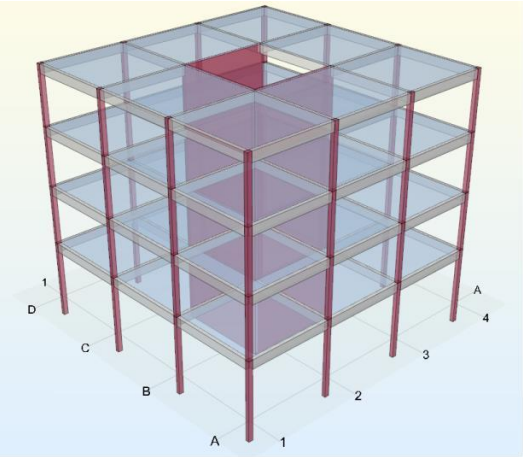
Getting started

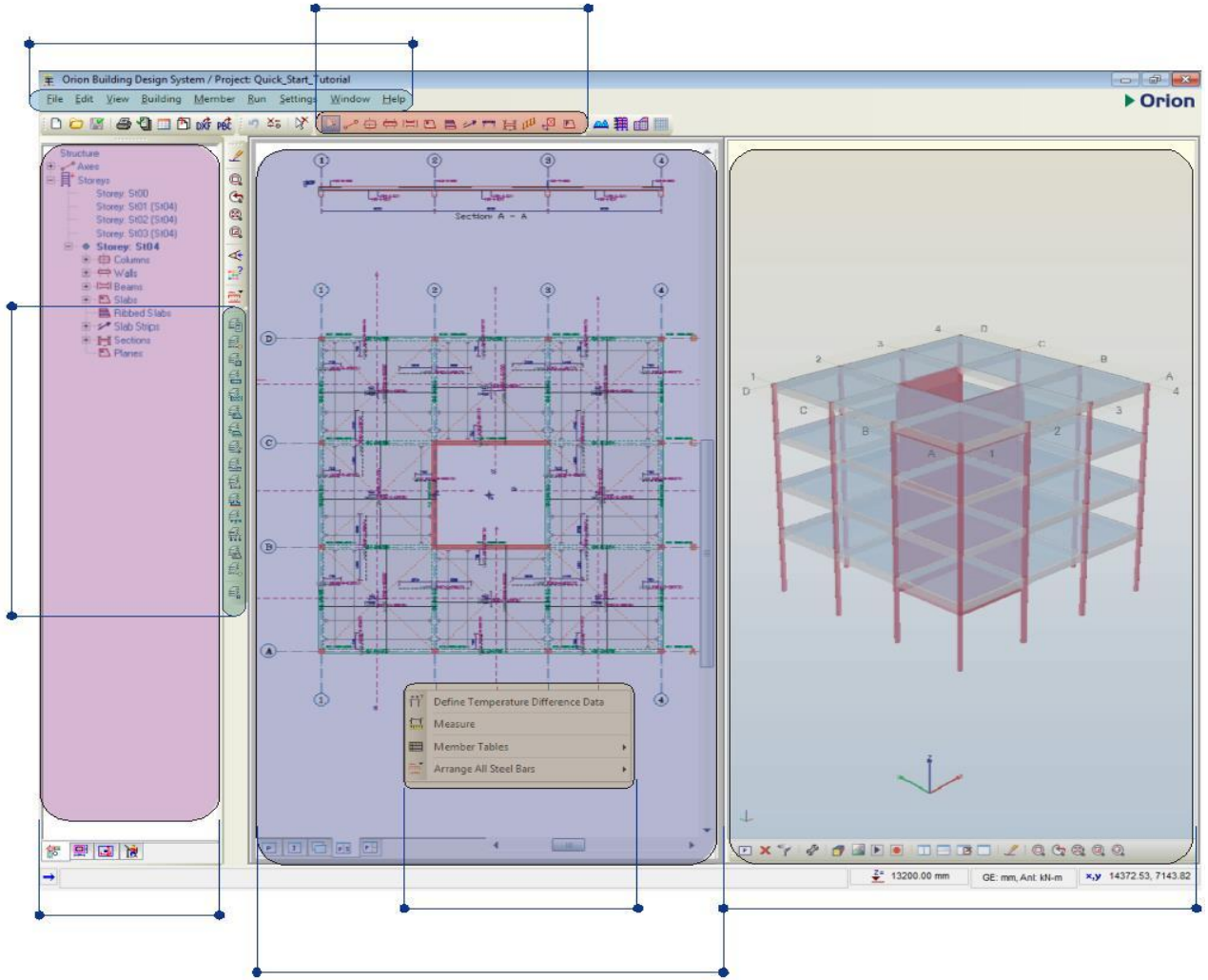
Over the following pages we will explain how to build the basic model shown below, analyse it and design its members.



The model is very simplified, but consistent with the aim of this guide — to give you a quick overview of Orion from start to finish.

User interface

The various components of Orion’s user interface are shown below:



Apart from the Pick icon , which you use to select any type of member, the Members Toolbar is arranged in a logical sequence.

Generally you create buildings by working from left to right along this toolbar.

You can also access the options on the toolbar from the Main menu although this is slower than using the icons.

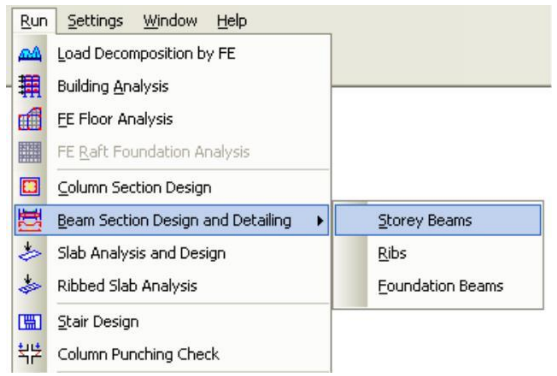
Quick Start conventions

The aim of these conventions is to enable you to work through this Quick Start example as quickly as possible.

Menus — For the purpose of this exercise when we refer to Main menu commands we do so using following syntax:

Pick Run / Beam Section Design and Detailing » Storey Beams

This means that you should pick item shown below from the Main menu.



Buttons — We shorten the command “Click the Button Name button” to “Click Button Name”.

Steps you must perform — In order to differentiate from any explanatory text, actions that you need to perform are numbered and shown in italics as shown below:

1. Leave the Codes tab unchanged and click Analysis.

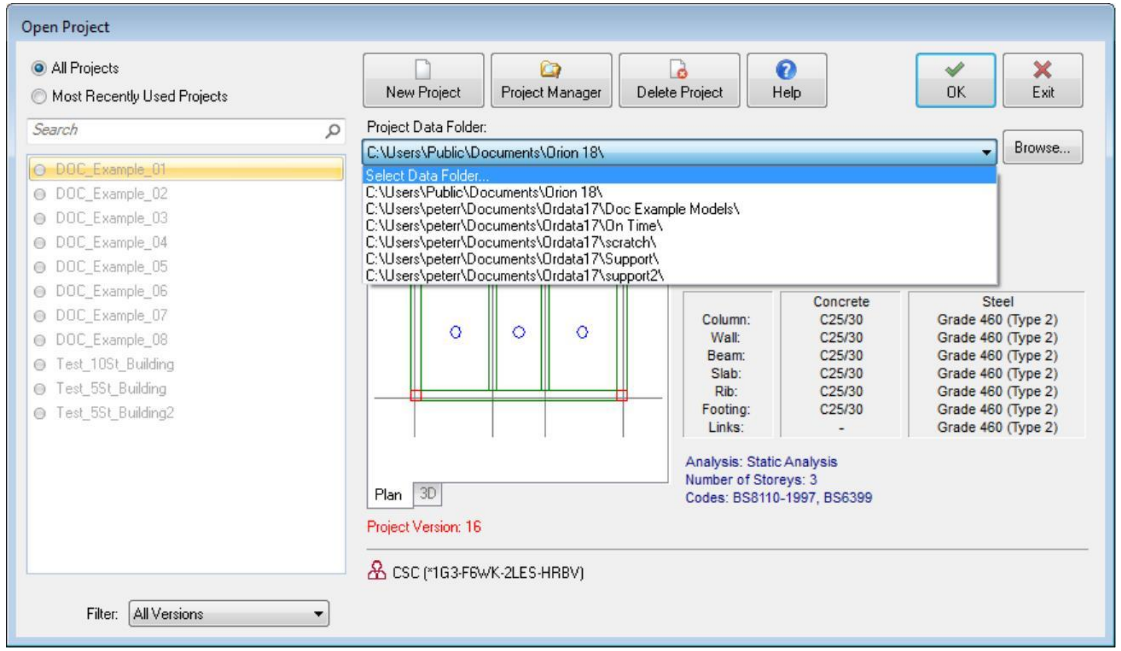
First Steps

If you have not yet installed Orion, insert the CD into your computer’s CD/DVD drive, and follow the on-screen instructions to install it. Ensure that you connect the hardware dongle to the appropriate port on your computer.

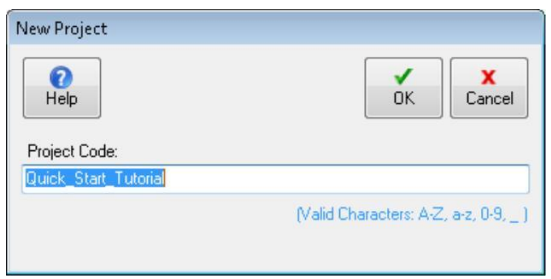
1. Click the Start button on the Windows task bar, then in turn click:

    - All Programs,
    - CSC/Orion,
    - Orion 18.

This loads Orion, and displays the Open Project dialog. You use this both to select an existing project or to start a new one. If required you can specify a different location by clicking the Data Folder dropdown and choosing Select Data Folder...



2. Click New Project and type the Project Code as shown using the '_' character to denote spaces.



3. Select OK.

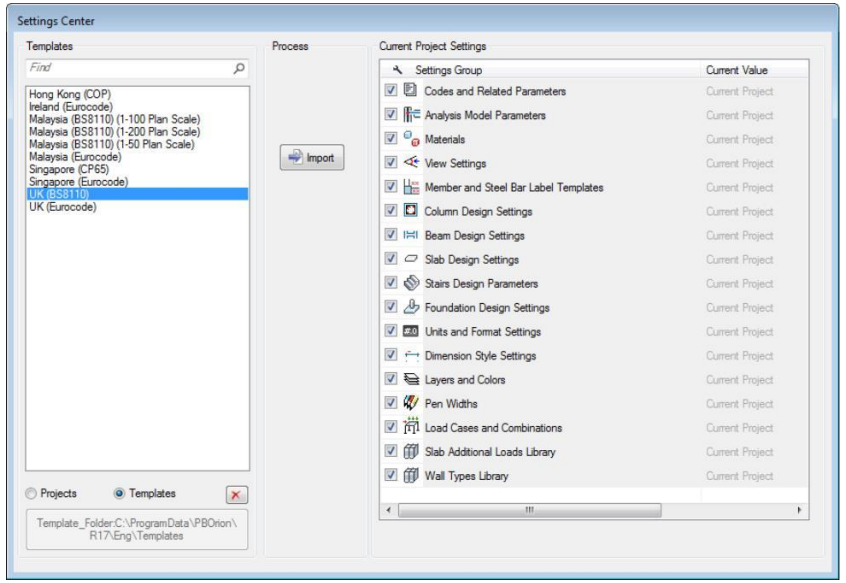
This creates a folder automatically called Quick_Start_Tutorial within the default Data Folder. This is where Orion stores all the data for this model.

Project Settings

We now need to choose the settings to be applied to the project.

Template Selection

The Settings Center dialog appears automatically.



1. Select the UK (BS8110) template and click Import.

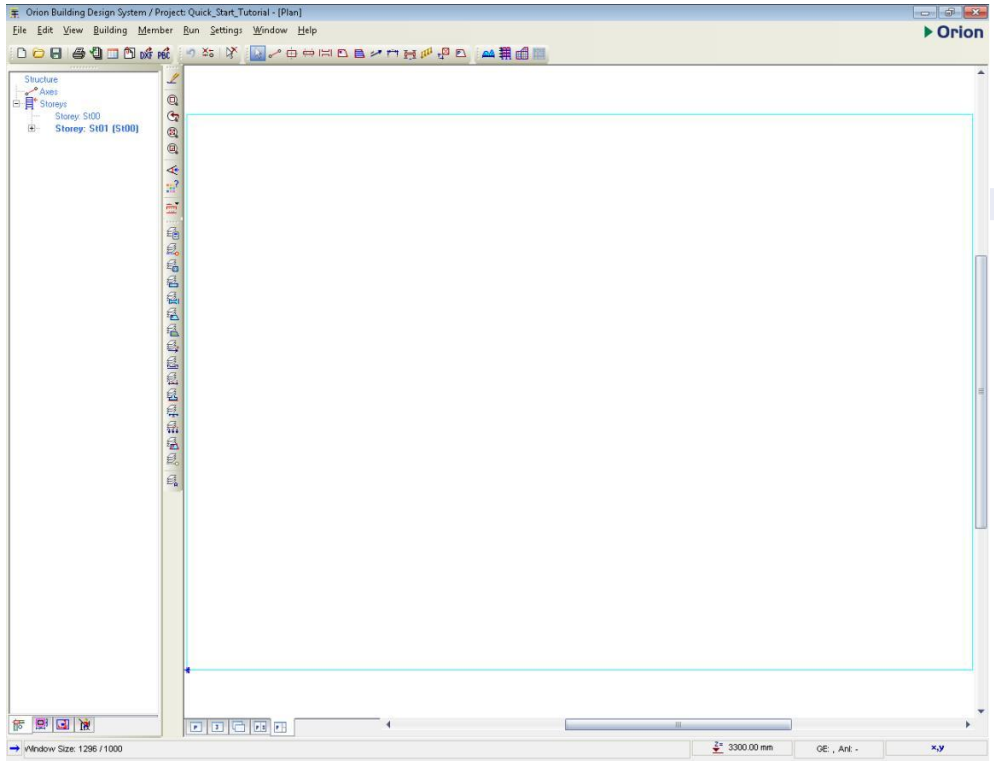
Templates are used to rapidly establish default model parameters: design codes; material properties; member design settings etc. for the project. Alternatively you can choose to duplicate settings from an existing Project.

Drawing Sheet Selection

The Drawing Sheet selection dialog appears automatically.

1. Select the standard sheet size A0 (1188 x 840 mm), then set the drawing and detail scales to 1/50 and 1/20 respectively, then click OK.

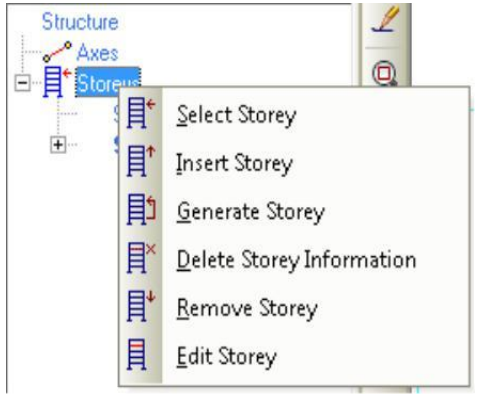
After you select the drawing sheet you will see the main drawing area (Graphic Editor). Initially the drawing area is completely empty.



Specifying the Storey Height

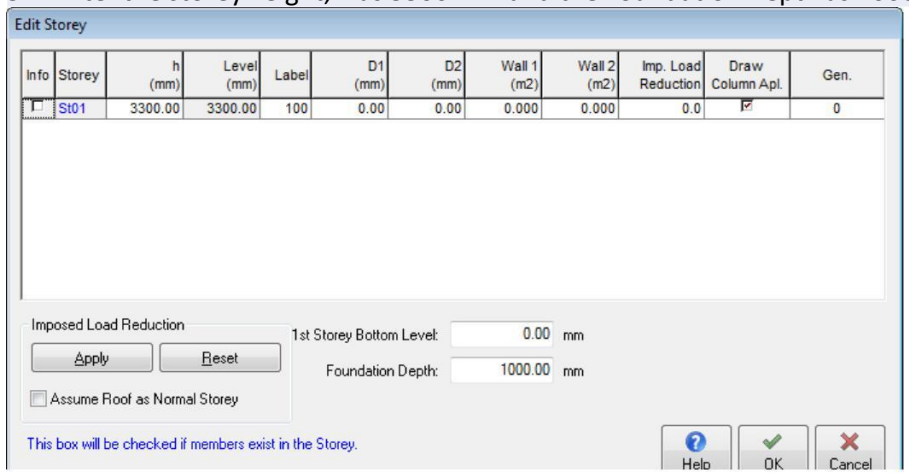
The storey height is specified as follows:

1. Right click on Storeys in the Structure Tree View to see the context menu.



2. Pick Edit Storey.

3. Enter the storey height, h as 3300 mm and the Foundation Depth as 1000 mm.

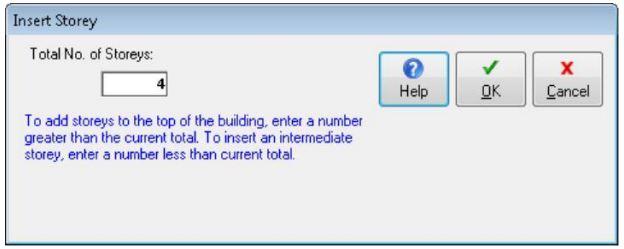


4. Click OK.

Inserting Additional Floors

Now we will proceed to define the building's total number of floors.

1. Right click on Storeys in the Structure Tree View once more and pick Insert storey from the context menu.



2. Enter 4 in the Total No. of Storeys box and click OK.

By entering 4 you are telling Orion that you want to add storey 4. Orion knows that your model only contains a base level at present, and it will therefore create not only storey 4, but will also create the intermediate storeys 1, 2 and 3 automatically.

If you were to use the dialog on a building which already has storeys in it, and you entered the number of an existing storey, then Orion would create a new storey at that storey number and would move the existing storey(s) up to cater for this.

Selection methods

Selection is performed using the first button in the Members toolbar. At the moment the model has no members for you to select. You will however need this general information as you continue to work through the example, so please take time to read it now.

You select all entities in the same way — using the Pick icon. Selected entities are also highlighted in the Structure Tree View.

If you click the right mouse button when an entity is selected you will see a pop-up (context) menu which allows you to edit that entity.

You can select several entities at the same time, simply hold down the CTRL key while you pick them. You can identify selected entities by the small squares or grips that appear at their ends.

You can also select entities directly from the Structure Tree View. Click the entity name in the tree view to select it. You can also hold down the CTRL key to selection multiple entities from the structure tree.

You can also drag with the mouse to access further selection options:

- drag from left to right and you will see a rectangular box. When you release the mouse button Orion selects all entities that are totally contained within the box.

- drag from right to left similarly and Orion selects any entities that are contained within the box and which cross its boundaries.

- hold the Shift key down and drag to create a line rather than a rectangle. In this case Orion selects any entities.
- hold Control and Shift keys down and drag to filter by element type the selected entities.

As you progress through this example please experiment with selecting using both the Pick icon and the Structure Tree View methods.

Zooming and Panning methods

As with selection there is nothing other than the sheet border with which to zoom and pan. You will need this general information later in the example. As with selection we would encourage you to experiment with zooming and panning as you work through the example.

The most useful zoom commands are:

Zoom Extents — zooms the display so that you can see the selected object(s).

Zoom Limits — zooms the display so that you can see all the objects that are within the drawing sheet border.

Zoom Window — Allows you to zoom into an area by dragging a rectangle to define its diametrically opposite corners.

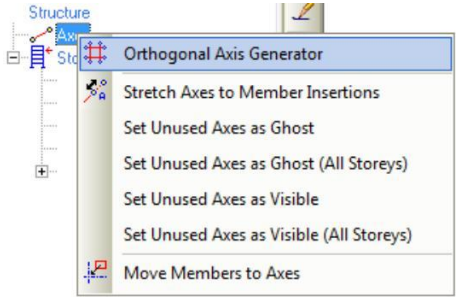
Pan — Allows you to pan the drawing image by holding down the scroll wheel on the mouse and dragging to a new location in the window.

Defining a grid pattern

Now to create the axes in our model. You can either define axes individually by clicking the Axis icon, or you may prefer to import them from a DXF drawing. Refer to the Orion Help for more details of how to import them from DXF.

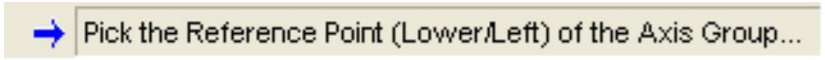
You can also define multiple axes in one go using the Orthogonal Axis Generator. This is the approach we shall adopt here.

1. Right click on Axes in the Structure Tree View to see the context menu (shown below).



2. Pick Orthogonal Axis Generator.

Look at the bottom of the window. The text displayed there tells you how to proceed.



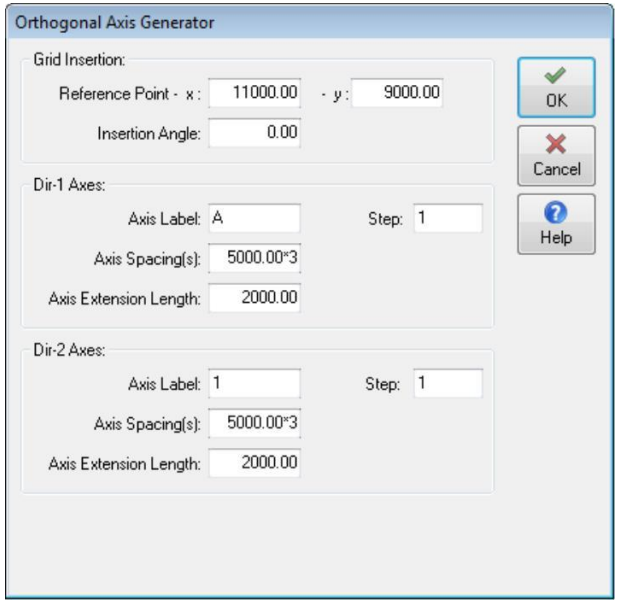
3. Hold down the CTRL key and pick a point in the lower left hand region of the drawing sheet.

Holding the CTRL key like this ensures the reference point is a sensible (i.e. whole number) offset from the

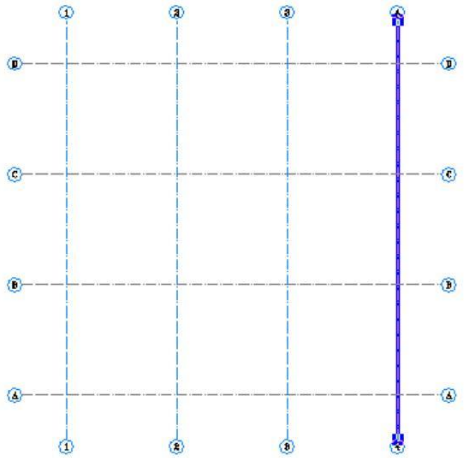
4. After you pick the reference point you will see the Orthogonal Axis Generator dialog.

The Orthogonal Axis Generator creates Direction 1 axes horizontally and gives them Alphabetic labels, It creates Direction 2 axes vertically with Numeric labels. By convention Orion will assign direction 1 to all axes within ±45 ° to the horizontal and direction 2 to all axes within ±45 ° to the vertical.

5. Complete the Orthogonal Axis Generator as shown, with both Dir-1 and Dir-2 Axis Spacings set to 5000*3.



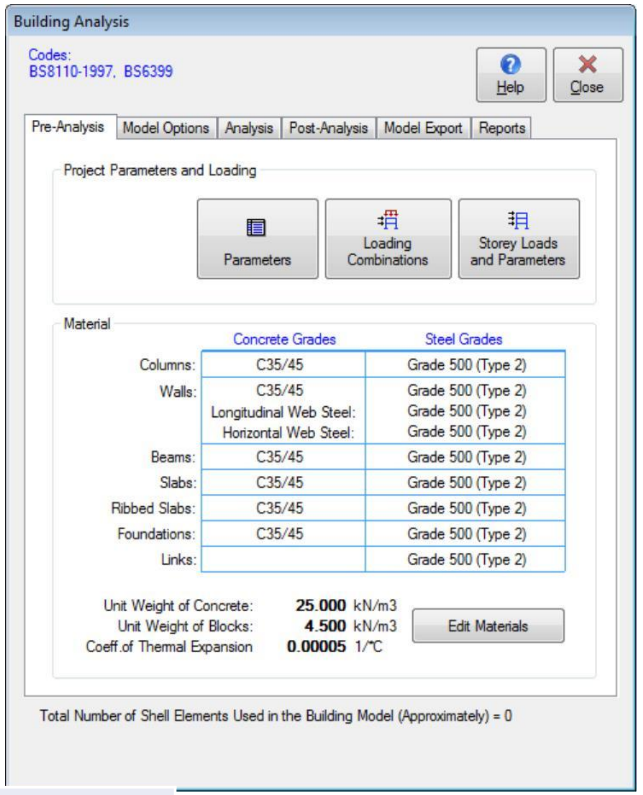
6. Click OK and you will see the axes shown below.



Defining Materials

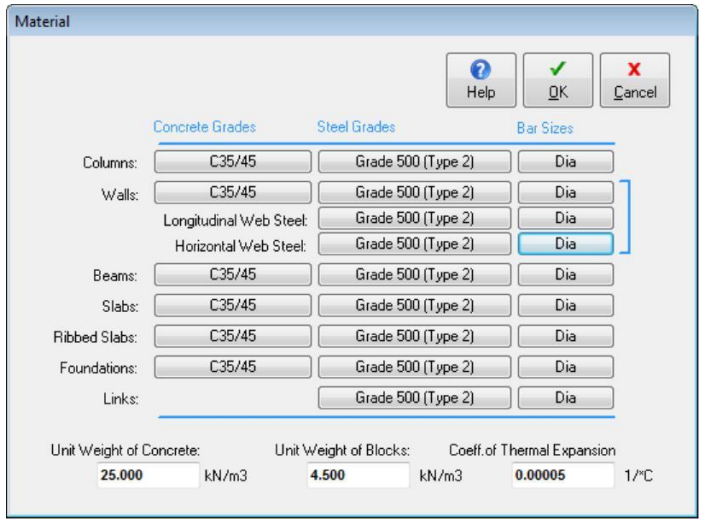
We now need to specify the materials to be used in the model.

1. Pick Run / Building Analysis to see the Building Analysis dialog.

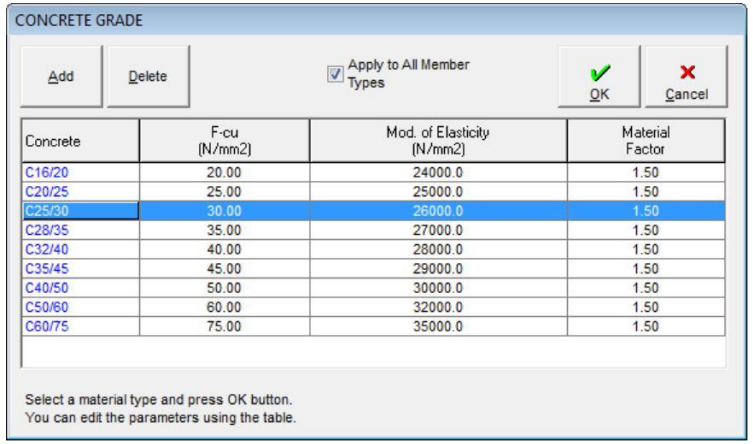


The default material grades are listed for each member type at the bottom of this form.

2. Click Edit Materials.



3. Click the Columns/Concrete Grades box, highlight C25/30 and check the box Apply to All Member Types.



You could apply an alternative grade or adjust the modulus of elasticity of an existing grade from here.

Note

4. Click OK to return to the Materials tab.

The unit weight of concrete is set on this screen also. In this example 25 kN/m is used.

5. Repeat the same process if necessary for the Steel Grade. All structural members should be set to use bars of Grade 500 (Type 2) and a Material Factor of 1.15.

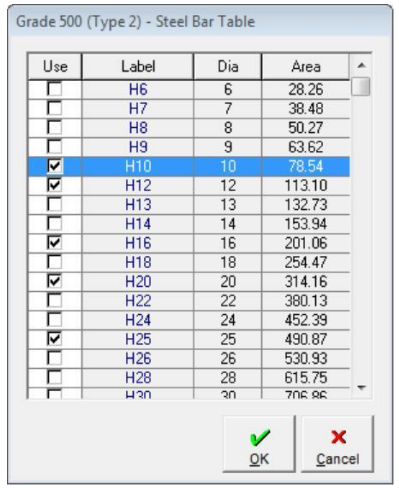
The above Grade and Material Factor are appropriate for design to BS 8110:1997 in the UK. For other codes/regions you may be required to adjust these.

Next we will set the bar diameters which we want Orion to consider when it performs the reinforcement design for each member type.

6. Click Dia (the one on the Slabs row).

You will notice that some bars sizes are selected by default. You can prevent Orion using a bar size by removing the tick in its Use box. Conversely you can allow Orion to use a bar size by ticking its Use box.

In this example we will choose not to use H8 bars in the slabs or beams.

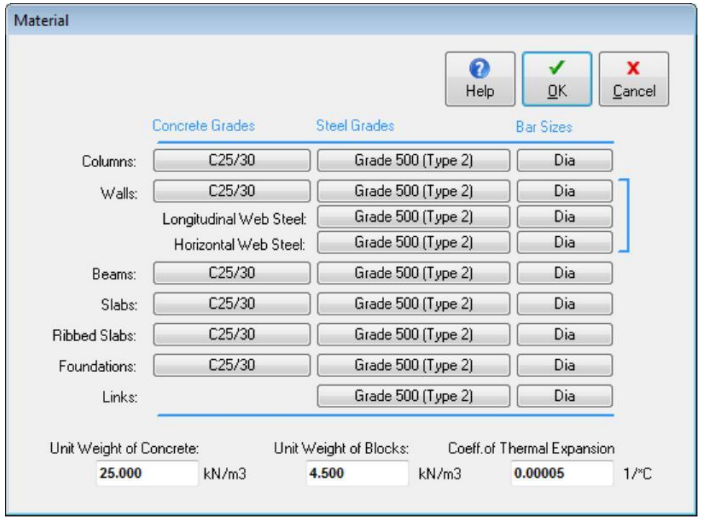


7. Make the settings shown above and click OK.

8. Repeat the above process for the beams and links also.

You may prefer to use different bar sizes, in which case you will achieve different results later in the example.

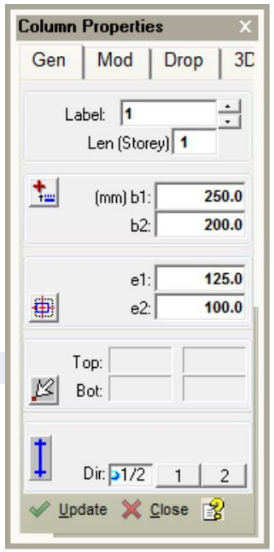
9. Select OK to return to the Materials dialog which looks as shown below.

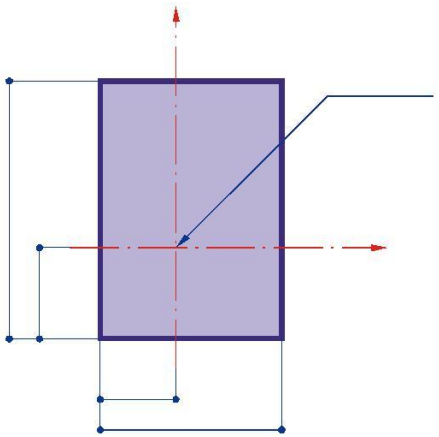


10. Select OK once more to return to the Building Analysis dialog and then Close.

Creating Columns

1. Click the Column icon or pick Member / Column.
2. Enter b1 and b2 as 250 and 200, then e1 and e2 as 125 and 100 as shown below. The dimensions are explained by the diagram to the right of the dialog.





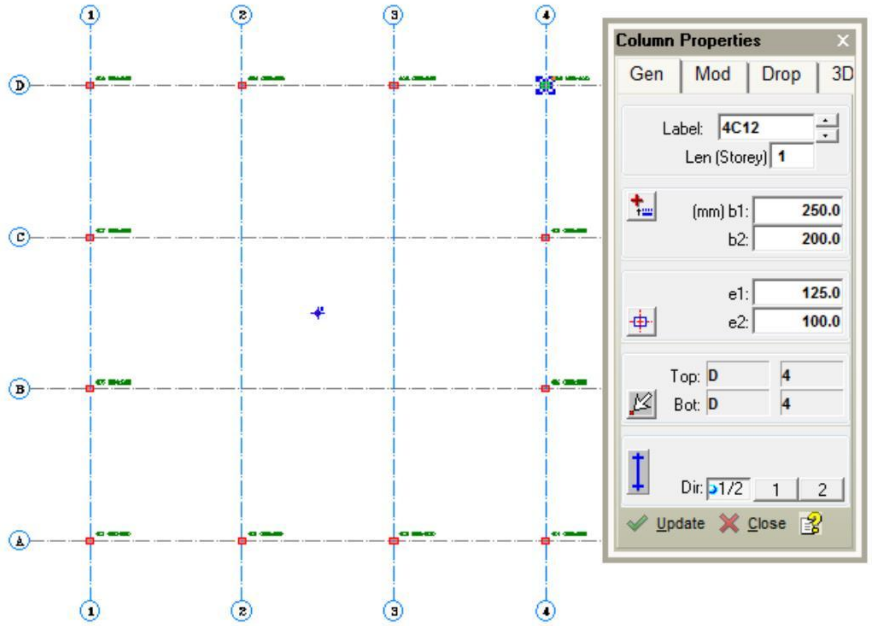
You can right click in each box and select from the available dimensions instead of typing.

These properties will create a 250 x 200 sized column with the 250 dimension in direction 1. These columns are also parallel to the grids in both directions 1 and 2.

3. To position columns click and drag from axis A-1 to axis A-4.

   You might like to try out some of the zooming and panning methods on page 12 to view the area of interest at this point.

4. Next position single columns by clicking at the following grid intersections: B-1, B-4, C-1, C-4.
5. To position the final row of columns, click and drag from Axis D-1 to axis D-4.

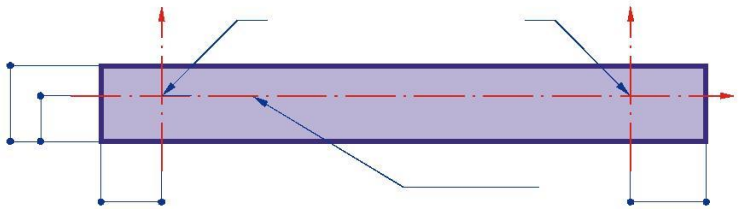


Creating Walls

1. Click the Wall icon or pick Member / Wall.

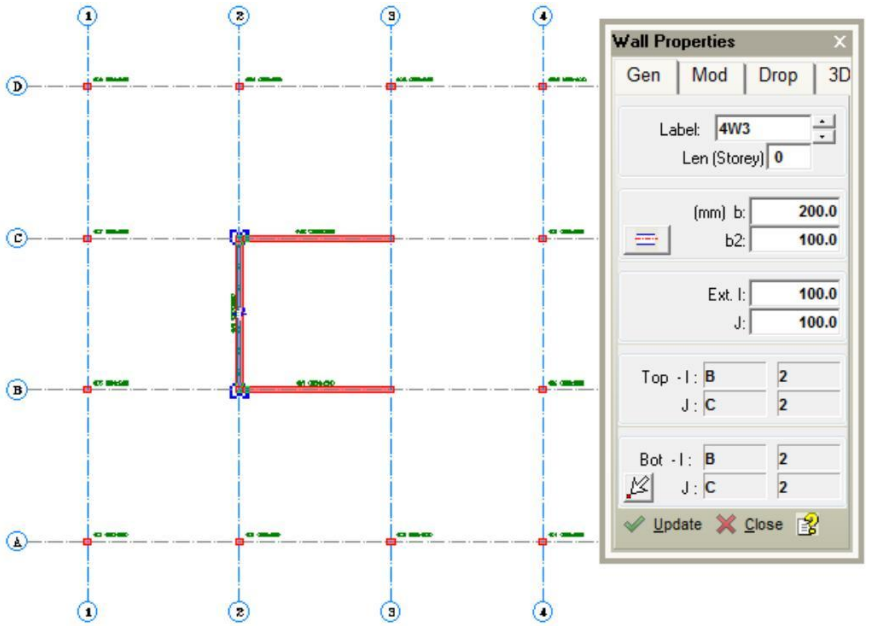
2. Enter a b: dimension of 200, and a b2: dimension of 100. Also enter a value of 100 for both Ext. I: and J:.

   The dimensions are explained by the diagram below.



3. Insert the wall by clicking and dragging from the start grid B-2 to C-2.

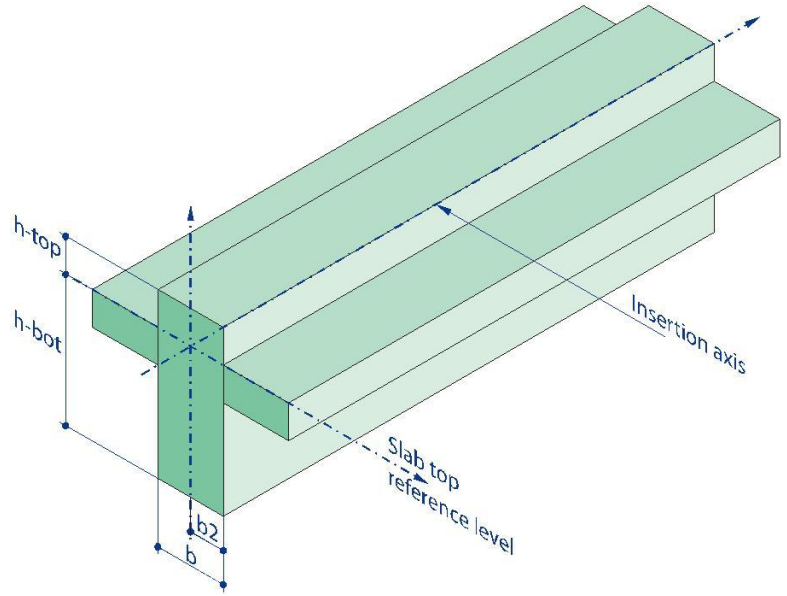
4. Do the same at Grid C-2 to C-3 and Grid B-2 to B-3. You should now have the following arrangement of walls.



Creating Beams

1. Click the Beams icon or, pick Member / Beam.

2. In the Beam Properties dialog ensure that b: is 200, h-bot: is 500 and h-top: is 0.0.



The dimension h-bot is the depth of the beam from the top of the slab, and h-top is the height of the beam above it. A negative value of h-top would drop the beam below the slab.

We want the beams are to be placed in the centrally along the axes so we want b2 to be half of b.

3. To ensure this click the alignment icon ( ) followed by the centre icon ( ). This sets b2 to 100.

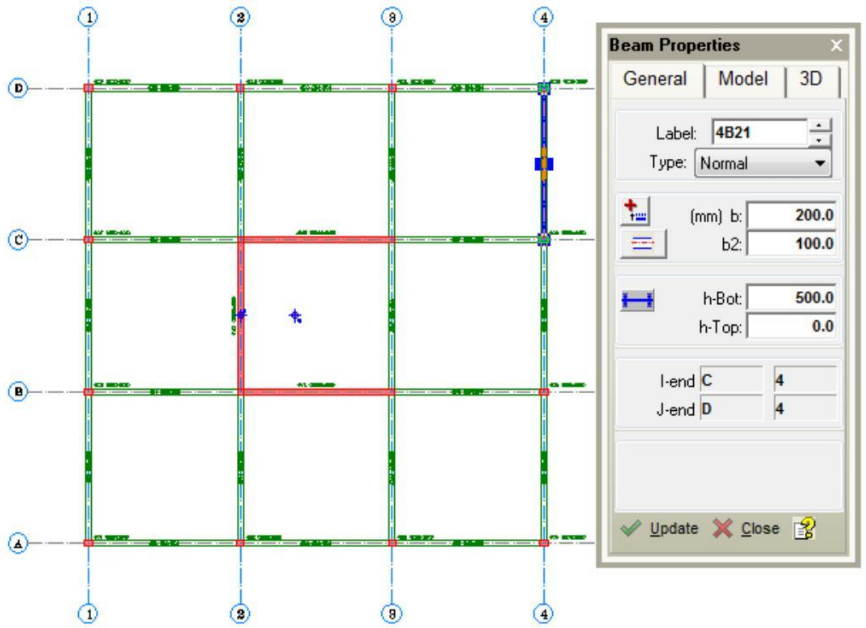
4. To position the beams click and drag along the following axes A-1 to A-4, B-1 to B-4, C-1 to C-4, D-1 to D-4, A-1 to D-1, A-2 to D-2, A-3 to D-3 and A-4 to D-4.

Like the columns the beams are automatically labelled based on the storey and numbered sequentially as they are entered.

Orion automatically splits the beam into three individual members between the columns.

A beam will not be placed where a wall already exists. For example, dragging along grid line B created two extra beams, not 3.

Your screen should now look like this.



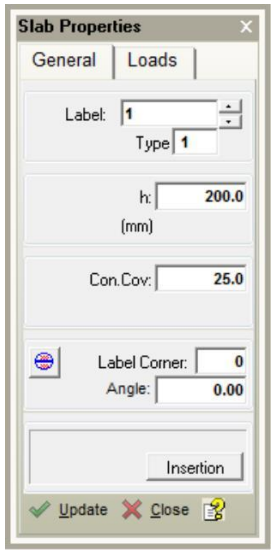
Creating Slabs

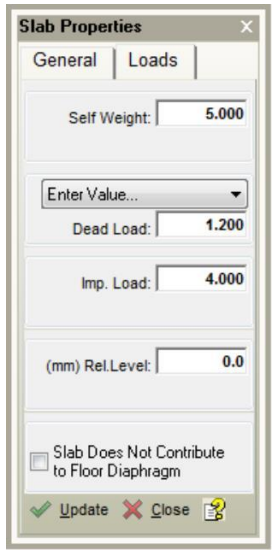
Now we shall create the 4 storey slabs.

1. Click the Slab icon or pick Member / Slab.

2. In the Slab Properties dialog enter the slab thickness h: to be 200 and the Concrete cover to be 25.
3. Click on the Loads tab and enter an extra Dead Load of 1.2 kN/m^2.

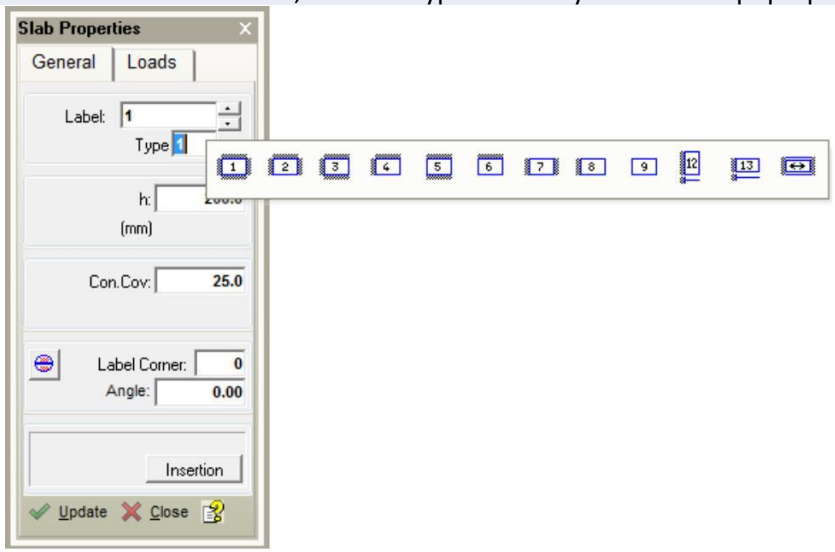
4. In the Imp. Load box do a right mouse click and select a value of 4.0 kN/m^2.





Orion calculates the self-weight of the slab automatically from its thickness and the concrete density (specified earlier when defining materials).

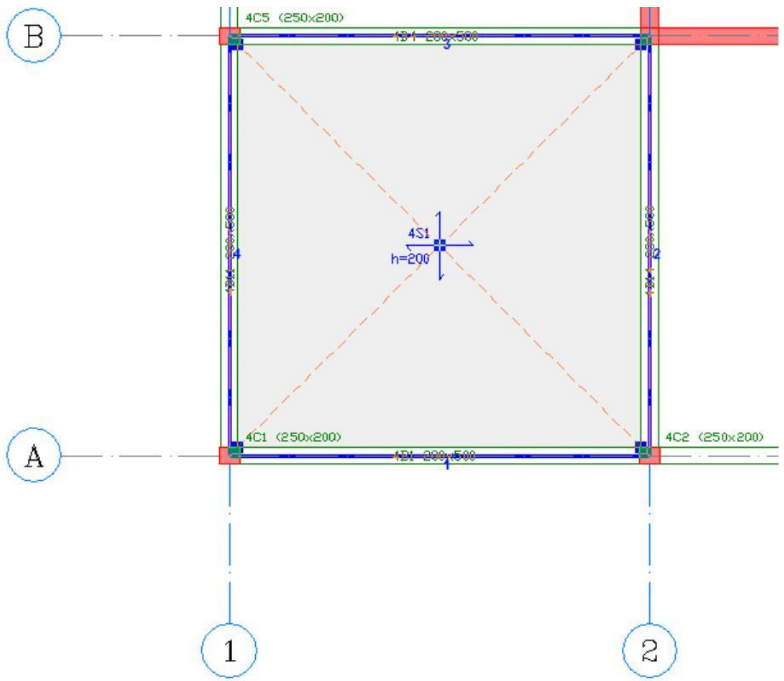
5. Return to the General tab, click the Type box and you will see a pop-up menu of all possible Slab Types.



The Slab Type relates to table 3.14 in the code and is used to obtain correct reinforcement values, based on the coefficient method. For ease in creating this model we will leave the Slab Type as 1 initially. Once all the slabs exist we can tell Orion to calculate the correct type for each slab automatically.

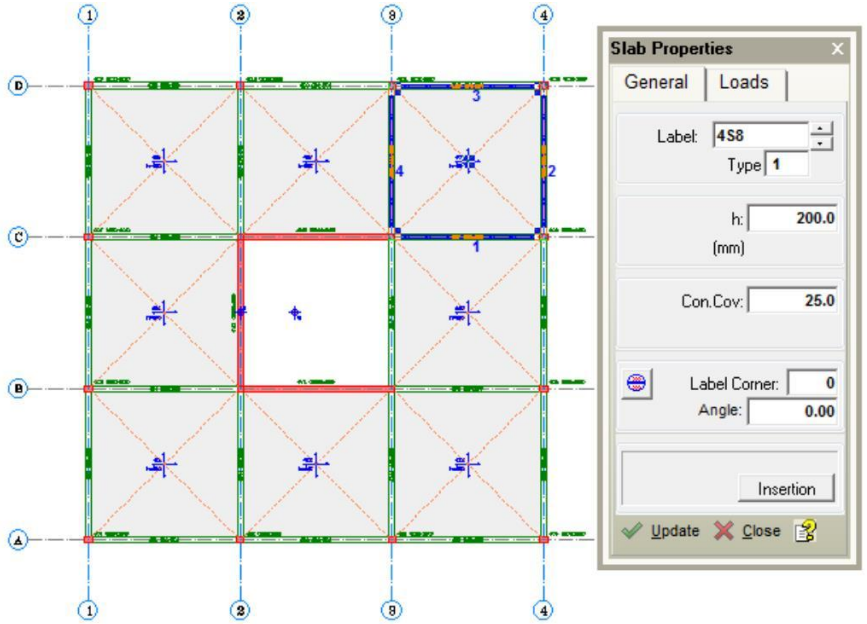
6. To create the first slab position the cursor in the square between grid points A-1 and B-2 and left-click.

You will see your first slab — 4S1 as below, this also includes the yield line for the slab load distribution.



If yield lines are not visible, these can be activated as follows. Pick Settings/View Settings/Slabs. Check the box Display Yield Lines.

7. Repeat the above process to define seven more slabs to achieve the layout shown below:

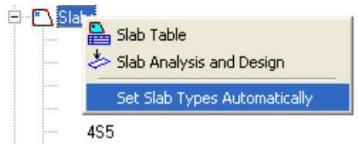


Setting Slab Types Automatically

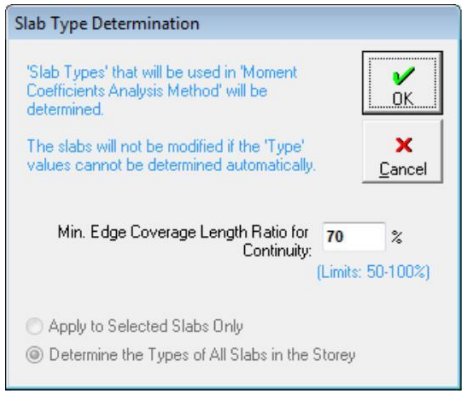
To set the slab types in accordance with BS 8110 Table 3.14 automatically, proceed as follows:

1. To clear the selection of any members click the Clear Selection Set icon.

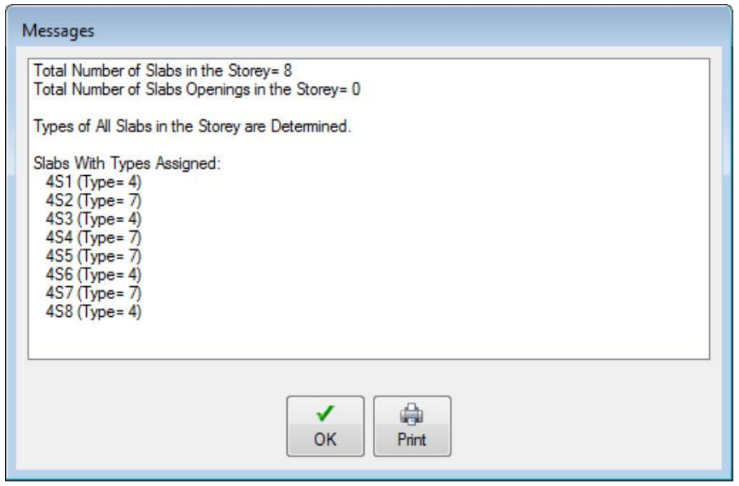
2. Right mouse click the Slabs folder in the Structure Tree and select Set Slab Types Automatically.



You will see the Slab Type Determination dialog.



3. Click OK to proceed. Orion determines the slab types.



4. Click OK once more to close the Messages dialog.

At this point the model is ready for various types of analysis and design. If you are in a hurry, you could skip to page 31 and do this. We are going to cover a few other useful options first.

Creating Ribbed Slabs

As an alternative to normal and flat slabs you can also define and use ribbed and waffle slabs. These slabs types are not within the scope of this Quick Start Guide. For further information on these types of slab refer to the Structure Modelling topic in the Orion Help system.

Slab Design and Detailing

For beam/slab models (as opposed to flat slab), you determine slab reinforcement requirements by inserting slab strips in the X & Y directions. This then automatically determines the reinforcement required for the different slab types by using Table 3.14 from BS 8110.

Strips provide flexibility to design and detail reinforcement in any plan orientation. This enables very complex floor layouts to be designed and detailed.

The slab strips parallel to the horizontal axes will be labelled X1, X2… and those parallel to the vertical direction axes will be labelled Y1, Y2…

First we will define a strip labelled X1 through the slabs between grid lines A-B and which cross grid lines 1 to 5.

1. Click the Slab Strip icon to see the Slab Strip Properties dialog.

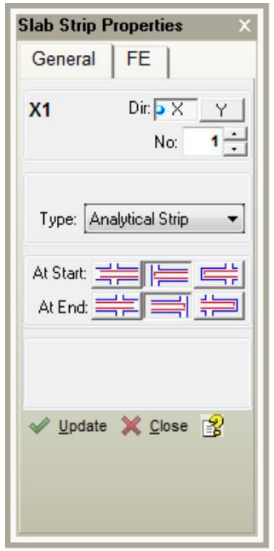
When drawing strips it is essential that you set the correct start and end conditions.

There are three options:

Slab — The strip starts or ends inside a slab. The bottom steel for the slab in question is not designed, but the span of the slab can be defined and this value is used in determining the support steel.

Bob — The strip starts or ends beyond an edge beam or wall. The support steel at the edge is bent down into the beam/wall.

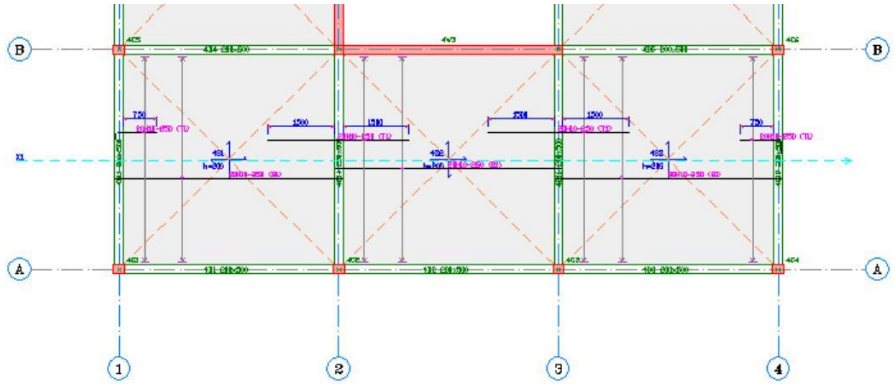
Cantilever — The strip starts or ends beyond a cantilever slab.



2. Ensure the label is X1 and indicate a Bob at both the start and end of the strip by clicking on the appropriate end conditions as shown above.

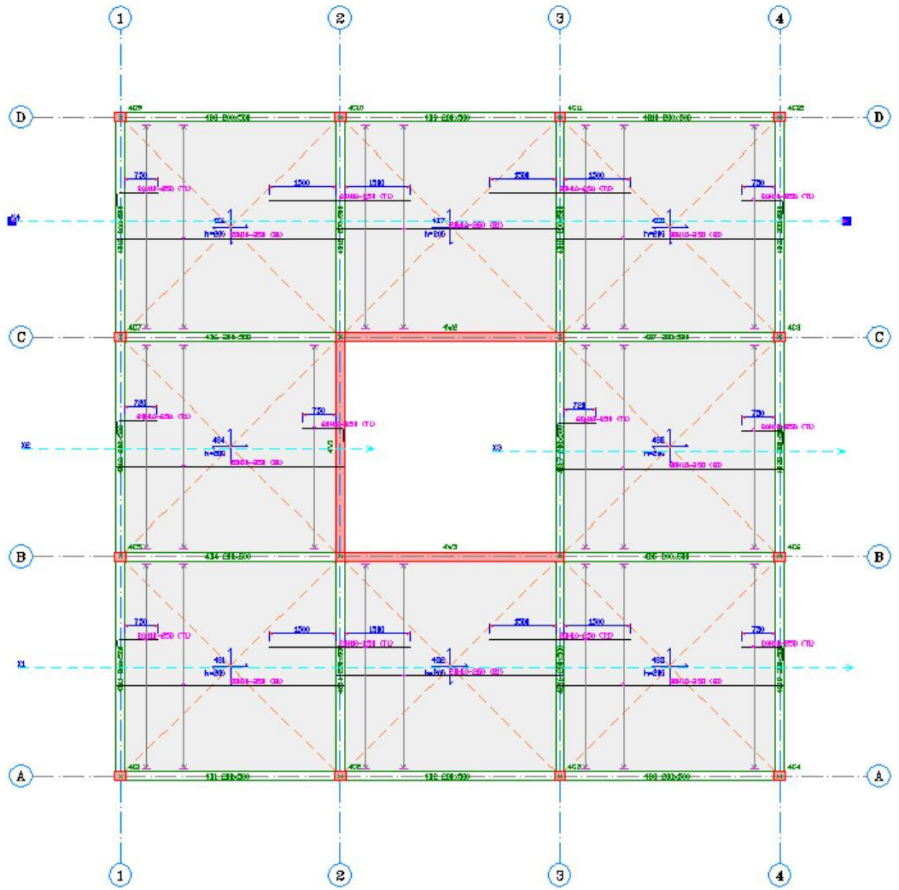
3. Position your cursor above Grid A but to the left of Grid 1 (so that it is not in the model). Now press and hold the CTRL key and at the same time click and drag to create a horizontal line which extends past Grid 4.

Your screen should look similar to this.



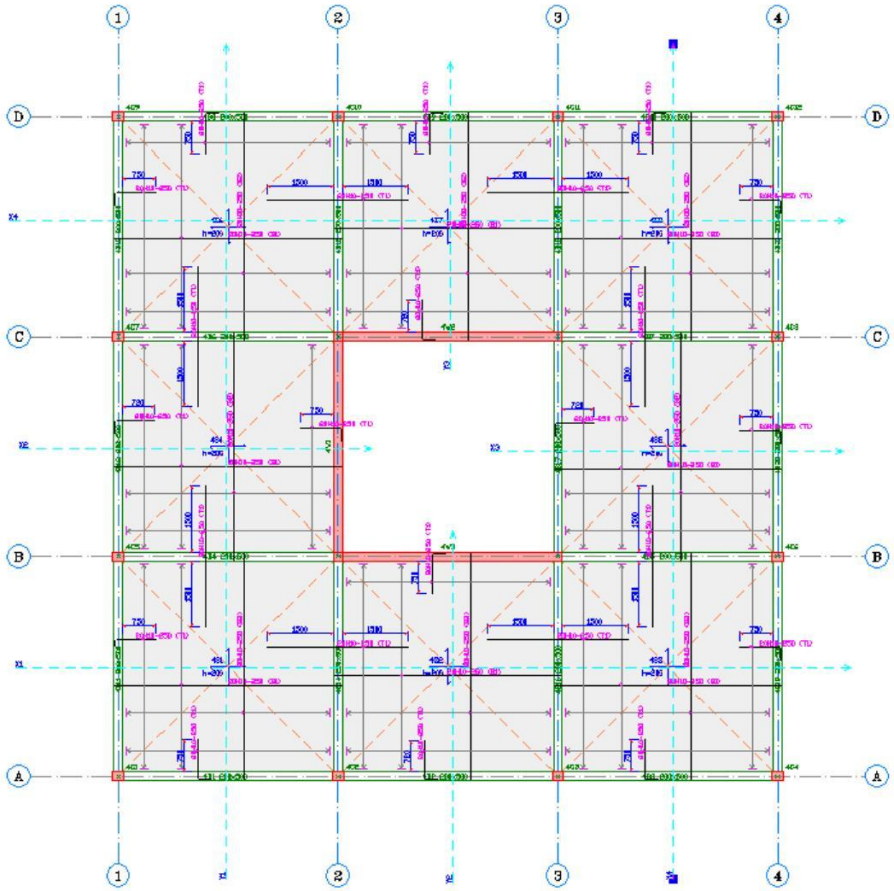
We specified the bar sizes which Orion can use in slab design earlier (see “Defining Materials” on page 15 .) You can also control the slab bar spacing range that will be applied by picking Settings / Slab Design Settings / Steel Bars.

4. Create another similar strip labelled X2 by repeating the process for the slabs between grid lines B-C starting to the left of grid line 1 and ending to the right of grid line 2.
5. Draw a third strip and a fourth strip in the X direction to achieve the layout below.



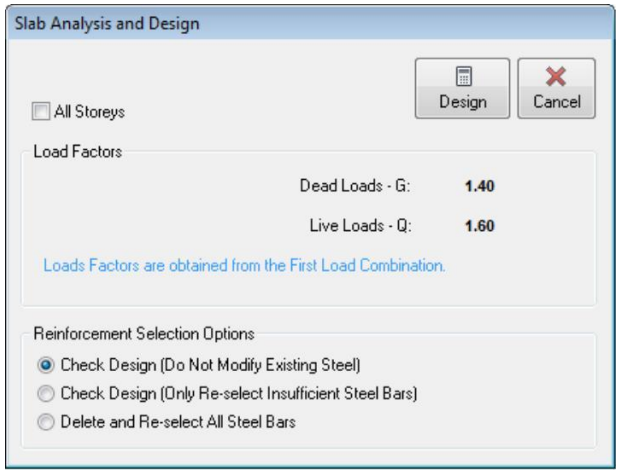
6. Now we can insert some vertical strips. Create strip Y1 between grid lines 1 and 2 and which extends from below grid line A to beyond grid line D.

7. Create additional strips Y2, Y3 and Y4 as shown below:

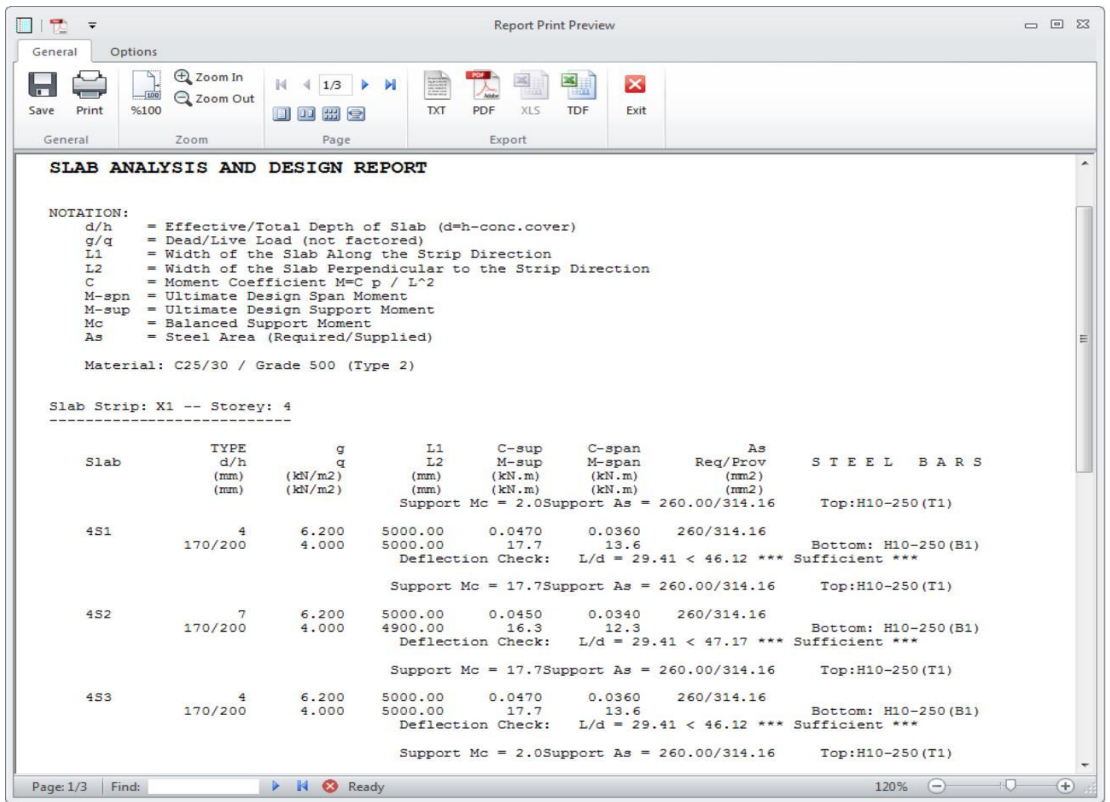


We can now re-check strips as a batch and create a report.

8. Pick Run / Slab Analysis and Design.



9. Ensure that the option Check Design (Do Not Modify Existing Steel) is selected and then click Design to see the report below:



10. Click Save followed by Exit to shut the Report Print Preview.
11. Click Cancel to close the Slab Analysis and Design dialog.

Dimensions

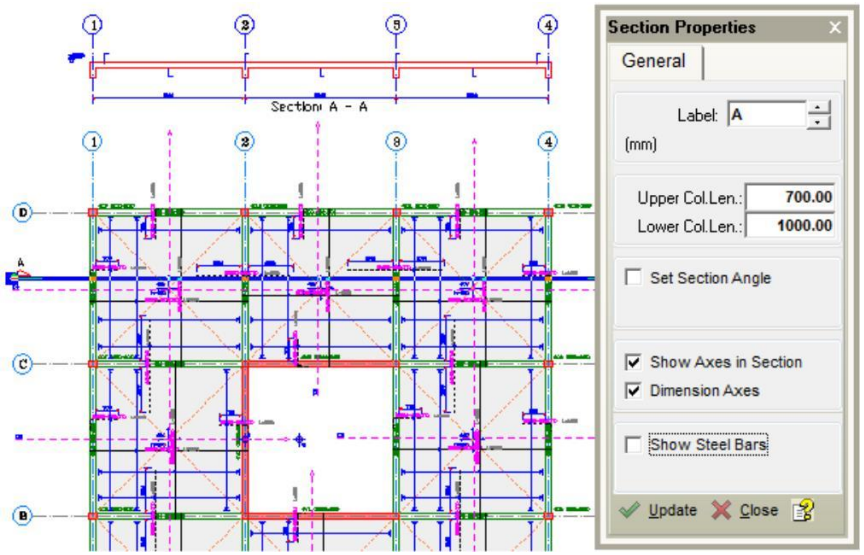
You can dimension the plan view at any time. Again this is not within the scope of this Quick Start Guide.
Please refer to the Structure Modelling topic in the Orion Help system for more information.

Sections

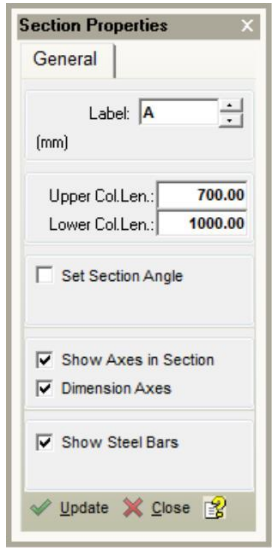
You can cut any sections through your model that you require. In this example we will create a horizontal section.

1. Click the Section icon or pick Member / Section.
2. Click and drag a horizontal line mid way between axes C and D. (Press the CTRL key down while dragging to ensure that the line is exactly horizontal.)

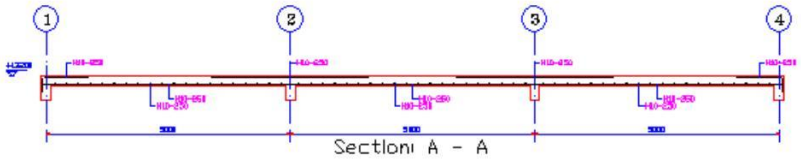
3. Now click above the plan to indicate the location where Orion is to place the cross section.



4. In the Section Properties dialog, tick Show Steel Bars and click Update.



You will now see the bars required at this cross section as shown below.



Slab Loads

You add Slab-point, -line and -patch loads using this command. This is not within the remit of this Quick Start Guide. Refer to the more comprehensive model in the Training Notes for an example and/or refer to the Structure Modelling topic in the Orion Help system for more information.

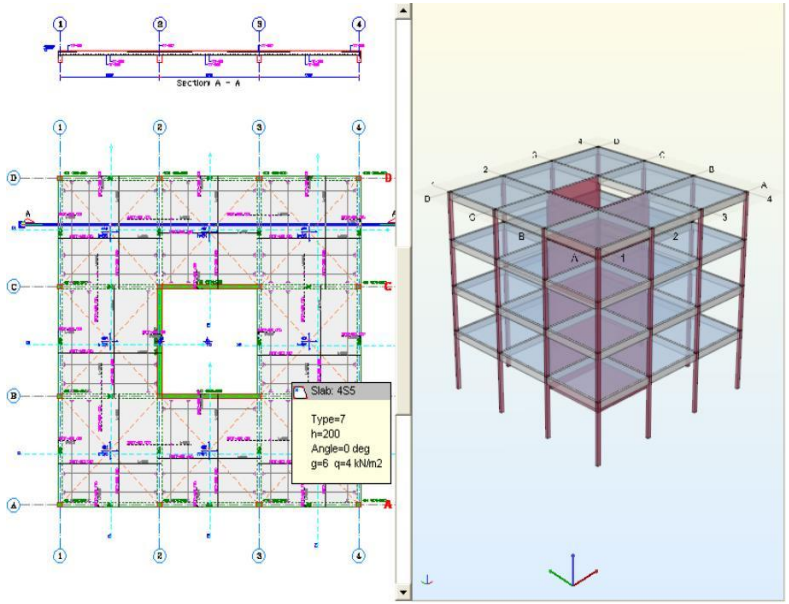
Generating a 3D View of the Model

Orion allows you to obtain a 3D view of the model, and to choose different layouts of Plan- (P) and 3D-view windows. You can create different 3D views in different windows. To switch between the different views use the tabs located at the bottom left of the Graphic Editor.

1. Choose the tab shown on the right to tile a 3D view and plan view horizontally as shown below: 

Although information has only been defined at ST04, all the lower storeys have been created automatically. The program assumes that unless a floor has at least one member defined, it is to be an exact duplicate of the floor above.

2. Click the 3D View to make its window active.



You can manipulate the 3D View in a number of ways:

- To spin, zoom or pan the image dynamically click and drag the right mouse button, turn the mouse scroll wheel or hold the scroll wheel down and drag.

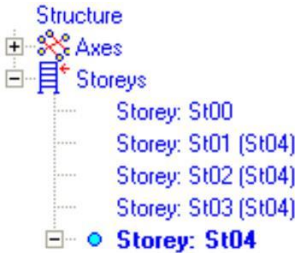
- To change the way the 3D View is displayed use the 3D View Settings icon.

- To display a menu showing various different filters use the Filters icon.
  Select Storey and Member Type Filters from this menu in order to filter by storey, member type or axis.

- Use the Animation icon to rotate the building about a vertical axis.

The Structure Tree Storey List

The Storey list menu in the Structure Tree is used to change from one floor to another. This list also indicates which storeys have had members defined (those with a blue circle mark adjacent to them). Storeys which have not had any members defined are shown without a circle mark. These storeys adopt the same member layout as the storey above.



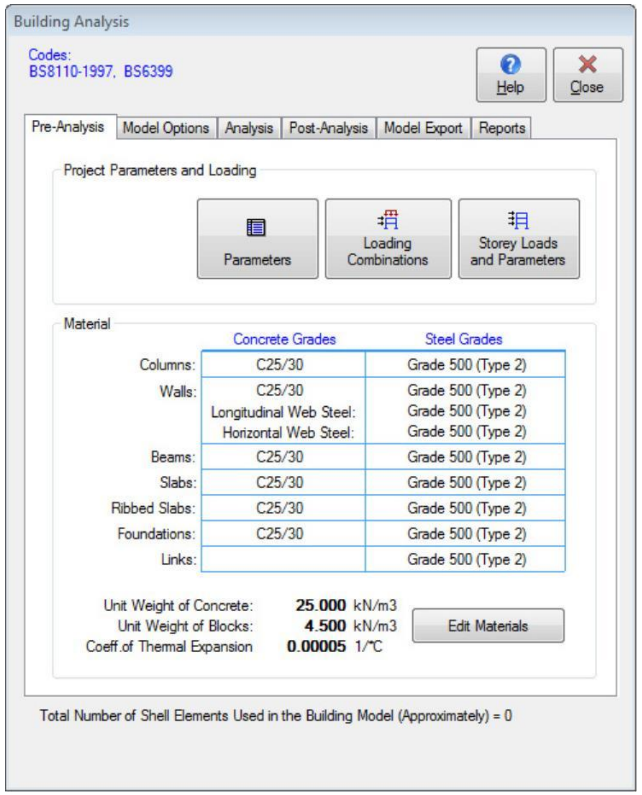
Hence in our model storeys St01, St02 & St03 are assumed to be identical to the 4 storey, and any changes we make to the 4 storey will also apply to the 3 , 2 and 1st storeys.

You double-click any floor in the tree view to switch to that floor.

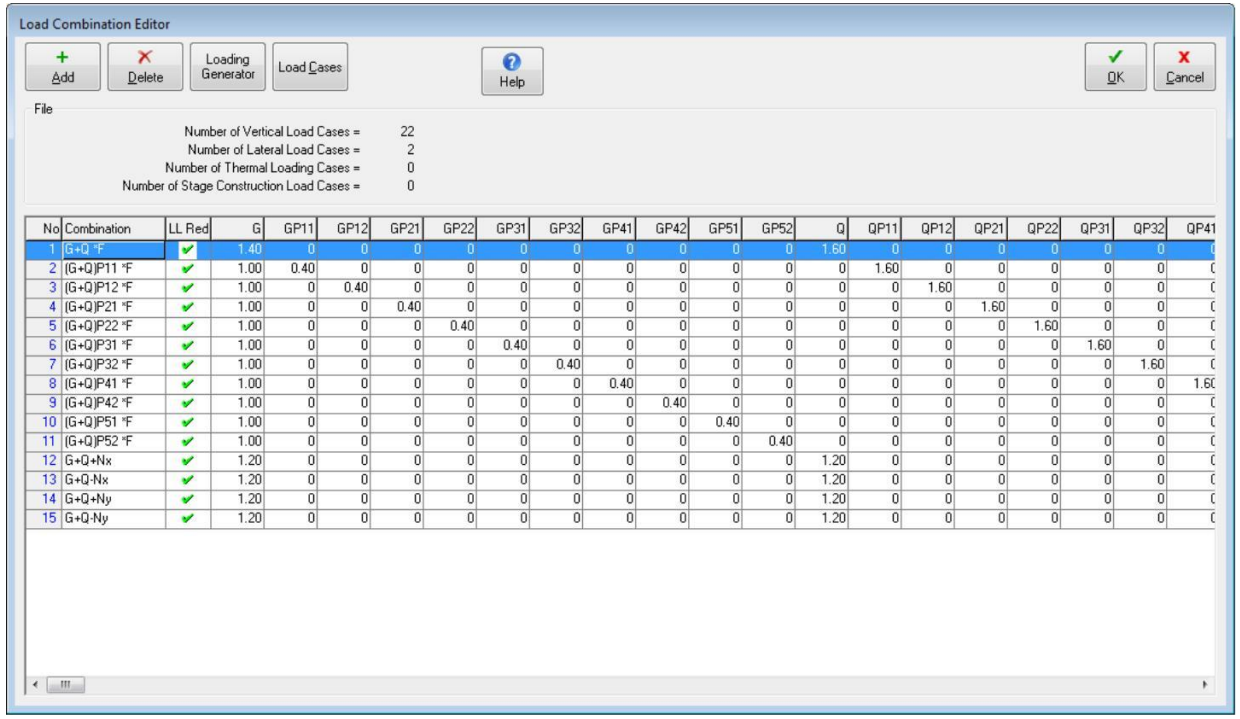
Defining Load Combinations

The template selected when the project was first started configures various settings including a default set of load combinations. These can be reviewed and edited if required from the Building Analysis dialog.

1. Pick Run / Building Analysis to see the Building Analysis dialog.



2. Click Loading Combinations to see the load combination set that was imported as part of the template.



No	Combination	LL Red	G	GP11	GP12	GP21	GP22	GP31	GP32	GP41	GP42	GP51	GP52	Q	QP11	QP12	QP21	QP22	QP31	QP32	QP41
1	G+Q *F	✔	1.40	0	0	0	0	0	0	0	0	0	0	1.60	0	0	0	0	0	0	0
2	(G+Q)P11 *F	✔	1.00	0.40	0	0	0	0	0	0	0	0	0	0	1.60	0	0	0	0	0	0
3	(G+Q)P12 *F	✔	1.00	0	0.40	0	0	0	0	0	0	0	0	0	0	1.60	0	0	0	0	0
4	(G+Q)P21 *F	✔	1.00	0	0	0.40	0	0	0	0	0	0	0	0	0	0	1.60	0	0	0	0
5	(G+Q)P22 *F	✔	1.00	0	0	0	0.40	0	0	0	0	0	0	0	0	0	0	1.60	0	0	0
6	(G+Q)P31 *F	✔	1.00	0	0	0	0	0.40	0	0	0	0	0	0	0	0	0	0	1.60	0	0
7	(G+Q)P32 *F	✔	1.00	0	0	0	0	0	0.40	0	0	0	0	0	0	0	0	0	0	1.60	0
8	(G+Q)P41 *F	✔	1.00	0	0	0	0	0	0	0.40	0	0	0	0	0	0	0	0	0	0	1.60
9	(G+Q)P42 *F	✔	1.00	0	0	0	0	0	0	0	0.40	0	0	0	0	0	0	0	0	0	0
10	(G+Q)P51 *F	✔	1.00	0	0	0	0	0	0	0	0	0.40	0	0	0	0	0	0	0	0	0
11	(G+Q)P52 *F	✔	1.00	0	0	0	0	0	0	0	0	0	0.40	0	0	0	0	0	0	0	0
12	G+Q+Nx	✔	1.20	0	0	0	0	0	0	0	0	0	0	1.20	0	0	0	0	0	0	0
13	G+Q-Nx	✔	1.20	0	0	0	0	0	0	0	0	0	0	1.20	0	0	0	0	0	0	0
14	G+Q+Ny	✔	1.20	0	0	0	0	0	0	0	0	0	0	1.20	0	0	0	0	0	0	0
15	G+Q-Ny	✔	1.20	0	0	0	0	0	0	0	0	0	0	1.20	0	0	0	0	0	0	0

The above set consists of 15 combinations; some being patterned gravity load combinations and others being lateral load combinations.

The Loading Generator can be used to easily re-generate a different set of combinations to replace the above if required.

3. As the default combination set is acceptable in this case, click Cancel to close the Load Combination Editor without making any changes and return to the Building Analysis dialog.

Building Analysis and Design

The Building Analysis dialog consists of a number of tabbed pages:

Pre-Analysis Settings

In addition to the definition of Load Combinations the Pre-Analysis tab is used to access a number of other function buttons:

Parameters allows you to review and/or modify any building parameters you may have defined.

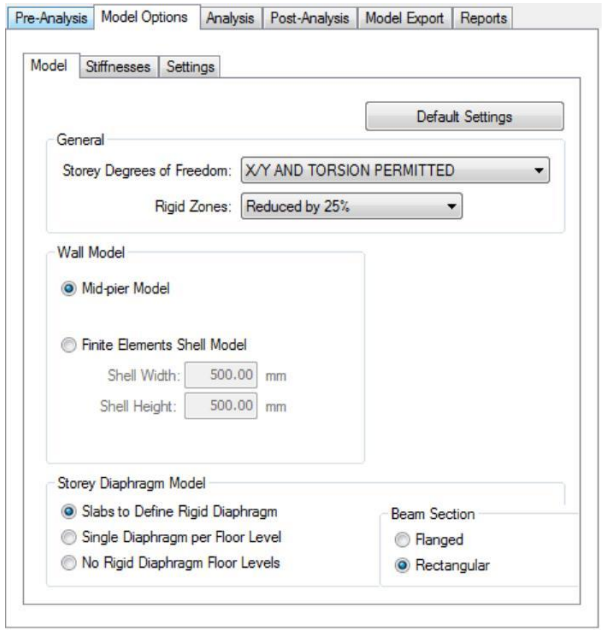
Loading Combinations allows you to review and/or modify an existing set of load combinations. You can also create new load combination sets from here using the Loading Generator.

Storey Loads and Parameters allows you to review and/or modify any lateral load cases applied at each storey. Automatically generated lateral loads are only available after completing the analysis data preparation stage.

Edit Materials allows you to review and/or modify the concrete and steel material properties.

Building Analysis Model Options

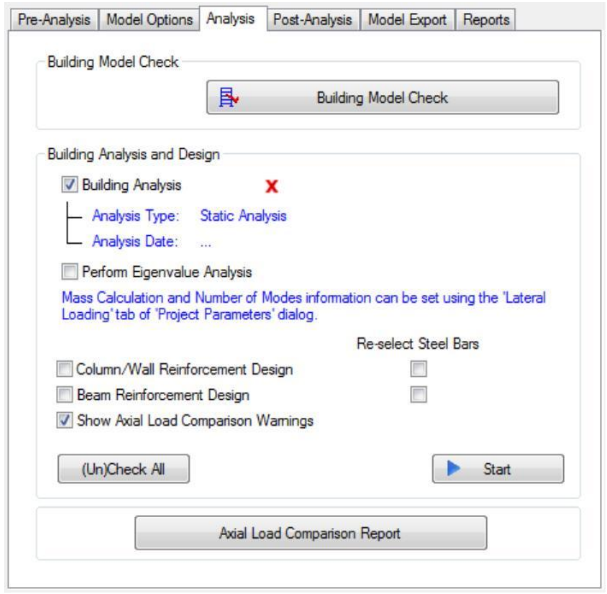
1. Click the Model Options tab and set the options as shown below.



Note the rigid zones option has been set to Reduced by 25%. Rigid Zones and the various other analysis parameters shown here are fully described in the Orion Help system and Engineer's Handbook.

The Analysis Tab

1. Pick the Analysis tab.



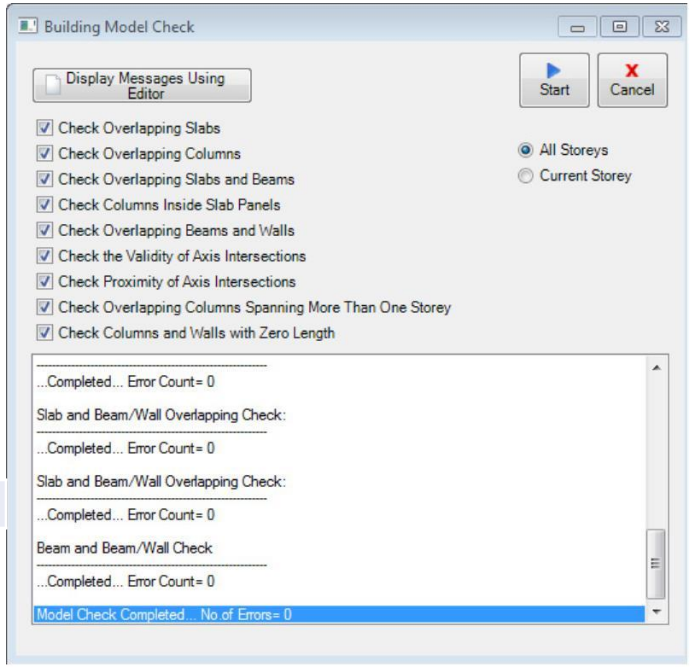
Before we analyse the model let's check its validity.

2. Click Building Model Check, choose All Storeys and then click Start.

This will check that the building is valid for those conditions indicated.

Even if this process doesn't find any errors, it doesn't guarantee that the building is modelled correctly.

3. Assuming that there are no errors click Cancel to shut the dialog.



Running the Analysis

Building Analysis is performed from the Analysis tab. Optionally after the analysis is complete, Orion can automatically perform Column/Wall Reinforcement Design and Beam Reinforcement Design for all members in the building.

1. Click (Un)Check All to activate the optional Column/Wall and Beam designs. Also activate the options to re-select Steel Bars.
2. Click Start to begin the batch analysis and design process.
   Orion analyses the building and designs all the columns, walls and beams.
3. When the analysis has completed click OK.

Cross Checking the Analysis Results

An important cross check on the validity of the analysis is the Axial Load Comparison Report. This report sums all the dead and live loads applied at each storey level and also displays the axial forces in the columns and shear walls. These values should be within a few percent of each other. If this is not the case, then you need to determine the reason for any discrepancy. The report can be automatically displayed at the end of the analysis process if any warnings have occurred during the cross checking process.

1. Pick Axial Load Comparison Report.

An example axial load comparison report is given below.

SUM OF APPLIED LOADS (Using Un-Decomposed Slab Loads):

G - Dead Loads:

Storey	Column (kN)	Wall (kN)	Beam (kN)	Slab (kN)	Rib (kN)	Total (kN)
St04	49.5	257.4	287.7	1142.8	0.0	1737.4
St03	49.5	257.4	287.7	1142.8	0.0	1737.4
St02	49.5	257.4	287.7	1142.8	0.0	1737.4
St01	49.5	257.4	287.7	1142.8	0.0	1737.4
Total						6949.5

Q - Live Loads:

Storey	Column (kN)	Wall (kN)	Beam (kN)	Slab (kN)	Rib (kN)	Total (kN)
St04	0.0	0.0	84.0	737.3	0.0	821.3
St03	0.0	0.0	84.0	737.3	0.0	821.3
St02	0.0	0.0	84.0	737.3	0.0	821.3
St01	0.0	0.0	84.0	737.3	0.0	821.3
Total						3285.1

SUM OF APPLIED LOADS (After Decomposing Slab Loads):

G - Dead Loads:

Storey	Column (kN)	Wall (kN)	Beam (kN)	Slab (kN)	Rib (kN)	Total (kN)
St04	49.5	257.4	1430.5	0.0	0.0	1737.4
St03	49.5	257.4	1430.5	0.0	0.0	1737.4
St02	49.5	257.4	1430.5	0.0	0.0	1737.4
St01	49.5	257.4	1430.5	0.0	0.0	1737.4
Total						6949.5

Q - Live Loads:

Storey	Column (kN)	Wall (kN)	Beam (kN)	Slab (kN)	Rib (kN)	Total (kN)
St04	0.0	0.0	821.3	0.0	0.0	821.3
St03	0.0	0.0	821.3	0.0	0.0	821.3
St02	0.0	0.0	821.3	0.0	0.0	821.3
St01	0.0	0.0	821.3	0.0	0.0	821.3
Total						3285.1

BUILDING ANALYSIS COLUMN/WALL AXIAL LOADS:

Storey	G (kN)	Delta G (kN)	Q (kN)	Delta Q (kN)
St04	1737.4	1737.4	821.3	821.3
St03	3474.8	1737.4	1642.6	821.3
St02	5212.2	1737.4	2463.8	821.3
St01	6949.5	1737.4	3285.1	821.3
Total		6949.5		3285.1

You can print this report, or save it for later inclusion in a batch print out of all the reports created by Orion for this model.

Normally the total “SUM OF APPLIED LOADS (Using Un-Decomposed Slab Loads)” values should be similar to those from the Decomposed Slab Loads table.

Provided that you can account for any difference between the un-decomposed and the decomposed values, you should compare the Total Decomposed Applied Dead Load with the Total Delta G value from the “BUILDING ANALYSIS COLUMN/SHEARWALL AXIAL LOADS” table.

Similarly, you should compare Total Decomposed Live Load against Total Delta Q.

You must be able to account for any differences in these values.

In more complicated models there are often small percentage differences, but they should never be more
than a few percent.

2. Click Save and then click to Exit the report.

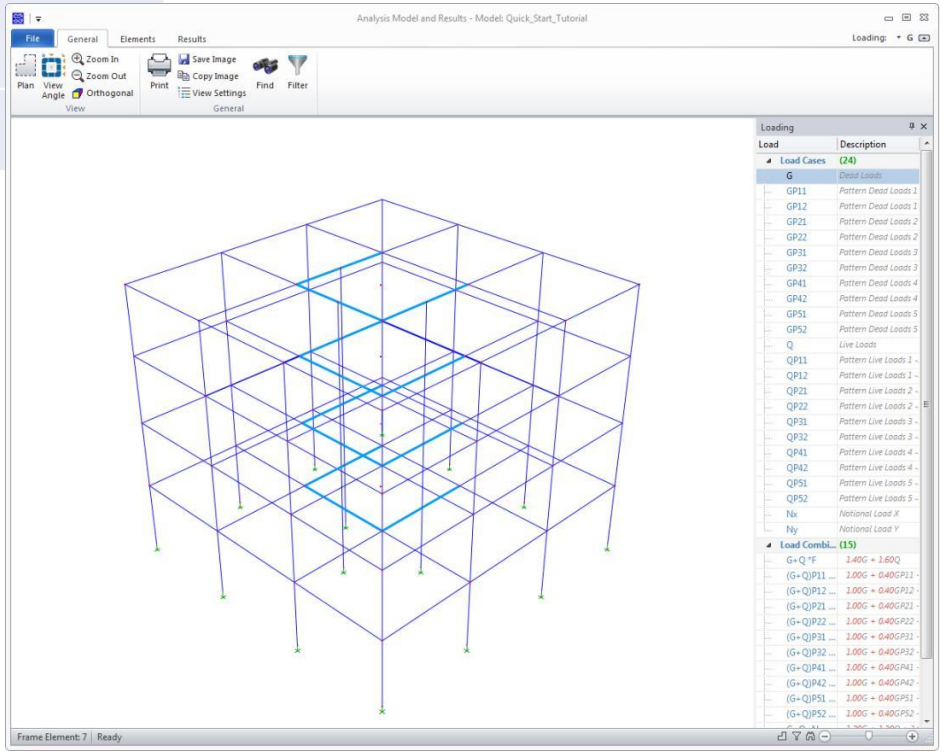
Post Analysis

1. Click on the Post-Analysis tab.

The buttons on the Post Analysis tab are as follows:

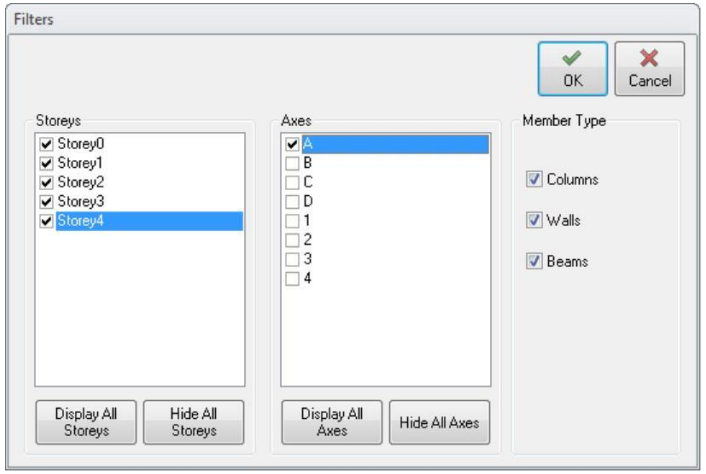
- Model and Analysis Results Display provides a graphical means of reviewing the analysis results.
- Analysis Results Report configures the numerical analysis results as required. These can then be can selected through the Output Reports: dropdown list, then previewed and printed through Preview Analysis Results Report.
- Column/Shearwall Design and Beam Reinforcement Design can be accessed from here also.

2. Pick Model and Analysis Results Display.

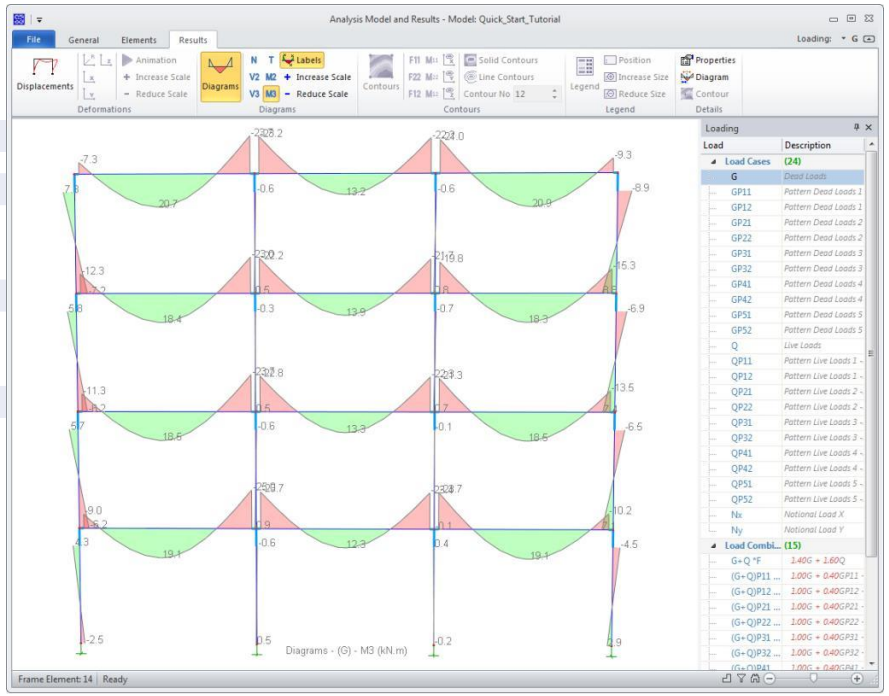


The most commonly used settings to control the appearance of the display are accessed from the buttons in the ribbon at the top of the window. Results are displayed for the chosen load case or combination that is selected from the Loading panel on the right of the window.

3. Click the Filter button (located on the General tab at the top of the screen) and hide all axes apart from axis A.



4. Use the right mouse button to rotate the frame to a front view, and use the scroll wheel to zoom in and out.
5. From the Elements tab click the Frame Element Loads button to view the decomposed element loads. Click the button once more to unselect.
6. From the Results tab click the Displacements button to view the deflected shape. Click the button once more to unselect.
7. Click the Diagrams button to view Frame Element Results and then choose the M3 button as the required effect to display.
8. Click the Increase Scale button multiple times until you obtain a plot similar to that shown below:



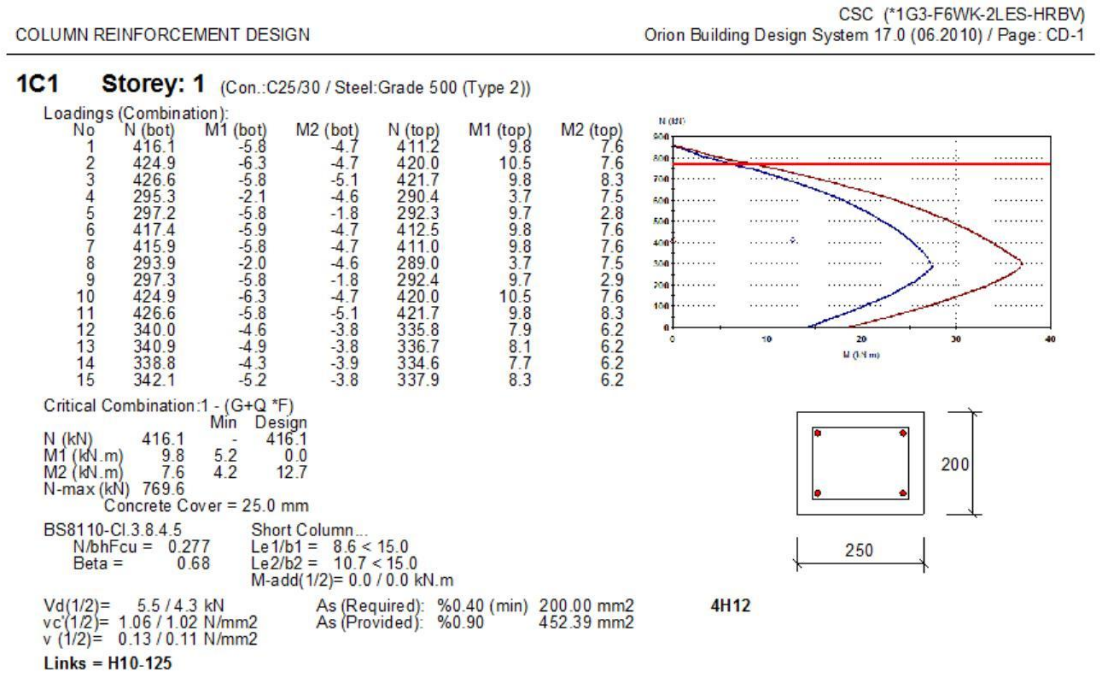
9. Experiment with the filters and effects to create other plots and then close the Analysis Model and Results window.
10. Close the Building Analysis dialog.

Column Design

1. Pick Run / Column Section Design.

Since you requested a column design as part of the building analysis process all the columns have already been batch designed. You can thus proceed to create a report immediately.

2. Click the Design Report button.



CSC (*1G3-F6WK-2LES-HRBV)
COLUMN REINFORCEMENT DESIGN — Orion Building Design System 17.0 (06.2010) / Page: CD-1

1C1 Storey: 1 (Con.:C25/30 / Steel:Grade 500 (Type 2))

Loadings (Combination):

No	N (bot)	M1 (bot)	M2 (bot)	N (top)	M1 (top)	M2 (top)
1	416.1	-5.8	-4.7	411.2	9.8	7.6
2	424.9	-6.3	-4.7	420.0	10.5	7.6
3	426.6	-5.8	-5.1	421.7	9.8	8.3
4	295.3	-2.1	-4.6	290.4	3.7	7.5
5	297.2	-5.8	-1.8	292.3	9.7	2.8
6	417.4	-5.9	-4.7	412.5	9.8	7.6
7	415.9	-5.8	-4.7	411.0	9.8	7.6
8	293.9	-2.0	-4.6	289.0	3.7	7.5
9	297.3	-5.8	-1.8	292.4	9.7	2.9
10	424.9	-6.3	-4.7	420.0	10.5	7.6
11	426.6	-5.8	-5.1	421.7	9.8	8.3
12	340.0	-4.6	-3.8	335.8	7.9	6.2
13	340.9	-4.9	-3.8	336.7	8.1	6.2
14	338.8	-4.3	-3.9	334.6	7.7	6.2
15	342.1	-5.2	-3.8	337.9	8.3	6.2

Critical Combination:1 - (G+Q *F)

		Min	Design
N (kN)	416.1	-	416.1
M1 (kN.m)	9.8	5.2	0.0
M2 (kN.m)	7.6	4.2	12.7
N-max (kN)	769.6		

Concrete Cover = 25.0 mm

BS8110-Cl.3.8.4.5 — Short Column...
N/bhFcu = 0.277 — Le1/b1 = 8.6 < 15.0
Beta = 0.68 — Le2/b2 = 10.7 < 15.0
M-add(1/2)= 0.0 / 0.0 kN.m

Vd(1/2)= 5.5 / 4.3 kN
vc'(1/2)= 1.06 / 1.02 N/mm2
v (1/2)= 0.13 / 0.11 N/mm2

As (Required): %0.40 (min) 200.00 mm2
As (Provided): %0.90 452.39 mm2

4H12

Links = H10-125

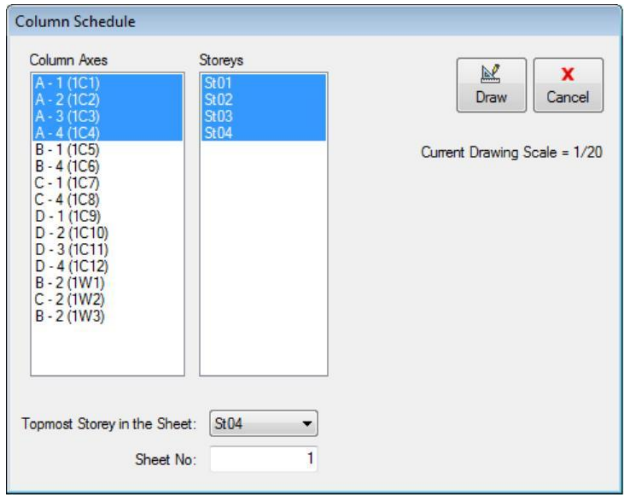
200

250

3. Click Save then close the report.

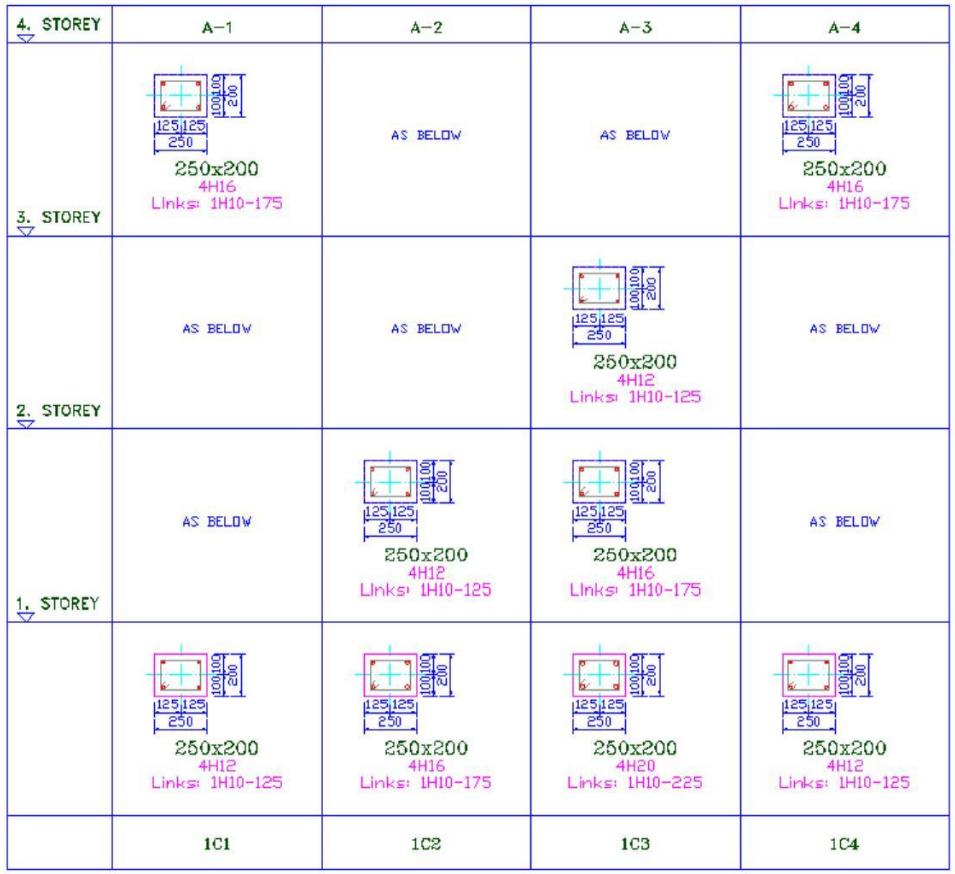
You can also generate a column schedule quickly.

4. Pick Detail Drawings/ Column Schedule. Highlight several of the columns as shown, and then click Draw.



5. A dialog appears telling you the size of sheet you require and the drawing scale, click OK to continue.

A schedule drawing is produced as follows:



6. Close the Schedule and the Column Reinforcement Design dialog.

If any of the columns fail, or if you want to modify the batch design, then you can run the column design interactively. Refer to the Orion Training Manual for further details on using the Column Design Editor to do this.

Beam Design

1. Pick Run / Beam Section Design and Detailing » Storey Beams.

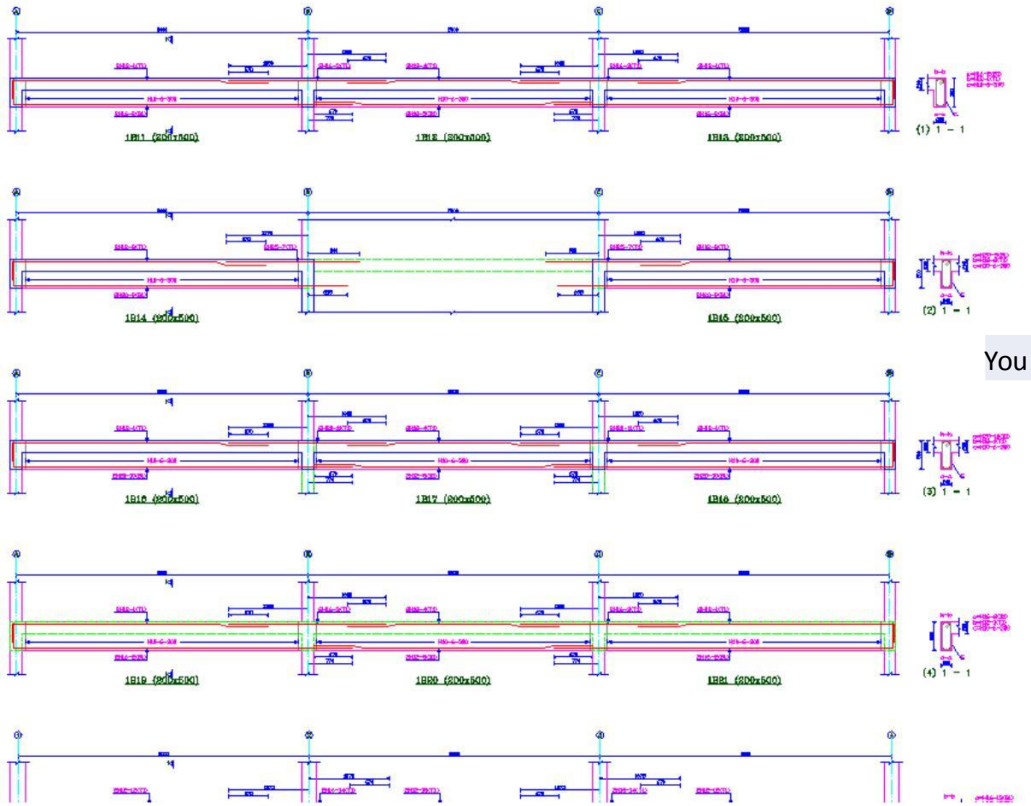
Since you set the option to batch design the beams as part of the building analysis you can view the results immediately.

As with the column design, If any of the beams fail, or if you want to modify the batch design, then you can run the design interactively. Refer to the Orion Training Manual for details of how to do this.

The designs you obtain are very dependant on the current Settings. You can adjust these to suit your own

Those beams that have been successfully designed can be placed on to a drawing sheet automatically.

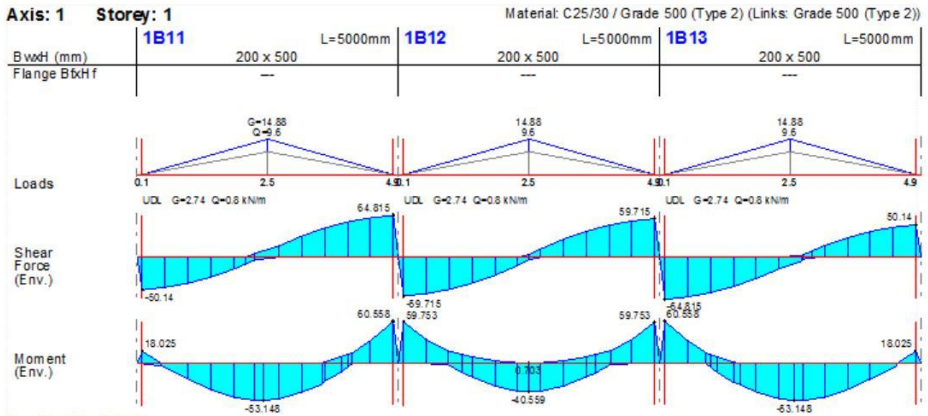
2. Pick Detail Drawings/Automatic Detail Drawings of All Axes and check Storey-1
3. Set the Number of Columns in the Sheet to 1 and click OK to create the drawing.



You can control the information presented on the detail drawings via the Settings menu.

4. Close the drawing.

5. Now to create a report pick Design / Design Report and then click OK to create it.



Bending (Top Edge) ...									
M (kN.m)	18.0	5.1	60.6	59.8	11.8	59.8	60.6	5.1	18.0
d (mm)	459.0	459.0	457.0	457.0	459.0	457.0	457.0	459.0	459.0
K/K'	0.09	0.03	0.31	0.31	0.06	0.31	0.31	0.03	0.09
x (mm)	51.00	51.00	57.82	57.01	51.00	57.01	57.82	51.00	51.00
As (mm2)	95.07	26.99	323.18	318.61	62.12	318.61	323.18	26.99	95.07
As' (mm2)	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
As-min	130.00	130.00	130.00	130.00	130.00	130.00	130.00	130.00	130.00
Bending (Bottom Edge) ...									
M (kN.m)	16.5	53.1		0.9	40.6	0.9		53.1	16.4
d (mm)	457.0	457.0		459.0	459.0	459.0		457.0	457.0
K/K'	0.08	0.27		0.00	0.21	0.00		0.27	0.08
x (mm)	50.78	50.78		51.00	51.00	51.00		50.78	50.78
As (mm2)	87.47	281.56		4.88	213.93	4.88		281.56	87.02
As' (mm2)	0.00	0.00		0.00	0.00	0.00		0.00	0.00
As-min	130.00	130.00	130.00	130.00	130.00	130.00	130.00	130.00	130.00
Shear and Torsion Design ...									
Vd (kN)	50.1		61.1	59.7		56.0	64.8		46.5
v (N/mm2)	0.55		0.67	0.65		0.61	0.71		0.51
vc (N/mm2)	0.42	0.51	0.51	0.51	0.42	0.51	0.51	0.51	0.42
v-max (N/mm2)	4.38		4.38	4.38		4.38	4.38		4.38
Vnom (kN)		150.7			143.1			150.7	
Td (kN.m)		0.0			0.0			0.0	
vt (N/mm2)		0.00			0.00			0.00	
vt-min(N/mm2)		0.00			0.00			0.00	
b-sup (mm)	0.0		0.0	0.0		0.0	0.0		0.0
Links	**H10-300**	**H10-300**	**H10-300**	**H10-300**	**H10-300**	**H10-300**	**H10-300**	**H10-300**	**H10-300**
Deflection Check ...									
L/d	10.94 < 41.54 OK			10.89 < 35.63 OK			10.94 < 41.54 OK		
Provided Steel Areas (mm2)									
Top Edge	226.19	226.19	402.12	402.12	226.19	402.12	402.12	226.19	226.19
Bottom Edge	402.12	402.12	402.12	226.19	226.19	226.19	402.12	402.12	402.12
Steel Bars ...									
Top Bar		2H12			2H12			2H12	
Top SupBar			2H16	2H16		2H16	2H16		
Top SupBar									
Bottom Bar		2H16			2H12			2H16	
Bottom Bar									
Bot SupBar									

6. Click Save, and then Exit the report and Exit the Beam Design window.

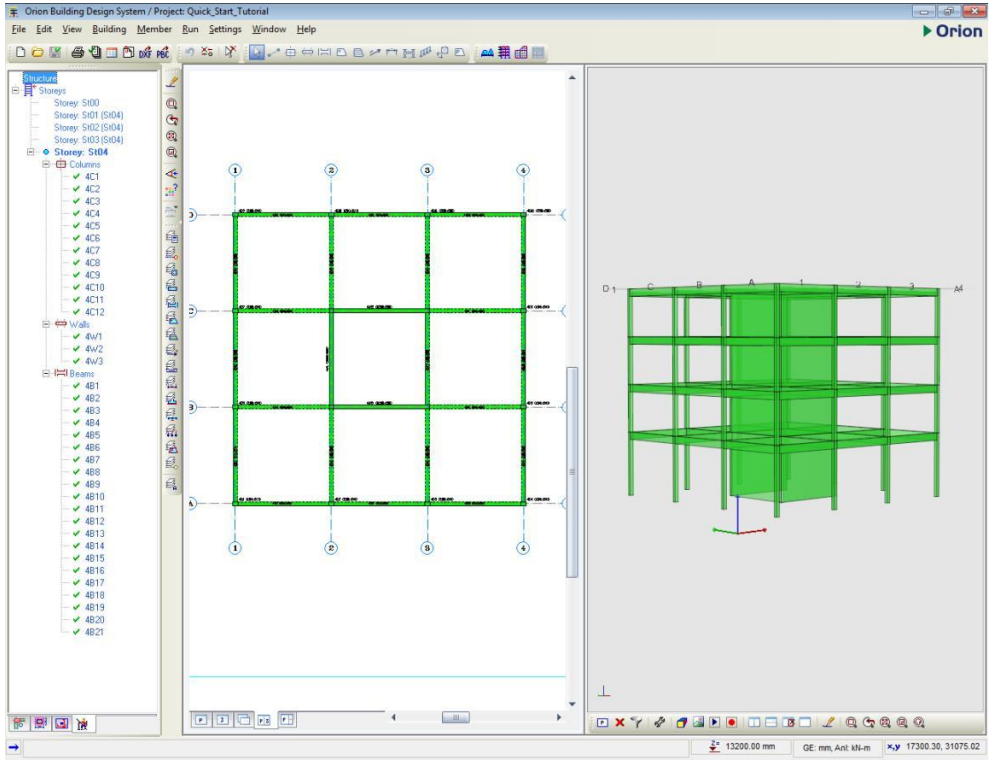
Design Status

You can display the member design status graphically in both the plan and 3D window.

1. Click Design Status from the bottom of the Structure tree view as shown.

The members are shown colour coded as follows:

- Green = PASS,
- Red = FAIL - insufficient area of steel provided,
- Unhatched = FAIL? - although a sufficient area of steel has been provided, the member still fails due to spacing requirements.

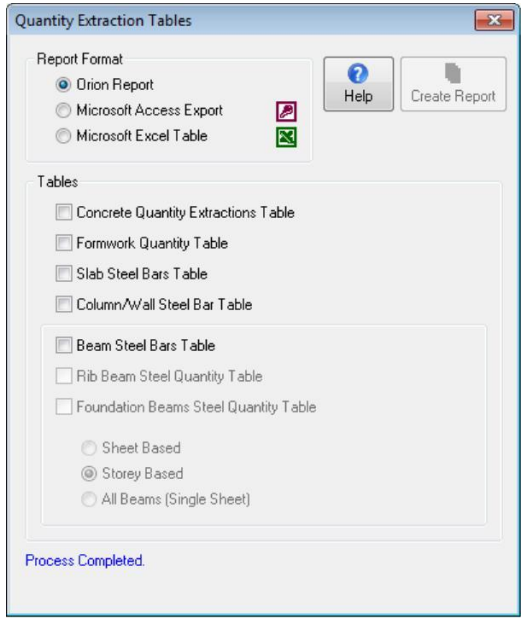


The design status of each member is also indicated in the Structure tree view.

Quantity Extraction Tables

You can extract quantity reports in various formats.

1. Pick File / Quantity Extraction Tables.



2. Check Slab Steel Bars Table and then click Create Report.

Shown below is an extract from the resulting slab steel bar quantity report.

SLAB STEEL BAR SCHEDULE CSC (*1G3-F6WK-2LES-HRBV)

TOTAL STEEL
4. STOREY SLABS

TYPE (mm)	UNIT WGT.(kg/m)	TOTAL LEN (m)	TOTAL WGT. (kg)
H10	.6165	2576.0	1588.2
Total			1588.2

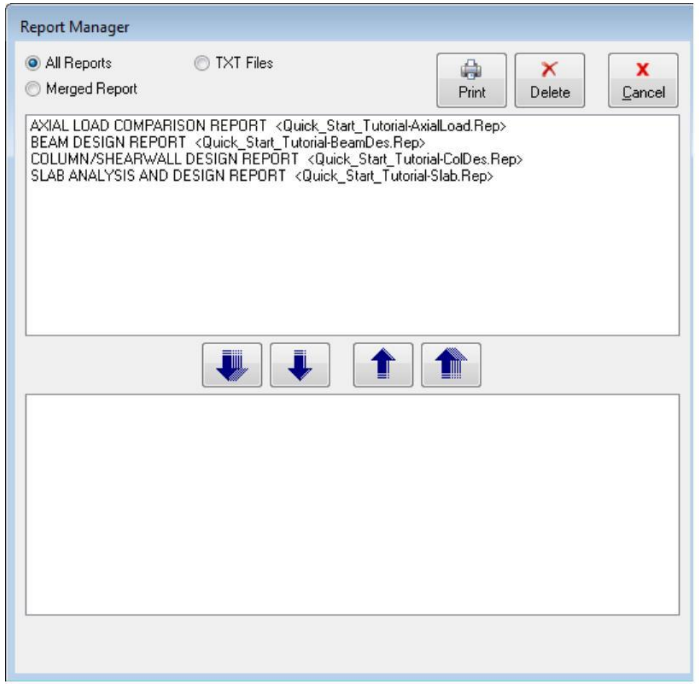
STEEL BAR TABLE - 4. STOREY SLABS

POS	TYPE (mm)	QTY	LENGTH (mm)	TOTAL LEN (m)	SHAPE	MEMBERS
1	H10	320	5200	1664.0	00	
2	H10	320	1250	400.0	11	
3	H10	160	3200	512.0	00	

Report Manager

You can merge all saved reports into a single report for printing in one go. This also ensures consistent page numbering.

1. Pick File / Report Manager.



You can combine the separate reports which you created earlier in any order and then print them.

What Next?

In this very simple example you have created and analysed a small model, this may give you the confidence to go on and try something for yourself.