Disclaimer

CSC Inc. 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 Inc.)
• 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 Inc. Software Licence Agreement.

Proprietary Rights

CSC Inc. 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 States.

Fastrak documentation: © 2009 CSC Inc. All rights reserved.
Fastrak software: © 2009 CSC Inc. All rights reserved.
Trademarks

Fastrak™ is a trademark of CSC Inc.
TEDDS™ is a trademark of CSC Inc.
The CSC logo is a trademark of CSC Inc.

Autodesk and Revit are registered trademarks or trademarks of Autodesk, Inc., in the USA and/or other countries.

Microsoft and Windows are either trademarks or registered trademarks of Microsoft Corporation in the United States and/or other countries.

Acrobat® Reader Copyright © 1987-2009 Adobe Systems Incorporated. All rights reserved. Adobe and Acrobat are trademarks of Adobe Systems Incorporated which may be registered in certain jurisdictions.

All other trademarks acknowledged.
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, analyze 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, you probably want to get the most out of Building Designer in the shortest possible time. To help you move from being a Building Designer "novice" to a Building Designer "expert" we would strongly recommend that you book yourself on to our Building Designer on-line training courses.

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 license Building Designer at this stage then do so. The Licensing help explains this process fully.

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

2. Close the Licensing dialog. Building Designer will start and initialize.
3 Introduction

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

When you have a project open, the screen will appear as shown above. You can customize 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.
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 tutorial building](image)

**Plan View**

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

Design Data
The example building is a 2 x 24 ft bay by 2 x 22 ft bay composite building which has:
• floor levels at 12’ and 22’ and an eaves level at 32’
• a sloping roof
• beams at ⅓ points
• edge girders which are of non composite construction
• internal girders and beams of composite construction

Loading Information
• Construction Stage; S.W of Slab (Wet) = calculated automatically
  Construction Live = 20 psf
• Composite Final Stage; S.W of Slab (Dry) = calculated automatically
  Live = 100 psf
  Roof Live = 20 psf
  Wall load-edge beams = 0.7 kip/ft

Composite Data
• Slab Depth = 5 in
• Concrete Strength = 4000 psi
• Reinforcement (slab) = Mesh A142 as necessary
• Decking = Vulcraft 2VLJ 24
• Gage = 20
• Standard Stud Layout,
  • Stud Height = 4 in
• Composite Beam steel grade = ASTM A992 Gr. 50

Non-Composite Beam Data
• Steel grade = ASTM A992 Gr. 50
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 pad 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 ….
1. Enter the project details as shown above.
2. 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. Thus it is effective in many instances to completely create a single floor, and then use this as a template for the other floors to which it is similar. 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 rest of the structure.

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 2 wizards which allow you to define entire grid systems quickly and easily. The two grid system types that these wizards allow you to create are:

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

\(^1\) Are automatically duplicated on.
1/ 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).

Note To help you visualize 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.

2/ Click Next > to accept these 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) Check that the settings are as shown above and then click Next >. The next page of the Wizard allows you to define the grid lines in the x-direction.

4) Make the settings shown above to define 2 bays at 24' and then click Next >.

5) To define the grid lines in the y-direction, enter the details shown above to define 2 bays at 22' and then click Next >.

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

Example

If you enter 0', 26', 39', 52' then Building Designer will create grid lines at 0, 26, 39, and 52 feet 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 feet.

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.

6] 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.

7] 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 will now add a new grid line which is mid way between grid lines 2 and 3.
Chapter 6: Create Grid Lines: 17

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 and the distance to the new parallel grid line.

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

Again, either enter the Distance, 8’ 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 2).

3] 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.
Chapter 6: Create Grid Lines

You should now have this grid arrangement.

Customize 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 7—it was the model’s 7th grid line. You only created this grid line to help you create a single infill beam, 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 Building Objects one.

2] Pick grid line 7 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.

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.
Chapter 7: Attributes

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 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.
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 to Column at 0 deg. 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
* any Design Properties (constraints) you wish to apply to this element.
Chapter 8 : Place Columns : 21

5] Click Design Properties…

You can customize the sections to be considered for design as shown above.

Note Because the sections to be considered can be set differently based on user preference, your design may adopt different sizes to those shown later in this guide.

6] Click OK.

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

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

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

9] Click OK to close the Attribute Set - Column dialog and then OK once more.

You will see the name of the first attribute set change in the Workspace window.

8 Place Columns

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 Building 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 Building 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 default level, 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).

3] Release the mouse button, and Building Designer creates columns at every grid intersection point as shown below.

These columns have their webs at $0^\circ$ – the setting in the current attribute set.
Delete superfluous columns

Review the plan for the model (see page 9), and compare this to the capture above, 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 Building Objects one.

Note When you are deleting elements and put the pointer over an element you can currently delete, Building Designer changes the pointer’s shape and the element’s color.

You can left-click and drag a window around a series of columns to delete them as follows:

2/ Position the pointer just below and to the left of the first surplus column.

3/ Click and hold the left mouse button.

4/ Drag the pointer till it is just above and to the right of the last surplus column.

5/ Release the mouse button.

Building Designer asks you to confirm that you really do want to delete the 3 columns.

6/ Click OK.
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 Building 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 attribute set. You might 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/ To keep the data held within the attribute set consistent with its name, 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 Building Objects one.

2] Select the column which lies at the intersection of grid lines 1 and B.

3] Now look at the Properties window, and you will see this column’s details. 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.

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

5] 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 9 selected columns, and the blanked details are inconsistent for some or all of these.

**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. You will do this later in the example.

6. Click the Clear Selection icon to clear the selection of all the columns.

**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 grid intersection A1 from the Workbook, Pick Edit GC A/1 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 both the Braced (Comp) and the Braced (LTB) tab are highlighted in amber, which indicates that there is a situation which needs your attention.
28 | Chapter 9: Create Beam Attributes

2] Click the Braced (Comp) tab to see what the problem might be.

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

3] Click Cancel.

9 Create Beam Attributes

Your model contains three types of beam – Composite, Non-Composite and General. You will define the attribute sets for these now.

Note

In Building Designer - the term *simple beam* specifically refers to a non-composite simple span beam. Also, no distinction is made between a beam and a girder - they are both referred to as a beam.

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.

Non-Composite Beams

<table>
<thead>
<tr>
<th>General</th>
<th>Attribute Set Title</th>
<th>Ticked</th>
</tr>
</thead>
<tbody>
<tr>
<td>Design</td>
<td>Automatic Design</td>
<td>Simple</td>
</tr>
<tr>
<td>Type</td>
<td>Fully braced</td>
<td>Not ticked</td>
</tr>
<tr>
<td>Size</td>
<td>Grade</td>
<td>A992-50</td>
</tr>
</tbody>
</table>

Note

In this example model, for attribute settings that are not specified you can accept the default values.

1] Once your attribute set details are correct click OK to create it.

2] Create a new set for the composite beams in your model (click Add…). The details are given below.
Chapter 10: Open the view of a floor: 29

Composite Beams

• General
  Attribute Set Title: Composite Beams

• Design
  Automatic 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: A992-50

• Studs - Standard Layout
  Stud Height: 4 in

Note: In any composite beam design personal preferences in relation to shear studs and transverse shear reinforcement mean that different engineers would prefer different answers in identical situations. Within the attribute setting options it is possible to apply these preferences and customize the automatic design to suit your personal or company taste. This more advanced customization is documented elsewhere.

1/ When you have set the details shown above click OK to return to the list of attribute sets. (Once again, other attribute settings that are not specified above can be left at the default values.)

2/ Create a new set for the roof beams in your model (click Add…). The details are given below:

Roof Beams

• General
  Attribute Set Title: Roof Beams

• Design
  Automatic Design: Ticked
  Construction Type: General

• Beam
  Grade: A992-50

1/ Once your attribute set details are correct click OK to create it.

2/ Click OK to close the Attributes Sets - Beams dialog.

10 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 next level, (currently named Roof, although you will rename it First Floor later). 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 customize 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.

The initial floor plan you are going to create is shown below.
You will create all the beams in your model initially as simple non-composite beams. 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).

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

You will see that the Non-Composite Beams attribute set 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.

**Note**

To switch to a different attribute set, right-click over the attribute set required and choose Set as Default.

### Place Beams

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.

**Place beams by the area method**

1. Pick Create from the Edit toolbar and Beam from the Building Objects one.
2] With the left mouse button click and drag over the entire grid area in the Workbook window (as shown below).

3] Release the mouse button, and Building Designer creates beams between all the columns as shown below.
Copy Elements

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 two rows of infill beams.

1/ 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.

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.

2/ Click over the beam that runs between grid intersections A1 and A2 (the beam will turn green to show that it is selected and its reference will appear in the Source Elements dialog).

3/ Click the beam between A2 and A3 (again it will go green and its reference will be added to the list in the Source Elements dialog).

4/ Repeat this for the two beams that lie along line B. The source elements dialog will now show the references of the four selected beams. These four beams will be included in the copy process.
5] Now you have picked the beams click Next > to continue with the copy process. You will see the 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.

6] 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).

7] 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.

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.

8] 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.

As well as mirroring you can also rotate the copies through any angle that is necessary.

9] For this example you don’t need to do any rotation, so click Next > to proceed to the final stage of the process.
10] 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 you can pick multiple targets for the copies, in this instance you need to replicate the 4 copied beams twice, 8 ft and 16 ft further right.

11] Move the pointer over the first beam on grid line 1 until the snap point for 1/3 of the member length is located as shown.
12] Click on this point and Building Designer adds this to the list in the Target Coordinates dialog. (Note that Building Designer shows a line representation of the elements that you are copying to help you visualize what you are achieving with your copy.)

13] Continue moving the pointer along the same beam until the snap point for 2/3 of the member length is located. Click on this point to add a second row to the Target Coordinates dialog.

14] To create your copies click Finish.
Place beams by start and end point

Individual beams can also be created by selecting 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 create the remaining beams in this way.

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

2/ Click the beam start point (C7).

3/ Click on the beam to the left (SB 2/2/23-2/3/23).

Snap points appear at quarter points, third points, member ends and also at the same Y point as the start of the beam.
4] **Click on the snap point Same Y point.**

Building Designer then creates the beam between the two points.

It is not necessary use grid lines at all to place members into your existing structure, as will now be shown.

5] **Click on the edge beam (SB 2/C2-C3).**

Again snap points appear at quarter points, third points, member ends making it easy to place a new beam. If you want to connect to a different point, this can be done also.
Instead of clicking on one of the displayed snap points start to type the distance you require, (16'). Building Designer pops up a dialog where you type the distance.

Click OK and then click on the beam to the left to define the beam end point, using the snap point Same Y point.

Building Designer then creates the beam between the two points.

Prior to creating the first slab in your building you will first define the required slab attributes.

Edit the existing Slab Attr 1 attribute set. Give it the title Vulcraft 2VLI 24 at 0 deg and set the following details: (other details can be left with the default values).

- **Angle of slab** = 0° (in relation to the x-axis)
- **Overall Slab Depth** = 5 in
- **Decking** = Vulcraft 2VLI 24
- **Gage** = 20
From the Plan View on page 9 you will see that your building has 2 separate slab areas spanning in different directions. You will create the two areas using the attribute set you created above, and then change the angles as required.

**Note** 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.

2/ Pick Create from the Edit toolbar and Slabs from the Building 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.

3/ Zoom the view of the Roof so that you can see it in its entirety.
4/ Set the View Options so that Building Designer shows the Text for the Slab Name.
5/ To place the first slab area click and drag as shown below.

6/ Next click inside the remaining zone at the right hand between grids 1 and 2 to place an additional panel spanning in the same direction.
7] 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 Building Objects toolbar.

8] Click in the steel bounded areas to add Slab 2 as shown below.

9] Right-click on the Slab 2 area and pick Edit from the context menu.
10] In the Slab Properties dialog set the angle of the slab to be 90°.

11] Click OK.
You will see that all areas of Slab 2 take the new angle, so that your slab arrangement should now be this.

13 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 (X) then you cannot 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 summarized on two separate pages. Review these now.

You can see that there is a single error and a single warning - both related to load combinations.

**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.

---

### Chapter 14: Apply an Attribute Set to a Selection

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

**Change to Composite Beams**

1/ Select the 11 non-composite simple beams which lie totally within the slab area as shown. (Do not select the beams around the slab opening.

**Note** If you only want to select those beams which are totally encompassed by the boxes shown below, then ensure that you don’t press the Shift key as you release the mouse button.
Pressing the Shift key selects all objects which are totally encompassed by a box, and also those which it crosses.

2) Use the View Options dialog to switch on the display of the beam names.
   Note All the beams currently have the prefix SB indicating they are simple beams.

3) In the Workspace, right-click over the title of the Composite Beams attribute set.
4) Pick Set as default from the context menu.

5) You will see that the attribute set’s title is now shown in bold text which indicates that it is the one in use.

6) Pick Apply Attribute Set from the Edit toolbar.

7) Confirm that you want to apply this attribute set to 11 beams.
   The prefix on the selected beams changes to CB indicating they are now composite beams.

8) Now click the Clear Selection icon to de-select the beams.
9) Validate your model again. Two new issues should arise.
10) Expand the Invalid composite (1) error so that you can double click on the invalid beam reference. This will locate it in the plan view.

   Building Designer immediately highlights the element associated with the error in blue, and (in case the member’s representation is small) points to it with a blue arrow.
Chapter 15 : Create Loadcases

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

11] Right-click on the beam in the plan view and pick the Edit context menu option. You will see the beam’s Properties.

![Beam Properties](image)

In the composite beam calculations, the program is restricted to considering one configuration of steel deck per beam only. For the majority of the beam length in question the deck in Slab 1 spans parallel to it, although for a portion of its length, Slab 2 is actually perpendicular. You need to specify which slab should be used in the beam calculations.

12] Choose Slab 1 from the drop down list and then click Yes to indicate that you wish to recalculate the beam effective width.

13] Click OK to close the beam’s Properties.

15 Create Loadcases

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

![Loadcases Icon](image)

You will see the Loadcases dialog.

![Loadcases Dialog](image)

From the Design Data details (see page 10) you can see that you need 6 loadcases in addition to the steel self-weight which Building Designer determines automatically):

- Construction stage – Slab Wet, and Construction Live,
- Composite stage – Slab Dry, Live, Wall and Roof Live.
18] Click Add… and define the Slab Wet loadcase name and type. Also check the box as shown so that Building Designer determines the load automatically.

16] Click OK to create it.

17] Repeat this process to define the Slab Dry loadcase in the same way.

18] Click Add… once more and define the Construction Live loadcase’s name and type.

19] Click OK to create it.

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

20] Repeat this process to define the Live loadcase. For this loadcase tick the Reductions box. (You will have to set the loadcase Type to Live first.)

21] Repeat again to define the Wall loadcase. (Set the loadcase Type to Dead.)

22] Finally create the Roof Live loadcase. Set the Type to Roof Live and again tick the Reductions box.

23] You will now see all your loadcases listed in the Loadcase dialog.
Chapter 16: Create Combinations

24) Click OK to close the 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.

Note: If the Floor Loads icon is grayed out, this could be because you have got either Floor Loads or Slabs switched off in View Options.

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

4) Enter the value of the load (20 psf) and then click OK.

5) Create the Live loadcase in the same way, using a floor load value of 100 psf.

Define Perimeter Loads

You will now define the loading within the Wall loadcase which applies to the wall load on the edge beam.

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

2) Pick Loading/Create Perimeter Load...

3) Enter the load value of 0.7 klf and then click OK. Building Designer creates element loads on all beams around the edge of the building.

16 Create Combinations

4) Pick Loading/Combinations…. or click the Loading toolbar’s Combinations icon.
You will see the Combinations dialog.

The first requirement is to create a combination for the construction stage loading.

6] Click Add Constr.

You will see that the steel Self weight and the Slab Wet loadcases are 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 complete.

7] Click OK to return to the previous window.
8] Now click Generate… to create the remaining gravity combinations.

Because the ASD code is being used, four load combinations are available. At this stage you can use engineering judgement to filter out those combinations that are unlikely to be critical for this type of building.

9] Untick the boxes for the first and third combinations and then click Next >
10] The next page of the dialog sets up the service factors.

11] Click Next >
The next page allows you create further lateral load combinations in which notional horizontal forces have been added. In this example we add these notional horizontal forces to the two combinations we are currently designing for gravity. Check the boxes as shown below - this will result in an additional eight lateral combinations being created,

12] **Click Finish.**

Eleven 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 then checked against all active combinations during the **Full Design** process.

13] **Check all three Gravity Combinations and the first Lateral combination as Critical before clicking on OK to close the Combinations dialog.**

14] **Validate your model again.**

You should only have one warning, indicating that a roof live loadcase has been defined but that it is currently empty.
17  Create Other Floors

Having completed your template floor, you can proceed to complete the rest of the structure.

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.

You will see your model in isometric view.

3] Pick Building/Levels…

The Construction Levels dialog lets you control the number of stories in your model.

Building Designer automatically created the pre-defined floor levels – Base (at a level of 0 feet) and Roof when you created a new model.

4] Click the line for the existing Roof level.

5] Click Insert Above three times to create 5 levels in all.

6] Define the upper floor levels at 12’, 22’, 32’ and 40’ and change their names to First, Second, Roof and Apex.
7] Only tick the boxes for floors at the First and Second floor levels. These levels should also contain single diaphragms as shown.

Don’t forget to untick the Floor box at the Apex and Roof levels.

8] Click OK to create the new floors.

Now to copy your template floor to the floor above.

9] Pick Building/Copy Floor…

10] Set the Source Level to First and indicate Second as the destination level by ticking the appropriate Copy To box.

11] Click OK.

Building Designer copies all the details from the first to the second floor, you can see this in the Structure window.

18 Extend the Columns

You now need to extend the tops of the columns to the correct levels.

1] Still in the Structure window, click the Column icon in the Building Objects toolbar and then select the 6 perimeter columns along grids 1 and 3.

2] Pick Building/Column Levels…

3] Specify the Level Start as Base and use the scroll buttons at the right of the box to set the Level End as Roof then click on OK.
4] Pick Select/Clear Selection to de-select the perimeter columns.

5] Repeat the above for the 3 internal columns on grid 2, only extend them to the Apex instead of the Roof level.

6] Again, pick Select/Clear Selection to de-select the columns.

**Note** For clarity while the remaining steel is being defined, in the above and subsequent views Slabs are switched off in View Options.

19 Add Roof Beams

Still in the Structure window, you will now define the roof beams.

1] From the Workspace locate the beam attributes and set Roof Beams as default.

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

3] Click the first roof beam start point (A1)
4] Click the beam end point (A2)

5] Building Designer creates the beam between the two points you identify

6] Create 5 more inclined roof beams in a similar manner.

7] From the Workspace set Non-Composite Beams as default for beam attributes.

8] Create two beams running horizontally along the apex and four beams along the two sides of the building to restrain the tops of the columns.
Chapter 20 : Create and Load the Roof

9] Validate your model again.
   The validation message informs you that the loadcase Roof Live is empty.

20 Create and Load the Roof

You will now create the two roof slopes and add the Roof Live loading.

1] Pick Create from the Edit toolbar and Roof from the Building Objects one.

2] Click the first corner of the first roof slope (A1).

3] Click the second corner of the roof slope (C1).

4] Click the third corner of the roof slope at the apex (C2).

5] Click the forth corner of the roof at the apex (A2) and then to complete it click once more on the first corner (A1).

6] The first roof should be created as shown below.

7] Repeat the above process to create the second roof slope.

Load the Roof

Now that you have defined the roof you can add the load to it.

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

Note If the Area Loads icon is grayed out, this could be because you have got them switched off in View Options.

3] Click anywhere over your first roof slope.

4] Enter the value of the load (20 psf) and then click OK.

5] Repeat the above for the second roof slope.

Note 20psf is added to the horizontal projection of the sloped roof.

21 Create Frames and 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 Hollow Section Braces. From the Design tab specify Automatic Design and click Design Properties to check that the orderfile used is RoundHSSOrderAISCimp. Also on the Size tab ensure that Hollow Section geometry is selected.
2] In the Base - 2D workbook select grid lines A, and 3 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 A, and then set the options from the toolbars which allow you to create braces.

8] Create the bracing system shown below for Frm A, and then repeat a similar process to create the bracing systems on Frm 3
Note

Ensure that each brace is defined starting at the lower floor and ending at the higher floor. This defines the brace local axis so that end 1 is always at the lower floor.

1] Validate the structure.

2] The validation warnings indicate that the brace connections to the mid-point of the beams could induce axial load and/or uplift in these beams. This can be resolved by vertically releasing the appropriate end of the braces.

Note

Alternatively, making the beams general as opposed to composite resolves the warning, as this enables them to be designed for the axial load and uplift forces from the braces.

3] Select all 6 braces in frame 3 and use the Properties window to vertically release end 2 of them all as shown.

4] Use the Building/Copy Elements command (see page 34) to copy the three bracing members from Frame A to Frame C.

5] Validate the structure again.
Chapter 22 : Gravity Sizing

22 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] Switch to the Structure view of your model.
2] 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.

Reviewing the Results Graphically

We shall now investigate the Show/Alter State dialog and the Analysis Results toolbar.

Show/Alter State

1] Take a moment to experiment with displaying the various feedback options on the left of the Design tab within the Show/Alter State dialog.

Design Status

The color coding allows you to see the status of each member in your structure instantly. For the model above all members achieve a green ‘pass’ or orange ‘warning’ status.

---

1. You can change to the other pages by clicking the tabs at the bottom of the Workspace.
Auto Design
Members that were initially set to *Auto Design Mode* have had a section size assigned to them and every member is now set to *Check Mode*. By hovering the cursor over each of the members you can see which section sizes have been assigned.

Gravity Only Design
This shows that all the horizontal beams are *Gravity Only Design* hence their section size will not be modified in the subsequent *Lateral Sizing* process.

Stability Coefficient
The stability coefficient will be calculated for all the columns.

Design Utilization
The color coding allows you to instantly assess the efficiency of the design for each member.

Analysis Results
You choose the effect you want to view from the Output Graphics toolbar and the loadcase or combination to view from the Loading toolbar.

Shown above is the major shear force diagram for the combination 1.0D+1.0L.
Chapter 23 : Modifying Element Designs : 61

1/ Take a moment to experiment with displaying the various analysis effects for the different loadcases and combinations.

To return to the Show/Alter State view pick Edit/Show Alter State.

Design Results

Building Designer makes it easy for you to examine the detailed design results.

1/ Hover the cursor over each of the columns, to see the section size(s) that have been chosen.

2/ Click the Column icon in the Building Objects toolbar and then right-click the column at the grid intersection A1. Pick Design Results from the menu that appears.

The ratios on the Summary tab give an indication of the utilization of the chosen section.

Note You might see different utilization ratios to those shown above, if, for example, your design order files include a different range of preferred sections for design consideration.

3/ Click Close to shut the Design Summary.

You can see the results for any of the other members in the same way.

23 Modifying Element Designs

From the above you will see that the design of the members in your structure is a fully automatic process – Building Designer has chosen suitable section sizes working within the design constraints you have applied. Now the structure has been analyzed, you can easily investigate alternative section sizes directly.

1/ Select the two ridge beams.
The currently assigned section is shown in the Properties window.

2/ Click on the button next to the section size and choose a W 12x16 from the Data List.

The two changed sections are redesigned immediately.

3/ Right-click on either of the changed beams and from the context menu pick Design Results once again.

These are updated to show the results for the new size that you selected.

Caution

These results are based on the current analysis of your model. Typically when you change section sizes in this way you would need to re-perform a gravity check design to determine the full effects on your model.

4/ Click Close to shut the Design Summary.

Refining the Design

If you want a greater degree of control than this, then you can transfer an element into the appropriate design program to get further options.
1] Right-click one of the sloping roof beams along grid line B and from the context menu pick Design Results.

You can see that the beam passes but with warnings. Investigating the warnings further you will find that the beam slenderness exceeds the advisory limit of 200. At this stage you could try substituting different section sizes until the warnings disappear, but a more powerful means of refining the design would be to extract the beam into the General Beam design application.

2] Click Close to shut the Design Summary then right-click on the beam again and this time from the context menu pick General Beam.

3] After a short delay General Beam will launch, and you will see that it contains the beam which you have selected.

You can use all General Beam’s features to further investigate your design options.

4] Pick Beam/Design Beam to revert from check design to automatic design mode.

5] Pick Design/Beam... to obtain a list of all the acceptable sections that satisfy your current design criteria.

Note The design results can be displayed for any section from any of the available files on the list using the Preview button.

6] Highlight the W 8x40 section from the WMOrderAISCImp file and click Preview to examine the result. If this is acceptable close the design summary and then click OK to choose this section.
7] Pick Beam/Return section to model to replace the existing section size in Building Designer.

8] Having finished refining the design you can close the General Beam program. There is no need to save the design so when prompted choose No.

You are returned to Building Designer and the section size has been updated – but only for the single roof beam that was investigated.

**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 of the roof beams to the new section size.

1] Pick Selection Groups from the Select toolbar to display the default groups.
2] Pick Select from the Select toolbar and click on any of the roof beams. Because they are all in the same group they will all be selected.

3] Now look at the Properties window, and you will see all the common details for the Roof Beams group.

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

   Some of the information is removed, for example *Section*. This is because the section size is not consistent for the whole group.

4] Click in the Section blank cell. Click again on the button that appears in order to call up the data list.

5] Select a W8x40 section. This will now be applied to all the roof beams.

6] Re-perform the gravity sizing.

## 24 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'.

### Set Auto Design Mode.

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.

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

2] Set all braces and all 'Gravity and Lateral Design' Columns to auto design mode as shown above.
3] To see which members have been affected by this change, click the Auto Design option in the Show/Alter State dialog.


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

5] On completion all members are set into Check Design mode and the Structure window shows the results of the design graphically:

Note Lateral Sizing is performed for strength only, not for stability. Hence you may still need to refine the design to get everything to pass. In this example the entire structure needs more stability. Increasing the column size increases the lateral stiffness and improves the stability.

6] Using the same technique as applied in the Gravity Sizing, (see Modifying Element Designs on page 61) increase all the columns to W8x40 and re-run the Lateral Sizing.
The Workspace Results tab

In addition to the graphical display of results, useful information is also presented in the Results tab of the Workspace.

At the top of the Workspace 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 stability co-efficient results for your model.

If a seismic design had been performed seismic drift results would also be available.

Next you will see the loading summary which is then followed by the reactions.

For each loadcase the loading summary shows you the load that you have applied to your structure. The reactions by combination show you the total load being transferred to the foundations.

Below this are the details of the design status and section size assigned to every member within the structure. Double-click on any member name in the workspace and you will see its Design Summary.

25 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 every member passes you can proceed to output the calculations.
26 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 on any beam and from the context menu that appears pick Report.
2. The report for the chosen beam appears in its own window. Review the details, (using the Report and Navigate toolbars at the right of your screen) 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 First floor.
2. Pick Show/Alter State from the Edit toolbar to see the Show/Alter State display in the First workbook.
3. Click the Results tab in the Show/Alter State dialog, then click the Report Level option.
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 color appropriate to the report level you assign.
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 numerous reports, including material lists, beam end reactions, and foundation load reports. You will create a material list now.

7. Pick File/Report/Material Listing, and after a few seconds you will see the material list report.
8] 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 Material Listing to Excel instead.

9] Close the open report windows.

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

27 Create a Drawing

You made need to include drawings along with your reports. You can create drawings in DXF format from any of the 3D- or 2D-windows.

1] From the First floor workbook pick File/Export/Export to DXF...

A dialog appears allowing you to indicate which details you want to include in the export.

2] Click Export.

You can then choose the destination for saving the dxf file.

28 Modify the Model

Modifying the position of any existing member, or grid line is easy within Building Designer.

Note

If the position of a gridline is moved, all the members connected to it move also (on all floors). This can be a very powerful technique for changing bay spacings etc.

1] Change to the First Floor workbook.

2] Pick Modify from the Edit toolbar and Beam from the Building Objects one.

3] Click the beam shown.
4] Hover the pointer over the left hand end of the beam.

5] You will see that the tooltip has changed to Move Point. Click again to do so, and then click on the supporting beam.

6] Instead of clicking on one of the displayed snap points simply type in the distance you require, (14').

7] Click OK and then move the right hand end of the beam, this time connecting to the snap point Same Y point.
8] Pick Slab from the Building Objects toolbar and then modify the slab shown so that it is once again properly supported by the beam.

After modifying your model, you simply re-perform the analysis and design process as previously illustrated before outputting the calculations once more.

Congratulations you have now finished this Quick Start Tutorial. We hope that you have found this introduction helpful.

29 What Next?

In this very simple example you have created and analyzed 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 course, where you will find that our expert instruction gives you a flying start, ensuring that you are fully productive in the shortest possible time.

You might also want to:

• look at the on-line documentation or help file which contain 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\Fasttrak folder. These technical notes deal with the sway resistance of models among other things.

1 Contact our Support Department for details.