add or remove dimensions.

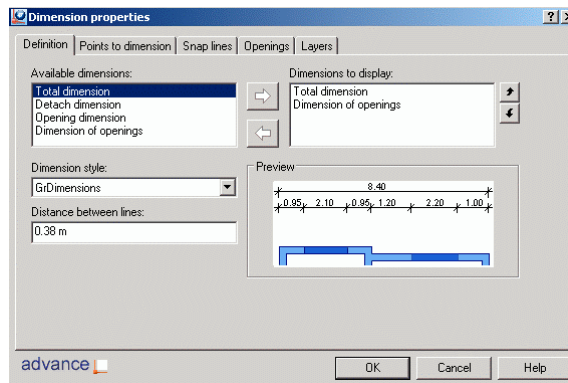


Figure 298: "Dimension properties" dialog box - **Definition** tab

- From the **Available dimensions** list, select **Total dimensions** and click to add the element in the **Dimension to display** list.
- Click to move the dimension up one level.

4. Select the **Openings** tab.

This tab is used to configure the associative dimension numerator and the denominator for the selected wall openings. It is possible to see 2 numerator and 2 denominator variables. The variables are Opening height, Opening width and Sill / threshold height.

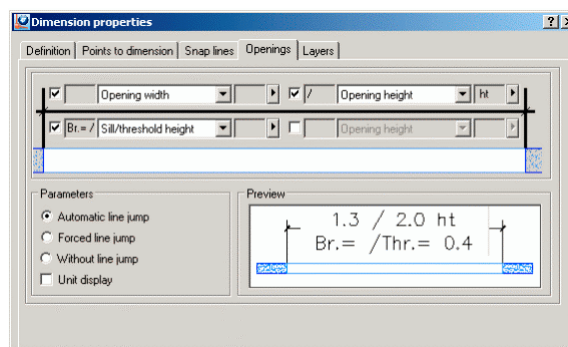


Figure 299: Dimension properties" dialog box – **Openings** tab

In our example, select the **Opening width** and the **Opening height**, above and under the dimension line.

- Check the option in the left-bottom side and select **Opening height** from the drop-down list to display the opening height in the top-left side of the associative dimension.
- Uncheck the option in the top-right side.
- Click the arrow next to the **Opening height** and select **Modify the prefix** from the menu.
- In the **Opening height prefix** field, erase the existing writing and click **OK**.



Figure 300: Adjusting the opening height

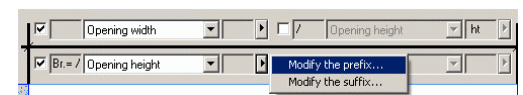


Figure 301: Setting the prefix

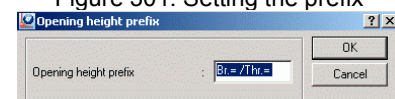


Figure 302: Setting the opening height prefix

5. Click **OK** to finish.

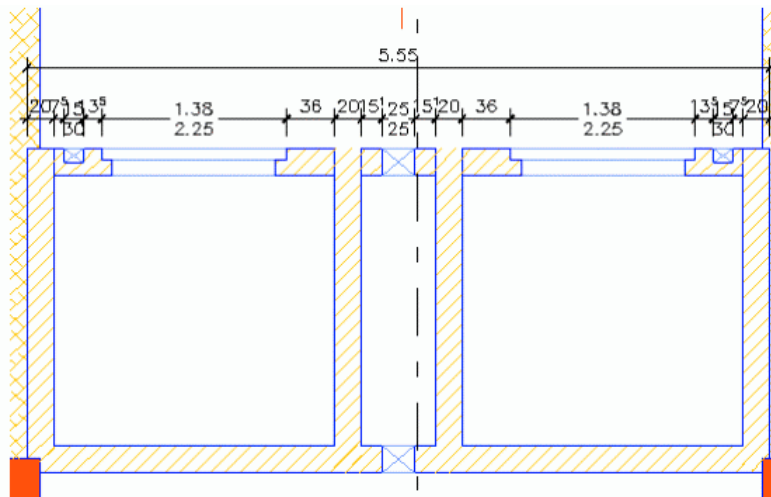


Figure 303: Intersection dimension

Create the second associative dimension:

1. On the **Views** toolbar, click .
2. On the Smartbar, click  to access the properties dialog box.
3. Using the same process, add and remove associative dimensions.

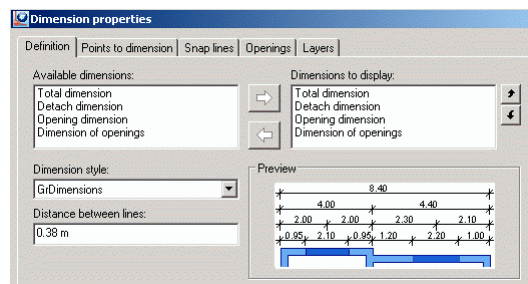


Figure 304: “Dimension properties” dialog box

4. Select the **D20** and **D21'** columns and press **Enter**.
5. On the Smartbar, click  to access the properties dialog box.
6. On the **Definition** tab, in the **Dimension to display** list, select **Total dimensions** and click the  arrow to remove the element from the list.

7. Place the associative dimension at the left bottom corner of the **D20** column.

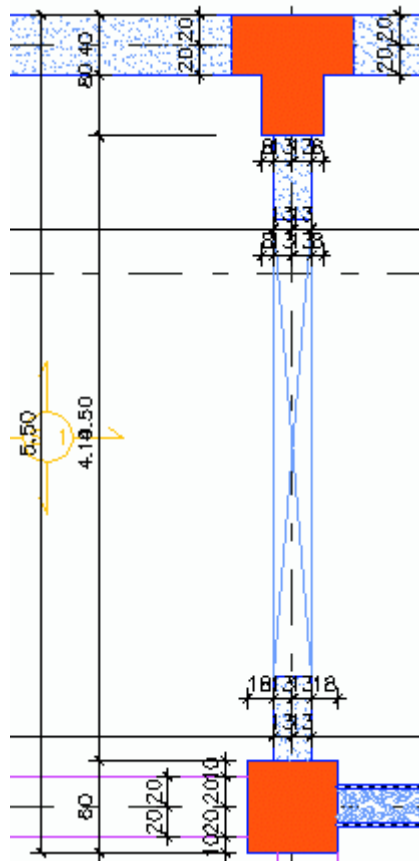


Figure 305: Associative dimension for the **D20** and **D21'** columns



Use the AutoCAD® **Repeat** command, to repeat the last executed command.

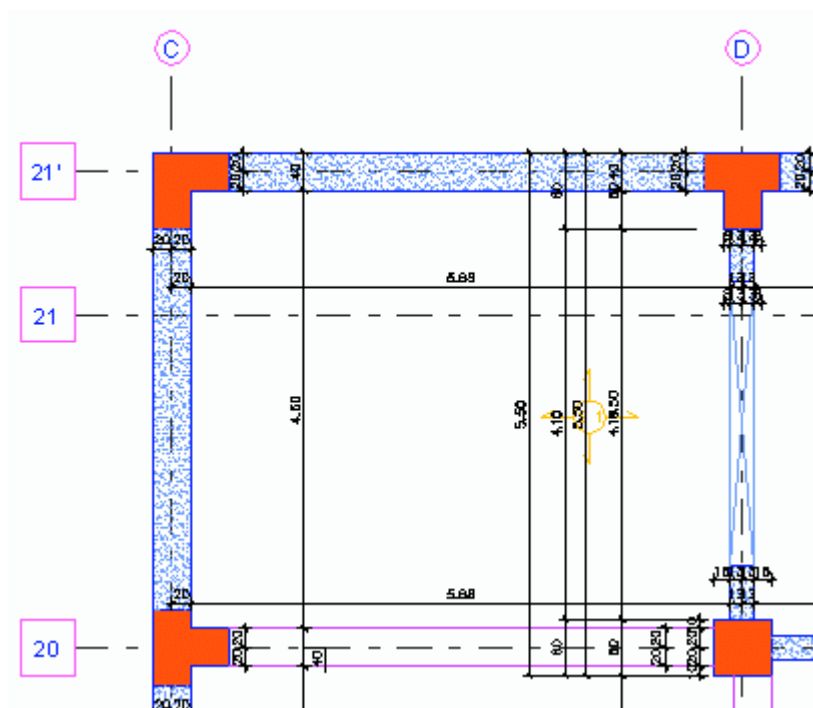


Figure 306: New associative dimensions

Step 5: Create a dimension by points

In this step, you will dimension the wall along the **21'** axis.

1. On the **Views** toolbar, click .
2. Click a point to define the dimension line position.

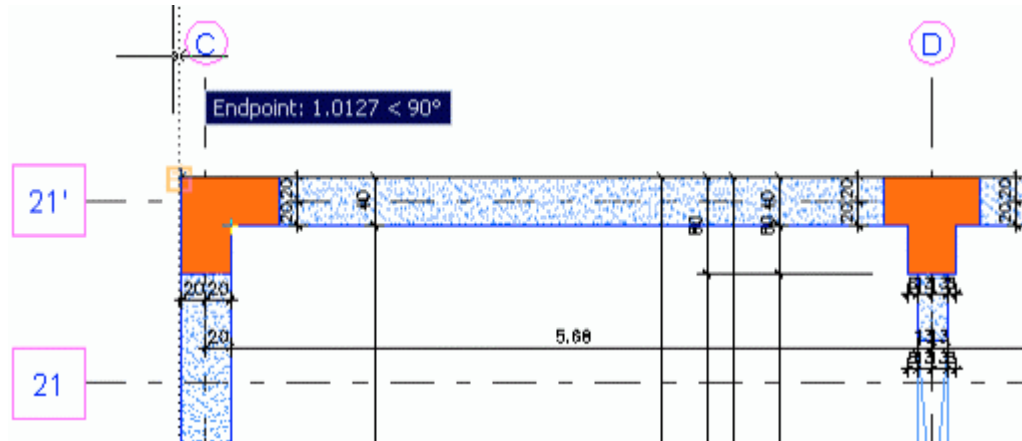


Figure 307: Defining the dimension line position

3. Click the first point of the dimension: the top left corner of the **C21'** column section.

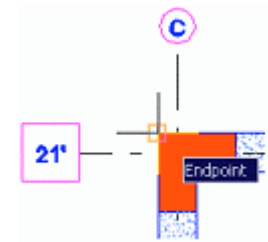


Figure 308: First point to dimension

4. Click the second point of the dimension line: the top corner of the **F'21'** column section.

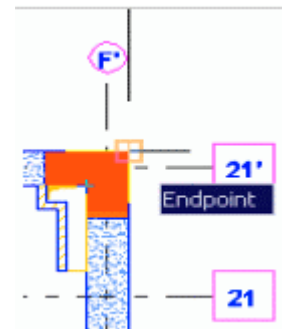


Figure 309: Second point to dimension

The dimension line is created.

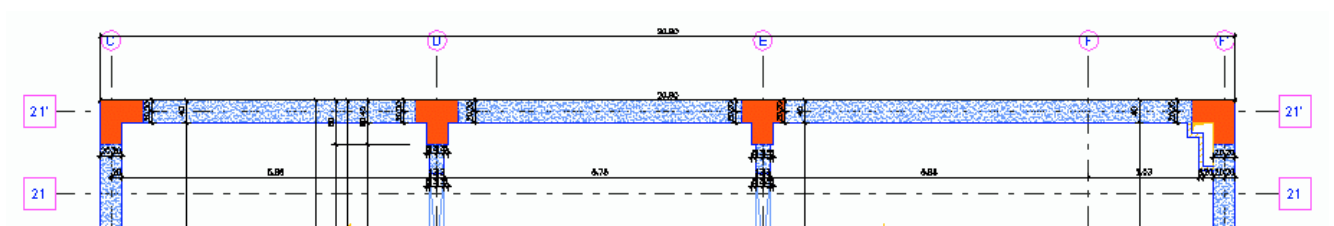


Figure 310: Dimension line

Using the same process, create dimensions by points as shown the following figure.

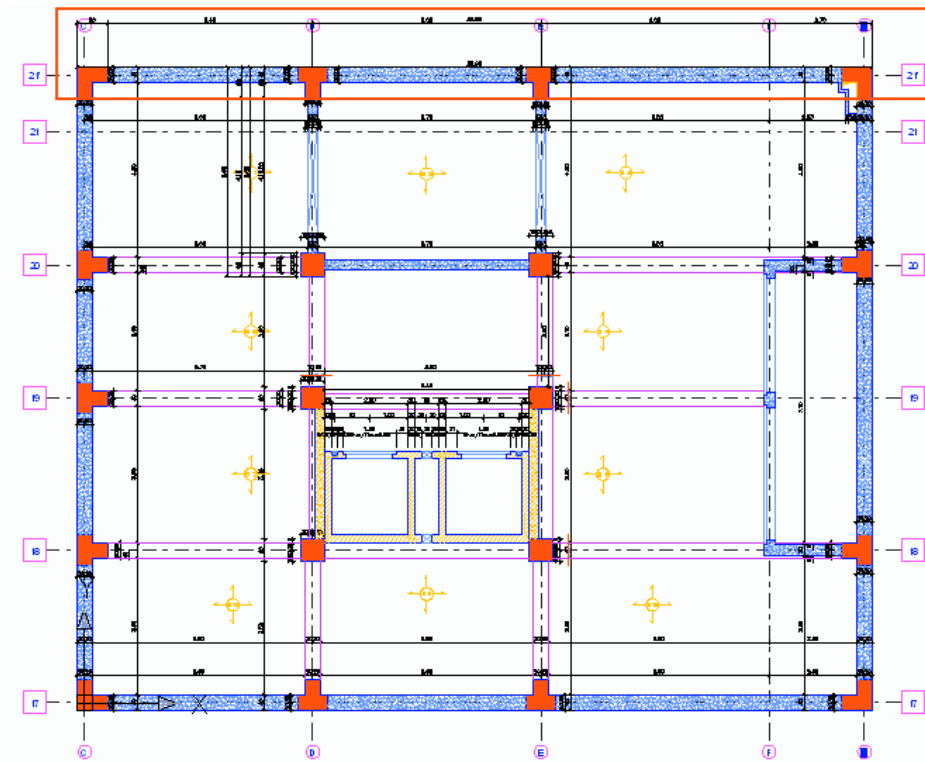


Figure 311: Dimension line

Step 6: Create a symbol

The symbol contains a certain number of attributes of the object to which it is linked. Symbols are saved in libraries, which can be used in other projects.

In this step, you will add a custom symbol for a column.

1. On the **Views** toolbar, click .
2. Select an object on which the symbol will be placed. In this example, it is the **D19** column.

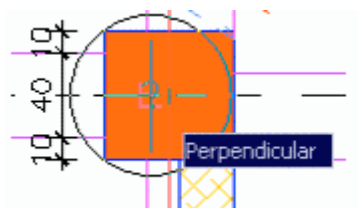


Figure 312: Placing the symbol on the **D19** column

3. On the Smartbar, from the **Name** drop-down list, select the desired symbol.

The symbols are grouped in folders, according to different country standards.

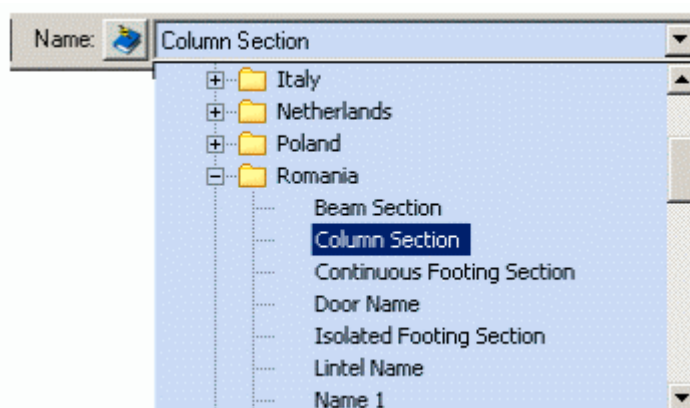


Figure 313: Selecting the symbol

4. Place the symbol next to the **D19** column.

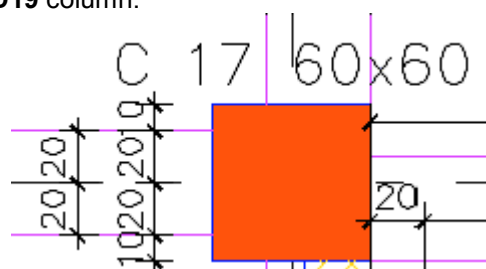


Figure 314: Symbol

Editing the symbol

1. Select the symbol.

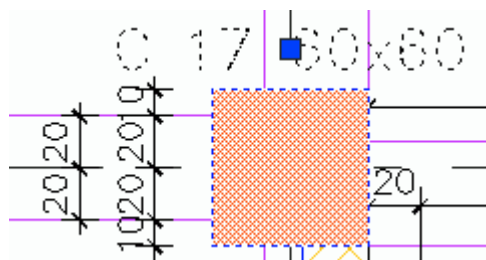


Figure 315: Selecting the symbol

2. On the Smartbar, click  to access the properties dialog box.
3. Select the **Definition** tab and click the **Edit** button to modify the symbol.

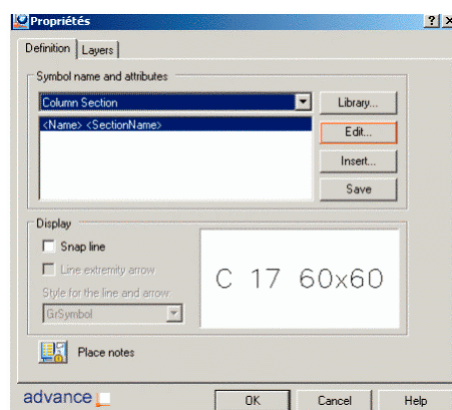


Figure 316: Symbol properties dialog box – Definition tab

The AutoCAD® text editor appears (you can modify the identifiers, the font, character size, add text, etc.).

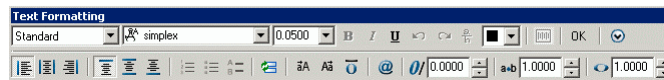


Figure 317: "Text Formatting" dialog box

4. In the edit area, type **Column**. This text appears on the drawing.



Figure 318: Setting a name for the symbol

5. Press **Enter**.
6. On the Smartbar, click  to access the properties dialog box. You can preview the symbol.

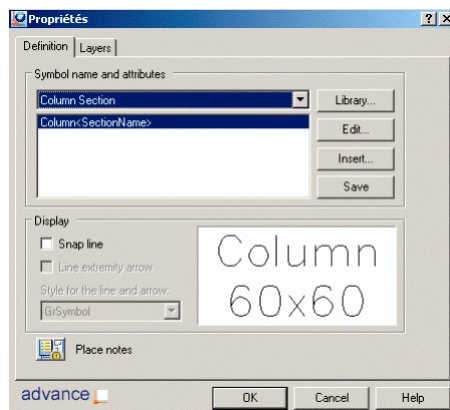


Figure 319: Properties dialog box – symbol preview

7. Press **OK**.

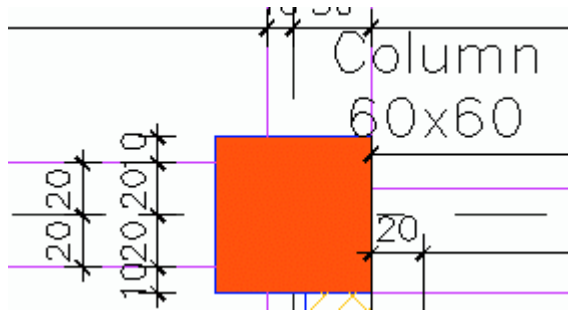


Figure 320: Column symbol - new representation

Modifying the symbol representation

In this exercise, you will add an arrow on the symbol.

1. Select the symbol.
2. On the Smartbar:
 - Select the **Snap line** option to link the symbol to the element.
 - Select the **Line extremity arrow** option to set the representation of the symbol using an arrow.

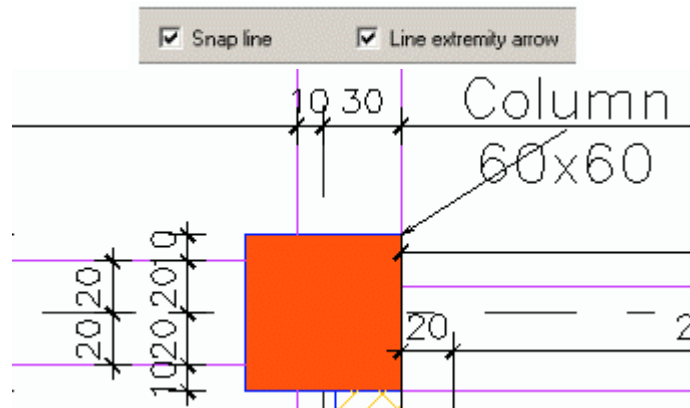


Figure 321: Symbol with line and arrow

Saving the symbol

1. Select the symbol.

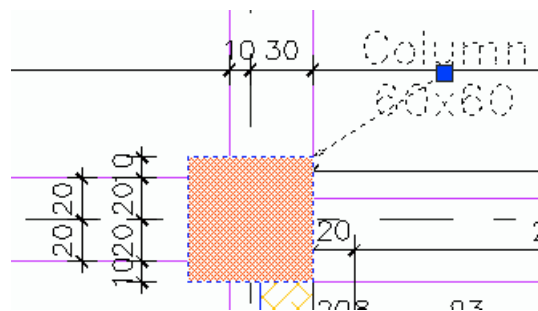


Figure 322: Selection of the symbol

2. On the Smartbar, click  to access the properties dialog box.
3. On the **Definition** tab, click **Save**.

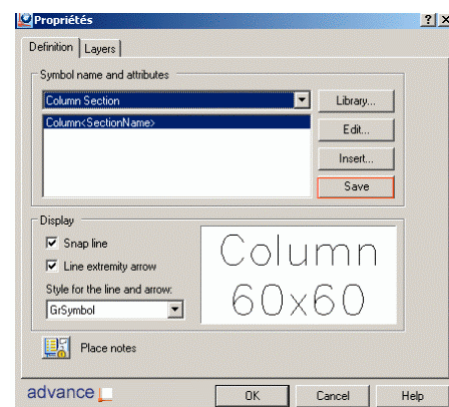


Figure 323: Symbol properties dialog box – Definition tab

- In the “Save symbol” dialog box, click **Browse** to select the folder where the symbol will be saved.

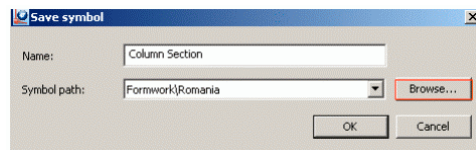


Figure 324: “Save symbol” dialog box

- Click **OK**.
- In the properties dialog box, click **OK**.

Step 7: Create multiple symbols

Symbols can be automatically created on drawings and views for a set of selected objects (e.g., beam, column, slab, etc.).

In this step, you will create symbols for all columns in the drawing.

- On the **Views** toolbar, click .
- Include the model in a window selection and press **Enter** to validate.

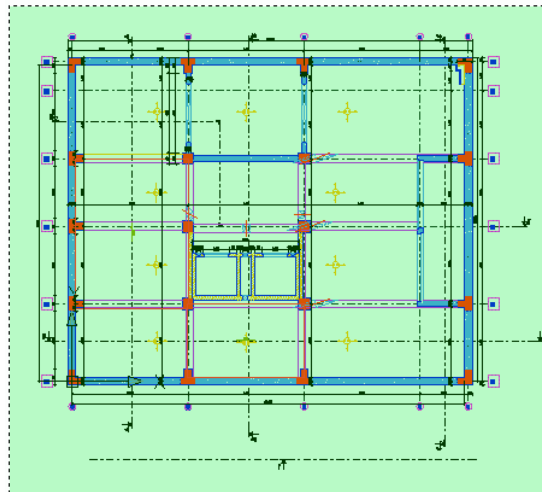


Figure 325: Selecting the drawing

- In the “Multiple symbols” dialog box, check only the **Column** row to add symbols only for the columns and click **OK**.

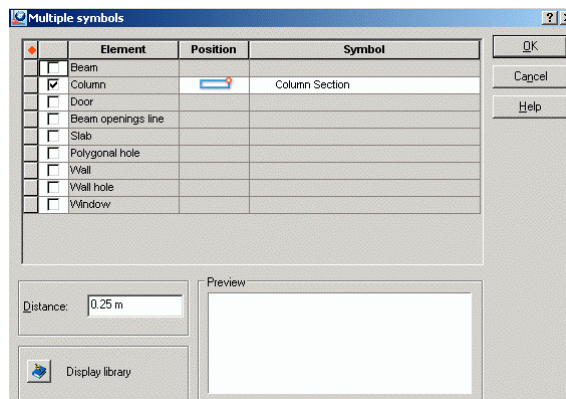


Figure 326: “Multiple Symbols” dialog box

Symbols are placed for all the columns on the drawing.

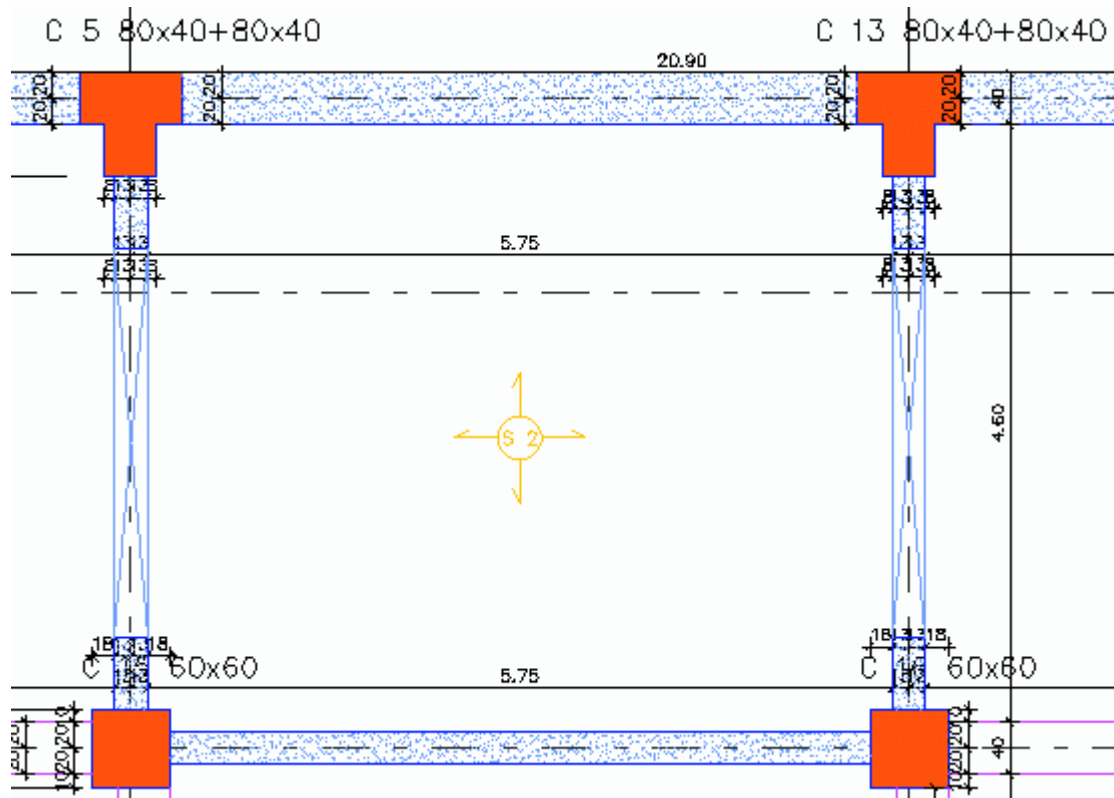


Figure 327: Symbols for columns

Moving the symbols

For a clear view, the symbol position must be adjusted manually using the symbol grip point.

1. Select a symbol.

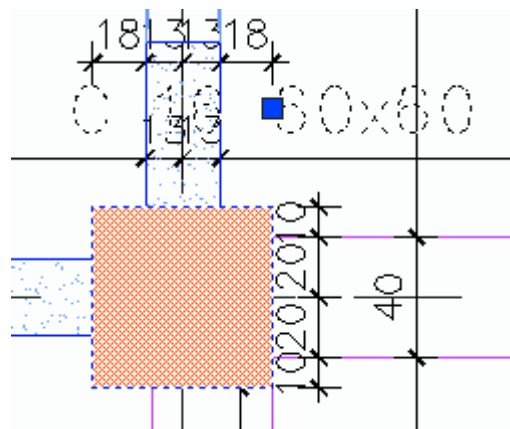


Figure 328: Selecting the symbol

2. Click the symbol again.

The grip point is red. The symbol is attached to the cursor.

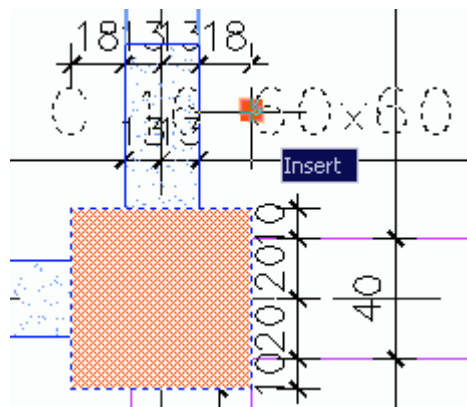


Figure 329: Moving the symbol

3. Click a point on the drawing to position the symbol.
4. Press **Esc** to finish.

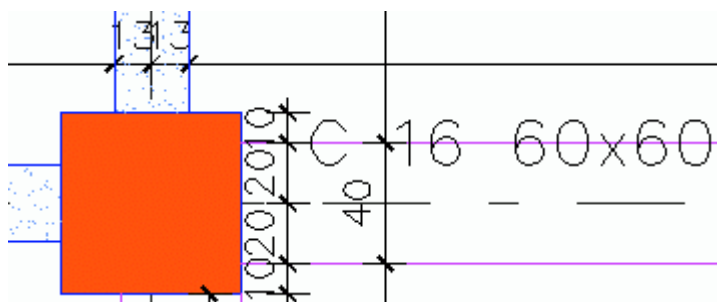


Figure 330: Symbol

Using the same process, move all symbols.

Lesson 13: Reinforcing a column

In this lesson, you will reinforce the **E20** column, in **Underground Level 2**.

You will learn how to:

- Create and modify a reinforcement drawing.
- Create the necessary dimensions on the reinforcement drawing.
- Create a rectangular frame.
- Create and modify a bar symbol.
- Create bar distributions: linear, free, one by one.
- Dimension a distribution.
- Create and modify a list.
- Create an annotated bar bending detail.
- Group a drawing in the column reinforcement.

Advance automatically creates the basic drawings for the reinforcement drawing starting from a selection of structural elements. According to the selected view arrangement type, **Advance** creates the cuts and elevations.

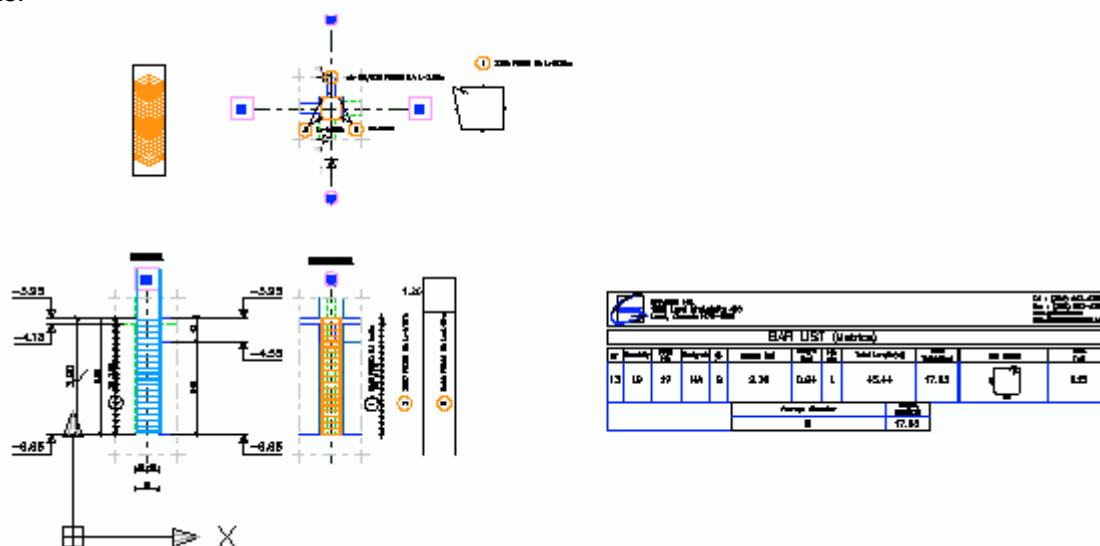


Figure 331: Final drawing of the column reinforcement

Before starting:

In our example, the reinforcement symbol, the bending detail annotation creation and the distribution dimension start automatically. This is possible by activating the options in the “User preferences” dialog box.

1. In the **Pilot**, in **Drawings** mode, right click **Project** and select **User preferences** from the context menu.
2. In the “User preferences” dialog box, **Visual elements** tab, check the options for creating reinforcement symbols, bending details, and a distribution dimension.

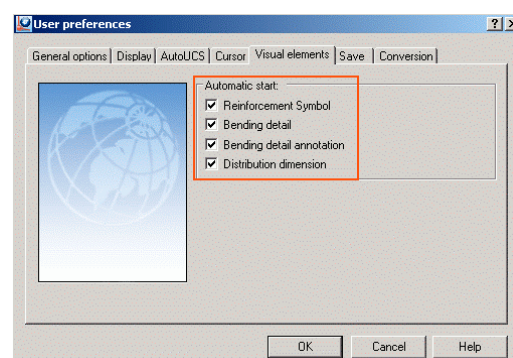


Figure 332: “User preferences” dialog box – **Visual elements** tab

Step 1: Create the reinforcement drawing

1. In the Pilot, click  to activate the **Model** mode.
2. Right click **Underground Level 2** and select **Enable** from the context menu.

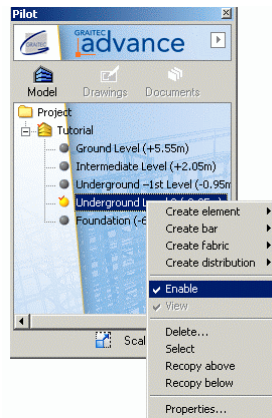


Figure 333: Activating **Underground Level 2**

3. On the **Reinforcement** toolbar, click .
4. Select the **E20** column and press **Enter**. The first dialog box of the reinforcement drawing creation wizard appears.

In the first step of the wizard, select the arrangement of the views calculated by Advance. The different arrangements are grouped by:

- Number of views (e.g., one view, two views, three views, four views)
 - View type (e.g., cut left, top view, elevation face, etc.)
5. Select the **Cut left, top view and elevation face** view template.

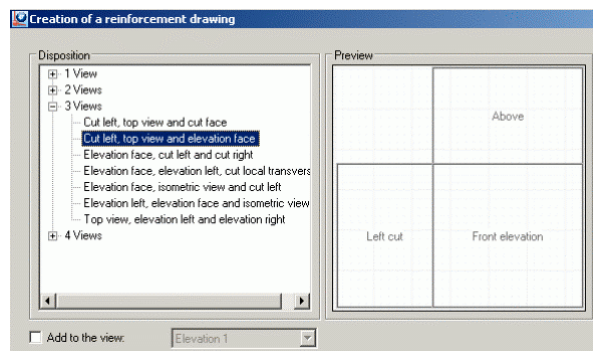


Figure 334: Reinforcement view template

6. Click **Next** until you reach the “Visible objects” dialog box.

7. In the “Visible objects” dialog box select the edge style for the visible objects.
 - In the “Definition of the cut visible edges” area, set the **Line thickness** to **0.50 mm**.

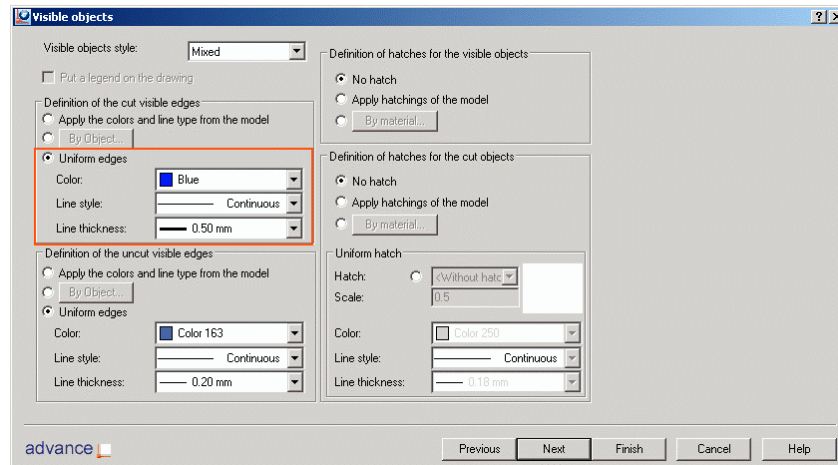


Figure 335: Setting the cut visible edges representation

8. Click **Next** to go to the “Hidden Objects” dialog box.
9. In the “Hidden Objects” dialog box set the edge style for the hidden objects.
 - In the “Definition of hidden objects” area, select **Uniform edges** to display the hidden object edges with uniform edges.
 - Next, set the edge color to **Green**.
 - Set the line thickness to **0.25 mm**.

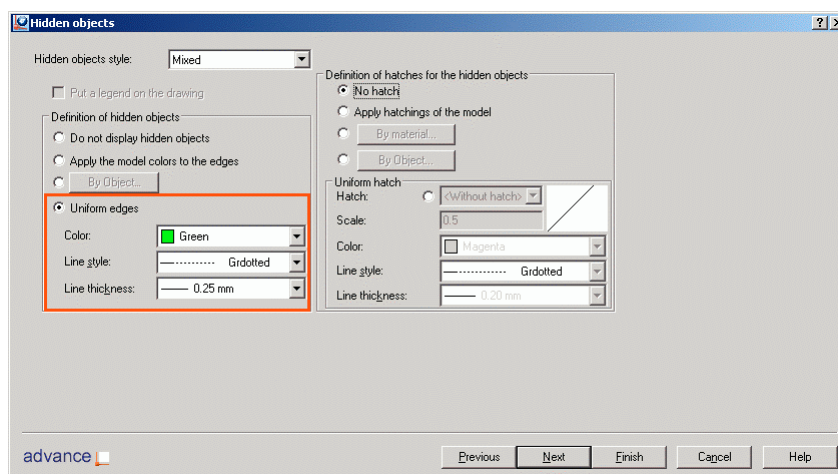


Figure 336: Setting the hidden objects properties

10. Click **Next** to go to the “Finishings” dialog box.

11. In the “Finishings” dialog box, select the **Interruption lines** option to display the cut line and set the **Length** to **0.20 m**.

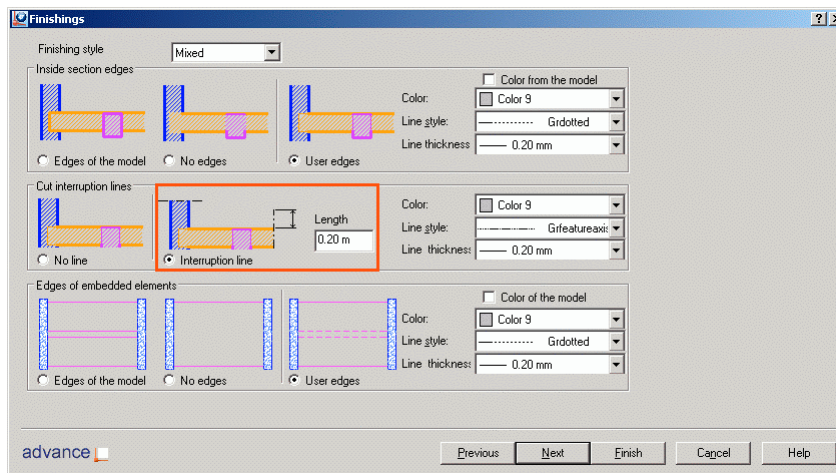


Figure 337: Setting the length of the interruption lines

12. Click **Next** until you reach the “Title” dialog box.
13. In the “Title” dialog box select the information to display on the drawing.
- In the **Prefix** field type the drawing name prefix: **Reinforcement Drawing-Column**. This name appears on the drawing, above all views.
 - Unselect the **Quantities** option.
 - Check the **Length and surface** option.
 - In the “Properties” area, set the text height to **0.25 m**.

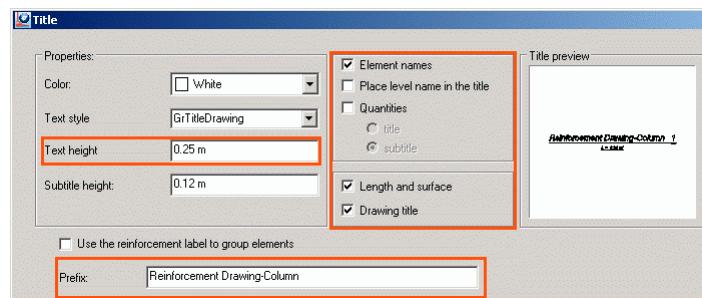


Figure 338: Setting the properties of the text

14. Click **Finish**.

The Pilot switches automatically to the **Drawings** mode and the reinforcement drawing is displayed.

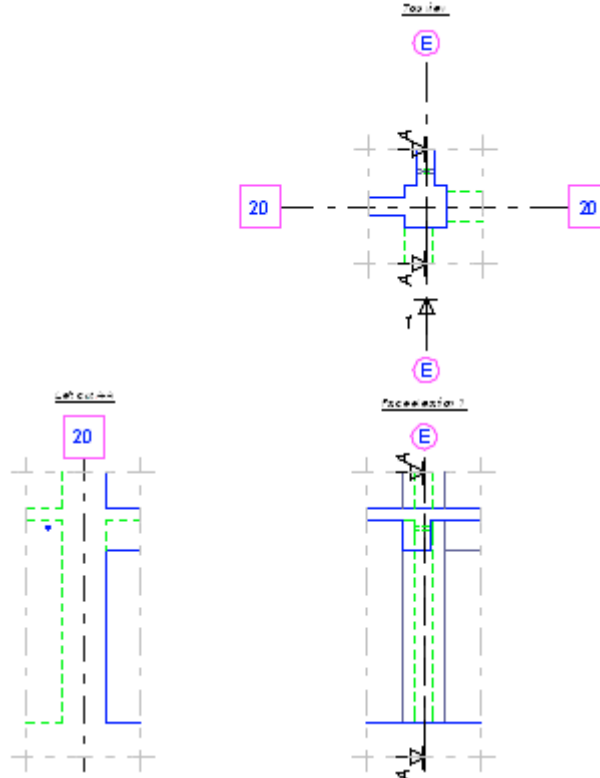


Figure 339: Reinforcement drawing of the **E20** column

Step 2: Create the necessary dimensions

This step briefly describes how to dimension a reinforcement view using intersection dimensions, point by point dimensions, and level dimensions. For details, see Lesson 12.

Creating an intersection dimension on Left cut A-A

Create intersection dimensions on Left cut A-A by taking the **Grid** element into account.

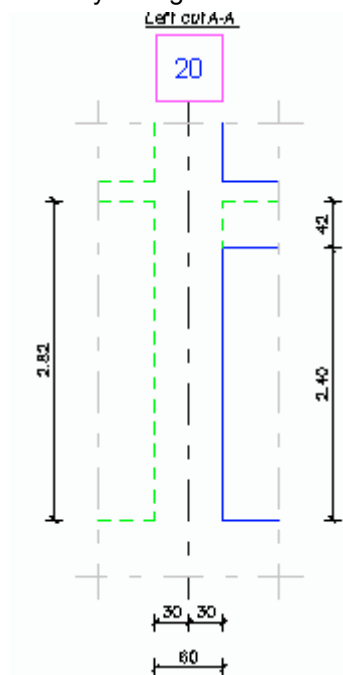


Figure 340: Left cut A-A - creation of the intersection dimension

Creating a point by point dimension on Left cut A-A

Create a point by point dimension on the right, left and bottom side of the Left cut A-A view.

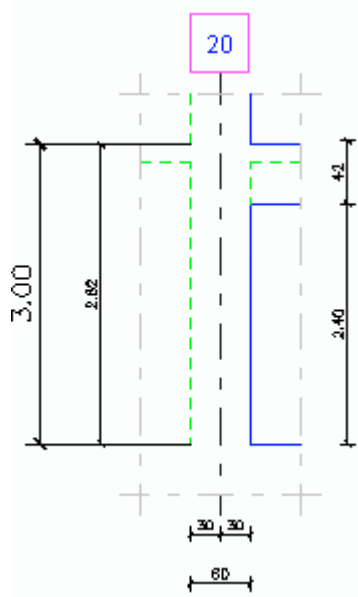


Figure 341: Left cut A-A - creation of the dimension by points

Creating a level dimension

Note: The level dimension is particularly useful to mark the slabs and foundations levels in a building. The command is accessible only in **Drawings** mode.

1. On the **Views** toolbar, click . A message appears on the command line: Dimension origin or [Mirror].
2. Type **M** (mirror) to reverse the arrow position and press **Enter**.
3. Select the dimension origin as shown in Figure 342.

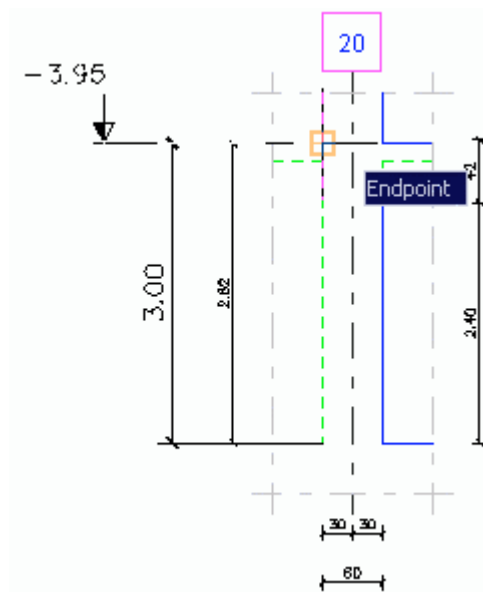


Figure 342: Left cut A-A - setting the dimension origin

- On the Smartbar, from the **Origin** drop-down list, select the **C16** column.

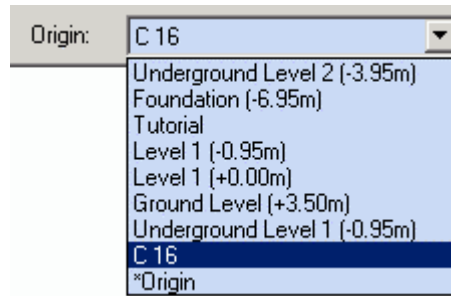


Figure 343: "Origin" drop-down list

- Click a point on the drawing to place the level dimension.
- Create all necessary level dimensions.

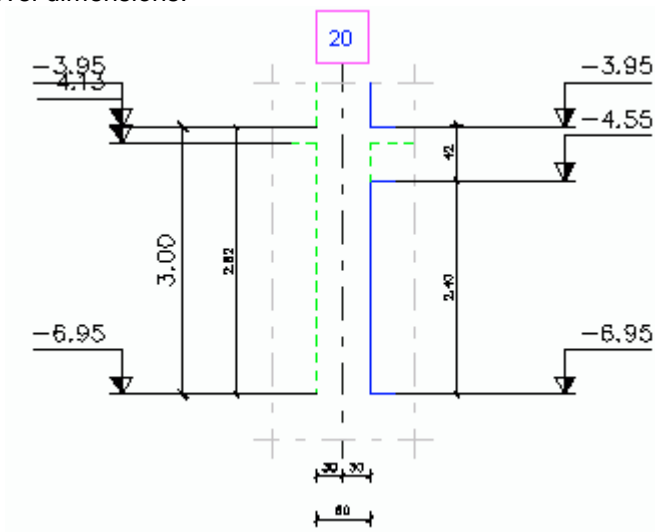


Figure 344: Left cut A-A - creating the level dimensions

- Press **Esc** to finish.

Modifying the level dimension properties

- Select the level dimensions.

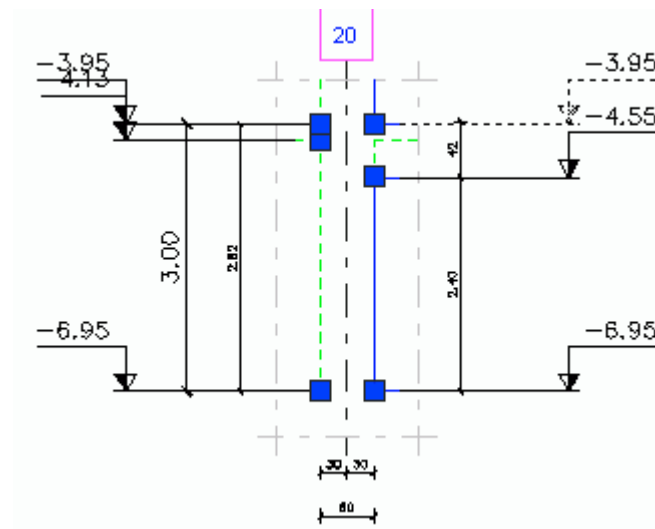


Figure 345: Left cut A-A - Selecting the level dimensions

- On the Smartbar, click to access the properties dialog box.
- On the **Definition** tab, check the options in the **Inverse** column to reverse the dimensions.

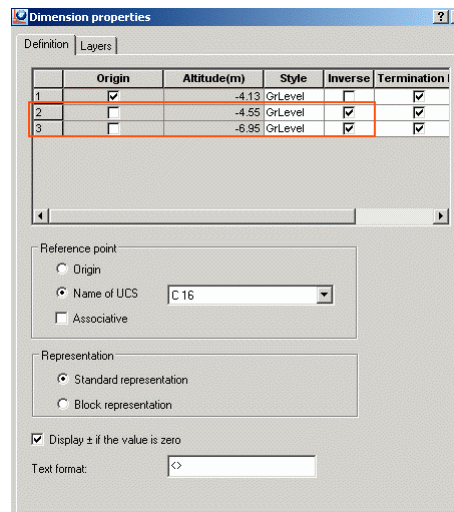


Figure 346: **Definition** tab - reversing the level dimension

- Click **OK**.

In the following steps, add the reinforcement elements.

Step 3: Create a rectangular frame on the top view

Zoom to the top view of the reinforcement drawing.

- On the **Reinforcement** toolbar, **Reinforcement Bars** flyout, click .
- On the Smartbar set the rectangular frame properties:
 - Bar diameter: **8Ø**.
 - Concrete cover: **0.02 m**.

Note: Advance distinguishes the different covers using a specific note. When the cover is associated to an edge, a letter is used to define the edge.

- Press **Enter**.

4. Place the frame by clicking two diagonal points of the **E20** column section.

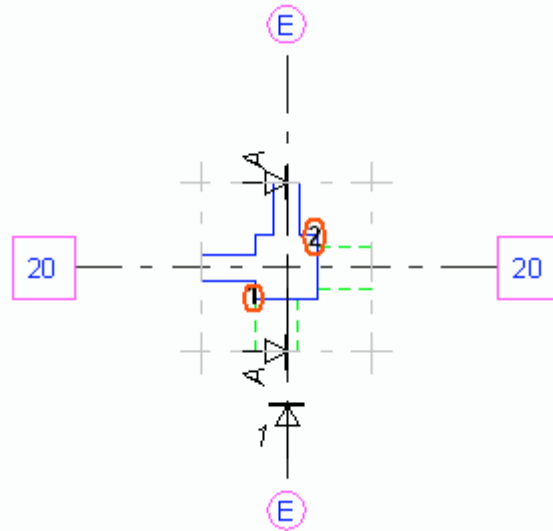


Figure 347: Top view - creating the rectangular frame

5. Click a third point to position the hook of the rectangular frame in the top-left corner. The rectangular frame is created.

Note: In our example, the symbol creation starts automatically after the bar creation. Press **Esc** if you do not want to create the symbol at this time.

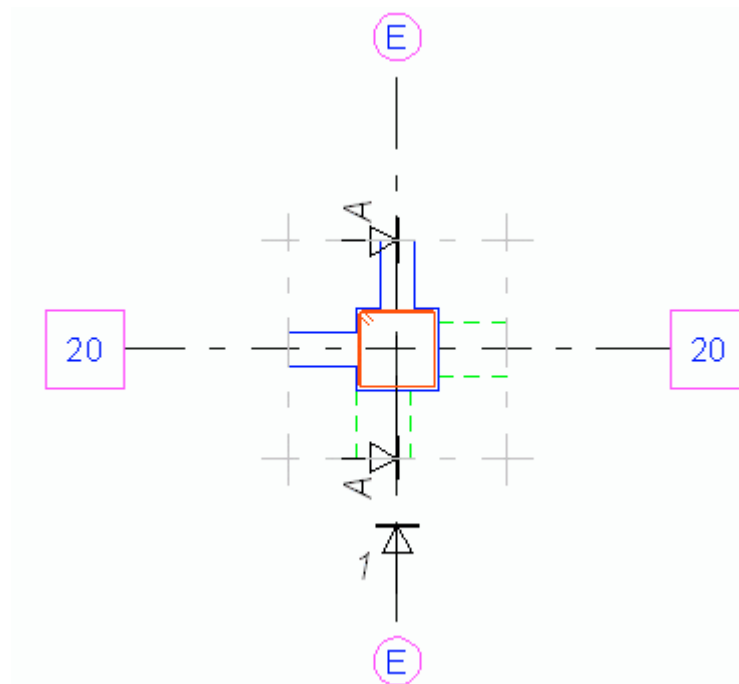


Figure 348: Top view - Rectangular frame

Creating a bar symbol

1. On the **Reinforcement** toolbar, **Symbols and annotations** flyout, click

Note: In our example, the symbol creation starts automatically after the bar creation.

2. On the Smartbar, select the symbol from the drop-down list.

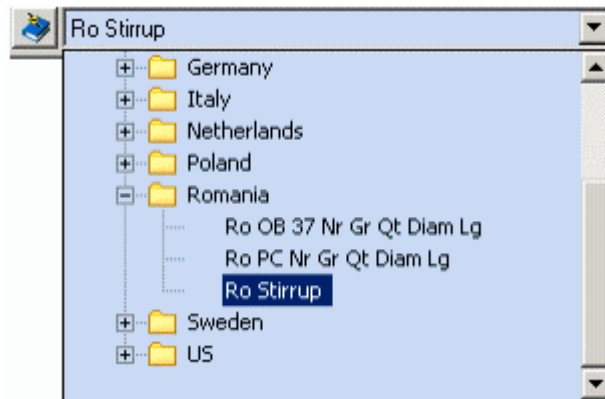


Figure 349: Selecting the symbol

3. Place the symbol as shown in Figure 350.

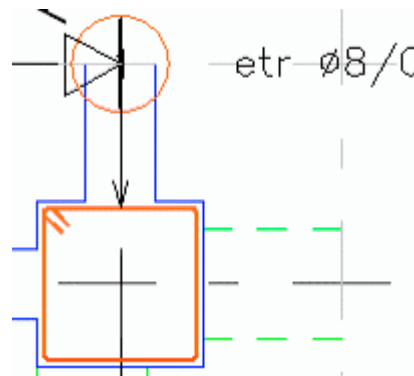


Figure 350: Placing the symbol

4. Press **Esc** to finish.

Modifying the symbol scale

1. Select the symbol.
2. In the Pilot, click and select **1:100** from the list.

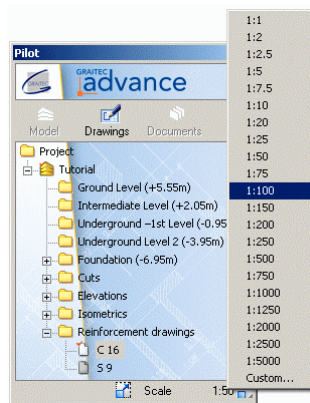


Figure 351: Selecting the scale

3. In the Pilot, click  to confirm.

Rounding the value displayed in the symbol

1. In the Pilot, right click **Project** and select **Project settings** from the context menu to display the **Project Settings** toolbar.

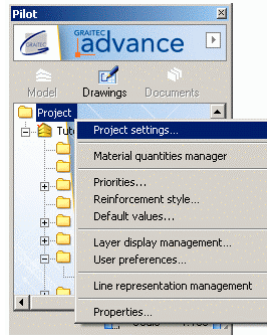


Figure 352: Accessing the **Project settings** menu

2. On the **Project Settings** toolbar, click  to access the rounding rules.
3. In the **Heavy** category, set the total length tolerance to **10 mm**.

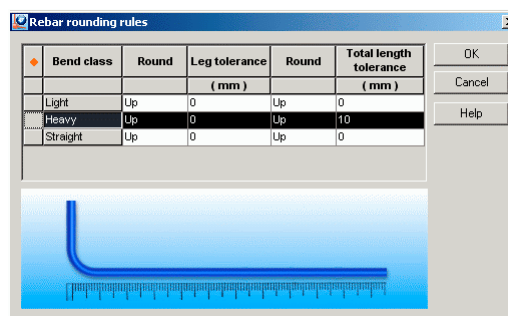


Figure 353: “Rebar rounding rules” dialog box

4. Click **OK**.

The value displayed in the symbol is rounded.

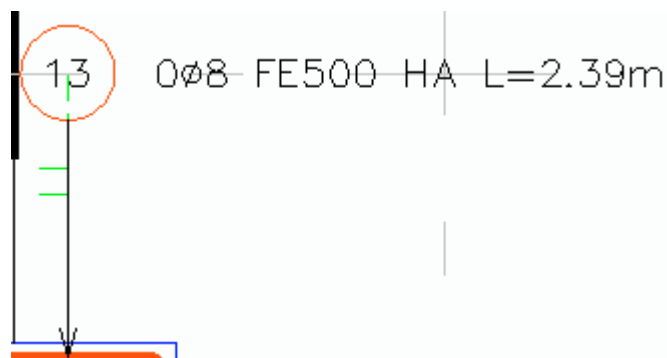


Figure 354: New value displayed in the symbol

Step 4: Distribute the rectangular frame

The rectangular frame will be distributed on the **Face elevation 1** view and on the **Left cut**.

Creating the distribution on the elevation face

1. On the **Reinforcement** toolbar, **Distributions** flyout, click .
2. Select the rectangular frame.
3. The Sideview tool automatically appears as two arrows. Click the vertical arrow to set the view direction.

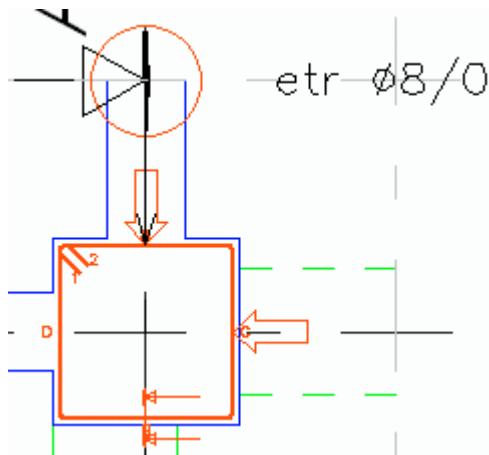


Figure 355: Top view - Setting the view direction

4. Click the first point at the left bottom corner of the Face elevation 1 view.

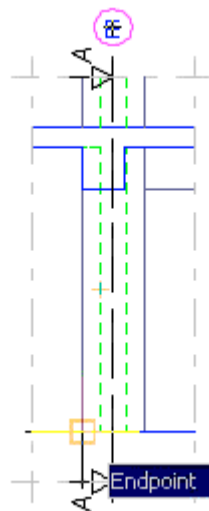
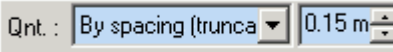


Figure 356: Face elevation 1 - Setting the first point of the cut

5. On the **Smartbar** set the number of bars in the distribution by spacing:

- In the **Qty.** field, enter **0.15 m** to set the distance between bars. 
- Set **0 m** for the **Offset**. 

 **Note:** The bars are bounded to the definition bar. The modification of the definition bar supposes the recalculation of the elements lengths from the attached distribution.

6. Click the second point of the distribution as shown in the following figure.

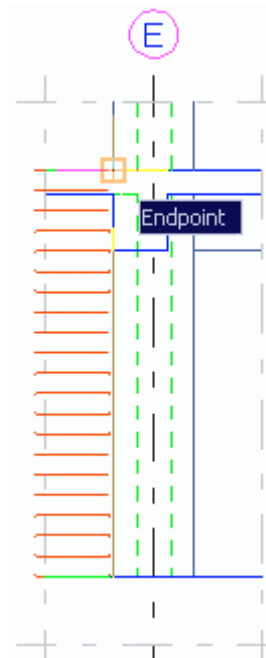


Figure 357: Face elevation - Setting the second point of the distribution

7. Next, click a point inside the elevation to define the distribution bar direction.

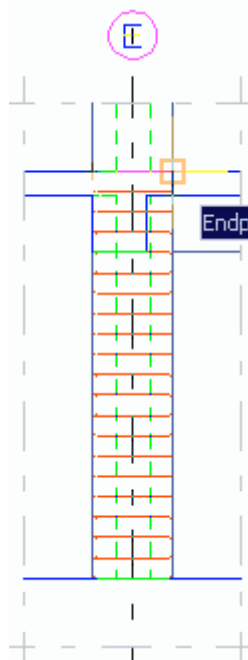


Figure 358: Face elevation 1 - Placing the distribution

8. Use the gray rectangular frame to set the bar position in the distribution. The hook should be on the left side.

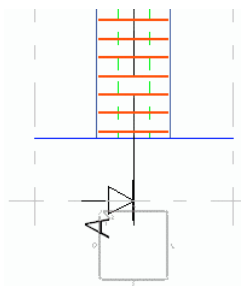


Figure 359: Setting the bar position in the distribution

9. Press **Esc** to finish.

The distribution is created.

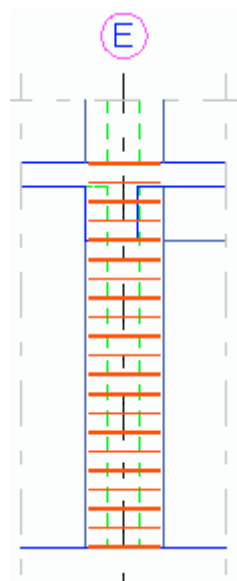


Figure 360: Face elevation 1 - Distribution

 **Note:** In our example, the distribution dimension creation starts automatically after the distribution creation.

The dimension line appears automatically hooked to the cursor.

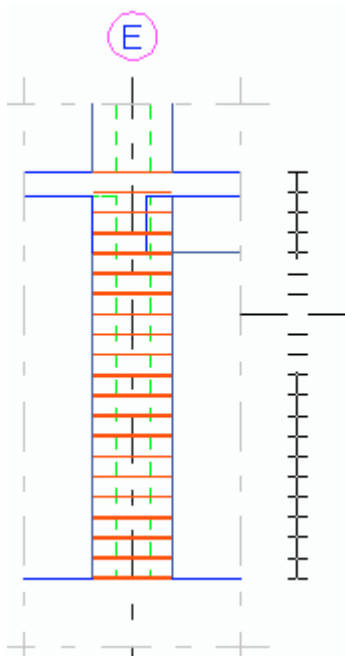


Figure 361: Face elevation 1- Placing the dimension line

10. Click next to the cut section to specify the dimension line position.
11. Press **Enter** to finish.

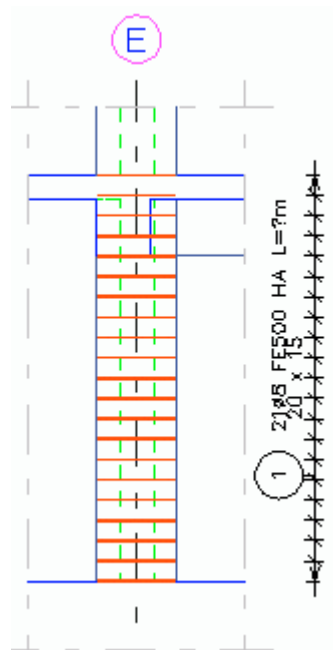


Figure 362: Face elevation 1 - Dimension line

Modifying the distribution dimension symbol

1. Select the distribution dimension.
2. On the Smartbar, click  to access the properties dialog box.
3. Select the **Reference mark and bending detail** tab to modify the symbol representation.
4. From the drop-down list select another symbol, for example, **Romania > Ro Nr Sp Diam**.

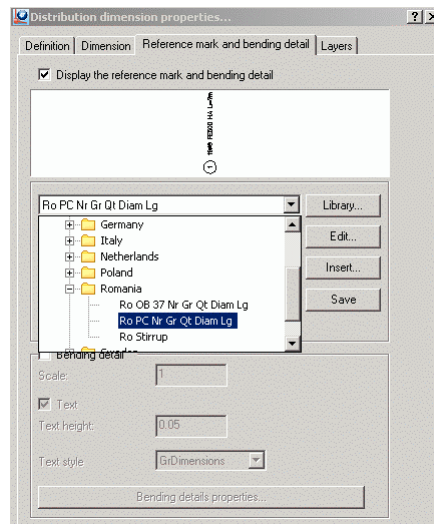


Figure 363: Selecting another symbol

5. Click **OK**.
- The symbol is changed.

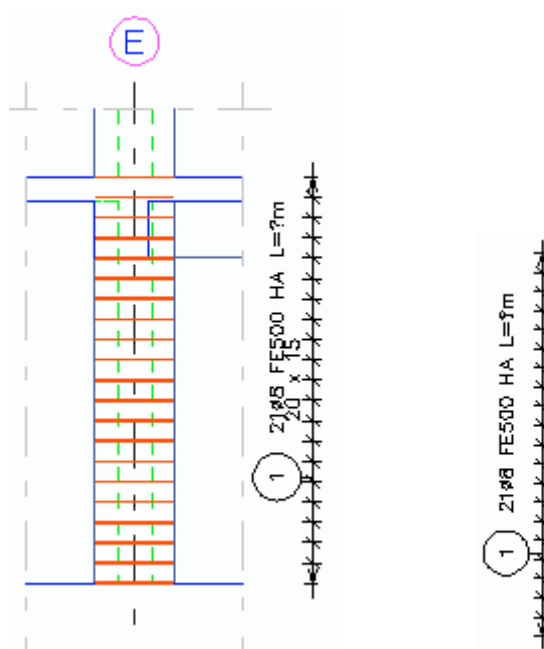


Figure 364: Distribution dimension and the symbol

Using the same process, distribute the rectangular frame on **Left cut A-A**. Using the gray rectangular frame set the bar position in the distribution. The hook should be in the top-left corner.

The drawing in the following figure is obtained.

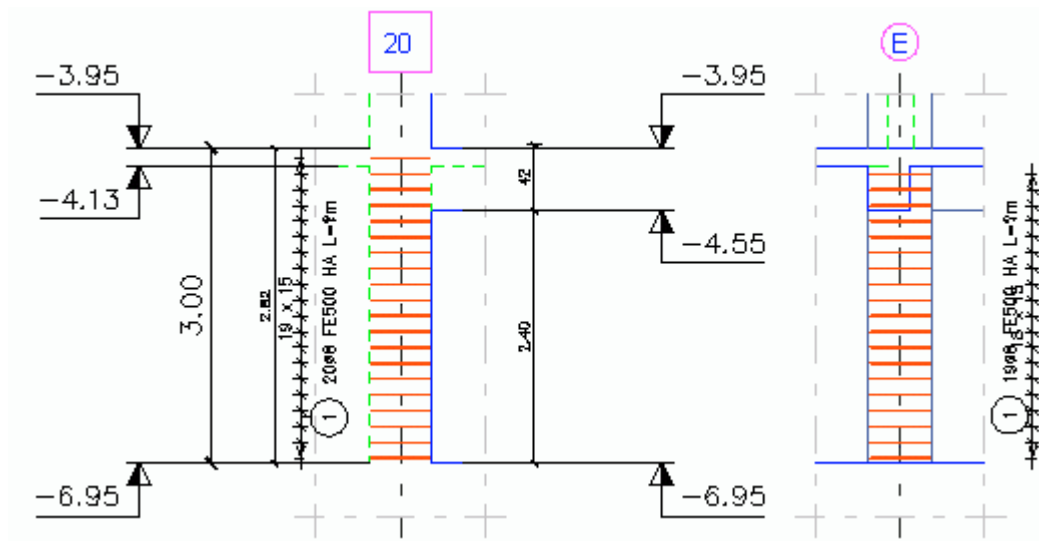


Figure 365: Bar distributions on Left cut A-A and Face elevation 1

Step 5: Create a list on the layout

1. On the **Reinforcement** toolbar, **Lists** flyout, click .
2. On the command line, enter **B** (bar) to consider the bars in the list and press **Enter**.
3. On the command line, enter **V** to select all the elements in the current view and press **Enter**.
4. On the Smartbar, from the drop-down list, select a bar list template and press **Enter**.

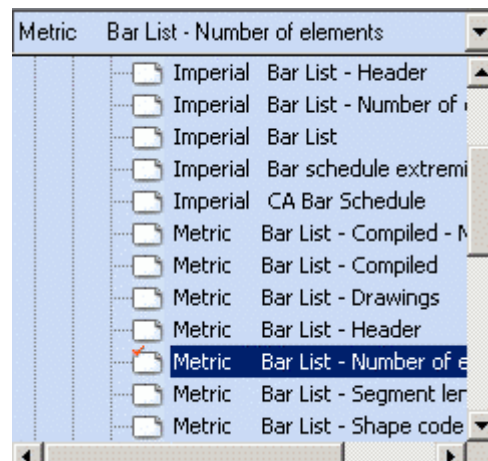


Figure 366: Selecting a list template

5. Click a point on the drawing to place the list.

 GRAITEC INC. 1989 Level Blvd: Suite 410 Level, Canada H7B-2N2 Tel : (450) 863-1383 Fax : (450) 863-8326 www.graitec.com info.ass@graitec.com											
BAR LIST (Metrics)											
N°	Quantity	Total Nb	Steelgrade	Ø	Length (m)	Weight (kg)	Nb bars	Total Length(m)	Total Weight(kg)	Bar Bends	Dist. (m)
1	42	42	HA	8	2.39	0.94	1	100.45	39.68		###
Average diameter:						Weight					
8						Bar(m)					
						39.68					

Figure 367: List

Updating the list

Note: After creating the list, it is often necessary to modify the element (e.g., bars and meshes) parameters, or to add or delete some of the elements. Advance allows updating the list automatically.

In this tutorial, the number of bars in the list is 42 but only 21 bars are used for reinforcing the column. The mismatch appears because the same bars are represented in two distributions and both are taken into account in the list.

In this step, you will "ignore" the bars from the first distribution, and next, update the list to reflect the modification.

Avoiding taking into account the distribution bars

1. Select the distribution on the **Left cut A-A** view.

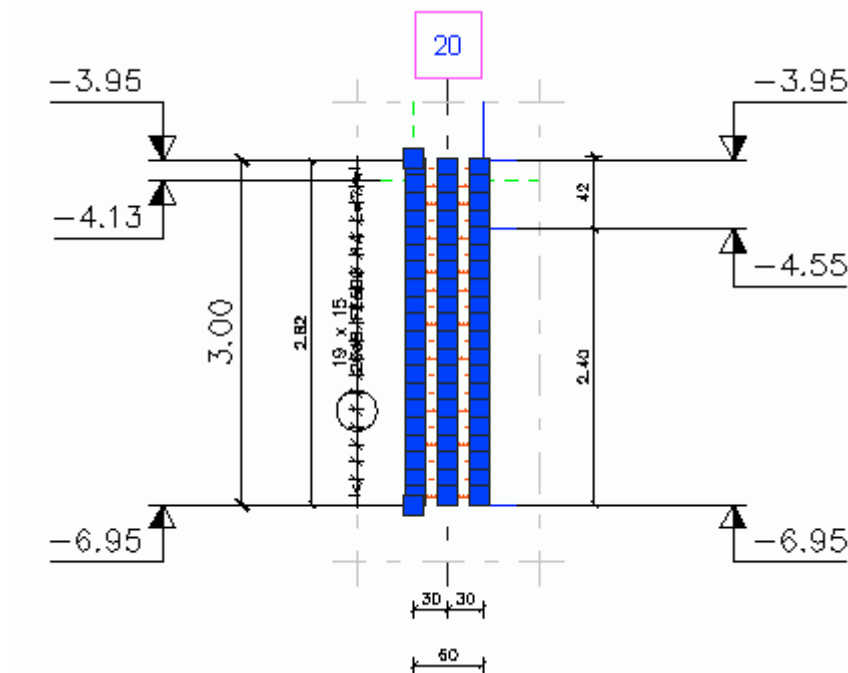


Figure 368: Left cut A-A - Selecting the distribution

2. On the Smartbar, click to access the properties dialog box.
3. On the **Definition** tab, unselect **To count** option to not consider the bars.

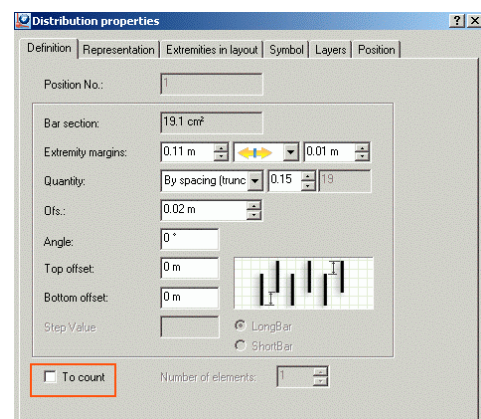


Figure 369: "Distribution properties" dialog box – Definition tab

4. Click **OK**.

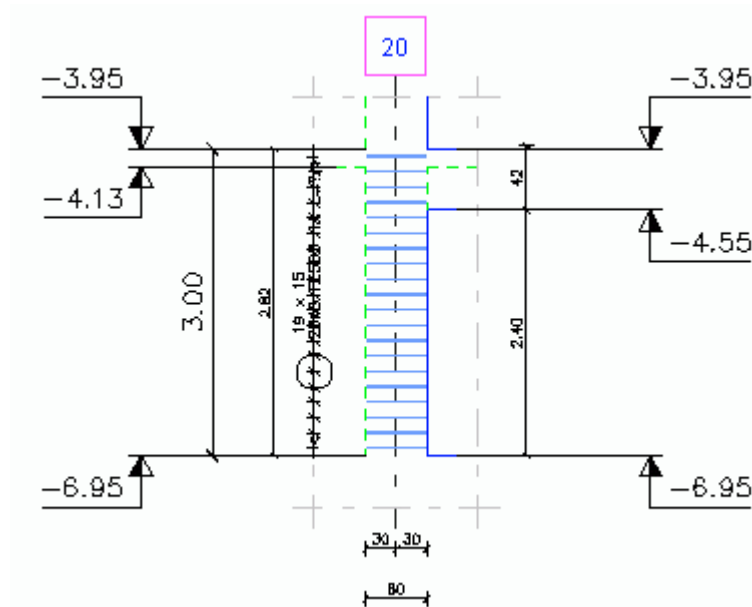


Figure 370: Left cut A-A - modified cuts

The list is displayed in red as it is affected by the modification.


GRAITEC INC.
1845 Laval Blvd., Suite 410
Laval, Canada H7S-2M2

Tel : (450) 883-1383
Fax : (450) 883-8325
www.graitec.com
info.canada@graites.com

BAR LIST (Metrics)											
Nr	Quantity	Total Nb	Reinforce	Ø	Length (m)	Weight (kg)	Nb. els.	Total Length(m)	Total Weight(kg)	Bar Bends	Dist. (m)
13	42	42	HA	8	2.39	0.94	1	100.45	39.68		###
						Average diameter		Weight Sum(kg)			
						8		39.68			

Figure 371: List must be updated

Updating the list

1. Select the list.
2. Right click and select **Update** from the context menu.

The number of bars is correct.



GRAITEC INC.
1845 Laval Blvd., Suite 410
Laval, Canada H7S-2M2

Tel : (450) 883-1383
Fax : (450) 883-8325
www.graitec.com
info.canada@graites.com

BAR LIST (Metrics)

Nr	Quantity	Total Nb	Reinforce	Ø	Length (m)	Weight (kg)	Nb. els.	Total Length(m)	Total Weight(kg)	Bar Bends	Dist. (m)
13	21	21	HA	8	2.39	0.94	1	50.23	19.84		###
					Average diameter			Weight Sum(kg)			
					8			19.84			

Figure 372: List

Note: Lists can be added on the drawing and deleted from the drawing at any time.

Step 6: Add the longitudinal reinforcement

In this step, you will create two straight bars on the Face elevation 1 view.

Creating a straight bar

Create a straight bar, with a diameter of $\varnothing 20$ and an **Ext.2** of the Concrete cover of **0 m**.

1. On the **Reinforcement** toolbar, **Reinforcement bars** flyout, click .
2. On the Smartbar, define the following settings:
 - Select a bar diameter of $\varnothing 20$.
 - Set the concrete cover to **0.035 m**.
3. Place the bar by specifying two points as shown in Figure 373.

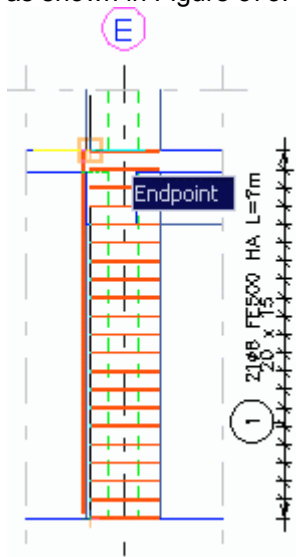


Figure 373: Face elevation 1 - Straight bars

Adding a bar anchor

1. Select the bar.

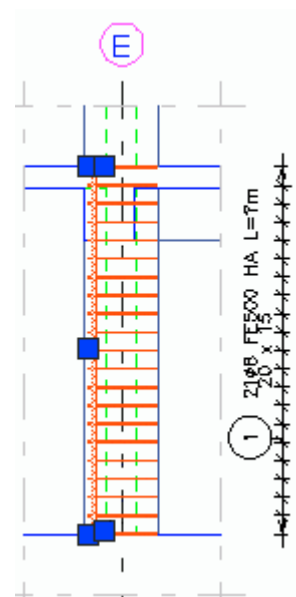


Figure 374: Face elevation 1 - Selecting the bar

2. On the Smartbar, click  to access the properties dialog box.
3. Select the **Hooks and anchors** tab.

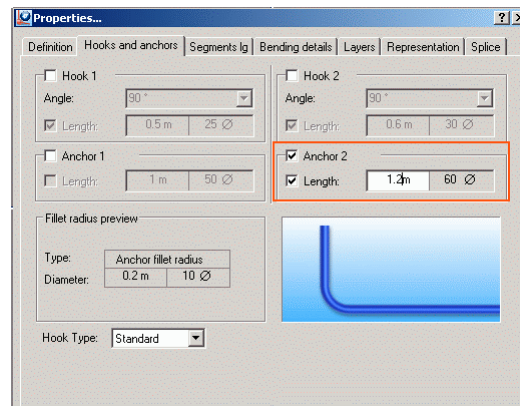


Figure 375: "Properties" dialog box - **Hooks and anchors** tab

- Check the **Anchor 2** option to add an anchor at the second extremity of the bar.
 - Next, check the **Length** option and enter an anchor length of **1.2 m**.
4. Click **OK**.
- The bar is modified.

Adjusting the bar representation

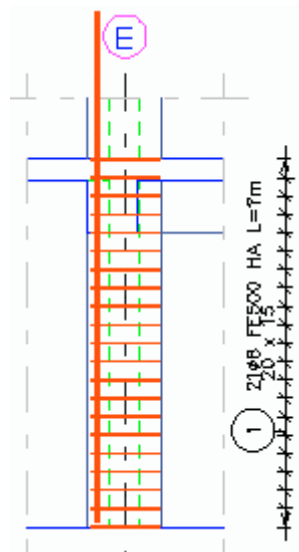


Figure 376: Face elevation 1 - Straight bar

1. Select the straight bar.

- Click the top grip point of the bar and move it to the necessary position to adjust the bar representation.

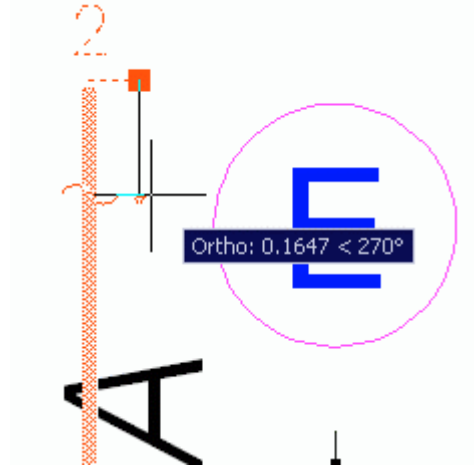


Figure 377: Face elevation 1 - Adjusting the bar

- Press **Esc** to unselect the bar.

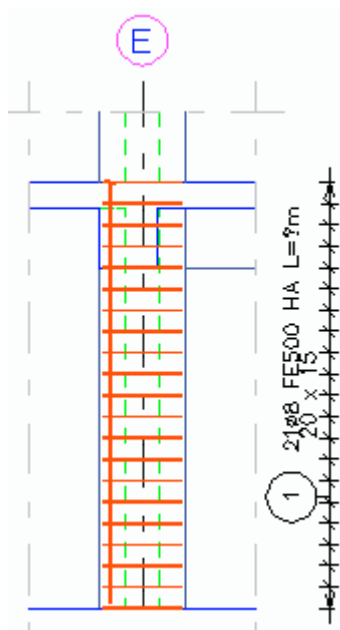


Figure 378: Face elevation 1 - Bar

Using the same process, create another straight bar (with the same properties) on the right side of the Face elevation 1 view.

 **Note:** The settings of the previous straight bar remain active.

As for the previous bar, adjust the bar representation.

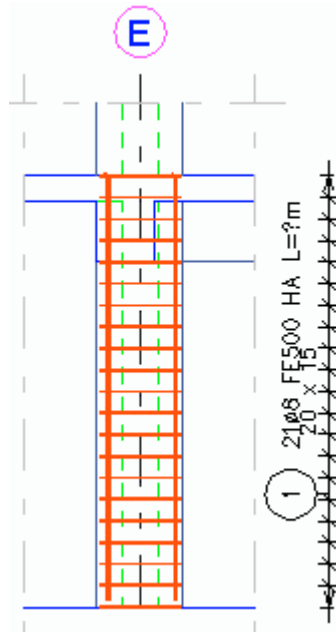


Figure 379: Face elevation 1 - Straight bars

Step 7: Distribute the longitudinal reinforcement

In this step, you will create a free distribution of the two straight bars created in Step 6.

Concept

A free distribution consists of bars represented as points. All point bars of a free distribution are linked.

1. On the **Reinforcement** toolbar, **Distributions** flyout, click .
2. Select the left straight bar on the Elevation face 1 view.

After having clicked the bar, a point bar is hooked on the cursor.

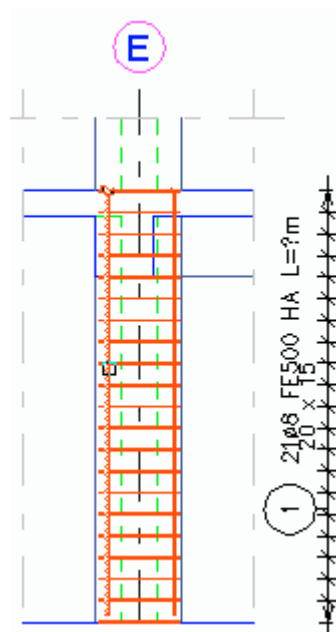


Figure 380: Elevation face 1 - Selecting the bar to distribute

3. Zoom to the rectangular frame.
4. Create a distribution of two point bars on the top left and the bottom left corners of the rectangular frame on the column section (the Top view – see Figure 381).

Note: To facilitate the placement of the two point bars, activate the "center" snap point on the AutoCAD® snap object.

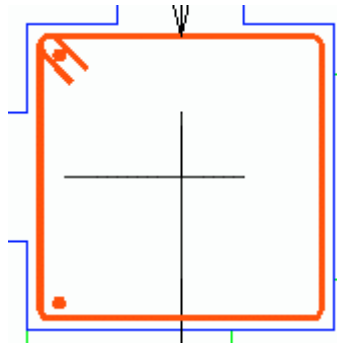


Figure 381: Free distribution of the first bar

5. Press **Esc** to finish.

Using the same process create a free distribution of the right straight bar.

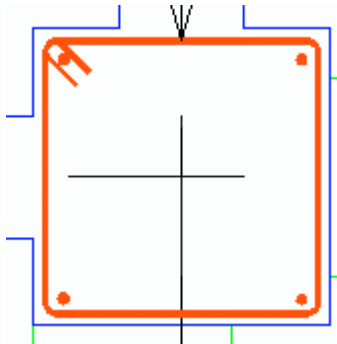


Figure 382: Two free distributions of the longitudinal bars

Step 8: Place a reference mark on the distribution bars

1. On the **Reinforcement** toolbar, **Symbols and annotations** flyout, click .
2. On the Smartbar, select a symbol.

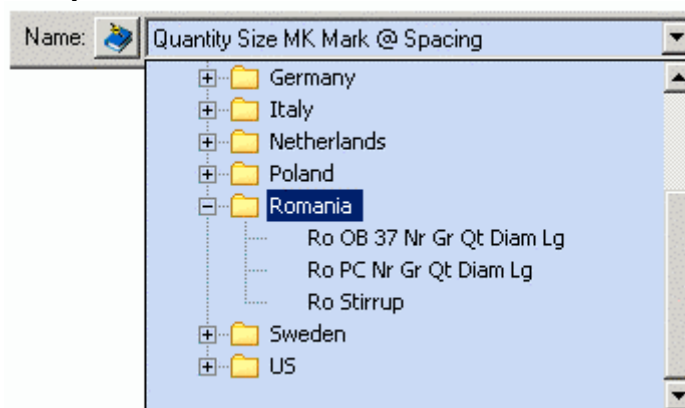


Figure 383: Selecting a symbol name and representation

- Click one of the two point bars of the right distribution and place the reference mark.
Using the same process, place a reference mark on the other distribution.

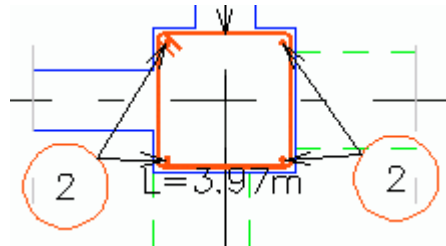


Figure 384: Placing the reference marks

Step 9: Create a one by one distribution of a straight bar

In this step, the straight bars are placed using the one by one distribution tool. All the bars from a distribution are linked.

- On the **Reinforcement** toolbar, **Distributions** flyout, click .
- Select the left bar on the **Face elevation 1** view.
- Click the horizontal arrow to select the view direction.

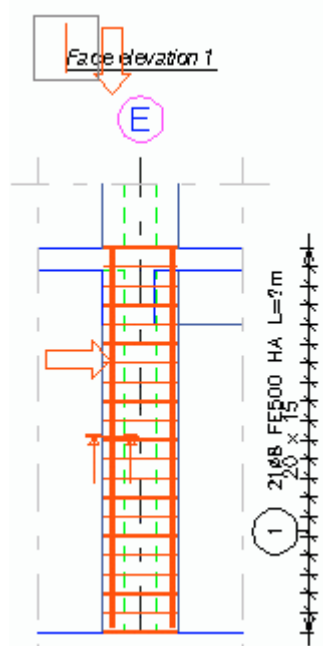
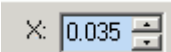


Figure 385: Face elevation 1 - Selecting a view direction

- On the Smartbar, define the following settings:

- Set the justification: .
- Set the offset value on the **X**-axis to **0.035 m**. .
- Press **Enter**.

- Click the bottom left corner of **Left cut A-A** to place the first bar of the distribution.

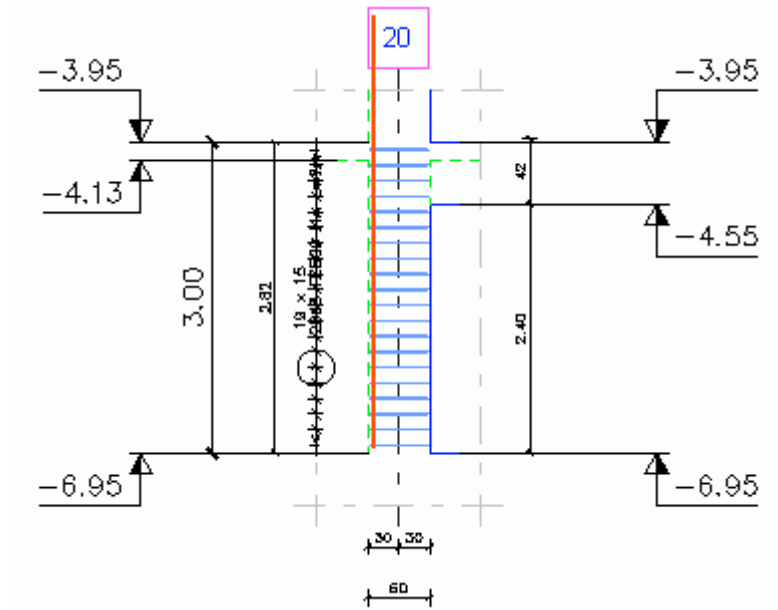


Figure 386: Left cut A-A - Placing the distribution

- On the Smartbar, set the offset value on the Y-axis to **-0.035 m** and press **Enter**.
- Click the right bottom corner of **Left cut A-A** to place the second bar of the distribution.
- Press **Esc** to finish.

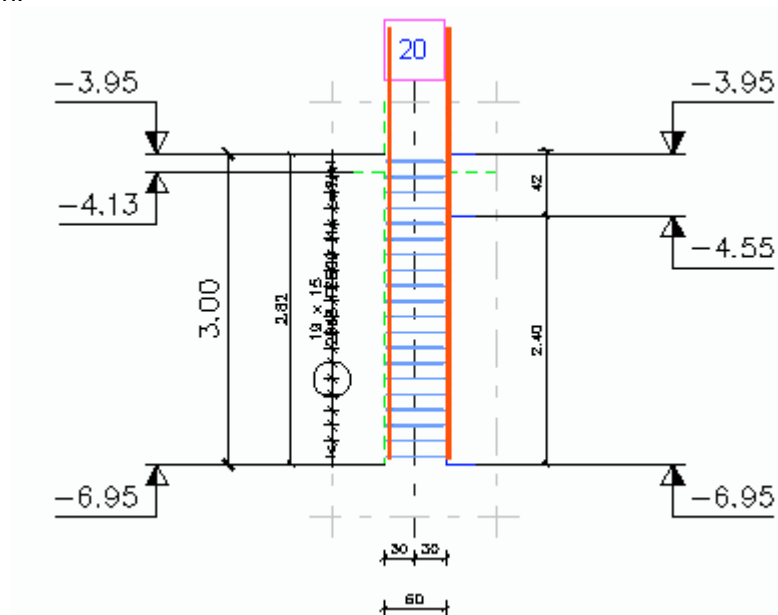


Figure 387: Left cut A-A - One by one distribution

 **Note:** To avoid an incorrect result, the created straight bars must not be taken into account in the lists.

Avoiding taking distribution bars into account

- Select the distribution.
- On the Smartbar, click  to access the properties dialog box.

3. On the **Definition** tab, unselect the **To count** option.

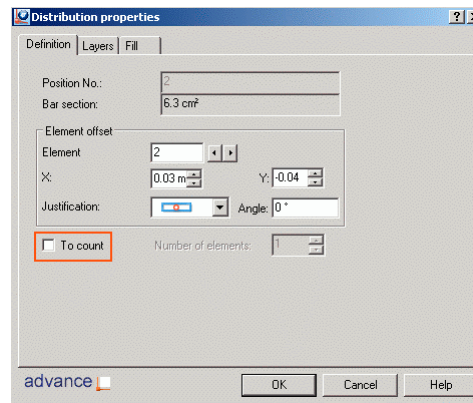


Figure 388: “Distribution properties” dialog box – **Definition** tab

4. Press **OK**.

Using the same process, create two other bars by placing the bar on the right side of the Face elevation 1 view. Also, avoid taking the bars into account.

Step 10: Create a bar bending detail on the right bar of the Face elevation 1 cut

Before creating the bar bending detail, set the scale of the drawing to **1:50**.

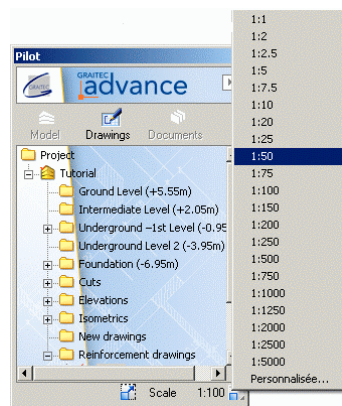


Figure 389: Setting the drawing scale

1. On the **Reinforcement** toolbar, **Distribution** flyout, click .
2. Select the bar on the right side of the **Face elevation 1** view.

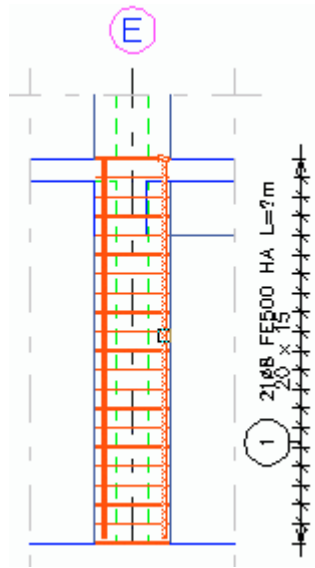


Figure 390: Face elevation 1- Selecting a bar

The bending detail is hooked to the cursor.

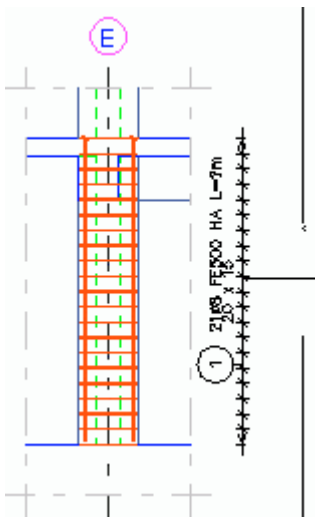


Figure 391: Face elevation 1: Placing a bar

The following message appears on the command line: Point or [Restricted POSitioning/Scale/Rotation/Align/Mirror/Finish].

3. On the command line, enter **pos** to select the restricted positioning mode. The bar schema can be placed only in an orthogonal position.
4. On the Smartbar, in the **Text ht** field, enter **0.01** to adjust the text height.



5. Place the bar bending detail next to the **Face elevation 1** view.

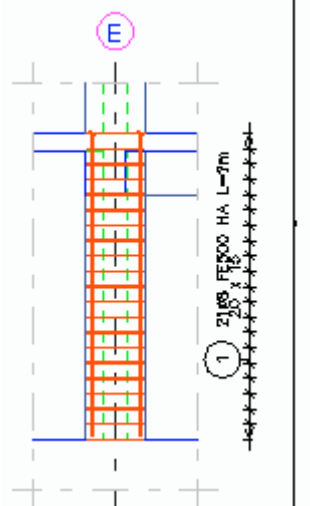


Figure 392: Bar bending detail

Note: The bending detail displays the bar full length.

The bending detail annotation creation starts automatically.

6. On the Smartbar, select the bending detail symbol: **Romania > Ro PC Nr Gr Qt Diam Lg**.

The command line automatically displays the following message: Point or [restricted Positioning/Rotation/Scale/Align/Finish].

7. To set the symbol scale:
- Type **S** (Scale) and press **Enter**.
 - Enter **2** for the symbol scale.
8. To align the symbol, type **A** and press **Enter**.
9. Place the symbol next to the bar bending detail.

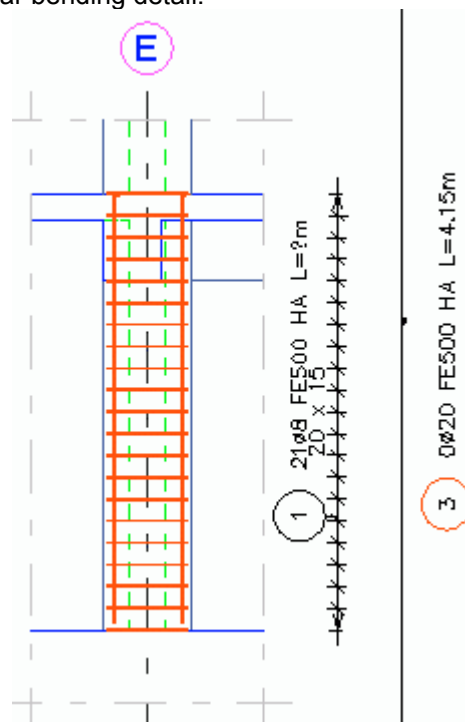


Figure 393: Face elevation 1 - placing the symbol

Using the same process, create a bar bending detail on the left bar of the Face elevation 1 cut.

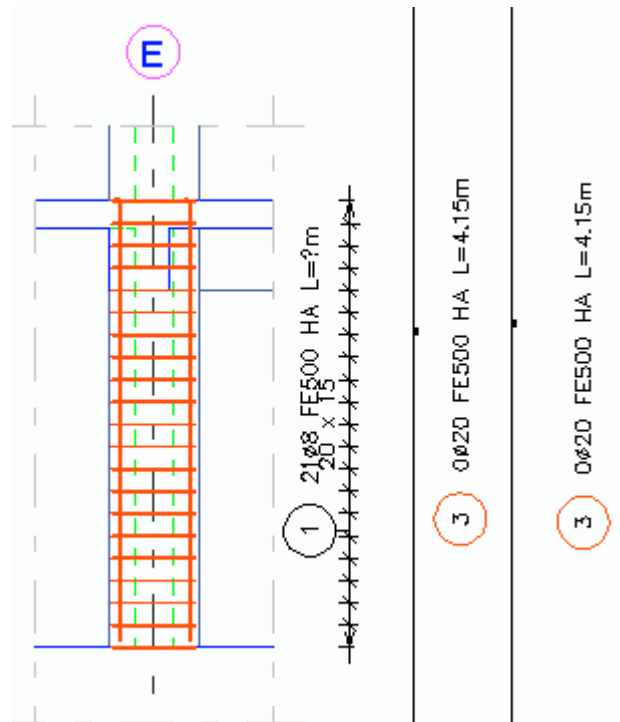


Figure 394: Face elevation 1 - Two bending details

Create a dimension by point on the bar bending detail

Note: The bending detail dimension starts automatically.

Create a dimension by point on the bar bending detail by selecting, as the first point, the previous created bar bending detail.

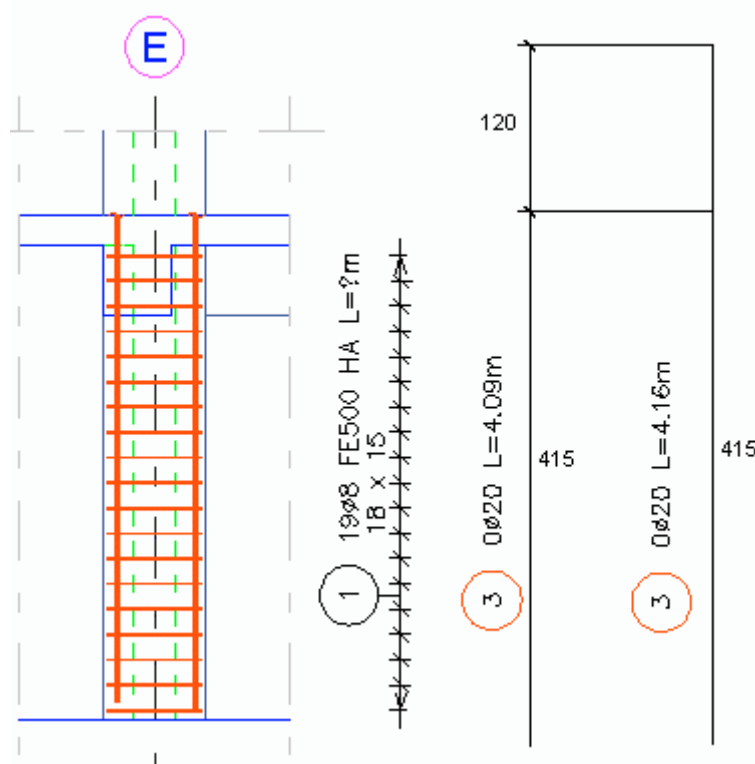


Figure 395: Face elevation 1 - Point by point dimension

Using the same process, create a bending detail for the rectangular frame on the top view. Using the properties dialog box, set the scale to **1** and select the **Unfold** representation. Place the bending detail next to the Top view cut.

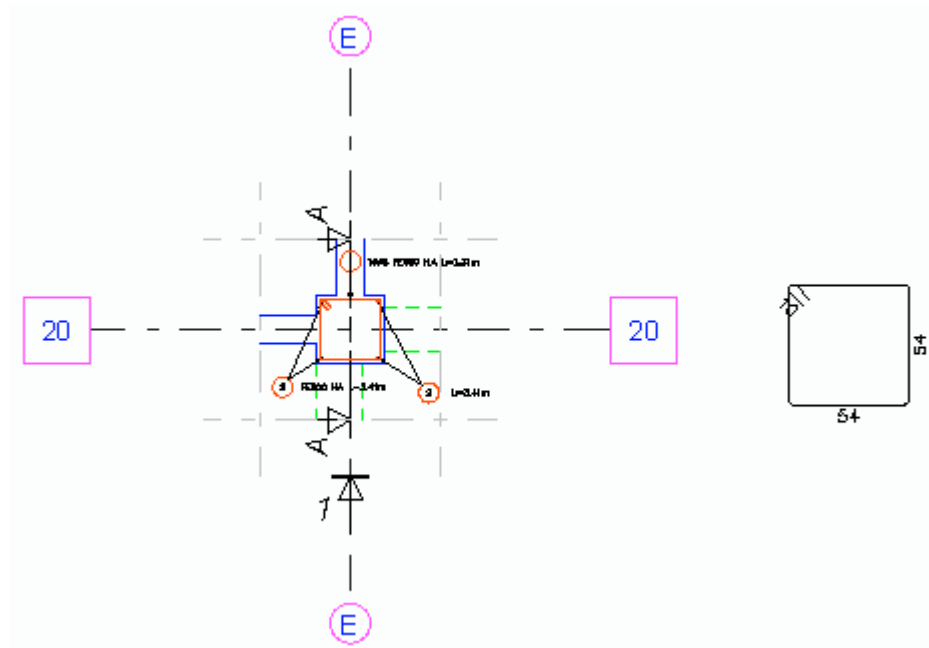


Figure 396: Top view - Bar bending detail

Additionally, create a bending detail annotation.

 **Note:** In our example, the annotation creation starts automatically.

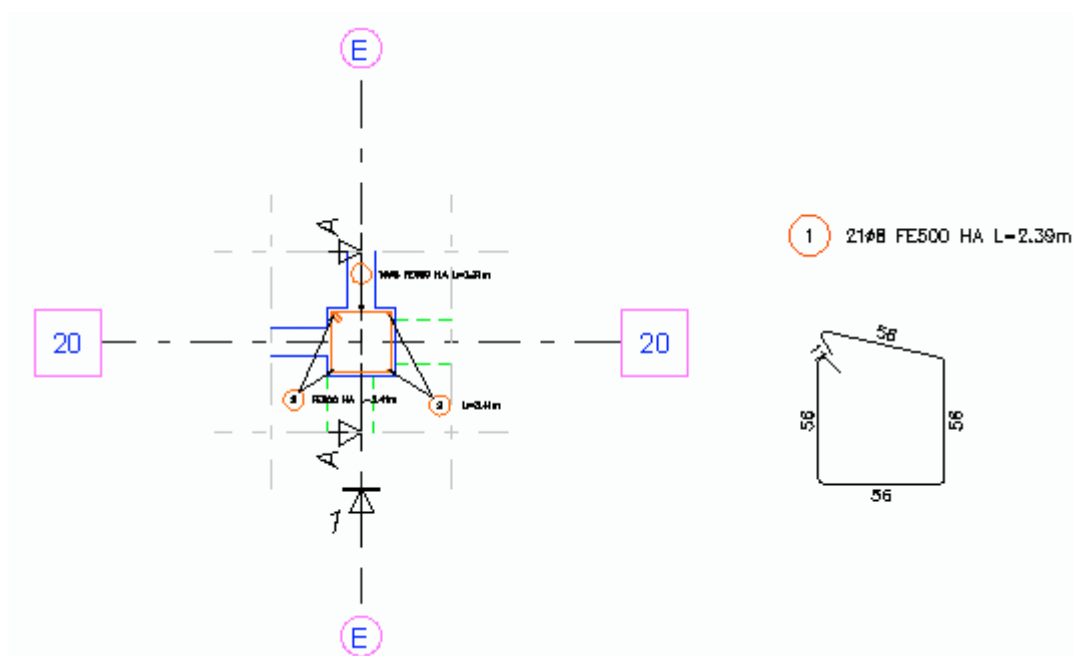


Figure 397: Top view - Bar bending detail symbol

Stretching the bar bending detail

1. Double click the bending detail to display all grip points.

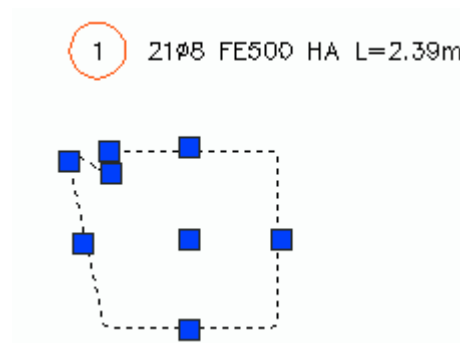


Figure 398: Bar bending detail – Grip points

2. Click a grip point of the bar bending detail and extend it as shown in Figure 399.

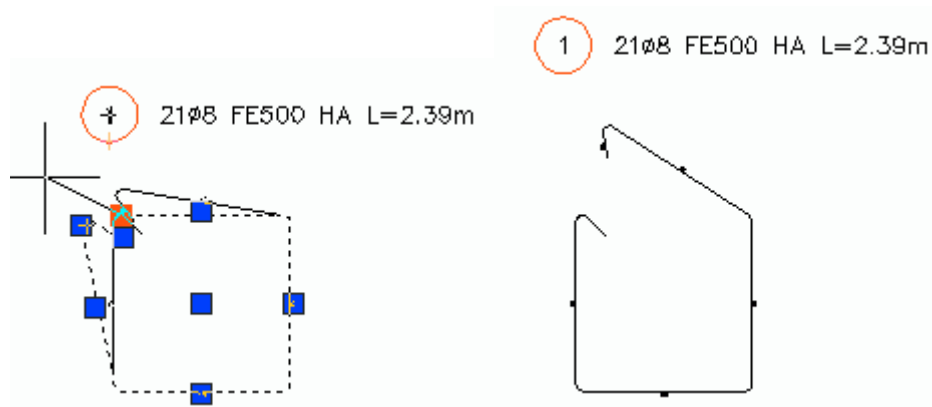


Figure 399: Stretching the bar bending detail

3. Press **Esc** to finish.

Adjusting the segment length displayed in the bar bending detail

1. In the Pilot, in **Drawings** mode, right click **Project** and select **Project settings** from the context menu.
2. On the **Project Settings** toolbar, click .
3. In the **Heavy** category, set the **Leg tolerance** to 10 mm.

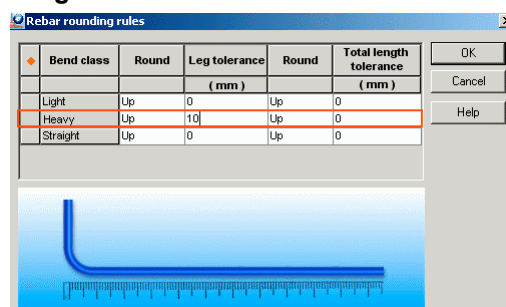


Figure 400: "Rebar rounding rules" dialog box

4. Click **OK**.

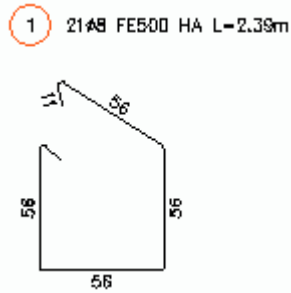


Figure 401: Bar bending detail – rounded lengths

Step 11: Create a list

1. On the **Reinforcement** toolbar, **Lists** flyout, click  to create the list.
2. On the command line, enter **b** (Bar) to take the bars into account in the list and press **Enter**.
3. On the command line, enter **v** (View) to select all the bars in the current view and press **Enter**.
4. Click a point on the drawing to place the list.

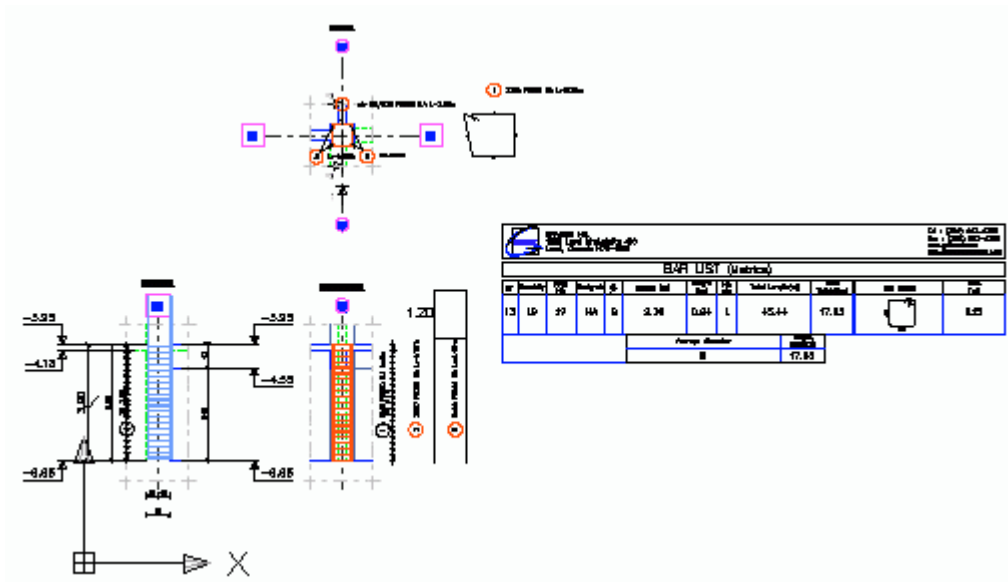


Figure 402: Reinforcement drawing

Modifying the list

Perform the following steps to modify the text dimension on the bending detail and to sort the list according to position number.

1. Select the list.

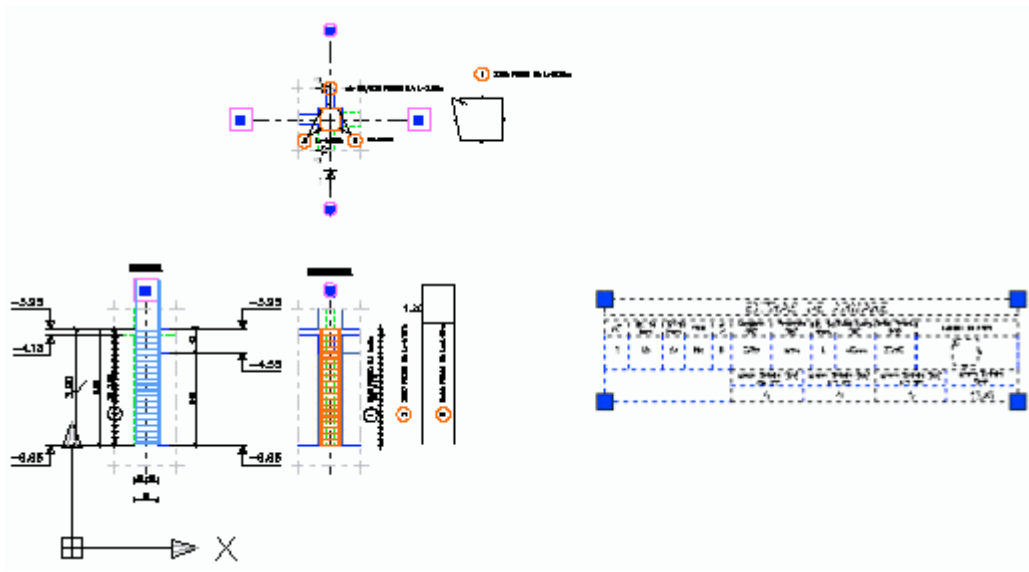


Figure 403: Selecting the list

- On the Smartbar, in **Ht. bending text**, set the text size to **0.075 m**.
- Click to access the properties dialog box.
- On the **Definition** tab, from the **Sorting column** drop-down list, select the **PositionNum** sorting method.

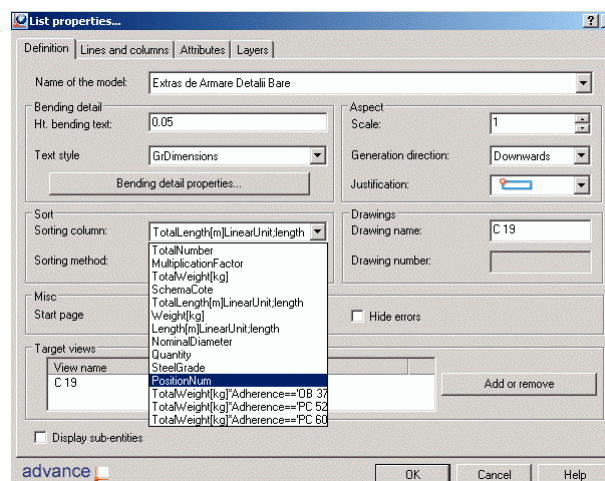


Figure 404: "List properties" dialog box - Definition tab

- Click **OK**.

GRAITEC INC.
1895 Level Blvd Suite 410
Laval, Canada H7S 2M2

Tel : (+1) 514 883-1363
Fax : (+1) 514 883-2325
www.graitec.com
info@graitex.com

BAR LIST (Metrics)

Nr	Quantity	Total No.	Steel grade	Ø	Length (m)	Weight (kg)	No. of bars	Total Length(m)	Total Weight(kg)	Bar Bends	Use (m)
1	21	38	HA	8	2.39	0.94	1	90.89	35.90		0.15
2	6	0	HA	20	4.09	10.10	0	0.00	0.00		
3	8	0	HA	20	4.18	10.25	0	0.00	0.00		
Average diameter									Weight Sum(kg)		
8									35.9		

Figure 405: List

Step 12: View the drawing in 3D

Using the **Reinforcement 3D Viewer** toolbar, you can view the reinforcement 3D drawings.



Figure 406: "Reinforcement 3D" viewer

1. On the **Reinforcement 3D** viewer toolbar, click  to start the **3D viewer**.
2. Include all the views of the reinforcement drawing in a window selection.

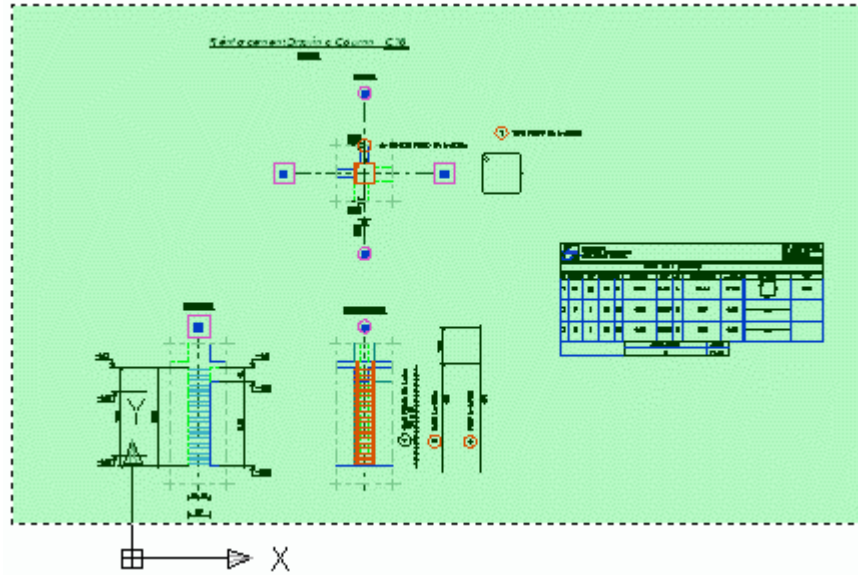


Figure 407: Selecting the reinforcement drawing

3. Press **Enter**.
- The 3D viewer appears.

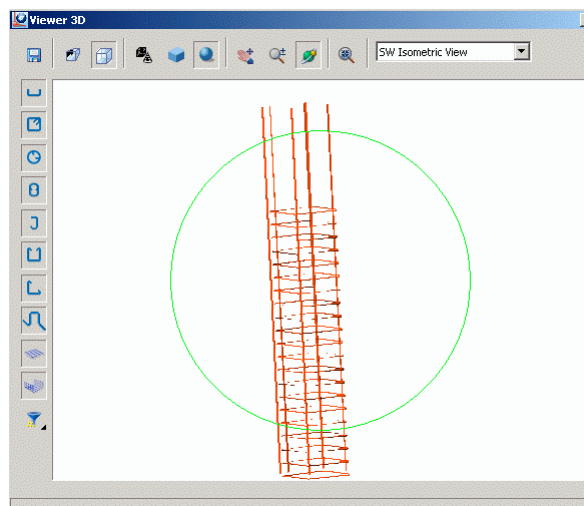


Figure 408: "Viewer 3D" dialog box

Click in the drawing area to view the reinforcement in different styles. From the **Isometric view** drop-down list, select **Top view** for a section view.

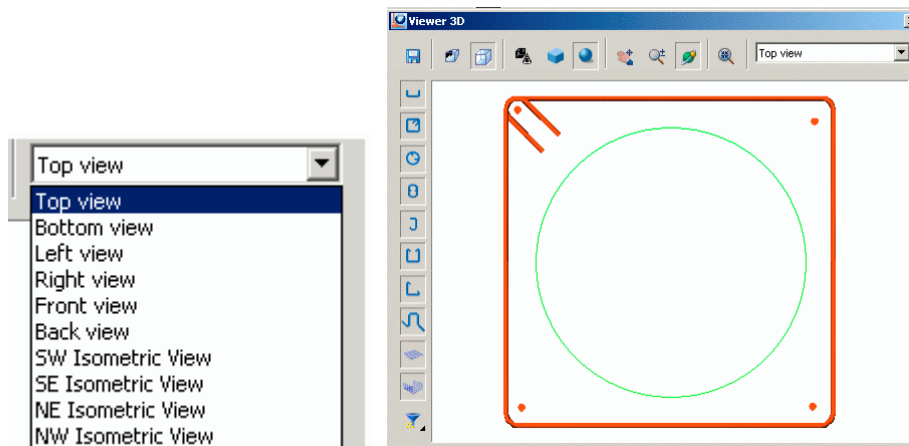


Figure 409: Selecting a predefined view

Saving the drawing

1. In the “Viewer 3D” dialog box, click .
2. In the “View name” dialog box, type the name: **Reinforcement Frame-Column C16**.

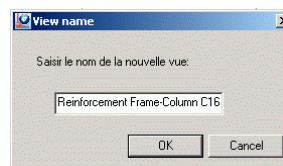


Figure 410: Setting a name for the view

3. Click **OK**.

The drawing is saved and displayed in the Pilot, in the **New drawings** folder.

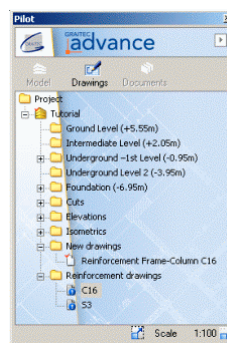


Figure 411: The Pilot displays the **C16** reinforcement drawing

Step 13: Group the 3D Reinforcement view with the Reinforcement column drawing

1. In the Pilot, in **Drawings** mode, in the **New drawings** folder, select the Reinforcement Frame drawing.

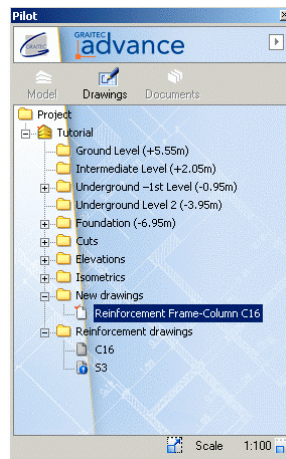


Figure 412: Selecting and grouping the **C16** reinforcement drawing

2. Using the mouse, drag the drawing in the **C16** reinforcement drawing.
3. Place the Reinforcement drawing frame on the **C16** drawing.

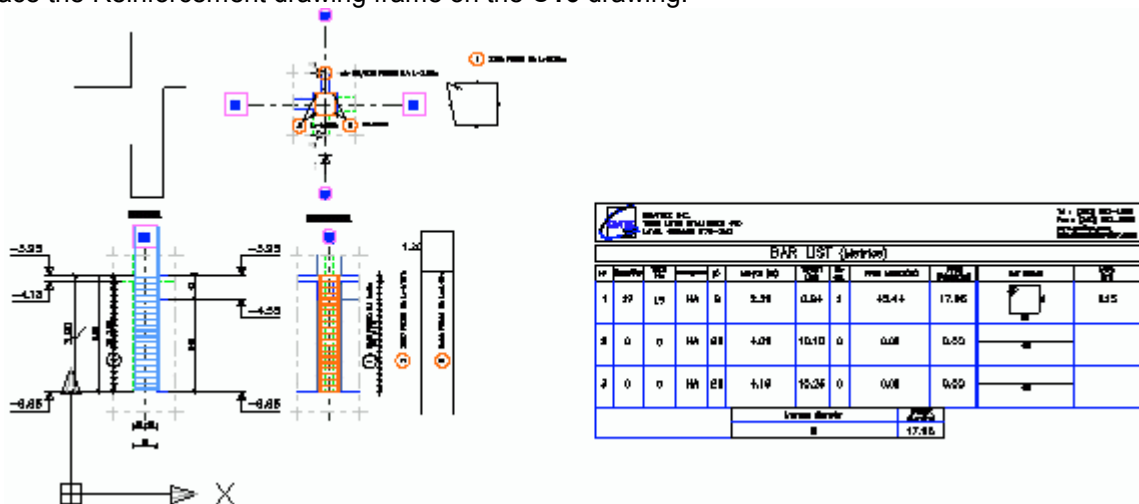


Figure 413: Placing the **C16** reinforcement drawing

The reinforcement drawing is grouped with the **C16** drawing.

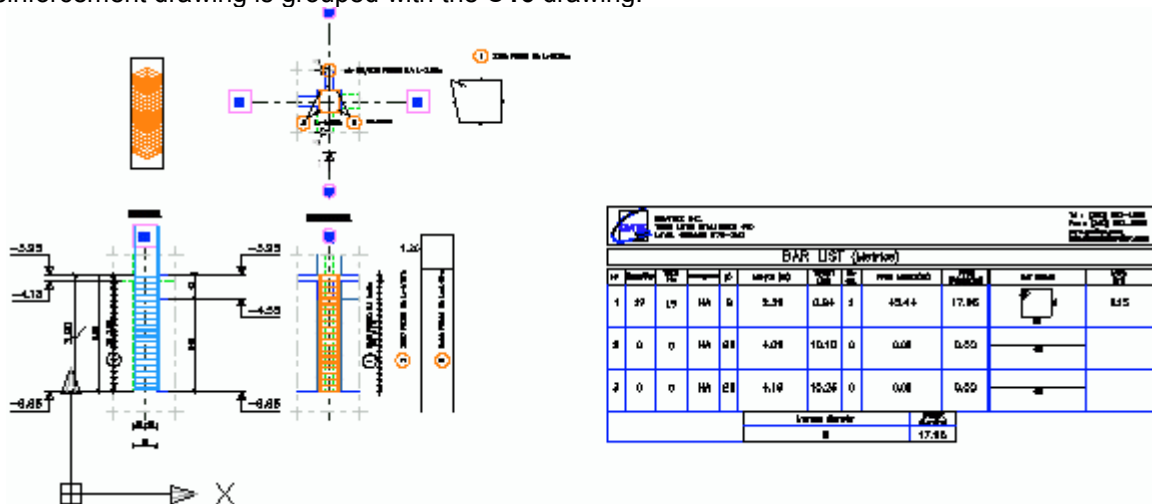


Figure 414: Reinforcement drawing

Lesson 14: Reinforcing a beam

Using **Advance**, you can create reinforcement drawings starting from a selection of elements. Depending on the selected view arrangement type, **Advance** creates cuts and elevations.

In this lesson, you will create the reinforcement drawing for a beam.

You will learn how to:

- Create a custom view arrangement.
- Create a bar bending detail.
- Create a free distribution.
- Create a rectangular frame.

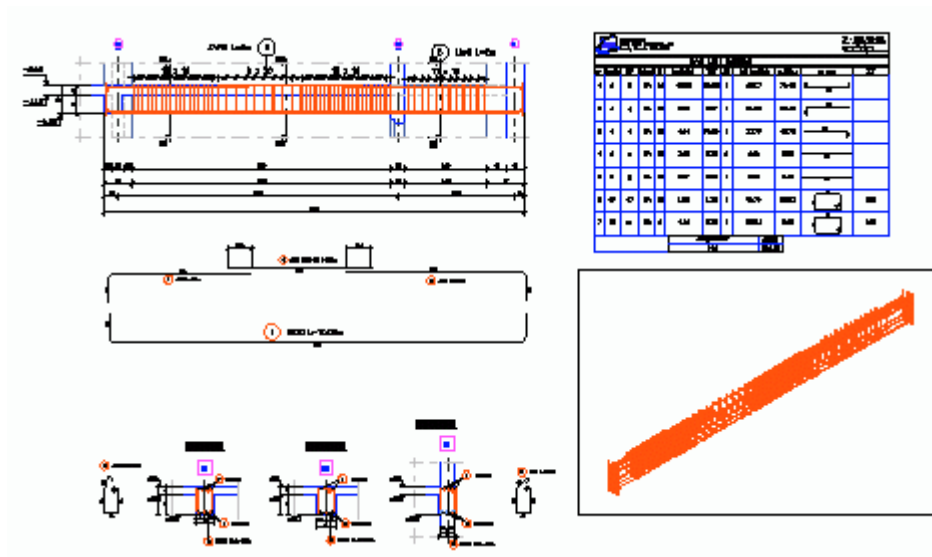


Figure 415: Final drawing of the beam reinforcement

Step 1: Activate the level

1. In the Pilot, click  to access the **Model** mode.
2. Right click **Underground Level 2** and select **Enable** from the context menu to activate the level.

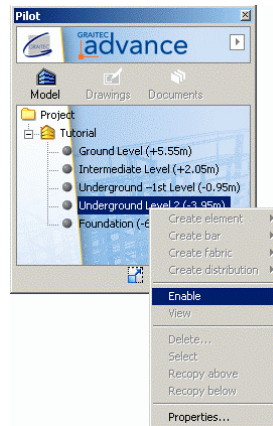


Figure 416: Activating **Underground Level 2**

Step 2: Create a custom view arrangement

The beam reinforcement drawing requires a custom view arrangement. In this step, you will create a custom view arrangement, with 3 cut sections, for the beam reinforcement drawing

For a better understanding, on the **Reinforcement** toolbar, **Drawings** flyout, click  to display the first dialog box of the reinforcement drawing creation wizard.

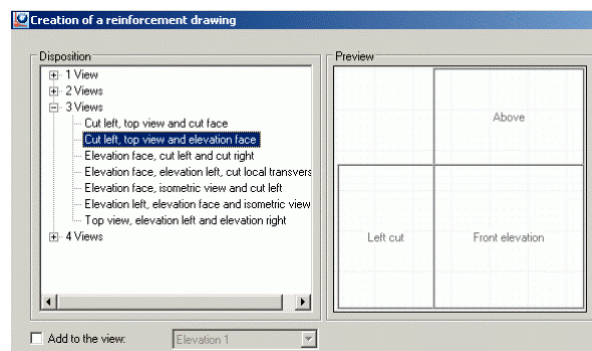
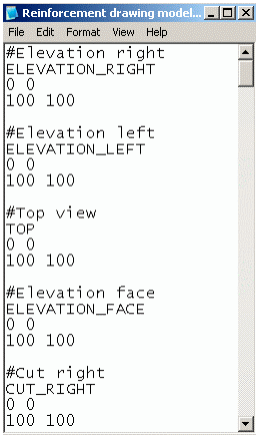


Figure 417: "Creation of a reinforcement drawing" dialog box – **3 Views** branch

The templates listed in the left panel of the dialog box are defined in the **Reinforcement drawing models.txt** file stored in the Program files\Graitec\Advance Concrete 8.1\Support\Template\ folder.

All the reinforcement drawing models are in the text file and marked with #. For each template, the view port frames are defined in percentage.



TOP
BOTTOM
CUT_LEFT
CUT_RIGHT
ELEVATION_LEFT
ELEVATION_RIGHT
ELEVATION_FACE

Figure 418: List of the view port names

For example, select **Dynamic reinforcement: left, front, top, isometric**. This template contains four views.

#Dynamic reinforcement: left, front, top, isometric
ELEVATION_LEFT
0 0
50 50
ELEVATION_FACE
0 50
50 100
TOP
50 0
100 50
ISOMETRIC
50 50
100 100

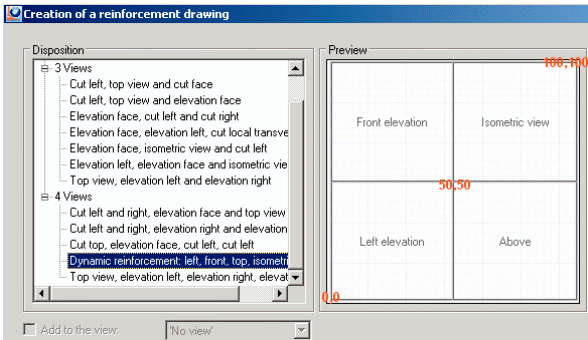


Figure 419: Dynamic reinforcement template

3. Open the **Reinforcement drawing Models** text file.

4. In the text file, enter the name of the new template, **#Reinforcement view for beams with 3 cut sections**, and the following coordinates (in percentage):

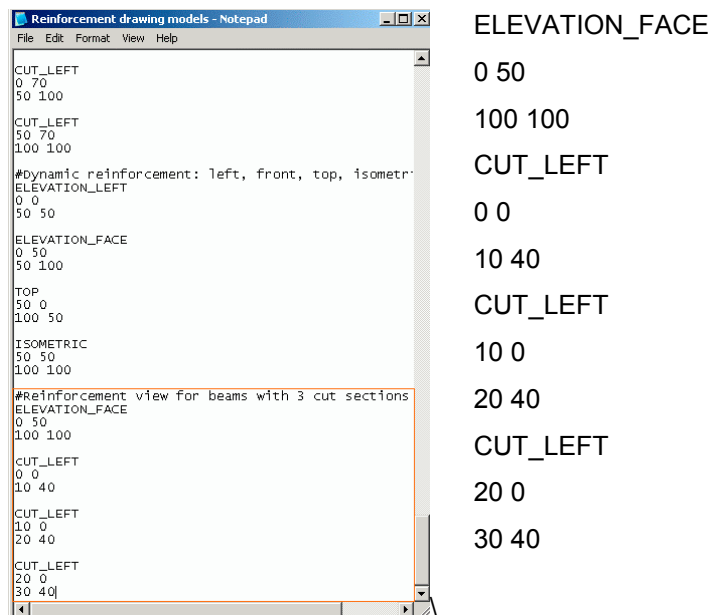


Figure 420: Reinforcement drawing model text file

5. Save and close the text file.

Step 3: Create the reinforcement drawing

1. Select the **F'20-E20** beam.

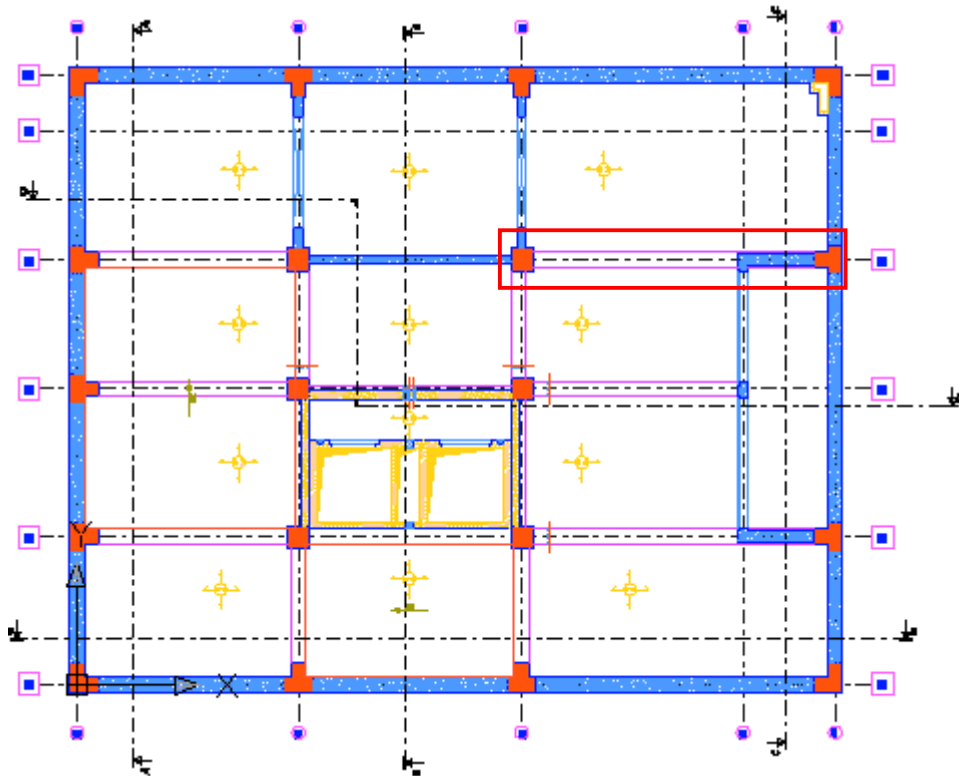
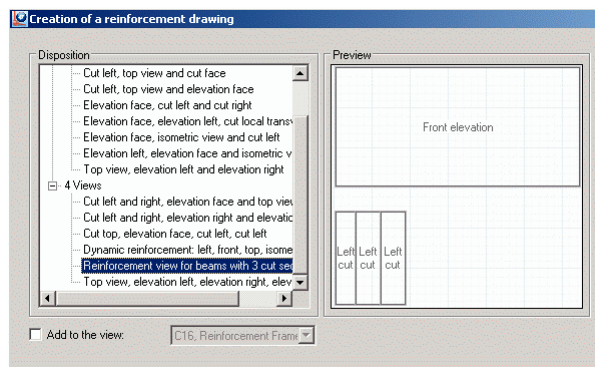


Figure 421: Selecting the beam

2. On the **Reinforcement** toolbar, **Drawings** flyout, click .
3. In the “Disposition” area, expand the **4 Views** branch, and select the **Reinforcement view for beams with 3 cut sections** template created in the previous step.

Figure 422: “Creation of a reinforcement drawing” dialog box – **4 Views** branch

4. Click **Next** to go to the “View parameters” dialog box.

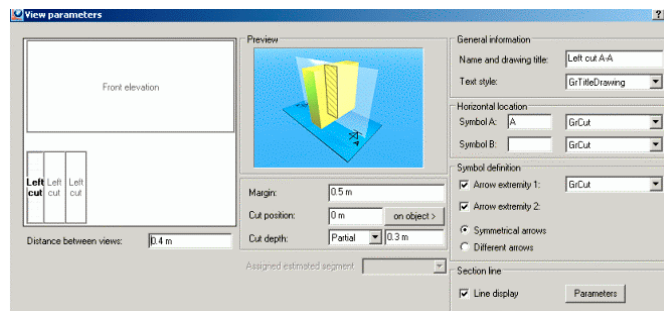


Figure 423: “View parameters” dialog box

5. Next, set the cut position. Click the first left cut.

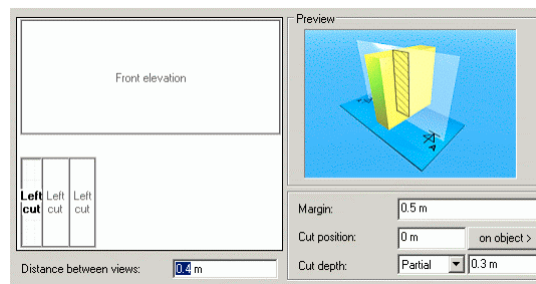
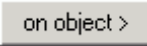


Figure 424: Adjusting the left cut

6. Click the **on object >** button to set the cut position on the drawing. 

7. Place the cut position on the **Face elevation 1** view.

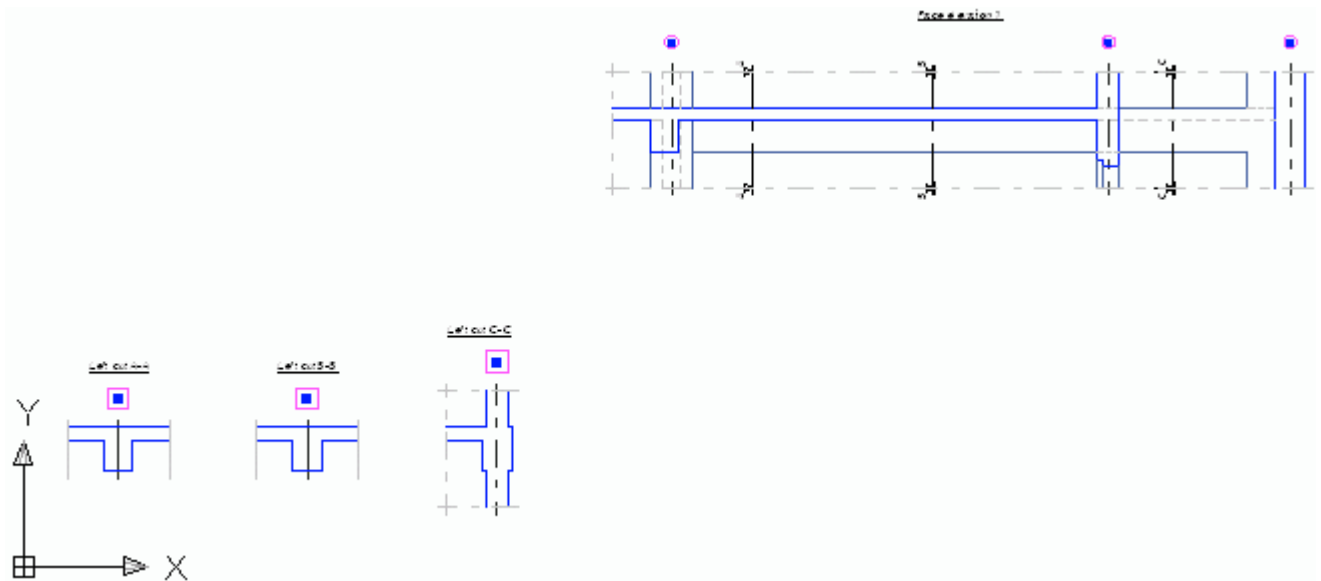


Figure 425: Placing the cut

The cut position is displayed in the **Cut position** field in the “View parameters” dialog box.

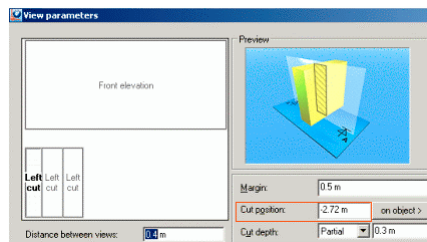


Figure 426: Setting the properties for the left cut

8. Click the third left cut and use the **on object >** button to set the cut position on the **Face elevation 1** view.

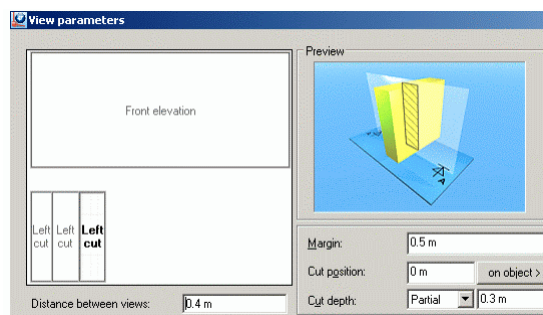


Figure 427: Setting the properties for the right left cut

9. Select the middle left cut from the “View parameters” dialog box and enter **0.5 m** in the **Cut position** field.

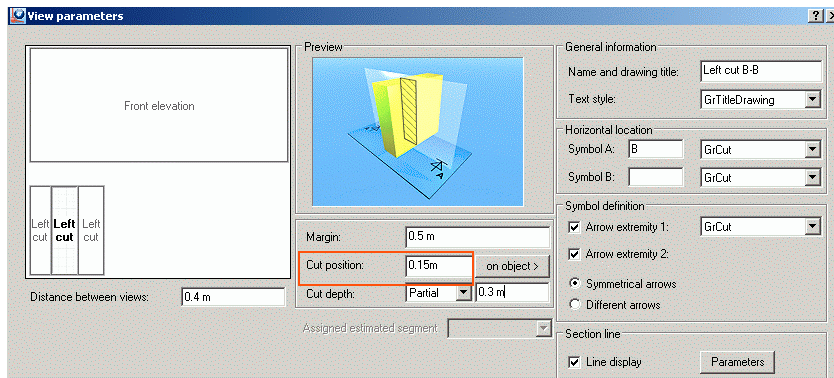


Figure 428: Setting the properties of the middle cut

The cuts are visible on Face elevation 1. The cut positions are set according to your previous options.

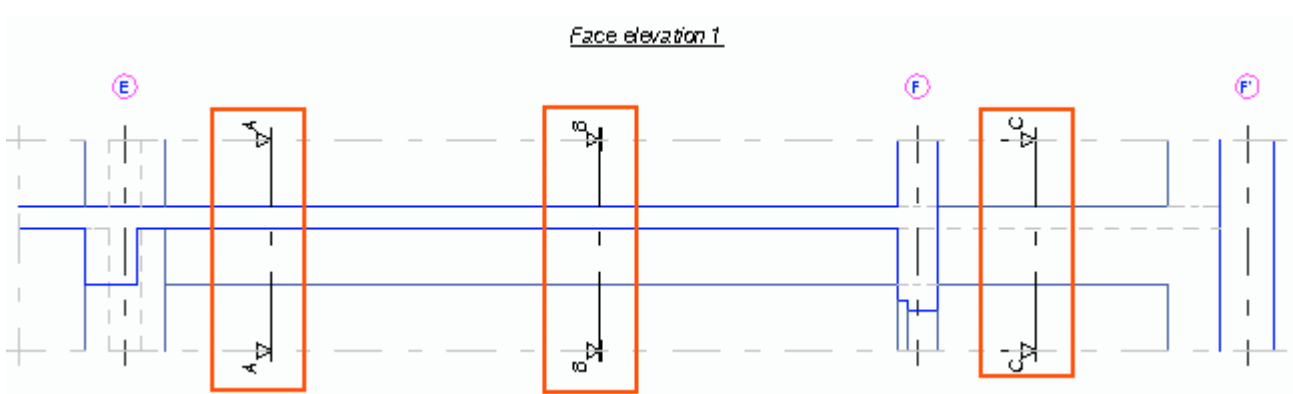


Figure 429: Face elevation 1 - Defined cuts

10. Click **Next** to go to the “Visible Objects” dialog box to set the edge style for the visible objects.
- In “Definition of the cut visible edges” area, set the **Line thickness** to **0.50 mm**.

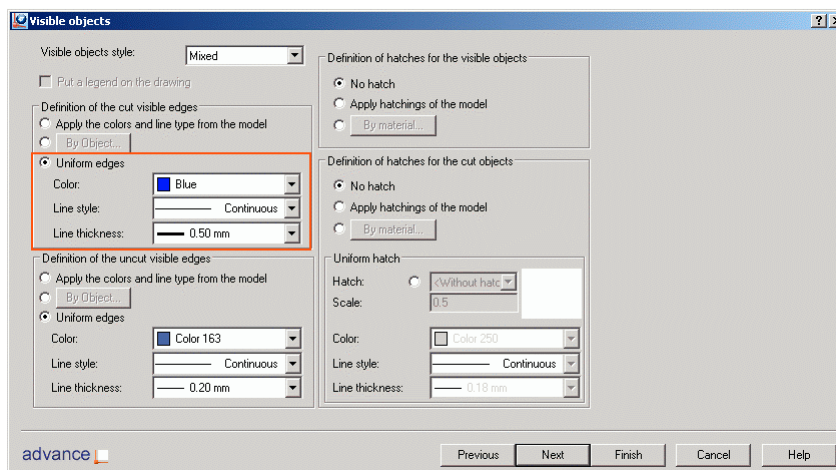


Figure 430: “Visible objects” dialog box

11. Click **Next** to go to the “Hidden objects” dialog box.

- In the “Definition of hidden objects” area, set the line thickness to **0.25 mm**.

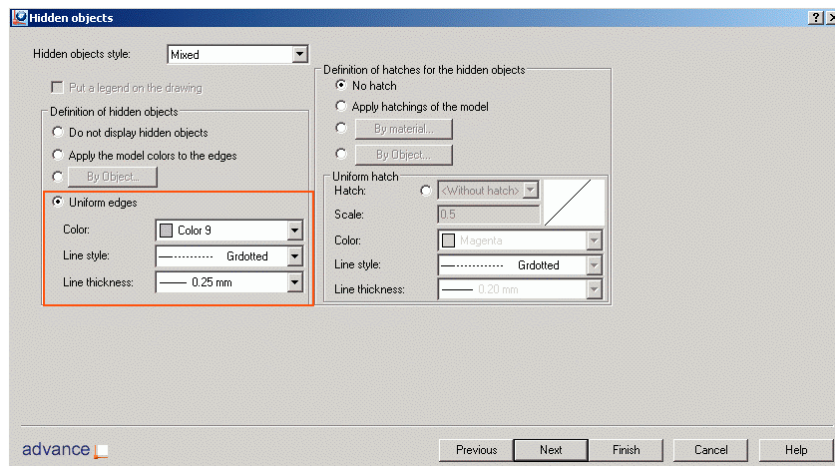


Figure 431: “Hidden objects” dialog box

12. Click **Next** to go to the “Finishings” dialog box to set the cut interruption line lengths.

- In the **Cut interruption lines** area, in the **Length** field, enter **0.1 m** for the cut line length.

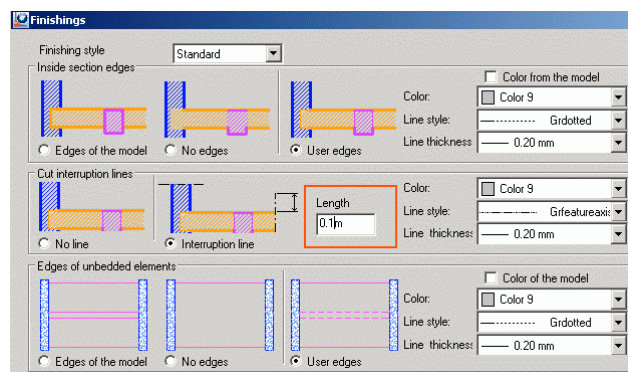


Figure 432: “Finishings” dialog box

13. Click **Next** to go to the “Visual elements” dialog box.

- Click the arrow of the **Symbol scale** drop-down list and set the drawing scale to **1:25**.

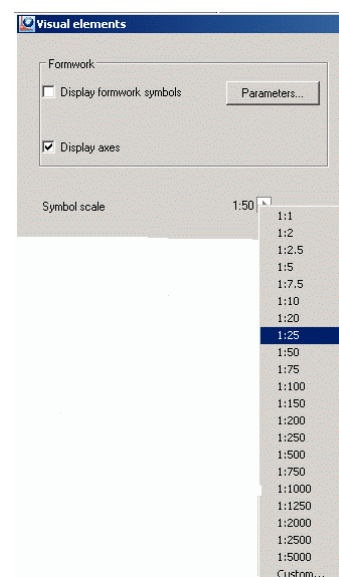


Figure 433: Selecting the symbol scale

14. Click **Next** to go to the “Title” dialog box.

- In the **Prefix** field, enter the view name prefix: **Reinforcement View-Beam**. This title appears on the drawing, above each view.
- Unselect the **Quantities** option so that the beam quantities are not included in the title.
- Select the **Length and surface** option.
- In the “Properties” area, in the **Text height** field, type **0.25 m** to set the text height.

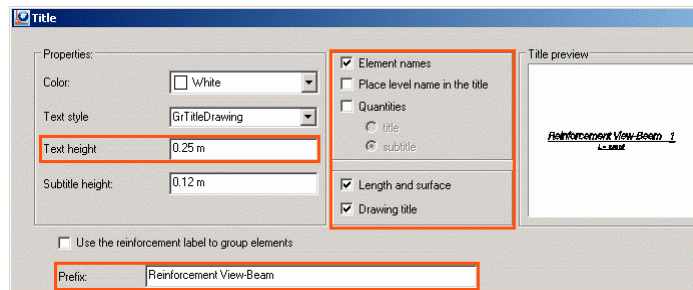


Figure 434: “Title” dialog box

15. Click **Finish**.

The Pilot switches automatically to the **Drawings** mode and the reinforcement drawing is displayed.

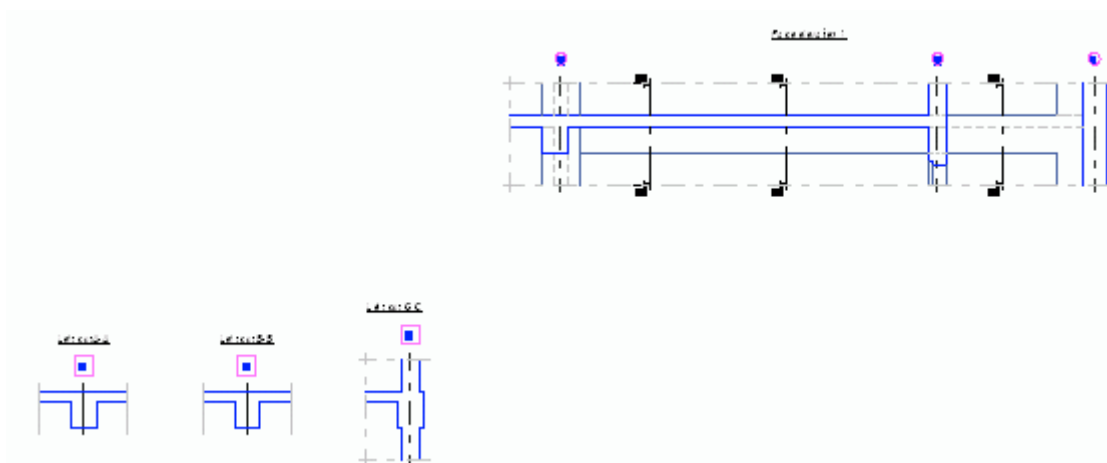


Figure 435: Reinforcement drawing

 Use the AutoCAD® **Move** command to modify the position of the cuts.

1. Include the three cuts in a window selection.

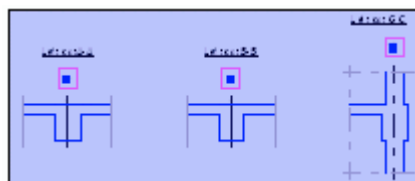


Figure 436: Selecting the cuts

2. On the command line, enter **move** and press **Enter**.
3. Click a point on the drawing to specify the displacement point.

4. Place the left cuts under the **Face elevation 1 cut**.

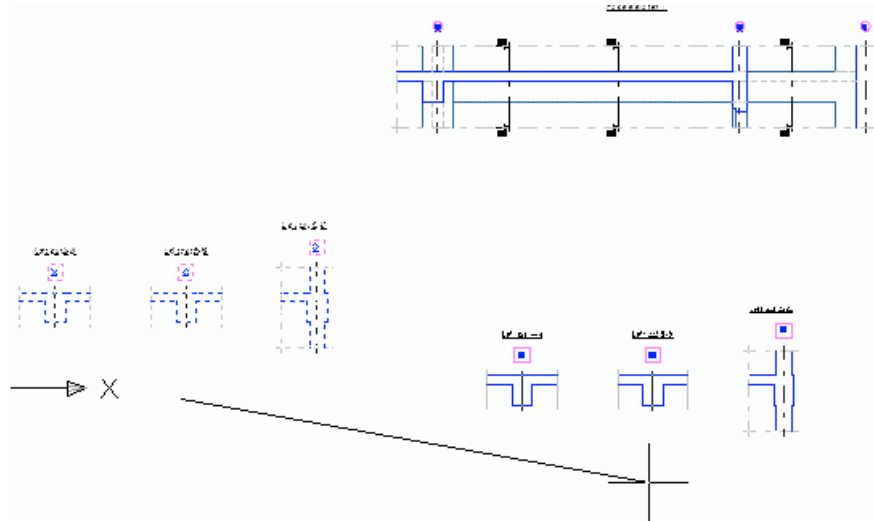


Figure 437: Moving the cuts

Using the same process, move the text in the middle of **Face elevation 1**.

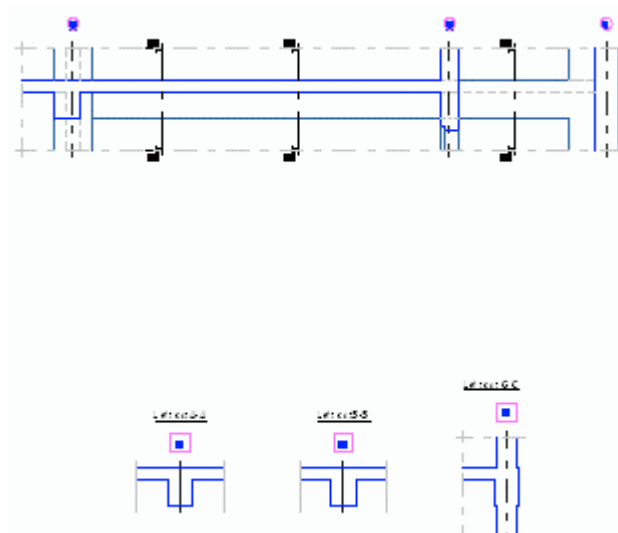
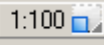


Figure 438: Adjusted reinforcement drawing

Changing the scale of the drawing

In the Pilot, in **Drawings** mode, click  **Scale**  and select the **1:25** scale from the list.

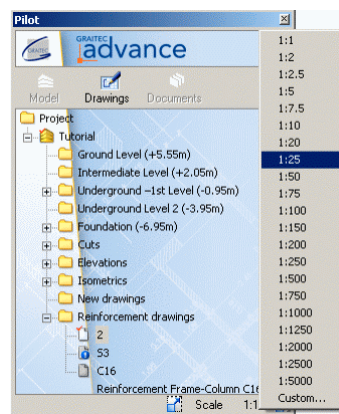


Figure 439: Setting the drawing scale

Step 4: Create all the necessary dimensions

This step briefly describes how to dimension the views of the reinforcement drawing using intersection dimensions, point by point dimensions and level dimensions.

Creating intersection dimensions

Create intersection dimensions on **Elevation face 1**. In the **Model elements** dialog box, select all elements to intersect the dimensions with all the elements of the drawing and take the grid into account.

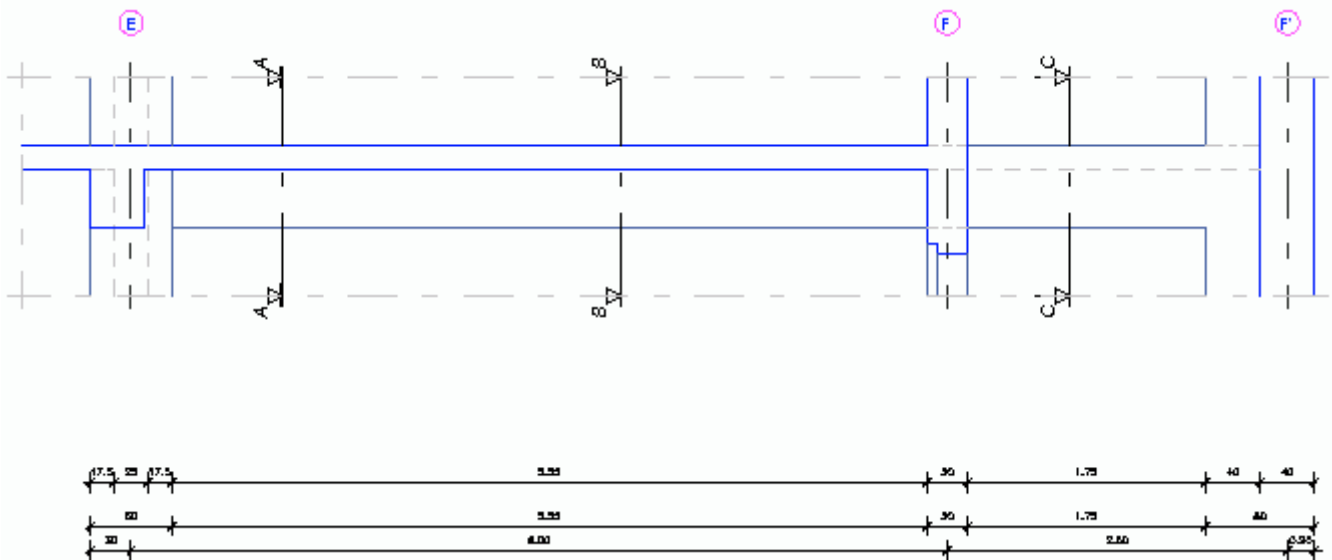


Figure 440: Elevation face 1- intersection dimensions

Creating a point by point dimension

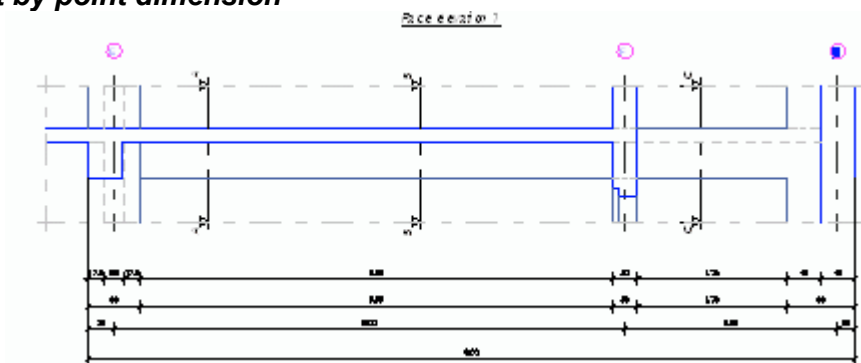


Figure 441: Face elevation 1- point by point dimension

Create intersection dimensions on the Face elevation 1 view.

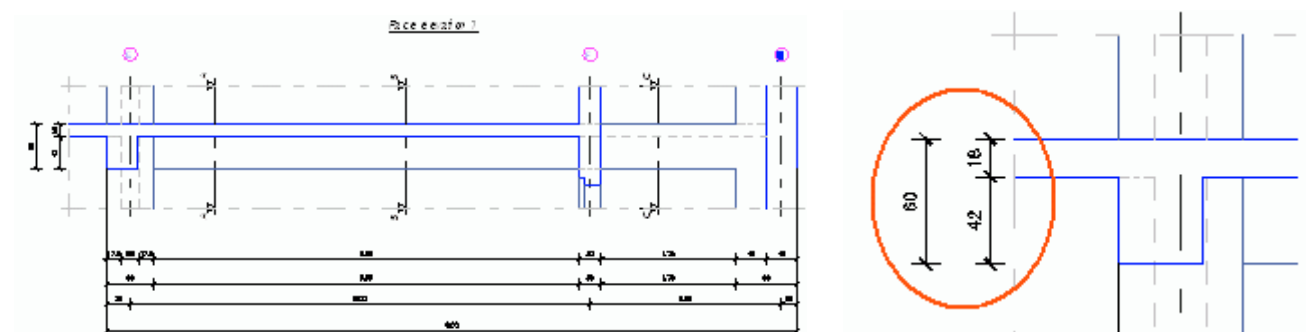


Figure 442: Face elevation 1- Intersection dimensions

Creating level dimensions

1. On the **Views** toolbar, click .
2. On the command line, enter **M** (mirror) to reverse the arrow positioning and press **Enter**.
3. Select the dimension origin by clicking the left corner of the Face elevation 1 view in this example.

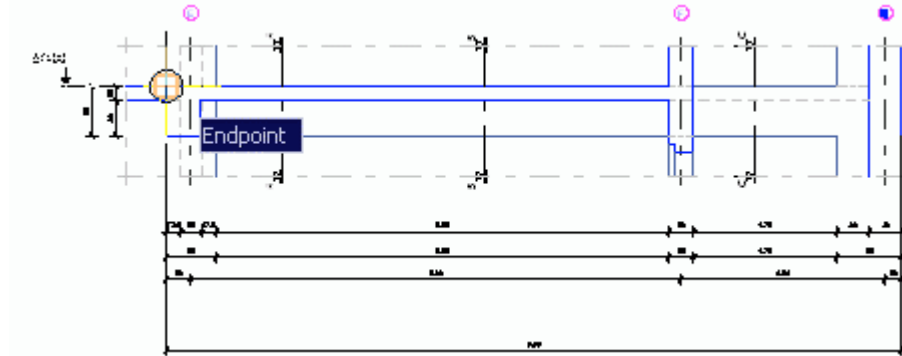


Figure 443: Elevation face 1- placing the dimension origin

4. Click a point on the drawing to create the level dimension.
5. Create level dimensions.
6. Press **Esc** to finish.

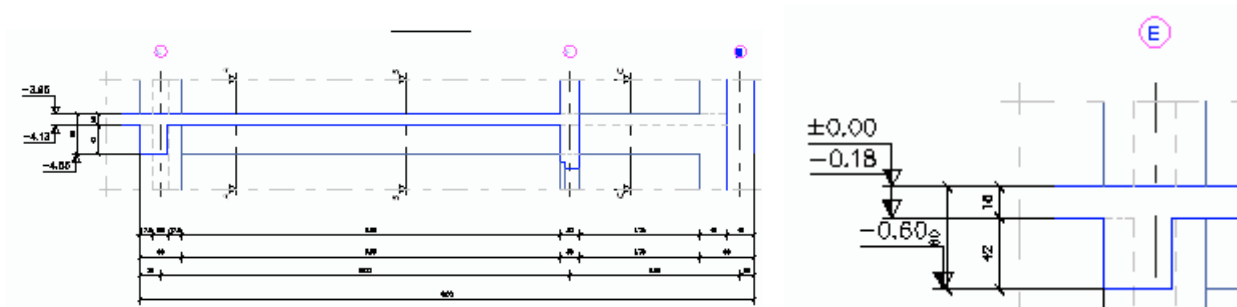


Figure 444: Face elevation 1- Level dimensions

Modifying the properties of the level dimension

1. Select the level dimension.

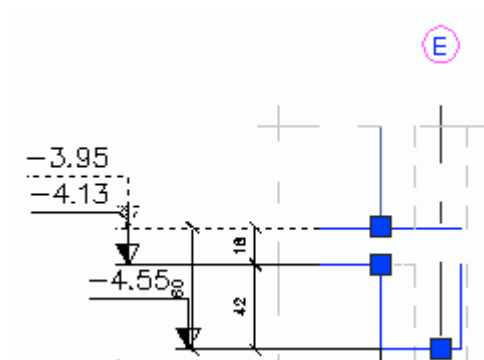


Figure 445 Elevation face 1- Selecting the level

- On the Smartbar, click to access the properties dialog box.
- On **Definition** tab, check the **Inverse** options on rows 2 and 3 to reverse the dimension.

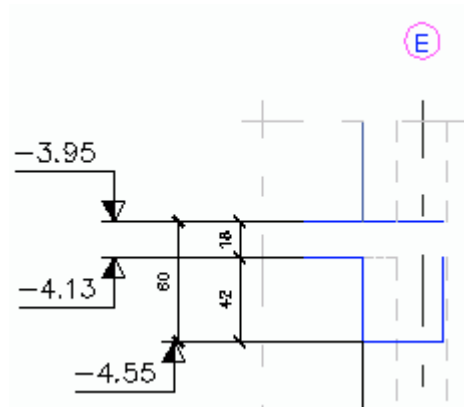
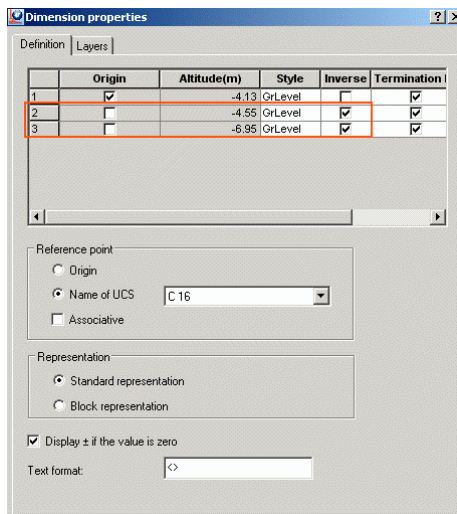


Figure 446: Reversing the level dimensions

- Click **OK**.

Using the process used for the Face elevation 1 view, create intersection dimensions on **Left cut A-A**.

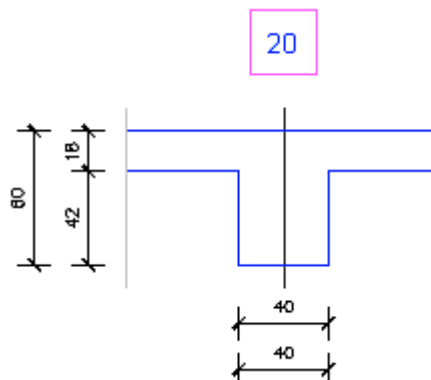


Figure 447: Left cut A-A: intersected dimension

Selecting the set of intersected elements

- Select the last created intersection dimension.

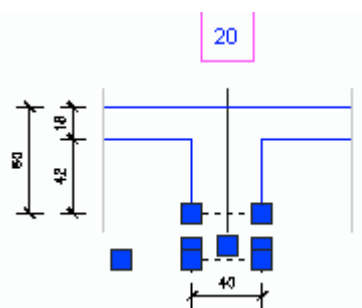


Figure 448: Left cut A-A - selecting the intersection dimension

- On the Smartbar, click to access the properties dialog box.

3. Select the **Definition** tab.
 - Expand the **Model elements** tree branch.
 - Unselect the **Grid** element.

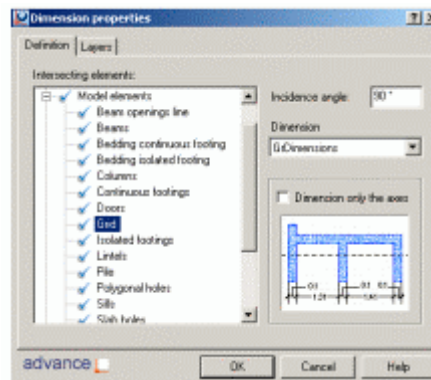


Figure 449: "Dimension properties" dialog box – **Definition** tab

4. Click **OK**.

The dimension line intersects all the drawing elements.

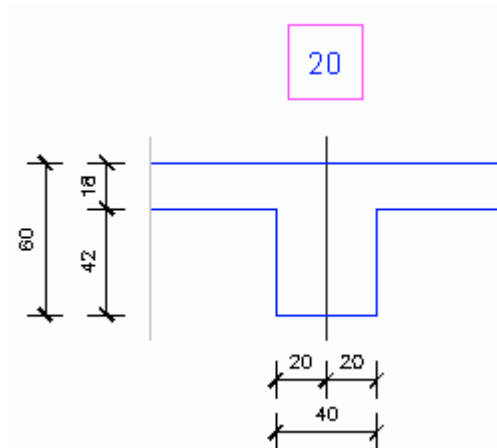


Figure 450: Left cut A-A - new representation of the dimension

Using the process used for the Elevation face 1 view, create level dimensions on **Left cut A-A**.

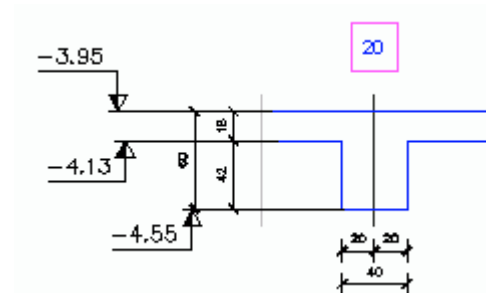


Figure 451: Left cut A-A - Level dimensions

Copying the intersection and the level dimensions of Left cut A-A to Left cut B-B

1. Include all the dimensions in a window selection.

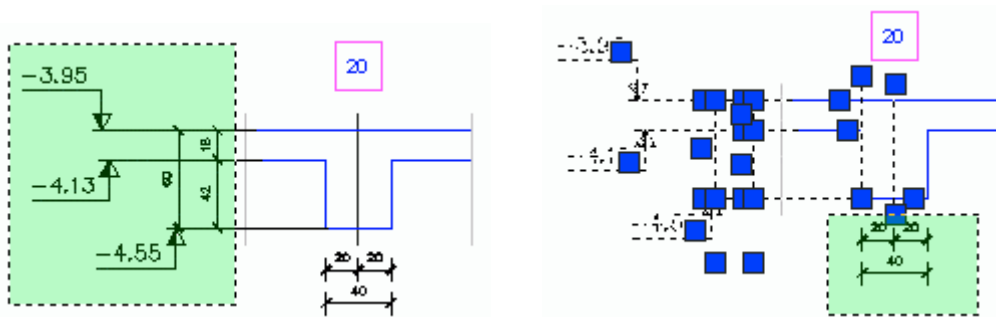


Figure 452: Left cut A-A - selecting the intersection dimension

2. Right click and select **Copy selection** from the context menu.
3. Specify a base point on **Left cut A-A** to copy the dimensions.

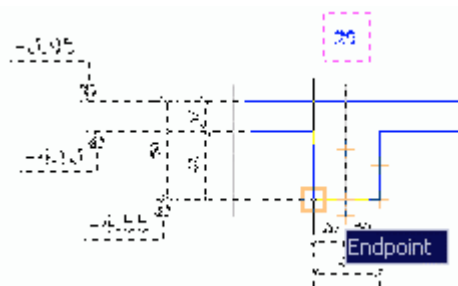


Figure 453: Left cut A-A and Left cut B-B - copying the elements to Left cut B-B

4. Position the dimensions similar to those on **Left cut A-A**.

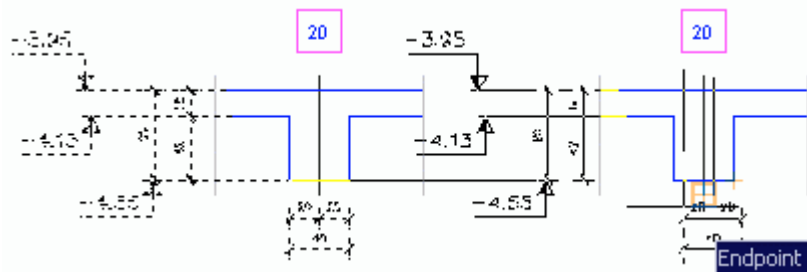


Figure 454: Left cut A-A and Left cut B-B - placing the elements

5. Copy the dimensions from **Left cut A-A** to **Left cut C-C**.

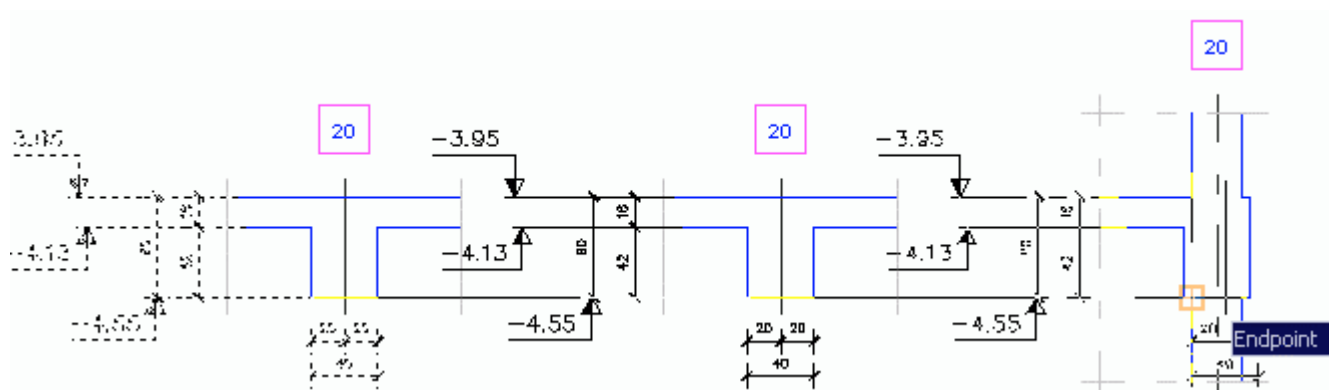


Figure 455: Left cut A-A, Left cut B-B and Left cut C-C - Copied dimensions

- Press **Esc** to finish.

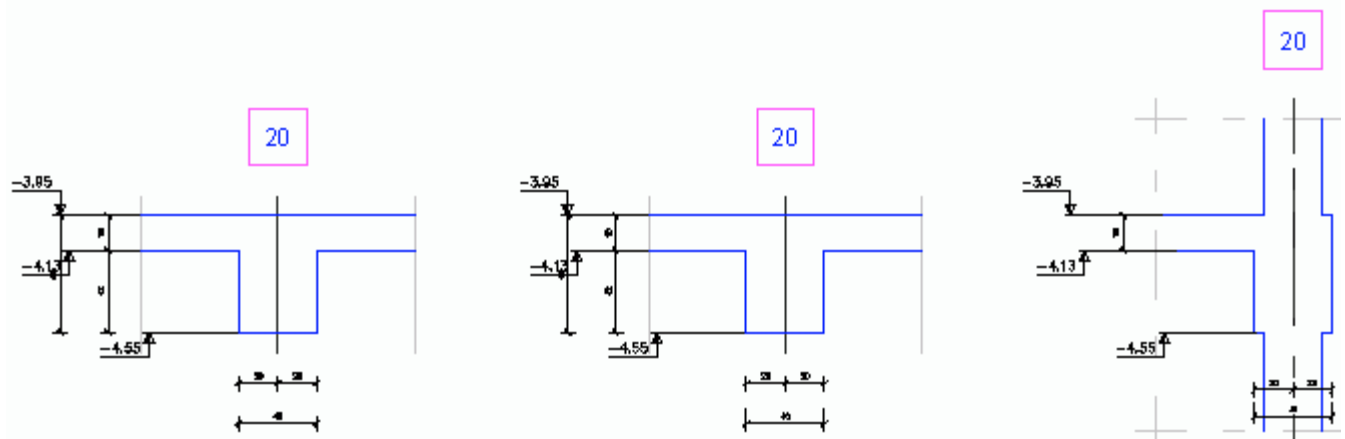


Figure 456: Left cut A-A, Left cut B-B and Left cut C-C

Recalculating the dimensions on Left cut C-C

Left cut C-C has different dimensions unlike the other views (i.e., Left cut A-A and Left cut B-B). It is necessary to recalculate the intersection dimensions of Left cut C-C to display the correct dimensions.

- Select the intersection dimension on the bottom side of **Left cut C-C**.

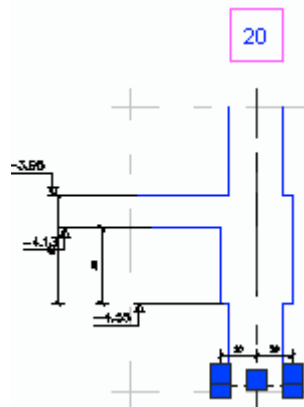


Figure 457: Left cut C-C - selecting the cut

- Right click and select **Recalculate the dimensions** from the menu.
- Press **Esc** to finish.

The dimension is recalculated and adjusted for **Left cut C-C**.

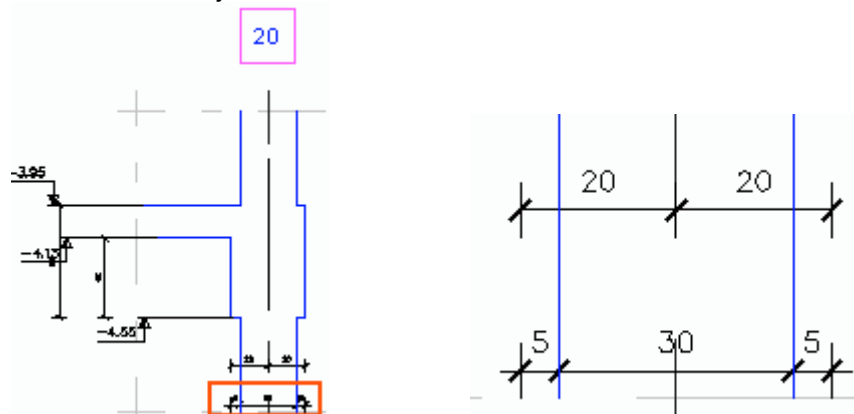


Figure 458: Left cut C-C - recalculated dimension

Create an intersection dimension at the bottom of **Left cut C-C**.

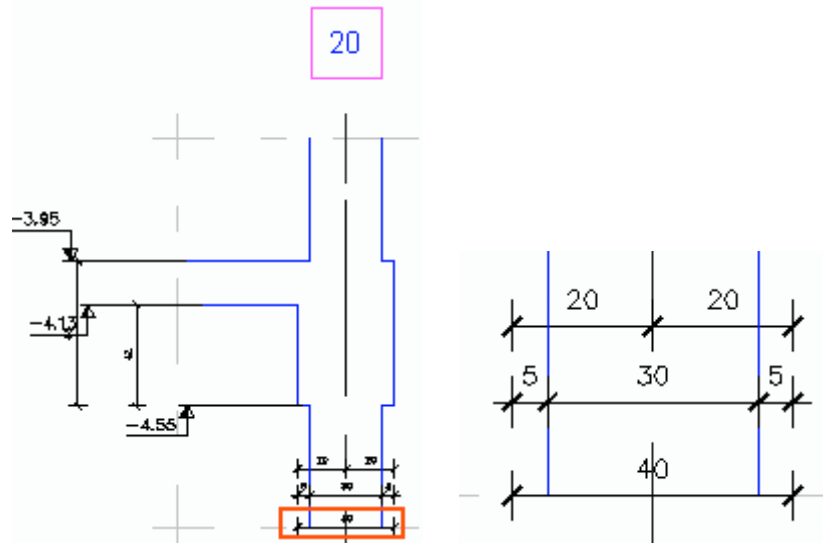


Figure 459: Left cut C-C - new intersection dimensions

Creating a reinforcement bar on the Face elevation 1 cut

- On the **Reinforcement** toolbar, **Reinforcement Bars** flyout, click .
- Click the first point of the bar on the bottom left corner of the Face elevation 1 view.

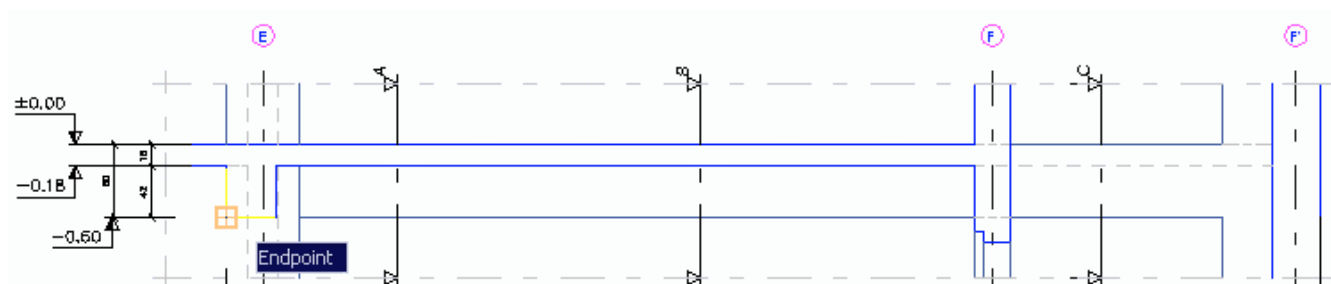
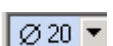


Figure 460: Elevation face 1: First point of the bar

- On the Smartbar:
 - Set the bar diameter to **20 Ø**. 
 - Click  to access the properties dialog box.
 - In the field next to the **Con. covers** drop-down list, enter **0.03 m** to set the concrete cover.

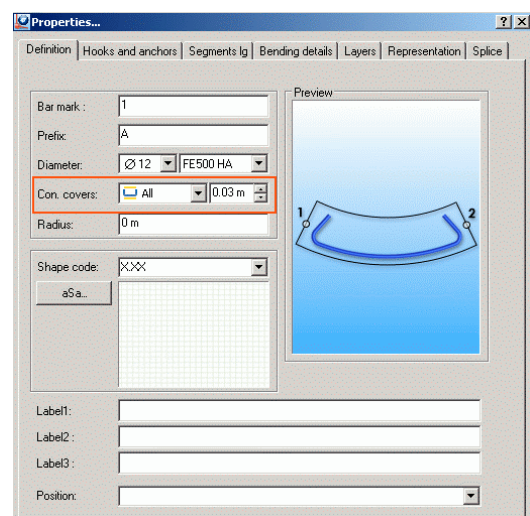


Figure 461: "Properties" dialog box – **Definition** tab

4. Select the **Hooks and anchors** tab to set hooks for the bar.
 - Unselect the **Anchor 1** option.
 - Check **Hook 1** and **Hook 2** to add hooks at both bar extremities.
 - Check **Length** and enter **0.60 m** for each hook to set the length.

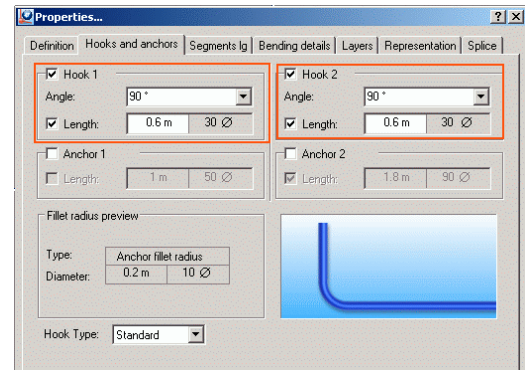


Figure 462: “Properties” dialog box – **Hooks and anchors** tab

5. Click **OK**.

Place the reinforcement bar as shown in the drawing below.

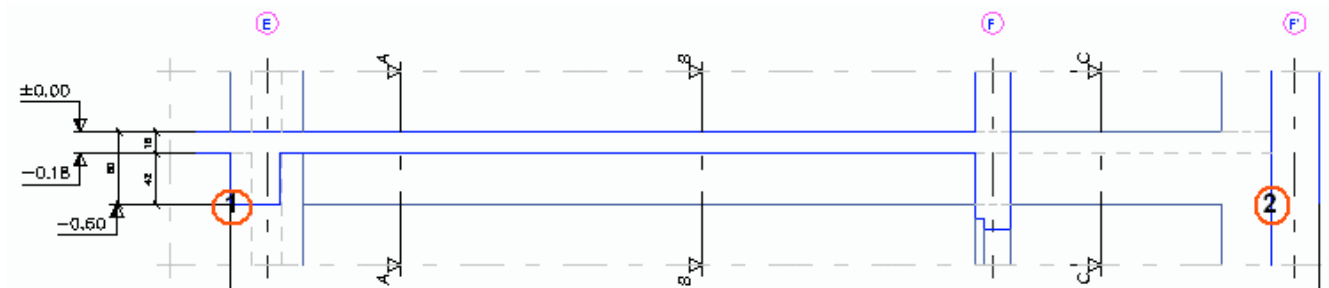


Figure 463: Elevation face 1 - first and second point for creating the bar

6. Press **Enter** to finish.

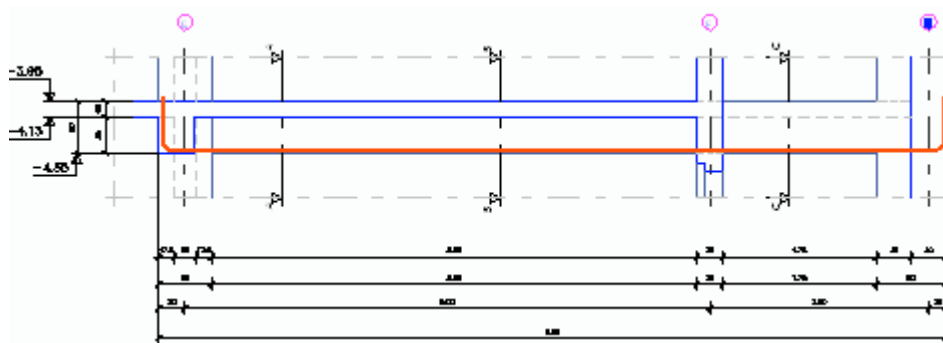


Figure 464: Elevation face 1 - Straight bar

Adjusting the hook length of the bar

1. Select the bar.
2. On the Smartbar, click  to access the properties dialog box.

3. In the properties dialog box, select the **Hooks and anchors** tab to modify the hook length.
 - In the **Length** field set the hook length to **0.5 m**.

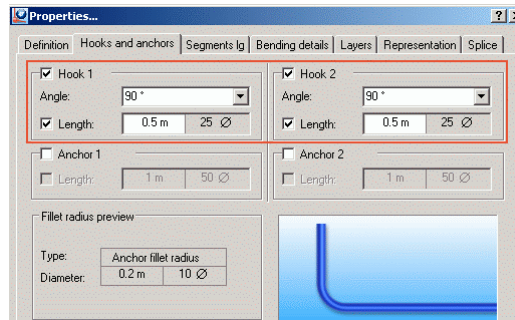


Figure 465: "Properties" dialog box – Hooks and anchors tab

4. Click **OK**.

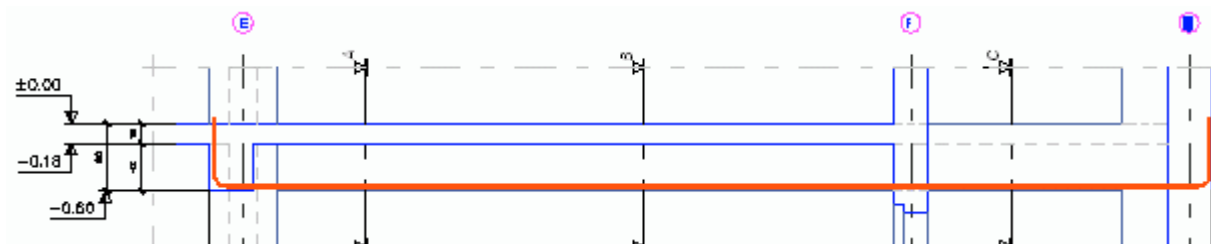


Figure 466: Face elevation 1 - Adjusted bar

Next, create two straight bars, with hook at one extremity.

1. On the **Reinforcement** toolbar, **Reinforcement Bars** flyout, click .
2. On the Smartbar, click  to access the properties dialog box.
3. In the properties dialog box, select the **Hook and anchor** tab.
4. On the **Hook and anchor** tab, define the following settings:
 - Unselect **Hook 2** to create a hook at the second extremity of the straight bar.
 - Set the length for **Anchor 2** to **1.8 m**.
5. Place the bar on the top side of the elevation.

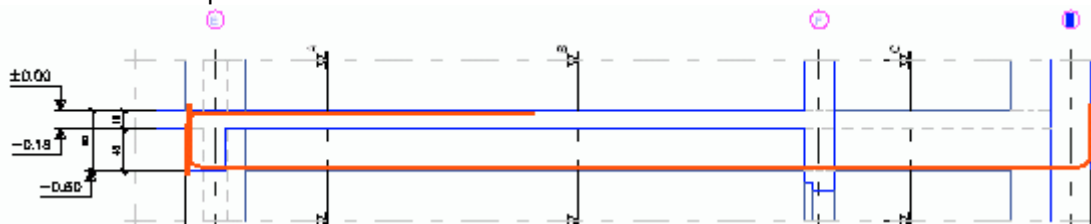


Figure 467: Elevation face 1 - First bar

Create another bar with the same properties on the left side of the elevation.

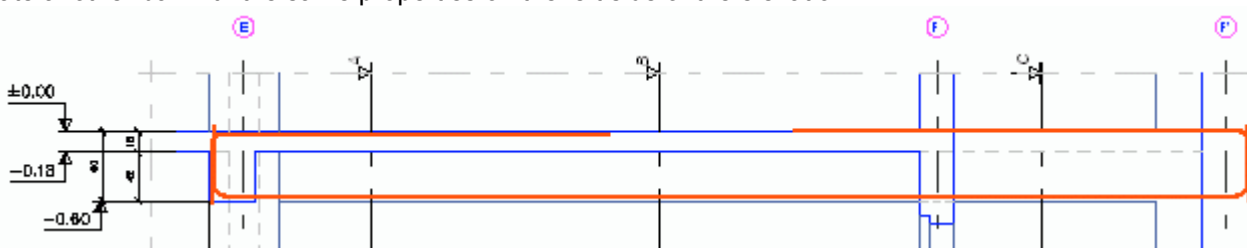


Figure 468: Elevation face 1 - Second bar

Modifying the hook length of the bar

1. Select the second created bar.

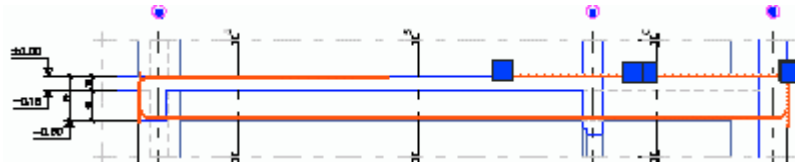


Figure 469: Elevation face 1 - Selecting the bar

2. On the Smartbar, click  to access the properties dialog box.
3. In the properties dialog box, select the **Hooks and anchors** tab.
4. Set the hook length to **0.5 m**.

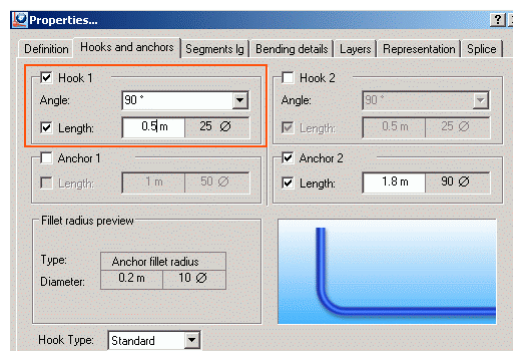


Figure 470: "Properties" dialog box - Hook and anchor tab

5. Click **OK**.

Creating a reinforcement bar anchor

1. On the **Reinforcement** toolbar, **Reinforcement Bars** flyout, click .
2. On the Smartbar, click  to access the properties dialog box.
3. On the **Definition** tab, from the **Diameter** drop-down list, set the bar diameter to **12Ø**.

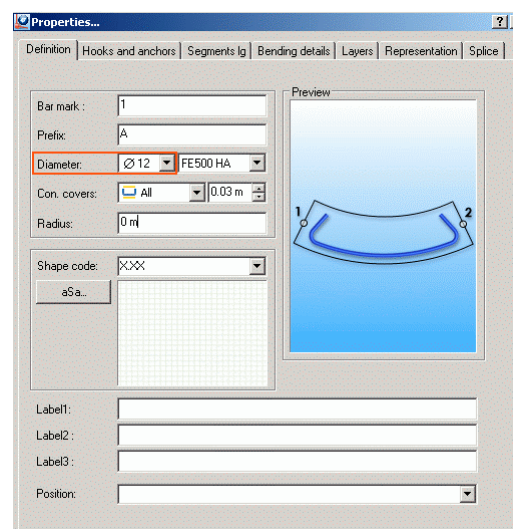


Figure 471: "Properties" dialog box – Definition tab

4. Select the **Hooks and anchors** tab to add anchors.
 - Unselect the **Hook 1** and **Hook 2** options (if checked) to remove the hooks of the bar.
 - Check the **Anchor 1** and **Length** options and set the length of the anchor to **0.50 m**.
 - Check the **Anchor 2** and **Length** options and set the length of the anchor to **0.50 m** also.

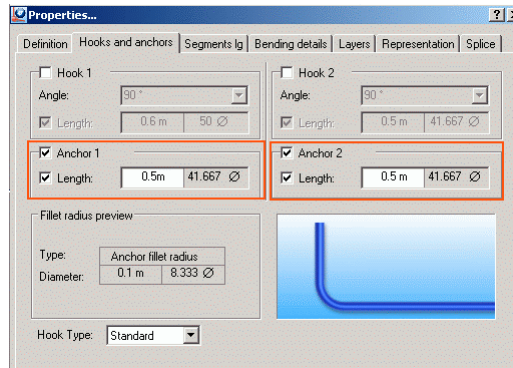


Figure 472: Bar properties – Adding anchors

5. Click **OK**.

Positioning the bar

1. Click the first point of the bar as shown in the following figure.

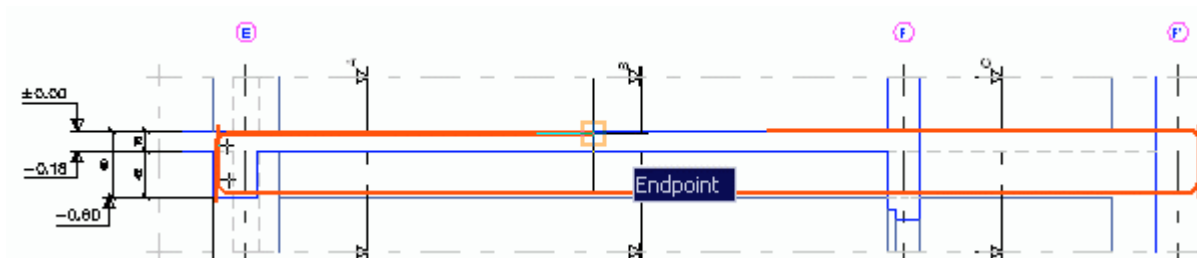


Figure 473: Elevation face 1 - First point of the bar

2. Place the second point at the end of the second created bar.
3. Set the orientation of the bar.
4. Press **Esc** to finish.

The straight bar is created.

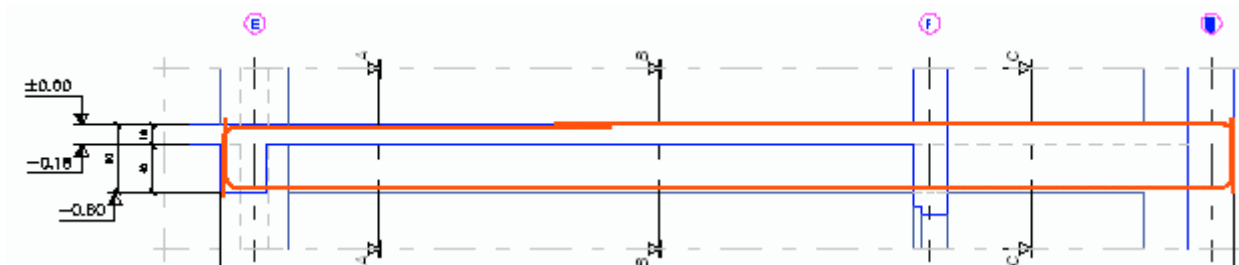


Figure 474: Elevation face 1 - Straight bar

Step 5: Create the bar bending details

Before creating the bar bending detail, set the scale of the drawing to **1:50**.

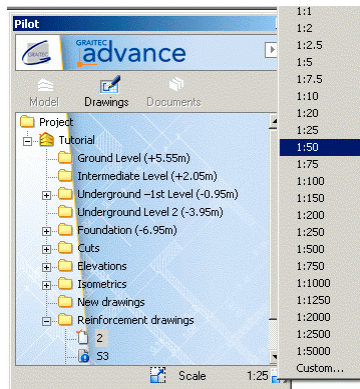


Figure 475: Setting the drawing scale

1. On the **Reinforcement** toolbar, **Distribution** flyout, click .
 2. Select the first created bar on the face elevation.
The bending detail is hooked to the cursor.
The following message appears on the command line: Point or [Restricted POSitioning/Scale/Rotation/Align/Mirror/Finish].
 3. On the command line, enter **pos** (Positioning) to select the restricted positioning mode.
The bar bending detail can be placed only in an orthogonal position.
 4. Place the bar bending detail under the elevation.
- The bar bending annotation creation starts automatically.
Set the symbol scale to **2**.

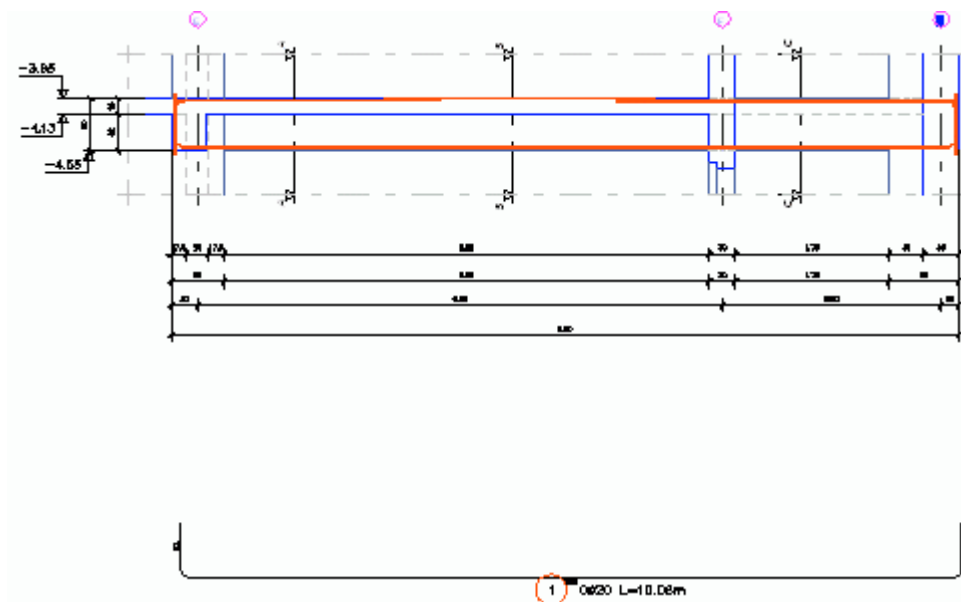


Figure 476: Face elevation 1- Bar bending detail

Editing the symbol text

1. Select the symbol.

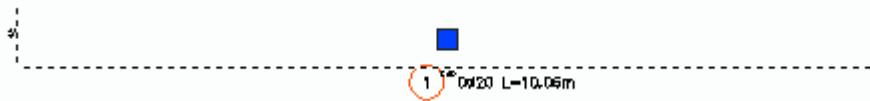


Figure 477: Selecting the symbol

2. On the Smartbar, click  to access the properties dialog box.
3. On the **Definition** tab, click **Edit**.

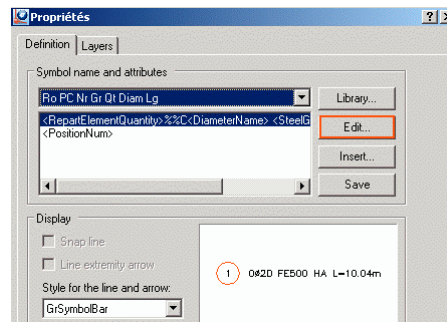


Figure 478: "Properties" dialog box – **Definition** tab

4. In the "Text Formatting" dialog box, add the Ø symbol to the representation.
5. Click **OK** to finish.



Figure 479: "Text Formatting" dialog box

Saving the symbol

1. On the **Definition** tab, click **Save**.
2. In the "Save symbol" dialog box, click **OK** to save the symbol.

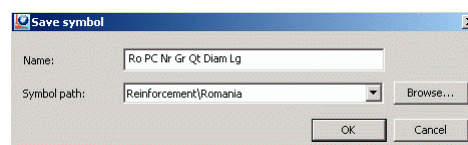


Figure 480: "Save symbol" dialog box

3. Click **OK** to finish.
- The symbol is saved.



Figure 481: Bending detail and symbol

Add bending details and symbols for the other 3 bars.

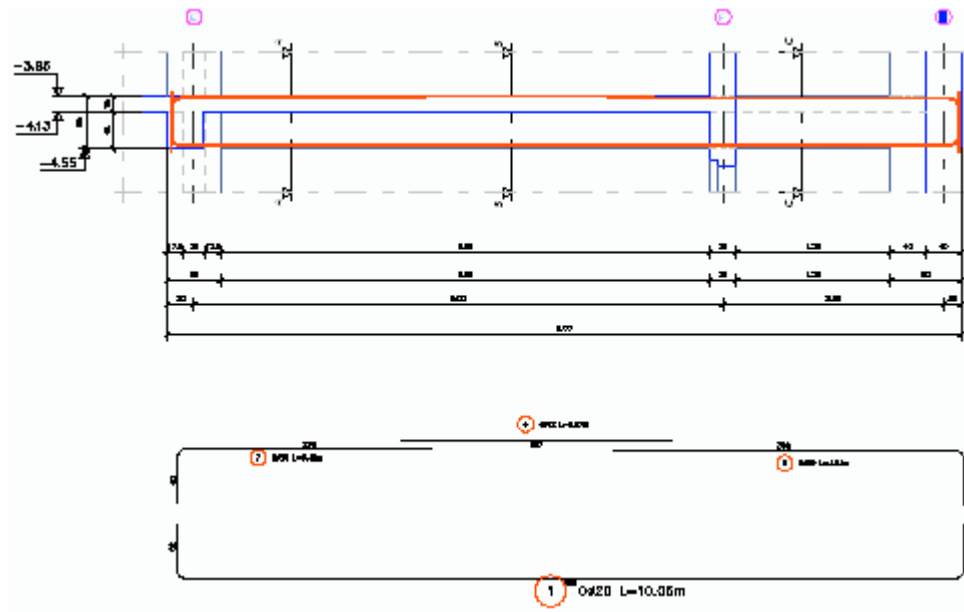


Figure 482: Bending details

Creating point by point dimensions on the bending detail

Next, create point by point dimensions to measure the overlapping distance on the bending detail.

1. On the **Dimensions** toolbar, click .
2. Click a point to define the position of the dimension line.

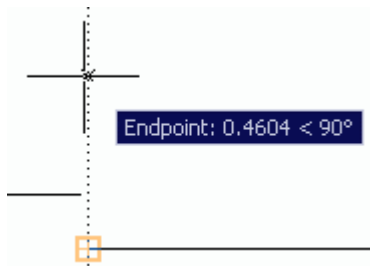


Figure 483: Setting the dimension line position

- Click the two points to dimension as shown in Figure 484.

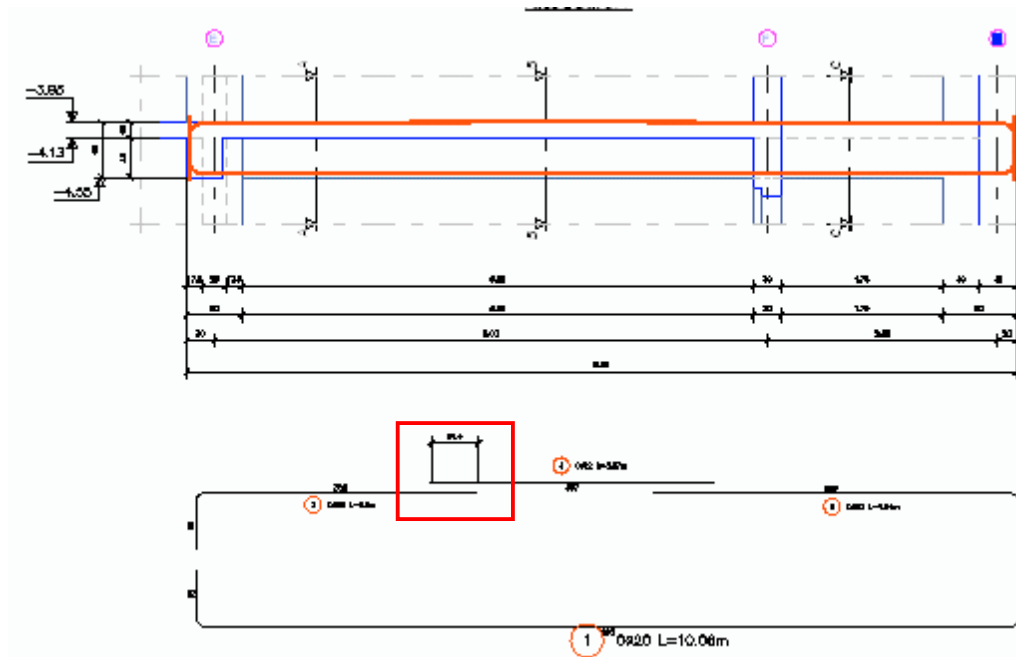


Figure 484: Dimension by points

Repeat the command and dimension the other extremity.

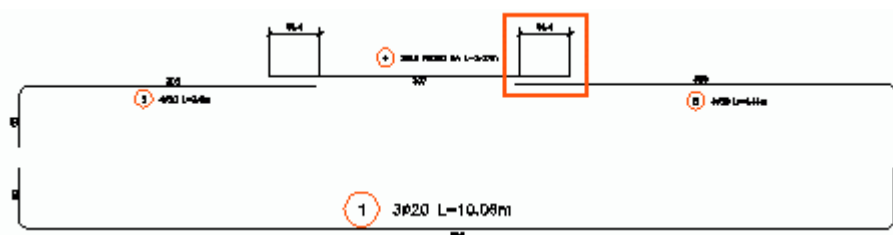


Figure 485: Dimensions by points

Step 6: Create a rectangular frame on Left cut A-A

- On the **Reinforcement** toolbar, **Reinforcement Bars** flyout, click
- Click the first point of the rectangular frame as shown in the following figure.

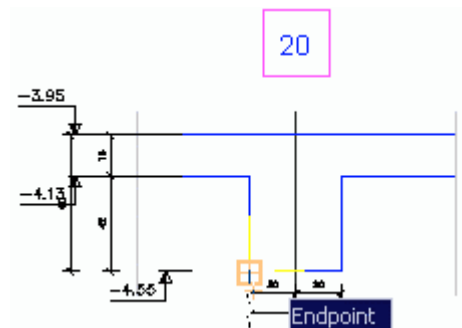


Figure 486: Left cut A-A - Setting the first point of the rectangular frame

- On the Smartbar, set the frame diameter to **10Ø**.

4. Place the second point of the bar diagonal as shown in the following figure.

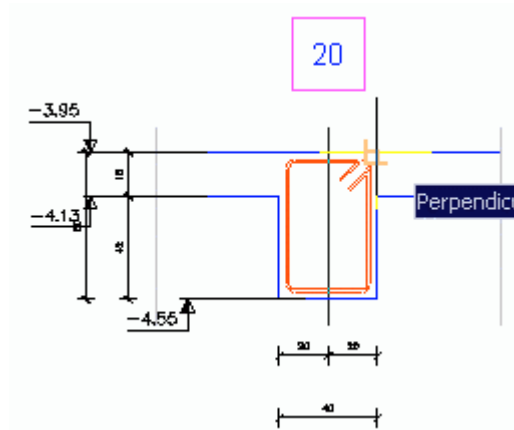


Figure 487: Left cut A-A - Second point of the bar

5. Click another point to set the hook position to the right.

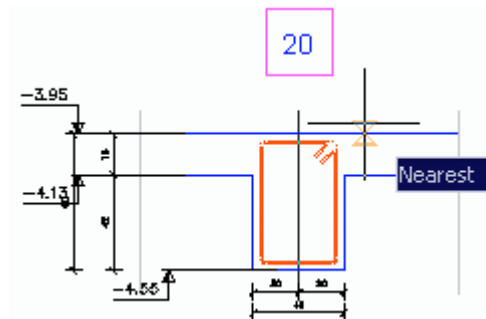


Figure 488: Left cut A-A - Setting the hook position

6. On the command line, enter **S** (scale) to define the symbol scale and press **Enter**.
7. On the command line, enter **2** to set the scale and press **Enter**.
8. Place the symbol at the bottom of the rectangular frame.

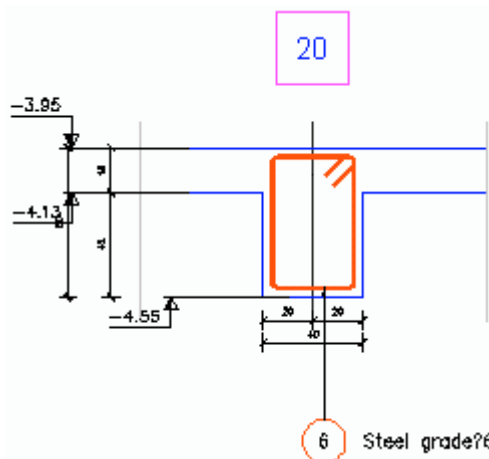


Figure 489: Left cut A-A - Symbol representation

Using the same process, create rectangular frames for the **Left cut B-B** and **Left cut C-C** views.

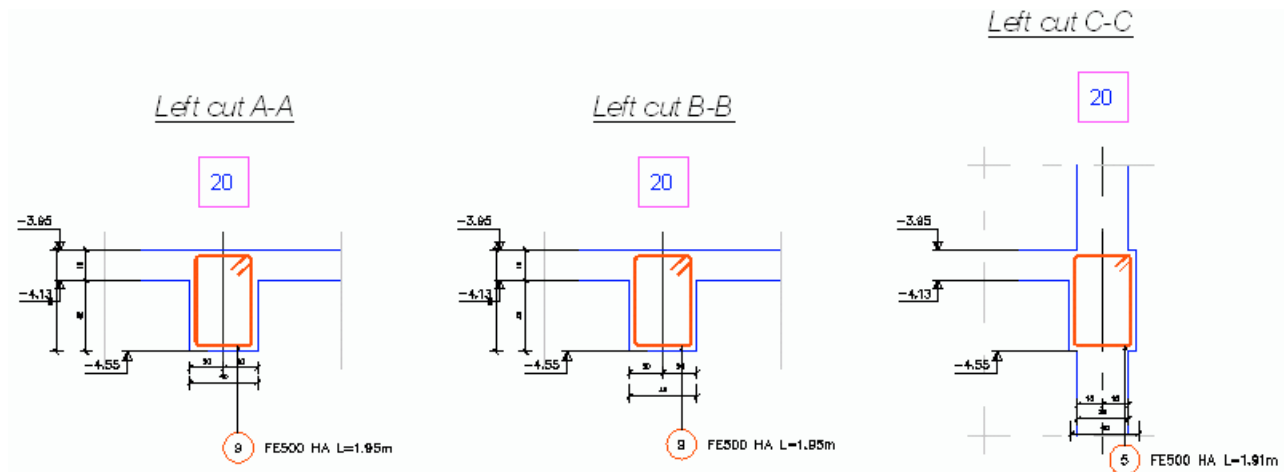


Figure 490: Rectangular frames on the cuts

Next, modify and save the symbol of Left cut A-A.

Creating a free distribution

1. On the **Reinforcement** toolbar, **Distributions** flyout, click
2. On the Face elevation 1 view, select the first created straight bar.

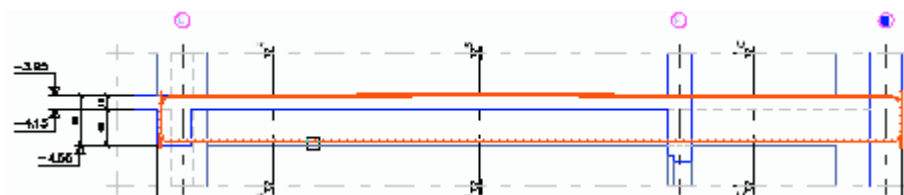


Figure 491: Face elevation1 - Selecting the straight bar

3. Place the first point of the distribution on the left bottom corner of Left cut A-A.

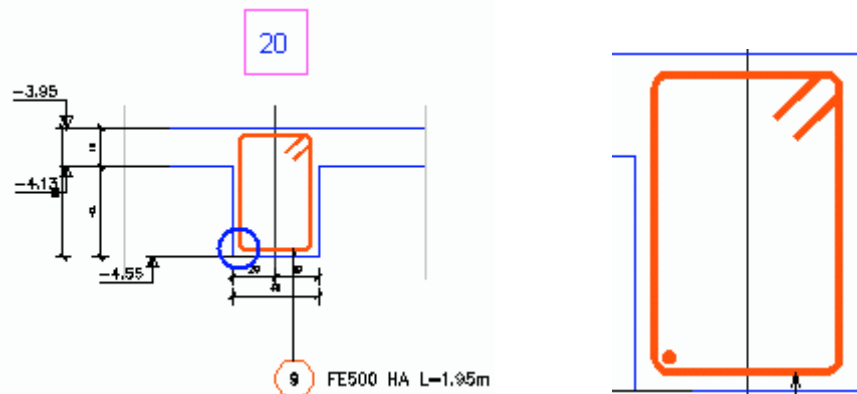


Figure 492: Left cut A-A - First point of the distribution

Create free distributions along **Left cut A-A** as shown in the following figure.

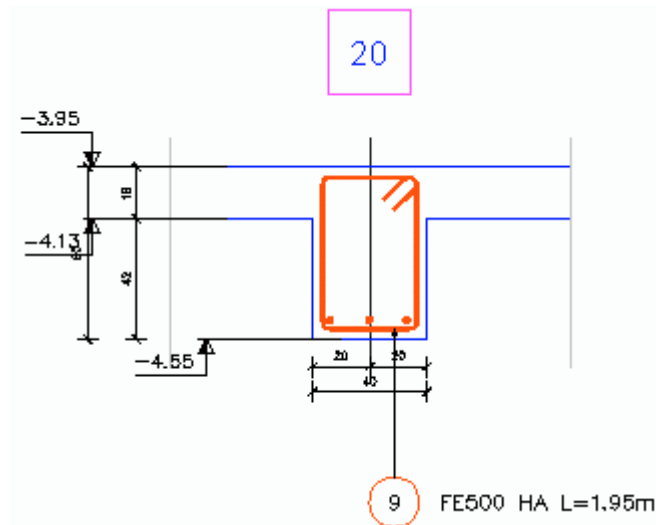


Figure 493: Left cut A-A - Free distribution

Creating symbols for the free distributions

1. On the **Reinforcement** toolbar, **Symbols and annotations** flyout, click .
2. Select an element. For this example, use the first created free distribution.

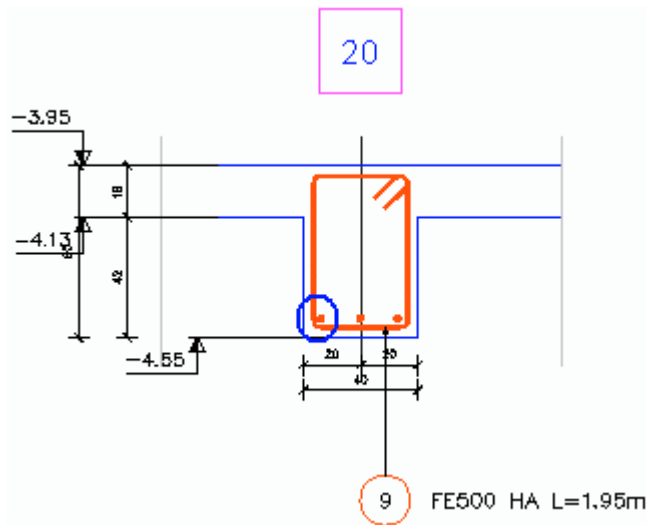


Figure 494: Left cut A-A - Creating the symbol for the free distribution

3. On the command line, enter the following settings:
 - Enter **S** (scale) and press **Enter**.
 - Enter the scale, **2**, and press **Enter**.
4. Place the symbol on the right side of the Left cut A-A view.

5. On the Smartbar, from the **Name** drop-down list, select the symbol.

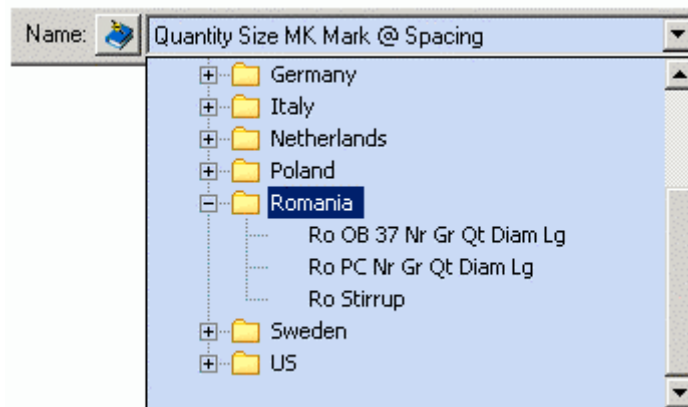


Figure 495: "Name" drop-down list

The symbol for the rectangular frame on Left cut A-A is created.

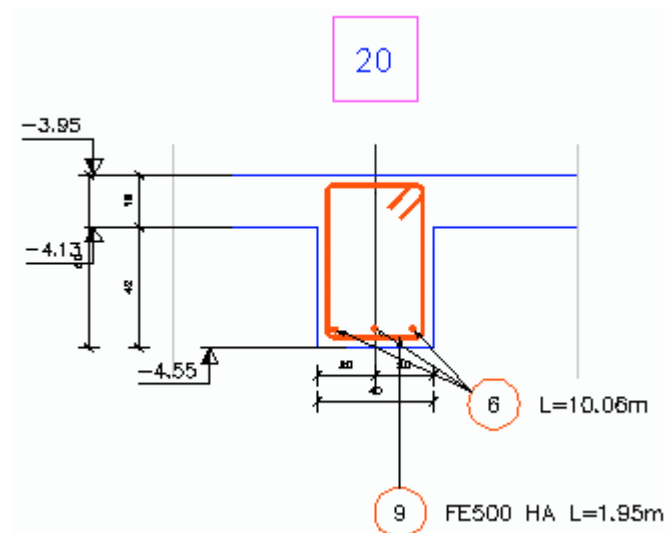


Figure 496: Left cut A-A - Symbol



Use the AutoCAD® **Copy selection** command and copy the symbol for the rectangular frame of **Left cut A-A** on **Left cut B-B** and **Left cut C-C**.

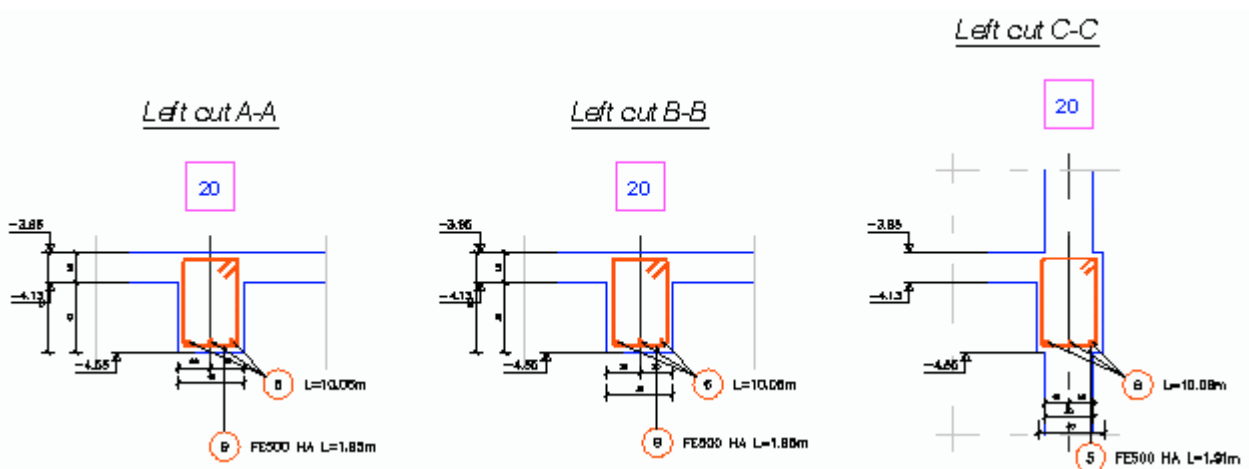


Figure 497: Symbols on views

Avoiding taking the bar distribution into account for Left cut B-B and Left cut C-C

1. Select the free distributions of the **Left cut B-B** and **Left cut C-C** views.

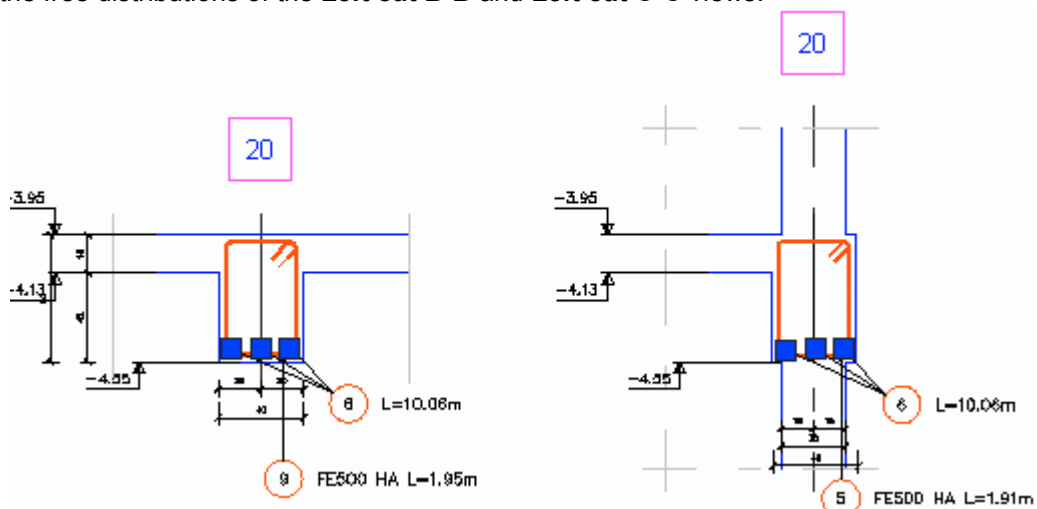


Figure 498: Left cut B-B and Left cut C-C - Selecting the distributions

2. On the Smartbar, click  to access the properties dialog box.
3. On the **Definition** tab, unselect the **To count** option.

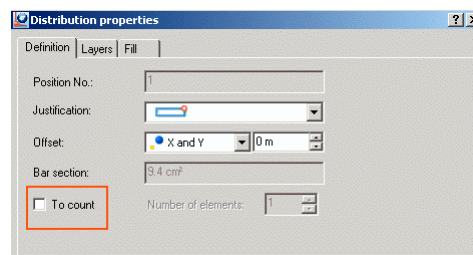


Figure 499: "Distribution properties" dialog box – **Definition** tab

4. Click **OK**.

Note: Notice the effect of not considering the bar distribution of Left cut B-B and Left cut C-C on the first created bending detail. The quantity of bars has changed from 9 to 3.

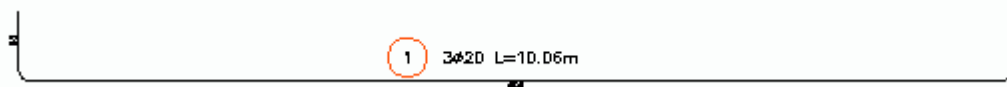


Figure 500: Number of bars displayed in the bar bending detail

Create distributions and symbols on the top of Left cut A-A, B-B and C-C for the straight bars created on the Face elevation 1 view.

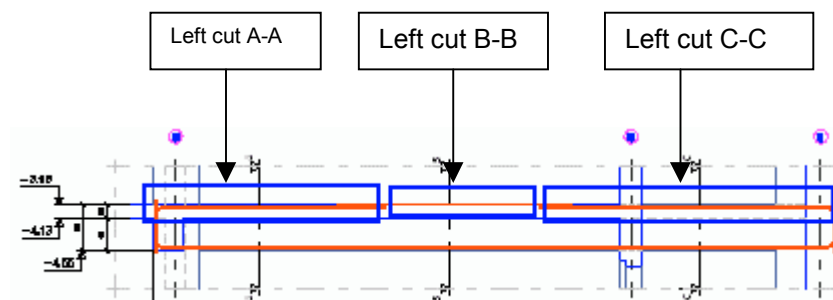


Figure 501: Creating free distributions using the straight bars of the Face elevation 1 view

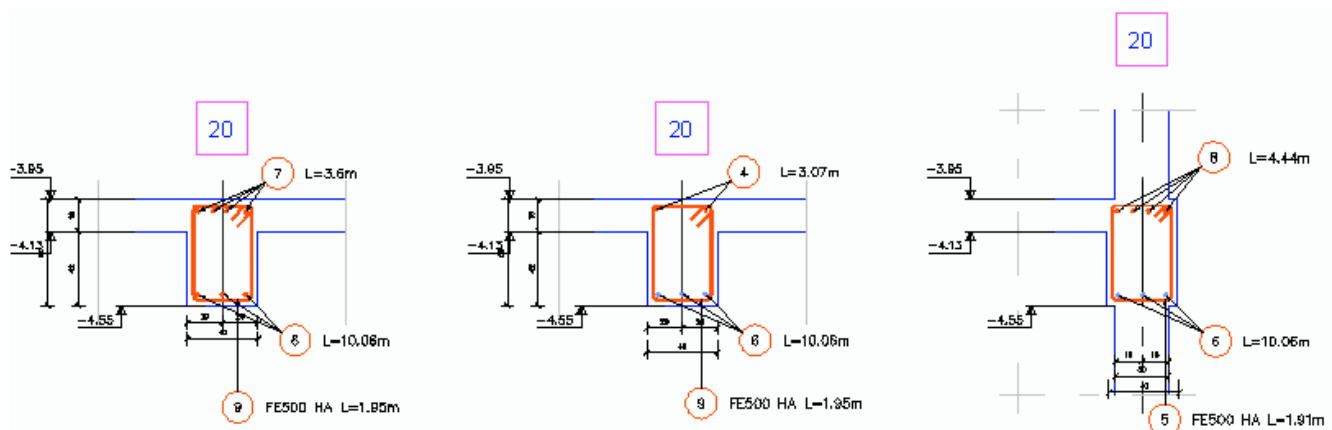


Figure 502: Left cut A-A, Left cut B-B and Left cut C-C - Free distributions and symbols

Creating a bar bending detail on Left cut A-A

1. On the **Reinforcement** toolbar, click
2. Select the rectangular bar on **Left cut A-A**.

The bar bending detail appears "hooked" to the cursor.

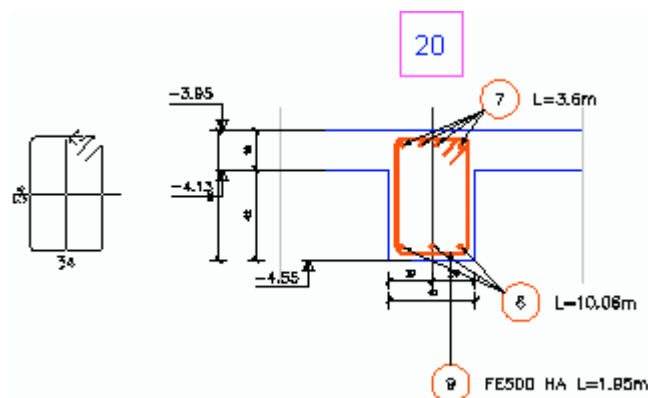


Figure 503: Creating the bending detail

3. Place the bar bending detail next to **Left cut A-A**.

The annotation creation starts automatically.

4. On the Smartbar, from the **Name** drop-down list, select the symbol for the bar bending detail: **Ro>OB37 Gr Nr**.
5. Place the symbol on the top of the bending detail.

The bending detail is created.

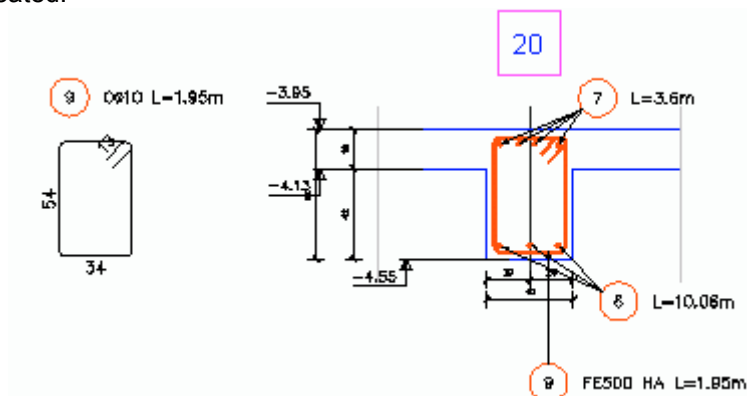


Figure 504: Left cut A-A - Bar bending detail for the rectangular frame

Modifying the hook representation of the bending detail

1. Select the bar bending detail.

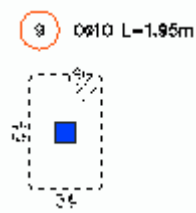


Figure 505: Left cut A-A - selecting the bending detail

2. On the Smartbar, click  to access the properties dialog box.
3. On the **Bending details** tab, in the **Representation** area, select the **Unfold** option.

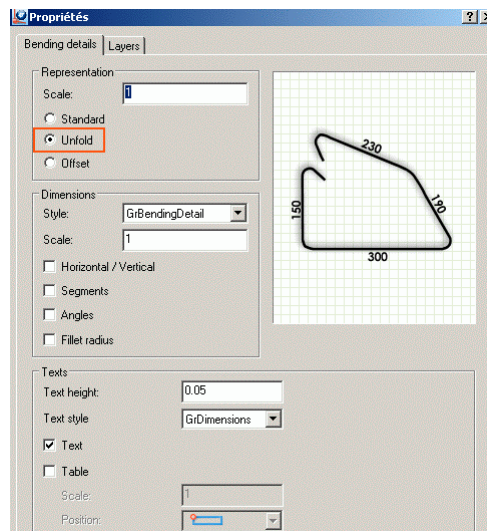


Figure 506: "Properties" dialog box – **Bending details** tab

4. Click **OK**.

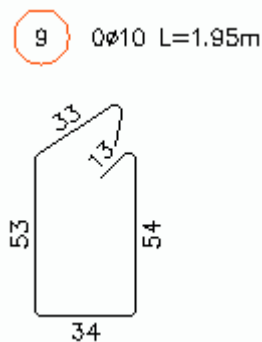


Figure 507: New symbol representation

Modifying the hook length of the bending detail

1. Double click the bending detail to display all the grip points.

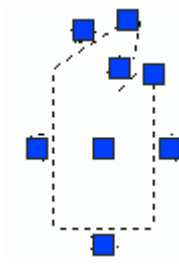


Figure 508: Bending detail – Grip points

2. Click the hook point and stretch it to modify its representation.

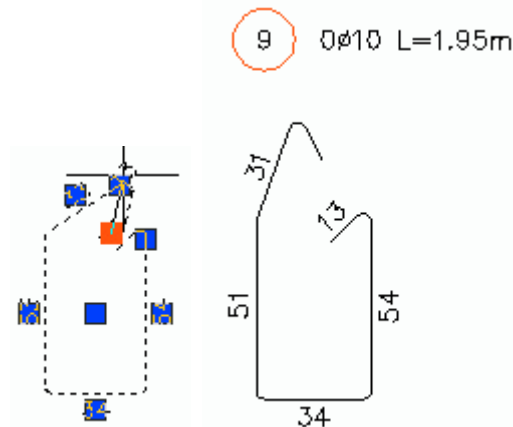


Figure 509: Stretching the bending detail

Step 7: Create a linear distribution on Face elevation 1

Distribute the rectangular frames on the **Left cut A-A** and **Left cut C-C** on the **Face elevation 1** view.

1. On the **Reinforcement** toolbar, **Distributions** flyout, click
2. Select a bar to distribute. For this example, use the rectangular frame on **Left cut A-A**.

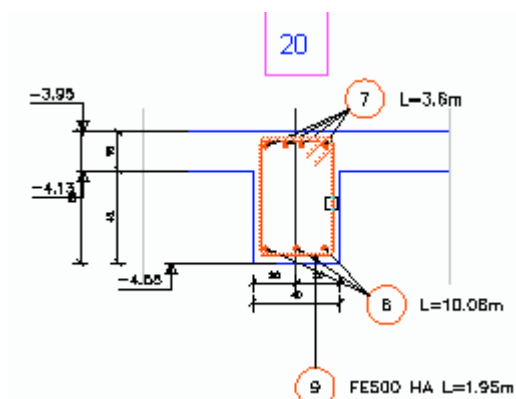


Figure 510: Left cut A-A - Selecting the rectangular frame

3. Click the horizontal arrow to select the view direction.

4. Place the distribution by two points as shown in the following figure.

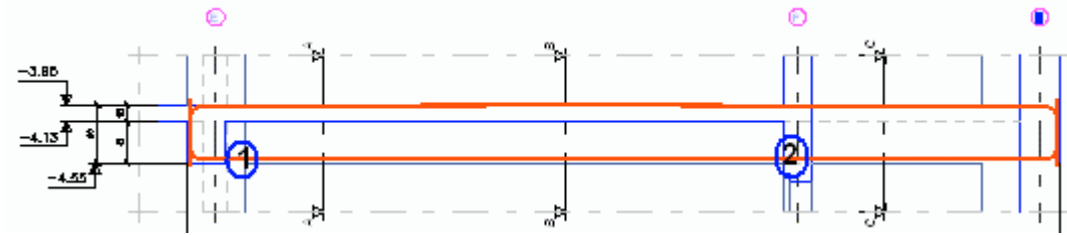


Figure 511: Elevation face 1 - Starting points

5. On the Smartbar, from the **Qty.** drop-down list, select **by quantity** to set the number of bars in the distribution using a combination of numbers and spacings. Next, set the quantity of bars, **18x0.1 9x0.2 19x0.1** , and press **Enter**.
6. Click the top of the elevation to position the distribution as shown in the following figure.

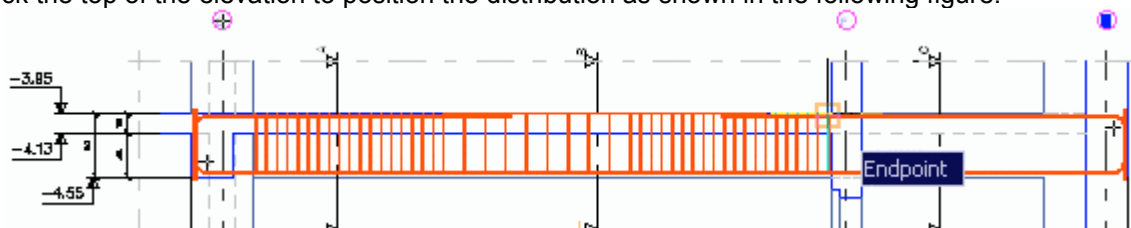


Figure 512: Elevation face 1 - Setting the distribution side

7. Press **Esc** to finish.

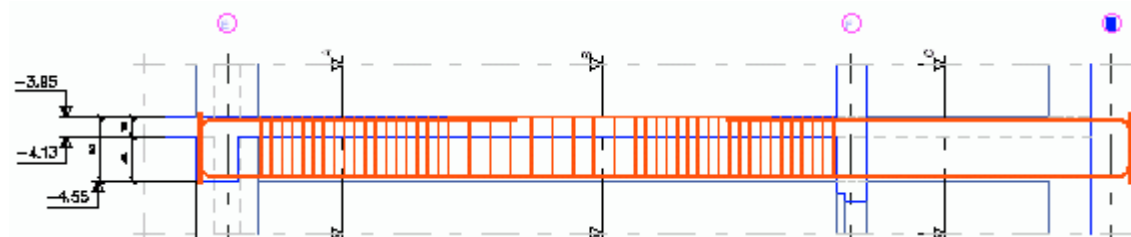


Figure 513: Elevation face 1 - Distribution

Using the same process, create another linear distribution. Select the frame on the **Left cut C-C** view as the bar to distribute. Set the definition type to **By spacing (truncated)** and the bar spacing to **0.15 m**.

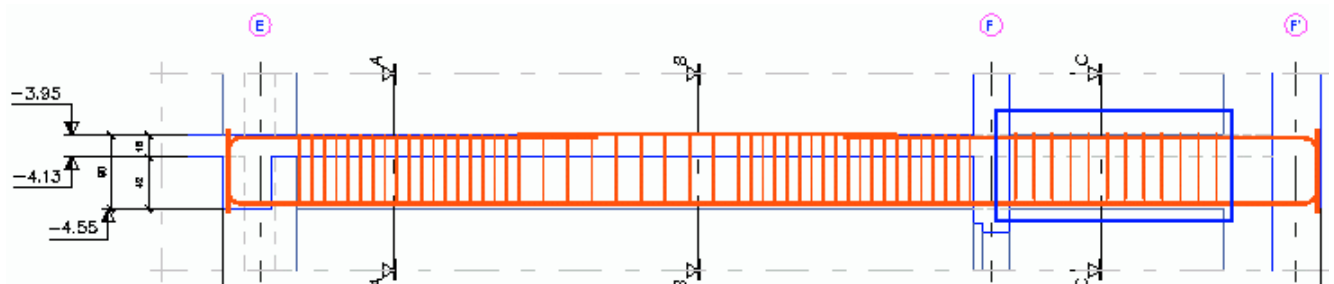


Figure 514: Elevation face 1 - Second linear distribution

Step 8: Add visual elements

In this step, add a distribution dimension on the **Face elevation 1** view and create a bar bending detail on **Left cut C-C** with the **Unfold** representation.

Creating a distribution dimension on Face elevation 1

Before creating a distribution dimension, change the scale of the drawing to **1:100**. In the Pilot, click  and select the **1:100** scale from the list.

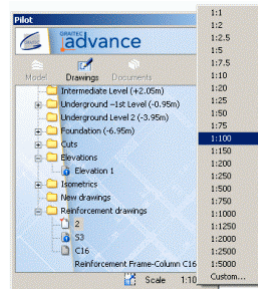


Figure 515: Setting the drawing scale

1. On the **Reinforcement** toolbar, click .
2. Select the distribution to dimension. For this example, use the first linear distribution.

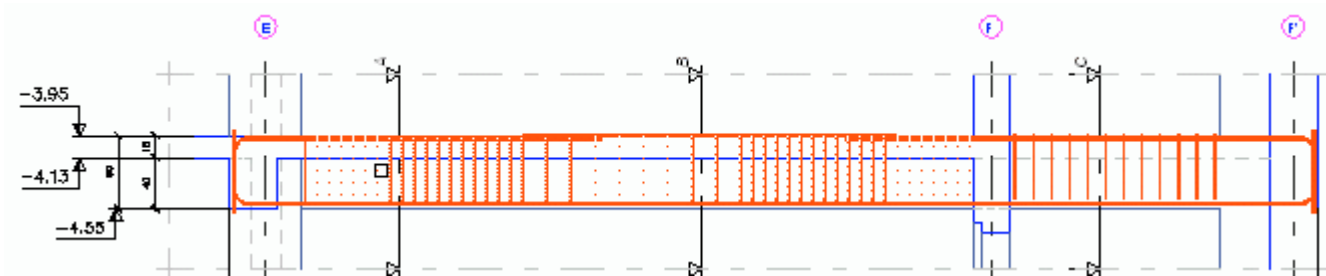


Figure 516: Elevation face 1 - Selecting the distribution

3. Place the dimension line on top of the distribution.

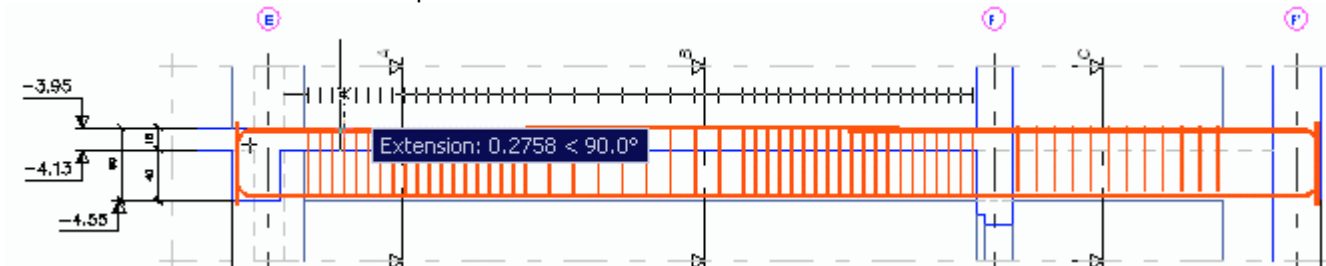


Figure 517: Elevation face 1 - Placing the distribution dimension

Modifying the symbol name and representation

1. On the Smartbar, click  to access the properties dialog box.

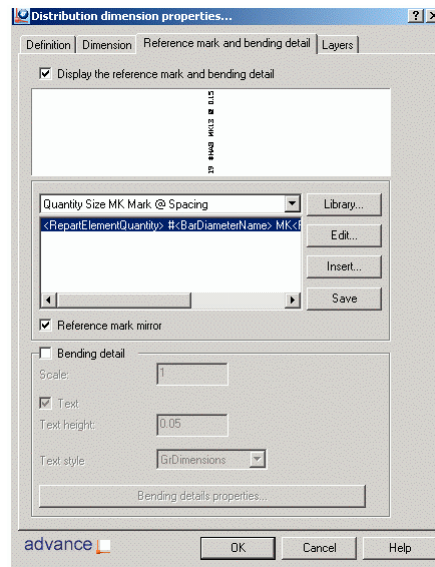


Figure 518: "Distribution dimension" properties dialog box – **Reference mark and bending detail** tab

2. Select the **Reference mark and bending detail** tab to modify the symbol.
 - From the **Name** drop-down list, select the symbol (e.g. **Romania > Ro Nr Sp Diam**).

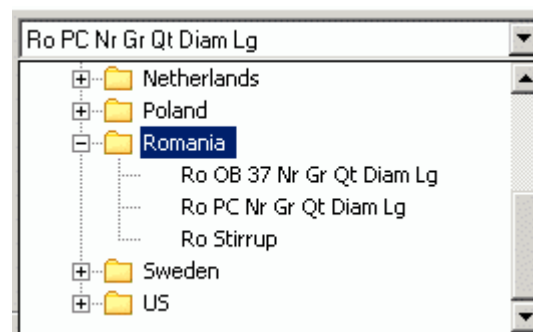


Figure 519: Selecting the symbol

3. Click **OK**.
4. Place the symbol on top of the Elevation face 1 view.

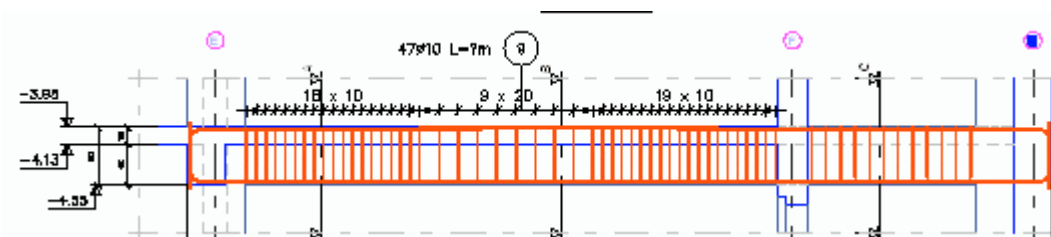


Figure 520: Elevation face 1 - Distribution symbol

Editing and saving the symbol

1. Select the distribution dimension.

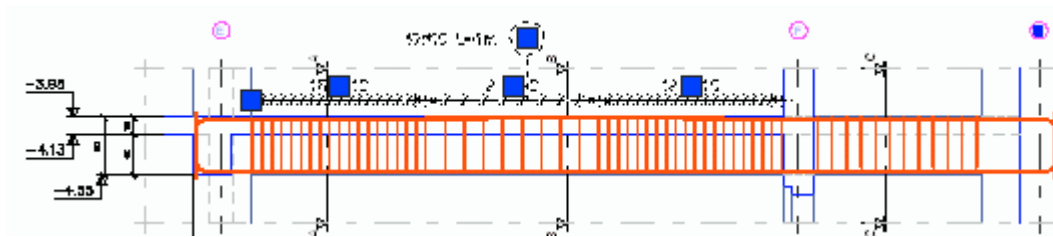


Figure 521: Elevation face 1 - Distribution dimension

2. On the Smartbar, click to access the properties dialog box.
3. Select the **Reference mark and bending detail** tab to modify the symbol.
4. Click **Edit** to display the **Text Formatting** bar.

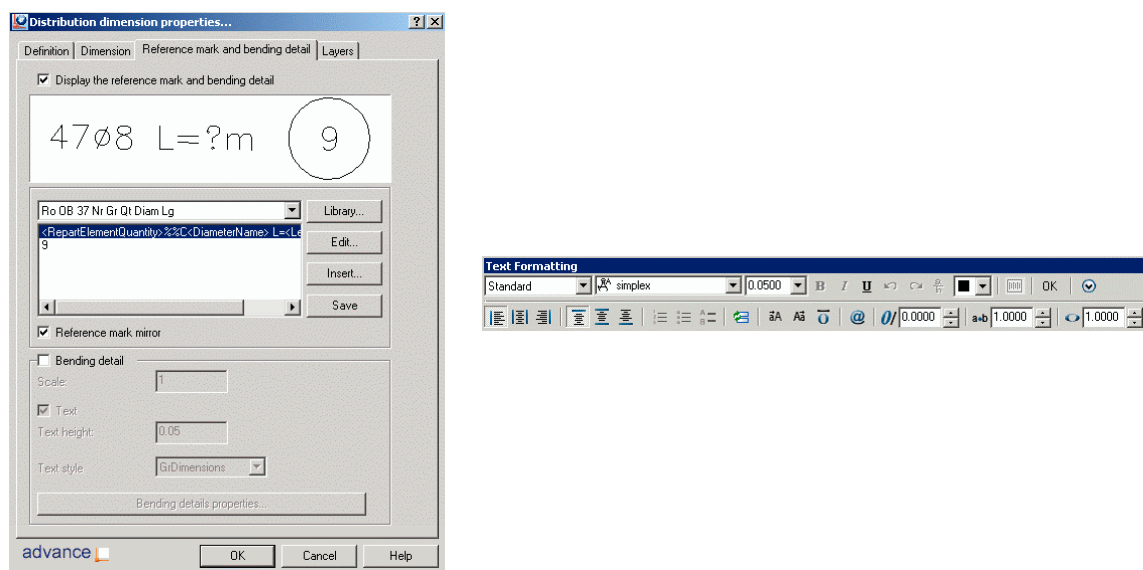


Figure 522: "Distribution dimension" properties - **Reference mark and bending detail** tab

5. In the text field, enter the name of the symbol: DiameterName>/Element distances[1]>/<ElementDistances[2]>.
6. Click **OK**.
7. In the "Distribution Dimension" dialog box, on the **Reference mark and bending detail** tab, click **Save**.
8. Enter the symbol name: **Ro Nr Sp Diam[1][2]**.

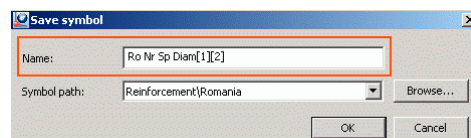


Figure 523: "Save symbol" dialog box

9. Click **OK** to save the s
10. In the properties dialog box, click **OK** to finish.

Dimensioning the second distribution

1. On the **Reinforcement** toolbar, click .
2. Select the distribution to dimension. For this example, use the second linear distribution.
3. On the command line, enter **M** (mirror) to reverse the symbol position and press **Enter**.
4. Place the dimension line on top of the distribution.

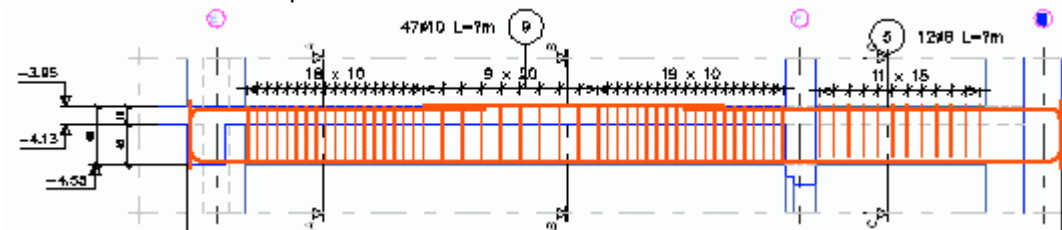


Figure 524: Face elevation 1 - Second dimension and symbol

Change the scale of the drawing to **1:50**.

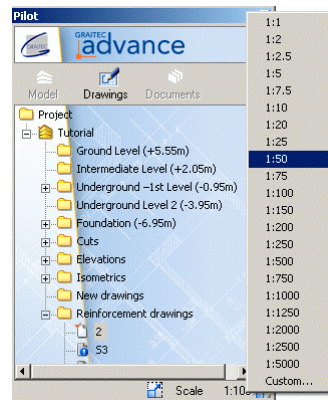


Figure 525: Setting the drawing scale

Creating a bar bending detail on Left cut C-C

Using the same process used for **Left cut A-A**, create a bar bending detail for the rectangular frame of **Left cut C-C**.

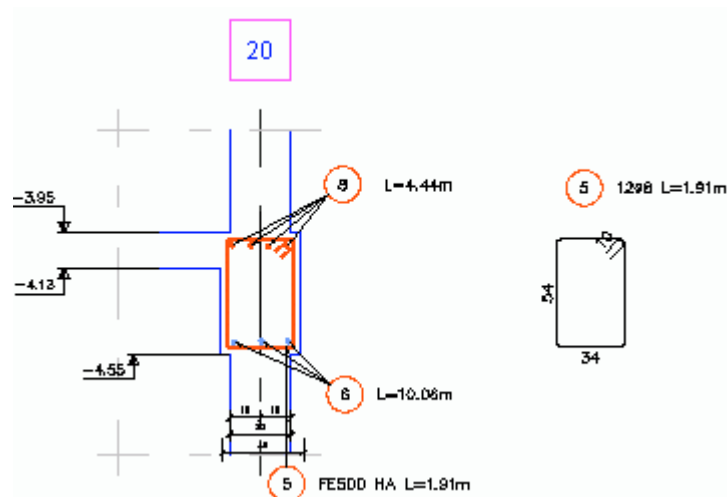


Figure 526: Left cut C-C - Bar bending detail

Modifying the representation of the bar bending detail

1. Select the bending detail.
2. On the Smartbar, click  to access the properties dialog box.
3. On the **Bending details** tab, in the **Representation** area, select the **Unfold** option to modify the representation of the bar bending detail.

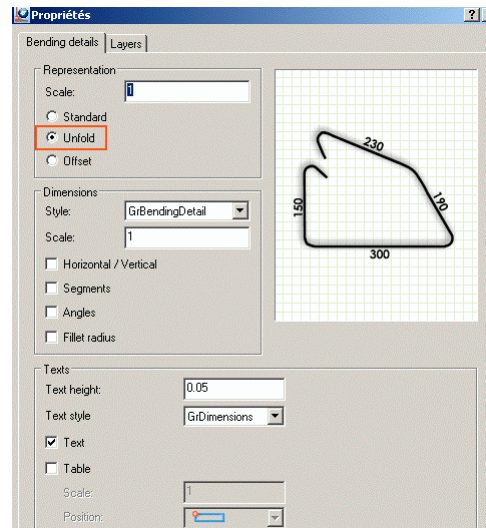


Figure 527: "Properties" dialog box - **Bending details** tab

4. Click **OK**.

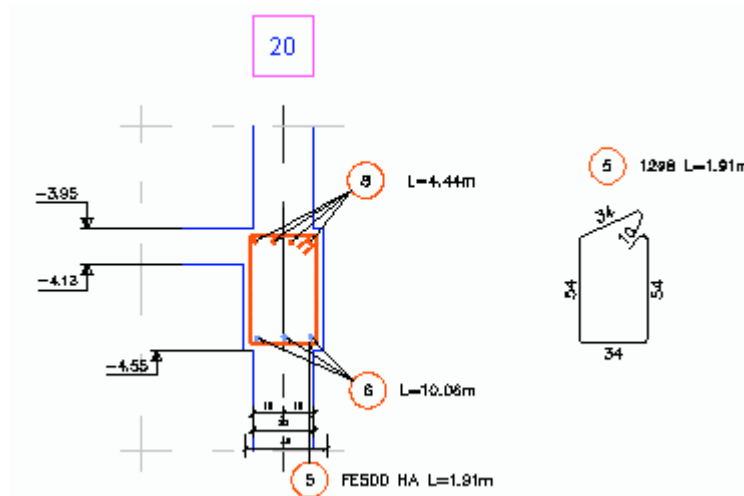


Figure 528: Left cut C-C - Symbol representation

Double click the bar bending detail to select all the grip points.

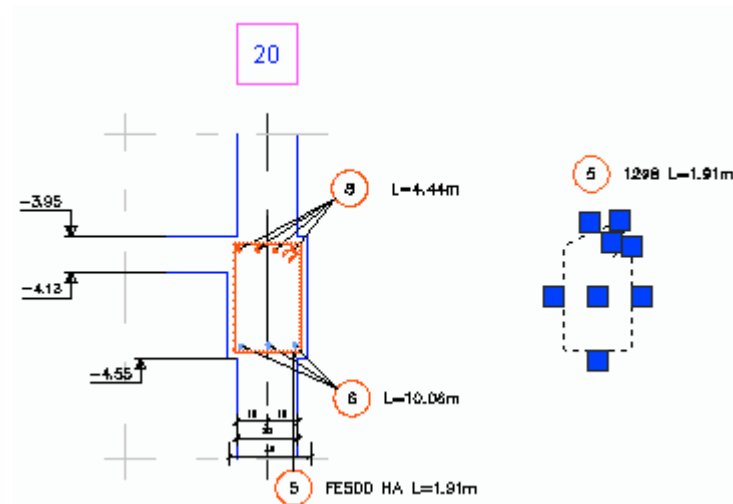


Figure 529: Left cut C-C - Displaying the bending detail grip points

Click the hook and stretch it to adjust its position.

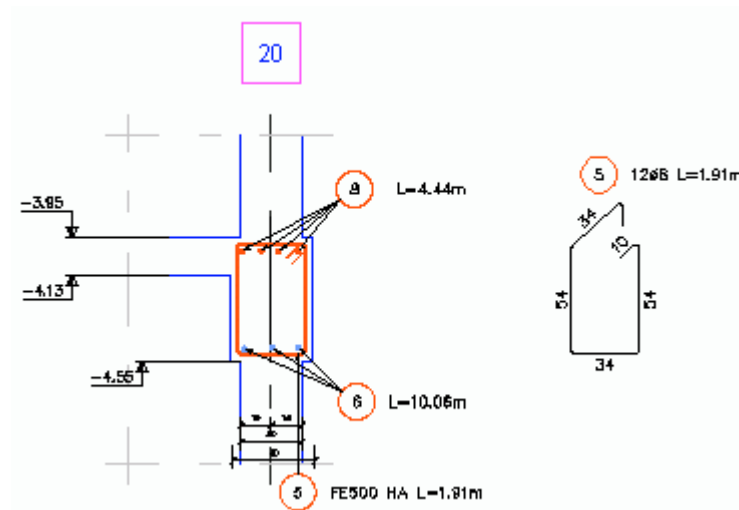


Figure 530: Left cut C-C - Adjusting the hook

Step 9: Renumber the bars

- On the **Reinforcement** toolbar, **Renumbering** flyout, click .
- In the “Numbering scope” area, select **the numbering applies to the current drawing** option.

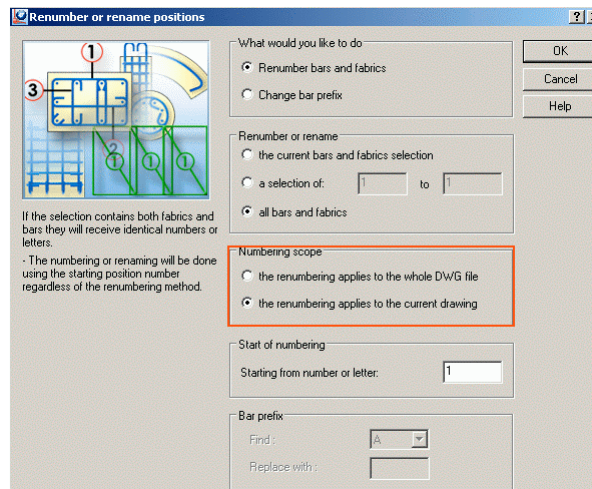


Figure 531: “Renumbering or rename positions” dialog box

- Click **OK** in the renumbering dialog box and in the message window.



Figure 532: Confirmation window

Step 10: Create a list on the layout

- On the **Reinforcement** toolbar, **Lists** flyout, click .
 - On the command line, enter **B** (Bars) to consider all the bars and press **Enter**.
 - On the command line, enter **V** (View) to select all the elements in the current view and press **Enter**.
- The list is hooked to the cursor.
- Place the list next to the **Face elevation 1** view.

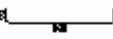
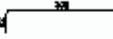
 GRAITEC INC. 1895, Level 10, Suite 1010 Laval, Québec H7S-2G5										Tel : (418) 835-1000 Fax : (418) 835-1000 www.graitec.com info.graitec@graitec.com
BAR LIST (Metric)										
Nr	Quantity	Total No	Reinforced	φ	Length (m)	Weight (kg)	Nr. sp.	Total Length(m)	Total Weight(kg)	Bar Shape
1	3	3	HA	20	10.06	24.80	1	30.17	74.40	
2	4	4	HA	20	3.60	8.87	1	14.39	35.49	
3	4	4	HA	20	4.44	10.94	1	17.75	43.78	
4	0	0	HA	12	3.08	2.75	0	0.00	0.00	
5	2	2	HA	12	3.07	2.73	1	6.15	5.46	
6	47	47	HA	10	1.85	1.20	1	81.76	58.82	
7	12	12	HA	8	1.81	0.78	1	22.94	8.06	
Amount Reinforced								Weight Summary		
14.1								224.81		

Figure 533: Creation of the list

Sorting the list

1. Select the list.

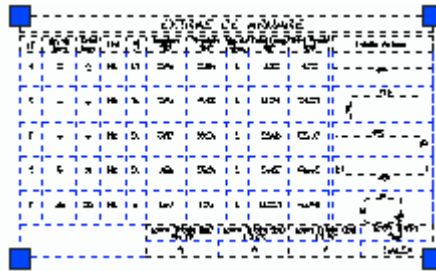


Figure 534: Selecting the list

2. On the Smartbar, click  to access the list properties dialog box.
3. On **Definition** tab, from the **Sorting column** drop-down list, select **PositionNum** as the sorting method.

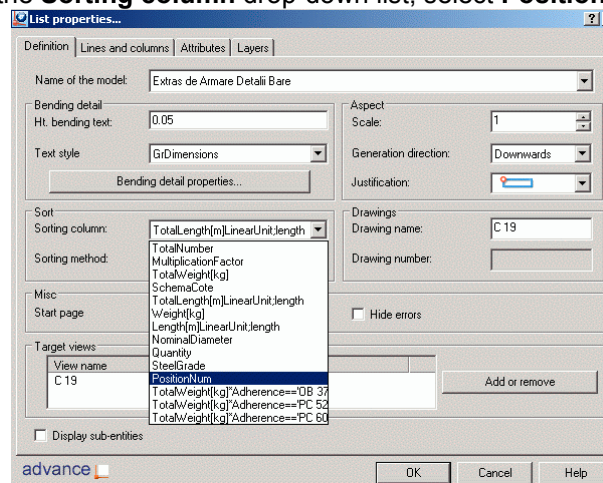


Figure 535: "List properties" dialog box - **Definition** tab

4. Click **OK**.

Step 11: View the reinforcement in 3D

1. On the **Reinforcement 3D** viewer toolbar, click .
2. Include the reinforcement views in a window selection.

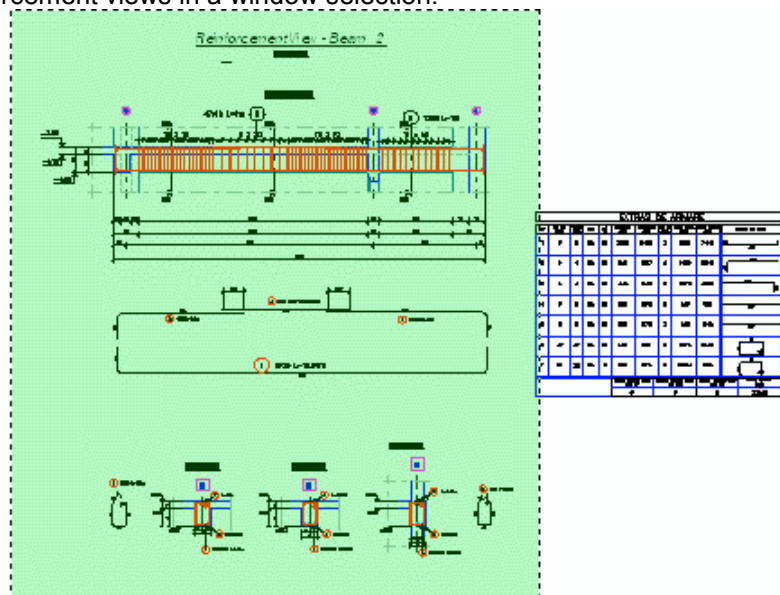


Figure 536: Selecting the reinforcement view

3. Press **Enter**.

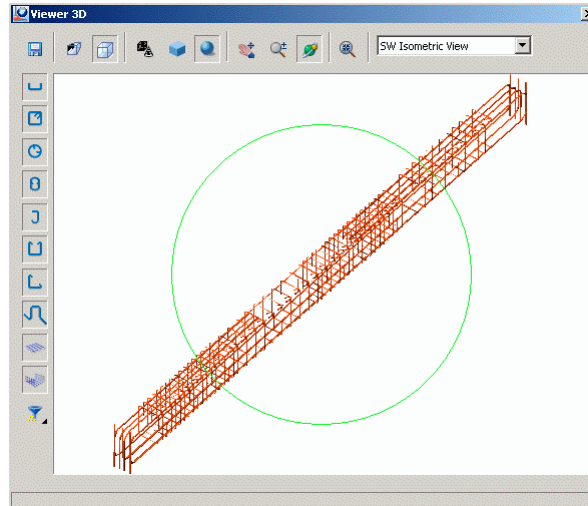


Figure 537: "Viewer 3D" dialog box

4. Use the cursor to rotate and view the drawing from different angles.

Saving the drawing

1. In the "Viewer 3D" dialog box, click .
2. In the "View name" dialog box, set the name of the drawing to **Reinforcement Frame - Beam B2**.

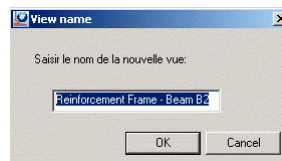


Figure 538: Setting a name for the drawing

3. Click **OK**.

The new drawing is displayed automatically in the Pilot.

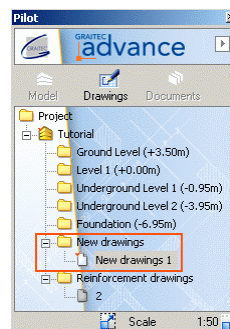


Figure 539: Pilot - New drawing

Using the "drag and drop" procedure, group the **Reinforcement Frame - Beam B2 3D** drawing in the **B2 reinforcement**.

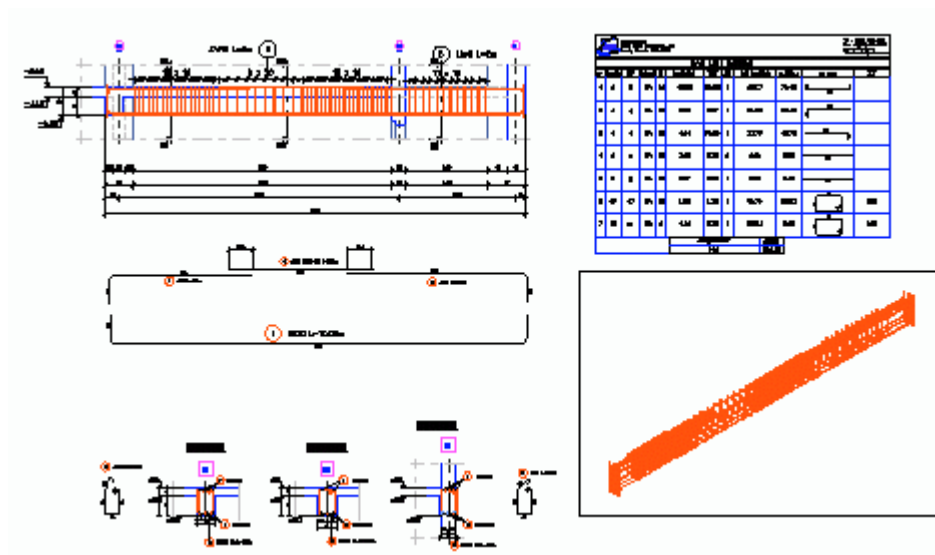


Figure 540: The reinforcement drawing

Lesson 15: Creating a layout

In Advance, you can automatically create layouts based on drawings. You can insert your title blocks and frames in the layout.

In this lesson you will learn how to:

- Create a drawing presentation.
- Prepare the page for printing.
- Modify the layout using the Smartbar.

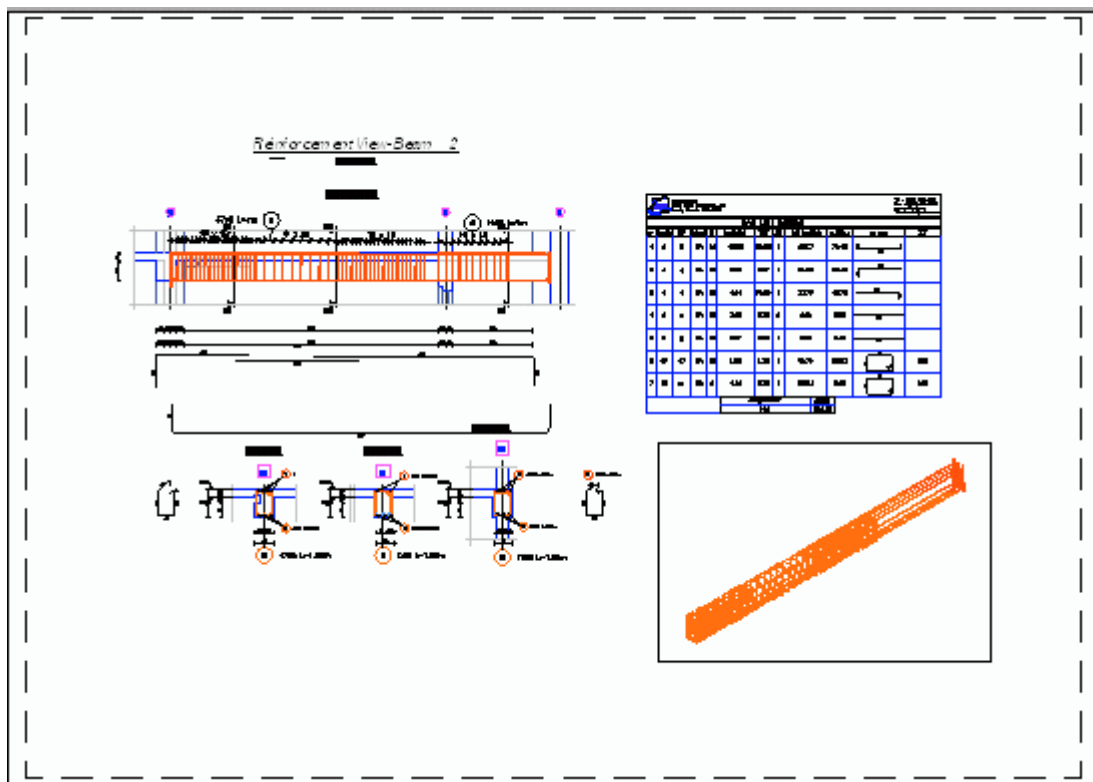


Figure 541: Layout of the reinforcement drawing

Setting the layout page for print

This step describes how to set up the layout page for printing and how to name it.

1. In the Pilot, in **Drawings** mode, right click **B2 Reinforcement Frame – Beam B2** and select **Create a layout from the drawing** from the context menu.

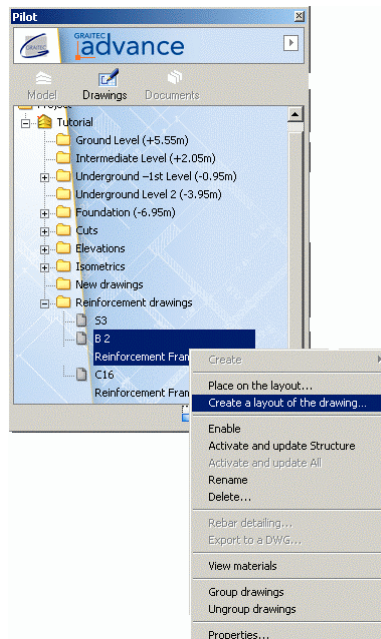


Figure 542: Pilot - Creating the layout

2. In the “Properties” dialog box, click **New** to create a new presentation style.

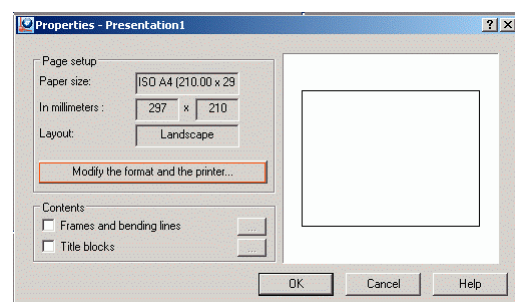
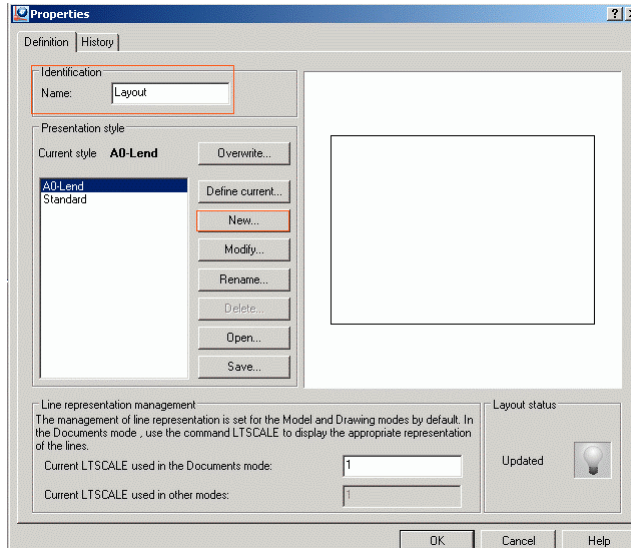


Figure 543: “Properties” dialog box – Definition tab

- Click **Modify the format and the printer** button to set the page format and the printer.

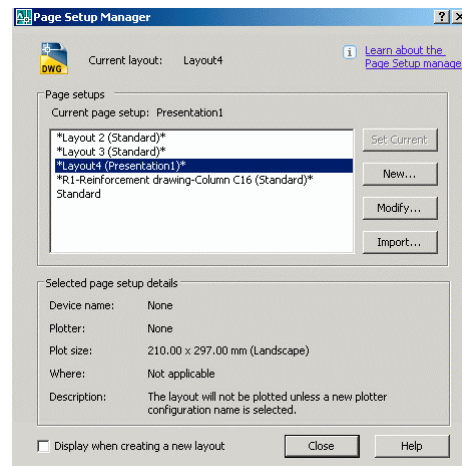


Figure 544: "Page Setup Manager" dialog box

- Click **New** to create a new page setup.

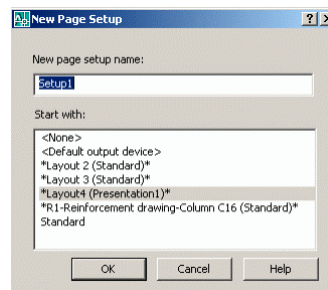


Figure 545: "New Page Setup" dialog box

- Set the New page setup name to **Setup for AO**.
- Click **OK**.

Preparing the page for printing

- In the "Page Setup" dialog box make the following settings:
 - From the Paper size drop-down list, select **ISO A0 (841.00 x 1189.00 MM)**.

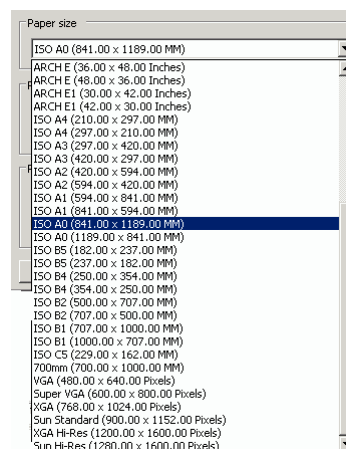


Figure 546: Selecting the paper size of the presentation

- In the “Drawing Orientation” area, select **Landscape** to set the drawing position.

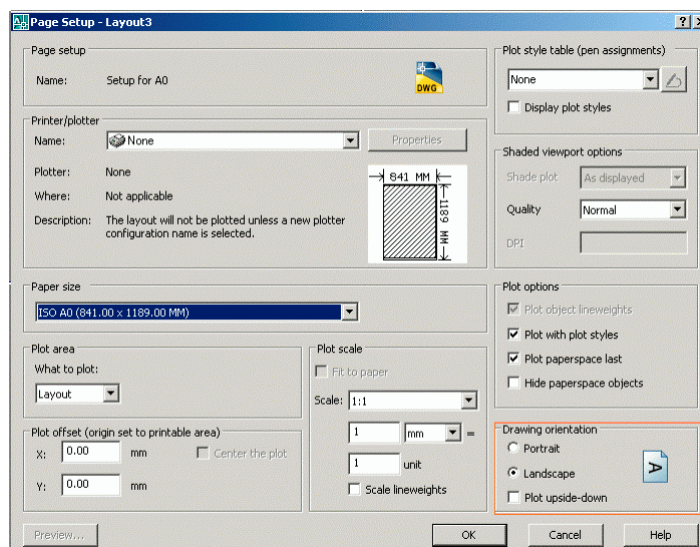


Figure 547: Selection of the drawing orientation

- Click **OK**.
- In the “Page Setup Manager” dialog box:
 - In the “Page Setup” area, select the previously defined page layout: **Setup for AO**.

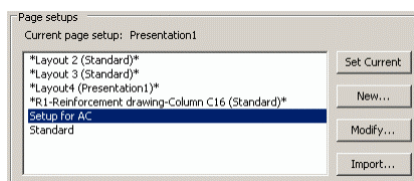


Figure 548: “Page Setup Manager” dialog box

- Click **Set Current** to apply the layout.
- Click **Close**.

Creating the page layout

This step describes how to set the page layout for the reinforcement drawing.

Adding a frame

- In the “Properties – Presentation 1” dialog box, in the **Contents** area:
 - Check the **Frames and bending lines** to add frames and bending lines to the layout.

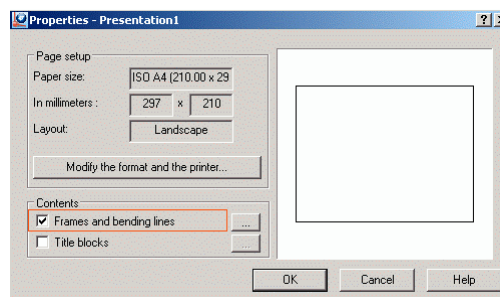


Figure 549: Adding a frame to the layout

- Click the  button next to the **Frames and bending lines** to select the frames presentation.

2. In the “Frames of Presentation 1” dialog box, click **Add** to add a frame.

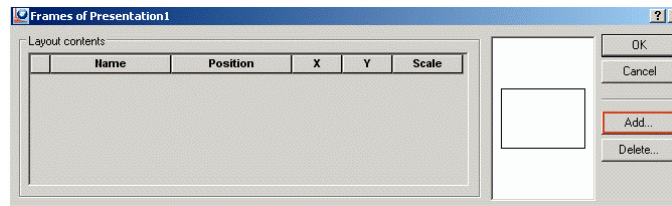


Figure 550: “Frames of Presentation 1” dialog box

- Browse and select a frame from the Advance folder tree.
 - Click **OK**.
3. In the “Properties – Presentation 1” dialog box, in the **Contents** area, make the following settings:
 - Check the **Title blocks** option.

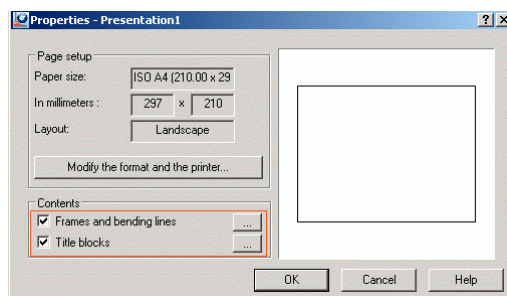


Figure 551: “Frames of Presentation 1” - Adding title blocks

- Click the  button to set the title blocks.

Adding a title block

1. In the “Title block of Presentation 1” dialog box:

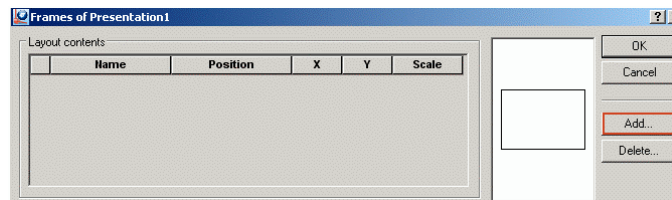


Figure 552: “Title block of Presentation 1” dialog box

- Click **Add** to add a title block.
 - Browse and select a title block from the Advance folder tree.
2. Click **OK**.
 3. Click **OK** to return to the “Properties” dialog box.

In the “Properties” dialog box, on the **Definition** tab, select the **Presentation1** presentation style.

Click **Define current** to apply the settings.

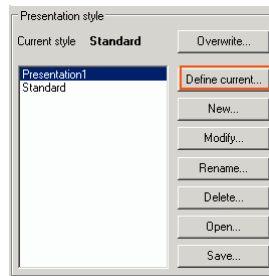


Figure 553: "Properties" dialog box.

Renaming the presentation style

In the properties dialog box, on the **Definition** tab:

1. In the "Presentation style" area, select **Presentation1** and click **Rename**.
 - In the **Style name** field, enter the new presentation name: **A0-Lend**.

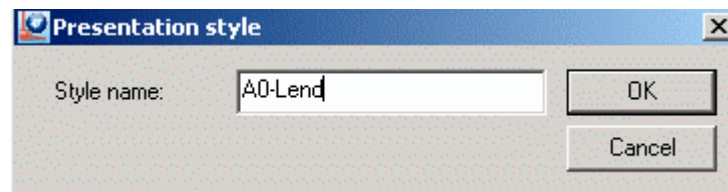


Figure 554: "Presentation style" dialog box

2. Click **OK**.
3. In the "Layout properties" dialog box, in the "Identification" area, in the **Name** field, set the layout name to **R-056.REINFORCEMENT DRAWING FOR BEAM B2**.
4. Click **OK**.

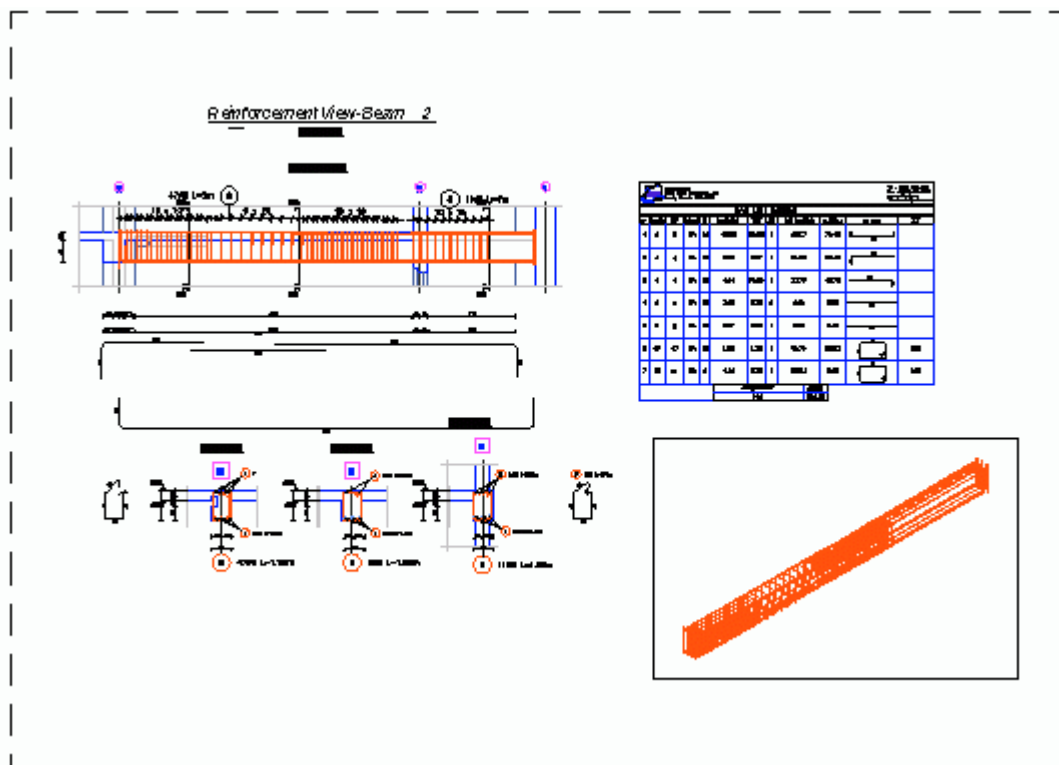


Figure 555: Layout

Modifying the layout properties using the Smartbar

1. Click to modify the layout.
2. On the Smartbar, make the following settings:
 - Set the drawing scale to **1/20** and press **Enter**.

Scale: 1/

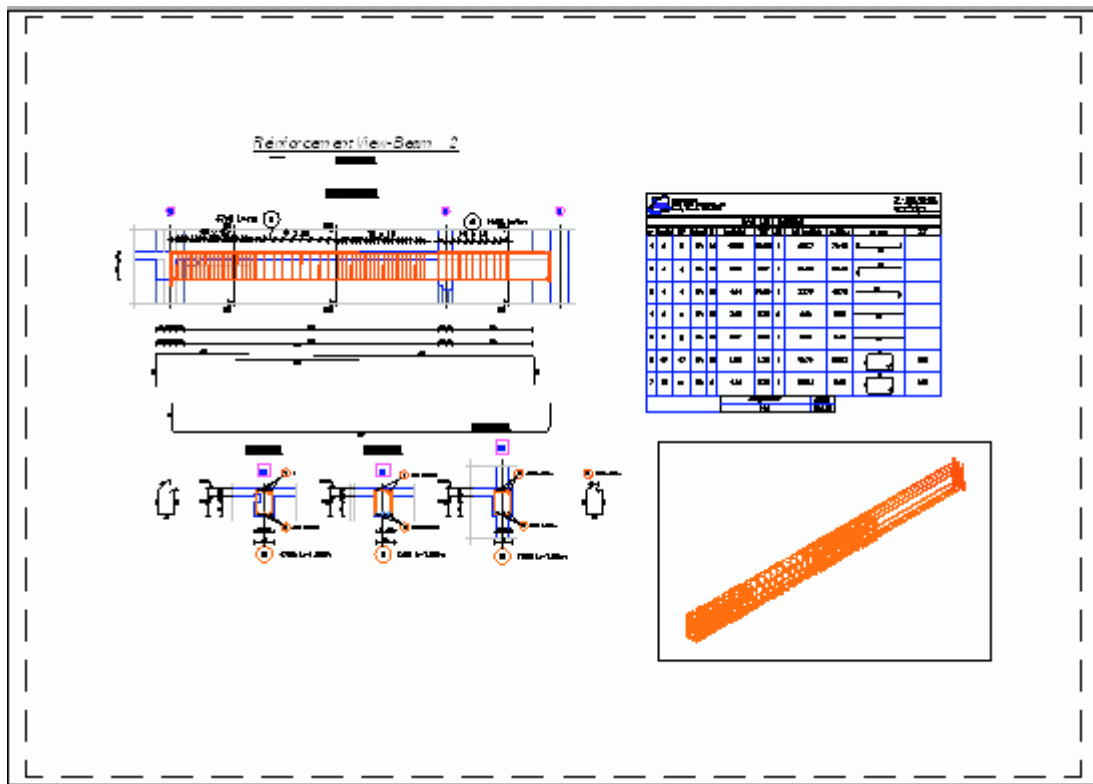


Figure 556: Layout of the reinforcement drawing

Canada

GRAITEC Inc.

183, St. Charles St. W.
Suite 300
Longueuil (Québec) Canada
J4H1C8
Tel. (450) 674-0657
Fax (450) 674-0665
Hotline (450) 674-0657
Toll free 1-800-724-5678
Web <http://www.graitec.com/En/>
Email info.canada@graitec.com

Germany, Switzerland, Austria

GRAITEC GmbH

Centroallee 263a
D-46047 Oberhausen Germany
Tel. +49-(0) 208 / 62188-0
Fax +49-(0) 208 / 62188-29
Web <http://www.graitec.com/Ge/>
Email info.germany@graitec.com

United Kingdom

GRAITEC UK Ltd.

The Old Forge
Suth Road
Weybridge
Surrey KT13 9DZ
Tel. +44 (0)1932 858516
Fax +44 (0)1932 859099
Email sales@graitec.co.uk

Romania

GRAITEC Roumanie SRL

Str. Samuil Vulcan, Nr. 10 Sector 5
București, Romania
Tel. +40 (21) 410 0119
Fax +40 (21) 410 0124
Mobile 0729 002 107
Web <http://www.graitec.com/Ro/>
Email sales@graitec.ro

France

GRAITEC France Sarl

17 Burospace
91573 Bièvres Cedex
Tel. 33 (0)1 69 85 56 22
Fax 33 (0)1 69 85 33 70
Web <http://www.graitec.com/Fr/>
Email info.france@graitec.com

Czech Republic and Slovakia

AB Studio spol. s r.o.

Jeremenkova 90a 140 00 PRAHA 4
Tel. +420/244 016 055
Fax +420/244 016 088
Hotline +420/244 016 050
Web <http://www.abstudio.cz/>
Email abstudio@abstudio.cz

United Kingdom

Adris Limited

Riverside House, Brunel Road
Totton, Southampton, Hampshire
SO40 3WX England
Tel. +44 023 8086 8947
Fax +44 023 8086 1618
Hotline +44 023 8086 9995
Web <http://www.adris.co.uk/>
Email sales@adris.co.uk