<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Thank you</td>
<td>5</td>
</tr>
<tr>
<td>Getting started</td>
<td>6</td>
</tr>
<tr>
<td>User interface</td>
<td>7</td>
</tr>
<tr>
<td>First Steps</td>
<td>8</td>
</tr>
<tr>
<td>Project Settings</td>
<td>10</td>
</tr>
<tr>
<td>Drawing Sheet Selection</td>
<td>11</td>
</tr>
<tr>
<td>Inserting Storey Height</td>
<td>12</td>
</tr>
<tr>
<td>Inserting Additional Floors</td>
<td>13</td>
</tr>
<tr>
<td>Selection methods</td>
<td>14</td>
</tr>
<tr>
<td>Zooming and Panning methods</td>
<td>15</td>
</tr>
<tr>
<td>Defining a grid pattern</td>
<td>16</td>
</tr>
<tr>
<td>Defining Materials</td>
<td>18</td>
</tr>
<tr>
<td>Defining Columns</td>
<td>22</td>
</tr>
<tr>
<td>Creating Walls</td>
<td>24</td>
</tr>
<tr>
<td>Creating Beams</td>
<td>25</td>
</tr>
<tr>
<td>Creating Slabs</td>
<td>27</td>
</tr>
<tr>
<td>Setting Slab Types Automatically</td>
<td>30</td>
</tr>
<tr>
<td>Creating Ribbed Slabs</td>
<td>32</td>
</tr>
<tr>
<td>Slab Design and Detailing</td>
<td>32</td>
</tr>
<tr>
<td>Dimensions</td>
<td>36</td>
</tr>
<tr>
<td>Sections</td>
<td>37</td>
</tr>
<tr>
<td>Slab Loads</td>
<td>38</td>
</tr>
<tr>
<td>Slab Openings</td>
<td>38</td>
</tr>
<tr>
<td>Generating a 3D View of the Model</td>
<td>39</td>
</tr>
<tr>
<td>Building Analysis and Design</td>
<td>41</td>
</tr>
<tr>
<td>Pre-Analysis Settings</td>
<td>41</td>
</tr>
<tr>
<td>Building Analysis Model Options</td>
<td>42</td>
</tr>
<tr>
<td>The Analysis Tab</td>
<td>43</td>
</tr>
<tr>
<td>Running the Analysis</td>
<td>45</td>
</tr>
<tr>
<td>Cross Checking the Analysis Results</td>
<td>45</td>
</tr>
<tr>
<td>Post Analysis</td>
<td>47</td>
</tr>
<tr>
<td>Column Design</td>
<td>51</td>
</tr>
<tr>
<td>Beam Design</td>
<td>55</td>
</tr>
<tr>
<td>Design Status</td>
<td>58</td>
</tr>
<tr>
<td>Quantity Extraction Tables</td>
<td>59</td>
</tr>
<tr>
<td>Report Manager</td>
<td>60</td>
</tr>
<tr>
<td>What Next?</td>
<td>61</td>
</tr>
</tbody>
</table>
Disclaimer

CSC (UK) Ltd does not accept any liability whatsoever for loss or damage arising from any errors which might be contained in the documentation, text or operation of the programs supplied.

It shall be the responsibility of the customer (and not CSC)

• to check the documentation, text and operation of the programs supplied,
• to ensure that the person operating the programs or supervising their operation is suitably qualified and experienced,
• to ensure that program operation is carried out in accordance with the user manuals,

at all times paying due regard to the specification and scope of the programs and to the CSC Software Licence Agreement.

Proprietary Rights

CSC (UK) Ltd, hereinafter referred to as the OWNER, retains all proprietary rights with respect to this program package, consisting of all handbooks, drills, programs recorded on CD and all related materials. This program package has been provided pursuant to an agreement containing restrictions on its use.

This publication is also protected by copyright law. No part of this publication may be copied or distributed, transmitted, transcribed, stored in a retrieval system, or translated into any human or computer language, in any form or by any means, electronic, mechanical, magnetic, manual or otherwise, or disclosed to third parties without the express written permission of the OWNER.

This confidentiality of the proprietary information and trade secrets of the OWNER shall be construed in accordance with and enforced under the laws of the United Kingdom.

Orion documentation: Orion software:
© CSC (UK) Ltd 2010 © CSC (UK) Ltd 2010
All rights reserved. All rights reserved.

Trademarks

Orion™ is a trademark of CSC (UK) Limited.
Fastrak5950™ is a trademark of CSC (UK) Limited.
TEDDS® is a registered trademark of CSC (UK) Limited.
The CSC logo is a trademark of CSC (UK) Limited.

Microsoft and Windows are either trademarks or registered trademarks of Microsoft Corporation in the United States and/or other countries.
Acrobat® Reader Copyright © 1987-2010 Adobe Systems Incorporated. All rights reserved. Adobe and Acrobat are trademarks of Adobe Systems Incorporated which may be registered in certain jurisdictions.
Thank you

Thank you for purchasing this copy of Orion. We know that you have made the right decision, and look forward to building a close relationship with you in the future.

We want to ease your use of this powerful product and ensure that you get maximum benefit in minimum time. That is what this Quick Start Guide¹ is all about. Using this guide you should be able to create, analyse and review results for this simple model in around 1 hour.

After that bigger (more realistic) models may be to some extent just a matter of repetition. Orion has numerous features to make defining models easier. As you turn from an Orion "novice" into an Orion "expert" you will probably want more information on other issues such as how best to model more complex or oddball structures. So when you have worked through the model in this Quick Start Guide we would recommend that you work through the Orion Standard Training Manual which is a printed version of the basic Orion training course.

Finally, we will always welcome your comments and ideas about Orion. Your input is very important to us. It enables us to ensure that our continuing development of Orion meets your requirements. Many of Orion's current features come from interaction with existing customers. We are committed to maintaining and updating Orion and your contributions are vital to this.

So, thank you again, both for purchasing Orion, and for the future input we anticipate you will provide.

The Orion team

¹ This document was prepared for Orion version 16.0. If you are using a newer version of the software some features may vary. The latest version of this guide is available from the Welcome Screen on Orion's Help menu.
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\(^1\), 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 on this toolbar. You can also access the options on this 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 the following syntax:

Pick Run / Beam Section Design and Detailing » Storey Beams'

\(^1\) Wherever we refer to an icon you will find a picture of that icon to the right of the text which refers to it.
This means that you should pick item shown below from the *Main menu*.

```
(File | Settings | Lightbox | Help)
(Load Decomposition by FE | Building Analysis | 2D Floor Analysis | FE Part Foundation Analysis | Column Section Design)
(Steel Section Design and Detailing | Structural Analysis and Design | Reinforced Slab Analysis | Slab Design | Column Punching Check | RESSO Programs)
```

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

*Steps you must perform* - In order to differentiate actions that you need to perform from any explanatory text we use the easily recognisable font shown below.

Leave the *Codes* tab unchanged and click *Analysis*.

### First Steps

If you have not yet installed *Orion*, then 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.

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

- *All Programs*,
- *CSC/Orion*¹,
- *Orion 16*².

This loads *Orion*, and displays the *Open Project* dialog. You use this both to select an existing project or to start a new one. You can specify the location of the project by clicking *Data Folder*.

---

¹ The start menu folder.
² The application.
1. **Click New Project.**

   Type the **Project Code** as shown using the '_' character to denote spaces.

   ![New Project dialog](image)

2. **Select OK.**

   This creates a folder automatically called *Quick_Start_Tutorial* within the default data directory. 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.

3. 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.
**Drawing Sheet Selection**

The *Drawing Sheet* selection dialog appears automatically.

1. Select the **Standard Sheet size AO (1188 x 840 mm)**, then set the drawing and detail scales to **1/50** and **1/20** respectively, then click **OK**.

![Drawing Sheet Selection Dialog](image)

You can also change the drawing and detail scales from this dialog.

**Note**

The sheet origin (0,0) is located at the lower left corner of the drawing sheet. If, after creating your model, you find it is too close to the edge of the sheet, then you can reposition it by clicking *Pick Sheet Origin*. 
Inserting Storey Height

The next dialog prompts for the Storey Height of the 1st storey.

1. Enter the storey height as \textbf{3300 mm} and click \textbf{OK}.

![Storey Information dialog]

After you enter the height of the 1st storey you will see the main drawing area (Graphic Editor). Initially the drawing area is completely empty.

![Main drawing area]
Inserting Additional Floors

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

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

2. Choose Insert storey.

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

   **Note** 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

This is the first button in the Members toolbar. At the moment the model has no members for you to select. You will 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 that cross the line.

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

Note For additional information about selecting single/multiple entities using the Pick command refer to the Orion Training Manual.
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.

**Note**

You can Zoom and Pan easily on-the-fly using the mouse.

- If you have a mouse with a scroll wheel, scroll it to zoom in and out.
- Hold the scroll wheel down and drag to pan the display.
- If you have a three button mouse you can use the middle button to zoom and pan.
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 Training Manual 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.

   Note: 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.

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

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

   Note: 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 below.

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 Analysis Form dialog.

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

Note: The concrete grade displayed in the above table will either display the cube strength only, or the cylinder/cube strength, depending on your current settings.
2. Click *Edit Materials*. 

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

**Note** You could apply an alternative grade or adjust the modulus of elasticity of an existing grade from here.
4. Click OK to return to the Materials tab.

This will set all structural members to have Grade 30 Concrete.

Note
The unit weight of concrete is set on this screen also. In this example 25kN/m$^3$ is used.

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

Note
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 Columns row).

You will notice that some bars sizes are selected by default, and you may find that the defaults you see may not be the same as those shown in the capture below. 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.

7. Make the settings shown above, or different settings if you prefer. Repeat this process for the other member types also.
8. Select **OK** to return to the *Materials* dialog which looks as shown below.

   ![Materials Dialog](image)

   **Concrete Grades**  | **Steel Grades** |
   ---------------------|-----------------|
   Column: C25/40        | Grade 500 (Type 3) |
   Wall: C25/40          | Grade 500 (Type 3) |
   Ribbed Slab: C25/40   | Grade 500 (Type 3) |
   Slab: C25/40          | Grade 500 (Type 3) |
   Links:                | Grade 500 (Type 3) |

<table>
<thead>
<tr>
<th>Unit Weight of Concrete</th>
<th>Unit Weight of Blocks</th>
<th>Coefficient of Thermal Expansion</th>
</tr>
</thead>
<tbody>
<tr>
<td>25.040 kg/m³</td>
<td>4.600 kg/m³</td>
<td>0.00015 °C</td>
</tr>
</tbody>
</table>

9. Select **OK** once more to return to the *Building Analysis* form and then **Close**.
Defining Columns

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

2. Enter $b_1, b_2, e_1$ and $e_2$ as shown below. The dimensions are explained by the diagram to the right of the dialog.

   ![Column Properties Panel](image)

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

   These properties will create a $250 \times 200$ 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.

   Tip: You might like to try out some of the zooming and panning methods on page 15 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 arrangement of walls shown below.
Creating Beams

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

2. In the Beam Properties dialog ensure that \( b \) is 200, \( h\text{-bot} \) is 500 and \( h\text{-top} \) is 0.0.

3. To ensure this click the alignment icon ( ) followed by the centre icon ( ). This sets \( b_2 \) to 100mm.

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

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

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

Your screen should now look like this.
Creating Slabs
Now we shall create the 4th 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 mm.
3. Click on the Loads tab and enter an additional Dead Load of 1.2 kN/m².
4. In the Imp. Load box do a right mouse click and select a value of 4.0 kN/m².

Note: 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.

![Slab Properties Window](image)

**Note** 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 — 45° as below, this also includes the yield line for the slab load distribution.

Note
If the slab does not appear as shown above, click the Insertion button and ensure the insertion method is set to Beam Region.

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**.
3. You will see the **Slab Type Determination** dialog.

![Slab Type Determination dialog]

4. Click **OK** to proceed. Orion determines the slab types.

![Messages dialog]

5. Click **OK** once more to close the **Messages** dialog.

**Note**
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 41 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.

![Slab Strip Properties dialog](image)
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.

   ![Diagram](image)

   **Note**
   
   We specified the bar sizes which Orion can use in slab design earlier (see “Defining Materials” on page 18.) You can also control the slab bar spacing range that will be applied by picking **Settings / Graphical Editor Settings / Slab Reinforcement 1**.

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.

---

1. This ensures that the strip you create is completely horizontal.
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 **Calculate** to see the report below:

![Slab Analysis and Design Report](image)

- 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.

1. Click the Section icon or pick Member/Section. We will now create a horizontal 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.

**Slab Openings**
Again refer to the more comprehensive model in the Training Notes for an example of applying slab openings, 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:

   **Note** 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, pan or zoom 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 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 4th storey, and any changes we make to the 4th storey will also apply to the 3rd, 2nd and 1st storeys.

Tip — You double-click any floor in the tree view to switch to that floor.
Building Analysis and Design

The building can now be analysed.

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

### Pre-Analysis Settings

The following function buttons are available from here:

- **Parameters** allows you to review and/or modify any building parameters you may have defined.
- **Edit Load 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.
*Edit Storey Loads* 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.

The various analysis parameters 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**.

![Building Model Check dialog box](image)

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

3. Choose **All Storeys** and then click **Check**.

![Building Model Check results](image)

Model Check Completed: No of Errors: 0
Even if this process doesn’t find any errors, it doesn’t guarantee that the building is modelled correctly. There may be other problems in the model that are not picked up by the validity checking process.

4. Assuming that there are no errors click Close 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. (At this stage there is no need 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.

An example axial load comparison report is given below.

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

<table>
<thead>
<tr>
<th>Storey</th>
<th>Column (kN)</th>
<th>Wall (kN)</th>
<th>Beam (kN)</th>
<th>Slab (kN)</th>
<th>Total (kN)</th>
</tr>
</thead>
<tbody>
<tr>
<td>S04</td>
<td>69.5</td>
<td>257.4</td>
<td>287.7</td>
<td>1142.8</td>
<td>1797.4</td>
</tr>
<tr>
<td>S05</td>
<td>49.5</td>
<td>257.4</td>
<td>287.7</td>
<td>1142.8</td>
<td>1797.4</td>
</tr>
<tr>
<td>Total</td>
<td>119.0</td>
<td>514.8</td>
<td>575.4</td>
<td>2285.6</td>
<td>3393.0</td>
</tr>
</tbody>
</table>

**SUM OF APPLIED LOADS (After Decomposing Slab Loads):**

<table>
<thead>
<tr>
<th>Storey</th>
<th>Column (kN)</th>
<th>Wall (kN)</th>
<th>Beam (kN)</th>
<th>Slab (kN)</th>
<th>Total (kN)</th>
</tr>
</thead>
<tbody>
<tr>
<td>S04</td>
<td>69.5</td>
<td>257.4</td>
<td>1430.5</td>
<td>9.6</td>
<td>1797.4</td>
</tr>
<tr>
<td>S05</td>
<td>49.5</td>
<td>257.4</td>
<td>1430.5</td>
<td>9.6</td>
<td>1797.4</td>
</tr>
<tr>
<td>Total</td>
<td>119.0</td>
<td>514.8</td>
<td>2861.0</td>
<td>9.6</td>
<td>3393.0</td>
</tr>
</tbody>
</table>

**BUILDING ANALYSIS COLUMN/WALL AXIAL LOADS:**

<table>
<thead>
<tr>
<th>Storey</th>
<th>S Delta 6 (kN)</th>
<th>0 Delta 6 (kN)</th>
</tr>
</thead>
<tbody>
<tr>
<td>S01</td>
<td>1797.4</td>
<td>821.3</td>
</tr>
<tr>
<td>S02</td>
<td>2474.0</td>
<td>821.3</td>
</tr>
<tr>
<td>S03</td>
<td>3215.5</td>
<td>821.3</td>
</tr>
<tr>
<td>S04</td>
<td>4949.5</td>
<td>821.3</td>
</tr>
<tr>
<td>Total</td>
<td>6949.5</td>
<td>3285.1</td>
</tr>
</tbody>
</table>

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.

Note: 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 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 **View Settings** menu is shown on the left of the window. This can be used to control the appearance of the display. The most commonly used settings can also be accessed from the icons and drop down menus at the top of the window.
3. Click the Filters icon from 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. Click the Displacements icon to turn off the display of the deflected shape.

6. Click the Frame Element Results Diagrams icon to select it and then choose the $M_3$ as the required effect to display from the drop down menu to the right of the icon as shown below.
7. Uncheck **Diagrams/Automatic Scale** from the View Settings menu to activate the **Display Scale**. Experiment with the scale to obtain a plot similar to that shown below:

8. Experiment with the filters and effects to create other plots and then close the **Analysis Model and Results** window.

9. Close the **Building Analysis** dialog.
Column Design

1. Pick Run / Column Section Design.

Note
The designs you obtain are very dependant on the current Settings. You can adjust these to suit your own preferences through the Settings menu. Hence don’t be surprised if your results differ from those shown below. For example in the results below the minimum column bar size has been set to H12.

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.

3. Close the report.

You can also generate a column schedule quickly.
4. Pick **File - Column Schedule**. Highlight several of the columns as shown, and then click **Draw**.

![Column Schedule Diagram]

**Note**: You may see a dialog telling you the size of sheet you require and the drawing scale. If you do, then simply click **OK** to continue.
A schedule drawing is produced as follows:

5. 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.

**Note**
The designs you obtain are very dependent on the current Settings. You can adjust these to suit your own preferences through the Settings menu. Hence don’t be surprised if your results differ from those shown below.

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.

Those beams that have been successfully designed can be placed on to a drawing sheet automatically.
2. Pick *Sheet - Beam Detail Drawings of All Axes (Single Sheet)* and check *Storey-1*

![Detail Drawing Arrangement](image)

3. Set the **Number of Columns in the Sheet** to 1 and click **OK** to create the drawing.

4. Close the drawing.
5. Now to create a report pick **File / Beam Reinforcement Design Report**.

6. Click **Save**, and then **Close** the report and **Close** the **Beam Design** window.
Design Status
You can graphically display the design status of all members on a particular floor.

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.

Note: The design status of each member is also indicated in the Structure tree view.
You can extract quantity reports in various formats.

1. Pick *File / Quantity Extraction Tables*.

Shown below is an extract from the Column/Wall steel bar quantity report.
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.

There is much more help and information provided with Orion.
- The Orion Training Notes – complete notes for introductory Orion training (particularly recommended).
- The Engineer’s Handbook – accessed from Orion’s Help menu. This provides a great deal of information on topics such as Finite Element analysis.

Alternatively, we strongly recommend that you attend one of our Orion training courses¹. Our expert, highly acclaimed, tuition will give you a flying start.

¹ Contact our Support Department for details.