Advance Concrete

Tutorial
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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.
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.

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](image)

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.
   - In the “Bubbles and texts” area:
     - 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 the Angle drop-down list select a 0° bubble rotation angle.
     - From the Position drop-down list select On the left and on the right to place bubbles on both ends of the grid axes.

   In the “Numbering” area, select 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.

   ![Figure 3: Bubbles tab – Numbering options](image)
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.

4. Click <OK> to confirm.

---

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

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.

---

**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 4: Numbering option for the Y axis

Figure 5: Labeled axis

Figure 6: Renumbering the G and 22 axes

Figure 7: 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.

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.
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.
3. In the properties dialog box, in the **Level name** field, enter the new level name: **Underground Level 2**.

**Note:** Make sure to keep @ in the level name as it displays the level altitude.

![Figure 10: Renaming Level 1](image)

4. Set the level elevation to **3 m**.

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

![Figure 11: The Pilot displays the renamed levels](image)

**Step 3: Activate the Layer Manager**

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

1. In the Pilot, in **Model** mode, right click **Project** and select **Project settings** from the context menu.

2. On the **Project Settings** toolbar, click ![Layer Manager](image) to access the "Project preferences" dialog box.

3. Select the **Layer naming** tab.

4. Select the **Automatic layer name management** option.

5. Click **<OK>**.

![Figure 12: “Project preferences” dialog box - Layer naming tab](image)
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 Settings** toolbar, click ![image](image) to display the working definition dialog box. Make the following settings:
   - Click lengths "Type" and select **Meter**. In the "Precision" column select the **0.00** 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 13: "Working units definition" dialog box](image)
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 14: Columns in the model](image)

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 15.

![Figure 15: Measurements for the L section columns](image)

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

![Figure 16: Creation of the first column](image)

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

<table>
<thead>
<tr>
<th>Rotation angle</th>
<th>0°</th>
<th>90°</th>
<th>180°</th>
<th>270°</th>
</tr>
</thead>
<tbody>
<tr>
<td>Result</td>
<td>![21']</td>
<td>![21']</td>
<td>![21']</td>
<td>![21']</td>
</tr>
</tbody>
</table>

![Figure 17: Columns orientation with rotation angle](image)

9. Press **Esc** to finish.

![Figure 18: L section columns](image)
**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 19: Moving the C21' column](image)

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 20.

![Figure 20: L columns after adjusting position](image)

**Step 2: Create columns with T sections**

This step describes how to create columns with T sections with different rotation angles.

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.
   - From the Material drop-down list, select the Steel/Concrete/Wood.
   - 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 0º.

7. Place columns at the following intersections: C 20 (90º), F’20 (270º) and D17 (180º).
As in the previous step, use the AutoCAD® Move command to adjust the column position 20 cm in the X or Y direction.

**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.

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.

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.
Step 3: 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, select a section type and modify the size.
   - From the Material drop-down list, select the Steel/Concrete/Wood.
   - From the Type drop-down list, select the Rectangular type.
   - Modify the section size: Set the Height and Width to 60.

![Figure 28: Measurements for the rectangular column sections](image)

4. Click <OK>.
5. Place the columns at the D18, D19 and D20 axis intersections, with a 0° rotation angle.
6. Place the other columns at the E18, E19 and E20 intersections.

The following figure shows the result.

![Figure 29: Creation of the D18, D19, D20 and E18, E19, E20 columns](image)
Use the buttons on the AutoCAD® View toolbar to change the view angle. For a more realistic presentation of the model, use the AutoCAD® Visual Styles.

Figure 30: 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.

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

![Figure 33: Setting the end point of the first straight wall](image)

5. 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, grtcwall.

![Figure 34: Walls along the C axis](image)

**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.

1. On the **Model** toolbar, **Structural** flyout, click ![icon]
2. Click the C17 axis intersection to define the start point of the wall.
3. Click D17 axis intersection to set the end point.

Figure 35: Creation of continuous wall between the C17 and D17 axis intersections

4. Draw the walls by clicking the following intersections: E17, F'17, F'21' and C21'.

5. Press Esc.

Figure 36: Underground level – columns and outer walls

Step 3: Create inner walls with 25 cm width

Next, create inner walls with 25 cm width between the D21', D20, E20 and E21 axis intersections (Figure 37).

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

2. On the Smartbar, enter 25 to draw a wall with a 25 cm width.

3. Press Enter.

4. Click the start point at the D21' axis intersection.

Figure 37: 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 38: Inner walls](image)

**Step 4: Create inner walls with 30 cm width**

1. On the **Model** toolbar, **Structural** flyout, click ![image](image).
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 39: Creation of the first inner wall](image)

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 40: Underground level – outer and inner walls](image)
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 \[\text{Material choice} \].
2. On the Smartbar, make 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. Click the D19 axis intersection to define the wall start point.
4. Click the E19 axis intersection to define the end point.
5. Press Enter.

Adjusting the wall position
Next, align the wall to the bottom edge of the column section 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 41: First brick wall: D19 – E19
Figure 42: Wall aligned to the bottom edge of the column section
Next, create another full brick wall between the E19 and E18 axis intersections.

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

Using the same process, draw another full brick wall between the D19 and D18 axis intersections, with 0.175 justification value.

![Figure 43: Underground level – outer and inner walls](image)

**Changing the wall representation style**

2. On the Smartbar, from the Style drop-down list, select **Full bricks** to change the wall representation style.
3. Press **Enter**.

The wall representation is changed.

![Figure 44: D19-D18-E18-E19 walls](image)

![Figure 45: New representation of the walls](image)
Use the buttons on the AutoCAD® View toolbar to change the view angle. For a more realistic presentation of the model, use the AutoCAD® Visual Styles.

Figure 46: 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 47: Final Model](image)

---

**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.

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, `grtcbeam`.**
Step 2: Create continuous beams

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.

Next, create beams along the D axis, between the D17, D18, D19 and D20 columns and along the E axis, between the E17, E18, E19 and E20 columns.
Step 3: Modify the beam properties
This step describes how to adjust the beam position.

Modifying the position of the beams
1. Select 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.
   Using the same method, define a custom justification of \(0.10\) m for the beam located on the E18-E19 columns.
Use the buttons on the AutoCAD® View toolbar to change the view angle. For a more realistic presentation of the model, use the AutoCAD® Visual Styles.

Figure 54: Final model - The beams
Lesson 6: Creating slabs

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

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 O (Option) and press Enter.
5. On the command line, type 1.00 for the extension value for the slab and press Enter.
6. Click the areas (as shown in the Figure 56) to create the slabs. The slabs are automatically created using boundary detection.
7. Press Esc to finish.
The slabs are numbered from $S_1$ to $S_{12}$.

*Figure 57: Slabs*

*Use the buttons on the AutoCAD® View toolbar to change the view angle. For a more realistic presentation of the model, use the AutoCAD® Visual Styles.*

*Figure 58: Final model*
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.

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 60 illustrates the ramp profile.

Step 1: Define the ramp slope

This step describes how to define the ramp slope using an AutoCAD® polyline.
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.

5. Press Esc to finish.

   Draw a segment to close the polyline using the Polyline tool 🔄.
Next, rotate the last segment to get the correct profile (see Figure 63).

1. Select the segment.
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.

**Step 2: Create the ramp surface**

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 64.

---

**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.

5. Delete the polyline.

*Use the buttons on the AutoCAD® View toolbar to view the model 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.

Use the AutoCAD® View toolbar to view the ramp from the side as in the Figure 67.

1. Select the ramp surface.

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.

Step 4: Bind the ramp to the structural elements

View the model in 3D to see the elements.

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

The selected elements adjust to the roof surface.
Use the buttons on the AutoCAD® View toolbar to change the view angle. For a more realistic presentation of the model, use the AutoCAD® Visual Styles.

To cancel the shading, click on the AutoCAD® Visual Styles toolbar.

Use the AutoCAD® View toolbar to return to the plan view.
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 70: Selection of the S12 slab.](image)

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 71: Setting the slab upper level](image)

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.

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 72: Beams for the ramp auxiliary element

Figure 73: Ramp
In the next steps, create the second flight of the ramp. The slope profile must be defined as in a front view of the ramp.

Step 6: 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 \[ \text{Polyline} \].
2. As the ramp is made up of the S11 slab, snap to the slab corners to define the ramp length.
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.
Step 7: 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 77: Draw the second ramp](image)

5. Delete the polyline.

Step 8: Adjust the position of the second ramp

*Use the AutoCAD® View toolbar to view the ramp from the side as in the Figure 78.*

1. Select the ramp surface.

![View toolbar](image)

![Figure 78: Selection of the second ramp](image)

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

![Value: -1.25 m](image)

3. Press Enter to finish.
The ramp is correctly placed.

Figure 79: Ramp in side view

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

Step 9: Bind the structural elements to the ramp surface

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.

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

Figure 80: AutoCAD® View toolbar

Figure 81: Ramp surface
Step 10: Hide the control surface

1. In the Pilot, click \( \text{ } \) to display the quick menu.

2. Select Display by type.

![Figure 82: Pilot – Quick menu](image)

3. In the "Element visibility" dialog box, select Control surface (roof/ramp). The sign \( \checkmark \) turns into \( \times \).

![Figure 83: “Element visibility” dialog box](image)

4. Click Apply.

5. Click Close to close the dialog box.

![Figure 84: 3D view of the model](image)
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 85: Openings in the model
Before starting

Before creating the openings, create the walls for the two elevators. Zoom to the model area as in Figure 86.

![Elevator area](image)

Figure 86: 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.

4. In the properties dialog box, on the Junctions tab, select the Extremity 1 and Extremity 2 options and enter 0 m.

![Properties dialog box](image)

Figure 87: "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 88: Example of Junction effects

5. Click <OK>.

Note: Activate the ORTHO mode to draw the walls.

6. Define the start point at the D18 axis intersection.

7. On the Smartbar, select a suitable justification:

8. Move the cursor in the X direction, enter 2.4 on the command line and press Enter.

9. Move the cursor in the Y direction, enter 2.4 and press Enter.

10. Move the cursor in the –X direction, enter 2.4 and press Enter.

11. Move the cursor in the –Y direction, enter 2.4 and press Enter.

12. Press Esc to finish.

Figure 89: The walls of the 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 90: Selecting the elevator walls
2. On the Smartbar, type 41 for the new priority.
3. Press Enter.

![Figure 91: The edge between the elevator walls](image)

**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.
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.

![Figure 92: Defining the symmetry axis](image)

![Figure 93: Left and right walls of the second elevator](image)
**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 94: Extending the elevator wall](image)

Use the same process for the upper wall.

![Figure 95: Elevator walls](image)

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

![Figure 96: Moving the slab symbol](image)
**Drawing the auxiliary polyline for 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 97: Additional polyline](image)

**Creating walls along the polyline**

1. On the Model toolbar, Structural flyout, click 🖼️.
2. On the Smartbar, make the following settings:
   - 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 98: Walls](image)
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.

3. Click <OK> to validate.
Step 1: 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 101: Selection of the D21’-D20 wall](image)

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

![Figure 102: Opening on the D21’-D20 wall](image)

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

![Figure 103: Wall openings](image)
Step 2: Create window openings

1. On the Model toolbar, Openings flyout, click \[\text{\textbullet} \text{\textbullet}\].
2. Select the wall along the F axis.

3. On the Smartbar, click \[\text{\textbullet} \text{\textbullet}\] to access the window library and select an opening type.
4. In the dialog box, click Add. A second panel appears. The Libraries tab lists the openings available in the library.
5. 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.
6. Select LD 160X135 C90 D8 EFGH8888.
7. Click Import. The opening is imported into the current project. All openings available in the project are listed in the openings list and on the Smartbar.
8. Click <OK>.

Figure 104: Wall along the F axis

Figure 105: Importing a window opening into the current project
Draw the window:

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

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

Using the same process, create another window opening of the same type on the **F19-F20** wall. The window opening has the following size: **LD 300x90 C130 D8 EFGH8888**

---

**Figure 106: Creating the window opening**

**Figure 107: Creation of the second window**

**Figure 108: Window openings**
Use the AutoCAD® View toolbar to view the window openings.

On the AutoCAD® View toolbar, click \( \text{ } \) to return to the plan view.

Step 3: Create the door openings

This step describes how to create the elevator door openings.

1. On the Model toolbar, Openings flyout, click \( \text{ } \).
2. Select the upper walls of the elevator.

3. On the Smartbar, click \( \text{ } \) to access the door library and select an opening type.
4. In the dialog box, click Add. A second panel appears. The Libraries tab lists the door openings available in the library.
5. From the drop-down list on the right side of the dialog box, select the Right lintel opening shape. The Libraries tab lists all doors with a right lintel.

6. Select LD 70x215 D8 EFGH8880.

7. Click Import.

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

8. Click <OK>.

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.

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

7. Press Enter.
The door opening is created.

![Image: Creation of the first door](image1)

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

![Image: Elevator doors](image2)

**Step 4: Create the slab openings**

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

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

![Image: Opening symbol](image3)

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.

6. Click a point to define the opening position.

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

Using the same process, create a rectangular slab opening for the space between the elevators. The opening size is **R60x200**.
Step 5: 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 119: Placing the depression](image)

Using the same process, create a depression for the control panel of the other elevator.

Next, 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.
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 120: Elevator openings](image)
Using the same process, create an opening of the same type and size in the bottom wall.

**Adjust the opening position**

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

Figure 121: Elevator area – Wall openings

Figure 122: Middle wall openings
Step 6: 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.
1. On the **Model** toolbar, **Openings** flyout, click 🟡.

2. Define the line start point in the middle of the **D19-E19** wall.

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**.

4. Press **Enter**.

5. Press **Esc** to finish.

   The first beam opening line is created.

   Using the same process, create two additional opening lines.

   ![Figure 126: Creation of the first beam opening line](image)

   ![Figure 127: Placing the opening](image)
Use the AutoCAD® View toolbar to view the beam openings in the front view.

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.

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 cursor parallel with the beam and click the second point.

![Figure 130: Create the beam opening line](image130.jpg)

6. Press **Enter** to finish.

![Figure 131: Openings](image131.jpg)

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 132: Representation of the openings](image132.jpg)
Step 7: 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 [image].

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 133.

4. Press Enter to finish.

The polygonal opening is created.
Step 8: Complete the model by adding all remaining openings

Create round openings with 5 cm 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, make the following settings:
   - Set the hole size to D5.
   - Set the wall sill: 265.
   - Select a crossing wall opening.

Using the same process, create round wall openings in front of the beam openings.

---

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

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Figure 135: Created openings

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Figure 136: Wall openings – Front view
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.

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.

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.

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**.
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).

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

5. Press Enter to finish.
For a clear view, move the slab symbol using the slab symbol grip point:

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

![Elevation pit parameters](image)

**Figure 146: 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).

![Elevator pit](image)

**Figure 147: Elevator pit**

*Use the AutoCAD® View toolbar to view the elevator pit.*

![View toolbar](image)

**Figure 148: Elevator pit – Front view**

*On the AutoCAD® View toolbar, click ![View button](image) to return to the plan 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, make 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 149.

   ![Figure 149: Creation of the first depression](image)

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

![Figure 150: Creation of the second hole](image)
In the Pilot, right click **Underground level 2** and select **View** from the context menu to display the **Underground** level.

*Use the buttons on the AutoCAD® **View** toolbar to change the view angle. For a more realistic presentation of the model, use the AutoCAD® **Visual Styles**.*

![View](image)

![Visual Styles](image)

Figure 151: View of the model in 3D

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

![Orbit](image)

Figure 152: AutoCAD® **Orbit** 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 Level 1**, **Intermediate Level** and **Ground Level** and copy the elevator elements from one level to another.

![Levels of the Model](image)

**Figure 153: 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.

![Recopy the Underground Level 2](image)

**Figure 154: 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.

Delete the elements that are not necessary in the new level: S5, S8 slabs and the D19 – E19 wall.

Step 2: Modify the elevator walls

Selecting the elevator walls
1. Include the elevator area in a window selection.
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 158: Selecting the walls using quick selection](image)

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 159: Properties dialog box – Definition tab](image)

2. Click **<OK>**.
Use the AutoCAD® View toolbar for a front view of the elevator walls.

Figure 160: Front view of the model

On the AutoCAD® View toolbar, click to return to the plan 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 161: Creating the top level

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

Figure 162: “Level properties” dialog box – Properties tab
3. Click <OK>.  
The new level is created.

Figure 163: Top level displayed in the Pilot

Step 4: Rename the levels

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

Figure 164: Accessing the Level 1 properties

2. On the Properties tab, enter the new name: Underground Level.

3. Click <OK>.

Figure 165: Renaming a level
Step 5: Add elements in the intermediate level

Next, create a slab in the top level, on the elevator walls.

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

![Figure 166: Access the "Slab" tool from the pilot context menu](image)

2. On the Smartbar, set the slab thickness to **18 cm**.

3. Input the slab corners, point by point as in Figure 167.

![Figure 167: Setting the points of the slab creation](image)

4. Press **Enter** to finish.

The slab is created.
Use the AutoCAD® View toolbar for a front view of the elevator walls.

![View toolbar](image)

Step 6: Create the Ground Level

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

![Context menu](image)

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>.

![Level properties](image)

**Note**: Notice that in the Pilot the level elevation is automatically displayed.
Step 7: Copy elevator elements to the Ground Level

Before starting, on the AutoCAD® View toolbar, click for a plan view.

1. Select 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. Select the Copy option.

5. Click <OK>.

The elevator elements are copied on the Ground Level.
Step 8: Create a slab on the Ground Level

Create a slab on the elevator walls.

1. On the Model toolbar, Structural flyout, click .
2. Click the slab corners as in the Figure 174.

3. Press Enter to finish.
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 as shown in the Figure 176.

![Figure 176: “Properties” dialog box – Defining slab edges](image)

4. Click <OK>.

![Figure 177: Slab edges](image)

Use the AutoCAD® View toolbar for a front view of the elevator walls.

![View toolbar](image)

![Figure 178: Front view of the building](image)
Step 9: Modify the elevator wall heights

1. Select the elevator 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>.

The height of the walls has changed.
Use the buttons on the AutoCAD® View toolbar to change the view angle. For a more realistic presentation of the model, use the AutoCAD® Visual Styles.

In the Pilot, right click Tutorial and select Enable to view the entire building in 3D.
Step 10: 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, double click **Level 1** to activate and display it in top view.

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

4. Select the middle beam opening lines and delete them.
5. Select the D19-E19 beam.

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

7. On the Definition tab, set the beam upper level, relative to the upper level, to 0.95 m.

7. Click <OK>.

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

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

3. On the **Definition** tab, make the following settings:
   - In the **Section** drop-down list, change the section of the beams to **R40x155**.
   - Set the beam upper level to **0.95 m**.

   ![Figure 191: “Properties” dialog box – Definition tab](image)

   3. Click **OK**.

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

   ![Figure 192: Adjusted beam](image)

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

   ![Figure 193: Adjusted wall](image)
**View the entire building in 3D**

In the Pilot, in Model mode, right click Tutorial and select Enable from the context menu to display all levels.

![Figure 194: Preparing to view the model in 3D](image)

Use the buttons on the AutoCAD® View toolbar to change the view angle. For a more realistic presentation of the model, use the AutoCAD® Visual Styles.

![Image of AutoCAD View toolbar and Visual Styles](image)

Notice that the column height must be adjusted.

![Figure 195: 3D View – the created slab](image)

Before starting, on the AutoCAD® View toolbar, click 🗋 for a plan view.
1. Select the columns on each side of the slab (D18, D19, D19 and E18, E19, E20).

Figure 196: Selection of the columns

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

3. On the Definition tab, set the value for the upper level to 0.95 m.

Figure 197: “Properties” dialog box - Definition tab

3. Click <OK>.
View the entire building in 3D

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

Use the buttons on the AutoCAD® View toolbar to change the view angle. For a more realistic presentation of the model, use the AutoCAD® Visual Styles.

Figure 198: 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 199: Drawings of the Model](image1)

**Step 1: Create section cuts**

*Creating a section cut*

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

![Figure 200: Creating the first cut](image2)
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 201 and click a point.

**Calculating the cut**

1. In the Pilot, click 🗿 to enter the Drawings mode.
2. In the Pilot, the Section A-A appears. A red mark indicates that the view is not updated.
3. Double click **Section A-A** to calculate it.

![Figure 203: Section A-A](image)

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

![Figure 204: Cuts](image)

**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. In the Pilot, click to activate the Model mode.
2. On the Model toolbar, Drawings flyout, click .
3. Draw the cutting plane as you draw an AutoCAD® polyline (see Figure 205).
4. Press Enter to finish.

Figure 205: Created cut

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.
4. Press Enter to confirm.
5. Click a point on the drawing to input the back cutoff plane.

The elevation is created.

Figure 206: Creating the elevation
Step 4: Create a top view

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

1. In the Pilot, in **Model** mode, activate the **Underground level 2**.

2. On the **Model** toolbar, **Drawings** flyout, click . The top/view creation wizard starts.

3. 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

4. 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.
   - 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.

   ![Setting the line thickness of the cut and uncut visible edges](image)

   ![Setting the view parameters](image)

5. 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

5. 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 210.
   - In the "Definition of hatches for the hidden objects" area, select **No hatch**.

6. 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**.

7. Click **Next** until you reach the "Visual elements" dialog box to modify the visual elements.
8. In the "Visual elements" dialog box, select **Display formwork symbols** to display symbols on the drawing.
9. Click **Parameters**.
10. In the "Multiple symbols" dialog box, select the set of objects for which to add symbols: select only **Beams** and **Columns**.

![Figure 212: "Multiple symbols" dialog box](image)

11. Set the symbol relative position according to the object for every object type:
   - For beams: ![Beam Symbol]
   - For columns: ![Column Symbol]

The preview area displays the selected attributes: the beam section.
12. Click **<OK>**.
13. Click **Finish**.

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

![Figure 213: Top view](image)
Step 5: Create a local top/bottom view

In the Pilot, in Model mode, enable the Ground level and display all other 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.

3. Click Finish.
4. Draw the local view definition polyline by three points and include the elevator walls as in Figure 216.

5. Right click the polyline and select Activate and update the drawing from the menu.
The top view is calculated and **Drawings** mode is activated.

**Step 6: Create an isometric view**

*Note:* Before creating the isometric view, it is necessary to select the view angle from the AutoCAD® **View** toolbar for a suitable 3D view of the model.

1. On the **Model** toolbar, **Drawings** flyout, click ![Isometric view button](image).

2. Click **OK** to validate.
Advance switches automatically to the **Drawings** mode. The isometric view is displayed.

![Figure 220: Isometric view of the building](image)

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

### Modifying the isometric view 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 221: Accessing the Isometric View properties](image)
2. In the properties dialog box, unselect the **Consider the slabs** option to hide the slabs from the isometric view.

![Image of properties dialog box with unselected Consider the slabs option]

**Figure 222: 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.

![Image of updated isometric view without slabs]

**Figure 223: Updated Isometric view – Without slabs**
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.

   ![Properties dialog box – Visible objects tab](image)

   Figure 224: “Properties” dialog box – Visible objects tab

3. Click <OK>.

   The cuts that have modifications are marked with ![modified cut symbols](image). It is necessary to update the drawing to see the result.

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

   ![Section A-A – updated](image)

   Figure 225: Section A-A – updated
**Updating several drawings**

1. On the Model toolbar, Drawings flyout click ![Ok](image). A dialog box displays the list of all created drawings. The updated elements are marked with ![Ok](image) and the elements that are not up-to-date are marked with ![Not updated](image).

   ![List of drawings table](image)

   Figure 226: List of drawings table

2. Select all the sections that are not updated.

   ![Selection of the cuts](image)

   Figure 227: 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.)

   ![Selection of cuts](image)

   Figure 228: 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**.

   ![Diagram](image)

   **Figure 229: 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.

   ![Ungroup screenshot](image)

   **Figure 230: Selection of the cuts**

2. In the “List of drawings” dialog box, select **Section D-D** and click **<OK>**.

   ![List of drawings](image)

   **Figure 231: Drawing selection**
The **Section D-D** is separated from the drawings group.

![Figure 232: Ungrouped section cut in the Pilot](image)

**Step 9: Create a reinforcement drawing**

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 233: Activating Underground level 1](image)
2. Select the wall along the 21' axis.

3. On the Model toolbar, Drawings flyout, click [Diagram Icon].

4. In the "Creation of a reinforcement drawing" dialog box, from the 3 Views branch, select the Cut left, top view and cut face.

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 236: Setting the line thickness](image)

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 237: “Hidden Objects” dialog box](image)

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.
    - Select Element names to display the wall name in the view title.
    - Select Place level name in the title box to display the level name.
    - Unselect Quantities.

   ![Figure 238: Setting the information to display in the title](image)

11. Click Finish.
The reinforcement drawing is created.

Step 10: Create a new empty drawing

1. In the Pilot, in Model mode, double click Tutorial to activate all the levels.
2. Right click Tutorial and select New drawing from the context menu.

The new drawing is displayed in Drawings mode.
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 242: “Drawing properties” dialog box](image)

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 a level and select Enable from the context menu.

2. On the Model toolbar, Drawings flyout, click .

3. In the “Select a level” dialog box, select Level 1.

![Figure 243: “Select a level” dialog box](image)

3. Click <OK>.

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

![Figure 244: Plan view](image)
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.

![Accessing the Plan view properties](image1)

Figure 245: 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.

!["Floor plan" dialog box](image2)

Figure 246: "Floor plan" dialog box

The elevator walls are displayed in the plan view.

![Plan view – Impact on upper level](image3)

Figure 247: Plan view – Impact on upper level
Lesson 12: Creating dimensions

In this lesson, you will dimension a plan view of the Underground level. You will learn how to:

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

Figure 248: Dimensions on the plan view
**Before starting**

Before starting, create a plan view for the first **Underground level**. It will be used it in this lesson.

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

![Figure 249: Creating a plan view](image)

3. Click **<OK>**.

The plan view is created.

![Figure 250: Pilot – Plan view](image)

*Use the AutoCAD® dimension settings to always keep the text between the extension lines.*

1. On the **Views** toolbar, click ![Views](image).
2. From the **Styles** list, select **GrDimensions**.

![Figure 251: “Dimension Style Manager” dialog box](image)
3. Click Modify.

4. In the "Modify dimension style" dialog box, on the Fit tab, make the following settings:
   - Select the **Always keep text between ext lines** option.
   - Unselect the **Suppress lines if they don’t fit inside** extension lines.

5. Click <OK> to validate and return to the "Dimension Style Manager".

6. Click Close to finish.

**Step 1: Create intersection dimensions**

In this step, you will dimension the elements along the C axis (see the red line in Figure 254), taking the columns into account.

1. On the Views toolbar, click 📸.

2. On the Smartbar, click 📜 to access the properties.

**Note:** The settings made during element creation are used as default settings in the current project.

3. In the properties dialog box, expand the **Model elements** branch and select the elements to dimension: only the columns.

**Tip:** The elements taken into account are preceded by a blue sign ✔️ while the elements not taken into account are preceded by a red ✗️. Clicking twice on the ✔️ sign unselects all elements. You can then select the elements to dimension.
4. Click <b>OK</b> to confirm.
5. Click two points to define the dimension line as shown in Figure 254.

![Figure 254: Create the first intersection dimension](image)

6. Press <b>Enter</b>.
7. Click a point on the drawing to set the intersection dimension position.

![Figure 255: Placing the intersection dimension](image)
The dimension line is automatically created.

Figure 256: Dimension line

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

Figure 257: Placing the intersection dimensions

**Note:** Make sure the line crosses the columns.
Using the same process and settings, create intersection dimensions as shown in Figure 258.

![Figure 258: Intersection dimensions](image)

**Taking the grid axis into account**

Next, dimension the columns taking the grid into account. Zoom to the D20 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.
   - Select Grid.
4. Click <OK> to validate.

![Figure 259: Selecting the intersected elements](image)
5. Click two points to define the dimension line.
6. Press **Enter**.
7. Click a point on the drawing to set the intersection dimension position.

The dimension line is automatically created.

Using the same process, dimension the **D20** column taking the grid into account. Also, dimension each side of the columns in the drawing as in Figures 260 and 261.

**Figure 260: Intersection dimensions for the C18, C19 and C20 columns**
Taking other elements into account

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

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.
   - Select Beams, Columns and Walls.
4. Click <OK> to validate and close the dialog box.
5. Click two points to define the dimension line parallel with the top side of the column. The line must cross the column and the wall.

![Figure 263: The dimension definition line](image)

6. Press **Enter**.

7. Click a point on the drawing to set the intersection dimension position.

![Figure 264: D20 column – Dimension by intersection taking the walls into account](image)

Using the same process and settings, dimension each side of the **D20** column.

![Figure 265: D20 column – Dimension by intersection taking the beams and walls into account](image)
Displaying decimals and millimeters in the exponent

Next, 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.

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.
   - Select the suppress trailing zeroes option to not display the trailing zeros.
   - Select the mm in exponent option to display millimeters in the exponent.

5. Click <OK>.

Note: When the distances are greater than 1 m, they are expressed in meters and if they are less than 1 meter, they are expressed in centimeters.
Step 2: 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 only possible in Drawings mode.

1. On the Views toolbar, click ![View Tool](image)
2. Select the middle, left and right elevator walls.

![Selecting the elevator walls to create the associative dimension](image)

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

The associative dimension is automatically created.

![Associative dimension](image)
Selecting the dimensions to display

1. Select the associative dimension.
2. On the Smartbar, click to access the properties.
3. Select the Definition tab to add or remove dimensions.
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.

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 text and click <OK>.

5. Click <OK> to finish.

Using the same tool, create other associative dimensions for the openings in the D20 – D21' and E20 – E21' walls.

1. On the Views toolbar, click .

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

3. On the Definition tab, use the arrows to select only Dimension of openings in the Dimension to display list.

4. Select the D20 – D21' wall and press Enter.
5. Place the associative dimension on the drawing.

*Figure 278: Associative dimension for the D20 – D21' columns*

*Use the AutoCAD® Repeat command, to repeat the last executed command – in this case, grtcdimassoc – and create all necessary opening dimensions.*

**Step 3: Create a dimension by points**

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

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

*Figure 279: Defining the dimension line position*
3. Click the first point of the dimension: the top left corner of the C21' column section.

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

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

Step 4: 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 D20 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 282: Selecting the symbol](image)

4. Place the symbol next to the **D20** column.

![Figure 283: Symbol](image)

**Editing the symbol**

1. Select the symbol.

![Figure 284: Selecting the symbol](image)

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 285: Symbol properties dialog box – Definition tab](image)

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

![Figure 286: “Text Formatting” dialog box](image)

4. In the edit area, modify the symbol. For example, type **Column**, delete the `<Name>` attribute and press **Enter** to display the section size on a new line. This text appears on the drawing.

![Figure 287: Setting a name for the symbol](image)

5. On the AutoCAD® **Text formatting** toolbar, click **OK**.

6. In the properties dialog box, click `<OK>`.

![Figure 288: Column symbol - new representation](image)
**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.

![Symbol with line and arrow](image)

Figure 289: Symbol with line and arrow

**Saving the symbol**

1. Select the symbol.
2. On the Smartbar, click ![Properties](image) to access the properties dialog box.
3. On the **Definition** tab, click **Save**.

![Symbol properties dialog box](image)

Figure 291: Symbol properties dialog box – **Definition** tab
4. In the “Save symbol” dialog box, enter the new name and click **Browse** to select the folder where the symbol will be saved.

![Figure 292: “Save symbol” dialog box](image)

5. Click <**OK**>.

6. In the properties dialog box, click <**OK**>.

**Step 5: 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.

1. On the **Views** toolbar, click ![image](image)

2. Include the model in a window selection and press **Enter** to validate.

![Figure 293: Selecting the elements](image)
3. In the “Multiple symbols” dialog box, check only the **Column** row to add symbols only for the columns and click **<OK>**.

![Figure 295: “Multiple Symbols” dialog box](image)

Symbols are placed for all the columns on the drawing.

![Figure 295: Symbols for columns](image)
**Moving the symbols**

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

1. Select a symbol.

![Figure 296: Selecting the symbol](image)

2. Click the symbol grip point.
   
   The symbol is attached to the cursor.

![Figure 297: Moving the symbol](image)

3. Click a point on the drawing to position the symbol.

4. Press Esc to finish.

![Figure 298: Symbol](image)

Using the same process, move all the symbols.
Lesson 13: 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. The beam is reinforced using straight bars and rectangular frames, distributed differently in the three areas of the beam. The reinforcement drawing requires a custom view arrangement, with 3 cut sections and an elevation.

You will learn how to:
- Modify the view arrangement.
- Delete a view from the reinforcement drawing.
- Add views to the reinforcement drawing.
- Create a straight bar.
- Create a rectangular frame.
- Create a free distribution.
- Create a linear distribution.

Before starting:

1. In the Pilot, click to activate the Model mode, and then enable Underground level 2.
Step 1: Create the reinforcement drawing

In this step, you will create a reinforcement drawing for the F’20-E20 beam using an existing template with a cut, an elevation and a top view. During the steps of this lesson, the template will be modified as the beam reinforcement drawing requires a custom view arrangement.

1. On the Reinforcement toolbar, click.
2. Select the F’20-E20 beam and press Enter.

![Figure 301: Selecting the beam](image)

5. Select the Cut left, top view and elevation face view template.

![Figure 302: Reinforcement view template](image)

4. Click Next to go to the “View parameters” dialog box where the view properties are set. In this example, the position of the Left cut view is modified.

5. In the “View parameters” dialog box, select the Left cut, then click [on object] to set the cut position on the drawing. The reinforcement view is calculated and displayed.
6. Using the mouse, place the cut line on the Top view, in the first half of the beam (see Figure 303). The cut position is displayed in the Cut position field in the “View parameters” dialog box.

![Figure 303: Top view – Defining the cut line position](image)

**Warning!** Make sure the 3D power is activated! It allows Advance to manage bar representation in all reinforcement views.

10. Click **Next**.

11. In the “Visible Objects” dialog box, in the “Definition of the cut visible edges” area, set the edge style for the visible objects:
   - Uniform edges
   - Line thickness: **0.50 mm**.

![Figure 304: “View parameters” dialog box – Displayed cut position](image)

![Figure 305: “Visible objects” dialog box](image)
11. Click **Next**.

12. In “Hidden objects” dialog box, in the “Definition of hidden objects” area, set the edge style for the hidden objects representation.
   - Uniform edges
   - Line thickness: **0.25 mm**.

13. Click **Next**.

14. In the “Finishings” dialog box, in the “Cut interruption lines” area, in the **Length** field, set the cut interruption line lengths: **0.1 m**.

13. Click **Next** until the “Visual elements” dialog box is displayed.

14. In the “Visual elements” dialog box, click the arrow next to the scale and select **1:25** from the displayed list.

14. Click **Next**.
15. In the “Title” dialog box, make the following settings:

- In the **Prefix** field, enter the view name prefix: **Reinforcement Drawing-Beam**. This title appears on the drawing, above each view.
- Unselect the **Quantities** option so that the bar 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 title text height.

![Figure 309: “Title” dialog box](image)

15. Click **Finish**.

The Pilot switches automatically to the **Drawings** mode and the reinforcement drawing is displayed.
Step 2: Customize the reinforcement drawing

Only the elevation and the cut are necessary, therefore, in this step, you will delete the top view. Additionally, you will create two more cuts.

 Deleting a view

1. On the Reinforcement 3D viewer toolbar, click .
2. Include the top view in a window selection.
3. Press Enter.

The reinforcement drawing contains only the elevation and the cut.

 Adding a view

Next, add another cut to the reinforcement drawing.

1. On the Reinforcement 3D viewer toolbar, click .
2. Define the cutting plane by drawing a vertical line across the beam, in the middle area.
3. Press Enter.
4. Move the cursor in the X direction and click a point, to define the cut depth.
5. Place the cut by clicking in the drawing in the desired position. Using a similar process, create another cut on the beam, in the right area (see Figure 314).

![Figure 314: Cut position on the beam](image)

There are three cuts on the reinforcement drawing.

![Figure 315: Reinforcement drawing](image)

**Note:** If necessary, the cut position can be adjusted in the reinforcement view properties dialog box, on the **View parameters** tab.

![Figure 316: Setting the properties for the left cut](image)

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

1. Include the three cuts in a window selection.
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 view.

![Figure 317: Moving the cuts](image)

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

**Creating a reinforcement drawing template with custom view arrangement**

*Note:* If during the project other reinforcement drawings with custom view arrangement are necessary, a new reinforcement drawing template should be created.

For a better understanding, on the Reinforcement toolbar, Drawings flyout, click ![icon](image) to display the first dialog box of the reinforcement drawing creation wizard.

![Figure 318: “Creation of a reinforcement drawing” dialog box – 3 Views branch](image)

The templates listed in the left panel of the dialog box are defined in the *Reinforcement drawing models.txt* file stored in the `\Documents and Settings\All Users\Application Data\Graitec\AdvanceConcrete\[release]\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.

<table>
<thead>
<tr>
<th>Port Name</th>
</tr>
</thead>
<tbody>
<tr>
<td>TOP</td>
</tr>
<tr>
<td>BOTTOM</td>
</tr>
<tr>
<td>CUT_LEFT</td>
</tr>
<tr>
<td>CUT_RIGHT</td>
</tr>
<tr>
<td>ELEVATION_LEFT</td>
</tr>
<tr>
<td>ELEVATION_RIGHT</td>
</tr>
<tr>
<td>ELEVATION_FACE</td>
</tr>
</tbody>
</table>

![Figure 319: List of the view port names](image)
For example, select **Dynamic reinforcement: left, front, top, isometric**. This template contains four views.

---

**Figure 320: Dynamic reinforcement template**

To create a custom view arrangement, with 3 cut sections, for a beam reinforcement drawing similar with the one presented in this lesson, enter the name of the new template, **#Reinforcement view for beams with 3 cut sections**, and the coordinates from the Figure 321 (in percentage) in the Reinforcement drawing Models text file.

---

**Figure 321: Reinforcement drawing model text file**

After saving the .txt file, the first dialog box of the reinforcement drawing creation wizard displays the new template.
Changing the scale of the drawing

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

![Figure 323: Setting the drawing scale](image)

**Step 3: 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.

- Use intersection dimension tool on **Face elevation 1** as in Figure 324.

![Figure 324: Elevation face 1- intersection dimensions](image)

- Use the point dimension tool as in Figure 325.

![Figure 325: Face elevation 1- point-by-point dimension](image)
• Use the level dimension to mark the slab and beam level as in Figure 326.
• Reverse the level dimension if necessary.

Figure 326: Elevation face 1- placing the dimension origin

Dimension the sections also using the intersection dimension and the level dimension tools. Figure 327 displays the result.

Figure 327: Dimensioned cuts

Step 4: Create straight bars on the Face elevation 1 cut

In this step you will place the necessary straight bars on the Face elevation 1 view of the reinforcement drawing.

Creating a straight bar with hooks

1. On the Reinforcement toolbar, Reinforcement Bars flyout, click.
2. On the Smartbar, click to access the properties dialog box.
3. On the properties dialog box, make the following settings:
   On the **Definition** tab:
   - Set the bar diameter to **20 Ø**.
   - In the field next to the **Con. covers** drop-down list, enter **0.03 m** to set the concrete cover.
   - For the concrete covers, select **All** and enter **0.03** to set the concrete cover on all bar segments.

   ![Image: Properties dialog box – Definition tab](image)

   On the **Hooks and anchors** tab:
   - Unselect the **Anchor** options.
   - Select both **Hook 1** and **Hook 2** to add hooks at both bar extremities.
   - For each hook, enter an angle of **90°**, then select **Length** and enter **0.60 m** to set the length.

   ![Image: Properties dialog box – Hooks and anchors tab](image)

4. Click **OK** to close the dialog box.
5. Place the straight bar by clicking two points as in the drawing below.

   ![Image: Elevation face 1 - first and second point for creating the bar](image)

6. Click a point above the elevation to define the bar orientation.
7. Press **Esc** to finish.

**Adjusting the hook length of the bar**

The straight bar is placed. You can make small adjustments to the properties if necessary. For example, adjust the hook length.

1. Select the bar.
2. On the Smartbar, click **** to access the properties dialog box.
3. On the **Hooks and anchors** tab, in the **Length** field, set the hook length to **0.5 m**.

![Figure 331: "Properties" dialog box – Hooks and anchors tab](image)

4. Click **<OK>**.

![Figure 332: Face elevation 1 - Adjusted bar](image)

**Creating straight bars with hooks and anchors**

Next, use the same tool to create two straight bars, with a hook at one extremity and an anchor at the other one.

1. On the **Reinforcement** toolbar, **Reinforcement Bars** flyout, click ![Reinforcement Bars](image).
2. On the Smartbar, click ![Properties](image) to access the properties dialog box.
3. In the properties dialog box, make the following settings on the **Hook and anchor** tab:
   - Select **Hook 1** to create a hook at the first extremity of the straight bar. Enter **0.6** for the hook length.
   - Select **Anchor 2** to create an anchor at the other extremity of the straight bar. Enter **1.8 m** for the anchor length.

![Figure 333: "Properties" dialog box – Hooks and anchors tab](image)
4. Place the straight bar by clicking two points as in the drawing below.

![Figure 334: Elevation face 1 – Bar definition points](image)

5. Click a point under the elevation to define the bar orientation.
6. Press Esc to finish.

The bar is created.

![Figure 335: Elevation face 1 – Bar with hook and anchor](image)

Create another bar with the same properties on the left side of the elevation.

![Figure 336: Elevation face 1 - Second bar definition points](image)

**Creating a straight bar with anchors**

Next, finish reinforcing the top side of the beam by creating a straight bar with anchors.

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, make the following settings:

   On the **Definition** tab, from the **Diameter** drop-down list, select **12Ø** for the bar diameter.

   ![Figure 337: “Properties” dialog box – Definition tab](image)

   On the **Hooks and anchors** tab:
   - Select only the **Anchor 1** and **Anchor 2** options (unselect the hooks if checked) to create anchors at both extremities of the bar.
   - For each anchor, select the **Length** option and enter **0.50 m** for the anchor length.

   ![Figure 338: Bar properties – Adding anchors](image)

4. Click `<OK>` to close the dialog box.

5. Place the straight bar using as definition points the extremities of the bars in the top side of the beam.

   ![Figure 339: Elevation face 1 – Placing the straight bar](image)

6. Click a point above the elevation to define the bar orientation.

7. Press **Esc** to finish.
The straight bar is created.

**Step 5: Create a rectangular frame**

In this step, create a rectangular frame on Section A-A and a bar symbol.

**Note:** In Advance Concrete, the reinforcement symbol creation can start automatically. This is possible by activating the option in the “User preferences” dialog box.

**Before starting:**

1. In the Pilot, right click **Project** and select **User preferences** from the context menu.
2. In the “User preferences” dialog box, **Visual elements** tab, select the option for creating reinforcement symbols.

   ![User preferences – Visual elements tab](image)

1. On the **Reinforcement** toolbar, **Reinforcement Bars** flyout, click ☑.
2. On the Smartbar, set the frame diameter to 10Ø.
3. Place the rectangular frame by clicking two diagonal points as shown in the following figure.

   ![Section A-A - Second point of the bar](image)

4. Click another point to set the hook position in the top-right corner.

   ![Section A-A - Setting the hook position](image)
5. On the command line, make the following settings to modify the symbol scale:
   • Enter S (scale) and press Enter.
   • Enter the scale, 2, and press Enter.

6. Place the symbol at the bottom of the rectangular frame.

7. Press Enter to finish.

Using the same process, create a rectangular frame for the Section C-C.

**Step 6: Create a linear distribution**
Distribute the rectangular frames on the Section A-A and Section C-C on the Face elevation 1 view.

1. On the Reinforcement toolbar, Distributions flyout, click ![Distributions](image)

2. Select a bar to distribute. For this example, use the rectangular frame on Section A-A.

3. Click the horizontal arrow to select the view direction.
4. Place the distribution by clicking two points as shown in the following figure.

![Figure 346: Elevation face 1 – Distribution definition points](image)

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. In this exercise, set 18x0.1 9x0.2 19x0.1, and press Enter.

6. Click a point above the elevation to position the distribution.

7. A gray bar (the original definition bar copy) is displayed as "hooked" to the distribution. The fake bar is oriented by moving the mouse on one side and the other of the distribution line. Select the bar direction by clicking a point on the right side as the hook should be on the right side.

![Figure 347: Selecting the bar position](image)

7. Press Esc to finish.

Note: Advance Concrete automatically manages the bar representation in all views of the reinforcement drawing, and Section B-B displays a rectangular frame with the same properties as the definition bar of the created linear distribution.

![Figure 348: Rectangular frames](image)
Using the same process, create a linear distribution of the rectangular frame on Section C-C. Set the distribution definition type to **By spacing (truncated)** and the bar spacing to **0.15 m**.

**Step 7: Creating a free distribution**

In this step, distribute the straight bar in the bottom side of the beam. We only need to create the distribution on Section A-A. Due to the 3D power, the bars are represented automatically in the other two sections of the reinforcement drawing.

1. On the **Reinforcement** toolbar, **Distributions** flyout, click ➧.
2. On the **Face elevation 1** view, select the straight bar in the bottom side of the beam (the first created straight bar).
3. Place the first point of the distribution on the left bottom corner of Section A-A.
4. Create bars along the bottom segment of the rectangular frame on Section A-A as shown in the following figure.

5. Press Enter to finish.

The bars also appear on Section B-B and Section C-C.

Using the same tool, distribute the three straight bars on the top side of the beam on Section A-A, Section B-B, and Section C-C, respectively.
Note: Note that the distributions on Section B-B and Section C-C are not taken into account in lists and symbols.

Step 8: Create the reinforcement symbols for the free distributions
In this step, you will create the symbols for the free distributions on the three sections on the reinforcement drawing.

1. On the Reinforcement toolbar, Symbols and annotations flyout, click.
2. Select an element of the bottom distribution on Section A-A.
3. On the command line, make the following settings to modify the symbol scale:
   - Enter S (scale) and press Enter.
   - Enter the scale, 2, and press Enter.
4. On the Smartbar, from the Name drop-down list, select the symbol.
5. Click a point to place the symbol on the right side of the Section A-A view.
   The symbol is created.

Figure 356: Distribution properties – The distributions are not taken into account

Figure 357: “Name” drop-down list

Figure 358: Section A-A – Symbol
Using the same process create all necessary bar symbols on Section A-A, Section B-B and Section C-C.

**Note**: Due to the 3D power, the number of bars is displayed correctly in the bar distribution symbol. The distributions are not taken into account.

**Step 9: Create the bar bending details**

**Note**: In Advance Concrete, the creation of the bar bending details and the bar bending annotation can start automatically after the bar creation. This is possible by activating the option in the “User preferences” dialog box.

**Before starting**:

1. In the Pilot, right click Project and select User preferences from the context menu.
2. In the “User preferences” dialog box, Visual elements tab, select the option for creating reinforcement symbols.

Before creating the bar bending detail, set the scale of the drawing to **1:50**.
1. On the **Reinforcement** toolbar, **Distribution** flyout, click ![Distribution](image).

2. Select the first created straight 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.

5. On the command line, make the following settings to modify the symbol scale:
   - Enter `S` (scale) and press **Enter**.
   - Enter the scale, 2, and press **Enter**.

![Figure 362: Face elevation 1- Bar bending detail](image)

Using the same process, add bending details and symbols for the other 3 bars as in the following figure.

![Figure 363: Bending details](image)
Dimensioning the anchors 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 [image].

2. Click a point to define the position of the dimension line.
3. Click the two points to dimension as shown in Figure 484.

Repeat the command and dimension the other extremity.

Figure 364: Dimensions by points for the overlapping distance

Step 10: Create bar bending details for the rectangular frames

Next, create a bending detail for the rectangular frame on Section A-A.

1. On the Reinforcement toolbar, click [image].
2. Select the rectangular bar on Section A-A.

The bar bending detail appears "hooked" to the cursor.

Figure 365: Creating the bending detail
3. Place the bar bending detail next to Section A-A. The annotation creation starts automatically.

4. On the Smartbar, from the Name drop-down list, select the desired symbol for the bar bending detail.

5. Place the symbol on the top of the bending detail. The bending detail is created.

Figure 366: Section A-A - Bar bending detail for the rectangular frame

Modifying the hook representation of the bending detail

1. Select the bar 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.

4. Click <OK>.

Figure 367: “Properties” dialog box – Bending details tab

Figure 368: Unfolded bar bending detail
Modifying the hook length of the bending detail

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

2. Click the hook point and stretch it to modify its representation.

Using the same process, create a bar bending detail for the rectangular frame on Section C-C.
Step 11: Dimension the distributions

In this step, add a distribution dimension on the Face elevation 1 view.

Before starting, change the scale of the drawing to 1:100. In the Pilot, click and select the 1:100 scale from the list.

Creating the distribution dimension

1. On the Reinforcement toolbar, click ![](image)
2. Select the distribution to dimension. For this example, use the first linear distribution (from the left).
3. Place the dimension line at the top of the distribution.
Modifying the symbol

1. On the Smartbar, click \(\text{\textbullet}\) to access the properties dialog box.
2. On the properties dialog box, select the Reference mark and bending detail tab to modify the symbol.
   From the Name drop-down list, select the desired symbol.

3. Click \(<\text{OK}>\).
4. Place the symbol at the top of the Elevation face 1 view.

Figure 375: “Distribution dimension” properties dialog box – Selecting the symbol

Figure 376: Elevation face 1 - Distribution symbol
Step 12: Renumber the bars

1. On the Reinforcement toolbar, Renumbering flyout, click.
2. In the “Numbering scope” area, select the numbering applies to the current drawing option.

![Image](Renumbering or rename positions dialog box)

3. Click <OK> in the renumbering dialog box and in the message window.

![Image](Confirmation window)

Step 13: Create a list on the layout

1. On the Reinforcement toolbar, Lists flyout, click.
2. On the command line, enter B (Bars) to consider all the bars and press Enter.
3. 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.
4. Place the list next to the Face elevation 1 view.

![Image](The list)
**Sorting the list**

1. Select 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.

4. Click <OK>.

**Step 14: View the reinforcement in 3D**

1. On the **Reinforcement 3D** viewer toolbar, click.

2. Include the reinforcement views in a window selection.

3. Press **Enter**.
The 3D viewer appears. The drawing can be rotated and viewed from different angles.

![Figure 382: "Viewer 3D" dialog box]

**Saving the drawing**

1. In the “Viewer 3D” dialog box, click ![button]
2. In the “View name” dialog box, set the name of the drawing to Reinforcement Frame - Beam B2.

![Figure 383: Setting a name for the drawing]

3. Click <OK>.

The new drawing is displayed automatically in the Pilot.

![Figure 384: Pilot - New drawing]
Using the “drag and drop” procedure, group the **Reinforcement Frame - Beam B2 3D** drawing in the B2 reinforcement.
Lesson 14: 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 386: Layout of the reinforcement drawing
**Setting the page layout for printing**

This step describes how to set up the page layout 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 387: Pilot - Creating the layout](image)

2. In the “Properties” dialog box, click **New** to create a new presentation style.

   ![Figure 388: “Properties” dialog box – Definition tab](image)

3. Click **Modify the format and the printer** button to set the page format and the printer.
4. In the “Page Setup Manager” dialog box, click **New** to create a new page setup.

5. Set the New page setup name to **Setup for AO**.

![Figure 389: Creating a new page setup](image)

6. Click **<OK>**.

**Preparing the page for printing**

1. 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)**.
   - In the “Drawing Orientation” area, select **Landscape** to set the drawing position.

![Figure 390: Selection of the drawing orientation](image)

2. Click **<OK>**.
3. In the “Page Setup Manager” dialog box, in the “Page Setup” area, select the previously defined page layout: Setup for AO.

![Page Setup Manager dialog box](image)

Figure 391: “Page Setup Manager” dialog box

4. Click Set Current to apply the layout.

5. Click Close.

**Adding a frame**

Next, add a frame to the layout.

1. In the “Properties – Presentation 1” dialog box, in the Contents area, select the Frames and bending lines option.

![Properties dialog box](image)

Figure 392: Adding a frame to the layout

2. Click the button next to the Frames and bending lines option to select the frames presentation.

3. In the “Frames of Presentation 1” dialog box, click Add to add a frame.

![Frames of Presentation dialog box](image)

Figure 393: “Frames of Presentation 1” dialog box

4. Browse and select a frame from the Advance folder tree.

5. Click <OK>.
Adding a titleblock

1. In the presentation properties dialog box, in the Contents area, select the Title blocks option.

![Image](https://example.com/image1.png)

Figure 394: Presentation properties – Adding title blocks

2. Click the ... button next to the Title blocks option to select the desired title block.

3. In the title block selection dialog box, click Add.

![Image](https://example.com/image2.png)

Figure 395: “Title block of Presentation 1” dialog box

4. Browse and select a title block from the Advance folder tree.

5. Click <OK>.

6. Click <OK> to return to the “Properties” dialog box.

7. In the “Properties” dialog box, on the Definition tab, select the Presentation1 presentation style and click Define current to apply the settings.

![Image](https://example.com/image3.png)

Figure 396: “Properties” dialog box.
Modifying the layout scale using the Smartbar

1. Select the layout.

2. On the Smartbar, set the drawing scale to \( \frac{1}{20} \) and press Enter.

![Diagram showing the Smartbar with scale 1/20 set]
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