BUILDING DESIGNER
CSC Fastrak™
Structural steelwork analysis and design
www.cscworld.com/fastrak
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.

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

Trademarks

Fastrak5950™ is a trademark of CSC (UK) Limited. Orion™ 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.
<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Thank you</td>
<td>5</td>
</tr>
<tr>
<td>First steps</td>
<td>5</td>
</tr>
<tr>
<td>Introduction</td>
<td>6</td>
</tr>
<tr>
<td>Design Information</td>
<td>8</td>
</tr>
<tr>
<td>Getting Started — the First Steps</td>
<td>9</td>
</tr>
<tr>
<td>Create Grid Lines</td>
<td>11</td>
</tr>
<tr>
<td>Understand Attributes</td>
<td>17</td>
</tr>
<tr>
<td>Review Column Attributes</td>
<td>18</td>
</tr>
<tr>
<td>Place Columns</td>
<td>20</td>
</tr>
<tr>
<td>Create Beam Attributes</td>
<td>29</td>
</tr>
<tr>
<td>Open the view of a floor</td>
<td>30</td>
</tr>
<tr>
<td>Place Beams</td>
<td>32</td>
</tr>
<tr>
<td>Create Curved Grid</td>
<td>40</td>
</tr>
<tr>
<td>Place Slabs</td>
<td>45</td>
</tr>
<tr>
<td>Validate Design Model</td>
<td>48</td>
</tr>
<tr>
<td>Change to Composite Beams</td>
<td>54</td>
</tr>
<tr>
<td>Create Loadcases</td>
<td>54</td>
</tr>
<tr>
<td>Create Combinations</td>
<td>57</td>
</tr>
<tr>
<td>Create Other Floors</td>
<td>59</td>
</tr>
<tr>
<td>Define Bracing</td>
<td>60</td>
</tr>
<tr>
<td>Gravity Sizing</td>
<td>62</td>
</tr>
<tr>
<td>Modify an Element Design</td>
<td>65</td>
</tr>
<tr>
<td>Lateral Sizing</td>
<td>68</td>
</tr>
<tr>
<td>Full Design</td>
<td>69</td>
</tr>
<tr>
<td>Create a Report</td>
<td>69</td>
</tr>
<tr>
<td>Modify the Model</td>
<td>71</td>
</tr>
<tr>
<td>What Next?</td>
<td>73</td>
</tr>
<tr>
<td>CSC Offices Worldwide</td>
<td>74</td>
</tr>
<tr>
<td>The Quick Start Guide continues</td>
<td>75</td>
</tr>
<tr>
<td>How's the structure working?</td>
<td>78</td>
</tr>
<tr>
<td>Add bracing</td>
<td>82</td>
</tr>
<tr>
<td>A cross-braced system</td>
<td>83</td>
</tr>
<tr>
<td>An inverted V-brace system</td>
<td>86</td>
</tr>
<tr>
<td>A partially braced solution</td>
<td>88</td>
</tr>
<tr>
<td>A rigid frame solution</td>
<td>89</td>
</tr>
<tr>
<td>Defining a sub-structure</td>
<td>92</td>
</tr>
<tr>
<td>Define roof trusses</td>
<td>100</td>
</tr>
<tr>
<td>Add roof plane</td>
<td>109</td>
</tr>
<tr>
<td>Define wind load</td>
<td>112</td>
</tr>
<tr>
<td>Add Dimensions</td>
<td>119</td>
</tr>
<tr>
<td>Controlling Composite Beam Design</td>
<td>122</td>
</tr>
<tr>
<td>Section</td>
<td>Page</td>
</tr>
<tr>
<td>--------------------------------------------</td>
<td>------</td>
</tr>
<tr>
<td>Make every beam composite</td>
<td>123</td>
</tr>
<tr>
<td>Review of Initial Design</td>
<td>131</td>
</tr>
<tr>
<td>Typical Beams</td>
<td>135</td>
</tr>
<tr>
<td>Decking Spans and Fixing</td>
<td>146</td>
</tr>
<tr>
<td>Other information</td>
<td>154</td>
</tr>
</tbody>
</table>
1  Thank you

Thank you for purchasing this copy of Building Designer. We know that you have made the best choice, 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 Guide is all about. You should be able to create, analyse and review the results for the simple model in this Guide in 1 - 2 hours.

After that, more realistic models may simply be a matter of repetition. However, to help you move from being a Building Designer “novice” to a Building Designer “expert” we would strongly recommend that you book on to the next Building Designer training course.

In addition, to properly understand Building Designer’s Scope and Limitations we would direct you to the Engineers Handbooks within the Help System.

Remember, we will always welcome your comments and ideas about Building Designer. Your input is very important to us. It enables us to ensure that our continuing development of Building Designer meets your requirements. We are totally committed to maintaining and updating Building Designer and your contributions are vital to this.

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

The Fastrak team

2  First steps

If you have not yet installed Building Designer, then insert the CD into your computer’s CD/DVD drive, and follow the on-screen installation instructions.

1/  “Click the Start button on the Windows task bar, then in turn click:
   • All Programs; CSC; Fastrak; Building Designer
   Building Designer starts and you will see a dialog relating to licensing. If you want to licence Building Designer at this stage then do so. The Licensing help explains this process fully.

   If you are using a local licence, then you do not have to wait before you can use Building Designer. If you are using a network licence, then you will have to wait until you receive your licence files before you can use Building Designer.

2/  Click Close to shut the Licensing dialog; Building Designer will start and initialise.
3 Introduction

The aim of this document is to enable you to become productive with Building Designer as quickly as possible.

After installation, if you have a project open, the screen will appear as shown above. You can customise this at will, placing and docking windows and toolbars wherever you prefer.

You can access all Building Designer’s commands either from the main menu pads, or by using the toolbar icons on the screen.

If you hover over an icon, then you will see a tool tip which indicates that icon’s operation.

You can create your model using either 3D- or 2D-windows, all commands work in both.

The Properties window allows you to view and modify the properties of single- or multiple-elements quickly and easily.

---

4. You get a grace period from now (the first time the Building Designer Licensing dialog appears) until midnight tomorrow during which you can use Building Designer without licensing it. After midnight tomorrow Building Designer will not launch until you have licensed it.
Tutorial Building

The tutorial building you will create following this *Quick Start Guide* is shown below. Please bear in mind that this example is not intended to be completely realistic, or to show you the best way to create a model. It simply aims to introduce you to *Building Designer* quickly.

**Isometric View**

![Isometric View of the Building](image)

**Plan View**

![Plan View of the Building](image)
4 Design Information

Design Data
The example building:
• is a 3 x 6m bay by 3 x 9m bay by 3 x 4m storey composite building which:
  • contains a semicircular floor area where the beams are of simple (non-composite) construction, and
  • has an inclined side on plan which is rotated 10° out of vertical,
  • has secondary steel beams at ⅓ points,
  • has edge beams which are of simple construction,
  • has internal primary (spine) and secondary beams of composite construction over the regular grid area,

Design Codes
• Steel Design Code; BS 5950-1: 2000,

Note Alternatively you may choose to use Eurocodes for this example, in which case some of the screen shots shown will differ.

Loading Information
• Construction Stage; S.W of Slab (Wet) = 2.5 kN/m²,
  Live Construction = 0.5 kN/m²,
• Composite Final Stage; S.W of Slab (Dry) = 2.0 kN/m²,
  Imposed Loading = 5.0 kN/m²,
  (incl. 1 kN/m² for partitions)
  Wall load on edge beams = 10.0 kN/m.

Composite Data
• Slab Depth = 150mm,
  • Concrete Grade = C30,
  • Reinforcement (slab) = Mesh A142 as necessary,
  • Decking = R150D RIBDECK AL,
  • Gauge = 0.9 mm,
  • Standard Stud Layout,
  • Stud Height = 100 mm,
  • Composite Beam steel grade = S355.

Simple Beam and Simple Column Data
• Steel grade = S355.
Chapter 5: Getting Started — the First Steps

5 Getting Started — the First Steps

New Project

In order to work on a model you need to create a new project to contain it.

1. Pick File from the menu pads and New Project… from the menu that appears1.

Note You can create a project this way, or you can click the New Project icon ( ) from the Standard toolbar. This icon is near the top left hand corner of the capture above.

You will see a dialog box2 in which you define the appropriate project details. As you can see on your screen, the title Job No. is highlighted in red3 and the OK button4 is dimmed. Building Designer requires that you define at least the Job No. before you can proceed.

1. In future we shall shorten this to ‘pick File/New Project…’
2. Shortened to dialog for the rest of this guide.
3. In general anything highlighted in red on the screen requires you to enter information or change an existing value.
4. From here on we shorten ‘the OK button’ to ‘OK’ — in this case the text would read … red and OK is dimmed……
2] Enter the project details as shown above.
3] Click OK.

You will see that:
- the main project Workbook window becomes active,
- the toolbar icons become active,
- a 2D plan view of the base is created (indicated in the top left hand corner of the Workbook window).
- The Workspace and Properties windows activate.

Before you can place any structural element in Building Designer you must first create the intersection points to which that element connects. You will normally define these by creating construction levels (which may or may not be floors), and then defining intersecting grid lines on these levels. The intersections of the grid lines are the points you require.

In many buildings the layout of the floors is largely consistent between different floors. When this is the case we recommend that you model a single floor in the building initially and then validate your model to check that no issues arise. Once you have resolved any issues, then you can use that floor as a template for the other floors to which it is similar.

You will use this approach in your structure.
6 Create Grid Lines

As we mentioned earlier, Building Designer requires nodes between which to place structural elements. You can define these on any level, however grid lines (and the intersections that they create) on the Base construction level are automatically shared\(^1\) with all other levels. Therefore you will create the grid lines for this model in the base view.

You could create the grid lines which the model needs singly, however Building Designer has two wizards which allow you to define entire grid systems quickly and easily. The two types of grid that these wizards allow you to create are:

- **Rectangular / Parallelogram** shaped grids, and
- **Radial grids** — Circular shaped, either faceted or smooth.

\*Note\* Wherever we show a toolbar, we show it in its un-docked state for clarity. If you have maintained the as installed settings the toolbars will all be docked to various points in the Building Designer window. These are shown below.

---

\(^1\) Are automatically duplicated on.
You are already working in the 2D Base Workbook so click the Rectangular Grid Wizard icon to create the rectangular grid system.

The first page of the Wizard asks you to set the origin of the grid. You will notice that the mouse pointer changes to display the coordinates of the pointer on the screen (these are respective to the origin of your model).

Building Designer lets you enter most information either graphically (by clicking in the Workbook window) or numerically (by entering the relevant information in a dialog box or directly into the Property window). 1

Note

To help you visualise the final result the current grid arrangement is shown graphically (as a shadow) in the Base Workbook.

The first page of the Wizard allows you to define the origin of the grid. You want to place this at your model's origin. Look at the Wizard and you will see that this is the default.

Click Next > to accept the default values.

1. The functionality is always appropriate to the current operation.
Chapter 6: Create Grid Lines

The next page of the Wizard asks you to choose which grid lines you want to create, and the line style that you want to use for them.

3] Click Next > to accept the default settings as shown above. The next page of the Wizard allows you to define the grid lines in the x-direction.

As well as controlling the placement of the x-grid lines you can also control their lettering or numbering.

4] Specify 3 bays of 9 (metres) as shown above and then click Next >.

If you pick the Irregular option, Building Designer allows you to enter a run of numbers which define the distances between successive grid lines.

**Example**

If you enter 6, 2, 4, 5 then Building Designer will create grid lines at 6, 8, 12 and 17 metres from the origin you set in the first page of the Wizard. You can also create runs of grid lines at the same spacing using the construct 3x6 (or 3*6) which would generate lines at 6, 12 and 18 metres.

5] Click Irregular.

6] Enter 1, 3x6 as shown below and then click Next >.

You have now defined the details for a rectilinear grid pattern in the x- and y-dimensions. In the following pages the Wizard allow you to rotate the grid about its origin or to change the angle between the x- and y-grid lines. You might want to...
investigate the effects of these options (as shown by the shadowing on the screen). For this example you want a rectilinear grid whose rotation is $0^\circ$ and whose grid lines are at an angle of $90^\circ$. These are the default settings in the two Wizard pages, ensure that you make these settings as you continue.

7] Click Next > to accept the default rotation of $0^\circ$.
You will see that you are now at the last page of the Wizard as Next > has changed to Finish.

8] Click Finish to accept the default axis angle of $90^\circ$, complete the Wizard and generate the grid system shown below.
Add an Extra Grid Line

You can now place additional grid lines anywhere in the Workbook window, and in any direction. You are not confined by the boundaries of the rectilinear grid pattern you have just created, or the points at which the grid lines cross each other.

You now need to create the node from which you will set out the radial grid system. To do this you will add a new grid line which is parallel to line C, 4.5 m from it and between it and line D.

1] Pick Grid/Grid Line Parallel, or pick the Grid Line Parallel icon from the Grids toolbar.

You will see the Add Parallel Grid Lines dialog.

As you can see (on your screen) the Base Line text is shown in red. This indicates that Building Designer has insufficient information to proceed. Create is dimmed (disabled) and thus you are unable to continue and generate the new grid line.

To proceed you must give Building Designer the information that it requires – the reference line from which the distance to the new parallel grid line is measured.

2] Either click on grid line C in the Base workbook window, or pick C from the Base Line drop list of available grid lines. You will see that other areas of the dialog become active.

3] Either enter the Distance, 4.5 (in m), in the dialog, or move the mouse pointer until you have set the position of the parallel grid line graphically, (the pointer tooltip tells you the exact distance from grid line D).

4] Now Building Designer has sufficient information to define the new grid line so click Create (in the dialog) to do so.

**Note**
If you set the position of the grid line graphically, then a double-click will automatically create it.
You should now have this grid arrangement.

Customise Grid Lines

If you look at your structure you will see that the grid line you created using the Parallel Grid Line option has been given the label 10 – it was the model's 10th grid line. You only created this grid line to help you create the radial grid system, so it would be good to hide the grid line's reference. To alter the details for an element in Building Designer you must first select it.

1/ Pick Select from the Select toolbar and Grid Line from the Objects one.

2/ Pick grid line 10 from the Workbook window (the selected grid line turns blue).

3/ Now look at the Properties window and you will see the selected grid line's details.

---

1. At least if no one has changed the pre-defined colours in Building Designer's Preferences.
You can change the grid line's details (Properties) at will. Changing them will affect the view of the grid line in any window in which it is visible.

4 | Click the right-hand side of the Label view line.
5 | Pick None from the drop-list that appears,
6 | When you have made the appropriate settings click the Clear Selection icon ( ● ) from the Select toolbar to remove the selection of the grid line.

Note When no elements of the current type are selected the Clear Selection icon is dimmed.

7 Understand Attributes

When you design by hand you will be aware of preferences and limitations that apply to the design of each member and you will apply these naturally. (For example, steel grades, maximum beam depth, and such like.) Building Designer needs you to tell it about all these attributes so that it can work to the same limitations/preferences as you.

Things like maximum beam depth will apply to all beams in a floor so you do not want to have to provide that information lots of times. Attribute settings are nothing more than an efficient way for you to store and apply several sets of attributes. The basic principles of attributes are:

1. For each member type Building Designer creates an initial set of attributes. You can edit these if you want.

2. There is always a default set of attributes that Building Designer uses as you create new members.

3. You can create as many additional sets of attributes as you want.

4. You can then swap and change between sets of attributes telling Building Designer which set – the Default set – you want to use for new members.

5. When you change an attribute set this never changes the properties of members that have been created using it. There are ways to achieve this, as we will show you later.

With the above in mind you can proceed with member creation and as you do so the principles of attributes will become completely clear.
8 Review Column Attributes

You can access the attributes from the Workspace, or via the menus as indicated below.

1] In the attributes list in the Workspace click the plus sign next to the Columns entry and you will see the default attribute set that Building Designer has created.

2] Double-click the Columns text and Building Designer shows a dialog which lists the current column attribute sets.

You can add a new set to the list, access a set to change its details, or delete a set which you no longer require using this dialog.

3] Column Attr 1 is highlighted so click Edit… and change its title as shown below.

4] Click the Design tab and you will see the Design page.

This page allows you to control the design process including:
• whether Building Designer will determine adequate section sizes for you, or will check specific section sizes which you set,
• the type of column construction, and
Chapter 8 : Review Column Attributes

5) Check Gravity Only Design as shown above, so that the members with this attribute set are not designed for any lateral load combinations.

6) Click Design Properties…

Review the options you can set, but do not change them.

7) Click OK.

Note: You can set individual Design Properties for each and every structural element in your model, in order to control its design uniquely.

8) Click the Alignment tab, take note of the details but don’t change them.

9) Take a moment or two to investigate the other tabs of the Attribute Set - Column dialog. Again leave the settings unchanged.

10) Click OK to close the Attribute Set - Column dialog.

11) Click Copy… and define a new attribute set titled **Column at 90 deg**.

12) On the Alignment page set the Angle to **90** (degrees).

13) Click OK to close the Attribute Set - Column dialog.

14) Click OK again to close the Attribute Sets - Columns dialog.

You will see the name of the first attribute set change in the Workspace window, and that Building Designer creates a line for the new attribute set you created.

Note: The current default attribute set (shown in bold text) is that for the columns at 0°.
Now to create the columns in your model. You need to follow a particular (simple) procedure common to all element types in order to do this:

- **First** — from the **Edit** toolbar pick the operation you want to perform.
- **Second** — from the **Objects** toolbar pick the type of element with which you want to work.
- **Third** — perform the operation on the element type.

1/ Pick **Create** from the **Edit** toolbar and **Columns** from the **Objects** one.

Look on your screen and you will see a small square at each grid intersection point. This indicates that there is an equivalent grid intersection point on a different level, and that **Building Designer** can therefore create a column at this point. This second point is on the **Roof** level which **Building Designer** creates automatically for new models. This is initially set to a level of 3 m, but you can change this at will and insert and add other floors as necessary, which you will do later.

2/ With the left mouse button click and drag over the entire grid area in the **Workbook** window (as shown below).
Release the mouse button, and Building Designer creates columns at every grid intersection point as shown below.

These columns have their webs at 0° – the setting in the current attribute set.
Delete columns

Review the plan for the model (see page 7), and compare this to the capture above (see page 21), you will see that you have generated columns at grid intersection points where they are not required. You need to delete the surplus columns (shown below).

1] Pick Delete from the Edit toolbar and Columns from the Objects one.

2] Click the column at grid intersection D5 and Building Designer deletes it immediately.

3] Repeat the process to delete the column at grid intersections D4.

4] Position the pointer just below and to the left of grid intersection A2.
5] Click and hold the left mouse button.

6] Drag the pointer till it is just above and to the right of grid intersection D2.

7] Release the mouse button.

Building Designer asks you to confirm that you really do want to delete the 5 columns.

8] Click OK.

9] Delete the columns on grid line 10 (the grid line between grid lines C and D).

Your floor plan now looks like this.

Note: You can delete any element type in a similar way, simply pick Delete from the Edit toolbar, and the Type of element from the Objects toolbar and start deleting.
The View Options Window

When you have several elements shown on the screen you may find that the model display becomes cluttered and difficult to use (particularly if you switch on the view of their textural-labels and -details). *Building Designer* allows you to configure exactly what is, and is not, displayed in the current Workbook window.

1/ Pick View Options from the Navigate toolbar.

2/ In the View Options dialog, click the Structure tab and switch off the display of Supports (remove the tick in the Support box).

You should see the view below.

3/ Pick View Options again to close the View Options dialog.
Edit an Attribute Set

When you created the columns you told Building Designer to use Column at 0 deg set. You actually want your columns to be such that their webs are parallel to the x-axis – a rotation of 90°, so what happens if you change the set?

1/ Double-click the name of the Column at 0 deg attribute set (that you defined earlier) from the Workspace.
2/ Edit the set and change the Alignment Angle to 90°.
3/ Click OK to get back to the Workbook window.

*This does not change your existing columns.* Their properties are still based on the attributes that were in force when you created them. It is important to understand and remember that the only link between the attributes and column properties is:

* the copy of the former to the latter when you create them, or
* when you tell Building Designer to apply the properties of the selected columns from an attribute set.

4/ As you already have an attribute set which has a 90° rotation set you can use this set to update your columns, so repeat steps 1 and 2 above, but this time change the rotation of the columns in the attribute set back to 0°.

Editing the Properties of a Selection

1/ Pick Select from the Select toolbar and Column from the Objects one.

2/ Select the column which lies at the intersection of grid lines 3 and 8.
3/ Now look at the Properties window, and you will see this column’s details.

4/ In the Properties window, set the Rotation of the column to be 90° (click the right-hand-side of the Rotation line, type the new value and press Enter).
You will immediately see the column rotated in the *Workbook* window.

6] **Now click the Clear Selection icon to clear the selection of the single column**.

6] **Window all the columns to select them.**

The *Properties* window will now look similar to that below.

You will see that some information has been removed, namely the *Reference* and *Rotation* of the columns. This happens because the *Properties* window is showing the details for all 14 selected columns, and the blanked details are inconsistent for some or all of these.

1. If you don’t clear the current selection first *Building Designer* will deselect column B3 as it selects the other columns. *Selection simply toggles the selection state of the elements involved in the selection process.*
7] In the Properties window enter the Rotation of the columns as 45° (don’t forget to press ENTER).
You will see that all the columns rotate to this angle. Since the columns all have the same rotation, the Properties window now shows the angle again. The Reference details for the columns remain empty, since Building Designer maintains the disparate settings.

**Note**
The Properties window only shows a subset of the details which are available for a particular element type. If you want to change details other than these, then you should impose an attribute set which contains the amendments which is what you will do now.

### Apply an attribute set to a selection

1] In the Workspace, right-click over the title of the Column at 90 deg attribute set.
2] Pick Set as default from the context menu.
You will see that the attribute set's title is now shown in **bold** text which indicates that it is the one in use.

3] Now click the Clear Selection icon to clear the selection of the columns.
4] Pick Apply Attribute Set from the Edit toolbar.

5] Window all the columns to include them in the Apply Attribute Set operation.
Building Designer asks you to confirm that you want to apply these attributes to 14 columns.

6] Click OK to do so.
Building Designer applies the attributes from the set Column at 90 deg to all the selected columns. You will see the change reflected in the Workbook and Properties windows.

Individual Element Properties
So far you have looked at attribute sets and the subset of these shown in the Properties window. We have only mentioned in passing the properties of an individual element. To address this you will look at the properties of a column now.

1. Right-click the column at intersection B3 from the Workbook, Pick Edit SSC B/3 from the context menu and you will see its Properties dialog.

Any changes you make in this dialog are automatically saved in your model.

If you look on your screen you will see that the Restraints tab is highlighted in red, which indicates that there is a situation which needs your attention.

1. SSC B/3 is the reference of the column. In future we shall omit the reference of members, and simply say "Pick Edit from the context menu", "Pick Design from the context menu" and so on.
2/ Click the **Restraints** tab to see what the problem might be.

![Restraints tab](image)

You can see that the problem arises because the top of the column is currently unrestrained. This is not surprising, because you have not yet created any beams in your model.

3/ Click **Cancel**.

### 10 Create Beam Attributes

Your model contains two types of beam – Composite and Simple. You will define the attribute sets for these now.

1/ To access the beam attributes either double-click the **Beams** title in the **Attributes** part of the **Workspace**, or pick **Attributes / Beams**… from the menu.

**Note** As with columns a pre-defined attribute set has been created automatically.

2/ The line for **Beam Attr 1** is highlighted, click **Edit**… and define the details below.

**Note** Where there are no details for a page, leave that page as it is.

#### Composite Beams

- **General**
  - Attribute Set Title: Composite Beams

- **Design**
  - Automatic Design: Ticked
  - Gravity Only Design: Ticked
  - Construction Type: Composite

**Note** When you change the **Construction Type** of the beam, you will see that new pages are added to the dialog. These are the pages which are only relevant to composite beams.

- **Size**
  - Grade: S355

- **Reinforcement**
  - Type: High yield steel, H
  - Autoselect: Ticked

- **Studs - Strength**
  - 1 Stud, 2 Studs: Ticked
  - Distance \( e \): 50 mm

- **Connectors Layout**
  - Auto-layout: Ticked, (Uniform)

---

1. The distance for your studs is the distance measured from the centre line of the stud to the nearest point on the profiled decking (the bottom of the trough for a trapezoidal profile and the top of the trough for a re-entrant one). This distance controls the interaction between the decking, the stud, and the concrete. For more information email support@cscworld.com.
11 Open the view of a floor

Now that you have defined the attribute sets that you need you can continue and place the beams which use these into the model.

Remember that you are initially creating a single floor and using that floor as a template for the other the floors in the model.

Look at the current Workbook, you will see that you are currently working on the Base (shown in the top-left-hand corner of the Workbook window). Obviously you don’t want to define beams at this level, you will add these to the Roof. You must therefore open the workbook for the Roof.

Open a different view

To open a new Workbook window you need to follow the procedure detailed below.

• First — in the Workspace click the plus sign next to Structure (if there is a minus sign by Structure, then omit this step).
• Second — click the plus sign next to Construction Levels (again if there is a minus sign by Construction Levels, then omit this step), this shows all the construction levels which you have created in your model.
• Third — double-click the construction level’s Name in the Workspace and Building Designer will open a 2D view of that floor.
Note

If you have a view open, but that view is obscured by other windows, then double-clicking its name in the Workspace will bring that window to the top of all other windows, maintaining the current view.

You can also obtain a 3D view of your entire structure, and a 3D view of a floor using a similar procedure:

- to obtain a 3D view of your entire structure double-click the text *Structure* (right at the top of the *Workspace*),
- to obtain a 3D view of a construction level right-click that construction level's *Name* in the *Workspace* and then click *Open 3D View* from the menu that appears.

You can work with frame workbooks in a similar manner, all frame views are in 3D, but these are shown using an orthographic view by default. You can customise this through *Preferences*.

- to obtain a 3D view of a frame double-click the frame’s name in the *Workspace*. If the isometric icons in the *View* toolbar are not enabled, then orthographic view is set. In this case access the *Grid* page of the *View Options* dialog, remove the tick against *Orthographic view* and click the isometric icons for the view you require.

1/ **Open up the Workspace until you can see the construction levels.**

2/ **Double-click the text Roof in the Workspace.**

*Building Designer* opens the roof’s *Workbook* with the default *Preferences* settings.
32 : Chapter 12 : Place Beams

8] Use the View Options dialog to switch on the display of the beam names.
The initial floor plan you are going to create is shown below.

You will create all the beams in your model initially as simple beams (note the prefix SB in the capture above). You can then check that your model is valid without introducing any complications due to composite construction (such as the necessity to create a slab over composite beams).

4] In the Workspace, click the plus sign to the left of the Beams title in the Attributes part of the tree.

5] Right-click over the Simple Beams attribute set which you have just created.

You will see that the attribute name is emboldened. This is the attribute set which the new beams you place will use as their properties' source. Now you can start to place your beams.

12 Place Beams

7] Pick Create from the Edit toolbar and Beam from the Objects one.

As you used a window to define the positions of the columns you can also use a similar technique to define beams which run along grid lines and between columns. For beams which do not meet this requirement, you select the start- and end-point of the beam.
These points can be grid line intersections, column positions, beam intersections, specific points along the beam or specific distances along it. You will create your first beams using the area method.

8] With the left mouse button click and drag over the entire grid area in the Workbook window (as shown below).

9] Release the mouse button, and Building Designer creates beams between all the columns as shown below.
You could start to create the infill beams between the beams that you have just created, however Building Designer also allows you to identify one or more elements and then copy and/or rotate these to other areas in your structure where they are required. You can do this in any Building Designer window.

You will use this feature to create the four rows of infill beams.

10] **Pick Building/Copy Elements** and you will see the Source Elements dialog.

You will see that the pointer tooltip is prompting you to **Select an element or truss**¹.

There are several methods of picking the elements you want to copy:

* you can click single elements (or trusses), or
* you can drag a window around the elements you want to copy. (You can’t use this option here, since you would also select the columns along grid line A.

You will use the first option now.

11] **Click over the beam that runs between grid intersections A1 and A3** (the beam will turn green to show that it is selected and its reference will appear in the Source Elements dialog).

12] **Click the beam between A3 and A4** (again it will go green and its reference will be added to the list in the Source Elements dialog).

13] **Repeat this for the beam between A4 and A5 and for the three beams that lie along line B.** The source elements dialog will now show the references of the six selected beams. These six beams will be included in the copy process.

¹ If you have defined trusses, then you can pick up the entire truss as a single entity by clicking on one of its members.
14] Now you have picked the beams click Next > to continue with the copy process. You will see the Base Point dialog.

![Base Point dialog]

This dialog allows you to identify the control point that you want to use as the origin of your copy. Your copied elements will be placed by picking equivalent points relative to this point.

15] Move the pointer over grid intersection A1 and Building Designer indicates that it has located this point by placing a small yellow square over it (note too that the tooltip shows the coordinate of the point).

16] Click over this grid point to use it as the base point for the copied elements. As soon as you do this the dialog changes to the Mirroring one.

![Mirroring dialog]

As you can see from this dialog, you can do much more than create a straightforward copy of elements in your structure. For this example we shall create simple copies of the selected elements.

17] Leave this dialog at its default setting of No Mirroring and click Next > to proceed with the copy process. You will see the Rotation dialog.

![Rotation dialog]

As well as mirroring you can also rotate the copies through any angle that is necessary.
18] For this example you don’t need to do any rotation, so click Next > to proceed to the final stage of the process.

19] This is to define the **Target Points** – the equivalent points to the base point which you selected earlier. To enable you to do this you will see the **Target Coordinates** dialog.

As with selecting elements to copy you can pick multiple targets for the copies. In this instance you need to replicate the 6 copied beams at the one third and two third points of the beams between grid intersections \( A1 \) and \( B1 \), and between \( B1 \) and \( C1 \). Since the beams are 9 m long, the copies need to be placed at 3 m and 6 m.

20] Move the pointer horizontally over the bottom-left-hand beam (the one which runs between grid intersections \( A1 \) and \( B1 \)).

21] Move the pointer horizontally along the beam until the tooltip shows the correct coordinate – 3.000, 0.000. (Note that Building Designer shows a line representation of the elements that you are copying to help you visualise what you are achieving with your copy.)

22] Click on this point and Building Designer adds this to the list in the **Target Coordinates** dialog.

23] Continue to move the pointer horizontally along the beam until the tooltip shows the correct coordinate for the next row of beams – 6.000, 0.000.

24] Click on this point and Building Designer adds this to the list in the **Target Coordinates** dialog.
To create your copies click Finish. You should now have the arrangement of beams shown below.

To define the other beams in your model you need to select their start- and end-points. These points can be grid line intersections, column positions, beam intersections, specific points along a beam or any distance along it. You will first create the remaining beams which lie between grid intersection points.

Pick Create from the Edit toolbar and Beam from the Objects one.

Click the beam start point (C5).
28) Click the beam end point.

Building Designer creates the beam between the two points you identify.

29) Repeat the above process to create the 2 further beams shown below.

Now you will use another Building Designer feature. This allows you to place new beams by trimming them into existing beams directly.
30] Click the beam which runs between grid intersections C1 and C3. You will see that boxes are added to the beam, representing the \( \frac{1}{4}, \frac{1}{3}, \frac{1}{2}, \frac{2}{3} \) and \( \frac{3}{4} \) points along the beam.

31] Click the point indicated above (the centre of the beam).

32] Use a similar process to specify the location of the other end of the beam.

33] Use a similar process to create the two additional beams shown below which are at the \( \frac{1}{3} \) and \( \frac{2}{3} \) points.

Building Designer also allows you to trim a beam into an existing beam at a specific distance along that beam.

34] Click the beam which runs between grid intersections C3 and D3. You will see the standard points along that beam.

35] Instead of clicking on one of these points start to type the distance you require (in this case 4). Building Designer pops up a dialog where you type the distance.

36] Type in the rest of the distance to place the point at 4.5 m.

37] Click OK.

38] For the end point of the beam click the grid intersection on grid line 4 vertically above the start point (this point is on grid line 10 you switched its label off earlier).

39] Place a column at this location.
13 Create Curved Grid

To do this you will use the *Radial Grid Wizard.*

Rather than returning to the *Base* level, you will create the radial grid at the *Roof* level. Remember that while grid lines created at the *Base* level are automatically shared with all other levels, those created on other levels are not. You need the radial grid system on both levels since you need to place columns within the grid which run between the *Base* and *Roof.*

Buildings Designer has an option which shares any new lines that it creates.

1/ **Click the Share New Grids icon.**

Any grid lines that you create while this icon is active (has a box around it) are shared across all construction levels.

2/ **Click the Radial Grid Wizard icon** to add a new radial.

3/ **Pick the point at which you have just created the column as the origin for the radial grid.**

Its coordinates are 22.500, 13.000 – these are shown in the tooltip on the screen.
4] Choose the option to create all lines, set the line representation asDash and then click Next >.

Now you need to define the dimensions (extent) of the grid system.

You must define the radius at which you want to place the circular grid lines. For this example you only need to create a single line at a radius of 9 m (7.5 m to the external intersecting column positions and a 1.5 m cantilever).

Grid lines A to D already exist, so set the start for the numbering of this grid as E.

Tick Use Arcs to make Building Designer draw the radial line as true arcs (otherwise it will create a series of straight lines making up the facets of the circle).

The details for the Extent page with these settings are shown below.

5] Make these settings and then click Next >

Now you need to define the extents of the grid in the other direction - the number of segments (pie slices) that you want to create.

6] You want to create six slices of 30° each, and this time you want to start the numbering of the grid system with 6.

7] Define these details and click Next >

8] Finally you specify the rotation of the grid system. You can either enter a value directly or click a point in the workbook. In this case click the existing grid intersection D3 (note that the angle of rotation is -53.1301°).

Tip

When you are working with angles you may need to increase the precision, since slight rounding of an angle can have significant impact on your model. You can change the precision in Preferences which you access from the File menu. We suggest that the minimum precision for angles should be 4 decimal places.
The new grid is created, and the *Workbook* window automatically zooms so that you can see the grid in its entirety.

9] Check that the new grid lines have been duplicated in the *Base* workbook, but return to the *Roof Workbook* before you proceed.

The last column that you created needs to lie with its web perpendicular to the two diagonal beams which trim into it. This means that its web needs to be parallel to grid line 9. You could calculate the appropriate angle, and apply this using the column’s properties, however *Building Designer* allows you to achieve this graphically.

10] Pick *Modify* from the *Select* toolbar and *Column* from the *Objects* one.

11] Click the column whose rotation you want to modify. *Building Designer* adds a handle to the column and changes the tooltip to read *Move Point*. You don’t want to change the location of the column, simply to change its rotation. Move the
pointer away from the column and Building Designer shows a line which is parallel to the column’s web. Move the pointer over this line and you will see that the tooltip changes to Change Rotation.

12] Click this line, and then move the pointer and click over grid line 9 (note that the tooltip shows the rotation of 306.8699°), and the column rotates so that its web lies along this grid line.

13] Double right-click to end the column rotation process.

14] Place and rotate the new columns shown below (remember to click on a grid line, and not over the central node when you are rotating the columns).

15] Continue and place the beams (shown below) which lie between grid intersection points. Start at grid intersection C5 and continue clockwise to intersection D3 (the reason for noting the direction will become apparent later).
16] Place the **Diagonal** beam annotated below.

You will now use a new technique to define the beams which are orthogonal to the diagonal beam (annotated above).

17] Click on grid intersection E10.

18] Click the **Diagonal** beam.

You will see the standard snap points on the beam, but there is also an extra snap point which is that where the incoming beam is exactly perpendicular to this one.
Click on this point to create the perpendicular beam.

Create the other beams as shown below.

Create an attribute set for the first slab in your building, taking the details from the Composite Data details (see page 8). Set the angle of the slab to 0° and give it the title RLSD AL 0.9 0 deg.

From the Plan View on page 7 you will see that your building has 3 separate slab areas spanning in different directions. You will create the three areas using the attribute set you created above, and then change the angles as required.

You must surround slabs by appropriate steelwork in order to get proper load distribution. If you don’t provide such steelwork, then your model will not validate.

Pick Create from the Edit toolbar and Slabs from the Objects one.

You can create slabs by:
- windowing the area covered by the slab,
- clicking zones which are completely surrounded by steel to create single areas, or
- defining the total area that the slab covers by identifying a series of points. You will use these three methods in turn now.

Zoom the view of the Roof so that you can see it in its entirety.

Set the View Options so that Building Designer shows the Text for the Slab Name.
To place the first slab area click and drag as shown below.

Your screen should now look like this:

Since the next slab area takes a different angle to that which you have just defined you must create this as a new slab, not as a continuation of the current one. Therefore pick New Slab… from the drop-list in the Objects toolbar.
27] Click in the steel bounded areas to add Slab 2 as shown below.

28] Pick New Slab… and then click on points D3, E6, E7, E8, E9, E10, E11, E12, C5, the point at the left-hand end of the Diagonal beam referred to above, and D3 again. You should now have this slab arrangement.

29] Right-click on any Slab 2 area and pick Edit from the context menu.

30] In the Slab Properties dialog set the angle of the slab to be 90°.

31] Click OK.
You will see that all areas of Slab 2 take the new angle.

32] Set the angle of Slab 3 to 36.8698°.
33] Switch the View Option to see the beam name text off.
Your slab arrangement should now be this.

34] Switch the View Option to see the beam name text on again.

15 Validate Design Model

Validation is a check of your structure which you must perform before you can design it. It checks all elements in your structure for a wide range of conditions. If any condition is not satisfied, then Building Designer tells you. Building Designer doesn't let you perform the design until you have sorted any problems.

1] Click the Validate icon from the Design toolbar.

You will immediately see the Output window. As the validation proceeds this will be populated with a list of any issues that arise.

Note
Validation can produce two types of condition – errors and warnings.
• If errors arise, and you see the error icon (×) then you can not proceed with the design until you have corrected the problem.
• If warnings arise, and you see the warning icon (⚠️) then you should take note of the condition which has caused the warning, and you must use engineering judgement as to whether or not to proceed with the design.
• You will also see that the errors and warnings are summarised on two separate pages. Review these now if you want.

You can see that there are four unsupported beams, and that because of these particular errors validation has not run to completion.

As each structural element is unique within your model, you could search in the Workbook to find the particular element which has an error, however Building Designer provides a much quicker way.

2/ Double-click the first validation error in the Output window. Building Designer immediately highlights the element associated with the error in blue, and (in case the member’s representation is small [as in this case]) points to it with a big blue arrow.
Double-click the other three validation errors, and you will see that they relate to the beam that trims into this cantilever beam, the cantilever beam that trims into point D3 and the edge beam that connects to this second cantilever beam.

The combination of the validation errors, and the location of the elements in the Workbook allows you to see the cause of the problems immediately.

Simple Construction - To this point we have been defining simple beams and simple columns which come under the heading of simple construction as defined in BS5950. In essence we have defined pin-ended beams which only induce eccentricity moments in the columns. In this example, at this location, we have something more complex than simple construction and so we have two choices:

Option 1 - It is possible to change the properties of the simple beams so that they become cantilevers and induce moments into the simple columns. The simple column design is then carried out in the normal way. The design is likely to result in warnings (most probably relating to shear forces that are not normally checked in simple column design), but it remains potentially valid. In practice it is a matter of some engineering judgement as to whether you allow the columns to remain as simple columns. Your decision would depend on the length of the cantilever, its loading, and hence the perturbing effect on the assumptions made in the code for columns in simple construction. Provided you are comfortable with this judgement this option will be the simplest to define and the fastest in design.

Option 2 - If you have large cantilevers putting significant moments into the columns you may want to change the members types to general. General Beams and General Columns can accommodate more complex loading/buckling cases, but will often require more detailed input from you in order to guide the design process.

Although probably un-necessary in this case you will change to general beams and general columns to illustrate the process and (coincidentally) to remove the validation warnings.
You will then take steps to remove the other validation warnings, to keep things as simple as possible throughout the rest of this worked example.

4] Pick Beam from the Objects toolbar.

5] Right-click the cantilever beam which trims into point C5, and pick Edit from the context menu. You will see the cantilever’s Properties.

You will see that the Construction Type of this beam is set to Simple, you could edit the supports to make this a simple cantilever, but as noted above you will change it to a general beam first.

6] Pick the General option from the drop-list.

7] Next click the Supports tab.

Remember that you created this set of beams in a clockwise direction (this is the reason for the earlier note to take heed of the direction in which the beams were created).

8] Set Support 1’s Type to Fully Fixed.

9] Set Support 2’s Type to Free.

Note
If you don’t remember the direction in which you created a member, you can find which are its start and end nodes by switching on the View Option to show the Element Direction. When you set this option Building Designer draws an arrow on the member which points from its start to its end.

10] Click the Restraints (LTB) tab.

Note the assumed (default) restraint conditions. For normal loads the effective length of the bottom flange (the compression flange) is assumed to be 1.0L. Within a framed building this will often be a reasonable or even a conservative assumed length. In this
example we will not change the default LTB or Strut buckling restraints but it is worth noting that this is something you should review when you use general beams or general columns. (It is also worth noting that when you change a simple beam to a cantilever the default assumed LTB buckling length is 3.0L. It is anticipated that when you use this option you will often do so with less attention to detail/defaults and so Building Designer defaults to the most conservative assumption.)

11] Repeat this process for the other cantilever beam, (in this case the support conditions need to be reversed – the cantilever end is at the start of the member).

Note Correcting the two cantilever beams also fixes the errors with the edge beams that connect to them.

12] Run the validation again and you will see that this time validation has completed, but that you have five errors and one warning.

Note These errors and the warning were not found before, since the previous errors in the structure stopped the validation running to completion.

You have eliminated the errors about unsupported members, but you now have new errors about the general beam cantilevers and their connections to the supporting simple columns. Clearly with the current column orientation the connection of the cantilever to the column would be extremely difficult. Two of the errors indicate that the simple column gravity only design will not be able to design for this situation, but as already discussed you are going to change these to general columns.

13] Right-click on one of the columns and pick the Edit context menu option. You will see the column’s Properties

14] Change the Construction Type to General and uncheck Gravity Only Design.
16] Click OK.
16] Repeat this for the column which supports the other cantilever.
17] Run the validation again, and you should find that the number of issues raised reduces as shown below.

![Validation Result](image)

Warnings relating to the orientation of the columns remain, if the orientation could not be changed we could take note of the potential difficulties with the detailing and fabrication of the connection, but carry on. In this example we will re-orientate the columns.

**Note**
Since validation is a speedy process we would recommend that you validate frequently as you deal with issues, so that you can deal with any other issues as they arise.

18] Rotate the columns which have warnings so that the cantilevers trim into their flanges.

19] Now you will deal with the errors relating to other simple columns. Use the Output window to determine the column which has invalid restraints.

20] Edit the **Properties** for this column.

21] You will find that the **Restraints** page shows in error, so click on its tab.

![Restraint Properties](image)

**Note**
Building Designer has found that there is no restraint to the major axis, since there is no incoming beam trimming at an angle of less than 45° to this axis. With the slab, the restraint will be adequate, so we tell Building Designer this.

**Caution**
Building Designer lets you stipulate that there is restraint at a point, but there is no restraining member. You should only use this option for modelling purposes. **If you set this option then you must ensure that, in your judgement as an engineer, you have provided the restraint that you have specified.**

22] Simply change the major axis restraint type to 5 (telling Building Designer that the column is restrained about the major axis by non modelled means).

23] Click OK.

24] Change the restraints for the other column in the same way.

25] Validate your model again.
You should find that the only validation issues that arise are: an error that you have no critical gravity combination, two warnings that the general columns should be in Moment Frames if they are to resist lateral load and a warning that you have no combinations which include NHF loads. You will address these when you load your model a little later.

16 Change to Composite Beams

Having validated your floor with simple beams, now would be a good time to effect the change to composite ones.

1. Select all the simple beams which lie totally within the area covered by Slab 1 (do not select the beams round the edge of Slab 1).
2. From the Workspace make the attribute set Composite Beams the default set.
3. Pick Attributes from the Edit toolbar.
4. Confirm that you want to apply this attribute set to 19 beams.
5. Deselect the beams.
6. Validate your model again.

You should find that a new validation issue arises: composite beams exist, but no Construction Stage Combination has been specified. You will address this now.

17 Create Loadcases

7. Pick Loading/Loadcases…, or pick the Loadcases icon from the Loading toolbar.

You will see the Loadcases dialog.

From the Design Data details (see page 8) you can see that you need 4 loadcases in addition to the self-weight - excluding slabs which Building Designer determines automatically):

* Construction stage – Slab Wet, and Construction Live,
* Composite stage – Slab Dry, and Composite Imposed.

You will create these now.
Chapter 17: Create Loadcases

8] Click Add... and change the loadcase's type to Slab Wet. An option is presented to automatically calculate the loading, leave this checked.

9] Click OK to create the loadcase.

10] Click Add... and change the loadcase's type to Slab Dry. Again leave the option to automatically calculate the loading checked and click OK to create the loadcase.

11] Click Add... and define the Construction Live loadcase's name and type Imposed.

12] For this loadcase only, untick the Include in Generated Combinations box.

13] Click OK to create it.

**Note** Don't define any loads in the loadcase yet, you will do this graphically momentarily.

14] Repeat this process to define the Composite Imposed loadcase. Ensure the Include in Generated Combinations box is ticked. Also tick the Reductions box. (You will have to set the loadcase Type to Imposed first.)

15] Click OK to create it.

16] Click Add... once more and define the Perimeter Wall loadcase's name and type Dead.

17] Click OK to create it.

You will now see all your loadcases listed in the Loadcase dialog.

18] Click OK to close that dialog.

Define Floor Loads

Now that you have defined the loadcases you can add the loads that these contain.

1] From the drop-list in the Loading toolbar pick the Construction Live loadcase.
2] Pick Create from the Edit toolbar and Floor Load from the Loads one.

3] Click anywhere over an area of your floor plan where there is a slab.

4] Enter the value of the load (0.5 kN/m²) and then click OK.
5] To create the floor loads in the Composite Imposed loadcase repeat this process, using the load value 5.0 kN/m².

Define Perimeter Loads

1] From the drop-list in the Loading toolbar pick the Perimeter Wall loadcase.
2] Pick Loading/Create Perimeter Load…

3] Enter the load value of 10 kN/m and then click OK. Building Designer creates element loads on all beams around the edge of the building.

Note: In the above captures the beam name and slab name text is switched off for clarity. You know how to remove text from the display, so in future text will be minimised to maximise clarity.
18 Create Combinations

4] Pick Loading/Combinations…, or click the Loading toolbar’s Combinations icon.

You will see the Combinations dialog

5] Click Add Constr.

You will see that the Slab Wet loadcase is already included, however you also want to add in the Construction Live loadcase.

6] Double-click the name of the Construction Live loadcase.

It moves from the left-hand list of Available Loadcases to the right-hand list of Included Loadcases. This combination is now complete1.

7] Click OK to return to the previous window.

8] Now click Generate… to create the remaining combinations.

The first page of the dialog sets up the ultimate limit state factors for the gravity combination that will be generated.

9] Click Next >

The next page of the dialog sets up the service factors.

10] Click Next >

1. Building Designer can automatically handle design for second-order effects using the ken approach. You might want to investigate the options on the Second order effects page, there is no space to consider them further at this point, but the subject is documented elsewhere.
The next page allows you to create further lateral load combinations in which notional horizontal forces have been added. In this example we add these notional horizontal forces to the combination we are currently designing for gravity. Leave all the boxes checked as shown below - this will result in an additional four lateral combinations being created.

11] Click Finish.

Six combinations should now exist as shown.

During both the Gravity and Lateral Sizing processes, only those combinations marked as Critical are considered. The member sizes chosen are subsequently checked against all active combinations during the Full Design process.

12] For the Gravity Sizing process the members should be sized for 1.4D+1.6I, therefore ensure this combination is checked as Critical (in addition to the Construction stage combination).

13] For the Lateral Sizing process it is not clear at this stage which direction is critical. A maximum of 4 lateral combinations can be marked as critical (essentially allowing for selection of one in each of the principal directions) so check all four Lateral Combinations as Critical.

14] Click on OK to close the Combinations dialog.
18] Validate your model again.

The only remaining warning is that you have two columns which have moment connections to flanges.

**Note** As you will not be using second-order two step iterative analysis you can safely ignore this warning.

19  Create Other Floors

Now you have completed your template floor, you can proceed and complete the rest of your structure.

**Note** In a real model you should consider designing the template floor at this time. This enables you to resolve any issues that might arise before they can be duplicated to other floors.

1] In the **Workspace** double-click the **Structure** text to open the **Structure** view of your model.

2] Pick **South West** from the **View** toolbar to see your model in isometric view.

3] Pick **Building/Levels…**

The **Building Levels** dialog lets you control the number of storeys in your model. **Building Designer** automatically created the pre-defined floor levels – **Base** (at a level of 0 metres) and **Roof** (at a level of 3 metres) when you created a new model.

4] Click the line for the existing **Base** level.

5] Click **Insert Above** twice to create 4 levels in all.

6] Define the **First floor** at 4m, **Second** at 8m and the **Roof** at 12m.

7] Specify the **Roof** as a **Floor** with a **Single diaphragm** as shown below.

8] Click **OK** to create the new floors.
You will see that the Roof’s level moves in the Structure window and that the other floor names are created in the Workspace.

Now to copy your template floor to these new floors.

9] Pick Building/Copy Floor…

10] Set the Source Level to Roof and tick the Copy To boxes that relate to the First and Second floors.

11] Click OK.

Building Designer copies all the details from the roof to these two floors, again you can see this in the Structure window.

12] Validate your model again.

You are warned that because new floors have been added it is necessary to recalculate the column load reductions.

13] Pick Building/Update Column Load Reductions

14] Validate your model again.

20 Define Bracing

At the moment you have defined no bracing for your structure. You will do this now. By now you should be familiar with defining members, so the instructions below will be more concise.

1] Edit the default attribute set for Braces, change the Attribute Set Title to 200x100x6 RHS and define a section size of RHS 200 x 100 x 6.0 (you will only use a single section size for all your braces).

2] In the Base - 2D workbook Select grid lines 1, 5, A, and D and then pick Building/Create Frame(s).

3] Clear the selection of the grid lines.

4] Move back up the Workspace, until you can see the Frames entry (it will now have a + sign to its left).

5] Open this entry and then double-click each frame reference in turn to open a workbook for that frame.
6/ To view the frames in isometric view as shown below use the View Options Grid page, remove the tick against Orthographic View, and click the South-West isometric view icon from the View toolbar.

7/ Pick the workbook for Frm 1, and then set the options from the toolbars which allow you to create braces. Create the bracing system shown below for Frm 1, and then repeat a similar process to create the bracing systems on Frm 5, Frm A and Frm D.
1] Validate the structure again.

You are warned that the braces are connected to columns which are set to be Gravity Only Design. The seven columns affected should be switched to consider both gravity and lateral combinations.

2] Switch to the Structure view of your model.

3] Set the options from the toolbars which allow you to select the seven simple columns with bracing attached that were referred to in the warning messages.

4] Now look at the Properties window and you will see the selected columns details.

5] Click the right-hand side of the Gravity Only Design line.

6] Pick No from the drop-list that appears,

7] Click the Clear Selection icon ( ).

8] Validate the structure again.

21 Gravity Sizing

The gravity sizing process will automatically design all the members for those gravity combinations marked 'critical'. On completion all members are set into Check Design mode.

1] Pick Perform Gravity Sizing from the Design toolbar.
The design will proceed, and you will see a progress bar charting this.

On completion of the design the **Structure** window shows the results of the design graphically and the **Workspace** switches to the **Results** tab.1

The colour coding allows you to see the status of each member in your structure instantly. As you can see some of the columns have a warning status. **Building Designer** makes it easy for you to find the cause of the warnings.

2 Right-click the column on grid intersection A3, and then pick **Design Results** from the menu that appears. **Building Designer** shows its summary results,

As you can see the warning is against the **Notes**, so click on the **Notes** tab, and you will find that the warning relates to shears which exceed specified limits. The shears are small, so in this case you choose to ignore them. If the shears were larger, then you

---

1 You can change to the other pages by clicking the tabs at the bottom of the **Workspace**.
would need to consider changing the affected columns into general columns so that the design of these forces could be considered. You should find the same warning has been applied to the other columns also.

3) Click Close to shut the Design Summary. Some of the beams also have a warning status, we shall examine one of these now.

4) Right-click on composite beam 4/A/4-4/B/4, and then pick Design Results from the menu that appears. Building Designer shows its summary results.

5) The warning is against Shear Connectors, so click on the Shear Connectors tab, and you will find that the warning relates to the degree of shear connection provided.

6) Highlight the orange warning text and you will notice that an Engineering Tip becomes available. Click on the 'light-bulb' icon to display the Engineering Tip.

The beam in question has point loads at third points so that the critical bending moment utilisation ratio occurs away from the position of maximum moment. The connector layout could be altered to resolve this; you could for example decide to:

- retain a uniform stud layout, but reduce the stud spacing.
- adopt a non-uniform layout, placing more studs at the beam ends and less in the middle.

To achieve either of the above you would simply adjust the Connectors - Layout information within the Composite Beams attribute set, and then re-apply the attributes to the affected beams.

However, for the purposes of this exercise we shall simply acknowledge the warning and continue with the example.

7) Click Close to shut the Design Summary.
22 Modify an Element Design

From the above you will see that the design of the members in your structure is a fully automatic process – you can constrain the design, but you don’t have an option to choose a particular section size during this process. For example the adjacent column on grid intersection A4 has adopted a smaller serial size. You can easily investigate different section sizes directly within Building Designer.

1] Right-click on the column at A3 again, and this time pick Edit from the context menu. You will see the column’s properties. Click the Size tab.

2] Click the button to the right of the section size to see the Data List appropriate to the column. From this list Select the column size UC 203 x 203 x 60.

3] OK the Column Properties dialog.

4] Right-click the column again and from the context menu pick Design Results.

These are updated to show the results for the new size that you selected. In the example above the column fails in Overall Buckling.

Caution When alternative section sizes are investigated in this way the results are based on the current analysis of your model. You would need to re-run the gravity sizing to determine the full effects on your model.
Selection Groups

Selection Groups allow you to associate members together so that you can quickly apply edits to the whole group. Default groups are created based on your attribute sets but you can add your own groups also. We will now use Selection Groups to update all similar perimeter columns to use one single section size.

1] Switch to the Selection Groups tab in the Workspace.
2] Right click on the Columns group and pick Create Sub-Group.

3] Name it Perimeter Columns
4] Pick Column from the Objects toolbar.
5] Select the 8 simple columns around the rectangular perimeter of the building (along grids A1 and 5). Take care not to select the General Columns at grid intersections D3 and C5.
7] Pick Perimeter Columns from the Existing Groups menu.

8] OK the Selection Group dialog.
9] Click the Clear Selection icon to clear the selection of the columns.
10] Now look at the Properties window, as you switch from one selection group to another in the Workspace the common details of the chosen group are displayed.

   Note You can set a colour for each group in the Properties window.

11] Click on the Perimeter Columns group.
Some of the information is removed, for example Section. This is because the section size is not consistent for the whole group.

12] Click in the Section blank cell. Click again on the button that appears in order to call up the data list.
13] Select a UC 254 x 254 x 73 section. This will now be applied to all the perimeter columns.
14] Re-perform the validation and gravity sizing.
Design Results

1/ Pick Show/Alter State from the Edit toolbar to return to the standard display in the Structure window.

Now that you have designed your model for the gravity combinations you are obviously interested in the results.

Note
In the above capture the Workspace has been un-docked and resized so that you can see the details more clearly.

You use the Results page to investigate these. At the top of the Results page you will see the deflections of your model. If you see very large values here, then the most likely explanation is that you haven't defined enough bracing or stiffness in your structure. Following on from the deflections you will see the most critical sway results for your model. Simply double-click the column name to see the full details for these.

2/ Double-click the line showing the reference of the column which is critical for sway in the X-direction and you will see its summary results,

This shows not only the deflections of the column, but also the $\lambda_{\text{crit}}$ values. The values for $\lambda_{\text{crit}}$ are both greater than 10, which means that a full second order analysis of the model is not required.

3/ Click Close to shut the Design Summary dialog.

4/ Move down the entries in the Workspace, and you will see the loading details.
For each loadcase Building Designer shows you the total load that you have applied to your structure, and the total load which has been transferred to the model.

Below this are the details of the various construction levels, and their beams.

6] Double-click on a beam name to see its Design Summary. Note that the beam size is shown in the title of the results dialog.

Note You can also right-click a member in the graphical display and then choose Design Results to see that member’s design results.

6] Click Close to shut the Design Summary dialog.

7] Other results follow these – feel free to investigate.

23 Lateral Sizing

Any members set to ‘Gravity Only Design’ are completely unaffected by the lateral sizing process. Other members are either designed automatically, or checked (depending on their design mode setting) - for the lateral combinations that you have nominated as ‘critical’.

Because, after the initial gravity design has been performed every member is set to Check Mode it is possible that you will want to return to Auto Design Mode for all or part of the structure before performing the Lateral Sizing. This can be achieved by either by picking Design/Set Auto Design Mode..., or via the Show/Alter State dialog.

In this instance, you will return just the two General Columns back to design mode.

1] Pick Design/Set Auto Design Mode...

2] Choose All for General Columns, (leaving Gravity Only unchecked). Choose None for other elements to ensure they remain in check mode.

8] Pick Perform Lateral Sizing from the Design toolbar.

The sizing process will proceed, and you will see a progress bar charting this.

4] On completion the Structure window shows the results of the design graphically.

Note that the ‘Gravity Only Design’ columns are assigned a status of Not Designed.
24 Full Design

When every member in the model is set to Check Mode, a final check must be performed for all members for every active load combination, based on up-to-date analysis results. This is required before you proceed to output the calculations.

1] Pick Perform Full Design from the Design toolbar.

Assuming none of the members fail you can proceed to output the calculations.

25 Create a Report

Now that your design is complete you may need to create a report for submission. You can obtain a report for a single member directly:

1] Right-click beam SB 4/A/5-4/B/5.
2] From the context menu that appears pick Report SB 4/A/5-4/B/5.
3] The report for the beam appears in its own window. Review the details, and then close the report window.

Alternatively Building Designer contains a sophisticated reporting option which allows you to create a report for many members and control its content exactly.

1] View the workbook for the Roof.
2] Pick Show/Alter State from the Edit toolbar to see the Show/Alter State display in the Roof workbook.
3] Click the Report Level option from the Results tab in the Show/Alter State dialog.
4] In the right hand list of report levels, pick the report level that you want to assign to the members you select.

5] Simply click or window the members to set that report level for them. They will change colour to that appropriate to the level they take.

6] Once you have set the details for all members that you want to include pick File/Report/Element Design and you will soon see the report ready for printing.
   In many cases you may not want to create a design report, but you may want to create different summaries. Building Designer allows you to create a material list, a beam end reaction, and a foundation load report for columns which you select. You will create a foundation load report now.

7] In the Reports tab of the Workspace right-click the Columns text.

8] Pick Include all Columns from the pop-up menu.

9] Also in the Reports tab, expand the Base Reactions branch.

10] Right-click the Lc - All text and pick Include from the pop-up menu.

11] Pick File/Report/Base Reactions, and after a few seconds you will see the results for all loadcases for the selected columns.

12] If you want to transfer these details to a Microsoft Excel spreadsheet, rather than creating a report, then you can do so. Instead of picking the menu option detailed above choose File/Export/Export Base Reactions to Excel instead.

13] Pick Show/Alter State to end the Show/Alter State display.

14] Close the open report windows.
26 Modify the Model

You may have noticed that we have not created the initial model as shown previously (see page 7 and compare it to the captures above and you will see that you need to rotate grid line A). This is intentional and will show the power of Building Designer.

1] Change to the Base workbook.
2] Pick Modify from the Edit toolbar and Grid Line from the Objects one.

3] Click grid line A.
4] Hover the pointer over grid line A between lines 4 and 5.
You will see that the tooltip has changed to Click to change angle. Do so, and then move the pointer to the left, you will see that the angle of grid line A changes in 5° intervals.

5] Once the angle of the grid line shows at 100° click again to accept this rotation.
After a short pause you will see that the grid line moves to the new rotation. If you look at the Workbook for any floor, you will see that the amendment has been applied to that floor too.

6] Pick Building/Recalculate Effective Widths…

7] Click OK.
This tells Building Designer to calculate the widths of all the composite beams, and therefore to calculate these for the composite beams affected by the modification1.

Now we need to perform the design again, but only for those beams in the part of the structure affected by the change we have just made.

8] Change to the Structure workbook, and change the view so that you are looking directly down on your model.
9] Pick Show/Alter State from the Edit toolbar to see the Show/Alter State display.
10] Click the Auto Design option from the Design tab.

---
1. Again for a more complex model you could select only those beams affected by the change, maintaining any particular effective widths that you have specified for beams elsewhere in your model.
11] Pick Beam from the Objects toolbar (so that we don’t change the existing column and bracing sizes).

12] Click and drag as shown above, and Building Designer sets all beams within that area to be auto designed. (You may want to switch back to an isometric view to confirm that this is true for all floors – an isometric view will also help you review the redesign more easily).

13] Pick Show/Alter State to end the Show/Alter State display.

14] Switch the Structure view back to South-West isometric.

Because some of the composite beams to be redesigned have changed in length, the existing shear stud layouts will be incorrect. It will therefore be necessary to set the stud layouts back to Auto-layout before redesigning.

15] Pick Design/Set Auto Design Mode...

16] Choose All for Composite Beams - Studs, (leaving Gravity Only unchecked). Choose None for other elements.

17] Validate your model.

18] Run the Gravity and Full design steps once more and review the results. You should find that the altered beams and columns all pass the design checks, and that all the unchanged beams do so too.

Note: A few warnings remain, you might choose to investigate these now and consider if further modifications are necessary in order to eliminate them.

Congratulations you have now finished this Quick Start Tutorial. We hope that you have found this introduction helpful.
27 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. However, we would suggest that you book yourself on the next available Building Designer training day, where you will find that our expert tuition gives you a flying start, ensuring that you are fully productive in the shortest possible time.

You might also want to:

- browse the on-line documentation accessible from the Help menu,
- take a look at the extended Quick Start Guide which contains a continuation to this worked example. This covers more of Building Designer’s features than has been possible in this short document and takes the design of your example building further,
- read the technical notes which are installed in the Documentation folder which you will find in the Program Files\CSC\Fastrak folder. These technical notes deal with the sway resistance of models among other things,
- look at exporting your model to a .dxf file,
- export your model to Revit Structure, and find that your structure is completely modelled,
- export your model to S-Frame, and investigate the analysis model fully.

---

1. Contact our Support Department for details.
2. You will need an application which can read this file format.
3. Assuming that you have purchased and installed Revit Structure.
4. Assuming that you have purchased and installed S-Frame.
## 28  CSC Offices Worldwide

<table>
<thead>
<tr>
<th>Company</th>
<th>Address</th>
<th>Telephone</th>
<th>Fax</th>
<th>Email Support</th>
<th>Internet</th>
</tr>
</thead>
<tbody>
<tr>
<td>CSC (UK) Ltd</td>
<td>Yeadon House, New Street, Pudsey, Leeds</td>
<td>Tel: (44) 113 239 3000</td>
<td>Fax: (44) 113 236 0546</td>
<td>Email: <a href="mailto:sales@cscworld.com">sales@cscworld.com</a>, <a href="mailto:support@cscworld.com">support@cscworld.com</a></td>
<td>Internet: <a href="http://www.cscworld.com">www.cscworld.com</a></td>
</tr>
<tr>
<td>CSC Inc</td>
<td>500 North Michigan Avenue, Suite 300, Chicago, IL 60611, USA</td>
<td>Tel: 877 710 2053</td>
<td>Fax: 312 321 6489</td>
<td>Email: <a href="mailto:sales@cscworld.com">sales@cscworld.com</a>, <a href="mailto:usa.support@cscworld.com">usa.support@cscworld.com</a></td>
<td>Internet: <a href="http://www.cscworld.com">www.cscworld.com</a></td>
</tr>
<tr>
<td>CSC WORLD (Malaysia) Sdn Bhd</td>
<td>Suite B-12-5, Block B, Level 12, North Point Offices, Mid Valley City, No.1, Medan Syed Putra Utara, 59200 Kuala Lumpur, Malaysia</td>
<td>Tel: (60) 3 2287 5970</td>
<td>Fax: (60) 3 2287 4950</td>
<td>Email: <a href="mailto:sales@cscworld.com">sales@cscworld.com</a>, <a href="mailto:asia.support@cscworld.com">asia.support@cscworld.com</a></td>
<td>Internet: <a href="http://www.cscworld.com">www.cscworld.com</a></td>
</tr>
<tr>
<td>Civil &amp; Structural Computing (Asia) Pte Ltd</td>
<td>3 Raffles Place, #07-01 Bharat Building, Singapore 048617</td>
<td>Tel: (65) 6258 3700</td>
<td>Fax: (65) 6258 3721</td>
<td>Email: <a href="mailto:sales@cscworld.com">sales@cscworld.com</a>, <a href="mailto:asia.support@cscworld.com">asia.support@cscworld.com</a></td>
<td>Internet: <a href="http://www.cscworld.com">www.cscworld.com</a></td>
</tr>
<tr>
<td>Civil &amp; Structural Computing Pty Ltd</td>
<td>Level 3, 349 Coronation Drive, Milton QLD 4064, Australia</td>
<td>Tel: 1300 882 393</td>
<td>Fax: +61 (07) 3378 5557</td>
<td>Email: <a href="mailto:sales@cscworld.com">sales@cscworld.com</a>, <a href="mailto:oz.support@cscworld.com">oz.support@cscworld.com</a></td>
<td>Internet: <a href="http://www.cscworld.com">www.cscworld.com</a></td>
</tr>
</tbody>
</table>
29 The Quick Start Guide continues

If you are have just completed the example in the printed *Quick Start Guide*, then your model is ready for you to follow the instructions below.

If you are picking up the example at this point, then you should open the file *Quick Start Example 29.fastrak* and then follow the instructions below.

**Saved files**

As we created this extension to the worked example we saved the model away at the start of each chapter and numbered these saved models according to the chapter number. Thus if you want to pick the example up at a particular point, rather than working through it in its entirety, you can simply open the appropriate file. If you want to load the example as it stands at this point (the start of chapter 29), then the file you need to open is *Quick Start Example 29.fastrak*, if you wanted to open the example as it stands at the start of chapter 30, then that's in the file *Quick Start Example 30.fastrak* and so on. You will find these files in the `documents and settings\All Users\Application Data\CSC\Fastrak\Examples` folder. You can open and use these files, but you can not save them away unless you change their names, this is done to protect the originals.

**Note**  
The Example Files were created prior to Amendment 1 of BS 5950-3:1:1990 so the results in this Guide are displayed accordingly.

**Reviewing sway**

For detailed information on the way that *Building Designer* handles sway refer to the "Building Designer Advisory Note – Designing for Second-order Effects". This comprehensive document contains essential information, and if you have not done so already we recommend that you acquaint yourself with it immediately. During the course of the following discussion we shall refer to this document.

In this extension of the example we shall initially remove some of the bracing in order to increase the building’s susceptibility to sway. This then allows us to investigate the different approaches available for increasing sway resistance.

1) *Pick the workbook for Frm D, and then set the options from the toolbars which allow you to delete braces. Delete the bracing system in this frame.*

   Before you proceed to using second order $P$-$\Delta$ analysis you should always design your structure using first order analysis. In this way you know that your structure actually works, before you begin to investigate any second order effects.

2) *Re-perform the validation and full design steps to check the structure.*

   Having checked the model using first order analysis, (do not worry if a brace member is failing at this point), you will now switch to second-order analysis.
3/ Pick Design/Analysis Options…

4/ From this dialog pick the options shown above (these switch on the second order analysis of your model and set the formula that you want to use in the automatic calculation of the $k_{amp}$ factor). Once you have made these settings click OK.

5/ Pick Loading/Combinations…, or click the Loading toolbar’s Edit Combinations icon.

You will see the Combinations dialog.

6/ Pick the 1.4D +1.6I + NHF X+ combination, click Edit and then click the Second order effects tab.

This allows you to control the second order analysis, and the values on which the automatic calculation of $k_{amp}$ will be based. See the Building Designer Advisory Note – Designing for Second-order Effects for further information.
7] For your model the default values are considered reasonable so click OK to close the dialog. (See the "Building Designer Advisory Note – Designing for Second-order Effects" for background on the options given in this dialog.)

8] Validate and check your model again. This time the lateral loads will be amplified and hence the design will automatically include for second order effects. On completion of the design the Project Workspace will show the results.

You will see that the Sway Y Critical value causes the $\lambda_{\text{amp}}$ value to show as a warning since Building Designer can not currently handle models where $\lambda_{\text{crit}}$ is less than 4. Essentially this is beyond the scope of the BS5950 approach, see the "Building Designer Advisory Note – Designing for Second-order Effects" for further information.

9] Double click the reference of the critical column (SSC E/8) to see its Design Summary.

10] Double click the line relating to $\lambda_{\text{crit}}$ to view the details of the check.
This shows that the sway under notional loads is \(7.8 \text{ mm}\) in the global Y direction, the stack height is \(4 \text{ m}\) and hence \(\lambda_{\text{crit}} (= \text{height}/200) = 2.56\). Note that this sway also equates to \(\text{height}/513\).

11] Close the column’s Design Results.
12] Switch off the Show/Alter State display.

### 30 How’s the structure working?

You obviously need to investigate what’s going on with this building, why is it swaying so much in the global Y direction? How does it actually sway? Building Designer provides you with several methods of understanding how your building is working. You will look at a couple of these now.

1] Ensure that you are viewing the Structure window.
2] Pick Combinations from the Loading toolbar, and pick the \(1.4D +1.6I + NHF Y+\) combination from the list.

Building Designer shows the axial forces in the structure graphically:

The axial forces in the columns are much larger than those in all other members, and thus you cannot see the loads in these other members clearly.
In order to see the loads in the other members clearly use the View Options Columns page to switch off the display of simple and general columns.

Building Designer automatically re-scales the axial force diagrams appropriately for the members that it is showing.

It is clear that the bracing resisting sway in the Y direction is not provided centrally in the building and therefore the building tries to twist when Y direction loads are applied. When this happens the bracing in the X direction tries to resist the twist resulting in the brace forces we see above. Assuming that you want to try and reduce this twisting you need to add more Y direction bracing towards the other end of the building.

Another way to see what is happening with the building is to look at the way in which it is swaying. Building Designer allows you to do this directly.

Switch the display of columns back on again.

Pick Sway from the Output Graphics toolbar.

Use the View Options to switch on the Sway Y and Abs. Sway Value options.
Building Designer shows the sway of the structure graphically.

You can use the View Options to see the text values for the sways in the X- or Y-directions, and either the absolute or relative values.

Note: If you hold the Ctrl key down and rotate your mouse wheel, then this scales the sway display.

Again it is clear from the graphics that the model is twisting, and that the critical sways are at the end closest to grid line D.

Pick Sway again from the Output Graphics toolbar to return to the normal display.

Discussion of Notional Load versus Wind Load Deflections

The general point to be drawn from the following discussion is that where $\lambda_{crit}$ drops to around 4 in any direction you should carefully consider the possibility that wind sway deflection could be critical before concentrating too much on designing for 2nd order effects.

Consider an idealised section of a building as shown below with an in-wind depth of $L\ m$. 

```
L
```

```
G = 5 kN/m
Q = 5 kN/m
```

Current floor

Floor above

Floor below

3.5 m

3.5 m
Assume a one metre width of building (that is a 1 metre wide slice with the section shown above) and a wind load of 1 kN/m² (the sum of the windward and leeward loads). With a 3.5 m floor to floor height this gives an unfactored wind load of 3.5 kN at each floor level. For the notional load to equal the unfactored wind load then

\[ 0.005 \times (1.4 \times G + 1.6 \times Q) \times L = 3.5 \]

Thus when \( L = 46.7 \text{ m} \) the notional forces equal the wind forces.

Now consider the deflections involved when \( \lambda_{crit} = 4 \).

\( \lambda_{crit} \) is derived from the equation \( \text{height}/200 \) so when \( \lambda_{crit} = 4 \) the maximum floor to floor deflection \( \delta = \text{height}/800 \).

Therefore, if we had a 46.7 m deep (in-wind depth) building with a total wind pressure of only 1 kN/m² and \( \lambda_{crit} \) was 4, the wind load condition would not be critical because the wind load deflection would also be \( \text{height}/800 \).

However, as the building gets narrower the notional loads decrease but the wind load stays much the same and so the possibility that wind load deflections are critical increases. If the limiting wind load deflection is \( \text{height}/500 \) then we can work out a limiting in-wind building depth below which wind deflections are more likely to be critical when \( \lambda_{crit} \) is approaching 4.

If \( L > 46.7 \times 500/800 \) (that is if \( L > 29.2 \text{ m} \)) wind deflections may be critical.

Repeating similar estimates for a range of floor loads and floor to floor heights the following table can be prepared.

### Approximate in-wind depth (assuming total effective wind load of 1.0 kN/m²) below which wind sway deflections are likely to be more critical than notional sway effects if \( \lambda_{crit} \) is approaching 4

<table>
<thead>
<tr>
<th>( 1.4G + 1.6Q )</th>
<th>Floor to floor height (m)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>3</td>
</tr>
<tr>
<td>10</td>
<td>37.5</td>
</tr>
<tr>
<td>15</td>
<td>25</td>
</tr>
<tr>
<td>20</td>
<td>18.7</td>
</tr>
</tbody>
</table>

|                  | 3.5                      |
| 10               | 43.8                     |
| 15               | 29.2                     |
| 20               | 21.9                     |

|                  | 4                        |
| 10               | 50.1                     |
| 15               | 33.4                     |
| 20               | 25                       |

If the wind load were 1.3 kN/m², then all the lengths in the table would increase by a factor of 1.3 – that is it is more likely that wind sway deflections will be critical.
As noted at the outset, the general point to be drawn from the above discussion is that where $\lambda_{crit}$ drops to around $\lambda$ in any direction the possibility that wind sway deflection control could be critical should be considered. This possibility becomes more likely as various factors change. Fairly obviously wind deflections are more likely to be critical than notional sway cases when:
1. Wind loads get higher,
2. Floor loads get lower,
3. The in-wind depth gets smaller, or
4. The floor to floor height increases.

### 31 Add bracing

In order to reduce the sways, you need to provide additional stiffness at this further end. You will look at some different ways of achieving this now.

1/ Open the view on frame D and recreate the original bracing system as shown below:

![Diagram of bracing system]

2/ Validate and recheck your model. On completion of the design review the sway results in the Workspace, and you will now find that the critical sway value in the Y-direction is greater than that in the X-direction, and thus your building is non-sway in both directions.

This exemplifies the fact that providing well-placed bracing is the best way of providing rigidity to a structure. On the other hand using bracing in itself does not guarantee that a frame is non sway, a structure with poorly placed, or inadequate bracing can be susceptible to sway.
32 A cross-braced system

The client does not want to use large bracing in this location, so let’s look at some other options.

1/ Delete the bracings on frame D.

2/ Create a new brace attribute set named 80 x 8 flat, and set the size of the braces to be 8 x 80 flat bars.

   Note that as soon as you pick a flat bar section size the Compression page is removed. This is because flat bars can only be designed for tension, so the compression details are not appropriate.

3/ Create the bracing system shown below.

4/ Validate the structure, and you will find that you have 3 warnings which relate to having cross-braces defined, both of which are active.

   When you define any brace you can choose to make that brace active or non-active. This enables you to investigate different scenarios quickly and easily. For cross-bracing the assumption is that you may want the system to be designed for "tension only” hence a warning is given. If you do not want it to be considered as tension only ignore the warning. If you do want it to be "tension only” then you need to deactivate braces so that only one of each pair remains active.

   If you analyse with both active then the load will be shared between the two braces and one will be checked for tension while the other is checked for compression. If you try to use crossed flats without deactivating one of them, at least one of them will develop compression and it will fail (flats are never designed for compression).
If you deactivate one cross brace, the remaining active one will be in compression for some cases and in tension for others. For the compression cases the design load will be reversed so it is checked for “tension only”.

5] Pick Show/Alter State from the Select toolbar to see the Show/Alter State display for frame D.

6] In the Show/Alter State dialog pick the Active Brace option from the Analysis tab, and then click on each of the braces which slope from top left to bottom right to make them inactive.

Note To move the display of the frame so that you can see the bracings clearly, pan the image by clicking and holding the middle mouse button (or mouse wheel) over the graphical part of the display and then drag the image to its new location.
Validate and recheck your model. On completion of the design review the sway results in the Workspace, and you will now find that the critical sway value in the Y-direction is 5.395 which is greater than 4, so this is a sway frame. Building Designer has been able to handle the sway calculations automatically using the k<sub>amp</sub> approach and shows the appropriate factors for each combination.

Note Since λ<sub>crit</sub> (and ultimately k<sub>amp</sub>) relate to the gravity loads in a combination these values are identical for each of the four combinations in this model (because they all contain the same gravity load component).

For the combination 1.4D + 1.6I + NHF Y+, the axial load in the bracing was -54.7 kN (negative load indicating tension) without taking k<sub>amp</sub> into account (switch off the amplification option and re-analyse if you want to check this). Now if you look at the axial force diagrams this load has been amplified to -61.6 kN.
For the case $1.4D + 1.6I + NHF Y$, the axial load in the bracing is amplified to 46.6 kN (compression). If you examine the design results for the brace you see the design check as shown below, the compression force is treated as tension.

![Design Summary](image)

Therefore, for this and many examples it is not essential that you correctly identify the active brace for each and every combination. However, where the bracing is not symmetrical a cross check may be more important.

**Note**
In the current version of Building Designer the capacity checks are based on the gross section area (there is no allowance for bolt holes), if this is likely to be critical then the workaround in the current release is to define a plate that has the correct nett (as opposed to gross) area noting that this will have some impact on $\lambda_{crit}$ and $k_{amp}$ as noted below.

**Note**
Where braces are checked by deactivating one element of a cross brace some care should be taken to ensure that the foundation at the base of the deactivated member is designed for the required shear loads (that is for the loads that would actually cause the deactivated member to be in tension).

Finally note that by decreasing the flat size $\lambda_{crit}$ drops (possibly below 4), conversely increasing the flat size increases $\lambda_{crit}$ and wind deflections are controlled.

### 33 An inverted V-brace system

You find that you need to have a door in the centre of the bottom bay, and so you opt to change the bracing system again and go for an inverted V system.

1. **Delete the bracings on frame D.**
2. **Create a new brace attribute set named V Brace, and set the size of the braces to be 60.3 x 3.6 CHS.**
3] Create the bracing system shown below. Start each brace at the lower floor, and end it at the centre point of the beam. (To get at this point, click on the beam to get the standard points, and then click on the centre one.)

4] Validate the structure, and you will find that you have 6 warnings which relate to having braces connected to beams, the brace not having a vertically released end, and that this may put uplift into the beam. Building Designer allows you to release a bracing so that it doesn't carry vertical load at its start or at its end (it does not allow you to release the bracing so that it doesn't carry vertical load at both ends). You will change the braces now so that they don't carry vertical load at their ends.

5] Select the 6 braces on frame D.

6] Use the Properties pane to change their properties so that Vert. release end is set to Yes.

7] Validate and redesign your model. On completion of the design you will find that the critical sway value in the Y-direction is 7.669 but that some of the braces fail the design checks.

8] Change the V Brace attribute set so that the section size is 88.9 x 5.0 CHS.

9] Apply the attribute set to the selected braces.

10] Validate and recheck your model. On completion of the design you will find that the critical sway value in the Y-direction is greater than 10, and that the braces pass the design checks.
11) Click Moment Major from the Output Graphics toolbar.

This emphasises the effect of releasing the bracing vertically.

12) Switch off the display of the major axis moments.

34 A partially braced solution

Your client accepts the inverted V-brace from ground to first floor, but will not accept it for the upper levels, and so you need to investigate the impact of this.

1) Switch back to the 200 x 100 x 6 RHS attribute set, change the size to 200 x 100 x 6.3 RHS and rename the set accordingly. Then apply this set to all 15 bracings in the model.
2] Return to Frame D and delete the bracings from the two upper bays so that you end up with the arrangement below.

3] Validate and recheck your model. On completion of the design you will find that the critical sway value in the Y-direction is just under 4.

4] Increase the size to 200 x 100 x 8 RHS in the attribute set and apply this to all 11 braces in the model.

5] Validate and recheck your model once more. This time you will find the critical sway value in the Y-direction is over 4. Simply adequately bracing the bottom bay reduces first storey lurch, and gives an acceptable solution.

It is worth noting that for many structures sway sensitivity (and overall deflection) is dictated by the stiffness between foundation and first floor levels. If you can only introduce better/stiffer bracing at this lowest level it may still have a significant benefit to your overall design.

Note also that this arrangement of bracing does not put additional loads into the columns, thus the external column (which could show a Fail status for some of the other bracing arrangements) has a Pass status.

35 A rigid frame solution

You take your solutions to the client who is still adamant that he doesn't want any bracing in this frame, and asks you to find what you would have to do to achieve an unbraced solution.

1] Delete the two remaining bracings.

2] Right-click on the simple column and pick Edit from the context menu that appears.

3] Change the Construction Type from Simple to General.
4. Using a similar process change the three Simple Beams in Frame D to be General Beams. At the same time set each of these beams back to Automatic Design.

6] Pick Show/Alter State from the Select toolbar to see the Show/Alter State display for frame D’s workbook.

6] From the list of options on the Show/Alter State Building tab pick Moment Frames.

7] Drag round all the members in the frame and Building Designer makes it a moment frame.

8] From the list of options on the Show/Alter State Design tab pick Auto Design and click on the columns in Frame D to set them back to be designed.

9] Pick Show/Alter State from the Select toolbar to return to the normal display. If the columns are to be as effective as possible in the moment frame they need to be orientated with so that their strong axis is acting in the direction of the frame.
10] Switch off the Orthographic View of Frame D (View Options Grid page) and view the frame from the South-West.

11] Click the right-hand column, and change its angle to 0°.
12] Click the left-hand column, and change its angle to 0°.
13] Validate and design your model again. Building Designer will calculate sizes for the members of frame D, however you will find that the critical value for sway in the Y-direction, although increased, is still less than 4, and in this case Building Designer shows a $k_{str}$ value of 1.0, but flags this with a warning. Despite the above warning Building Designer has calculated sizes for the members of Frame D and most of them are indicated as passing.

This occurs because Building Designer designs the members of the frame without applying any amplification so that you can see some sort of result and decide what you want to do next. Most of the members pass on this basis, but the stiffness of the frame is still insufficient to give the rigidity you probably need. (see the ‘Discussion of Notional Load versus Wind Load Deflections’) Assuming that you want to force $\lambda_{crit}$ up (and reduce wind sway) you have several choices.

1. Find another place to get some bracing into the building.
2. Apply a user defined $k_{str}$ value (probably something in excess of 30%) and set the members back to auto design. The higher lateral loads might force the selection of bigger members, thus increasing stiffness hence nudging $\lambda_{crit}$ above 4.
3. Or simply manually increase the member sizes in a trial and error fashion. Since the frame is rigid, the sway is dependent on the sizes of both the columns and the beams. This means that there are many alternative sets of beam and column section size which will achieve a critical sway value greater than 4. You will do this now.

14] Right-click each of the two columns, and pick Edit from the context menu that appears.
15] Pick the Size tab, and set a section size of 356 x 368 x 177 UC.
Using a similar process set a beam size of 610 x 229 x 125 UB (you set the size on the Beam tab).

17] Validate and design the frame again, and you should now find that the critical sway value in the Y-direction has increased to just over 4. Note that \( k_{\text{prop}} \) is now calculated (1.208) and so the sway loads are amplified and the design is complete – if you are happy with these large sizes.

In this example column sections were used to increase the sway resistance. Since the important attribute of the columns in this regard is the major axis moment of inertia, then from a pure efficiency point of view universal beam rather than universal column sections would have been a better choice.

**Provide fixity at supports**

These are very big increases in section size - so is there any way to reduce these. You could try providing fixity at the supports.

1] Select the two supports and use the Properties pane to set their Stiffness Major to Nominally fixed.

2] Deselect the two supports.

3] Validate and check the frame again. You will see that \( \lambda_{\text{crit}} \) increases to just over 6 (basically a 50% reduction in deflection) — once again this emphasises the importance of focussing on the stiffness of your structure at the lowest level in order to influence \( \lambda_{\text{crit}} \) positively.

4] Set a section size of 305 x 305 x 97 UC for each of the two columns, and a size of 457 x 191 x 74 UB for the beams.

5] Validate and check the frame again, and you should now find that the critical sway value in the Y-direction is still over 4 with these much reduced sections. However, bear in mind that the foundation design requirements for the fixed bases may have a significant impact on foundation costs.

**36 Defining a sub-structure**

Sub-structures allow you to view or work on a particular part of your structure in detail without having to contend with the entire model. You will create a sub-structure that just shows the braced and rigid frames now.
1] Click the Project tab in the Workspace, and scroll the display to the top left. Close the Frames branch, and the Workspace should look like this.

2] Right-click the Sub-structures entry and pick Create Sub-structure from the context menu that appears.

3] You will see that a new entry is created with the name Sub-structure 1, simply type the name you require – Braced Frames – and press Enter.

4] Right-click the Braced Frames entry and pick Open 3D View from the context menu that appears. Building Designer creates a new workbook for the sub-structure, this is currently blank.

5] Click the Base entry in the Construction Levels branch of the Workspace, and drag it over the Braced Frames workbook. Building Designer adds all the elements that lie on this level, and any elements that connect to them to the sub-structure.
Pick South West from the View toolbar to see your sub-structure in isometric view.
7] Repeat the above process to add the Roof, Second and First levels to the sub-structure. When you have finished your sub-structure will contain your entire structure!

8] Use View Options to add the display of Slabs, Slab Span Directions and Axes to the view.
9] Now to prune this to contain just the objects in which you are interested. Click Top from the View toolbar to view the sub-structure from above.

10] Pick Remove from the Edit toolbar and ensure that no object type is picked from the Building Objects one.

11] Now click and drag round the entire structure.

12] Building Designer asks you to confirm the elements to remove.

13] Remove the ticks against all lines other than the one relating to Slab Item and then click OK and Building Designer removes all the slabs from your sub-structure. This makes it easier to delete the other elements you don’t require.

Note You could also have picked Remove from the Edit toolbar and Slab from the Building Objects one, in which case only the slab would appear in the Confirm Remove dialog.
14) Now click and drag as shown below, while holding the **Shift** key down.

**Caution** With the Shift key pressed **Building Designer** removes any object which lies totally within the boundary, or which crosses it. Take care that you don’t touch the bases just outside the box shown above with the rectangle you create.

15) **Building Designer** asks you to confirm the elements to remove - you could use this option, for instance just to leave the beams in the sub-structure, removing everything else.
16) You want to remove all the objects, so click **OK**.

17) **Caution** Again watch out for the bases.
18) Again confirm the removal of all the object types.

19) Either use a similar process to delete the cantilever beams, or simply click over each set of beams 3 times (to remove the beam from each of the 3 floor levels).

20) Finally view the sub-structure isometrically from the South-West.

Review sub-structure results
For this Quick Start Example you simply review the results for the sub-structure. You can also use a sub structure to work on parts of your structure, for instance you could use a sub-structure to define a complex region of steel crossing several floors.

1] Pick Combinations from the Loading toolbar, and pick the 1.4D + 1.6I + NHF Y+ combination from the list.

2] Pick Sway from the Output Graphics toolbar.
Building Designer shows the sway of the sub-structure graphically.

Again View Options has been set to show the absolute values in the Y-direction.

Referring back to the previous captures it is clear that sway has been reduced. It is also clear to see that using a sub-structure makes it easier to see the details of interest.

You may want to use the other options from the Output Graphics toolbar to review the other results for the sub-structure. As before you can use View Options to switch off the display of particular member types to show particular details of interest.

37 Define roof trusses

In the current version of Building Designer you can define truss members in your model and then check their adequacy. Truss members will attract loads and participate in the 3D structural analysis, but elements of the truss can only be checked (that is not yet be designed). Therefore you might define important trusses such as “transfer trusses” where columns from upper levels are supported on a deep truss. By modelling the truss you can design the structure above and below it correctly. In the following example you will define some roof trusses which will pick up roof loads and carry them to perimeter columns. As you will see there is a certain level of complexity when dealing with trusses and sloped roofs, it is therefore appropriate to consider the extent of effort and detail to go to when defining truss work. In this example, could you achieve the same (or a similar) result in a fraction of the time by idealising the truss work with a handful of long-span beams?
Building Designer has a Truss Wizard to help you define many different types of truss. You will now add trusses to your model, extend columns to carry them and then add bracing to achieve a stable structure. When you have finished you will have created the new steelwork shown below.

1] Pick Building/Levels… and add new level Truss Base above the Roof, and set this at a level of 16 m, ensure that the Floor box is ticked and specify No diaphragm.
A roof level is a good example of where loading should be included in the calculation of imposed load reduction, but where there is no floor to tie the steelwork together, and thus where diaphragm action should not be considered. The settings you have just made model this.

2] Use the Project Workspace to open a 2D view of this level. This will appear as shown below.
If you look at the view you have opened, you will see that the red dots are missing at some intersections along grid line A. This means that there is no longer a grid intersection at the point where there is no red dot. This has occurred because you rotated grid line A. When you did so, although Building Designer correctly moved and adjusted all affected objects, it simply rotated grid line A about its centre, and did not adjust any other grid line in any way. This was intentional, and designed to prevent wider ramifications to your structure. Before you can place the trusses you require you need to modify the grid system manually to reinstate the grid intersections.

3] Pick Modify from the Edit toolbar and Grid Line from the Building Objects one.

4] Click grid line A. You will see that Building Designer selects the grid line and adds 3 boxes to it.

6] Hover the pointer over the top box.
   You will see that the tooltip has changed to Click to move end point. Do so, and then move the pointer upwards, you will see that the extension of the grid line changes. Once you have an extension of 10.000 m or more shown click again to extend the line.

6] Repeat the process for the bottom box - an extension of 10.3 m or slightly more is adequate.

7] In a similar manner extend grid lines 4 and 5 to reinstate their intersections with grid line A.

You are going to add parallel boom trusses along grid lines 1, 3, 4 and 5. In order to create a fall of 1 in 20 across the building, these are going to have different heights as tabulated below.

<table>
<thead>
<tr>
<th>Grid line</th>
<th>Height (m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>2.500</td>
</tr>
<tr>
<td>3</td>
<td>2.850</td>
</tr>
<tr>
<td>4</td>
<td>3.150</td>
</tr>
<tr>
<td>5</td>
<td>3.450</td>
</tr>
</tbody>
</table>

In this version Building Designer does not design truss members, it simply checks the member sizes which you have specified. Building Designer allows you to define up to four section sizes for a truss members. These are for the top boom, bottom boom, end posts and internals of your truss, and fully design or check a truss. You will start by defining the truss on grid line 1.

1] Building Designer defines the elements of trusses as truss members, which take the current default truss member attribute set. Use the Project Workspace to access the default Truss Members attribute set, specify the section sizes tabulated below, ensuring that the Alignment of each member type is Centre/Centre.
Chapter 37: Define Roof Trusses: 103

- Internal – SHS 150x150x6.
- Side – SHS 150x150x6.
- Bottom – SHS 150x150x10.
- Top – SHS 150x150x10.

2/ Pick Building/Truss Wizard… You will see the Truss Wizard.

This page of the Wizard allows you to pick the basic shape of the truss, (and also to define its location numerically should you desire, however you can also do this graphically).

3/ Pick Parallel Chord from the dialog.

4/ From the graphical display of the Truss base level click on grid intersection A1 (you will see that the Wizard sets the appropriate location for the start point).

5/ Now move over grid intersection D1. Click on this point and Building Designer shows the next page of the Wizard.

6/ This page shows the available configurations of truss of this shape that are available, and allows you to define the required details. You may want to investigate the various Types that are available. Before you continue ensure that
you return to the Type Parallel 1. For your example define a 10 panel truss with a height of 2.5 m. Once you have done so the picture reconfigures to show you the layout you will achieve.

7) Click Finish to create the truss. You will see a progress dialog which tells you how many truss elements the Wizard is generating, and how many of these it has created so far. On completion of the process you will see the truss in the Truss base workbook.

If you scroll to the top of the Project Workspace, and look at the list of levels, you will see that Building Designer has automatically created a new level for you at the top of the members in the trusses you have created. It has named these levels Parallel Booms 1 - L 1.
8/ Add the remaining trusses shown below, maintaining the Type, Panel Number and Height values given above. For your convenience the Height values are tabulated below:

<table>
<thead>
<tr>
<th>Grid line</th>
<th>Height (m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>2.850</td>
</tr>
<tr>
<td>4</td>
<td>3.150</td>
</tr>
<tr>
<td>5</td>
<td>3.450</td>
</tr>
</tbody>
</table>

When you have finished your screen should look like this.

If you switch to the Structure workbook, you will see the trusses in an isometric view. You will also see that none of the trusses are supported, the columns must be extended to pick up the truss ends. You will do this now.
1] Switch to the **Base** workbook, and select the eight columns shown below.

2] Switch to the **Structure** workbook.

3] Pick **Select** from the **Select** toolbar and **Columns** from the Building Objects one. The column selection you made in the **Base** workbook is maintained.

4] Choose Building/Column Levels…

5] Set **Truss Base** as the end level and click **OK**. The selected columns are extended and shown in the **Structure** window:

6] Deselect the columns.
7) Set the **Simple Beams** attribute set as the default one from the **Workspace**.

8) Define beams to tie the top corners of the trusses together as shown below.

![Diagram of trusses with beams](image)

**Note** The beams which you have just created are sloping. It’s as easy to define sloping beams as it is to define flat ones!

9) Define beams between the tops of the columns (immediately beneath the six beams you have just created).

![Diagram of additional beams](image)

You would obviously need more bracing than this in a real structure. Again you know how to define this, so you won’t do that here.
Update bracing systems

1/ Update the bracing systems on grid lines 1, 5, A and D as shown below.

Frame 1

Frame 5
Chapter 38 : Add roof plane : 109

For this exercise convert the rigid members on frame D to make them non-rigid and set them to be redesigned automatically.

38 Add roof plane

1/ Move to the Structure workbook,
2/ Use the View Options Floors tab to switch off the display of the Base, First, Second, Roof and Truss Base levels.

1/ Pick Create from the Select toolbar and Roof from the Building Objects one.

2/ Click each vertex of the roof area as indicated below.

As you click the vertices, you will see that a rubber polygon follows the pointer indicating the roof plane that you are creating. When you click point 6 (the start point) Building Designer recognises this automatically and creates the roof plane for you.
In an actual structure you would provide purlins spanning between the trusses. For this example you will achieve the same loading on the structure below by spanning the roof in the other direction.

3] Pick Select from the Select toolbar, select the roof, then use the Properties pane to change its Span Dir to 90° and then deselect the roof.

4] Validate your model. (This checks that the roof plane that you have defined is co-planar. If you make a mistake and define a non-co-planar roof plane, then you should not continue to add loads, since when you delete the roof plane Building Designer also deletes the loads automatically.)

**Define roof loads**

1] From the drop-list in the Loading toolbar pick the Perimeter Wall loadcase.

   **Note**
   The roof dead load is being added to this loadcase for convenience only. In a real model it is expected that a new loadcase would be created and the combinations edited accordingly.

2] Use View Options to ensure that the display of Area Loads is switched on.

3] Pick Create from the Edit toolbar and Area Load from the Loads one.

4] Click anywhere over your roof.

5] Enter the value of the load (0.35 kN/m²) and then click OK.

6] Create another area load in the Composite Imposed loadcase with a value of 1.5 kN/m².

Having made changes to many of your columns you will need to redesign these. You could pick these out individually, but Building Designer allows you to set whole groups of members back to be automatically designed very quickly.

7] Pick Design/Set Auto Design Mode…
You could change every designable member in your structure back to automatic design with 7 clicks!

8/ Set the options shown above to change all the columns in your model back to automatic design.

9/ Reinstate the display of the floors you switched off earlier

10/ Validate and design your structure again and review the results at your leisure.

39 Define wind load

Building Designer comes with a Simple Wind Load option which can calculate and apply loads to your structure via its walls. Obviously before you can use this tool you need to define the walls that transfer the load to your structure.

There is also an optional add-in for Building Designer – the Fastrak Wind Modeller. Fastrak Wind Modeller uses the BREVe database to determine the appropriate wind details for all locations in the United Kingdom, and then calculates the appropriate wind loading details for your structure.

You also define the wind directions in which you are interested, and Fastrak Wind Modeller automatically calculates the appropriate wind zones on the roofs and walls of your structure. You can set the type of each roof to achieve the correct zoning, and can define the internal and external faces of the walls. If you want to tailor the zoning to account for particular features in more detail, then you can also do this.

Fastrak Wind Modeller can automatically define standard wind loadcases for you based on the usual internal pressure coefficients, or you can define the loadcase information yourself. In both cases Fastrak Wind Modeller calculates the appropriate loading on each zone.

Fastrak Wind Modeller has its own Quick Start Guide, and is therefore not considered here.
1] Switch to the Frm A workbook and view the frame from the left.

2] Pick Create from the Edit toolbar and Wall from the Building Objects one.

3] Click the nodes in the order shown below.
As you hover over the third and fourth nodes, *Building Designer* indicates the wall that it will create using highlighting. When you click node 5 *Building Designer* creates the wall shown below.

4] *Switch to the Frm 1 workbook and view it from the front.*

5] *Pick the options to create walls for this frame.*
6] Again click on the nodes in the order shown below.


In practice you would need to create the other walls around your structure, so that you could consider wind coming from the back and right. You would also create walls around the hemispherical area to pick up the wind loads that apply to it. You will not do this for this example.

When you create walls Building Designer assigns an inside and an outside face to each wall. You should always check that walls in your building have the correct face to the wind, since the inner face of a wall is treated as though it is totally permeable! The wind just passes straight through it!
8] Pick Show/Alter State from the Select toolbar to see the Show/Alter State display for the structure.

9] From the list of options in the Show/Alter State dialog pick Wall Surface.

10] This view shows that the outer face of both walls is correctly defined.

11] Switch off the Show/Alter State view.

12] Create two new loadcases for the wind loads. Call these Wind at 0 deg and Wind at 90 deg (don’t forget to change their type to Wind).

13] Pick Wind at 0 deg from the list in the Loading toolbar.

14] Pick Loading/Simple Wind Loading…
18] **Type in a Length** of 19.45 m (the maximum height of the truss on line 5), an **Area Load** of 1.0 kN/m² and click OK. Building Designer calculates the loads that apply to your model, and shows these graphically. If you zoom into the wall and make the appropriate View Options settings you will be able to see the individual nodal loads that simple wind load generation has created.

**Note**

Even though you have applied the wind load up to the top of the truss, Building Designer only calculates and applies the loading up to the top of the columns. This is a limitation of the current version and will be addressed shortly. If you wanted to apply loads from the area above the columns, then you would need to apply additional nodal loads to the top of the columns and also to the nodes at the top of the truss.

---

1. Provided that the View Option to see the **Wind Pattern** is checked. You will need to make this setting yourself since the option is not checked by default.
16] Use a similar process to define the wind loads in the other loadcase. This time set the Length to 18.5 m, an Area Load of 1.0 kN/m² and a wind Angle of 90°.

17] Create new combinations to contain these wind loads. Call these Wind Combination 1 and Wind Combination 2. The appropriate settings for these are shown in the captures below.
Chapter 40 : Add Dimensions : II9

Having added additional loading to your structure you obviously need to redesign the columns and re-check the bracings.

18) **Pick Design/Set Auto Design Mode…** and set all the columns back to auto-design mode.
19) Validate and design your structure again and review the results at your leisure.

### 40 Add Dimensions

A valuable tool in *Building Designer* is the *dimensioning* one. You can create dimensions:

- on plan in a 2D floor view,
- on elevation in a 2D frame view,
- in any direction that you choose in the 3D Structure view.

1) Switch to the *Base* workbook.
2) Pick *Create* from the *Edit* toolbar and *Dimension* from the *Building Objects* one.

3) In the *Base* workbook pick the first of the two points (point 1) between which you want to create a dimension, the appropriate points are shown below.

![Diagram showing points 1, 2, and 3](image)

**Note**

When you are over a point at which you can click, you will see that a small yellow square appears to indicate this and you will also see a tooltip which gives the reference of the point.

4) **Now move the pointer to point 2.**
   As you do this you will see a dashed line following the pointer. This line is attached to the first point you selected and helps you confirm that you picked the correct point.
5] Pick point 2.
Now you will see a dotted line on the screen, parallel to the imaginary line between points 1 and 2. You will also see that the tooltip shows a Distance which is that from the imaginary line between these two points, to the line on which the dimension will be placed.

6] Move to the vicinity of point 3 and click again to set the distance at which you want Building Designer to place the dimension it creates.

Note: There is another way to find dimensions (linear and angular) in your model. To measure linear dimensions pick Grid / Measure, and then click on points on your model, and you will see the appropriate dimension in the tooltip. To measure angular dimensions pick Grid / Measure Angles, and then click on points on your model, and you will see the appropriate angle in the tooltip.
7] You can also add a run of co-linear dimensions to your model. Pick the first of
the points between which you want to create co-linear dimensions, the
appropriate points for this operation are shown below.

8] Now move the pointer to the second of the two points and click it.
9] Now press the Ctrl key down and move to the vicinity of point 3 and click again
to set the distance at which you want Building Designer to place the dimension it
creates.
10] Keep the Ctrl key pressed and click on point 4. As you do you will see an entire
dimension line representation following the position of the pointer.
11] Release the Ctrl key and click on point 5 to generate the run of dimensions.

1. Where the dimension line is always at the same distance from the line connecting the start and end points which
you are dimensioning between.
12] You may want to repeat similar processes to create the dimensions shown below.

41 Controlling Composite Beam Design

In this section we will not delve too deeply into the design detail of individual composite beams, that sort of information is all available within the composite beam Information for Engineers (see “Scope of composite beam” and “Worked Example”). In this example you will focus on controlling and standardising composite beam design for a complete floor system and for this purpose it is worth emphasising two points:

• for any member design it is possible that individual engineering preferences can lead to the selection of one result in preference to another. For individual composite beams this is particularly applicable, beam size, studs, transverse reinforcement, and even assumptions relating to effective flange widths can all interact so that several different designs can be achieved/preferred for one beam.

• once the above is considered in the context of a floor system there is also the possibility that different engineers will apply different preferences as to which beams are composite and which are not.

In this example we will look at a number of factors that you might consider as you make settings to influence a floor design. As noted in the “Summary of Review and Standardisation”, this may seem potentially tedious, but it should not be if you use the information in this example in conjunction with your own experience and expectations to drive Building Designer in the anticipated fashion.

Starting Model

We would always recommend that you perform this sort of exercise on a single floor (or on a representative part of that floor), once your settings yield the floor design that you require, you can use these for the rest of the floor, and then duplicate the floor to other levels, so that you don’t have to design them from scratch later.
In practice you would have performed these investigations much earlier in the design process, so you may prefer to load the example files which we have created, rather than destroy the model you have created so painstakingly.

If you want to continue with your model, you will need to delete the upper floors and construction levels before starting to work on the composite beams.

1] Pick Building/Levels… and delete all the levels until just the Base and First remain, then click OK.

2] Delete the two wind loadcases and their associated combinations.

3] Validate your model and take corrective action to resolve the errors and warnings that occur. Your structure should then look like this.

42 Make every beam composite

This would appear to be the simplest solution - make all the beams composite to get the benefits of composite action throughout.

1] Switch to the workbook for the First floor:

2] Make the Composite Beams attribute set the default one.
3] Pick Set Selection from the Select toolbar.

4] Use the Set Selection dialog to pick Beams whose Type is Simple, and whose Fabrication Method is Rolled as shown below and then click OK.

6] Building Designer selects all the simple beams.

6] Click Apply Attribute Set and OK the application of the attributes to all 31 simple beams.

7] Validate your model. You will find that validation runs to completion, however you do get 19 errors all of which relate to composite beams.

These errors tend to highlight cases where composite design is entirely impossible, or where you need to use engineering judgement to resolve the issues raised. Let's take a look at a few examples now.
Composite Beam with no Slab

There is only one example of this in this model. It seems an obvious thing to avoid, but this is just the sort of thing that can go wrong when you apply an attribute set too sweepingly.

You can not use a composite beam where there is no slab to act compositely with it.

1] Edit the beam and change it back to a simple beam.

2] On the Type page you also need to remove the tick against Fully Restrained.

Note You could also select the beam, and apply the Simple Beams attribute set to it. This set includes the above settings.
Beam Supporting Angled Slab

In actual fact this case, highlighted below, is picked up because Building Designer has not automatically determined the effective width of slab to use for the beam. The acute angle at the left-hand end of the beam means that the effective width (which is normal to the beam) of the beam at this point must be zero. Building Designer therefore requires that you exercise engineering judgement, and specify the width that it is to use.

However if you think about the troughs of the profiled decking¹ crossing the beam at this sort of angle you can anticipate another problem – you will be limited as to the number of studs which you can physically provide on this beam. This problem is not as acute for the other beams around this “curved” perimeter, but change all of these back to simple beams.

1] Make the Simple Beams attribute set the default one.
2] Select the beams round the “curved” perimeter.
3] Pick Apply Attribute Set from the Select toolbar.
4] Click OK to apply attributes to the 6 beams.
5] Deselect the beams.

¹ Which spans parallel to CB 2/4/10-2/9/E)
Beams Supporting Slabs Spanning in different Directions

The beam highlighted below, and the beam above it on the same grid have a slab which spans perpendicular to them on one side and a parallel to them on the other.

In this case you have to exercise engineering judgement and decide whether to idealise the concrete flange (for design purposes) assuming the deck is parallel or perpendicular to the beams. This is your choice, and you need to tell Building Designer what that choice is, but there are a few things that you need to consider carefully:

- there is a lot going on in this area and you need to check at some stage that the beam here can support both decks and that you can fit the studs that each deck requires adequately,
- if you choose to use the perpendicular spanning slab for design purposes you limit the concrete flange depth to that above the deck only so this may seem to be the more conservative choice,
- however if you do choose to use the parallel spanning slab, then the transverse shear requirements may seem more difficult to satisfy and you are more likely to require transverse shear reinforcement.

The issues of deck span and fixing are discussed in more detail in the section “Decking Spans and Fixing”. For now you will set these two beams to use the perpendicular slab (Slab 1) for design, which as the discussion shows is likely to provide an acceptable design. This is the only choice for the highlighted beam since the parallel spanning slab only runs part of its length anyway. In a real world situation you may wish to check the other beam for the alternate choice.
1] Right-click the beam indicated above. From the menu that appears Pick Edit.

2] From the Slab list pick Slab 1. Note that because the slab to the right of the beam doesn’t run the full length Building Designer sets this beam to the Edge condition.

3] Building Designer asks if you want to recalculate the effective width of the beam. Click Yes.

4] Click OK to close the Beam Properties dialog.

5] Repeat this process to use Slab 1 for the beam immediately above the one you have just edited. In this case Building Designer sets the Internal condition since there are slabs to both sides of the beam for its full length.

---

**Beams supporting Tapering Slabs**

For the beam highlighted below, not only are there different slab span directions to each side, but the slab to the right of the beam tapers down to zero width.

1] Repeat the previous process to pick Slab 1 as the one to use for the beam indicated above.
2] This time when you click Yes to recalculate the effective width of the beam you will see that the **Floor Construction** tab shows in error. Click on this tab.

3] Because the slab tapers **Building Designer** will not calculate the effective width automatically. You need to exercise your engineering judgement and assign an appropriate value yourself. Clearly the flange becomes increasingly effective as the moments increase, so in this case it would seem to be reasonable to allow the full effective width of span/4 - the span is 6m, so enter a value for the **Effective Width** of 1.5 m.

**Note** If you let the mouse pointer hover over the **Effective Width** box you get advice on maximum values.
Some Plainly Silly Cases!

This was never intended to be a completely realistic example model, but the beam highlighted below is not supporting any slab – it probably does not even need to be there, and it certainly does not need to be composite.

There are two more short trimmer beams below the slab opening which fall into the same category.

At the moment you have told Building Designer that you want to design these beams as composite ones, and you will get what you ask for, whether it is engineeringly reasonable or not!

Most models include similar beams which may be slightly less obvious cases for preferring non-composite design – typically you would tend to find that relatively short and lightly loaded beams, or beams with very heavy offset loading (such as those supporting a column near an end) could be more economical if you designed them as simple beams. Some examples of this are highlighted later.

1. Edit the beams referred to above and change them back into simple beams.
2. There are several other beams where you either need to pick the slab to use, to set the effective widths manually or both, sort these out now. (In all cases set the effective width to the nearest 0.1 m below the maximum acceptable to Building Designer).
3. Correct your model until it validates with no errors or warnings.
43 Review of Initial Design

4/ Design your model. On completion of the design process you will see that you have five beams with warnings - you can see this is indicated in the Warning colour on your screen.

We would strongly recommend that you investigate warnings after any design process. You are responsible for accepting that your design is not adversely affected in any way.

6/ Right-click the vertical beam (CB 2/C/1-2/C/3) with a warning and pick Design Results from the context menu.
6] You can see that the warning relates to longitudinal shear. Double click the line for that entry. This “drills down” to the details for the longitudinal shear check.

7] In the Details dialog click the line relating to the warning, and the Engineering Tip icon activates. Click this icon to see information for this warning.

The warning reminds you that the beam is treated as an edge beam and therefore should be provided with U-bars. You obviously need to take note of this.

8] Click Close in the Design Summary dialog, this also closes the Details and Engineering Tips… dialogs.

If you were to investigate the warnings on the other four beams, you would find that they all relate to another issue, i.e. the degree of shear connection provided – this will be discussed in detail in the next chapter.
Would some beams be better as Simple Beams?

There are some beams carrying very little load, if you switch the option to show the beam attributes text on, then you will see that you have beams whose size is UB 203 x 133 x 25 – but they are composite and have studs welded on top.

Two such beams are indicated above. Is it possible to make savings by designing these beams as non-composite? Building Designer makes it easy for you to investigate this.

1] In the Show/Alter State dialog click the Non Composite line.
2] Click the two beams above to switch them into Non-Composite mode.
3] Perform the design again - Building Designer checks the same section sizes. On completion of the design the beams which you toggled still pass, so you have saved some studs.

**Note** When you toggle composite beams to treat them as non-composite Building Designer assumes that their top flanges are fully restrained for the composite condition, and no construction stage checks are performed. If you use this feature, but you are not happy with having the top flange fully restrained, then you need to Edit each beam to change its Properties to reflect this. Alternatively you can make the beam a simple beam directly. The attributes defined for the other simple beams in this model were set so that they were (a little unusually) not fully restrained.

The advantage of being able to toggle between composite and non-composite is that the composite design information is not lost – you can change repeat the above process to reinstate the composite design. However, once you have made the decision that a beam is not going to be composite you may prefer to change it to a normal simple beam.

4] Right-click and Edit each of the beams shown above, making them into simple beams, and removing their Fully Restrained setting and setting them to be automatically designed.

5] Validate the structure and perform the design again. This time Building Designer redesigns the beams. On completion of the design the beams which you changed have increased in size to 406 x 140 x 39UB.
Secondary Beams

These are the beams which support the deck spanning onto them – typically perpendicularly. In this model the span of these beams is up to around 7.8 m with a maximum size of UB 356 x 127 x 33. Some of these beams are indicated below.

Building Designer shows the number of studs that are on these beams (the number in brackets after the beam grade) but notice that the number of studs is the same for secondary beams of the same span. This is because the design procedure attempts to provide studs in a series of simple arrangements. In this example the Auto-layout feature has been invoked to provide an efficient design in a ‘uniform’ (as opposed to a ‘non-uniform’) layout.

- **uniform** – for perpendicular decks, studs are placed in every rib, alternate ribs, or every third rib etc. The number of studs in each group is the same along the whole length of the beam.
- **non-uniform** – for perpendicular decks, studs are again placed in every rib, alternate ribs, or every third rib etc. However, knowing the number of studs necessary to achieve the required level of interaction, it is possible that placement at a given rib interval could result in a shortfall; the program will then attempt to accommodate this by working in from the ends. If every rib is occupied and there is still a shortfall, the remainder are ‘doubled-up’, by working in from the ends once more.
In this model, all the secondary beams have 1 stud in every trough (the troughs are at 300 centres). Obviously this is a very easy arrangement to install and check on site.

The number of studs in each group can be controlled by adjusting the limits on the **Studs - Strength** page. If you were to provide more than one stud per rib then you may want to consider adding notes to ensure that the site installation requirements are clear.

**Note**

In many cases the requirement of more than one stud per rib is an indication that a non-composite design may be more economic. Although not shown by this model a good example of this is the case where a large load is applied relatively close to the support of a beam. In such a case the moment has to develop over a short length resulting in the need for many studs over that same short length.

**Note**

With the introduction of Amendment 1 of BS 5950-3.1:1990 you are restricted to no more than 2 studs per trough. The same restriction also applies when designing to Eurocodes.

So for this model the initial results seem very reasonable for all the internal secondary beams.
Primary Beams

These are the larger main beams which in this model are typically supporting secondary beams at third points.

Notice that there are different numbers of studs on each beam. Also notice that all sorts of different sizes and spacings of transverse reinforcement have been calculated in the design of these beams. In a nutshell the design of these primary beams has been optimised for each beam but not standardised for the floor. This might be acceptable, but it could lead to:

- an increased chance of incorrect installation,
- an unnecessarily time-consuming checking process on site.

To standardise the design for the floor you need to make some changes to the composite beam attributes so that you take greater control over the design.

1) In the Workspace double click the name of the Composite Beams attribute set.
2) Click the Reinforcement tab.
The attribute set has Bar spacing as a multiple of stud spacing ticked. Since you have not forced any standard stud spacing this means that the reinforcement spacing will also be irregular.

3] Untick the Bar spacing as a multiple of stud spacing option and then click the Design tab.
4] On the Design page click the Design Properties (...) button and then click the Reinforcement tab.

This page allows you to control the bar diameters and spacings that Building Designer will consider for reinforcement during the design process. You can use these settings to cut down the choices of bar diameters and spacings. Bear in mind that an A type mesh is being used as general anti-crack reinforcement – it has bars at 200 mm spacing. If you stick to multiples of 200 mm spacings for any additional reinforcement then it will be easy to install and check.

6] Remove the ticks against all but the 10.0 diameter bar, and the 200 and 400 spacings. Click OK to close the Design Properties dialog.
6] Click OK again to close the Attribute Set dialog.
7] Right click the Composite Beams attribute set and pick Set as Default from the context menu. You now want to apply this attribute to all the composite beams on the floor.
8] Pick Set Selection from the Select toolbar.

8] Use the Set Selection dialog to pick Beams whose Type is Composite, and whose Fabrication Method is Rolled and then click OK.
10] Since all our composite beams are rolled sections Building Designer selects all of them.
11] Pick Apply Attributes from the Edit toolbar and Building Designer asks you to confirm the application of the attributes to 38 beams. Click OK.
Validate and design your structure. The result is as shown below.

You now have regular transverse reinforcement along with a minimum number of studs on the primary beams.

So far, the stud spacing on the beams has been determined automatically by making use of the Auto-layout feature. The way in which this is applied to parallel decks is as follows:

- **uniform** – (as in this example) - studs are placed at a uniform spacing along the whole length of the beam. If the point of maximum moment does not occur at mid span, the resulting layout is still symmetric.

- **non-uniform** – if optimization has been checked studs are placed at a suitable spacing in order to achieve sufficient interaction without falling below the minimum allowed by the code. If optimization has not been checked, studs are placed at a suitable spacing in order to achieve 100% interaction. If the point of maximum moment does not occur at mid span, the resulting non-uniform layout can be asymmetric.

You will now look at how Building Designer also allows you to control the stud spacing manually.

In the Workspace double click the name of the Composite Beams attribute set.
14] Click the Connectors - Layout tab and remove the tick against Auto-layout.

In the perpendicular direction the design is generally picking a stud in every rib, so you decide to enforce that setting everywhere.

In the parallel direction a quick calculation on one of the primary beams (9 m span, approximately 40 studs) reveals that the studs must be at just over a 200 mm spacing. Bearing in mind that an A type mesh has bars at 200 mm spacing you decide to try applying a 200 spacing throughout. This is going to make checking on site a simple visual exercise.

15] Make the appropriate settings (shown above) and then click OK.

16] Repeat the process you used above to apply the attribute set to the 38 composite beams.
Validate and design your structure. The result is as shown below.

This change has some interesting side effects. For all beams where the decking is parallel you now have studs at 200 centres – for some beams e.g. CB 2/A/3-2/B/3 this is insufficient, (this shows the value of letting the software provide the optimum design in the first pass). For some edge beams H10 at 200 centres is no longer enough and they have started to fail.

To correct the design for CB 2/A/3-2/B/3 right-click on the beam and pick Edit... from the context menu. On the Design tab set the beam back to automatic design and on the Connectors - Layout tab select a uniform auto-layout.

Click OK then validate and design once more.
You will see that some of the beams still have a warning status.

20] **Right-click CB 2/8/3 - 2/C/3**, and then pick **Design Results from the context menu.**

In this case the warning is issued for the shear connectors.

21] **Either click the Shear Connectors tab or double-click the Shear Connectors line.**

22] **Click the line relating to the warning, and then click the Engineering Tip icon.** There are several reasons why this warning might arise, scroll through the tip, and you will see the condition which is creating the warning in this instance (if you read the tip through, then you will see that none of the other conditions apply).

23] **Close the Engineering Tips... dialog, and return to the Shear Connectors check details, (noting in passing that the position of the check is at 3.000 m).**
As you can see the degree of shear connection achieved at this point (3.000 m) is 0.395. This generates a warning because it is less than 0.4 which might be regarded as the lower limit specified by BS5950:Part 3:clause 5.5. However, this clause indicates that the minimum requirement applies to the point of maximum moment and does not indicate a need to check interaction levels achieved at other points.

For this beam the maximum moment does not occur under the point loads, as a result of the small influence of self weight it occurs at mid span (4.500 m) and if you check the design moment at this position the degree of shear connection achieved is 0.58. However, the moment capacity at any point is dependent on the interaction ratio and the critical moment capacity utilization ratio is found to exist under the point load at 3.000 m (with an identical case at 6.000 m) along this beam. It may be arguable whether the code requires you to check the degree of shear connection at any point other than the maximum moment, but Building Designer will always check it at both the point of maximum moment and the point of maximum moment capacity utilization. If the degree of shear connection drops below 0.4 at the point of maximum moment the check will indicate a fail. If it drops below 0.4 at the point of maximum moment capacity utilization the check will indicate a warning (as in this case).

Note that on almost any beam (even a beam with a simple UDL) it would be possible to find other sections (particularly near to the supports) where the degree of shear connection drops below 0.4. In fact, even in this model you can select beam 2/A/4 – 2/B/4 and examine the shear connector calculations.

1. (if you see a value of 0.40, then you will need to change the precision for ratios in the Preferences to 3 decimal places).
When the check at the point load position of 6.617 m is examined, the check details show that the degree of shear connection from this point to the end of the beam is 0.369. This is not deemed worthy of any warning as this position is not the point of maximum moment, or the point of maximum moment capacity utilization.

In the case of the composite beam with the warning, including an extra stud within the length being checked would be sufficient to increase the ratio beyond 0.40, and thus you might exercise engineering judgement and choose to ignore the warning.

**Edge Beams**

To cure the problems with the edge beams created by the steps you took above you could create a different attribute set for the edge beams where the studs are at say 400 centres. However for edge beams transverse reinforcement is particularly important and needs to be detailed and provided in the form of U-Bars (for further information see "Decking Spans and Fixing", and in particular the notes on the assumptions relating to lap positioning). Potentially this is more costly and, some would argue, is more prone to incorrect installation. Where height permits many engineers will choose to use non-composite beams at the slab perimeter to avoid this. Other engineers will try to make edge beams non-composite to improve flexibility for future change of use – difficult to allow for future openings adjacent to edge composite beams.

Taking the above considerations into account you decide to change the edge beams back into simple beams.

1. **Apply the Simple Beams attribute set to the 9 beams shown below.**
Validate and design your structure. You should get the result shown below.

The beam sizes have obviously increased, but there are now no studs or transverse reinforcement to consider at the perimeter. If you wanted to limit the beams to the 533 UB range, then you could have applied depth limitations to the simple beams to ensure this.

Summary of Review and Standardisation

During this example we have shown that:

- It is not logical or economic to make every beam a composite beam.
- If you do not constrain the design process using attributes Building Designer will tend to produce optimum design for composite beams in that the stud numbers and areas of transverse reinforcement provided will be reasonably minimised.
- By applying more constraints in the attributes you can achieve more standardised arrangements that will be simpler to install and simpler to check on site, clearly the penalty for this is increased material usage.
- However, an advantage of such standardisation is that you will tend to have introduced more flexibility to introduce openings and so on, if future re-fits require it.

If you were to optimise your design in the sort of step by step manner that has been shown in this example, then this may seem to be a lot of work. However, whether you produce hand calculations or use a computerised design your experience as an engineer will enable you to
skip most or all of these steps. By examining the steel layout you can decide in advance which beams should be composite and which should not. For the composite beams you can then define different attribute sets for different beam types, for example:
  - Secondary Beams (Perhaps forcing the studs/trough setting and the type and spacing of transverse reinforcement if it is required and if that is your preference.)
  - Internal Primary Beams (Perhaps forcing the stud spacing and the type and spacing of transverse reinforcement if it is required and if that is your preference.)
  - Edge Beams? - If you want these to be composite you might use different attributes than apply to internal beams.
  - You might make different attribute sets for major and minor beams in each of the above categories if there are significantly different span/loading conditions.

We hope that by considering the points made in this quite detailed example you will be able to apply your own experience and expectations more quickly and effectively and therefore become more productive in your use of Building Designer.

45 Decking Spans and Fixing

Earlier in this example we noted the case where the decking spans change at a beam support line and in such cases we chose to assume that the perpendicular decking would be assumed for design purposes. This is likely but not certain to be a conservative assumption, and although we will focus on a specific case, this discussion is not restricted to situations where one deck is truly perpendicular and another is truly parallel.

The particular case we are going to consider in more detail is highlighted below.
For this beam, for the purposes of design, you chose to assume that the decking was perpendicular to the beam. To decide on the validity of this design the assumptions behind it must be clear, to find these out you need to review the beam’s design properties.

**Note** Fastrak does not check or design the slab.

### Decking Attachment

1. **Edit** the beam shown to access its **Properties**, and then click on the **Profiled Metal Decking** tab.

On this page the decking is set to be **Effectively attached** (the box is checked). This relates to the requirements indicated in BS5950: Part 3: clause 5.6.4, in which effectively attached sheeting can contribute to the transverse shear reinforcement required to ensure the concrete flange maintains its integrity.

Also note that the decking is not assumed to be **Continuous**. This has the effect of reducing the extent to which the decking can contribute to longitudinal shear capacity. It is also worth noting that where the spans are discontinuous at a beam line then **Effectively attached** sheeting must be effectively attached on both sides - that is the studs must be staggered and alternately welded through the sheets on each side of the beam. This requirement therefore applies even in the simple case where there is a butt joint between perpendicular spanning decks on both sides. (For further guidance refer to BS5950 and also to industry guidance such as SCI publication P300.)

The full longitudinal shear capacity is provided by several constituent parts:

1. The concrete itself.
2. Mesh reinforcement in the slab if you deem it to be effective and have defined it as part of the slab attributes.
3. The decking (if it is effectively attached).
4. Additional Transverse Reinforcement if the sum of the above is not sufficient.
What happens if we take a more conservative view of this beam and do not assume that the decking is effectively attached. Let’s look at the current results.

2/ Right-click the beam and pick Design Results from the menu, then drill down to view the longitudinal shear results.

The results show that the longitudinal shear resistance is 243.5 kN/m and when the details are examined this includes a contribution from the decking of 79.8 kN/m. Note that the decking contribution would be considerably higher if we had assumed it to be continuous, but normally this sort of information is not known at the design stage.

3/ Close the Design Summary, Edit the beam and change the Profiled Metal Decking so that it is no longer Effectively attached.
4] Right-click the beam and pick Design Results from the menu, then drill down to view the longitudinal shear results - this time without the decking effectively attached.

The results now show that the longitudinal shear resistance drops to 163.7 kN/m and when the details are examined this is because the contribution from the decking is now 0 kN/m.

In this particular example the beam still works but in many cases making the conservative design assumption that the decking is not effectively attached would have introduced the need for transverse shear reinforcement.

Change Assumed Decking Span

If the decking is effectively attached and you do not want to make the conservative assumption that it is not, you can consider checking the beam assuming that the parallel decking condition applies.

1] Edit the beam shown to access its Properties, and then click on the Profiled Metal Decking tab.

2] Set the decking to be Effectively attached.
3) Click on the Design tab.

4) Set the Slab to Slab 2 and click the Profiled Metal Decking tab.

Note: Although the Effectively attached setting is removed from the page, it is still in force. If you had not changed the setting previously, then Building Designer would treat the decking as the distance to the lap were zero, irrespective of the value you specify here.
5] Ensure that the Minimum distance to lap is set to 0 mm (the default) and click the Connectors - Layout tab.

6] For the sake of this comparison you decide to keep the same number of studs as on the other beams in the run (20), you therefore adopt single studs at a spacing of 300mm (studs at 150, 450, 750, 1050, …, 5550 and 5850). To achieve this layout click Insert, uncheck the No. of studs box and then enter a Group spacing of 300 as shown above.

7] Click OK.
Review the longitudinal shear design results for the beam which are now as shown below.

The longitudinal shear resistance has gone up to 454.7 kN/m. This is much higher than the 243.5 kN/m for the perpendicular case because at this point the deck is assumed to be continuous. However, there is now an extra check at the decking lap position. Where the decking laps the decking no longer contributes to the shear resistance - refer to BS5950: Part 3: 5.6.4. At design stage the lap positions are rarely known and so a conservative default lap position is assumed - 0 mm offset indicates that the lap occurs right beside the studs and so we are effectively checking for the condition where the deck on one side of the beam is not effectively attached.

In this case notice how the concrete sectional area has dropped to 100000 mm$^2$ for the parallel condition - the minimum concrete section above a trough is used. For the perpendicular condition it was 126667 mm$^2$ - the total concrete section is used. This results in the capacity at the lap dropping to 139.7 kN/m.

Once again, in this particular example the beam still works but in many cases the assumption that the lap occurs at 0 mm offset will introduce the need for transverse shear reinforcement.

For your example beam, if the parallel decking is going to be effectively attached then can you ensure that the offset to the first lap will be a sheet width away, or at least a large proportion of a sheet width way. Let’s assume that you can ensure that the first lap will be at least 300 mm offset – what effect does this have on the beam design?

Edit the beam Properties and change the Minimum distance to lap to 300 mm.
10] Review the revised Design Results.

The capacity at the lap is unchanged at 139.7 kN/m, but the force at the lap has dropped to 80 kN/m. Basically the lap force drops linearly to zero as the lap position is moved out from the beam centre line to the edge of the allowable concrete flange.

Therefore, provided you intend to ensure that the first parallel sheet adjacent to any beam is effectively fixed to the beam the critical lap position can be moved and the requirement for additional transverse reinforcement is significantly reduced or eliminated. For this and most examples this would mean that the original assumption of using the perpendicular decking and assuming effective attachment is reasonable.

**Note**

The advantage of assuming the perpendicular case is that the number of studs will relate to the number of troughs. One stud per trough in this example will mean that the perpendicular deck is fixed down in alternate troughs and that the parallel deck is fixed down alternately at the same spacing. If the parallel deck were assumed then as has been shown elsewhere in this example the design will determine a minimum number of studs that will not relate to trough centres and so effective attachment may become very awkward to achieve.

**Practicalities**

The BS and SCI guidance indicates minimum edge and spacing distances relating to the through deck welding of studs. Where sheeting is discontinuous with a simple butt joint occurring on top of a flange the recommendations indicate a minimum flange width of about 135 mm. It is perhaps worth considering applying a minimum flange width when designing such beams, and perhaps where decking is meeting at angles the minimum width should be increased to allow more tolerance for construction.

Congratulations you have now finished this extension to the Quick Start Tutorial.

We hope that you have found the additional features it has covered both interesting and helpful.
46 Other information

Select toolbar
The operations you can perform with the Select toolbar (working from left to right) are:
• Select,
• Clear Selection,
• Set Selection,
• Show All/Show Only Selection,
• Selection Groups, and
• Selection Group Filter,

Edit toolbar
The operations you can perform with the Edit toolbar (working from left to right) are:
• Undo
• Redo
• Create,
• Apply Attribute Set,
• Modify,
• Split and Join,
• Copy Attributes,
• Delete,
• Remove from sub-structure, and
• Show/Alter State.

Building Objects
The available building objects are:
• Grid,
• Column,
• Beam,
• Truss Member,
• Brace,
• Support,
• Slab,
• Slab overhang,
• Roof,
• Wall,
• Shear Wall,
• Shear Wall Extension,
• Shear Wall Opening,
• Truss,
• Portal Frame,
• Parapet,
• Dimension, and
• Inclined Plane.

Load Types
The available load types are:
• Floor Load,
• Area Load,
• Point Load,
• Line Load,
• Patch Load,
• Variable Patch Load,
• Nodal Load,
• Element Load,
• Wind Load, and
• Wind Zone Load.