

Tekla Structural Designer Fundamental Training

TSD 2016 Manual

Copyright © 1992 - 2016 Trimble Solutions Corporation – part of Trimble Navigation Ltd. All rights reserved.

Disclaimer

This Software Manual has been developed for use with the referenced Software. Use of the Software, and use of this Software Manual are governed by a License Agreement. Among other provisions, the License Agreement sets certain warranties for the Software and this Manual, disclaims other warranties, limits recoverable damages, defines permitted uses of the Software, and determines whether you are an authorized user of the Software. All information set forth in this manual is provided with the warranty set forth in the License Agreement. Please refer to the License Agreement for important obligations and applicable limitations and restrictions on your rights. Trimble does not guarantee that the text is free of technical inaccuracies or typographical errors. Trimble reserves the right to make changes and additions to this manual due to changes in the software or otherwise.

In addition, this Software Manual is protected by copyright law and by international treaties. Unauthorized reproduction, display, modification, or distribution of this Manual, or any portion of it, may result in severe civil and criminal penalties, and will be prosecuted to the full extent permitted by law.

Tekla, Tekla Structures, Tekla BIMsight, BIMsight, Tekla Civil, Tedds, Solve, Fastrak and Orion and are either registered trademarks or trademarks of Trimble Solutions Corporation in the European Union, the United States, and/or other countries. More about Trimble Solutions trademarks: http://www.tekla.com/tekla- trademarks. Trimble is a registered trademark or trademark of Trimble Navigation Limited in the European Union, in the United States and/or other countries. More about Trimble trademarks: http://www.trimble.com/trademarks.aspx. Other product and company names mentioned in this Manual are or may be trademarks of their respective owners. By referring to a thirdparty product or brand, Trimble does not intend to suggest an affiliation with or endorsement by such third party and disclaims any such affiliation or endorsement, except where otherwise expressly stated.

Portions of this software:

D-Cubed 2D DCM © 2010 Siemens Industry Software Limited. All rights reserved.

EPM toolkit @ 1995-2004 EPM Technology a.s., Oslo, Norway. All rights reserved.

Open CASCADE Technology © 2001-2014 Open CASCADE SA. All rights reserved.

FLY SDK - CAD SDK © 2012 VisualIntegrity[™]. All rights reserved.

Teigha © 2003-2014 Open Design Alliance. All rights reserved.

PolyBoolean C++ Library © 2001-2012 Complex A5 Co. Ltd. All rights reserved.

FlexNet Copyright © 2014 Flexera Software LLC. All Rights Reserved.

This product contains proprietary and confidential technology, information and creative

works owned by Flexera Software LLC and its licensors, if any. Any use, copying, publication, distribution, display, modification, or transmission of such technology in whole or in part in any form or by any means without the prior express written permission of Flexera Software LLC is strictly prohibited. Except where expressly provided by Flexera Software LLC in writing, possession of this technology shall not be construed to confer any license or rights under any Flexera Software LLC intellectual property rights, whether by estoppel, implication, or otherwise.

To see all the third party licenses, go to Tekla Structures, click Help --> About and click the Third party licenses button.

The elements of the software described in this Manual are protected by several patents and possibly pending patent applications in the United Stated and/or other countries. For more information go to page http://www.tekla.com/tekla-patents.

Table of Contents

1	Introduction	13
1.1	Welcome	13
1.2	Design Code	13
1.3	Important Notes Regarding this Documentation	13
2	The Graphical User Interface	15
2.1	Introduction	15
2.2	The Ribbon	15
2.3	Screen Layout	15
2.4	Customising the Graphical Interface	
2.4.1 2.4.2	Enabling and moving windows Side-by-side views	
2.5	Navigating the Interface	
2.5.1	The ribbon	
2.5.2	The project workspace and properties window	
2.5.3	Generated views	
2.5.4	Element select and properties window	
2.5.5	Clear selection	
2.5.6	Process window	
2.5.7	Scene contents	
2.5.8	Model status information	
2.5.9	Modelling view control	
2.5.10		
2.5.11	Modelling navigation cube	21
2.6	Mouse Control	22
2.6.1	Zoom, pan and rotate – 3D	22
2.6.2	Zoom and pan – 2D	
2.7	Selection Techniques	
2.7.1	How do I select an individual entity?	
2.7.2	How do I add further individual entities to the current selection?	
2.7.3	How do I deselect a single entity from the current selection?	
2.7.4	How do I select multiple entities?	23
2.8	How Do I Deselect All Entities?	25
2.9	Selection Entities via Structure Tree	25
2.10	Properties Window	
2.10.1		
2.10.2	How do I edit the properties of multiple entities?	27

2.11 2.11.1	Property Sets How do I save properties to a named property set? How do I create a property set prior to creating an entity in the model? How do I create a property set based on an existing entity in the model?	.29 .29
2.11.2		.29 .29
2.11.3		
3	Model Validation	.33
3.1	Introduction	.33
3.2	How Do I Run Model Validation?	.33
3.3	What are Validation Errors and Warnings?	.33
3.4 3.4.1	How do I Locate Validation Errors? Working with the status tab in the project workspace	
3.5 3.5.1	Common Validation Errors "Panel is not surrounded by load carrying members" RI (Roof Item) SI (Slab Item)	.34 .34
3.5.2 3.5.3 3.5.4	"Members collision" "Member Intersection" "Slab Overlap"	.37 .37
3.6	Right Click Context Menu	.41
3.7	Command Activation	.41
4	Tekla Structural Designer Help	.43
5	Multi Material Modelling 2D	.45
5.1	Introduction	.45
5.2 5.2.1 5.2.2 5.2.3 5.2.4 5.2.5 5.2.6 5.2.7	Model and Global Settings Settings Settings Sets Design Codes Section Defaults Analysis Options Concrete Design Options Drawings	.45 .45 .46 .47 .47 .48
5.3 5.3.1	Materials Concrete Grades	.49
5.3.2 5.4	Reinforcement	

5.5 5.5.1	Starting a New Project and the Project Wiki Model Settings	
5.6 5.6.1 5.6.2 5.6.3 5.6.4	Modelling – Establishing the Structural Geometry Construction Levels Grid and Construction Lines Graphically displaying Grids Adding additional gridlines	51 52 54
5.7 5.7.1	Selecting Elements for Editing Gridline Properties	
5.8 5.8.1 5.8.2	Dimensioning and Measuring Tools Dimensions Measuring tool	57
5.9 5.9.1 5.9.2	Deleting Elements Delete command Prior selecting before delete	57
6	Creating Structural Elements	59
6.1 6.1.1 6.1.2 6.1.3 6.1.4 6.1.5	Concrete Columns Editing column properties – levels Editing column properties – alignment Changing column characteristics – central steel column Changing column properties using property sets Managing property sets	.60 .60 .61 .61
6.2 6.2.1 6.2.2	Obtaining Model Views Obtaining a 2D frame / elevation view Obtaining a plan / level view	.63
6.3 6.3.1	Concrete Beams Creating a property set from a new element	
6.4	Steel Beams	65
7	Construction Lines and Free Form Truss	67
7.1	Construction Lines	67
7.2 7.2.1 7.2.2 7.2.3 7.2.4	Steel Truss Free Form truss properties Modelling booms Modelling internals Member Releases	70 72 73
8	Mirror Command	75
9	Loading	77
9.1	Loadcases	77

9.2	Applying Member Loads	78
9.3	Applying Nodal Load	79
10	Load Combinations	81
11	Validation	83
12	Analysis and Viewing Graphical Results	85
12.1	Running 1 st Order Linear Analysis	85
12.2	Loading Summary Table	86
12.3	View Analysis Results	86
12.4	Filtering Results Graphically	89
12.5	Tabulated Results	89
13	Right Click Context Menu	91
14	The Solver View	93
15	Creating a More Complex 3D Structure	95
15.1	Copy Command	95
15.2	Placing Concrete Beams	96
15.3	Placing Steel Beams	97
15.4	Placing Intermediate Steel Beams	97
15.5 15.5.1 15.5.2	Creating Slabs Concrete slabs Composite slabs	98
15.6	Deleting and Rotating Slabs	100
15.7	Openings for Two-way Slab	100
16	Floor Panel Loading	103
16.1	Displaying Loading Text	104
17	Creating 3D Roof Structure	105
17.1	Creating Eaves Beams	105
17.2	Creating Purlins	106
17.3	Placing Roof Panels	108

17.4	Loading the Roof Structure	.109
18	Steel Floor	.111
18.1	Placing Steel Beams	.111
18.2	Placing a Steel Deck	.112
18.3	Steel Floor Loading	.113
18.4	Combinations	.113
19	3D Model Validation and Analysis Results	.115
19.1	Validation	.115
19.2	Analysis Options	.115
19.3	Analysis Process	.116
19.4	Loading Summaries	.117
19.5	Deflections	.117
20	The Design	.119
20.1 20.1.1	Design Options Analysis	
20.1.2		
20.1.3	Design Forces	
20.1.4	Design Groups	
20.1.5	Autodesign	
20.2	Gravity Design	.122
20.3	Design Review Mode	.123
20.3.1	Design Status	.124
20.3.2	Design Ratio	.124
20.3.3	Show\Alter State – Auto\Check Design	.125
20.3.4	Show\Alter State – Steel button	.126
20.3.5	Design Tabular Data	.126
20.4	Right Click Context Menu	.127
20.4.1	5	
20.4.2	Check Member – Concrete	.127
20.4.3		
20.4.4	Design Interactive Reinforcement Concrete Design	.129
20.4.5	Generating Detailed Drawings	.130
20.4.6	Report for Member	.131
20.5	Steel Elements	
20.5.1	Check Member – Steel	
20.5.2	Design Member – Steel	.132

20.5.3 20.5.4	Generating Detail Drawing – Steel Report for Member – Steel	
20.6 20.6.1	Design Status Analysis warnings	
21	Lateral Loading	137
21.1	EHF's – Equivalent Horizontal Forces	.137
21.2	EHF Control	.137
21.3	Applying Equivalent Horizontal Forces (EHF's)	.138
21.4	Renumbering Loadcases and Combinations	.140
21.5	Wind Loading – Manual Input	.141
21.5.1	Nodal Loading	
21.5.2	Element Loading	
21.5.3	Panel Loading	
21.5.4	Wall Panel Orientation and Span Direction Decomposition	
21.5.5	Wall Panel Changing Orientation.	
21.5.6	Applying Panel Loading	. 145
21.6	Automated Wind Loading	1/6
21.6.1	The Wind Wizard	
21.6.2		
21.6.3	Wind Loadcases	
21.6.4	Reviewing the Decomposed Wind Loads	
21.6.5	Generating Combinations	
211010		
22	Output	153
22.1	Predefined Reports	.153
22.2	Individual Report Components – Model Reports	.153
22.2.1	Report Component Control	
		457
22.3	Structural Element Reports	
22.3.1	Quick Filter Report Options	.157
22.4	Exporting Reports	.159
22.5	Saving Pictures	.159
22.6	Engineering Drawings	160
22.6.1	Creating 2D Multi Material Drawing	
22.6.2		
0.2		
23	Analysis and Design of Concrete Structures	163
23.1	Analysis and Design Options	.163

23.2	Analysis and Design Procedure	
23.2.1	5 5	
23.2.2	5 1	
23.2.3	Complete the Analysis and Design	165
23.3	Reviewing the Overall Design Status	166
23.3.1	Review tab	
201011		
23.4	Reviewing the Analysis Results	167
23.4.1	How to View the Results	
23.4.2	Viewing Results for the Whole Model	168
23.4.3	Slab Strips	169
23.4.4	Viewing Frame Element Analysis Results	171
24	Interactive Design of RC Columns and Walls	173
24.1	Check Member	173
24.2	Design Member	174
24.3	Interactive Design	175
24.3.1	Interactive Design	
24.3.2	Interaction Diagrams	
24.3.3	Detail Drawing	
24.3.4	Updating the Design Group	
24.3.5	Interactive Design of Walls	
25	Interactive Design of RC Beams	181
25 25.1	Interactive Design of RC Beams	
25.1	Check Member	181
		181
25.1	Check Member Design Member	181 181
25.1 25.2	Check Member	181 181 182
25.1 25.2 25.3	Check Member Design Member Design Interactive Design	181 181 182 182
25.1 25.2 25.3 25.3.1	Check Member Design Member Design Interactive Design Detail Drawing	181 181 182 182 183
25.1 25.2 25.3 25.3.1 25.3.2	Check Member Design Member Design Interactive Design Detail Drawing Correcting the Warnings	181 181 182 182 183 183
25.1 25.2 25.3 25.3.1 25.3.2 25.3.3	Check Member Design Member Design Interactive Design Detail Drawing Correcting the Warnings	181 181 182 182 183 183 184
25.1 25.2 25.3 25.3.1 25.3.2 25.3.3 25.3.4 26	Check Member Design Member Design Interactive Design Detail Drawing Correcting the Warnings Updating the Design Groups RC Slab Design	181 181 182 182 183 183 184 185
25.1 25.2 25.3 25.3.1 25.3.2 25.3.3 25.3.4 26 26.1	Check Member Design Member Design Interactive Design Detail Drawing Correcting the Warnings Updating the Design Groups	181 181 182 182 183 183 184 185 185
25.1 25.2 25.3 25.3.1 25.3.2 25.3.3 25.3.4 26 26.1	Check Member Design Member Design Member Design Interactive Design Detail Drawing Correcting the Warnings Updating the Design Groups RC Slab Design Introduction to Slab Design	181 181 182 182 183 183 184 185 185 186
25.1 25.2 25.3 25.3.1 25.3.2 25.3.3 25.3.4 26 26.1 26.1.1 26.2	Check Member Design Member Design Interactive Design Detail Drawing Correcting the Warnings Updating the Design Groups RC Slab Design Introduction to Slab Design Overall Slab Design Procedure Checking the Background Reinforcement with No Patches	181 181 182 182 183 183 184 185 185 186 186
25.1 25.2 25.3 25.3.1 25.3.2 25.3.3 25.3.4 26 26 26.1 26.1.1	Check Member Design Member Design Interactive Design Detail Drawing Correcting the Warnings Updating the Design Groups RC Slab Design Introduction to Slab Design Overall Slab Design Procedure	181 181 182 182 183 183 184 185 185 186 186 187
25.1 25.2 25.3 25.3.1 25.3.2 25.3.3 25.3.4 26 26.1 26.1.1 26.2 26.3	Check Member Design Member Design Interactive Design Detail Drawing Correcting the Warnings Updating the Design Groups RC Slab Design Introduction to Slab Design Overall Slab Design Procedure Checking the Background Reinforcement with No Patches Types and Applications of Patches Wall Patches	181 181 182 182 183 183 184 185 185 186 186 187 187
25.1 25.2 25.3 25.3.1 25.3.2 25.3.3 25.3.4 26 26.1 26.1.1 26.2 26.3 26.3.1	Check Member Design Member Design Interactive Design Detail Drawing Correcting the Warnings Updating the Design Groups RC Slab Design Introduction to Slab Design Overall Slab Design Procedure Checking the Background Reinforcement with No Patches Types and Applications of Patches Wall Patches	181 181 182 182 183 183 184 185 185 185 186 186 187 187 188

26.6	Design the Slab Patches	190
26.7	Review and Rationalise the Patch Design	190
26.8 26.8.1	Flat Slab Design Wall Patches	
26.8.2	Column Patches	193
26.8.3	Completing the Design of the Floor	
26.9	Punching Shear Checks	194
27	RC Members Detail Drawings	
27.1	Drawing Settings	197
27.1.1	Layer Configurations	
27.1.2		
27.1.3	Options	
27.1.4	Detailing Groups	
27.2	Detail Drawings of Individual Elements	199
27.3	Schedules	200
27.4	Slab Detail Drawings	
27.4.1	General Arrangements	
27.4.2	0	
<u>от г</u>	Drowing Management	202
27.5	Drawing Management	
27.5.1	Concrete Member Detailing	
27.5.2	Other Drawing Management Options	
27.5.3	Schedule Management	206
28	RC Members Design Reports	207
28.1	Individual Member Design Reports	
28.1.1	Report Content	
00.0		000
28.2	Model Reports	209
28.3	Other Report Options	210
29	Design Groups	211
29.1	How are Design Groups Formed?	211
29.2	The Design Procedure for a Design Group	211
29.3	Working with Design Groups	212
29.3.1	Interrogating the Design Groups	
29.3.2	Renaming Design Groups	
29.3.3		
29.3.4		
29.3.5	5 5 5 1	

29.3.6	Re-Group All Model Members	215
30	Analysis and Design of Steel Structure	217
30.1	Combination Classes	217
30.2	Properties	217
30.2.1	•	
	Autodesign	
30.3	Design Options	
30.3.1	Analysis	
	What is Geometric (P-delta) Non-linearity?	
	First-order (Elastic) Analysis	
	Second-order (P-delta) Analysis – Amp. Forces Method	
	Second-order (P-delta) Analysis	
	Table of Analysis Options and Non-linearity consider	224
30.4	Design Steel (Gravity)	224
30.4.1	How do I run Design Steel (Gravity)?	224
30.5	Check the Status	225
30.6	Check the Loading Summary	225
30.7	Review View	226
30.7.1	Design – Status	
30.7.2	Design – Ratio	226
30.7.3	Show\Alter State – Auto\Check Design	
30.7.4	Show\Alter State – Diaphragm On\Off	
30.7.5	Show\Alter State – Restrained\Unrestrained	
30.7.6	Show\Alter State – Fixed\Pinned	
30.7.7	Show\Alter State – Steel	
30.7.8	Show\Alter State – Copy Properties	231
30.8	Interrogate and Review Individual Members	
30.8.1	Right Click Context Menu – Edit Member	
30.8.2	Right Click Context Menu – Open Load Analysis View	
30.8.3	Right Click Context Menu – Open Member View	
30.8.4 30.8.5	Right Click Context Menu – Show Member Loading	
30.8.5	Right Click Context Menu – Check Member Right Click Context Menu – Design Member	
30.8.7	Right Click Context Menu – Report for Member	
30.9	How to Check Member/ Design Member/ Edit	
30.9.1	Scenario 1 – Check Member/ Design Member	
30.9.2	Scenario 2 – Check Member/ Edit Member/ Check Member	237
30.10	Design Steel (Static)	239
	1 How to run Design Steel (Static)	
30.10.2	2 Using the Review View again	240
30.11	Using Result View to Investigate Solver Warnings	241

	1 Project Workspace – Status 2 The Result View	
30.12.	Tabular Data 1 Design Tabular Data 2 Analysis Tabular Data	246
31	Foundation Design	249
31.1	Introduction	249
31.2	Using Cutting Planes	249
31.3	Model of Pad Footing	251
31.4 31.4.1	Design of Pad Footing Rationalise bases	
31.5	Model of Strip Footing	254
31.6	Foundation Drawings	255
31.7	Foundation Design Report	256
32	Changing the Design Code (For Information only)	257
32.1	Introduction	257
32.2	Head Code	257
32.3	How to Configure the Default Design Codes to be applied to New Projects?	257
32.4	How to Change Design Codes in an Existing Project?	257
32.5	Model Changes	257

1 Introduction

1.1 Welcome

Welcome to the Tekla Structural Designer - Fundamentals training course.

This comprehensive two day course will teach you how to model, analyse and design steel and concrete buildings effectively using Tekla® Structural Designer. Referencing your existing knowledge, you will gain a full understanding of how to model with physical objects and undertake gravity and lateral design. In addition you will grasp the many useful shortcuts that our experts use daily.

1.2 Design Code

This manual has been written based on Eurocode to the UK National Annex. Screenshots throughout this manual reflect the chosen Eurocode and variations will exist if a different Code of Practice is selected.

Please refer to Chapter 31 'Changing the Design Code' should you need to design to an alternative Code of Practice and to understand the implications of switching design code.

1.3 Important Notes Regarding this Documentation

The following conventions have been utilised in the manual;

Step 1. You should undertake these steps so that you learn by "doing" rather than reading.

Tekla Structural Designer from now on will be referred to as "TSD"



A tip might introduce a shortcut, or suggest alternative ways of doing things.



A note draws attention to details that you might easily overlook. It can also point you to other information in this guide that you might find useful.



You should always read very important notes and warnings, like this one. They will help you avoid making serious mistakes, or wasting your time.



This symbol indicates advanced or highly technical information that is usually of interest only to advanced or technically-oriented reader.

14 (257)

2 The Graphical User Interface

2.1 Introduction

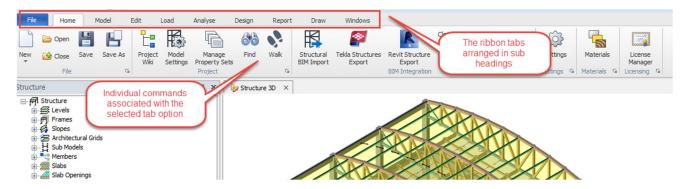
In this section you will understand the Graphical User Interface of TSD and the manipulation of the screen to enhance your TSD experience.

Step 1. Open the example model file Fundamental Training Completed.tsmd

2.2 The Ribbon

The graphical user interface has been designed so that any aspect of the program can be easily found.

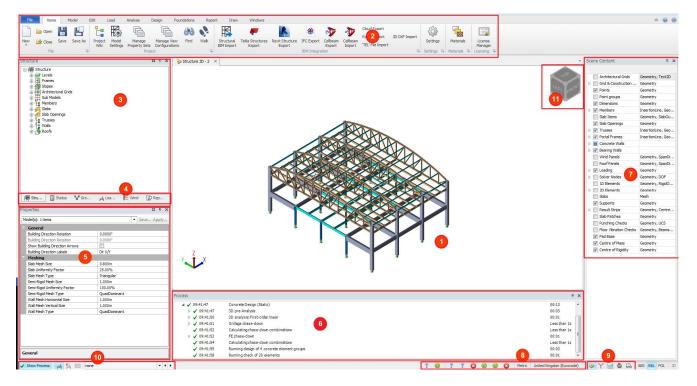
Tabs are created as sub headings with individual commands associated with those sub headings.



2.3 Screen Layout

The initial installation will create a standard program setup similar to that shown below.

Note that individual windows can be resized, repositioned and hidden as per the user's requirements.



Briefly the main areas of the interface are:

- 1. Modelling View 3D, plans, elevations, inclined planes al created in a separate tab.
- 2. Ribbon command and settings buttons arranged in a number of tasked based tabs.
- 3. Project Workspace Detailed information (split into sub tabs) about your model.
- 4. Project Workspace Sub Tabs Detailed information arranged in categories.
- 5. Properties Window Displays the properties of a selected item which can be viewed or edited.
- 6. Process Window Displays the process of analysis and design.
- 7. Scene Content On/off settings for displayed objects in the modelling view.
- 8. Status Information Current model situation in analysis and design.
- 9. Modelling View Control Displays alternative views of your model.
- 10. Loading Toolbar Loading control.
- 11. 3D View Navigation Cube Displays views by clicking on the corner edges or faces with a rotate option.

2.4 Customising the Graphical Interface

2.4.1 Enabling and moving windows

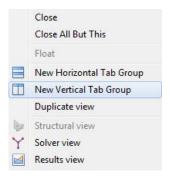
The graphical interface windows can be turned on and off via the buttons on the Windows tab of the ribbon. Every window or sub tab (not on main ribbon) can be unlocked and clicked and dragged to various positions.

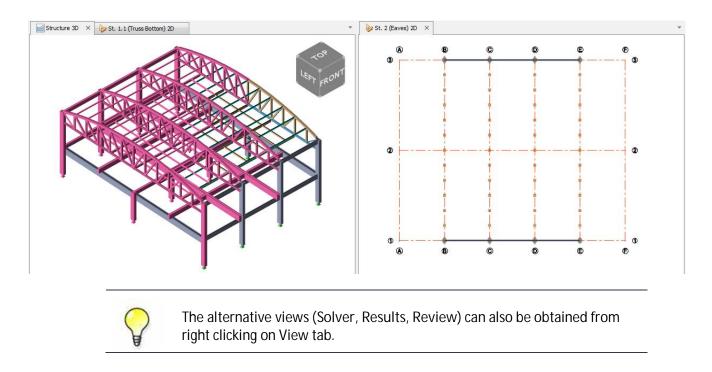
File Home Model Edit Load Analyse Design	Report Draw Windows Results		a 😠 🛈
1st Order Linear - ID Forces V Fx Mx Fxyz	Axial Force Torsion X Dir1 X Y Fix Fixz My Mdx.bottom X Pefection Axial Force Torsion	Deflection Support Reaction	
None * Strength Factors 2D Forces Py My Moy	EHF- Design Moment	Axial Force Torsion	
Convice Exchange	Seismic Create AsBen		
Reduce Axial Force Deflections Deflections Fz Mz Tota Result Type G Scale Settings G Support Reacti		Shear Moment Text G	
	Structure 30 di St. () 20 ×	 Scene Content 	ά×
Stuture ■ P × X Stuture ■ P × X State = State = Sta	State 2 10 ×	 Scene Content Architectural Gride Grid & Construction Ponts Ponts Ponts Bab Items Sab Items Sab Items Sab Items B Concrete Nation Pont of Pounds Sab Items Centre Of Nass Centre Of Nass Centre Of Nass Pound 	0 x Georetry, Tot20 Georetry Georetry Georetry Georetry, Boun, Georetry, Boun, Georetry, Sach, Georetry, Sach, Georetry, Sach, Georetry, Sach, Georetry, Sach, Georetry, Sach, Georetry, Sach, Georetry, Coll Georetry,
Show Process 🛃 🎇 🖂 none 🔻 🗸	and a second secon	1001	486 08 001 20
A A LO INIC	Sector Chicked Million (Chicked Million (Chicked Million) (Chicked Million)		nee rot 30

2.4.2 Side-by-side views

Views can be displayed side-by-side or above/below each other (tiled).

When views are opened right-click the vertical tab and choose New Vertical Tab Group from the context menu.





2.5 Navigating the Interface

2.5.1 The ribbon

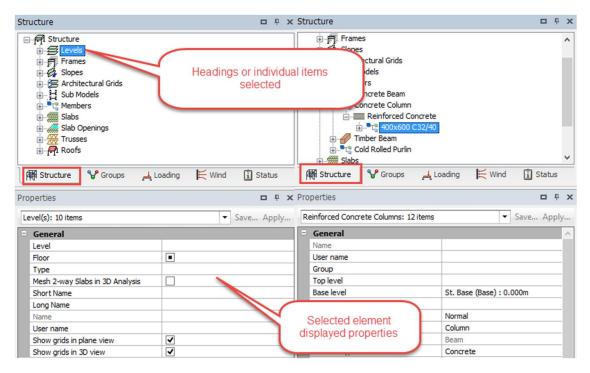
File	Home	Model	Edit	Load Analy	rse	Design	Bern		
Loadcases	Combination	Envelope	Update Patterns	Wind Wizard	Update Zones	a Wind Load	Click a ribbon tab to activ Corresponding comman associated with that tab displayed.	nds 🦾	Line Load

2.5.2 The project workspace and properties window

Under the Structure sub tab of the project window, individual items can be selected and properties displayed.

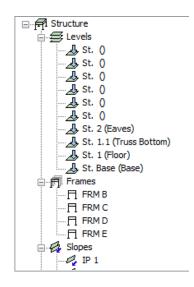


Note the structure tree can be expanded and individual items selected.



2.5.3 Generated views

As the model is created, TSD will automatically create views and are listed in the Project Workspace.



To display any view, simply double left click mouse button on it under the Structure tree.

2.5.4 Element select and properties window

By default when no command is activated you are in Select mode.

• Select entities in the visible scene by left clicking on the element – the individual element properties are displayed in the properties window.

	Save A	piy		
General				
Name	CB FRM B V1			
User name				
Group	CRB8-D2			
Plane	FRM B			
Concrete type	Normal			
Characteristic	Beam			
Element type	Beam			
Material type	Concrete			
Construction	Concrete beam			
Fabrication	Reinforced			
Autodesign				O ment and a state of a state in
Rotation	0.0000°			Current selected entity is
Rotation angle	0.0000°			highlighted
Allow automatic join end 1				
Allow automatic join end 2	~	00		
All spans		15 000		
Name				
User name			X	
Section	400x800			
Concrete dass	C32/40	Properties of that entity is	00	
Linearity	Straight	displayed	15 000	
🗄 Releases				
🕑 Alignment				Ar

2.5.5 Clear selection

To clear a selection – press the keyboard Esc key or double right click the mouse button.

2.5.6 Process window

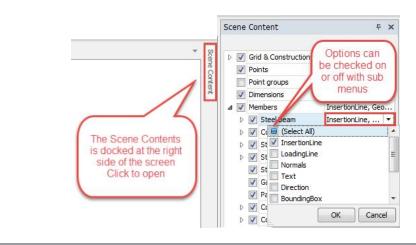
The process window can be activated to appear or permanently docked on the screen. To activate press this button.

Show Process	<u>بلم</u>	5월 🖂	none	•	•	+	
--------------	------------	------	------	---	---	---	--

2.5.7 Scene contents

When you have several elements displayed in a modelling view along with their text labels and rendering styles, the scene can become cluttered and difficult to use.

The Scene Contents allows you to configure what is displayed in the current view.



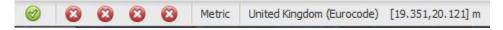


Each individual modelling view will be controlled by individual Scene Contents option.

2.5.8 Model status information

The green ticks or red crosses will show the current status of the model with respects to:

- Analysis Condition
- Concrete / Steel Design Condition
- Modelling System Units
- Model Design Code Adopted
- Coordinate Cursor Position



2.5.9 Modelling view control

The current modelling view can be visualised in several ways.

The default view is the structure view (rendered 3D of the physical model).

- Structural View for modelling the structure as a physical model.
- Solver View a simplified view of the underlying analysis model.
- Results View for viewing analysis results.
- Review Mode For reviewing and amending certain aspects of the model.

Toggle buttons for changing the view mode can be found at the bottom right of the screen.



[25.977,4.727] m 😡 🍸 🛃 🔓

2.5.10 Loading tool bar

The loading toolbar allow for the visualisation of loadcases and combinations.



2.5.11 Modelling navigation cube

When in a Structure 3D View – move the cursor over the Navigation Cube.

Click on a Corner, Edge or Face to move the view to the one you wish to see.



2.6 Mouse Control

2.6.1 Zoom, pan and rotate – 3D

When in a structure 3D view you can navigate the screen as follows:

- Scroll mouse wheel to Zoom
- Hold down mouse wheel and move cursor to Pan
- Hold down right mouse button and move cursor to Rotate
 - o Move cursor vertically to rotate about horizontal axis
 - o Move cursor horizontally to rotate about a vertical axis
- When an object is selected in the Structure 3D view the model will rotate around that object

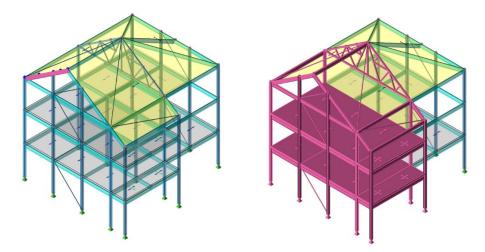
2.6.2 Zoom and pan – 2D

When in a 2D view you can navigate the screen as follows:

- Scroll mouse wheel to Zoom
- Also holding the right mouse button allows you to Zoom
- · Hold down mouse wheel and move cursor to Pan
- 2.7 Selection Techniques

Step 1. Open the example model file 1_Selection_Properties_Sets_Start.tsmd

When no command is operational you are automatically in Select mode. This will allow you to select multiple or individual items and will be highlighted in pink when in a 2D or 3D view.

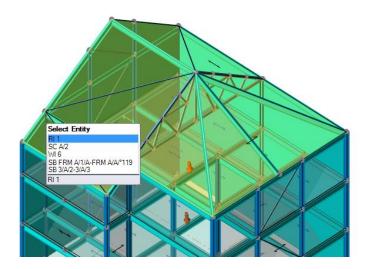


2.7.1 How do I select an individual entity?

Move the cursor over the required entity in one of the 3D or 2D Views.

• If the entity is the only one at that location it will become highlighted (it will also be the one listed in the Select Entity tooltip).

If several entities exist at the same location they will all be listed in the Select Entity tooltip, with only the first one being highlighted. If this is not the required entity, use the Tab key or the Up/Down arrow keys or the Ctrl + mouse wheel to scroll through the list.



When the required entity is highlighted, you can either press the Enter key or left click to select it.



The selected entity's properties are displayed in the Properties window.

2.7.2 How do I add further individual entities to the current selection?

Hold the Ctrl key whilst clicking on each subsequent entity.

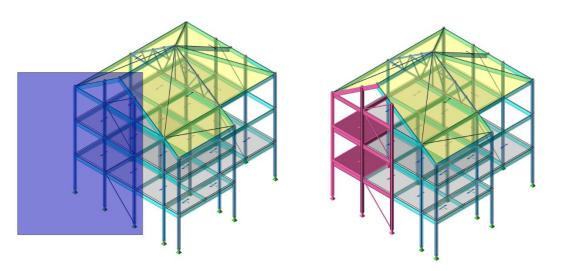
2.7.3 How do I deselect a single entity from the current selection?

Hold the Ctrl key whilst clicking on the entity you want to de-select.

2.7.4 How do I select multiple entities?

There are three methods available which will be discussed below.

- (1) If you only want to select those entities which are totally encompassed by the box, then:
- Move the cursor to the left corner of an imaginary box which will encompass the entities that you want to select.
- Click and hold the left mouse button.
- Drag to the opposite right corner (you will see a purple rectangle box on the screen which follows your mouse movements and helps you to check the area you are selecting).
- Release the mouse button.

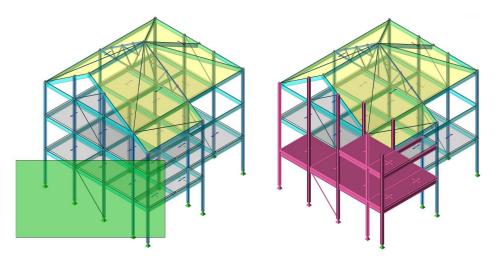


This will only create a selection for those entities encompassed by the box.



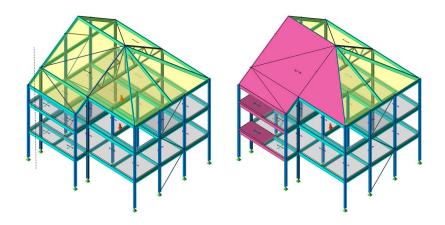
Hold the Ctrl key to add to an existing selection. If you drag a box to encompass a stack of a column the entire column will be included in the selection.

- (2) If you want to select those entities which are totally encompassed by the box, and which it crosses, then:
- Move the cursor to the right corner of an imaginary box which will encompass the entities that you want to select.
- Click and hold the left mouse button.
- Drag to the opposite lower left corner creating a green rectangle box.
- Release the mouse button.



This will only create a selection for those entities encompassed by the box and which it crosses.

- (3) If you only want to select those entities which cross a line,
- Move the cursor to the start of an imaginary line which will encompass the entities that you want to select.
- Hold the Shift key and click and hold the left mouse button.
- Release the mouse button then the Shift key.



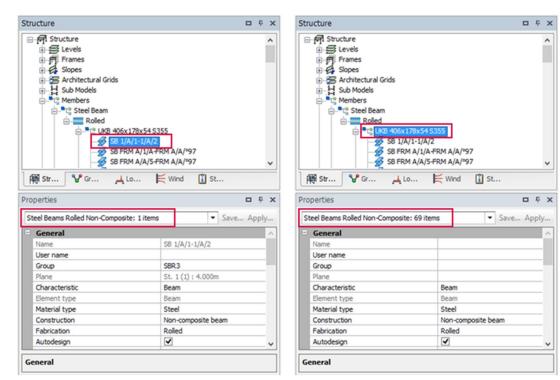
2.8 How Do I Deselect All Entities?

Double right click mouse button or press the Esc key will deselect all currently selected entities.

2.9 Selection Entities via Structure Tree

Only individual entities or groups of similar entities can be selected from the Structure tree, (it is not possible to select multiple members in this way).

- Expand the Members branch of the Structure tree and then the appropriate sub-branches until the member references are displayed.
- Click either a single entity from the list (to select just this entity) or the folder (to select all).





Right click the required member's reference or folder and then pick Select in visible views from the context menu to highlight in the view.

Structure	⊐ # ×
⊡ नि Structure	-
ia ∰ Levels ia ∰ Frames ia ∰ Slopes	
	E
🗄 📲 📲 Members	
🖨 📲 Steel Beam	
UKB 406x1/8x54 S355	
58 1/A/1-1/A/2	
SB 1/A/1-1/A/2 Edit SB 1/A/2-1/A/3 SB 1/A/2-1/A/3 Edit SB 1/A/3-1/A/4 SB 1/A/3-1/A/4 Deleter SB 1/A/4-1/A/5 Open Open	
SB 1/A/1-1/A/2 □	
SB 1/A/1-1/A/2 Edit SB 1/A/2-1/A/3 SB 1/A/2-1/A/3 Edit SB 1/A/3-1/A/4 SB 1/A/4-1/A/3 Delett SB 1/B/1-1/B/2 SB 1/B/1-1/B/2 Open SB 1/B/2-1/B/3 SB 1/B/2-1/B/3 Select	view

2.10 Properties Window

When a new entity is created, it adopts the properties that are displayed in the Properties window at that particular time. You should therefore ensure the properties are correct before you place the entity.

When an existing entity is selected its properties are displayed in the Properties window where they can be edited.

2.10.1 How do I edit the properties of a single entity?

Select an entity and change the properties as required using the Properties window.

teel Beams Rolled Non-Composite	e: 1 items Save Apply
General	· · · · · · · · · · · · · · · · · · ·
Name	SB FRM A/1/A-FRM A/A/97
User name	
Group	SBR5
Plane	FRM A
Characteristic	Beam
Element type	Beam
Material type	Steel
Construction	Non-composite beam
Fabrication	Rolled
Autodesign	
Design section order	UKB Beam
Gravity only	
Rotation	0.0000°
Rotation angle	0.0000°
Major snap level	Тор
Major offset	0.0mm
Minor snap level	Centre
Minor offset	0.0mm



Individual properties can also be edited via the Properties dialog, by right clicking an entity in the View and choose Edit from the context menu or by double left clicking on an entity in the View.

		 Undo Redo Copy Delete Element Zoom Out Create propert Edit SB 1/B/2-3 Open Load Art 	y set L/C/2	
ies - SB 1/B/2-1/C/2	General	Beam		
General Size Alignment	Element type	Beam	• •	
Alignment Releases	Material type	Steel	•	
Charles and the second s	Construction type	Non-composite bear	•	
Lateral restraints Strut restraints	Consu dedon type			
Lateral restraints	Fabrication	Rolled	•	

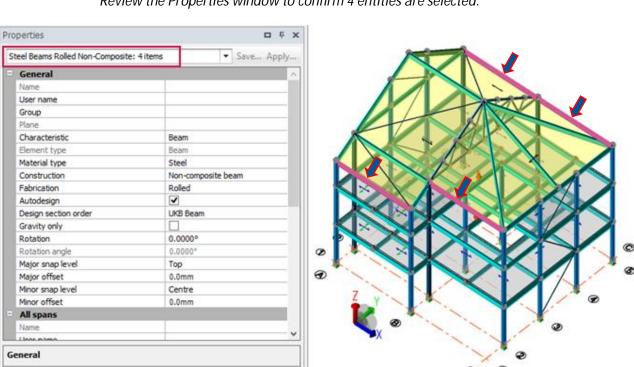
2.10.2 How do I edit the properties of multiple entities?

- If multiple entities of different types are selected, then properties information is displayed separately for each type.
- A dropdown menu at the top of the Properties window is used for moving between types.

roperties	_
Steel Beams Rolled Non-Composite: 71 items	ave Apply
Architectural Grid(s): 1 items	
Grid Lines: 8 items	
Roof(s): 3 items	
Slab Item(s): 24 items	
Steel Beams Rolled Non-Composite: 71 items	
Steel Braces Rolled: 10 items	E
Steel Columns Rolled Non-Composite: 14 items	
Steel Truss Internals: 11 items	
Steel Truss Members Bottom: 1 items	
Steel Truss Members Side: 2 items	
Steel Truss Members Top: 2 items	am
Support(s): 14 items	-
Support(s): 14 items Autoaesign	

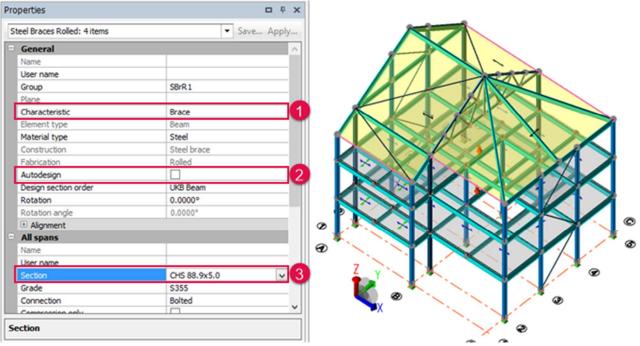
The Properties window will show a blank where an item (Reference format, Alignment, Offset, Report etc.) is not identical for all the selected entities.

- If you change a blanked item, the new setting are applied to all the selected entities.
- If you leave a blank item blank, then the current diverse settings for the selected entities are maintained.



Step 1. Select the roof eaves beams highlighted in below (using Ctrl key for multiple selections). Review the Properties window to confirm 4 entities are selected.

Step 2. Change the characteristic to a Brace (1), untick Autodesign (2) and assign the Section as a CHS 88.9x5.0 (3) section. The View is updated to reflect the changes.



Step 3. Undo 3 times the previous changes.



Pressing Undo once undoes the last property change i.e. Section. Thus you will need to press undo three times to revert back to the characteristic Beam!

2.11 Property Sets

In a typical model you may want to apply the same properties to similar members at a number of different locations. To do this efficiently, once properties have been set up for a member you can save them away to a named Property Set for subsequent recall.

2.11.1 How do I save properties to a named property set?

Property Sets can be created before you place an entity in the model or can be created from an existing entity that is already in the model.

How do I create a property set prior to creating an entity in the model?

Properties can only be saved to a Property Set from the Properties window when there are no items selected - this ensures that unique entries exist for each of the properties in the set.

- Step 1. Click on the Steel Beam command on the Model tab.
- Step 2. The drop list at the top of the Properties window should read '<unsaved set>'.
- Step 3. Specify the properties as required, then click the Save... button.

Properties		• • ×
<unsaved set=""></unsaved>	Save	Apply

Step 4. Enter a name called Trimming Beam for the saved set.

ve State	
Specify the the name of the state to save: 🔞	ОК
	Cancel

How do I create a property set based on an existing entity in the model?

Properties for an existing entity in the model can be saved as a property set by.

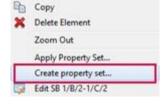
- Step 1. Right clicking on an entity (say an edge beam) you wish to save the properties for.
- Step 2. Select Create property set... from the context menu.
- Step 3. Enter a unique property set name called Edge Beam.

2.11.2 How do I recall a previously saved property set?

To create a new entity

Provided you have previously saved a property set, you can recall it again later from the Properties window - but only when it is applicable to the current command.

For example, assume different steel beam properties have previously been assigned to the main, secondary and edge beams in a structure, each being saved to a set for re-use. You subsequently want to re-use the edge beam properties.



To recall the edge beam property set:

- Step 1. Click the Steel Beam command on the Model tab
- Step 2. Click the drop list at the top of the Properties window only the previously saved steel beam property sets are displayed.
- Step 3. Choose the saved Edge Beams property set.



Properties previously defined and applicable to steel beams only are displayed in the Properties window.

To apply to an existing entity

Step 1. Select a few edge beams in the model



If multiple entities of different types are selected, then property information is displayed separately for each type. A dropdown menu at the top of the Properties window is used for moving between types.

Step 2. Select the type to work with.

Properties		0 ₽ X
Steel Beams Rolled Non-Composite: 2 items		Save Apply
Steel Beams Rolled Non-Composite: 2 items	-	
Steel Columns Rolled Non-Composite: 1 items		

Step 3. Click Apply...

Properties			Π÷Χ
Steel Beams Rolled Non-Composite: 2 items		Save	Apply
Steel Beams Rolled Non-Composite: 2 items	-		
Steel Columns Rolled Non-Composite: 1 items		-	

Step 4. Select on Edge Beam from the list of the property set and click OK to apply the change.

ect property set	ĺ
Frimming Beam Edge Beam	ОК
	Cancel

Property sets can be managed from the Home tab using the Manage Property Sets command.





The different property set types such as line elements are listed. Click the + icon to expand the list to show the sub-items.

To import or export saved property set, select the required property set in the list and click the Export or Import button. Give a filename and save it.

⊒-√ [Line Element	ОК
	Cancel
	Delete
	Export.

To delete a saved property set, ensure the property set is ticked and click the Delete button.



32 (257)

3 Model Validation

3.1 Introduction

The purpose of Validation is to trap errors that will cause the solver to fail before the model is submitted for analysis.

3.2 How Do I Run Model Validation?

Validation is automatically performed as part of an analysis or design. You can however, run a manual validation at any time from either the Model or Load tab with the Validate command.

The actual validation checks that are performed can be set within Model Settings accessible from the Home tab.

- Step 1. Open the example model file 1_Model_Validation_Start.tsmd.
- Step 2. Review Model Settings from the Home tab to see the validation checks that are included in a validation check.

···· Units]·· References				2D element quality			OK
	Error limit for length	10.0	mm	Error limit for quality	0.10	%	Cance
- Drawings	Warning limit for length	100.0	mm	Warning limit for guality	5.00	%	
 Grouping Material List 	warning innerior lenger			warning innertor quarty	1.500.00	10	Save
• Beam Lines	Check for validation warnings:						Load
 Rigid Zones Curved Beams 	Model						
Validation	📝 Wall overlap						
- Load reductions	Point, line or patch load applied to a meshed wall						
EHF	Curved Member axially loaded					=	
	Area load not defined within a panel area						
	Loads outside the member length						
	Temperature load defined on rigid diaphragm						
	Temperature load defined to non-meshed slab						
	Member may not be supported as intended						
	Column solver node is outside the slab plane						
	Beam centre-line is outside slab plane / boundary						

Step 3. Click Validate command on either the Model or Load tab.

The validation checks are performed on the model and if any issues exist these are displayed as Errors or Warning messages within the Status tab in the Project Workspace.

3.3 What are Validation Errors and Warnings?

The results of model validation are broken down into two distinct types; Errors and Warnings.

- Error messages must be rectified. You are unable to complete an analysis or design without resolving.
- Warning messages are for your information and usually relate to a design assumption. Review the message and take action if required or ignore if you are happy with the assumption.

- 3.4 How do I Locate Validation Errors?
- 3.4.1 Working with the status tab in the project workspace

The Status is used to review the validation status for the model and analysis. The analysis model validation is performed automatically as part of the analysis process so we will not look at analysis validation in this session.

- Step 1. Go to Status tab (1) in the Project Workspace and expand the Validation > Model branch by clicking on the "+" icon to display the model validation errors.
- Step 2. All model validation errors are listed under the Model sub-branch.
- Step 3. Ensure the Structure 3D View is active (2).
- Step 4. Double left click the error or warning message you wish to investigate (3).

Status	0 7 ×	🦻 Structure 3D 🛛 🗙 🥻
 Validation Model General Member collision Member intersection Slab/roof overlap A member may not be supported as intended Panel is not surrounded by load carrying members. RL2 X 13 	Double left click the error or warning message to highlight issue in the active view	the 3
 Analysis Analysis Model Geometry Wind Model Meshing Decomposition BIM Design 	load distribution cannot be carried out	
🏘 Structure 😵 Groups 🛁 Loading 🗮 Wind 🚺 Status	1	

Step 5. Provided the item is visible in the active view it will be highlighted ready for investigation.Step 6. Use Scene Content to make the view clearer if necessary.



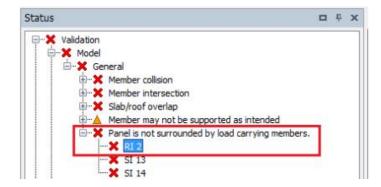
Investigate the highlighted item in the View to determine the issue in conjunction with the validation message.

3.5 Common Validation Errors

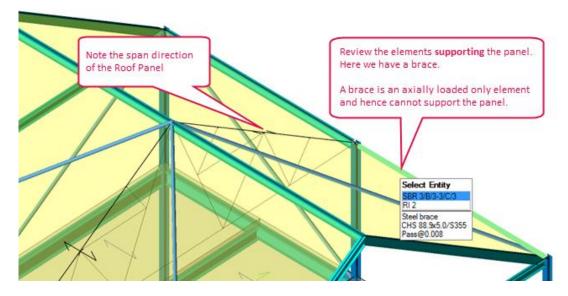
3.5.1 "Panel is not surrounded by load carrying members"

RI (Roof Item)

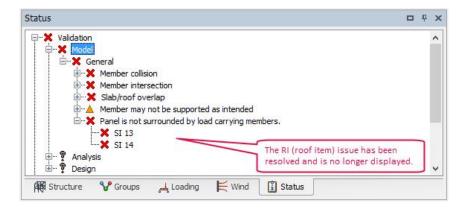
Step 1. Locate the reported roof item (RI 2) by double clicking on it.



Step 2. Review the elements supporting this roof panel. The panel must be bounded by beam members.

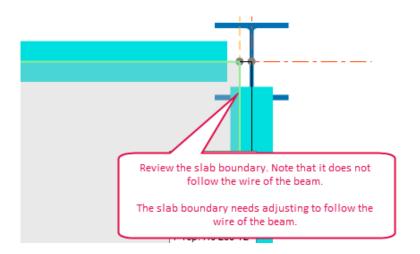


Step 3. To resolve, change the supporting element from a Brace to a Beam element. Step 4. Validate again to see if the issue has been resolved.



SI (Slab Item)

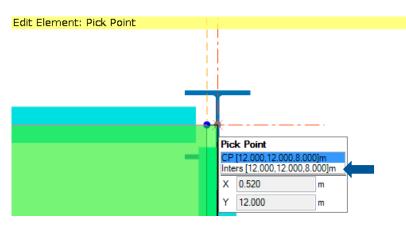
- Step 1. Open plan view of St.1 (2) 2D
- Step 2. Locate the slab item (SI 14) by double clicking on it in the error message.
- Step 3. Review the elements supporting this slab panel. The panel must be bounded by members.



Step 4. To resolve, select the slab panel. Hover over the top right corner of the panel and using the Select Entity tooltip select the Node on SI 14.



Step 5. Move the cursor over the beam and column intersection and use the Select Entity tooltip to highlight the required location - use tab to pick on "Inters" (intersection point).



Step 6. Hit Enter or left mouse click move the panel node to the new location.

Step 7. Repeat Step 3-5 on the other node of the panel to align the panel edge with the beam. Step 8. Validate again to see if the issue has been resolved.



Both panels have been resolved because the 1st and 2nd levels are duplicate levels. Change something on one level and the change also occurs on the other level.

3.5.2 "Members collision"

itatus	🗆 🕂 🗙 🥪 Structure 30 🛛 🖗 St. 2 (2) 20
X Valdation X Valdation X General B 3/A/1-3/B/1 SR 3/A/1-3/B/1 X SB 3/A	
All Structure V Groups A Loading	E Wind 👔 Status
Properties	D \$ X
Member Edge(s): 2 items	Save_ Apply_
General Name User name Section Grade Unearity	The two entities are highlighted. Hover over the entities and note the Select Entity tooltip shows two items in the same location.

Step 1. Double click on the reported member under 'Member Collision' error list to locate it.

Note there are two entities that collide with each other.

- Step 2. Select the entity that is not required, in this case the Brace and Delete.
- Step 3. Validate again to see if the issue has been resolved.

-X Validation		6	~
E-X Model	The member collision has been resolved		1
Member intersection Slab/roof overlap Amalysis	ted as intended		

3.5.3 "Member Intersection"

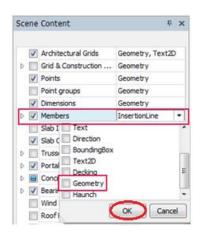
Step 1. Double click on the member under 'Member Intersection' error list to locate it in the View.

	E	🖬 🖡 🗶 Structure 30 🛛 😺 St. 2 (2) 20		
X Valdston X General X General X General X Seneral	Double cl	issue in		
Structure V Groups A Los	ding 🗮 Wind 📋 Status			
roperties		n f x	ア (い)	
Member Edge(s): 4 items	e Save		3	
Member Edge(s): 4 items			B	
Member Edge(s): 4 items General Name			U	
Member Edge(s): 4 items General Isame User name				
Nember Edge(s): 4 items General Name Uter name Section	• Save_ (B	7
Nember Edge(s): 4 items General Users User name Section Grade	• Save_ (P	
Member Edge(v): 4 items General Name User name Section Grade Linearity	 ■ Save_ 1 S355 Straght 		B	
Member Edge(s): 4 Rens General Name User name Section Grade Unearity Solation	Save_ / S355 Straight O*		B	
Mender Edge(Q): 4 Rems General Jame Jame Section Grade Linearity Linearity Rotation Rotation Rotation Rotation	▼ Save / 5355 Steight 0* 9.0000*		G	
Mender Edge(3): 4 Hens Ceneral Jann Jaan Jaan Jaan Jaan Secton Grade Lineerty Rotation Rotation Rotation Rotation Rotation Rotation	Save		G	
Mencher Edge(§): 4 Jenns Ceneral Tenne Section Grade Linearly: Rotation Rotation Rotation Canton angle Canton angle Canton angle	Save_ / Save_ / S355 Sreight 0.000* 0.000* 0.000* 0.000*		3	
Mender Edge(3): 4 Hens General Name User name Section Grade Linearity Rotation Rotation Rotation Rotation Gamma angle Gamma angle Top finge cont. rest.	Save		3	
Member Edge(s): 4 Items Ceneral Jame Uter name Section Grade Unearity Rotation	Save_ / Save_ / S355 Sreight 0.000* 0.000* 0.000* 0.000*		3	
Mecher Edge(§): 4 Nems General Name User name Section Grade Linearity Rotation Ratation angle General angle General cost. Top Sange cost. rest. Bottom fange cost. rest. Bottom fange cost. rest. Dettom limits	Save_ / Save_ / S355 Sreight 0.000* 0.000* 0.000* 0.000*		3	
Vender Edgr(3): 4 Hans General Jame Uaer name Secton Grade Unearly Rotation Rotat	Save_ / Save_ / S355 Sreight 0.000* 0.000* 0.000* 0.000*		3	
Mecher Edge(§): 4 Nems General Name User name Section Grade Linearity Rotation Ratation angle General angle General cost. Top Sange cost. rest. Bottom fange cost. rest. Bottom fange cost. rest. Dettom limits	Save_ / Save_ / S355 Sreight 0.000* 0.000* 0.000* 0.000*		3	

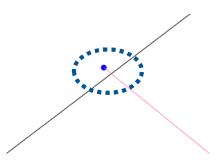
Grid & Construction Lines	Geometry, Text2D
Points	Geometry
Point groups	Geometry
V Dimensions	Geometry
Members	InsertionLine, Geometry, Haunch
Slab Items	Geometry, SlabOutline, SpanDire
Slab Openings	Geometry
Trusses	InsertionLine, Geometry, Haunch
Portal Frames	InsertionLine, Geometry, Haunch
Concrete Walls	
Bearing Walls	
Wind Panels	Geometry, SpanDirection
Roof Panels	Geometry, SpanDirection

Step 2. Use the Scene Content to turn off Slab Items, Trusses, Wind Panels & Roof Panels.

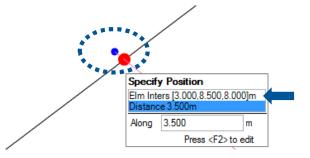
Step 3. Also turn off Geometry of the Members to change the solid shape display to single line display.



Step 4. Zoom in to where the entities meet. Note that one entity over sails the other. One entity should support the other.



Step 5. Select the node at the end of the entity and move the node to snap on the intersection point of the entities - use tab to pick on "Elm Inters"



Step 6. Validate again to see if the issue has been resolved.



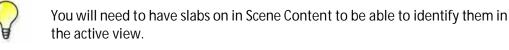
Both level member collisions have been resolved because the 1st and 2nd levels are duplicate levels, i.e. change something on one level and the change also occurs on the other level.

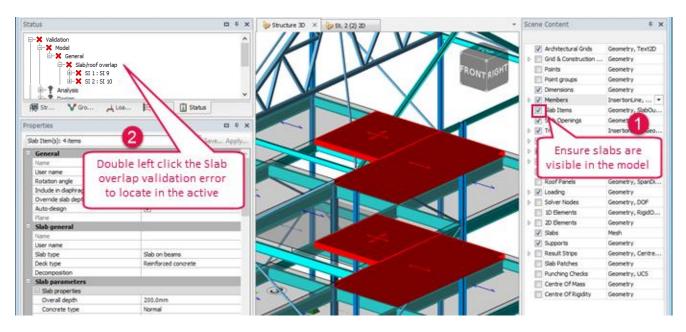


Step 7. In Scene Content check the options to display Members Geometry, Slabs Items and the Trusses again.

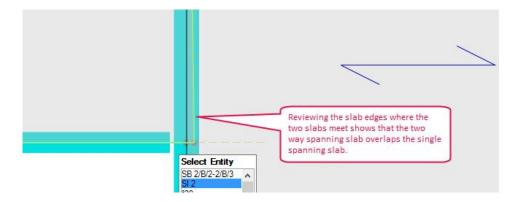
3.5.4 "Slab Overlap"

Step 1. Double click on 'Slab Overlap' to highlight the reported slabs in the View.

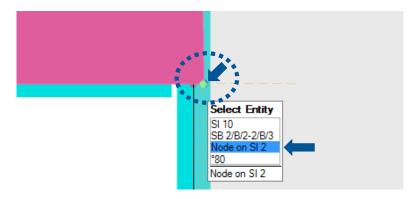




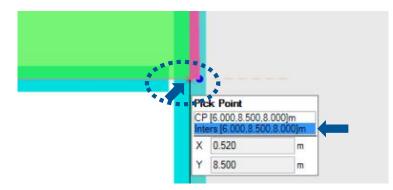
Step 2. Switch to the plan view of St.1 (2) 2D and review the slab boundary edges where the two slabs intersect.



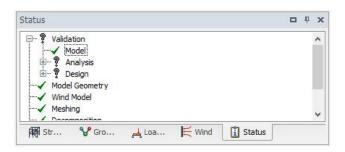
Step 3. Select the Node on SI 2.



Step 4. Move it to the intersection point of the two beam entities using the Pick Point - "Inters" tooltip.



Step 5. Validate again to see if the issue has been resolved



3.6 Right Click Context Menu

Right clicking the mouse either on a selected item or free space of the screen will bring up an additional menu at the cursor position.

			5	Undo	Ctrl+Z
			2	Redo	Ctrl+Y
			-	Сору	
			×	Delete Element	t
				Zoom Out	
				Create propert	y set 🔸
			5	Edit CB 1 H5	
				Open Load An	alysis View
				Open Member	View
				Show Member	Loading
			1	Check Membe	r
5	Undo	Ctrl+Z		Design Membe	er
3	Redo	Ctrl+Y		Design	
Ъ	Сору			Generate Detai	ling Drawing
	Zoom (Dut		Report for Mer	nber
	Check S	labs		Check Slabs	
	Design	Slabs		Design Slabs	
	Redraw			Redraw	
	Save Sc	reenshot		Save Screensh	ot

3.7 Command Activation

When any command is activated a yellow band will be seen across the top of the active view.

 Image: Structure 3D ×
 Image: Structure

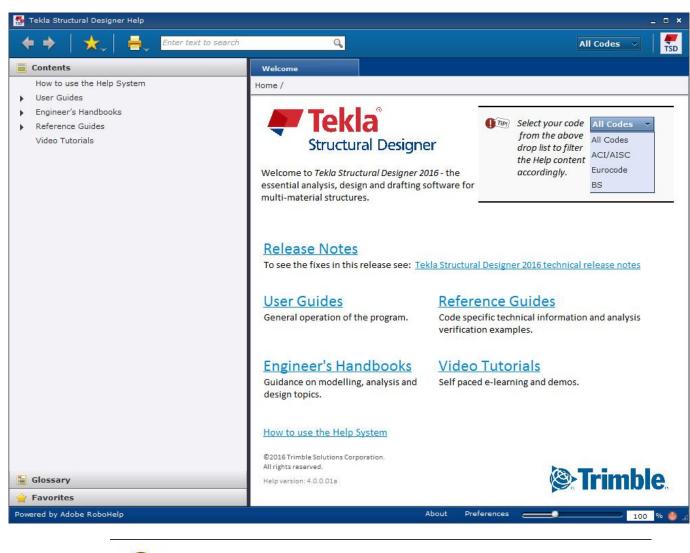
Press the Esc key or double right mouse click to exit a command.



Use the Scene Contents to simplify the view when it becomes too cluttered.

42 (257)

4 Tekla Structural Designer Help



To active the help menu press F1 on the keyboard



You can also click on the *(i)* button at top right corner of the program screen.

44 (257)

5 Multi Material Modelling 2D

5.1 Introduction

- Pre model settings
- Construction levels and gridlines
- Placing structural elements
- Construction lines and free form truss
- Element releases
- · 2D loading loadcases / combinations
- · Validation, analysis and viewing graphical results
- Tabulated results

5.2 Model and Global Settings

5.2.1 Settings

Before creating a new model, the Settings command on the Home tab should be reviewed to ensure the required options are selected. All new models that are created take these settings to generate their own model settings for the new file, and includes options for default design codes, analysis and design options, default section sizes, label and colour settings and detail drawing options.

Step 1. Launch Tekla Structural Designer 2016.

Step 2. Click on the Settings command on the Home tab.

File	Home	- 1	Nindows												
	🔁 Open		B	Ľ	B	B	66	2	R	9	R	Cloud Export Robot Export	3D DXF Import	<u>نې</u>	
New	🔛 Close	Save	Save As	Project Wiki	Model	Manage Property Sets	Find	Walk	Structural BIM Import	Tekla Structures Export	Revit Structure Export	'TEL' File Import	JO DAT IMPORT	Settings	Materials
	File		la			Project		l⊒			BIM Integration		Fa.	Settings 🛱	Materials 🗔

5.2.2 Settings Sets

When first opening TSD, you will be prompted to select the country to import the settings for. This specifies the default active Settings Set, which defines all of the default options in the Settings window.

The selected Settings Set can be changed by selecting Settings Sets on the left hand side of the Settings window, then by clicking the Import button. You can import multiple different Settings Sets for different regions, create new, customised sets to suit specific clients or project types, and edit existing sets. Settings Sets are edited by selecting the required set in the drop list, and then making the required changes on the other pages of the Settings window. The Settings Set that is made Active is the one that will be used to create any subsequent new models. The active settings set can also be imported into existing models.



For more information on this topic, please refer to the Help system.

Structure Defaults Section Defaults References Drawings Solver Solver Help	Settings Settings Sets Settings S	Select the settings set to edit UK Settings (active)	Add >> Active Import Rename Remove Open Folder	OK Cancel
	… Structure Defaults … Section Defaults … References … Drawings … Solver … Scene			

Step 3. Ensure you have "UK Settings" set selected and make it Active.

5.2.3 Design Codes

As some regions have multiple different design codes, or with the possibility of you designing a structure that will be located in a different country, you may need to change the design code to be used. This is done by selecting Design Codes on the left hand side of the Settings window. You can then select the Head Code option to make an overall change to all Action and Resistance Codes to be used, and then adjust the specific design codes and years for specific areas if required.



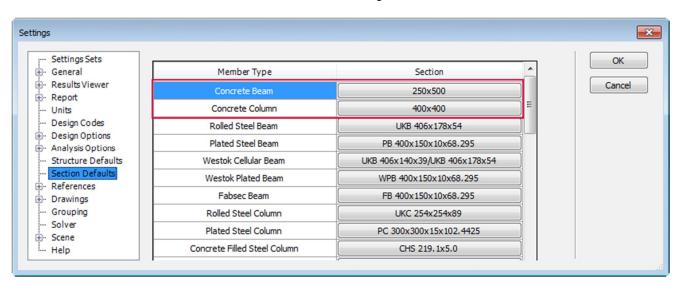
Currently Design is only available for British Standards, Indian Codes, Eurocodes, ACI/AISC codes.

General Results Viewer	Head Code Select the head code United Kin	ngdom (Eurocode) 🛛 👻					OK
Report Units Design Codes Design Options	Design Codes Action Codes Resistance Codes						
Analysis Options	Resistance Code	Design Code		Year		Info Text	
Structure Defaults Section Defaults	Steel Design	BS EN 1993-1-1 + UK NA	-	2005	-		
References	Concrete Design	BS EN 1992-1-1 + UK NA	-	2004	-		
Drawings	Composite Design	BS EN 1994-1-1 + UK NA	-	2004	-		
Solver Scene	Timber Design	BS EN 1995-1-1 + UK NA	-	2004	-		
Help	Masonry Design	BS EN 1996-1-1 + UK NA	-	2005	-		
	Foundation Design	BS EN 1997-1 + UK NA	-	2004	-		

Step 4. Ensure "United Kingdom (Eurocode)" head code is selected.

5.2.4 Section Defaults

If there are certain section sizes that you use regularly, or a section that will be commonly used in a project, you can also specify default section sizes for concrete beams and columns by selecting Section Defaults on the left hand side of the Settings window.



Step 5. Accept the default section sizes.

5.2.5 Analysis Options

As well as options for the various different analyses, the key settings for concrete structures are the concrete Modification Factors, found under Analysis Options > Modification Factors > Concrete. There are factors for the Building Analysis (i.e. the 1st Order Linear Analysis), the Grillage Chase-down and the FE Chase-down, and allow you to make adjustments to the stiffness of the various element types, for cracked and uncracked sections, to allow for long term effects. These adjustments will naturally have an impact on the analysis results, so it is important to set them as required.

Design Codes	Element Type	E	G	I torsion	I major	I minor	Area	A minor	A major	t	OK
Design Options Analysis Options	Mid Pier Wall Cracked	0.200	0.200	1.000	1.000	1.000	1.000	1.000	1.000		Cance
- 1° Order Non-Linear	Mid Pier Wall Uncracked	0.400	0.400	1.000	1.000	1.000	1.000	1.000	1.000		
- 2 nd Order Non-Linear	Meshed Wall Cracked	0.200	0.200							1.000	
- 1 [°] Order Vibration 2 nd Order Buckling	Meshed Wall Uncracked	0.400	0.400							1.000	
- Modification Factors	Column Cracked	1.000	1.000	0.200	0.200	0.200	1.000	1.000	1.000		
Concrete Building Analysis	Column Uncracked	1.000	1.000	0.400	0.400	0.400	1.000	1.000	1.000		
Grillage chase-down	Beam Cracked	1.000	1.000	0.010	0.200	0.200	1.000	1.000	1.000		
FE chase-down	Beam Uncracked	1.000	1.000	0.010	0.400	0.400	1.000	1.000	1.000		
⊕- Steel ⊕- Timber	Flat Slab	0.200	0.200							1.000	
	Beam and Slab	0.050	0.050							1.000	

Step 6. Review the various Modification Factors.

5.2.6 Concrete Design Options

All automated concrete designs are completed based on the concrete Design Options. There are settings available for the designs of beams, columns, walls and slabs, controlling minimum and maximum bar sizes, spacing, links, reinforcement arrangements and detailing options, amongst others.

Settings			
 Preport Units Design Codes Design Options General Analysis Concrete Reinforcement Parameters Beam Detailing Options Top Longitudinal Bar Pattern Bottom Longitudinal Bar Pattern Link Settings General Parameters Column Reinforcement Layout Detailing Options General Parameters Stab Design Forces Design Forces Autodesign 	General Country Longitudinal bars Minimum bar size Maximum bar size Minimum side bar size Minimum top steel clear spacing Minimum bottom steel clear spacing Maximum tension steel spacing Maximum compression steel spacing Use single bars when beam width <= Short Span Short span maximum length	50.0 r 200.0 r 200.0 r 150.0 r	mm mm mm mm mm

Step 7. Review the numerous concrete Design Options.

5.2.7 Drawings

A wide variety of detail drawing settings are also available within the Settings window. These can also be set up in anticipation of creating all new models to control content, layers, colours and more, but these settings will be discussed in more detail in a later section of this course.

Units	Drawing Variant		Availa	ble Styles									OK
Design Codes	Concrete Beam Deta	il 🔽		Sec. 2 and the second			-					1	
Design Options General			Deam	Decan		Add							Cancel
- Analysis	Concrete Beam Deta				6	Add copy							
+- Concrete	Concrete Column Dei Concrete Wall Detail						2						
Composite Beams	Non-concrete Beam I				3	Remove	100						
- Design Forces	Non-concrete Colum				100								
Design Groups	Active Style	Decun	1										
Autodesign			_										
Display Limits	Name Bean	Detail							A	pply	to All.	•	
Steel Joists	Layers:												
Analysis Options								_			Font	A	
Structure Defaults	E Description	Name	Is Merged	Merged with	Color	Report	Line Type		Font		Size		
Section Defaults			Merged			Color					[mm]	-	
References Drawings	Grid Lines	Grid Lines	100				Dash Dot	-				=	
- Export Preferences	Axis Text	Axis Text	100				Solid	-	Txt	- 5	.0		
≟- Layer Configurations ≟- Layer Styles	Axis Balloons	Axis Balloons					Solid	-					
- Planar Drawings	Section Text	Section Text					Solid	-	Txt	- 2	.5		
Member Details	Dimensions	Dimensions					Solid	-	Txt	• 2	.5		
— Member Schedules — Foundations	Thick Symbol Lines	Thick Symbol Lines					Solid	-					
- Options	Thin Symbol Lines	Thin Symbol Lines					Solid	-					
🗄 Planar Drawings	Small Text	Small Text	m				Solid	-	Txt	- 2	.5		
	Steel Bar Lines	Steel Bar Lines				Ì	Solid	-				+	
±- Member Schedules													

Step 8. Review the Drawing settings, then click OK to confirm all setting changes.

5.3 Materials

As well as the variety of analysis, design and general settings, the Materials command on the Home tab should also be reviewed to ensure the required options are selected. All new models that are created also take these material settings to generate their own model settings for the new file, and includes options for default concrete and reinforcement grades and rebar sizes.

Step 1. Click the Materials command on the Home tab.

5.3.1 Concrete Grades

The Head Code setting under the Material options allows you to choose concrete grades from a variety of different regions. You can then set the default concrete grade to be used, and add additional grades if the one you want to use is unavailable.

Step 2. Review the Material settings.

Sections				Close
	Current database	version: 12		
- Shear Connectors	Head Code	United Kingdom (I	Eurocode) 👻	
- Model	Material Type	Concrete	-	
	Concrete Type	Normal	-	
	Available Grades			
	Default Grade	C32/40		
	C16/20 C20/25		Add	
	C25/30 C28/35		View	
	C30/37 C32/40		Delete	
	C35/45 C40/50 C45/55 C50/60 C55/67 C60/75		>> Default	

5.3.2 Reinforcement

As with the Material options, the Head Code setting under the Reinforcement options allows you to choose steel grades from a variety of different regions. You can then set the default rebar steel grade to be used, and select which bar sizes are available for the design of your structure. Additional bar sizes can also be assed to the list for selection.

Materials Sections Material						Close
- Reinforcement Metal Decking	Current data	base version: 10				
- Shear Connectors	Head Code	United Kingdom (Er	urocode) 👻			
I Model	Country	UK	•			
	Type	Loose bars	-			
	Rib Type	Type 2	•			
	Available class Default Class		Add	Available sizes	Add	
			View	12 16 20 25 32 40	View	
			Delete	20 25	Delete	
			>> Default	32 40		

Step 3. Review the Reinforcement settings, then click Close to confirm them.

49 (257)

5.4 Templates

As well as being able to use the Settings and Materials options to generate the Model Settings for a new project, another option is to use Templates. A Template can be used to specify all of the same settings as detailed above, but they can also include modelling, such as grid lines and elements. This could useful for when you have multiple models to create that all use the same gridlines, or perhaps have a common core area.

New templates are created by first creating a new file by clicking the New command on the Home tab. Once this blank project is created, you can then edit the Model Settings as required, as well as insert any particular grids, section sizes and elements that you want in the template. Once this is done, simply go to File > Save As, choose a name and location of where you want to save the template, and set the Save as Type file format to *.tsmdt.

File name:	Test1.tsmdt 🗸	
Save as type:	Tekla Structural Designer template file (*.tsmdt)	

Once a template file has been created and saved, you can then use the drop down arrow under the New command on the Home tab and choose your template. This will actually open the template file, so you then just need to save this as a *.tsmd file so that you can continue to model the structure and leave the template file as it is.

File	Home	Windows		
	🕞 Open		Ľ	B
New	Close Sa	ve Save As	Project Wiki	Model Settings
T	est1	6		
a 0)pen Template			

5.5 Starting a New Project and the Project Wiki

Step 1. Click on the New command on the Home tab to start a new project.



To enter details about the project into TSD click the Project Wiki command on the Home tab.

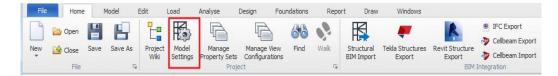
File Home Model Edt Load Analyse	e Manage View Find Walk Structural Tekla S	Indows	Cloud Export 30 D/xF Import TBL' File Import 50 D/xF Import 50 Settings 6 Materials 6 Uncome
ructure Project Wild	Project Name Job Structure Engineer teyka History Created: 29/(2015 Un:13-47 AM by teyka (v. 16:0.0.35) Last saved: <	Checked By Name Date 28/ 3/2016 * Check Approved By Name Date 28/ 3/2016 * Approve	Cancel



Within the Project Wiki the Metrics of the model can be displayed – number count of analysis and physical elements

5.5.1 Model Settings

When a new model is created the settings and parameters of that open model is contained in the Model Settings.



The Model Settings are specific to each model and are based on the previously defined Settings and Materials options, as detailed previously. This means when they're edited, they will only affect the currently open model. If required, you can load your currently active global settings into an existing model by clicking the Load button in the Model Settings window. This may also be necessary in other windows, such as the Analysis or Design Options windows.

Units	Object	Reference Format	Edit
References General	Steel Beam	M C L1 / P1 - L2 / P2	Edit E
- Formats	Steel Column	M C P1	Edit
Texts	Steel Brace	M C L1 / P1 - L2 / P2	Edit
Drawings General	Steel Joist	M C L1 / P1 - L2 / P2	Edit
Types	Steel Truss Member Top	M C L1 / P1 - L2 / P2	Edit
Layer Styles Options	Steel Truss Member Bottom	M C L1 / P1 - L2 / P2	Edit

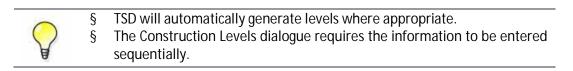
5.6 Modelling – Establishing the Structural Geometry

5.6.1 Construction Levels

The methodology we are going to adopt to create this structure is to create an individual frame and copy and repeat to produce the building. First let's establish the initial levels of the building.

- Step 1. Select on the Construction Levels command from the Model tab.
- Step 2. Click the Insert Above button to generate the levels as seen below.
- Step 3. Enter the details as shown.
- Step 4. Press OK when finished.

File	Home	Model	Edit	Load	Analyse		Design	Report	Draw	1	Nindows				
	K				 ←→		-	5				\cap			
Construction Levels	Frame	Sloped Plane ~	Grid Line	Construction	Dimensio	n	Steel Column 🔻	Steel Beam 🔻	Steel Brace *	Steel Joist		Portal Frame	Concrete Wall	Concrete Column	Concrete Beam
Le	vels	Tai .	Grid 8	& Construction I	lines	15			Steel			Es.		Concrete	1
Structure	ture	Constru	uction Level	s											3
⊡ 🛱 Struc	evels .		uction Level Ref	s Name	Туре		Level[m]	pacing [m]	Source		Slab Th. [mm]] Floor		бк	3
⊡ 🛱 Struc				Name	Type T.O.S	_	Level[m] 9.000	pacing [m]	Source -unique-	e 1	Slab Th. [mm]				
⊡ 🛱 Struc	evels .		Ref Eav	Name		•			22	-	Slab Th. [mm]	v	c	ОК	
່≣∰ເ	evels .	3	Ref Eav Tru	Name /es	T.O.S T.O.S	•	9.000	1.500	-unique-	-	Slab Th. [mm]		C	OK ancel	



5.6.2 Grid and Construction Lines

To place any structural element in the modelling environment an intersection position has to be created.

An intersection point can be created using Grid Lines or Construction Lines and the crossing of two lines will allow a structural element to be placed at that point.



§

Grid Lines are created on a horizontal plane.

§ Construction lines are created on sloped, vertical and inclined planes.

We will used a wizard to create a rectangular grid system in the model.

Step 1. Click on the drop list of the Grid Line command and select Rectangular Wizard.

File	Home	Model	E	dit	Load	Analys	se	Desig	n
$\not\equiv$	ß		E						
Construction Levels	Frame	Sloped Plane 🕆		angular ard ▼	Constructio	n Din	nension		Steel
L	evels	Fai	1	Grid L	ine		5		
Structure			11	Paralle	el		• P	×	S 🔊
🖃 🛱 Stru	cture		11	Paralle	el (quick)				
			\times	Perpe	ndicular				
±	Sub Mode	s	Ħ	Recta	d				
				Sector	r Wizard				
			7	Arc					
				Impor	t DXF				
			_			_		- 11	

Step 2. The first page of the wizard allows you to enter a name for the grid and set which levels it will appear (this can be edited after creation) press Next.

Grid		
	St. 3 (Eaves)	9.000
Color	 St. 3 (Eaves) St. 2 (Truss Botto St. 1 (First Floor) St. Base (Base) 	om) 7.500
	St. 1 (First Floor)	4.000
	St. Base (Base)	0.000
		Finish
Cancel Pr	evious Next	

Step 3. Set the grid origin position at the defaulted values of X=0, Y=0 and press Next.

	gular Grid Wizard	
Sele	ect Origin	
x	0.000	m

- Step 4. The next dialogue will ask the directions of lines to be generated accept the defaults (All Lines) and control the style of the lines press Next.
- Step 5. Generating lines in the X direction enter Count=5 & Length=6m and press Next.

ays					Direction
Regular Count	5	Length	6.000	m	Positive X
) Irregular Lengths				m	O Negative X
Name Grids by	Alphanum	ieric	~		
Name Grids by	Alphanum	ieric	~		

§ Labelling of the gridlines can be selected by letters or numbers.
§ If an irregular pattern of grids is required – use a "," as a separator or "*" for multiples.

Step 6. Generating lines in the Y direction enter Count=2 & Length=12m and press Next.

tangular Grid Wizard				83	
Bays Regular Count Irregular Lengths Name Grids by	2 Length Numeric	12	m	Direction Positive Y Negative Y	
Cancel	Previous	Next		Finish	

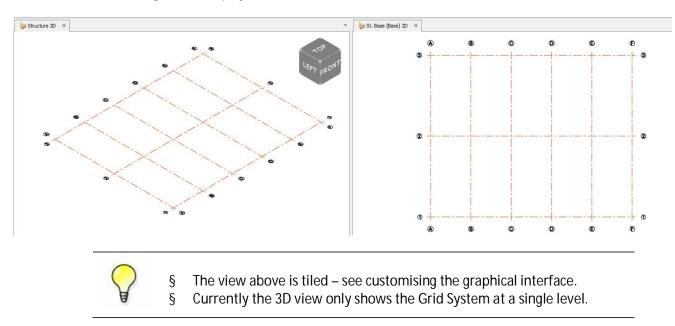


§

As information of the grid is being placed into the wizard the background screen (base level) will adjust accordingly.

§ Pan and Zoom will be functioning as mentioned in the mouse controls earlier.

- Step 7. Grid Rotate accept default values of 0 degree.
- Step 8. Axis Angle (rectangular grid skewed) accept default values of 90 degrees relative to X axis and press Finish.



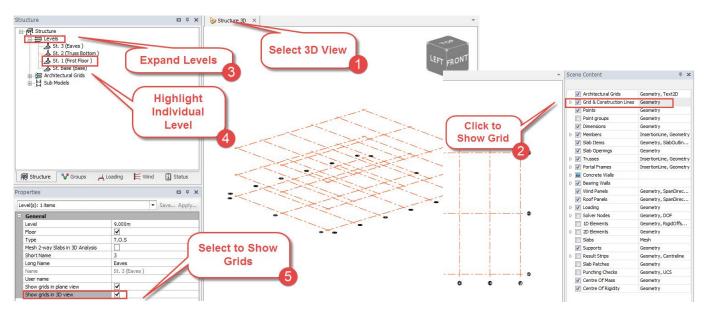
The following will be displayed in a 2D and 3D view:

5.6.3 Graphically displaying Grids

Currently TSD is showing the Architectural Grids at the lowest level in the 3D view.

To make the gridlines visible on the level you wish.

- Step 1. Select the Structure 3D view.
- Step 2. Turn on Grids and Construction Lines display from the Scene Content.
- Step 3. Expand the Levels branch under the Structure in the Project Workspace Structure tab.
- Step 4. Highlight an individual level.
- Step 5. In the Properties window, tick the Show grids in 3D view option.







The same process can be adopted for the plan 2D views.

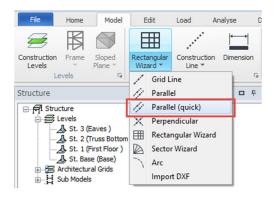
Vertical lines can be displayed to make intersection viewing easier between levels.

- Step 6. Open the Scene Content.
- Step 7. Next to Grids and Construction Lines, click on the 'arrow head' to display more options.
- Step 8. Select Vertical Lines and press OK to display.

5.6.4 Adding additional gridlines

In this section we will add two additional gridlines to the main grid.

- Step 1. Go to St. Base (Base) 2D view and select the Model tab.
- Step 2. Pick the drop list option under Rectangular Wizard / Grid Line command.
- Step 3. Pick the option Parallel (quick).



Launching this operation will place a yellow information bar across the active window.

Step 4. Select on gridline 2 to act as a reference position (position to measure from).Step 5. Move the cursor vertically to specify an offset direction.



You can click any position graphically to specify a parallel gridline on screen.

Step 6. Specify an actual distance by pressing the F2 key and enter a value of 6m.

Step 7. Place parallel gridlines above and below the reference line.

File Home Model	Edit Load	Analyse	Design	Report	Drav	v Wi	indows								
🕖 🛱 🖾	11 1		-	55				\cap					2	1	+
Construction Frame Sloped	Parallel Construction	Dimension	Steel	Steel	Steel	Steel	Steel	Portal	Concrete	Concrete	Concrete	Slab on	Slab Opening	Slab Split	
Levels * Plane *	(quick) V Line V	Dimension	Column *		Brace *	Joist	Truss *	Frame	Wall	Column	Beam	Beams *	Siab Oper in ig	Sido Spire	Sido Join
Levels 🖬	Grid 8 Construction	Lines 5	a		Steel			G		Concrete	Gi		Slabs		(
tructure				>> Structure			ase (Base)								
- A Structure			Ac	ld Paralle	Copy:	Specify	Distanc	ce(s)							
E Evels															
St. 3 (Eaves)	2					A		B		©	1	D	Ð		Ð
St. 1 (First Floor)						ø		Ð		0	4				U
St. Base (Base)					3	+									6
😥 🚝 Architectural Grids						1									
						i .		- i		- i -		i	1		- i
						1									
						1				1					
					۲	+									(
						1				1					1
						i i		- i		i i		i	i.		- i
						1									
						1					4				
					2	-					φ				0
Structure V Groups	Loading 📙 Win	d 🚺 Stat	us												
roperties			ά×			1			6	6.0 <mark>0</mark> 0					
		 Save Aj 	oply			1		1	Spe	cify Dista	nce(s)		1		
General						†				ance 6.000					÷
Architectural Grid	Main Grid					1				ances 6.		m			
Detect Name Type	✓					1		1			Press <f2></f2>		1		1
Vertical Grid Name	Alphanumeric										1000 1727	to out			
LineStyle	DashDot														
Extend Line						1									
Extension Length	500				O	+						+··			(
											- Cl				
						A		B		Ô	4	D	Ē		Ð

Step 8. Press Esc or double right mouse click to exit the operation.

- 5.7 Selecting Elements for Editing
- 5.7.1 Gridline Properties

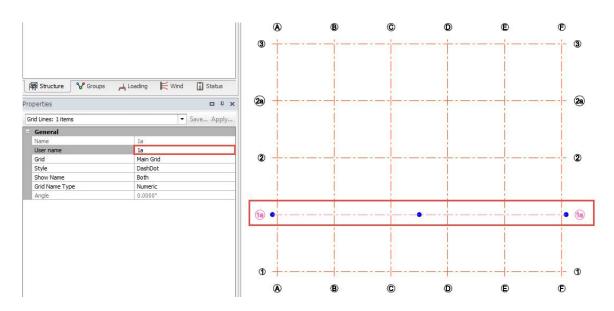
The new horizontal lines have grid label out of sequence with the rest of the grid system.

To change the properties of any gridlines the element need to be selected and the grid label rereferenced.

Step 1. Select on gridline 4 to be changed by left clicking on the item (notice the pink selection colour).

Look at the Properties window and the editing options for the selected gridline are displayed.

- Step 2. Go to User name and enter the new name as 1a.
- Step 3. Press Enter to apply the change.
- Step 4. Press Esc to exit the selection process.
- Step 5. Repeat the same procedure to rename gridline 5 to 2a.



5.8 Dimensioning and Measuring Tools

5.8.1 Dimensions

A dimension can be added to the scene in either the 2D or 3D view.

- Step 1. Go to Base 2D view.
- *Step 2.* Click on the Dimension command and follow the instructions given in the operation yellow bar create some dimensions.
- Step 3. Press Esc to exit the operation.

File	Home	Model	Edit	Load	Analyse	Desig	'n	Report	Drav	v W	indows				
Ø	ß		111		↓			S				\square		1	
Construction Levels	Frame	Sloped Plane T	Parallel (quick) 🔻	Construction Line 🔻	Dimension		eel mn ▼	Steel Beam 🔻	Steel Brace ▼	Steel Joist	Steel Truss 🔻	Portal Frame	Concrete Wall	Concrete Column	Concrete Beam
L	evels	G	Grid 8	& Construction I	ines 🖬				Steel			G		Concrete	5
Structure						φx	D	Structure	3D	🧽 St. Ba	ase (Base)	2D ×			
	cture						Cre	ate Dim	ension:	Pick Po	int 1				
9	Levels														

5.8.2 Measuring tool

Under the Home tab there are options for Measure and Measure Angle command – follow the instructions given in yellow information bar.

Timber Column	Timber Beam	Timber Brace ▼	Timber Truss 🔻	Column	Truss	Roof Panel Wa	all Panel	Support Element	Measure Measure Angle Bearing Wall	Validate	
	Tim	her	5	Cold Formed	5	Panels	Fai	Miscella	neous 5	Validate	

5.9 Deleting Elements

5.9.1 Delete command

There are several methods to delete an established object in the model. To delete any object (dimension), from the Quick Access toolbar click the Delete button.





The quick access toolbar is an access point for popular commands, use the drop down arrow to modify.

Step 1. Follow the instructions given in the yellow command bar – delete a dimension.

Ν	odel	Edit	Load	Analyse I	Design Fou	indations	Repor	t Draw	Windows							
			B	B	B	66	2	R		R			1	Cloud Export		ŝ
		1.00 20 00	10.517707							<u>s</u>				Robot Export	3D DXF Import	
ave	Save As	Project Wiki	Model Settings	Manage Property Sets	Manage View Configurations	Find	Walk	Structural BIM Import	Tekla Structures Export	Revit Structure Export	IFC Export	Cellbeam Export	Cellbeam Import	'TEL' File Import		Settings
	F₂i			Proje	ect		r _a				BIM Integra	ition			6	Settings

Options for Delete:

- Single Left Click on an individual element.
- Left Click and Hold Create a blue box (upper left to lower right) and delete any element totally contained within the box.
- Left Click and Hold Create a green box (upper right to lower left) and delete.
- When multiple elements are selected for deletion a filter box will appear for confirmation.

4 🔽	Dimension	
	V Dim B/3-C/3	
	Construction Line	
	A	
	V D	
	2	
	✓ 2a	

5.9.2 Prior selecting before delete

In selection mode, elements can be pre-selected and then the command of delete activated.

- Step 1. Make a multiple selection of different elements.
- Step 2. To delete the selection press the Delete button on the keyboard or click on the Delete icon in the quick access toolbar.
- Step 3. Delete all dimensions.
- Step 4. Press Esc to exit the operation.

6 Creating Structural Elements

6.1 Concrete Columns

We wish to create five concrete columns on the intersections of gridline B.

The outer columns extending to eaves level and the internal columns to the first floor level.

Step 1. Activate the 2D Base view.

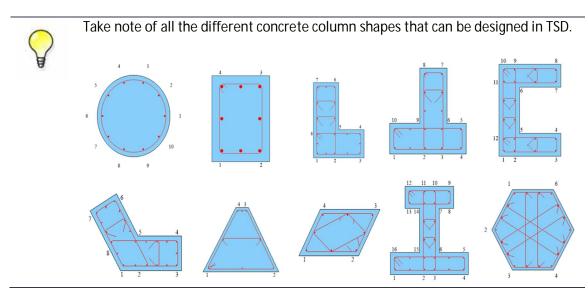
Step 2. Click on the Concrete Column command on the Model tab.



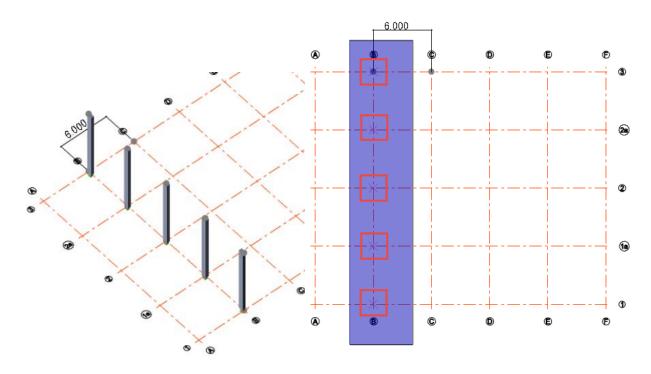
Step 3. In the Properties window under Section pick the option New/Edit. Step 4. Edit the rectangular section to have a depth of 600mm.

			Geometry		
Rotation angle	0.0000*			Destances	
Concrete class	C32/40		Shape	Rectangular 🗸 🗸	
Autodesign	~		Breadth	400.0 mm	
Select bars starting from	Minima				
Section	400x400	~	Depth	600.0 mm	
Automatic alignment	400x400				
Major alignment offset	<new\edit></new\edit>				
Minor alignment offset	0.0mm				
Reinforcement					
Rib type - vertical	Type 2				
Class - vertical	500				

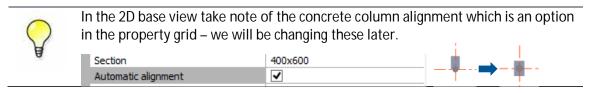
Step 5. Then press the OK button to accept the new section size.



Step 6. In a 2D Base view create a box around gridline 2 – take note of the cross hairs at the intersection points indicating a column is going to be created.



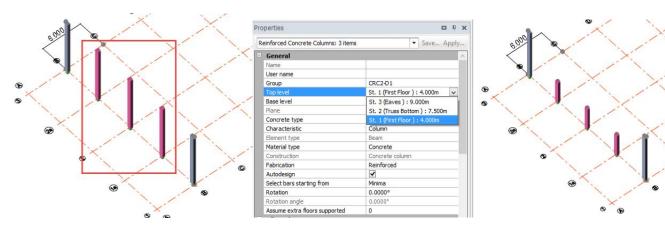
Step 7. Press Esc to exit the operation.



6.1.1 Editing column properties – levels

Changing the column heights of the internal columns.

- Step 1. In either the 2D or 3D view select the internal columns.
- Step 2. In the Properties window adjust the Top level of these columns to St. 1(First Floor).
- Step 3. Press Esc to exit the selection process.



6.1.2 Editing column properties - alignment

Ensure that the outer concrete columns Alignment is set to Centre / Centre.

Step 1. Select the two outer columns.

Step 2. In the Properties window, under Alignment set Major snap level and Minor snap level to be both Centre.

Alignment	
Major snap level	Centre
Major offset	0.0mm
Minor snap level	Centre
Minor offset	0.0mm

- 6.1.3 Changing column characteristics central steel column
 - Step 1. In either the 2D or 3D view, select the central column.
 - Step 2. In the Properties window and change the Material Type to Steel.
 - Step 3. Set the Construction to Non-composite column.
 - Step 4. Tick the Autodesign option.

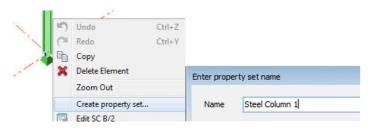
Plane		
Characteristic	Column	
Element type	Beam	
Material type	Steel	
Construction	Non-composite column	
Fabrication	Rolled	X X
Autodesign	v	•
Design section order		
Gravity only		

Step 5. Press Esc to exit the operation.

6.1.4 Changing column properties using property sets

Now that we have established the properties of the central column, we can create a Property Set and use those properties to apply to other elements.

- Step 1. Either in a 2D or 3D view, right click over the central steel column (highlights green).
- Step 2. Select Create property set..." from the context menu.
- Step 3. Enter the name Steel Column 1 for the property set.
- Step 4. Press OK.



Step 5. Select the remaining two internal columns and check the property grid that you have 2 columns selected.

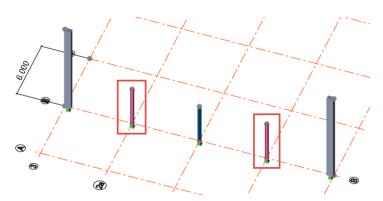
🕅 Structure 🛛 😵 Groups 🛁 L	oading 🗮 Wind 📋 Status
roperties	□ Ŧ X
Reinforced Concrete Columns: 2 items	✓ Save Apply
General	
Name	
User name	
Group	CRC2-D1
Top level	St. 1 (First Floor) : 4.000m
Base level	St. Base (Base) : 0.000m
Plane	
Concrete type	Normal
Characteristic	Column
Element type	Beam
Material type	Concrete
Construction	Concrete column
- 1	

- Step 6. With the columns selected press the Apply button.
- Step 7. The pop up screen will allow you to pick the Property Set you want to apply select on Steel Column 1 and press OK.

Select property set	
Steel Column 1	ОК
	Cancel

Step 8. Press Esc to exit the selection operation.

The Internal columns will have the property set now of steel, created from an element already established in the model.



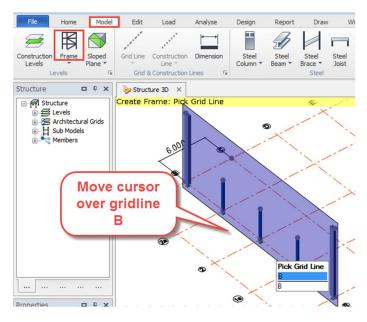
6.1.5 Managing property sets

All created property sets can be deleted or imported/exported to other projects.

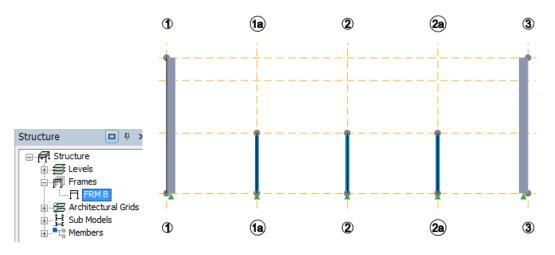
Step 9. On the Home tab, select Manage Property Sets command.

File New	Home	save	Iodel I Save As	Edit Project Wiki	Load Model Settings	Analyse Manage Property Sets Project	Design	Report Walk	Draw EXAMPLE Structural BIM Import	Windows	Revit Structure Export BIM Integration	Cloud Export Robot Export 'TEL' File Import	3D DXF Import	Settings
÷.	Structure					며 무 Manage Prop	`	>> Structur	e 3D ×					
	문 Architec 님 Sub Moo 다 Member	lels	ls				e Element							OK Cancel
														Delete

- 6.2.1 Obtaining a 2D frame / elevation view
 - Step 1. Go to Structure 3D view and in the Model tab click the Frame command.
 - Step 2. Move the cursor over Grid Line B and left click to execute (blue boundary box will appear to show the elevation you are creating).



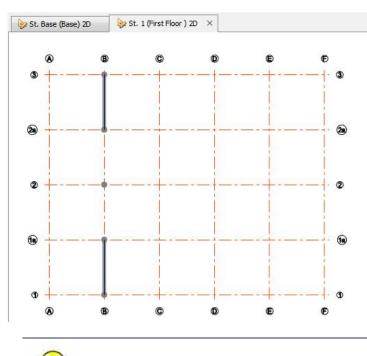
Step 3. In the Project Workspace – Structure tab, under Structure tree expand the Frames branch. Step 4. Double click FRM B to display its 2D view.



6.2.2 Obtaining a plan / level view

Information entered into the Construction Level dialogue will automatically generate a 2D plan level which can be extracted as a working view.

Step 1. In the Project Workspace – Structure tab, under Structure tree expand the Levels branch. Step 2. Double click the level name St. 1(First Floor) to open the 2D plan view.



Zooming into the plan view will show the definition of the structural elements placed into the project

6.3 Concrete Beams

In this section, we will place the following concrete and steel beams at the first floor level along gridline B.

- Concrete Beam size Breath 400mm x Depth 900mm
- Steel Beam Auto Design
- Step 1. Obtain either a 2D elevation\frame view of gridline B or a 2D plan view of the First Floor.
- Step 2. On the Model tab, click the Concrete Beam command.

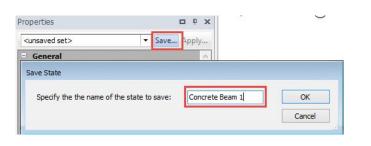
Step 3. In the Properties window, create a new section size of 400mm breath and 900mm depth. Step 4. Press OK.

Select bars starting from	Minima		Geometry	
Section	250x500	¥	Shape	Rectangular 🗸
Automatic alignment	250x500		Breadth	400.0 mm
Major alignment	<new\edit></new\edit>		breaduri	
Major alignment offset	0.0mm		Depth	900 mm

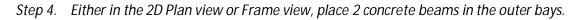
6.3.1 Creating a property set from a new element

When a new element is being placed into the model the properties can be saved.

- Step 1. Press the Save button in the Properties window.
- Step 2. Enter the Property Set name to be Concrete Beam 1.
- Step 3. Press OK.



This property set can be applied to element in the structure by pressing the Apply button.





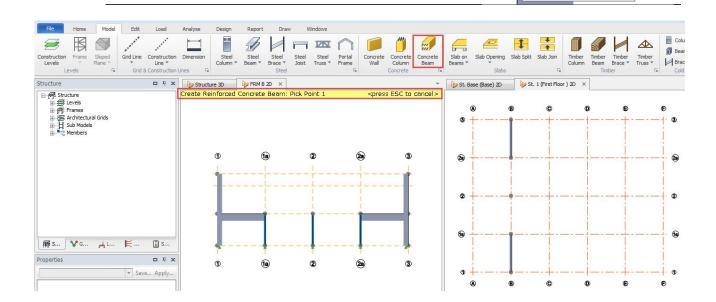
If in a frame view use the Intersection option in the context menu (select by the arrow keys or tab key to pick the point required).

Pick Point 1
Distance 3.995m
Inters [6.000,0.00
X 0.500

4.000

m

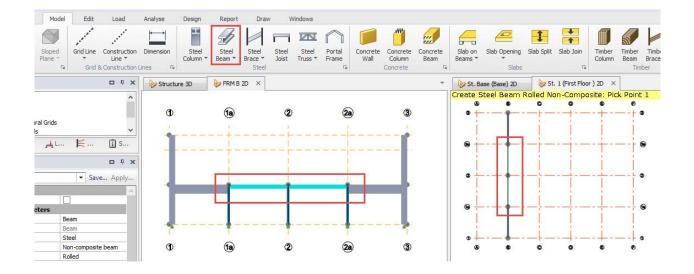
m



6.4 Steel Beams

Using the same methodology, we will place first floor steel beams in the remaining proportion of the frame.

- Step 1. Obtain either a 2D Plan or Frame view of gridline B.
- Step 2. On the Model tab, click the Steel Beam command.
- Step 3. Tick the AutoDesign option in the Properties window.
- Step 4. Place the steel beams in the remaining area of the floor.
- Step 5. Press Esc to exit the operation.



In this section, we will be creating a faceted freeform truss comprised of steel members. We will be using construction lines to create the form of steel truss.

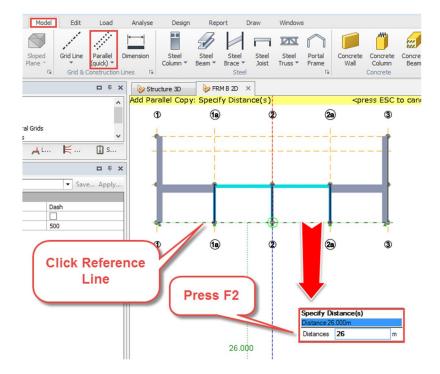
- · Gridlines are placed on the horizontal level.
- Construction lines placed on sloped or vertical views.

Steel Sections to be placed in the model – these can be saved as property sets.

7.1 Construction Lines

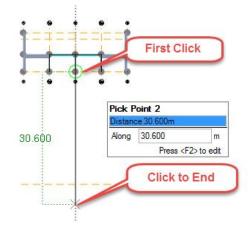
We will be using construction lines to create the faceted curve of the truss.

- Step 1. Select the Frame View of gridline B.
- Step 2. On the Model tab, click the Construction Line command.
- Step 3. From the drop list option, pick Parallel (Quick).
- Step 4. On the frame view of gridline B, click the gridline for the base level to act as a reference line then move the mouse cursor down to indicate the offset direction.
- Step 5. Press the keyboard F2 button to access the distance entry box.
- Step 6. Type an offset distance of 26m and press the Enter key to create that line.
- Step 7. Press Esc to exit the operation.



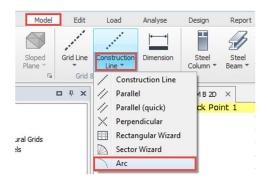
Now we need to create a vertical line to intersect this line.

- Step 8. From the same drop list option, choose Construction Line.
- Step 9. Click on the bottom of the existing middle vertical line to start.
- Step 10. Move the cursor vertically down to pass the new created horizontal line.
- Step 11. Click to create a new vertical line.



Creating an arc to describe the top of the truss.

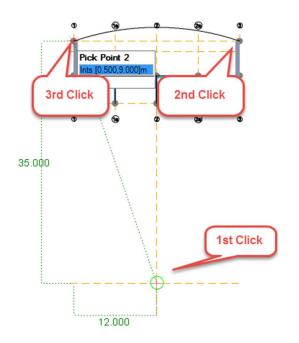
Step 12. On the same drop list option, pick Arc.

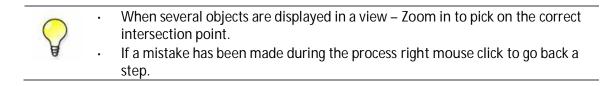


Step 13. To create the arc – click on the lower intersection just created.

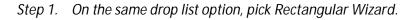
Step 14. Then click on the Right Hand Eaves position to define the beginning of the radius.

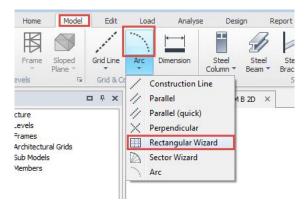
Step 15. Finally click on the Left Hand Eaves position to define the end of the radius.



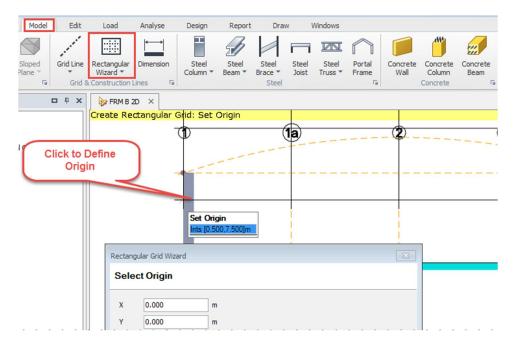


To create a series of vertical lines to define the truss bays, we use the rectangular wizard in the same frame view.





Step 2. When picking the Origin position pick the bottom left of truss bottom level.



Step 3. Generate Y Direction Lines Only to construct vertical lines at a distance of 4m.

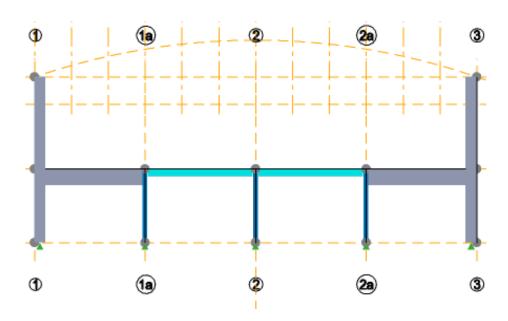
○ X Direction Lines Only			Length	24.000	m
Y Directi	on Lines Only		Length	4	m
All Lines					
Line Style	DashDot		¥		

Step 4. Press Next to continue.



Step 5. In the X direction extents, generate 11 Count of vertical lines at 2m intervals.

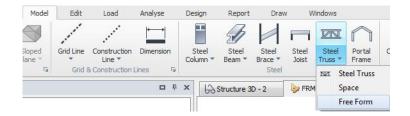
- Step 6. Press Next.
- Step 7. Accept the defaults for Rotation Angle and Axis Angle.
- Step 8. Press Finish to create the vertical lines.



The arrangement of the construction lines to define the truss is now complete.

7.2 Steel Truss

- 7.2.1 Free Form truss properties
 - Step 1. On the Model tab, click Steel Truss command.
 - Step 2. From the drop list option, pick Free Form.



Step 3. In the Properties window, establish the following sections and save individual Property Set:

- Characteristics Truss member top
- Grade \$355
- Section 150x150x8.0
- Design section order SHS Hollow
- Property set name Truss Top Boom

Properties	<u>-</u>	0 ⁶ ×		
<ursaved set=""></ursaved>	-	Save opply		
General	-			
Truss	-New truss-			
Element Parameters				
Characteristic	Truss member top			
Element type	Beam	Beam		
Material type	Steel			
Construction	Steel truss member	100		
Fabrication	Rolled			
Linearity	Straight.			
Use Automatic Grouping	12			
Rotation	0.0000*			
Rotation angle	6.00007			
Grade	\$355			
Autodesign	83			
Design section order	SHS Hollow			
Section	8H8 150x150.8.0			
Major alignment	0			
Major alignment offset	Save State			
Minor alignment				
Minor alignment offset	Specify the the name of th	e state to save: Truss Top Boom OK		
Releases				
Deflection limits		Cancel		
3 Size constraints	1			

- Characteristics Truss member bottom
- Grade S355
 - Section 150x150x8.0
- Design section order SHS Hollow
- Property set name Truss Bottom Boom

Properties			n f x		
<ursailed set=""></ursailed>		-	Seve		
General				ril	
Trues	1	New Blues-			
Element Parameters	-		-		
Overacteristic		Truss member bottom			
Element type		10.00			
Material type		Steel			
Canalituction		Steel truss member bottom			
Pabrication		Roled			
Linearity		Stript			
Use Automatic Grouping		2			
Ratation		0.0000+			
Rotation angle		0.0000*			
Grade	5	395			
Autodesign	18	1			
Design section order		PE Holow			
Sector		H8 190x190x8 0			
Major alignment					
Major algoment offset	Specify the the name of the state to save:				
Mnor algrment			and the second second second second	-	
Hinor alignment offset			Trues Bottom Boom	OK	
Releases Deflection limits		Cancel			
					Carce
Size constraints	5				

.

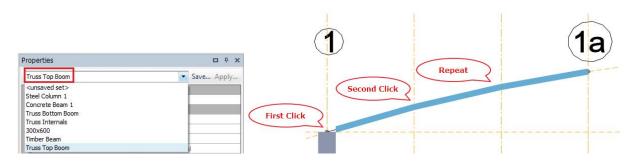
- Characteristics Truss Internal
- Grade S355
- Section 100x100x4.0
- Design section order SHS Hollow
- Property set name Truss Internal

roperties		e + ×	c		
<unsaved set=""></unsaved>		· Save_ apply			
General		<u> </u>	1		
Truss	-New truss		1		
Element Parameters					
Characteristic	Trues internal				
Element type	(Dearn				
Material type	Steel				
Construction	Steel truss intern	al			
Fabrication	Rolled				
Linearity.	Straight				
Use Automatic Grouping	121				
Rotation	0.0000*				
Rotation angle	in consider				
Grade	\$355				
Autodesign	12				
Design section order	SHS Hollow				
Section	8H8 100x100x4.0				
Connection					
Major alignment	Save State				
Major alignment offset	the second of			-	
Mnor alignment	Specify the the name of	the state to saves	Truss Internal	OK	
Mnor alignment offset				Count	
E Releases				Cancel	
Compression	1				

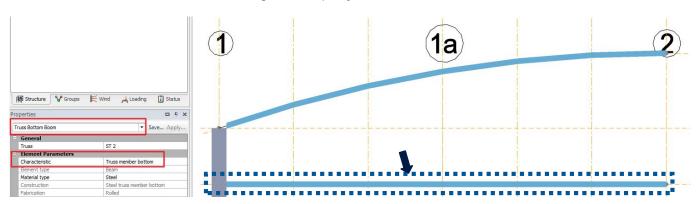
7.2.2 Modelling booms

Place individual faceted members to create the top boom of the truss.

- Step 1. Ensure that the saved property set Truss Top Boom is selected from Properties window drop list.
- Step 2. Click on individual intersections to pick point to insert truss element.
- Step 3. Repeat the placing of the truss elements until the centreline (at gridline 2).

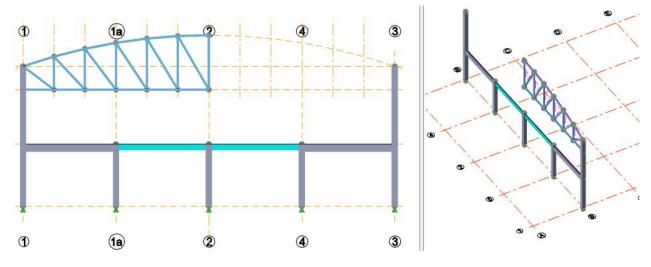


Step 4. Create a single truss element to the centreline (at gridline 2) for the bottom boom – remember to change the Property Set to Truss Bottom Boom.



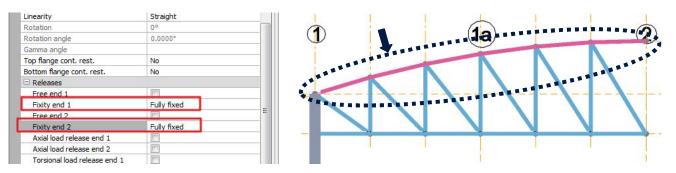
7.2.3 Modelling internals

Step 5. Set to use the Truss Internal property set in the Properties window grid and place the following truss elements up to gridline 2.



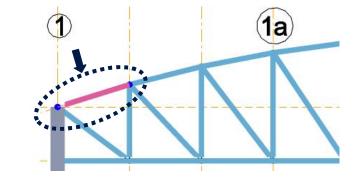
7.2.4 Member Releases

Step 1. Select all the elements of the top boom and ensure the releases of both ends are Fully Fixed.



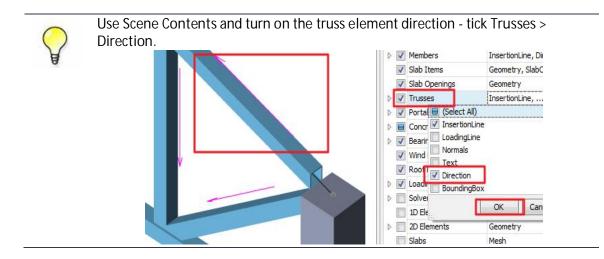
Step 2. Now select the first (individual) member that is connected to the column and set releases at Fixity end 1 to Pin.

Kolauon	U-	
Rotation angle	0.0000°	
Gamma angle	0.0000°	
Top flange cont. rest.	No	
Bottom flange cont. rest.	No	
🗆 Releases		1
Free end 1	101	
Fixity end 1	Pin	
Free end 2	100 to	
Fixity end 2	Fully fixed	
Axial load release end 1	10 ⁻¹⁰	
Axial load release end 2		
Torsional load release end 1		
Torsional load release end 2		

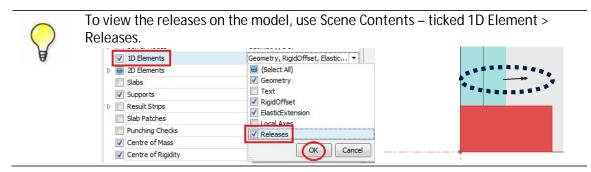


 \mathbf{Q}

End 1 – start of the member / End 2 – end of the member.



Step 3. Repeat the same process for the bottom boom to set it fixed at its end along gridline 2 and pinned at the end at the column.

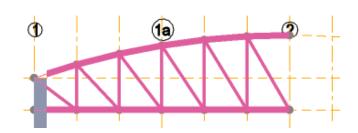


8 Mirror Command

We will use the Mirror command to establish the remaining elements of the truss.

Step 1. In the frame view of gridline B, select all elements of the truss.

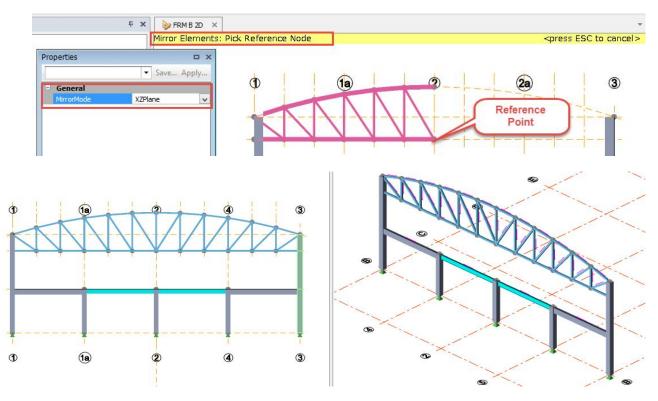
The Select operation will give you the option to pick individual members of the truss or the whole truss.



Step 2. Under the Edit tab, pick Mirror command.

File	Н	ome	Model	Edit	Lo	ad A	nalyse	Design
Þ	Y	52	×	₫È		↔		X
Сору	Move	Mirror	Delete	Join	Split	Reverse	Beam Lines	Cutting Planes

Step 3. Ensure that the Mirror Mode is in the "XZPlane" and you pick the reference point at gridline B bottom boom level.



Step 4. Place in the final Internal truss member and Save the Project.

76 (257)

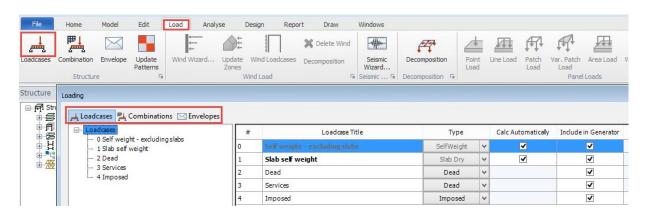
9 Loading

The following loading will be placed on the frame considering centres at 6 meters (unfactored).

- Frame Self Weight Automatically Calculated
- Dead (line load on first floor beams) considering 200 thick slab 0.2 x 25 x 6 = 30 kN/m
- Floor Imposed (line load on first floor beams) 3.5 kN/m^2 -3.5 x 6 = 21 kN/m
- Roof Imposed (point loads on top boom of truss) $0.6 \text{ kN/m}^2 0.6 \text{ x} 6 \text{ x} 2 = 7.2 \text{ kN}$

9.1 Loadcases

Step 1. Obtain a Frame/Elevation view of the structure. Step 2. On the Load tab, click the Loadcases command.



Step 3. Turn off the option of Calculate Automatically for the "Slab self weight" loadcase.

#	Loadcase Title	Туре		Calc Automatically	Include in Generator	Imposed Load Reductions	Pattern Load
0	Self weight - excluding slabs	SelfWeight	~	~	~		
	Slab self weight	Slab Dry	¥		✓		
2	Dead	Dead	~				
3	Services	Dead	Dead 🗸		Un-check this Option		otion
4	Imposed	Imposed	V				and a second

Step 4. Delete the "Services" loadcase by highlighting the line and pressing the Delete button.

#	Loadcase Title	Туре		Calc Automatically	Include in Generator	Imposed Load Reductions	Pattern Load	Cancel
D	Self weight - excluding slabs	SelfWeight	~	•	•			
1	Slab self weight	Slab Dry	~	~	~			1
2	Dead	Dead	Y		v			Add
3	Services	Dead	~		✓			Сору
1	Imposed	Imposed	~		~			Delete

Step 5. Change the name and type of the Imposed to Roof Imposed.Step 6. Add a "Floor Imposed" loadcase

0	Self weight - excluding slabs	SelfWeight	~	✓
1	Slab self weight	Slab Dry	~	
2	Dead	Dead	~	
4	Roof Imposed	Roof Imposed	~	
5	Floor Imposed	Imposed	~	

Step 7. Press OK to exit this dialogue



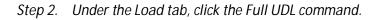
In this dialogue, unlimited number of loadcases can be created and automatically merged into combinations for a specific design head code.

9.2 Applying Member Loads

All loading is applied graphically by selecting the individual loadcase and then applying the load.

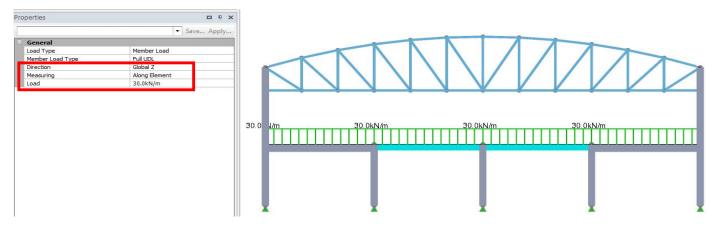
Step 1. Select the Slab self weight loadcase from the drop list option.

		0 Self weight - excluding slabs	-	_	-
		1 Slab self weight			
		 2 Dead			
		4 Imposed			
Show Process	10.00	1 Slab self weight	_		1



E	Load Analy	se D	Design Repo	rt Draw	Windows												
	E.	S E		💥 Delete Wind		Æ	4	I	₩.	F.	₽₽₽	A	ţţţţ	rffr	<u></u>	1	<u>+</u> +
	Wind Wizard	Update Zones	Wind Loadcases	Decomposition	Seismic Wizard	Decomposition	Point Load	Line Load	Patch Load	Var. Patch Load	Area Load	Var. Area Load	Level Load	Slab Load	Full UDL	UDL	VDI
2		N	/ind Load	5	Seismic 🛱	Decomposition 🖼				Panel	Loads			5			

Step 3. Enter the load value 30 kN/m in the Properties window and set the direction to Global Z. Step 4. Graphically left click on the beam in the frame view to apply the load.



Step 5. Use the Scene Contents to display the loading text on the screen.

D 📄	Loading	Geometry,	Text	-
> 🔲	Solver 🗹 (Select All)			
	1D Ele 🗹 Geometry			
Þ 🔲	2D Ele Text			
	Slabs	ОК	Canc	el
V	SUDDOLES	GEUITIEU Y		

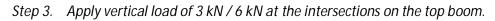
Step 6. Set to Floor Imposed loadcase and apply Full UDL of 21 kN/m on the same beams.

9.3 Applying Nodal Load

On the top boom of the truss, we will place individual nodal loads at the intersection of internals.

- Step 1. Set to Dead loadcase.
- Step 2. Under the Load tab, pick the Nodal Load command.

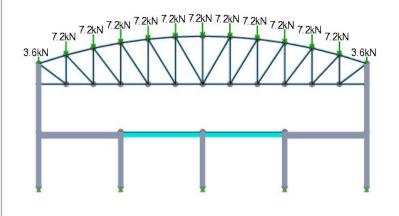
Load Analy	se l	Design Re	port	Draw	Window	VS							
Wind Wizard	Update	Wind Loadcase	X Decor	П	₽	<u>11</u>		<u>+</u>	÷	Į.	₽ <mark>₽</mark>	K	Nodal Load
	Zones	Vind Load		Full UDL	UDL	VDL	Trapezoidal Load	Point Load	Moment Load	Torsion Full UDL	Torsion UDL	Torsion VDL	Settlement Load
							Mem	per Loads	100000			5	Structure Loads



perties	□ 早 × Save Apply	6.0kN
General		
Load Type	Nodal Load	
🖃 Load	[0.0, 0.0, 3.0] kN	
х	0.0kN	
Y	0.0	
Z	3.0kN	
🗆 Moment	[0.0, 0.0, 0.0] kNm	
х	0.0kNm	
Y	0.0kNm	
Z	0.0kNm	

- Step 4. Set to Roof Imposed loadcase.
- Step 5. Apply vertical load of 3.6 kN / 7.2 kN at the same intersections on the top boom.

operties	D P
	▼ Save Apply
General	
Load Type	Nodal Load
🗆 Load	[0.0, 0.0, 3.6] kN
X	0.0kN
Y	0.0kN
Z	3.6kN
🗆 Moment	[0.0, 0.0, 0.0] kNm
X	0.0kNm
Y	0.0kNm
Z	0.0kNm



80 (257)

10 Load Combinations

Tekla Structural Designer has an inbuilt wizard which will produce code based combinations.

- Step 1. Under the Load tab, click the Combination command.
- Step 2. Click on the Generate... button in the combination dialogue.
- Step 3. Pick the option of Table A1.2(B) Eq 6.10a&b.
- Step 4. Press Finish and OK to exit the combinations dialogue.

File	Home	Model	Edit	Load An	alyse	Design	Report	Draw	Windows				
Loadcases			Update Patterns	Wind Wizard.	Zones		1 1	Composition	Seismic Wizard	Decomposition	Point Load	Line Load	Patch Load
Structu	Structure	s dons Comb	ination Generation Generatio Generation Generation Generation Generation Generation Gene	# [merator ameters .2(B) - Eq 6.10 .2(B) - Eq 6.10a binations - Table al combinations - Table	e A1.2(C) - Table A2.	Eq 6.10 5 - Eq 6.11a	R) 18b	Class		re Strength	Service	Ca A C	OK OK
Loa												Gene	erate

This will generate two gravity combinations to the model – they can be inspected and edited by selecting the individual combination in the list on the left.

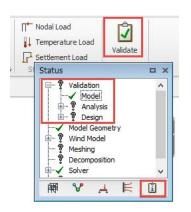
Loadcases			The second				OK
Combinations 1 STR1-1.35G+1.5\v_0Q+1	Name STR1-1.35G+1	5ψ0Q+1.5ψ0RQ					Cancel
2 STR ₃ -1.35§G+1.5Q+1.5	Parameters Seco	ond order effects					
	Active						Add
	Pattern						Сору
	Strength						
	Service						Delete
	Available Loadcases	>>	<<			Included	
	Loadcase Title	Туре	Loadcase Title	Strength	Service	SLS Quasi	
	EHForte		0 Self weight - excluding sla	1.350	1.000	1.000	
	EHF _{Dirt}		1 Slab self weight	1.350	1.000	1.000	
	FUE		2 Dead	1.350	1.000	1.000	
	EHF _{Dir2+}			1.050	1.000	0.300	
	EHF _{DI/2+}		5 Floor Imposed	1.050	Landa de la composition de la		
			5 Floor Imposed 4 Roof Imposed	1.050	1.000	0.000	

82 (257)

11 Validation

Before running any analysis or design it is recommended to validate the model – TSD will check for any modelling errors that can cause problems with the model.

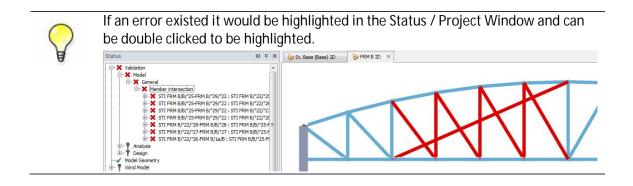
- Step 1. Click the Validate command on either Model or Load tab.
- Step 2. Go to the Project Workspace and click on Status tab to check if there is any issue reported on the model.



Validation has the following status:

§ §

- § 🖌 Status OK, no issues found
 - 🔋 Status unknown / Not yet checked
 - FYI (For your information) issues to consider but will not prevent analysis/design
- § X Errors that must be resolved



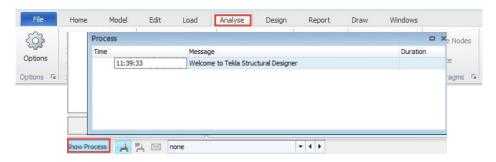
84 (257)

12 Analysis and Viewing Graphical Results

Even after performing a validation check there still maybe issues with your model in analysis such as large deflections, degrees of freedom, buckling etc.

12.1 Running 1st Order Linear Analysis

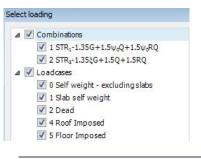
Step 1. Select the Analyse tab and click the Show Process button (at left bottom corner of the screen).



Step 2. In the Analysis tab, click the 1st Order Linear command.

File	Home	Model	Edit	Load	Analyse	Design	Report	Draw	Wind	lows		
	1st Order I	.inear	2nd Orde	er Linear	FE chase	-down		Mech	Slabs		Toggle Node	
Options	1st Order I	Non-linear	2nd Orde	er Non-linear			Tabular Data	3				
Options	1st Order	/ibration	2nd Orde	er Buckling	Grillage d	hase-down		Upda	ite Wall Bea	ams	Update	
Options Ta	1st Order /	Analysis G	2nd Orde	er Analysis	Chase-	-down G	Solver	154 1	leshing	G	Diaphragms	G

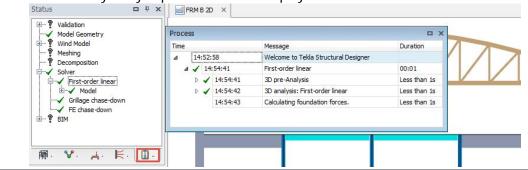
Step 3. Select all loadcases and combination to be analysed.



.

Take note of the Process window which will reveal any potential problems with the model.

When the analysis is complete the Project Workspace will switch to the Status tab – any analysis problems will be displayed.



12.2 Loading Summary Table

Do the applied loads equal the reactions? TSD automatically does this mathematical check for you.

The Loading tab within the Project Workspace is used to organise the loadcases and combinations into a hierarchical structure. It also provides a summary of each loadcase that can be used to cross check against the load applied.

- Step 1. Select the Loading tab in the Project Workspace.
- Step 2. Highlight a loadcase or combination to examine the Total Load on Structure vs Total Reactions.

I Loading I Loadcases ✓ 0 Self weight - exclu ✓ 1 Slab self weight I Slab se	<u>Q+1.5w,RQ</u> Q+1.5RQ
	▲ Loading
perties	
perties ombination(s): 1 items General	■ ¥ Save Apply.
ombination(s): 1 items	
ombination(s): 1 items	▼ Save… Apply.
ombination(s): 1 items General Name	▼ Save Apply. 1 STR_{1}-1.35G+1.5\psi_{0}Q+1.
ombination(s): 1 items General Name User name	▼ Save Apply. 1 STR_{1}-1.35G+1.5\psi_{0}Q+1 STR_{1}-1.35G+1.5\psi_{0}Q+1
ombination(s): 1 items General Name User name Member Loads	 ▼ Save Apply. 1 STR_{1}-1.35G+1.5\psi_{0}Q+1 STR_{1}-1.35G+1.5\psi_{0}Q+1 [0.0, 0.0, 1758, 1] kN
ombination(s): 1 items General Name User name Member Loads Nodal Loads	 ▼ Save Apply. 1 STR_{1}-1.35G+1.5\psi_{0}Q+1 STR_{1}-1.35G+1.5\psi_{0}Q+1 [0.0, 0.0, 1758, 1] kN [0.0, 0.0, 187.9] kN
ombination(s): 1 items General Name User name Member Loads Nodal Loads Total NHF Dir 1	 ▼ Save Apply. 1 STR_{1}-1.35G+1.5\psi_{0}Q+1 STR_{1}-1.35G+1.5\psi_{0}Q+1 [0.0, 0.0, 1758.1] kN [0.0, 0.0, 187.9] kN [0.0, 0.0, 0.0] kN
ombination(s): 1 items General Name User name Member Loads Nodal Loads Total NHF Dir 1 Total NHF Dir 2	 ▼ Save Apply. 1 STR_{1}-1.35G+1.5\psi_{0}Q+1. STR_{1}-1.35G+1.5\psi_{0}Q+1 [0.0, 0.0, 1758, 1] kN [0.0, 0.0, 187.9] kN [0.0, 0.0, 0.0] kN [0.0, 0.0, 0.0] kN
ombination(s): 1 items General Name User name Member Loads Nodal Loads Total NHF Dir1 Total NHF Dir2 Decomposable Loads	 ▼ Save Apply. 1 STR_{1}-1.35G+1.5\psi_{0}Q+1. STR_{1}-1.35G+1.5\psi_{0}Q+1 [0.0, 0.0, 0.758.1] kN [0.0, 0.0, 0.87.9] kN [0.0, 0.0, 0.0] kN [0.0, 0.0, 0.0] kN [0.0, 0.0, 0.0] kN
ombination(s): 1 items General Name User name Member Loads Nodal Loads Total NHF Dir 1 Total NHF Dir 2 Decomposable Loads 1 Way Decomp Results	 ▼ Save Apply. 1 STR_{1}-1.35G+1.5\psi_{0}Q+1. STR_{1}-1.35G+1.5\psi_{0}Q+1 [0.0, 0.0, 1758.1] kN [0.0, 0.0, 1758.1] kN [0.0, 0.0, 0.0] kN
ombination(s): 1 items General Name User name Member Loads Nodal Loads Total NHF Dir 1 Total NHF Dir 2 Decomposable Loads 1 Way Decomp Results 2 Way Decomp Results	 ▼ Save Apply. 1 STR_{1}-1.35G+1.5\psi_{0}Q+1. STR_{1}-1.35G+1.5\psi_{0}Q+1 [0.0, 0.0, 1758.1] kN [0.0, 0.0, 187.9] kN [0.0, 0.0, 0.0] kN



If there is discrepancy between total load on structure and total reaction, you need to investigate and check on the model for any incorrect input or modelling mistake.

12.3 View Analysis Results

Results for the model are available both graphically and numerically.

Step 1. In the current view, select the Analysis Results View found at the bottom of the screen. Activating this view will open the ribbon tab for Results.

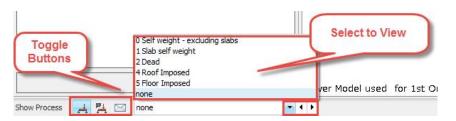


File Home Model Edit Load Analyse Design 1st Order Linear ▼ Strength Factors 1D Forces * Fx Mx Fx/2	Report Draw Windows Res Axial Force Torsion X Dir1	xults X Y EHF	Fx Fxz My Mdx bottom	
Nor 2D Forces Fy My Mxyz Red Axial Force Deflections Fz Mz Total	Shear Major Moment Major Y Dir2 Shear Minor Moment Minor Z Total	X Relative Y Relative Dir1 Shear Dir2 Shear Seismic	Fy Fyz Mxy Mdy top Fxy Mx Mdx top Mdy bottom	x bottom
Analysis Type for Viewing Results	FRMB 20 X	Sway and Storey Shear 🕼 Notion 🕼	20 Results 5	AsReq
Properties Y Save Apply				



- § 1D Results represents single wire elements (such as beams and columns), while 2D Results represents area elements which are finite element results (such for slabs and walls).
- § Multiple results can be displayed on the screen at any one time, example support reactions along with axial forces and deflections.
- § Please remember to pick the appropriate loadcase or combination.

Select a loadcase or combination of the result you would like to view using the toggle buttons and drop down list.



Create the following graphical results display: Z deflections, axial, vertical support reactions for the combination 1 STR₁-1.35G+1.5Q+1.5RQ.

- Step 2. Under Result Type select 1st Order Linear.
- Step 3. Under Deflections click on Z.
- Step 4. Under 1D Results click on Axial Force.
- Step 5. Under Support Reactions click on Fz.

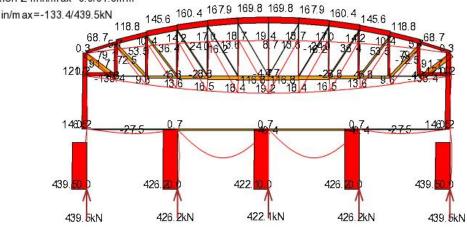
File Hor	ne Model	Edit Load	d Analyse	Design	Report	Draw Wi	ndows R	esults	
1st Order Linear 🔻	Strength Factors	1D Forces	Fx	Mx Fxyz	Axial Force	Torsion	X Dir1	x	Y
None -		2D Forces	Fy	My Mxy	Shear Major	Moment Major	Y Dir2	X Relative	Y Relative
Reduce Axial Force	Service Factors	Deflections		Mz Tota	Shear Minor	Moment Minor	Z Total	Dir 1 Shear	Dir 2 Shear
Result	Type G	Scale Setti	ings 🖬 Supp	port Reacti	5 1D R	lesults 🛛 🖓	Deflections !	Sway and Sto	orey Shear ଢ

Step 6. Set the combination to $1 STR_1$ -1.35G+1.5Q+1.5RQ from the drop down list option.

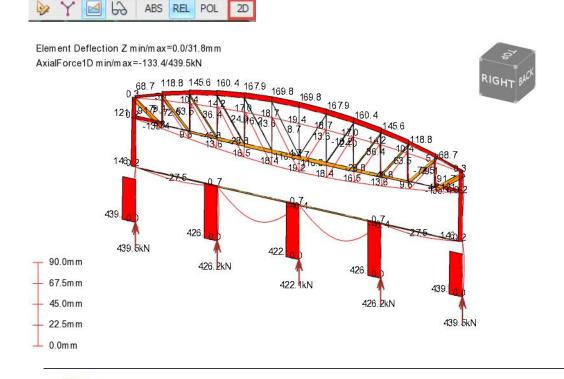
_	-				
,	\boxtimes	1 STR1-1.35G+1.5Q+1.5RQ	•	4	+

The following results is displayed:

Element Deflection Z min/max=0.0/31.8mm AxialForce1D min/max=-133.4/439.5kN



Step 7. Toggle the 2D button to view the fame in 3D.



The forces diagram and deflection diagram can be animated or scaled using the sliders.



When viewing deflections remember to select the appropriate Strength / Service Factor.

1st Order Linear 🔻	Strength Factors	1D Forces	9
None -		2D Forces	
Reduce Axial Force	Service Factors	Deflections	$\neg \bigcirc \neg$
Result T	ype 😼	Scale Set	tings



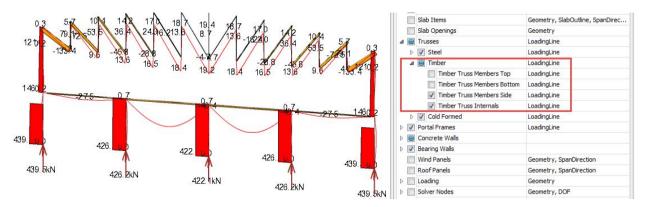
The steel beams were initially set to Autodesign, TSD will assign an initial size for the purpose of analysis (notice the large deflections). Home tab > Settings > Section Defaults.

We will design an appropriate size later in the exercise.

12.4 Filtering Results Graphically

You can further filter/simplify the current view by using the scene contents to turn off/on elements.

Step 1. Open the Scene Contents and turn off the top and bottom booms display.



Step 2. Turn them back on when finished.

12.5 Tabulated Results

Numerical input and results data can be obtained in tabulated form.

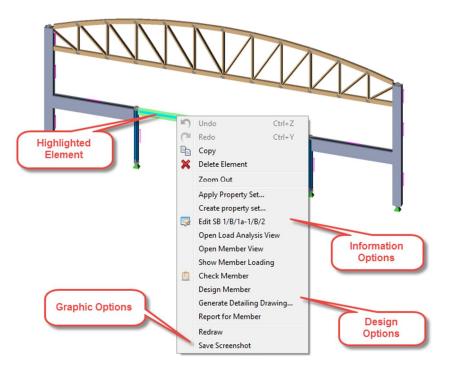
- Step 1. Select the Analysis tab and then Tabular Data.
- Step 2. Then select a number of inputs and results data tables from the drop down list on the ribbon.

Model Edit Load	Analyse D	esign Report	Draw	1	Vindow	s			
Linear 2nd Order Linear Non-linear 2nd Order Non-linear Vibration 2nd Order Budding Analysis 5a	FE chase-down Grillage chase-d Chase-down	Illage chase-down Tabular Data Update Wall Beams None Chase-down Ta Solver Ta Meshing Ta Result Type Tabular Data		r▼ Nodes ▼ S Nodes ✓ Elements					
□ # x	FRM B 2D	Solver Model	Data	×					◎ _# Nodal Forces
	Node Number	Coordinates [m]	F _x DOF	F _y DOF	F <u>.</u> DOF	M _x DOF	M, DOF	M ₂ DOF	 ◇ Nodal Deflections ✓ Element End Forces Wall Lines
	1	[6.000,0.000,4.000]	Free	Free	Free	Free	Free	Free	
	2	[6.000,6.000,4.000]	Free	Free	Free	Free	Free	Free	
	3	[6.000,12.000,4.000]	Free	Free	Free	Free	Free	Free	
	4	[6.000, 18.000, 4.000]	Free	Free	Free	Free	Free	Free	
	5	[6.000,24.000,4.000]	Free	Free	Free	Free	Free	Free	

13 Right Click Context Menu

In any view, hovering the mouse pointer over any element will shade the outline of that element.

Right click on the mouse, hover over an element to produce more options for modelling / information / design.



Step 1. Edit Member – Same options as the Properties window, individual modification of the element but in a pop up dialogue.

4	:neral	3 -			
General	Characteristic	Beam	~		
Size Alignment	Element type	Beam	~		
Releases	Material type	Steel	~		
Lateral restraints					
Strut restraints	Construction type	Non-composite bear	×		
Haunches	Fabrication	Rolled			
End plates	Fabrication	Rolled	Y		
Deflection limits	Automatic design	Section order	UKB Beam	~	Gravity onl
Size constraints					,
Camber					
Natural frequency					
Instability factor					
Seismic					

Step 2. Open Load Analysis View – Choose this option to see detailed individual analysis results from the altered ribbon options.

Remember to pick the appropriate loadcase/combination.

File Home Model Edit Load	Analyse Design Report Draw Windows Loading Analysis
Refresh Loading Strength Factors Axial Next Strength Factors Minor Prev Data 52 Result Type 53 Direction 53	
Structure 🗖 🕈 🗙	😼 FRM B 2D 🔠 SB 1/B/1a-1/B/2 loading 🗡 👻
□····································	We FRM B 2D
Properties ••• Arr F. •• ••• ••• ••• ••• ••• ••• ••• ••• •	
General A Distance 1221.4mm Span Whole beam Shear left 123.8kN	Graphically slide the bar to investigate positions along the
Shear right 123.8kN Moment left 203.2kNm Moment right 203.2kNm	element
Relative deflection 18.8mm Applied load left 51.5kN/m Applied load right 51.5kN/m	
Applied force 0.0kN Applied moment 0.0kNm Show loading Show shear	313.3 kNm
Show moment Show relative deflection Show dimensions	
Show extremes V Y	31.1 mm
Show Process 🛃 🔀 2 STR ₃ -1.35ξG+1.5Q+1.	RQ • • • •



You can also adjust the Distance property by hovering the cursor over the vertical red line in the loading analysis view. When the cursor changes to the hold the left mouse button down and drag the cursor to a new location. The Distance property is adjusted and co-existent values are displayed for the distance set.

14 The Solver View

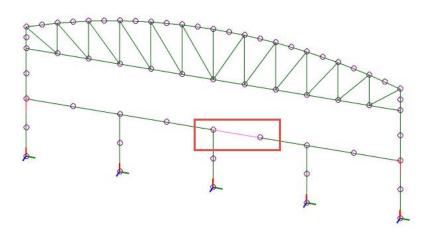
The underlying creation of the solver model can be viewed and settings for support fixities, member releases etc. can be interrogated / edited.

Step 1. To open the Solver View click the toggle button in the bottom right of the window.



Step 2. To see the properties of any element – left click and look to the Properties window.

So	olver Element(s): 1 items	▼ Save Apply					
-1	General						
	Active	True					
	Туре	Beam					
	Fabrication	Rolled					
	Construction	Non-composite beam					
	Material	S355					
	Gamma angle	0.0000*					
	Assume cracked	No					
	Length	3000.0mm					
	Start releases						
	Fx	Fixed					
	Fy	Fixed Fixed					
	Fz						
	Mx	Fixed					
	My	Released					
	Mz	Released					
	End releases						
	Fx	Fixed					
	Fy	Fixed					
	Fz	Fixed					
	Mx	Fixed					
	Му	Fixed					
	Mz	Fixed					

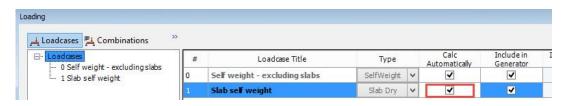


15 Creating a More Complex 3D Structure

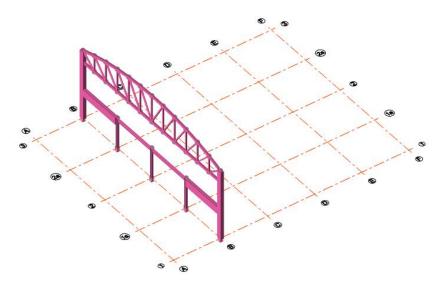
15.1 Copy Command

We will use the frame to produce the rest of the 3D Structure.

- Step 1. Go to Load tab, click on Loadcases command.
- Step 2. Delete all the existing loadcases (as shown below) and tick the Calc Automatically option for "Slab self weight" loadcase.



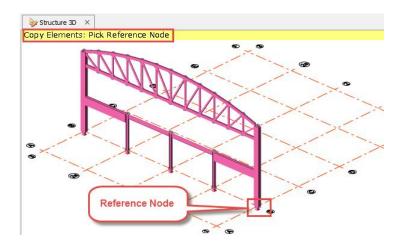
Step 3. In Structure 3D view and turn on the base Grid Lines display from Scene Contents.
Step 4. Select all the elements of the Frame – **Do not select any gridlines.



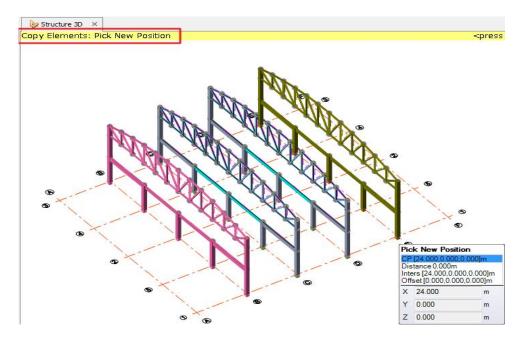
Step 5. Go the Edit tab and click the Copy command.



Step 6. Follow the instructions in the yellow information bar – pick the Reference Node at the location as shown in below.



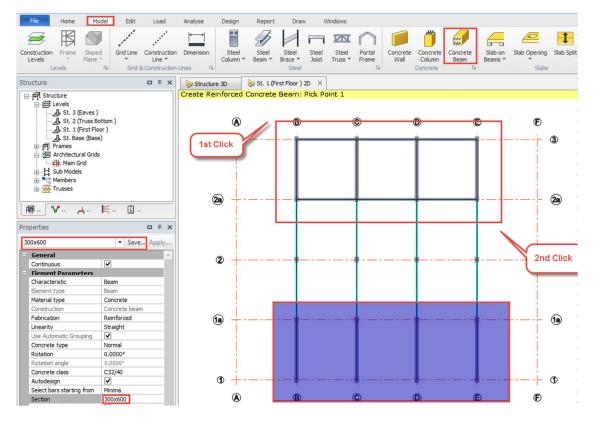
Step 7. Pick the new positions of 1/C, D and E.



Step 8. Press Esc twice to exit the operation and clear the selection.

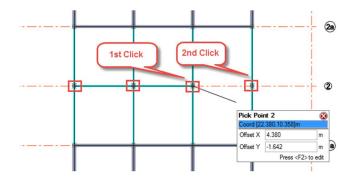
15.2 Placing Concrete Beams

- Step 1. Open St. 1 (First Floor) 2D plan view.
- Step 2. Go to Model tab and click the Concrete Beam command.
- Step 3. Edit the section to be Rectangular 300 x 600 then save as a property set.
- Step 4. Click and drag two boxed to connect the frames.



15.3 Placing Steel Beams

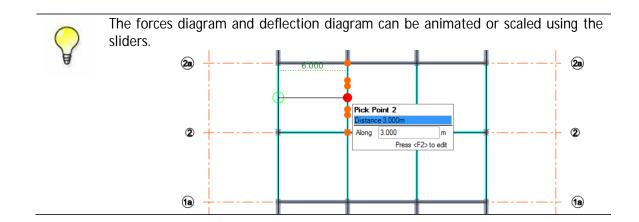
- Step 1. In the current view, click the Steel Beam command
- Step 2. Set to AutoDesign in the Properties window
- Step 3. Individually pick the intersection points to place them along gridline 2.



15.4 Placing Intermediate Steel Beams

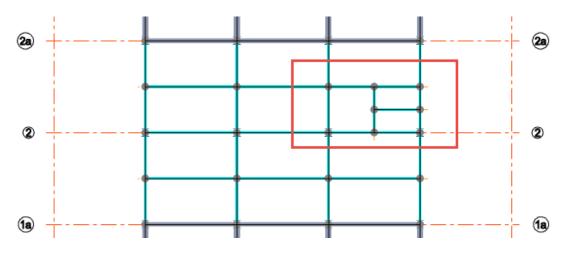
Place intermediate steel beams horizontally between gridlines 1a, 2 and 2a.

Step 4. Still in create beam mode place the cursor over the vertical beam and pick the central node point and pick the similar point on the other corresponding beam.



Step 5. Repeat this process for all steel bays.

Step 6. Add addition steel beams as shown below, all split at a 50% ratio.



15.5 Creating Slabs

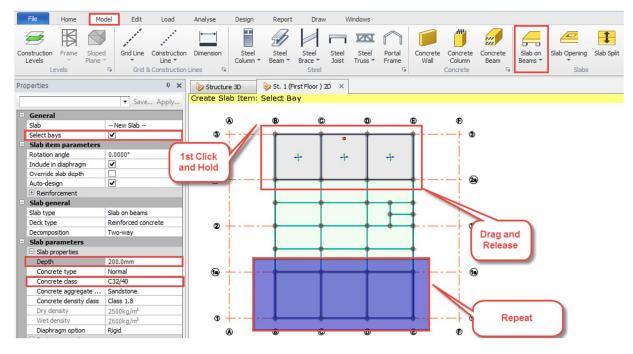
In the concrete area of the floor we will be placing Slabs on Beams and in the steel area placing a Composite Slab.

15.5.1 Concrete slabs

- Step 1. Still in the 2D plan view, click the Slab on Beams command.
- Step 2. In the Properties window, enter slab depth to be 200mm, C32/40, with Two-way Decomposition.
- Step 3. Use the click and drag method to create all slabs in individual bays.



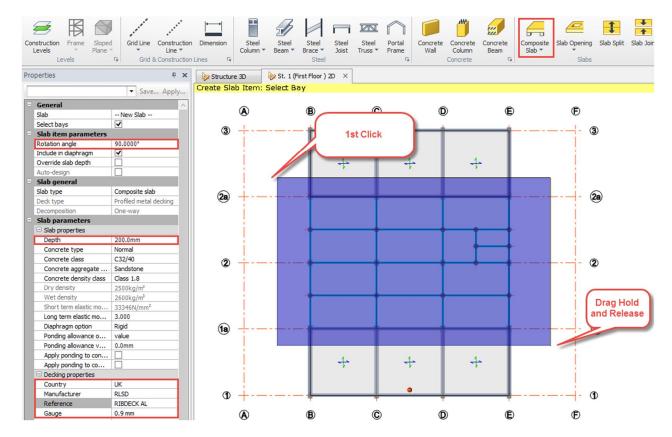
Slabs can be placed by picking the individual points – untick the option Select bays.

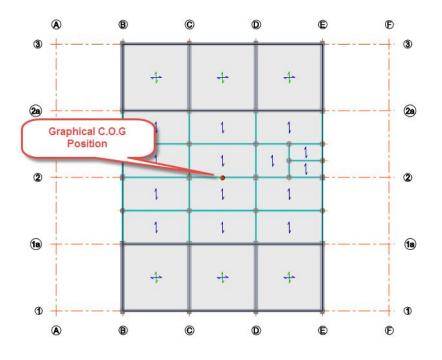


15.5.2 Composite slabs

Step 1. From the drop list, select the Composite Slabs command.

- Step 2. In the Properties window, set to the followings:
 - · Composite Slab One-way Decomposition
 - Depth 200mm, C32/40
 - Rotated 90 degrees
 - · Country of Origin UK, manufacturer RLSD
 - Reference Ribdeck AL @ 0.9 Gauge

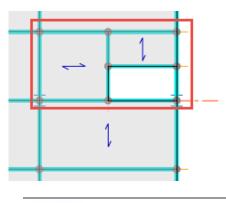




Take note of the graphical C.O.G (Centre of Gravity) position as you build the model.

15.6 Deleting and Rotating Slabs

Step 1. Delete and Rotate the following slabs in the model.



Openings which exist in one way decomposition slab are usually framed with beam elements.

15.7 Openings for Two-way Slab

When the decomposition of the slab is two-way, openings can be placed anywhere within the area.

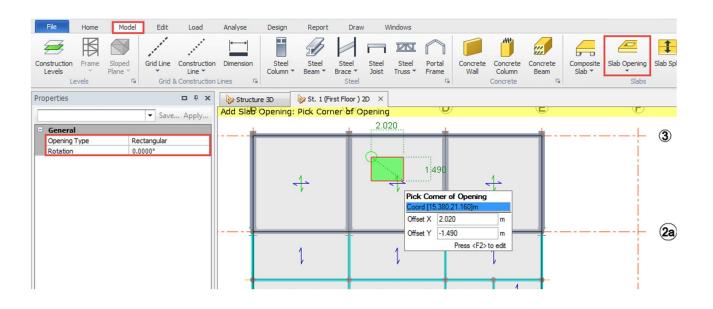
- Step 1. In the Model tab, click the Slab Opening command.
- Step 2. Within a two-way slab, single click to pick 1st corner of opening, move the cursor and single click to pick the diagonal corner of opening.



The F2 button can be pressed to define offsets and dimensions of the opening.



In the Properties window you have options for shape (rectangular and circular) and any rotation.



102 (257)

16 Floor Panel Loading

- Step 1. Go to Loads tab and click the Loadcases command.
- Step 2. In the Loadcases dialogue, delete the existing Dead, Roof Imposed and Floor Imposed loadcases.
- Step 3. Then create two new loadcases:
 - First Floor Additional Dead Type set as Dead
 - First Floor Imposed Type set as Imposed

Step 4. Ensure for Slab self weight the option Calc Automatically is ticked.

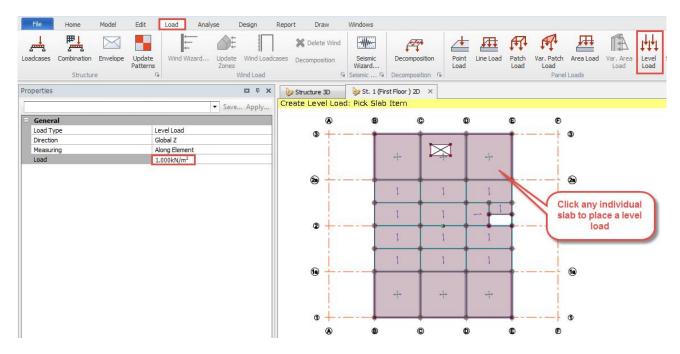
ading										
Loadcases Combinations - Loadcases - 0 Self weight - excluding slabs - 1 Slab self weight - 7 First Floor Additional Dead - 8 First Floor Imposed	# Loadcase Title		Туре		Calc Automatically	Include in Generator	Imposed Load Reductions	Pattern Loa		
	0	Self weight - excluding slabs	SelfWeight	Y	-	-				
	1	Slab self weight	Slab Dry	¥	✓	~				
	7	First Floor Additional Dead	Dead	*		~				
	8	First Floor Imposed	Imposed	×		~				

Step 5. Press OK to confirm the above.

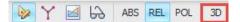
Step 6. In the Loading List option, select the loadcase First Floor Additional Dead.

Show Process 🛄 🔛 🖂	none	-	17	T.	
	8 First Floor Imposed				
	7 First Floor Additional Dead				
	0 Self weight - excluding slabs 1 Slab self weight				

Step 7. Click the Level Load command, set the load value in the Properties window to be 1 kN/m^2 . Step 8. Click on any slab panel to apply this load.



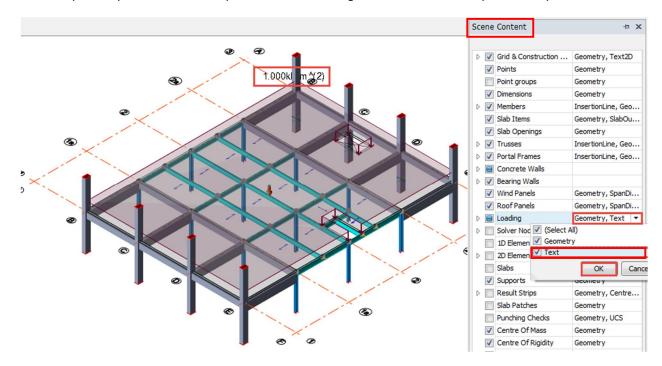
Step 9. Switch the 2D plan view into a 3D view – click on the 3D icon in the bottom right corner.



16.1 Displaying Loading Text

Use the Scene Content to display the magnitude of the loads applied.

- Step 1. Open the Scene Contents window and find the Loading option.
- Step 2. Open the addition options next to loading and select the Text option then press OK.



Step 3. Switch the loadcase to First Floor Imposed and apply 3.5 kN/m² on the floor as a Level Load.

File Home Model Image: Combination Envelope Structure Structure	Edit Load Analyse Design		Windows Seismic Decomposition Wizard 16 Decomposition 16	Point Line Load Pate		Var. Area Load
Properties	• + ×	Structure 3D	≽ St. 1 (First Floor) 2D 🛛 🗙			
	▼ Save Apply	Create Level Loa	d: Pick Slab Item			
General Load Type Direction Measuring Load	Level Load Global Z Along Element 3.500kN/m ²			DM m (12)	slab to	any individual place a level load

17 Creating 3D Roof Structure

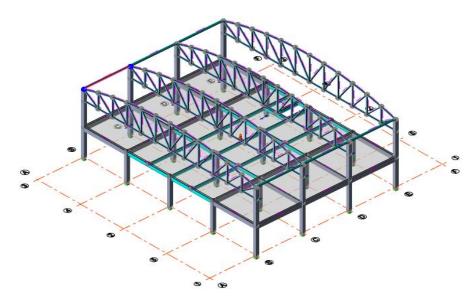
17.1 Creating Eaves Beams

We will now use purlins and eaves beams to connect the remaining frames.

- Step 1. Switch to the Structure 3D View.
- Step 2. Create a single Steel Beam at the eaves level.
- Step 3. Select the steel beam and change its properties to the followings:
 - · Characteristic to Eaves Beam
 - Material type to Cold rolled
 - Section to KingSpan Eaves Beams 265x2.0x21 (UK)
 - Grade to S450GD

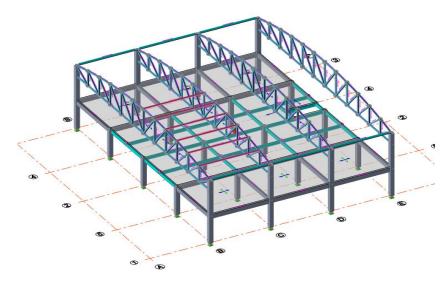
operties 🗖 म 🗙			2 0											
old Rolled Eaves Beams: 1 items	▼ Save Apply					<u></u>								_
General					Select Section									
Name	CREB 3/B/1-3/C/1					E 265>	2.0x21							
User name					UK				100	section	filters			_
Group	Eaves Beams				3.0	-		1	110	section				
Plane	St. 3 (3) : 9.000m				<no filter="" geometry=""></no>	D	t				Pitch	2		
Characteristic	Eaves beam				Albion Eaves Beams	185	1.4	-6	-3	0 3	6	9	12	1
Element type	Beam				Albion Eaves beams			18	21	24 27	7 30			
Material type	Cold rolled					215		0.000	122.04		1 140,000			
Construction	Eaves beam				Hi-Span Eaves Beams		1.7	-6	-3	0 3	6	9	12	200
Fabrication	Cold rolled							18	21	24 27	7 30			
Autodesign					Kingspan Eaves Beams	265	1.5	-6	-3	0 3	6	9	12	
Design section order								18	21	24 27	7 30			
Rotation	0.0000°				Metsec Eaves Beams		-		122.0		1 101/20			_
Rotation angle	0.0000°						2.0	-6	-3	0 3	6	9	12	100
Major snap level	Bottom				Structural Sections Eaves Beams			18	21	24 27	7 30			
Major offset	0.0mm						-		_					_
Minor snap level	Centre			100	Tegral Eaves Beams									
Minor offset	0.0mm													
All spans Steadmans Eaves Beams														
Name	CREB 3/B/1-3/C/1				Steaumans Laves Deams									
User name														
Section	E 265x2.0x21													
Grade	S450GD													_
Linearity	Straight													
Rotation	0°				Select Add Delete De	tails								Ca
Rotation angle	0.0000°												_	-
Gamma angle	0.0000*								-			-		-

- Step 4. Right click on the eaves beam, select Create property set > Span 1 in the context menu.
- Step 5. Save this property set as Eaves Beam.
- *Step 6. Either create and place new eaves beams using the saved property set or use the Copy function.*



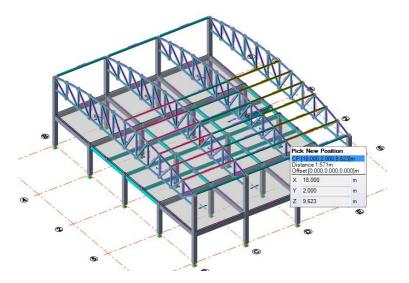
Between the intersections of the curved top boom truss we will place purlins to connect the frame.

- Step 1. In the Structure 3D View.
- Step 2. Create a single Steel Beam at the first top boom truss node position.
- Step 3. Select the steel beam and change to the following properties:
 - Characteristic to Purlin
 - Material type to Cold rolled
 - Section to KingSpan MultiBeam 205x2.0
 - Grade to S450GD
- Step 4. Save this property set as Purlin (using right click context menu option).
- Step 5. Place purlins throughout the structure with a technique of placing place new purlin beam and then using the Copy / Mirror function.



Copy function:

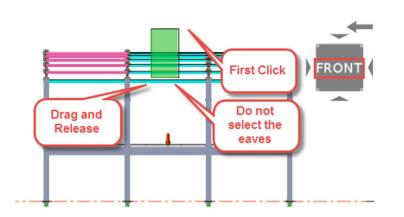
- Step 1. Select all the purlins (using Ctrl key for multiple selection).
- Step 2. Following the instructions given in the information bar first pick on a reference point and then pick on new location to paste the purlins.



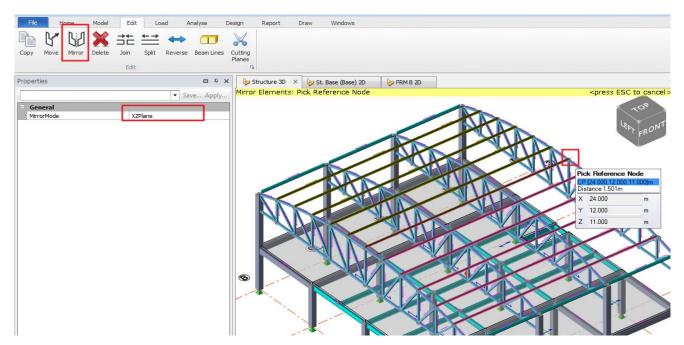
Mirror function:

- Step 1. Pick a Front view from the 3D view cube.
- Step 2. Holding the Ctrl key (adding to the selection) and using select entity by window crossing to select on all purlins.

Drag a window from right to left, having it crosses the purlins – careful not to select the eaves beam



- Step 3. Switch back to 3D perspective view.
- Step 4. Go to Edit tab, click the Mirror command.
- Step 5. In the Properties window, set MirrorMode to XZPlane.
- Step 6. Pick on the central node in the truss as the Reference Node.



Step 7. Press Esc twice to exit the operation and clear the selection.

17.3 Placing Roof Panels

Roof panels are loading decomposition panel which span in a single direction. They offer no additional stiffness to the model and must be supported in the span direction by a beam element.

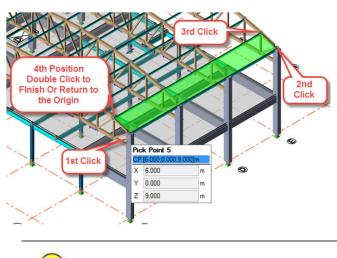
Step 1. In the Model tab pick the Roof Panel command.





Roof or wall panels must be placed in a planer position (no creases in the panel) otherwise load decomposition cannot occur.

Step 2. Click each corner of the panel in sequence to place the panel.

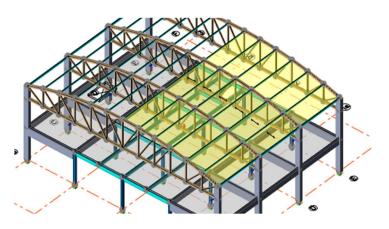


The panels will highlight in Red and Green when the appropriate node position is selected.

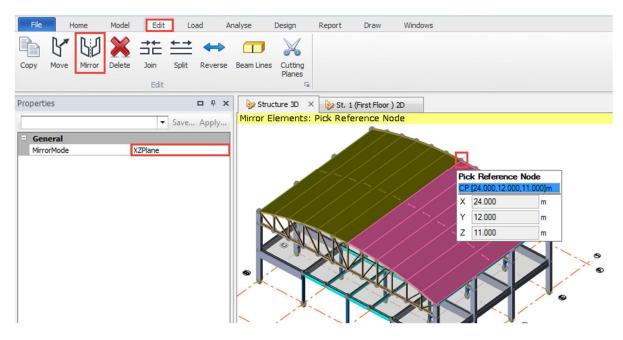
Step 3. The span direction of the panel of the panel needs to be altered – Select the roof panel item. Step 4. In the Properties window set the rotation to 90 degrees.



Step 5. Repeat this process and add half of the roof panels taking account of the rotation.



Step 6. Select the roof panels and use the Mirror command in the XZplane to place the remaining panels. Pick the central top boom node as the Reference Node.



Step 7. Press Esc twice to exit the operation and clear the selection.

17.4 Loading the Roof Structure

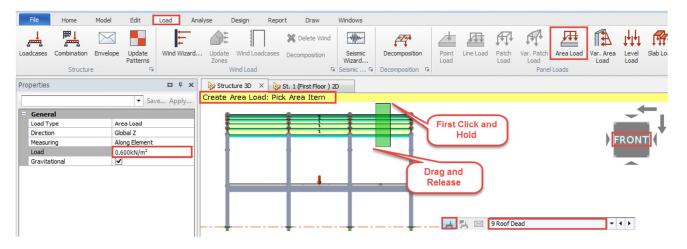
- Step 1. Go to the Load tab, click the Loadcases command.
- Step 2. In the Loadcases dialogue, create another two new cases -
 - Roof Dead Type set as Dead
 - Roof Imposed Type set as Roof Imposed

Loadcases O Self weight - excluding slabs	#	Loadcase Title	Туре		Calc Automatically	Include in Generator	Imposed Load Reductions	Pattern Load
- 1 Slab self weight	0	Self weight - excluding slabs	SelfWeight	Y	•	-		
	1	Slab self weight	Slab Dry	~	•	~		
 8 First Floor Imposed 9 Roof Dead 	7	First Floor Additional Dead	Dead	¥		~		
10 Roof Imposed	8	First Floor Imposed	Imposed	×		~		
	9	Roof Dead	Dead	¥		-		
	10	Roof Imposed	Roof Imposed	v		~		

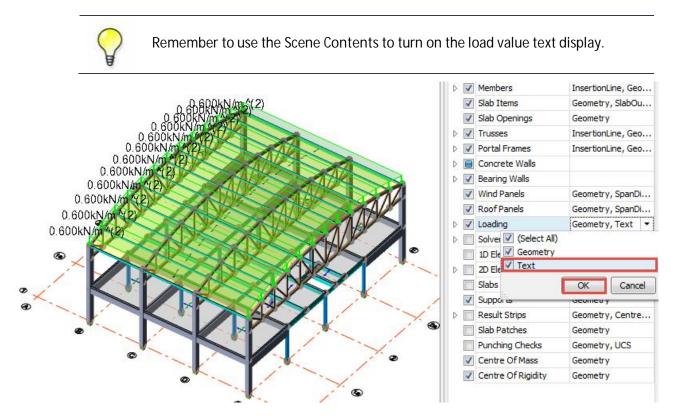
Step 3. Press OK to exit the dialogue.

We will be using Area Loads to place loading on the roof structure.

- Step 4. In the Structure 3D view.
- Step 5. Select the loadcase Roof Dead from the Loading List option.
- Step 6. In the Loads tab, click the Area Loads command.
- Step 7. Adjust the load value in the Properties window to 0.6 kN/m^2 .
- Step 8. Pick a Front view and drag a crossing window (from right to left) across the roof panel items.



Step 9. Repeat the same process to apply Area Load of 0.5 kN/m^2 for Roof Imposed loadcase. Step 10. Check the applied loading in the Structure 3D View.



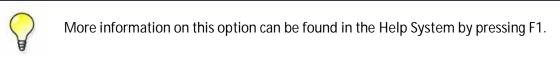
Step 11. Set the loadcase to None to remove the loading from the view.

18 Steel Floor

18.1 Placing Steel Beams

At the truss bottom boom level we will place a floor comprised of steel beams and steel deck of 200 mm thick.

When any floor is established in the model the Floors option must be ticked in the Construction Levels.



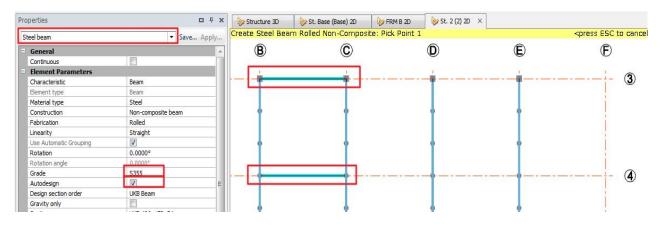
Step 1. In the Model tab, click the Construction Levels command.Step 2. Place a tick next to Truss Bottom in the Floors option.

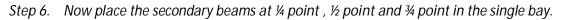
File	Ho	ome Model	Edit		Load	Analyse	Design		Report [Draw	Wi	ndows	
	Juction Fr	ame Sloped Plane *	Grid Line	Con	istruction	Dimension	Steel		Steel Steel Brace		eel	Steel	Portal Frame
	Levels		Grid &		inte Instruction Li	nes 🗔	Coldmin			eel	151	11055	Traine 5
~	onstruction	Lauala						_					F
C	onstruction	i Leveis											
	Ref	Name	Тур	e	Level[m]	Spacing [m]	Source		Slab Th. [mm]	Floor	^		ОК
			T.0.S	•	10.946	0.163	-unique-	•				Ci	ancel
			T.0.S	•	10.783	0.273	-uniqu <mark>e</mark> -	•				Inser	t Above
			T.O.S	•	10.510	0.385	-unique-	•					
			T.0.S	•	10,125	0.502	-unique-	÷				Inser	rt Below
			T.0.S	•	9.623	0.623	-unique-	•				Quick	Above
	3	Eaves	T.0.S	•	9.000	1.500	-unique-	•		-		Quick	Below
	2	Truss Bottom	T.O.S	•	7.500	3.500	-unique-	•		-		-	
	1	First Floor	T.O.S	•	4.000	4.000	-unique-	+		-		D	elete
		a monore service s	200.000	_	10000000			-		~			

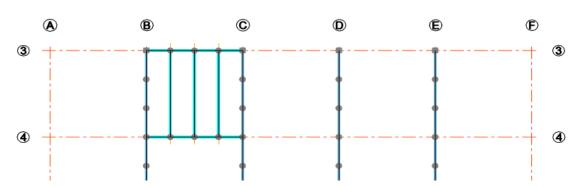
Step 3. Open a 2D view of the Truss Bottom level.

Step 4. Use the Steel Beam command in the Model tab, set in Properties window the followings:

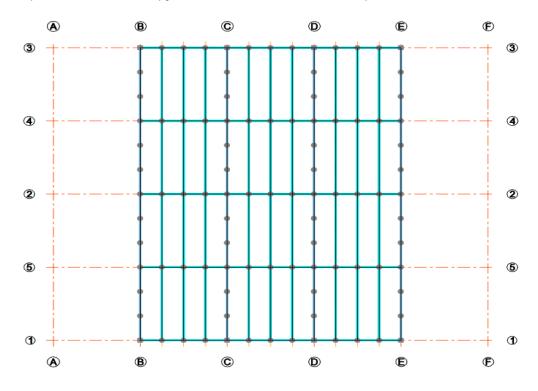
- Grade to be \$355
- Tick on the option Autodesign
- Save this property set as Steel Beam
- Step 5. Establish the floor layout shown below Place the primary beam as shown below in a single bay.







Step 7. Now use the Copy command to establish the floor plate shown below.



18.2 Placing a Steel Deck

Step 1. In the Model tab, in the slab area use the drop list to select the Steel Deck command.



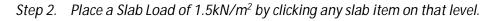
Slab parameters	
Slab properties	
Depth	200.0mm
Diaphragm option	None

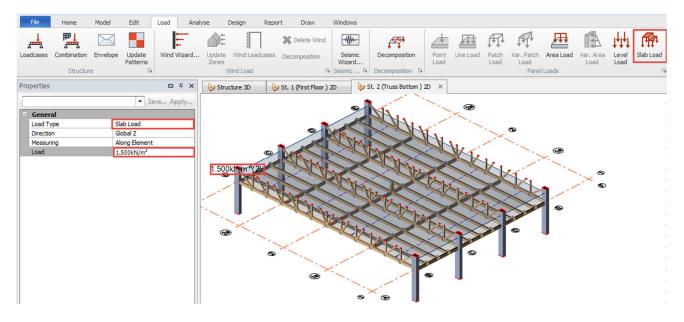
- Step 2. In the Properties window, set the slab Depth to be 200mm.
- Step 3. Drag a window encompassing the whole floor area to place the steel deck items.

18.3 Steel Floor Loading

- Loadcases		#	Loadcase Title	Туре	_	Calc	Include in	Imposed Load	Pattern Load
- 1 Self weight - excluding slabs		1	Self weight - excluding slabs		-	Automatically	Generator	Reductions	
2 Slab self weight 3 First Floor Imposed		2	Slab self weight	Slab Dry	-	V	V		
4 Roof Dead 5 Roof Imposed		3	First Floor Imposed	Imposed	-		V		
6 First Floor Additional Dead		4	Roof Dead	Dead	-		V		
7 Imposed Slab Floor		5	Roof Imposed	of Impo	•		V		
	(6	First Floor Additional Dead	Dead	-		V		
	Ξ.	7	Imposed Slab Floor	Imposed	-		V		100

Step 1. Create a new loadcase Imposed Steel Floor with Type set as Imposed.





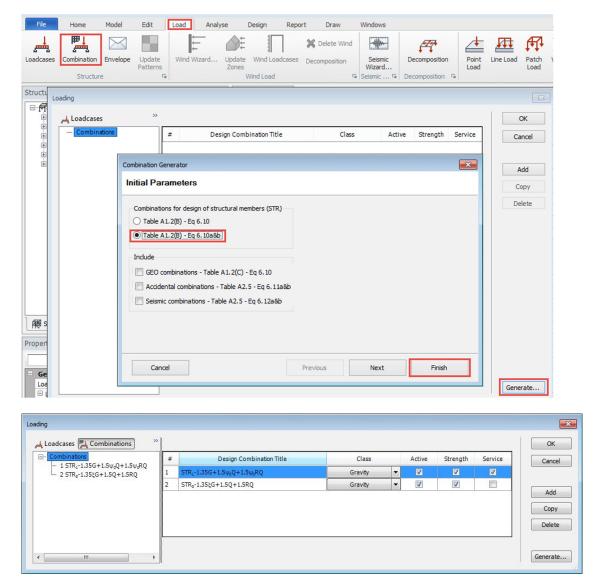
18.4 Combinations

Since new loadcases have been added to the model, the current load combination list are not up to date.

Available Loadcases	>>	<<			Included		
Loadcase Title	Type	Loadcase Title	Strength	Service	SLS Quasi		
7 First Floor Additional Dead	Dead	0 Self weight - excluding slabs	1.350	1.000	1.000		
8 First Floor Imposed	Imposed	1 Slab self weight	1.350	1.000	1.000		
9 Roof Dead	Dead	Loadca	eee Not	-			
10 Roof Imposed	Roof Imposed	Loadcases Not Included					
11 Imposed Slab Floor	Imposed		1000				

- Step 3. Pick the same options as previous.
- Step 4. In the Load tab, click the Combination command.
- Step 5. Delete the previous established combinations.
- Step 6. Click the Generate... button in the combination dialogue (at bottom right).

- Step 7. Pick the option of Table A1.2(B) Eq 6.10a&b.
- Step 8. Untick all other combinations options.
- Step 9. Press Finish.



Step 10. Check the individual combinations that the combinations contain all the loadcases.

Available Loadcases	>>	<<			Included
Loadcase Title	Туре	Loadcase Title	Strength	Service	SLS Quasi
EHFors-		0 Self weight - excluding slabs	1.350	1.000	1.000
EHF _{birb}		1 Slab self weight	1.350	1.000	1.000
EHF _{Dr2+}		7 First Floor Additional Dead	1.350	1.000	1.000
EHFore		9 Roof Dead	1.350	1.000	1.000
		8 First Floor Imposed	1.050	1.000	0.300
		11 Imposed Slab Floor	1.050	1.000	0.300
		10 Roof Imposed	1.050	1.000	0.000

Step 11. Press OK to exit the combinations dialogue.

19 3D Model Validation and Analysis Results

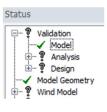
With the 3D model created we will perform the validation to check for modelling errors and then the analysis for any issues in the solver.

19.1 Validation

Step 1. Press the Validate button either off the quick access button or the end of the ribbon.

File	Hon	Model Edit	Load	T Analyse	Design	Report	Draw Win	dows						a 😠 👔
Loadcases Structure	**	Delete Wind Decomposition	Seismic Load *	Decomposition	Point Load	Line Load	Patch Load	 Var. Area Loa Level Load Slab Load 	d Sa	E Full UDL UDL VDL	Trapezoidal Load Point Load Moment Load Member Loads	R Torsion UDL	# P	Validate 74

Step 2. Check in the Status tab under Project Workspace to see if there are any issues reported.





The model has passed validation showing green ticks next to Model and Model Geometry.

19.2 Analysis Options

Before we undertake the analysis we will have a look at the analysis option for the model created.

Step 1. In the Analyse tab, click the Options command.

File	Home Model	Edit Load	Analyse Des	ign	Report	Draw Windows	Results	
ŝ	1st Order Linear 1st Order Non-linear	2nd Order Linear 2nd Order Non-linear	FE chase-down Grillage chase-down		Tabular Data	Mesh Slabs	Toggle Nodes Update	
Options	1st Order Vibration	2nd Order Buckling				Update Wall Beams		
Options 🗔	1st Order Analysis 🗔	2nd Order Analysis	Chase-down	Fai	Solver 🗔	Meshing 🖬	Diaphragms 5	

The following dialogue is displayed – here is where we can control the different options for analysis and the control of inertias in the model. These settings will be carried through to the design module

As shown below we have made alterations to inertial properties concerning concrete and separated them to be 'Cracked' and 'Uncracked'. Each element will allow you to specify which option is applicable.

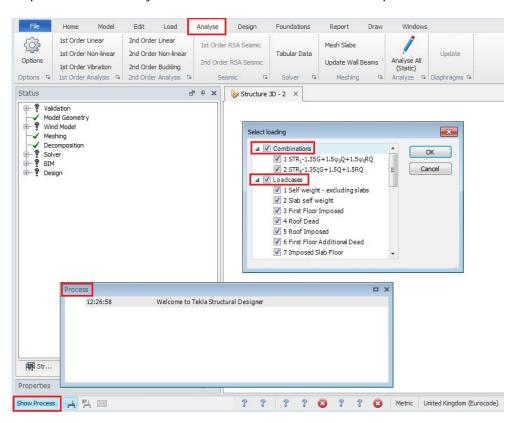
Assume cracked	Yes

116 ((257)
-------	-------

1 ^{se} Order Non-Linear 2 nd Order Non-Linear	Element Type	E	G	I torsion	I major	I minor	Area	A minor	A major	t
1 [⊄] Order Vibration 2 nd Order Buckling	Mid Pier Wall Cracked	0.200	0.200	1.000	1.000	1.000	1.000	1.000	1.000	
1 [#] Order Seismic	Mid Pier Wall Uncracked	0.400	0.400	1.000	1.000	1.000	1.000	1.000	1.000	Ĵ
Modification Factors	Meshed Wall Cracked	0.200	0.200							1.000
Concrete Building Analysis	Meshed Wall Uncracked	0,400	0.400							1.000
Grillage chase-down	Column Cracked	1.000	1.000	0.200	0.200	0.200	1.000	1.000	1.000	
 FE chase-down Vibration Analysis (and RSA) 	Column Uncracked	1.000	1.000	0.400	0.400	0.400	1.000	1.000	1.000	Ĵ.
- Steel	Beam Cracked	1.000	1.000	0.010	0.200	0.200	1.000	1.000	1.000	
Building Analysis	Beam Uncracked	1.000	1.000	0.010	0.400	0.400	1.000	1.000	1.000	
E. Timber Building Analysis	Flat Slab	0.200	0.200							1.000
⊡ Cold formed	Beam and Slab	0.050	0.050		6	1				1.000

19.3 Analysis Process

- Step 1. Open the Show Process window.
- Step 2. Run the 1st Order Linear analysis in the Analyse tab.
- Step 3. Select to analyse all loadcases and combinations and press OK.

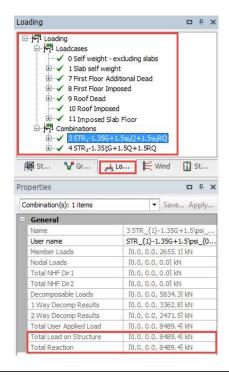


Step 4. Check the process window to ensure the analysis has completed.

Proces	ss			D X
Time			Message	Duration
4	18:57:30		Welcome to Tekla Structural Designer	
D	1	20:34:13	First-order linear	00:03

19.4 Loading Summaries

Step 1. Check the Loading Summaries in the Loading tab under Project Workspace. Check for load applied on the structure against the total reaction.





The green ticks in the Project workspace window indicates correct loading decomposition.

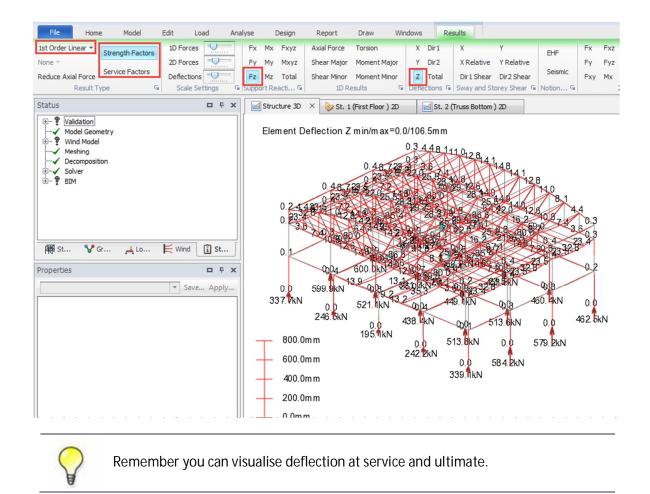
19.5 Deflections

Step 1. Check the deflections of the structure by switching to the Results View.

😓 🍸 🛃 🖧 ABS REL POL 2D

Step 2. Display the following results in the 3D view:

- Vertical Z Deflections
- Combination STR₃-1.35**x**G+1.5Q+1.5RQ
- Vertical Z Reactions



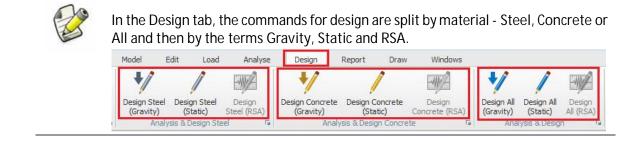
Step 3. Explore other diagrams and tabulated results – ensure all deflections are less than 1m.



It is always recommended that you have a stable model (deflections less than 1m in all directions) before you conduct any form of design.

20 The Design

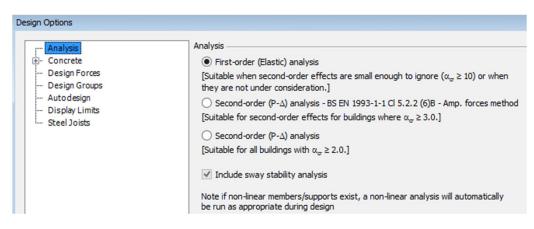
TSD will combine the Analysis and Design of elements and reinforcement into a single operation. Currently the only materials which are allowed for design are Steel and Concrete.



20.1 Design Options

Before any design is undertaken the Design Options for the model must be reviewed.

• Click the Options command and the following dialogue is displayed – this area is explained in more detail in the material specific courses but we will review some of the options.



20.1.1 Analysis

As mentioned before the analysis and design is performed as a single operation, but this dialogue you have control of which type of analysis is performed (1st Order, P-Delta, 2nd Order).



TSD will pick the appropriate analysis type depending if non-linear geometric objects existed in the model. Example – tension-only bracing, compression/tension only spring or support.



More information on this topic will be discussed in the later chapter.

20.1.2 Concrete

This is the reinforcement settings for individual concrete elements.

Analysis Concrete	General		
- Reinforcement Parameters	Country	UK	~
Reinforcement Settings Detailing Options	Longitudinal bars		
Top Longitudinal Bar Pattern	Minimum bar size	12	
Bottom Longitudinal Bar Patt Link Settings	Maximum bar size	32	
General Parameters	Minimum side bar size	16	
Reinforcement Layout	Minimum top steel clear spacing	75.0	mm
Detailing Options General Parameters	Minimum bottom steel clear spacing	50.0	mm
Wall Reinforcement Layout	Maximum tension steel spacing	200.0	mm
Detailing Options	Maximum compression steel spacing	200.0	mm
- General Parameters	Use single bars when beam width \leq =	150.0	mm
Reinforcement Layout Patches	Short Span		
General Parameters	зногезран		
esign Forces	Short span maximum length	3500.0	mm

20.1.3 Design Forces

During the design process the individual elements can encounter other (secondary) forces preventing the design (example, axial force encountered in a concrete beam design).

Any design forces smaller than the values defined here are ignored in the design calculations.

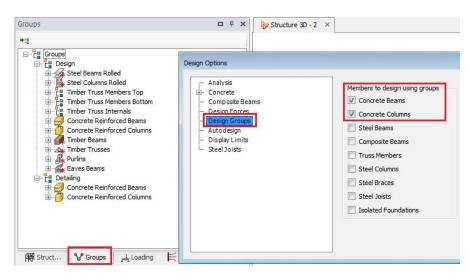
esign Options			
i Analysis ⊕ Concrete	Ignore Forces Below		
Design Forces Design Groups	Torsion Force	0.1	kNm
Autodesign	Axial Force	0.5	kN
Display Limits Steel Joists	Minor Axis Shear Force	1.0	kN
	Minor Axis Moment	0.1	kNm
	Major Axis Shear Force	0.5	kN
	Major Axis Moment	0.1	kNm
	Slab Design Moment	0.000	kNm/m
	Concrete Beams		
	Torsion force % of concrete resistance	20.00	%

This dialogue gives a	permissible limit which	is controlled by the user.
-----------------------	-------------------------	----------------------------

Some forces greater than the values set above will be ignored by the individual member design, since they cannot be handled by the element type. The element will be flagged as a warning (a yellow triangle) to advise of this and the member design results will clearly detail this.

20.1.4 Design Groups

Design groups are based on the geometric properties of the element, vertical or horizontal. These elements are placed together in a group and the worst case design of section or reinforcement applied to all members of the same group.





Concrete elements are grouped together while steel elements are designed individually.

Individual groups can be controlled via the Project Workspace – Groups tab.



More information on this topic will be discussed in the later chapter.

20.1.5 Autodesign

TSD predominately has two modes for design:

- Design Mode automatically select a section size / reinforcement which is adequate to pass the design of the element
- Check Mode a section size / reinforcement is already prescribed and the program will check if they are adequate to pass the design of the elements.

The Autodesign option controls what happens to individual steel and concrete member Autodesign settings at the end of the design process.

Analysis Concrete Composite Beams Design Forces Design Groups Autodesign Display Limits Steel Joists	Steel Members Reset Autodesign to off Always Never When check status is at worst Pass	Isolated Foundations Reset Autodesign to off Always Never When check status is at worst Pass
	Concrete Members Reset Autodesign to off Always Never When check status is at worst Pass	

The first two options determine if the individual members Autodesign setting are cleared or not as follows:

- Always The Autodesign setting is automatically cleared at the end of the design process putting every member into check mode.
- Never The Autodesign setting (either checked, or unchecked) is always retained at the end of the design process.

The third option "When check status is at worst" makes the change from Autodesign, conditional upon the design status (following a design) as follows:

- Pass The Autodesign setting is only automatically cleared at the end of the design process for members with design status: Pass.
- Warning The Autodesign setting is only automatically cleared at the end of the design process for members with design status: Pass, or Warning.
- Fail The Autodesign setting is only automatically cleared at the end of the design process for members with design status: Pass, Warning, or Fail.
- Invalid The Autodesign setting is only automatically cleared at the end of the design process for members with design status: Pass, Warning, Fail or Invalid.
- Beyond Scope The Autodesign setting is only automatically cleared at the end of the design process for members with design status: Pass, Warning, Fail, Invalid or Beyond Scope.



The most practical use of the "When check is at worst" status would be to set it to Pass and start with all members in Autodesign mode. At the end of the first design run passing members would be set to "Check mode", allowing you to focus on the remaining members still in "Autodesign mode".

For Concrete elements Auto design mode is default set to 'Never' – reinforcement is checked and if failing a manual integration must be performed.

For Steel elements Auto design mode is default set to 'Always' – a passing section is always sought for the design.

20.2 Gravity Design

For initial sizing of steel members and reinforcement we can perform a gravity design which will only analyse and design elements/reinforcement to the gravity combinations.



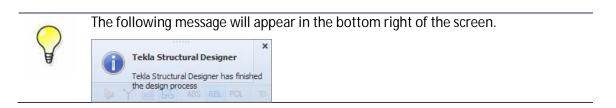
Only 2 combinations have been created and classified as Gravity, therefore only these two combinations will be analysed and the design produced from the results.

ding							
🛓 Loadcases 📕 Combinations 🔌							
Combinations	#	Design Combination Title	Class		Active	Strength	Service
	3	$STR_1-1.35G+1.5\psi_0Q+1.5\psi_0RQ$	Gravity	~	-	✓	✓

- Step 1. Open the Show Process window.
- Step 2. In the Design tab, click the Design All (Gravity) command.



This button will design both steel and concrete elements (for gravity combinations).



Step 3. Review the Show Process window to ensure all aspects of the design have been completed.

16	:59:37	Steel & Concrete Design (Gravity)	00:02
Þ 🗸	16:59:37	3D pre-Analysis	Less than 1s
Þ 🖌	16:59:38	3D analysis: First-order linear	Less than 1s
Þ 🖌	16:59:38	Grillage chase-down	Less than 1s
- 1	16:59:38	Calculating chase-down combinations	Less than 1s
Þ 🖌	16:59:38	FE chase-down	Less than 1s
- 🗸	16:59:39	Calculating chase-down combinations	Less than 1s
- 1	16:59:39	Running design of 4 concrete element groups	Less than 1s
- 🗸	16:59:39	Running check of 20 elements	Less than 1s
- 1	16:59:40	Running check of 31 elements	Less than 1s

- Step 4. Check the Loading Summaries in the Loading tab under Project Workspace. Check for load applied on the structure against the total reaction.
- Step 5. Check the deflections for the structure in the Results View.



20.3 Design Review Mode

Once a design has been undertaken you are automatically switched to the Review View where the model design status and other information can be displayed on the screen and reviewed.



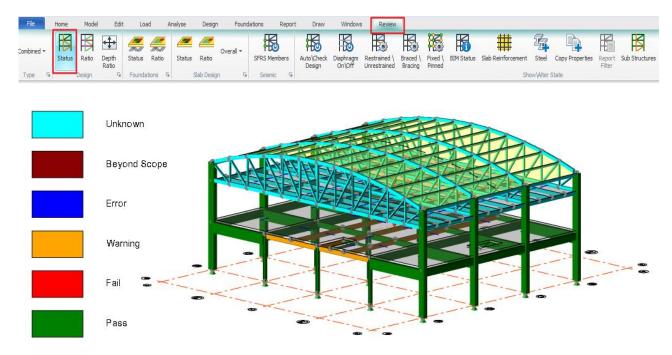


Most these options will be covered in detail in the later chapters, but we will review a few of them now.

20.3.1 Design Status

The Design - Status command uses colour codes to graphically display the design status of each member.

The key displays Pass, Fail, Warning, Error, Beyond Scope or Unknown status of the entities.



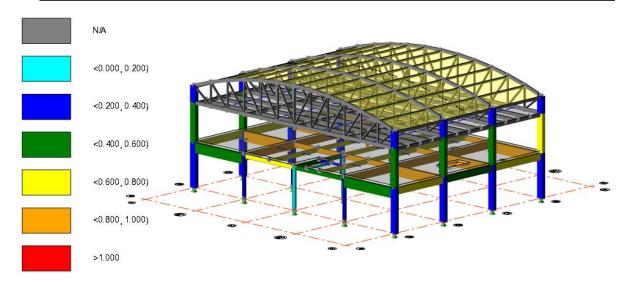
20.3.2 Design Ratio

The Design - Ratio command uses colour codes to graphically display the design ratio of each member.

Home	Model	Edit	Load	An	alyse	Design	Fou	und	ations R	eport	: Draw	Windows	Review				
Status	Ratio	Depth Ratio		latio	Status	Aatio	Overall 🔻		SFRS Membe	ers	Auto\Check Design	Diaphragm On/Off	Restrained \ Unrestrained	Braced \ Bracing	Fixed \ Pinned	BIM Status	S
	Design	Gi	Foundation	ns 🗔	S	lab Desigr	1	6	Seismic	G							



The 'N/A' colour code is assigned to those members that are either beyond scope or have yet to be designed.



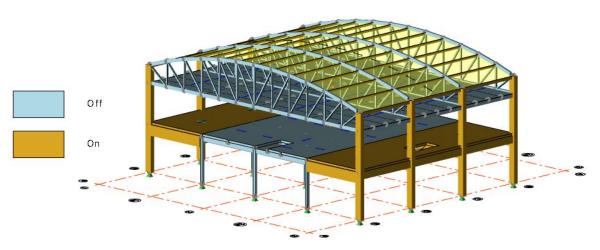
20.3.3 Show\Alter State - Auto\Check Design

Graphical switch for concrete/steel elements from Design to Check Mode.



Each member is colour coded to indicate its Autodesign setting (On or Off).

- · Clicking on an individual member toggles its Autodesign setting.
- Dragging a box from left to right toggles the Autodesign setting for all members totally enclosed by the box.
- Dragging a box from right to left toggles the Autodesign setting for all members that are either enclosed by the box, or are cut by the box perimeter.
- Depressing the SHIFT key and dragging a line toggles the Autodesign setting for all members that cross the line.



20.3.4 Show\Alter State - Steel button

The Steel command provides a means to graphically review and modify the section size and or grade applied to steel members.



To graphically copy the section size and grade between members:

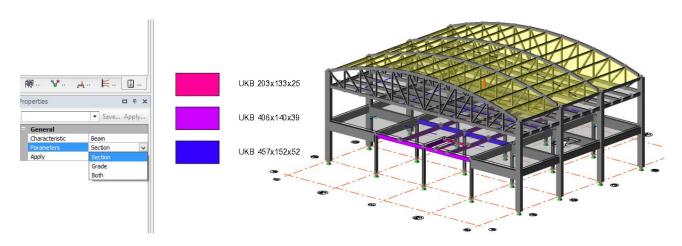
- 1. Click Review > Auto\Check Design and ensure that Autodesign is off for the members in question then click Review > Steel
- 2. In the Properties window, select the parameter to copy (Section, Grade or Both).
- 3. Click on the member containing the steel to be copied.
- 4. Click on the member(s) to which you want to apply the steel.



The member clicked on has to be of the same type (beam, column, or brace) as the source member.



To change the source being copied from, press Esc and then select a different member.



20.3.5 Design Tabular Data

Tabular design information such as Design Summary, Sway, and Storey Drift etc. can be displayed on the screen.

Report Draw	Windows	Review									
Restrained Unrestrained	Fixed (Pinned	BIM Status	Slab Reinforcement	Steel	Copy Properties	Report Filter	Sub Structures	Punch Check Position	Concrete Beam Flanges	Tabular Data	
			Show\Alter State						5	Design Data	5



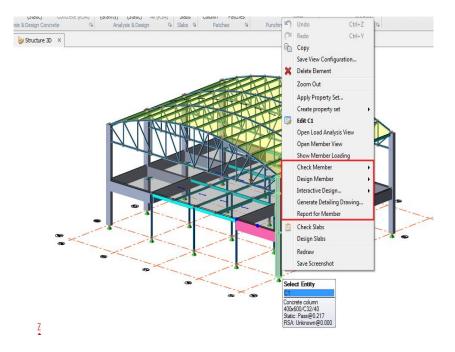
This will open up another window - select the relevant options.

File Home	Model	Edit	Load	Analyze	Design	Report	Draw	Window	s	Review Data						
	Steel	Cold Form	ed Bear	ns Purlin	s Wa	ills		Composite	R	olled	Westok Plated	Concrete	Encased	Members	Type Structure	•
esign Summary 🔹	Concrete		Colu	mns Rails	Sla	bs	Trusses			ated	Fabsec Plated	Reinforce	d l	Loose Bars		
	Timber	Cold Rolle	d Brac	es Eave	s Beams Por	tal Frames		Non-compos	ite V	/estok Cellular	Concrete Filled	Precast		Mesh	Items	*
View Type 🛛 🖓	Materi	ial Type	Fai		Characterist	c	Gi	Construction			Fabrication		Fai	Content 🖼	Filter	
tatus		• • ×	62 Stru	ture 3D	Revie	w Data 🛛 🗙	1									
🕂 – 🤋 Validation															Design Summary	
Model Geometry			Member	Reference	Group Ref.	Span No.	Section	n Grade	Length	No. Connector	Utilization	Status	Results			
Meshing			SB 1/B/1a	-1/B/2	SBR1	1	UKB 457x1	52x52 S355	6.000		0.864	✓ Pass	Results			
Decomposition			SB 1/B/2-	1/B/2a	SBR1	1	UKB 457x1	52x52 S355	6.000		0.864	✓ Pass	Results			
Solver Solver BIM			SB 1/C/1a	-1/C/2	SBR1	1	UKB 457x1	52x52 S355	6.000		0.853	✓ Pass	Results			
± ? Design			SB 1/C/2-	1/C/2a	SBR1	1	UKB 457x1	52x52 S355	6.000		0.853	✓ Pass	Results			
			SB 1/D/1a	-1/D/2	SBR1	1	UKB 457x1	52x52 S355	6.000		0.853	✓ Pass	Results			
			SB 1/D/2-	1/D/2a	SBR1	1	UKB 457x1	52x52 S355	6.000		0.841	✓ Pass	Results			
			SB 1/E/1a	1/E/2	SBR1	1	UKB 406x14	40x39 S355	6.000		0.649		Results			
			3D 1/C/18	-1/1/2	our ca	-						_ manning		-		

20.4 Right Click Context Menu

Individual element design information can be displayed by right click on an element in the model.

20.4.1 Concrete Elements



20.4.2 Check Member – Concrete

Check Member will display the design results for that element to the set design code of practice.

	Longitud	inal Bars S	ummary					
Links Summary	Stack	Section	Longitudinal Bars	Analysis	Combination	Critical position	Ratio	Status
Stack 3 400x600 Stack 2 400x600	3	400x600	10H12	3D Building Analysis	4	Bottom	0.207	🖌 Pass
Stack 2 400x600	2	400x600	10H12	FE Chase Down	4	Bottom	0.478	🖌 Pass
	1	400x600	10H12	FE Chase Down	4	Тор	0.297	🖌 Pass



Showing summary as well as individual element information by expanding the tree. Highlight any option in the tree to display the information.

4 STR 1 35%G+1 50+1 5P(Braced stack in this direction ▷ Second-order moment M₂ = I ▷ Top end design moment ▷ Bottom end design moment ▷ Mid-fifth design moment	MIN[M _{2,curv} / M _{2,stiff}] = 0.0 kNm M _{Ed,top} = M _{min,2} = 2.9 kNm M _{Ed,btm} = M _{btm,2} = 10.9 kNm M _{Ed,mid} = M _{mid,2} = 7.2 kNm	
-------------------------	--	---	--

20.4.3 Design Member – Concrete

An individual element can be auto-designed to find a reinforcement design configuration to all combinations.

─✔ Links Summary	Stack	Section	Longitudinal Bars	Analysis	Combination	Critical position	Ratio	Statu
	3	400x600	10H12	3D Building Analysis	4	Bottom	0.207	🖌 Pas
 ✓ Stack 3 400x600 ✓ Stack 2 400x600 ✓ Stack 1 400x600 	2	400x600	10H12	FE Chase Down	4	Bottom	0.478	🖌 Pas
Stack I howeve	1	400x600	10H12	FE Chase Down	4	Тор	0.297	🖌 Pas
Settinas							Ĩ	Close



The blue exclamation mark indicates the critical option in the design, along with any relevant causes in the codes of practice.

FE Chase Down	Stack 2 400x600 - Shear Links - 3D Building Analysis - 4 STR ₃ -1.35	ξG+1.5Q+1.5RQ - Major Load Direction
😟 🖌 3 STR ₁ -1.35G+1.5ψ	Largest design shear force in stack	∨ _{Ed,max} = 22.0 kN
	Stress state factor	α _{ew} = 1.000 EN 1992-1-1:2004 Section 6
Bar Limit Checks	Concrete design compressive shear strength	f _{cwd} = 21.3 N/mm ² EN 1992-1-1:2004 Section 3
🖃 🎻 Shear Links	Cracked concrete reduction factor	ν ₁ = 0.523 EN 1992-1-1:2004 Section 6
∃ 3D Building Analysis ∃ 3 STR ₁ -1.35G+1.5ψ	Maximum angle of compression strut	θ _{max} = 45.0000 ° EN 1992-1-1:2004 Section 6
	Maximum allowed shear stress v _{Rd.max} =0.9 × v	$v_{row} \times v_1 \times f_{rowd} / (\cot(\theta_{max}) + \tan(\theta_{max})) = 5.0 \text{ N/mm}^2$
Major Load Dire	Shear area	A _w = 220400 mm ²
Minor Load Dire FE Chase Down	△ Maximum shear resistance of section	V _{Rd.max} = v _{Rd.max} × A _w = 1107.0 kN
Grillage Chase Down	Unreinforced shear resistance of section	$\bigvee_{Rd,c} = v_{Rd,c} \times A_{w} = 108.8 kN$
🗸 🖌 Bar Limit Checks 🗸	Unreinforced shear resistance sufficient	
· · · · · · · · · · · · · · · · · · ·	<	



Individual combination design results will be listed in the tree – but the critical combination listed in the summary.

20.4.4 Design... - Interactive Reinforcement Concrete Design

Manual interactive reinforcement design can be conducted for all concrete elements in the project.

Longitudinal Bar Design

	Longitudinal	Links	Interaction Diag	grams					
C16 - 2, max UR = 0.478 C16 - 1, max UR = 0.297	Longitudinal E	Bars							400x600
	Principal bar	size	H12 🗸						Stack length = 4000 mm
	Intermediate	bar size	H12 ¥						Containment status: Pass 🖌
	Int. length	Count	Ctr spacing [mm]		Int. length	Count	Ctr spacing [mm]		4 3
	1-2	1	151.0	1	3-4	1	151.0	1	
	2-3	2	167.3	1	4-1	2	167.3	1	
M _{Ee} M _{res}			4						600 mm
					Longitudina	l Bars		^	
	Position			Тор	Mid-fift	h Botto	om		\sim
	M _{Ed} [kNm]			51.0	31.9	9.6	5		
	M _{res} [kNm]]	1	171.	7 173.1	165.	7		1 2
	Ratio	Ratio		297	 0.184 	0.058	1		400 mm
	Ivacio								

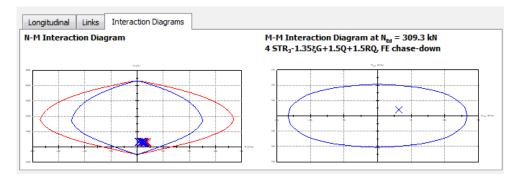
Link Design

.

.

Interactive Column Design	Longitudinal Links Totaraction Discreme	
	Longitudinal Links Uinks Use support link regions Link spacing 125.0 Ink size H8<	400x600 Stack length = 4000 mm Containment status: Pass ✓ 4 3 4 3 1 00 1 00 1 00 0 0 0 0 0 0 0 0 0 0 0 0

Interaction Diagrams

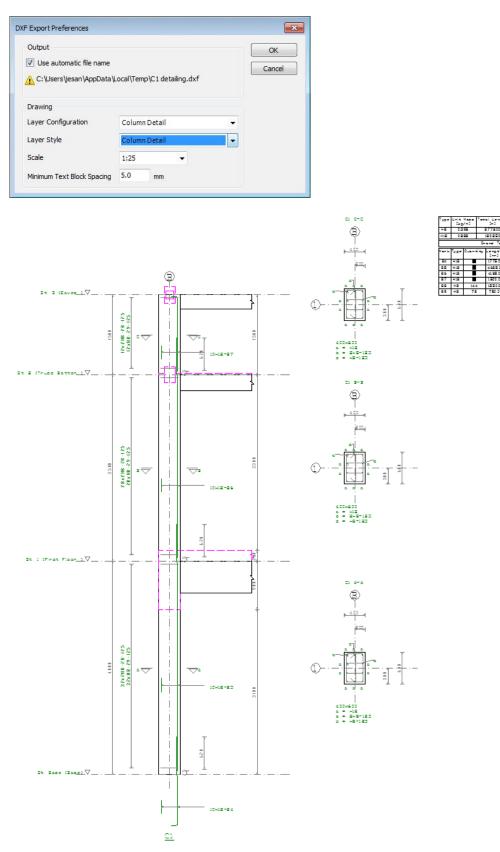


45.22

15.000 222.200 54.000

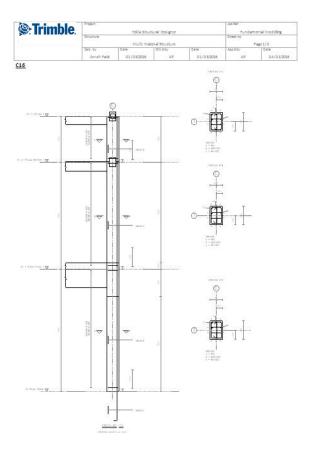
20.4.5 Generating Detailed Drawings

Single element DXF drawing can be produced showing reinforcement details for the element.



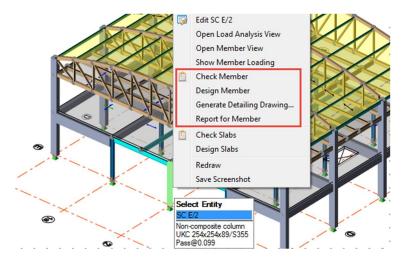
20.4.6 Report for Member

A simple defaulted element report can be obtained – these can be modified as explained later in the manual.



0	Trim	hle	Project		Tokia Sie	ctural	Designer			Jais		dam	ental M	odelen
		IUIC.	Structure		Muls N	storia	Studium			574	Sheet no. Fage 2/2			
3		Calc. by Anialt	Cate Cate 01/05/2005		10	SKEBY AF	0	01/05/2015		App d by		54	te 24/01/202	
ongitu	idinal Bars	Summary	1									-		
Stack	Section	Longitudi	inal Bars	1	Analysis	Con	nbination	Critic	al position	Rat	io S	tatu	15	
3	400x600	10H	12	3D Bui	ilding Analysis		4	3	Bottom	0.20	07 🖋	(P	855	
2	400x600	10H	12	FE C	Chase Down		4	1	Bottom	0.47	78 🦋	/ P	855	
1	400x600	10H	12	FE (Chase Down		4		Тор	0.25	97 🖋	/ P	855	
Links Se	ummery	2		0		22				83 - E	- 22-		1	
Stack	Section	Top supp links		inks	Bottom support lin		Analys	is	Combinati	on F	Ratio	St	atus	Ĩ
3	400x600	8	21	8-125	1999 (B)		3D Build Analys		4	1	0.182	1	Pass	
2	400×600		2+	8-125	8		3D Build Analys		4	0	0.202	1	Pess	
1	400x600	10.00	21	8-125			FE Chase 0	Down	4	(0.083	1	Pass	е

20.5 Steel Elements



20.5.1 Check Member - Steel

This provides the steel section size design results to a specific code of practice.

Summary UKC 254x254x89(S355)	Sun	mary UKC 254x254x89(5355	5)					
Classification		Design Condition	Combination Name	Design Value	Design Capacity	Units	U.R.	Status
Shear Major Shear Minor	\triangleright	Classification	3	Class 1			-	🖌 Pass
Buckling Shear Web	100	Shear Major	No	Significant	Forces	kN	23	Not required
Moment Major		Shear Minor	No	Significant	Forces	kN	52	Not required
Moment Minor		Buckling Shear Web	-	21.913	59.423		-	🖌 Pass
Axial	\triangleright	Moment Major	4	-0.5	422.2	kNm	0.001	🖌 Pass
Axial Bending Combined	\triangleright	Moment Minor	4	7.7	198.5	kNm	0.039	🖌 Pass
Buckling Lateral Torsional Buckling Compression		Axial	4	191.9	3909.2	kN	0.049	🖌 Pass
Buckling Compression Buckling Combined		Axial Bending Combined	4	2	<u>10</u>	-	0.039	🖌 Pass
erening comprised		Buckling Lateral Torsional	4	-0.5	410.3	kNm	0.001	🖌 Pass
	\triangleright	Buckling Compression	4	191.9	2617.5	kN	0.073	🖌 Pass
		Buckling Combined	4	<u>0</u>	. I <u>e</u>	1	0.099	🖌 Pass



All aspects of the design can be interrogated and the blue exclamation mark indicated the critical condition.

Glassification Major Axis Shear Major Design value Moment, M _{yEd} =-0.5 kNm Shear Major Design value Moment, M _{yEd} =-0.5 kNm Moment Major Plastic section modulus, W _{pky} = 1223.9 cm ³ Moment Major Plastic section modulus, W _{pky} = 345.0 N/mm ² ✓ 4 STR ₂ -1.35G+1.5v ₄ CH 1.5v ₄ R Plastic section modulus, W _{pky} = 345.0 N/mm ² ✓ 4 STR ₂ -1.35G+1.5v ₄ CH 1.5v ₄ R Design resistance, M _{c,yRd} = 422.2 kNm EN 1993-1-1: 2005 CI 6.2.5(2) Moment Minor Pass	
Buckling Combined Bit of the second seco	

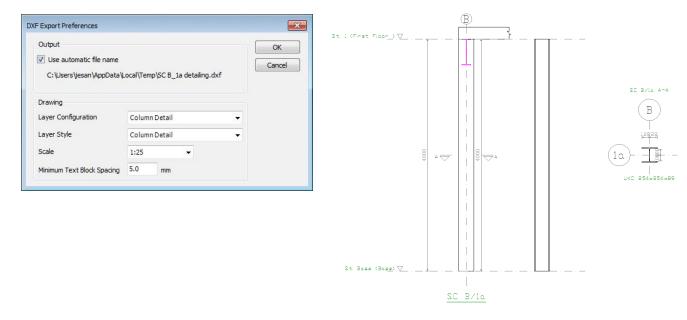
20.5.2 Design Member – Steel

An individual steel element can be auto-designed to find a section size, satisfying all design combinations which have been analysed.

Summary UKC 254x254x89(S355	Sull	mary UKC 254x254x89(5355	-)		~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~			
 Classification 		Design Condition	Combination Name	Design Value	Design Capacity	Units	U.R.	Status
- En Shear Major - En Shear Minor - ✓ Buckling Shear Web	\triangleright	Classification	3	Class 1	-	-	-	🖌 Pass
		Shear Major	No	Significant	Forces	kN	-	Not required
Moment Major		Shear Minor	No	Significant	Forces	kN	-	Not required
 Moment Minor 	\triangleright	Buckling Shear Web		21.913	59.423			🖌 Pass
🖌 Axial	\triangleright	Moment Major	4	-0.5	422.2	kNm	0.001	🖌 Pass
Axial Bending Combined	\triangleright	Moment Minor	4	7.7	198.5	kNm	0.039	🖌 Pass
 Buckling Lateral Torsional Buckling Compression 		Axial	4	191.9	3909.2	kN	0.049	🖌 Pass
 Buckling Complexition 	\triangleright	Axial Bending Combined	4	0	12	-	0.039	✓ Pass
Combined	\triangleright	Buckling Lateral Torsional	4	-0.5	410.3	kNm	0.001	🗸 Pass
		Buckling Compression	4	191.9	2617.5	kN	0.073	🖌 Pass
		Buckling Combined	4	8	12		0.099	🖌 Pass

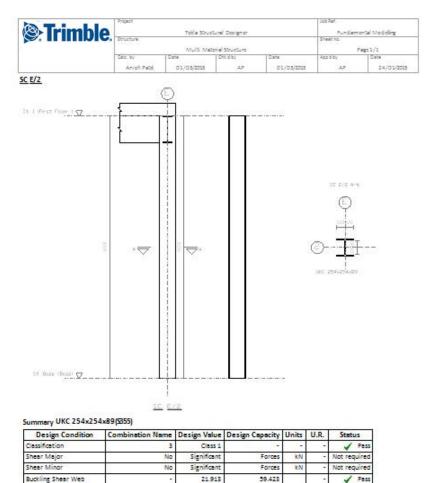
20.5.3 Generating Detail Drawing - Steel

Individual element drawing.



20.5.4 Report for Member - Steel

A simple defaulted element report can be obtained – these can be modified as explained later in the manual.



20.6 Design Status

Once a design has been conducted the status of the model can be visualised.



- (1) Load Condition Wind / Seismic
- (2) Analysis Condition
- (3) Design Condition Steel Gravity
- (4) Design Condition Steel All
- (5) Design Condition Concrete Gravity
- (6) Design Condition Concrete All

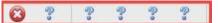


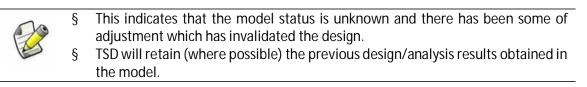
Red Cross – Not valid Green Tick – Valid

Step 1. Edit a steel beam by double left clicking on it - change the section size to UKB 533x165x66 and press OK.

	Size				
General	Span	6000.0 mm			
Size Alignment					
Releases	Steel	S355 V			
Lateral restraints	Steel	S355 V			
Strut restraints	alex-				
Haunches	Section	UKB 533x165x66			
End plates Web openings	Seculi	00X0122222000			
Deflection limits					
Size constraints					
Camber					/
Natural frequency					
Instability factor Seismic					
Scientic					
			OK Cancel	1	
			1		

The graphical status area shows question marks.





Step 2. Right click on the same steel beam as in above and select Check Member.

Summary UKB 533x210x138(S3)	55) Sun	1mary UKB 533x210x138(535	5)					
Classification		Design Condition	#	Design Value	Design Capacity	Units	U.R.	Status
イ Shear Major Ŋ Shear Minor	\triangleright	Classification	3	Class 1	_	-		🖌 Pass
Buckling Shear Web	\triangleright	Shear Major	4	113.3	1685.7	kN	0.067	🖌 Pass
Moment Major		Shear Minor		No	Forces	kN	87	Not required
Moment Minor	>	Buckling Shear Web		34.143	59.423			🖌 Pass
Axial	\triangleright	Moment Major	4	169.9	1246.4	kNm	0.136	🖌 Pass
Axial Bending Combined		Moment Minor	-	No	Forces	kNm	35	Not required
的 Buckling Lateral Torsional Buckling Compression		Axial	-	No	Forces	kN	6	Not required
Buckling Combined		Axial Bending Combined	22	No	Forces	12	1	Not required
/ Deflection		Buckling Lateral Torsional	35	No	Forces	-	17	Not required
		Buckling Compression	-	No	Forces	-	(-)	Not required
		Buckling Combined	-	No	Forces	12	28	Not required
	\triangleright	Deflection Self weight	3	0.1	-	mm		-
	\triangleright	Deflection Slab	3	1.3	24.0	mm	0.055	🖌 Pass



§ Since the previous results are retained, TSD can perform a check on this member.
 § A word of caution if the end fixities are changed and the analysis and design not

A word of caution if the end fixities are changed and the analysis and design not updated, the element may show design results not to the correct forces.

Step 3. Right click again on the same steel beam and select Design Member.



The individual element will run through the Autodesign process and select a section size which is adequate for the previous analysis results.

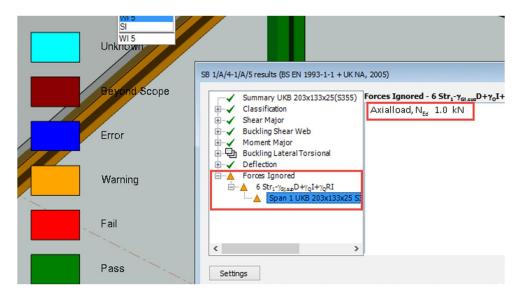
20.6.1 Analysis warnings

As mentioned previously the design procedure can encounter other analysis forces which will invalidate the design – these are control in the design options.

Analysis Concrete	Ignore Forces Below		
Reinforcement Parameters	Torsion Force	0.1	kNm
E- Column	Axial Force	0.5	kN
 Reinforcement Layout Detailing Options 	Minor Axis Shear Force	1.0	kN
General Parameters	Minor Axis Moment	0.1	kNm
. Slab	Major Axis Shear Force	0.5	kN
Design Forces Design Groups	Major Axis Moment	0.1	kNm
Autodesign Display Limits	Slab Design Moment	0.000	kNm/n
···· Steel Joists	Concrete Beams		
	Torsion force % of concrete resistance	20.00	%

If any element has a force exceeding the values stated above, it will be highlighted as a warning.

The design will still continue but ignoring this offending force value. This value can be seen in the Forces Ignored area of the design results.



Step 4. Re-run the Design All (Static) for the model to update the analysis and design results.

21 Lateral Loading

21.1 EHF's – Equivalent Horizontal Forces

The application of EHF's (Equivalent Horizontal Loads) and wind will be covered in the following sections.



For a gravity design a model can have all elements passing, but when lateral combinations are applied and checked, these elements can now fail.

EHF's are automatically calculated by TSD and can be easily applied to the model

EHF's are used to represent frame imperfections. The Eurocode requires they are applied to all combinations. (Lateral wind combinations therefore should also have EHF's applied).

EHF's are automatically derived from the factored load cases within the current combination. They are applied in the analysis as a horizontal force at each beam column intersection with a magnitude of 0.5% of the vertical load in the column at the column/beam intersection.

They are applied to the structure in X and Y global directions as follows: EHF X+, EHF X-, EHF Y+ and EHF Y-

However they are then combined to act in the +Dir1, -Dir1, +Dir2 and -Dir2 directions (because many structures do not have their primary axes along X and Y).

This is achieved by applying the loads themselves in global X and Y as above, and then using the combination factors to set them in Dir1 and Dir2 as required.



Example if the angle between X and Dir1 is +60 degs - TSD applies +X factor 0.5 and +Y factor 0.866. (i.e. cos 60 degs and sin 60 degs).

The net result is that any combination is able to have up to 2 EHFs applied within it - one from X (+ or -) and one from Y (+ or -).

In addition, you are able to set up the combinations manually and apply factors to each as required.

21.2 EHF Control

The control for EHF can be found in the Model Settings > EHF and this is used to control the magnitude of the EHF applied to the structure.

Design Codes				
Units	Height of the structure	0.000	m s	Set Defaul
References	Number of columns in X direction	1		
Drawings	Number of columns in X direction	-		
Grouping Material List	Number of columns in Y direction	1		
Beam Lines	Global initial sway imperfections			
RigidZones CurvedBeams	α"	1.000		
Validation Load reductions	α _{mix}	1.000		
EHF	α _{miY}	1.000		
	$\varphi_X = \varphi_0 \times \alpha_n \times \alpha_{m_X}$	0.50	%	
	$\phi_{v} = \phi_{0} \times \alpha_{n} \times \alpha_{mv}$	0.50	%	

- Height of structure to specify the effective height of the structure to be used in the EHF calculations.
- Set Default button sets the height to the highest construction level.
- Number of Columns in X and Y directions default set as 1



In the Eurocode CI 5.2(5) the calculation of the reduction factor a_m depends on the number of contributing members, m.



Valid input for "m" in the X and Y directions is any whole number from 1 to 1000. The default value is 1 which results in $a_m = 1.000$. If a value of 1000 is entered then a_m would reduce to 0.707.

- Global initial sway imperfections Displays the calculated alpha (a) and phi (f) values for the above input.
- 21.3 Applying Equivalent Horizontal Forces (EHF's)

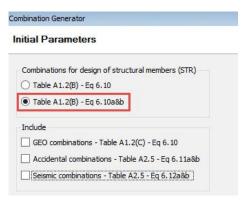
Keeping the defaulted values in the EHF settings, we will apply EHFs to our structure.

- Step 1. In the Load tab, click the Load Combination command.
- Step 2. In the combinations dialogue, delete all previous combinations.
- Step 3. Press the Generate... button.

Loading							×
📥 Loadcases	>>						ОК
Combinations	#	Design Combination Title	Class	Active	Strength	Service	Cancel
							Add
							Сору
							Delete
							Generate

Step 4. Pick the option "Table A1.2(B) – Eq 6.20a&b".

Step 5. Un-tick the options Geo, Accidental and Seismic combinations - press Next.



Step 6. Accept the defaults on the combinations equation generator – press Next.

Combinations

Combination	Generate	Dead	Imposed	Roof Imposed
STR1-1.35G+1.5\0Q+1.5\0RQ	✓	1.350	1.500	1.500
STR ₃ -1.35&G+1.5Q+1.5RQ	~	1.249	1.500	1.500
STR ₅ -1.35G+1.5ψ ₀ Q+1.5ψ ₀ RQ+EHF	~	1.350	1.500	1.500
STR7-1.35&G+1.5Q+1.5RQ+EHF	~	1.249	1.500	1.500

Step 7. Accept defaults on the service factor generator – press Next.

Service

Combination	Generate	Dead	Imposed	Roof Imposed
STR:-1.35G+1.5\v0Q+1.5\v0RQ	Y	1.000	1.000	1.000
STR3-1.35&G+1.5Q+1.5RQ	4			
STR ₅ -1.35G+1.5\u03c6_0+1.5\u03c6_0RQ+EHF	4	1.000	1.000	1.000
STR7-1.355G+1.5Q+1.5RQ+EHF	~			

Step 8. Tick all directions (Dir1+, Dir1-, Dir2+ & Dir2-) to apply EHF's to the repeated gravity equations.

EHF

Combination	Dir1+	Fac _{Dir1+}	Dir1-	Fac _{pirt} .	Dir2+	Fac _{Dir2+}	Dir2-	Fac _{Dir2}
STR1-1.35G+1.5W0Q+1.5W0RQ	Gr	avity E	quatio	ns Rep	eated			1
STR3-1.35&G+1.5Q+1.5RQ	-		1		-			
STR ₅ -1.35G+1.5\u00fc0Q+1.5\u00fc0RQ+EH		1.000	-	1.000	•	1.000	~	1.000
STR1.35&G+1.5Q+1.5RQ+EHF	~	1.000	~	1.000	~	1.000	~	1.000

Step 9. Press Finish to exit the generator and OK to exit the combination dialogue.

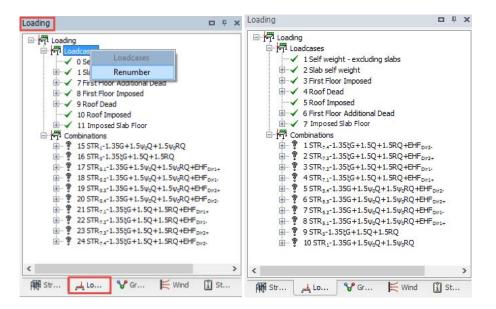
Total 10 combinations are created from the Combination Generator.

ombinations 		Design Combination Title	Class		Active	Strength	Service
	15	STR1-1.35G+1.5wQ+1.5wRQ	Gravity	~	~	~	✓
- 17 STR _{5.1} -1.35G+1.5\u03c6_Q+1.5\u03c6_0RQ+EHF _{Dir1+}	16	STR3-1.35&G+1.5Q+1.5RQ	Gravity	4	~	~	
 18 STR_{5.2}-1.35G+1.5ψ₀Q+1.5ψ₀RQ+EHF_{Dirb} 19 STR_{5.3}-1.35G+1.5ψ₀Q+1.5ψ₀RQ+EHF_{Dirb} 	17	STR _{5.1} -1.35G+1.5\u03c6_0Q+1.5\u03c6_0RQ+EHF_0m1+	Lateral	~	-	~	-
 20 STR_{5,4}-1.35G+1.5ψ₀Q+1.5ψ₀RQ+EHF_{DIR2} 	18	STR _{5.2} -1.35G+1.5\u03c6_0Q+1.5\u03c6_0RQ+EHF _{DIP2} .	Lateral	~	~	~	~
21 STR _{7.1} -1.35&G+1.5Q+1.5RQ+EHF _{Dir1+}	19	STR _{5.3} -1.35G+1.5\u00fcqQ+1.5\u00fcqRQ+EHF _{DIr2+}	Lateral	~	-	•	~
 22 STR_{7.2}-1.35\getG+1.5Q+1.5RQ+EHF_{DIF2} 23 STR_{7.3}-1.35\getG+1.5Q+1.5RQ+EHF_{DIF2} 	20	STR _{5.4} -1.35G+1.5\u03c8_0Q+1.5\u03c8_0RQ+EHF _{DIG2} -	Lateral	4	~	~	~
24 STR7.4-1.35&G+1.5Q+1.5RQ+EHFDIG2	21	STR7.1-1.35&G+1.5Q+1.5RQ+EHFDIr1+	Lateral	~	-	~	
	22	STR7.2-1.35&G+1.5Q+1.5RQ+EHFDirt	Lateral	~	-	-	
	23	STR73-1.35&G+1.5Q+1.5RQ+EHFDIr2+	Lateral	~	-	•	
	24	STR7,4-1.355G+1.5Q+1.5RQ+EHFpir2	Lateral	4	-	-	

21.4 Renumbering Loadcases and Combinations

The Loadcase and Combination numbering has continued from the previous deleted set, we will renumber them to start from 0 again.

Step 1. In the Project Workspace, select the Loading tab.Step 2. Right click over the title Loadcases and pick the option Renumber.



Step 3. Repeat the step above for combinations.

- Step 4. Perform a complete design on all loadcases and combinations.
 - a. Check the Loading Summary
 - b. Check the Deflections of the Structure
 - c. Check the Element Design
 - d. Check the Sway Results from the Tabulated Data

pading	α÷×	Node Number	Mov _x [mm]	Movy [mm]	Mov _z [mm]	Rot _x [°]	Rot, [°]	Rot. [°]
- 취 Loading 글 취 Loadcases		1	0.2	-1.8	0.2	-0.0946	0.0822	0.0010
1 Self weight - excluding slabs		2	0.1	-1.8	0.8	0.1295	0.2884	0.0010
		3	0.0	-1.8	0.8	0.0253	-0.0005	0.0010
		4	-0.2	-1.8	0.8	-0.1269	0.2899	0.0010
		5	-0.3	-1.8	0.2	0.1081	0.0828	0.0010
6 First Floor Additional Dead 7 Imposed Slab Floor		6	-0.5	-2.6	0.3	0.0304	-0.0022	0.0011
Combinations		7	0.0	0.0	0.0	-0.0256	-0.0572	0.0000
1 STR7.4-1.35&G+1.5Q+1.5RQ+EHF Der2-		8	-0.1	-2.1	14.9	-0.0001	-0.0203	0.0012
 2 STR₇₃-1.35\cup{G}+1.5Q+1.5RQ+EHF_{DIF} 3 STR₇₃-1.35\cup{G}+1.5Q+1.5RQ+EHF_{DIF} 		9	0.0	0.0	0.0	0.0957	-0.1469	0.0000
		10	0.2	-2.1	0.3	0.0092	-0.0198	0.0012
		11	0.0	0.0	0.0	0.0253	-0.0008	0.0000
		12	0.1	-1.8	4.9	-0.0185	0.1826	0.0010
		13	0.0	0.0	0.0	-0.0228	-0.1417	0.0000
⊕ 9 STR ₃ -1.35ξG+1.5Q+1.5RQ		14	0.0	-1.7	0.3	-0.0296	-0.0019	0.0012
		15	0.0	0.0	0.0	0.0992	-0.0472	0.0000
		16	-3.9	-2.8	0.4	0.0400	-0.0377	0.0005
		17	0.0	-0.9	0.4	0.0253	-0.0007	0.0005
🕅 Str 📃 🖌 Gr 🗮 Wind	👔 St	18	-3.7	0.4	0.4	0.0153	-0.0342	0.0005

Member Reference	Group Ref.	Span No.	Section	Grade	Length	No. Connectors	Utilization	Status	Results
SB 1/B/1a-1/B/2	SBR1	1	UKB 457x152x52	S355	6.000		0.864	🖌 Pass	Results
SB 1/B/2-1/B/2a	SBR1	1	UKB 457x152x52	S355	6.000		0.864	✓ Pass	Results
SB 1/C/1a-1/C/2	SBR1	1	UKB 457x152x52	S355	6.000		0.853	✓ Pass	Results
SB 1/C/2-1/C/2a	SBR1	1	UKB 457x152x52	S355	6.000		0.853	V Pass	Results
SB 1/D/1a-1/D/2	SBR1	1	UKB 457x152x52	S355	6.000		0.853	V Pass	Results
SB 1/D/2-1/D/2a	SBR1	1	UKB 457x152x52	S355	6.000		0.841	✓ Pass	Results
SB 1/E/1a-1/E/2	SBR1	1	UKB 406x140x39	S355	6.000		0.649	🔺 Warning	Results
SB 1/E/2-1/E/2a	SBR1	1	UKB 406x140x39	S355	6.000		0.593	🔺 Warning	Results
SB 1/B/2-1/C/2	SBR3	1	UKB 406x140x39	S355	6.000		0.640	🔺 Warning	Results
SB 1/C/2-1/D/2	SBR3	1	UKB 406x140x39	S355	6.000		0.640	🔺 Warning	Results
SB 1/D/2-1/E/2	SBR3	1	UKB 406x140x39	S355	6.000		0.526	🔺 Warning	Results
SB 1/B/°102-1/C/°103	SBR3	1	UKB 406x140x39	S355	6.000		0.640	🔺 Warning	Results
SB 1/C/°103-1/D/°104	SBR3	1	UKB 406x140x39	S355	6.000		0.640	🔺 Warning	Results
SB 1/D/°104-1/E/°105	SBR3	1	UKB 406x140x39	S355	6.000		0.605	🔺 Warning	Results
SB 1/B/°106-1/C/°107	SBR3	1	UKB 406x140x39	S355	6.000		0.640	🔺 Warning	Results
SB 1/C/°107-1/D/°108	SBR3	1	UKB 406x140x39	S355	6.000		0.640	🔺 Warning	Results
SB 1/D/°108-1/E/°109	SBR3	1	UKB 406x140x39	S355	6.000		0.640	A Warning	Results

Reference	Combination Dir 1	Stack Dir 1	α _{Dir1}	Combination Dir 2	Stack Dir 2	CL _{D(r2}	Combination Dir 1/2	Twist	Status	Details
SC	1 STR7.4-1.35&G+1.5Q+1.5RQ+EHFpirp	1	4.581	1 STR7,4-1.35&G+1.5Q+1.5RQ+EHF0rg	1	11.145	5 STR _{3.4} -1.35G+1.5\u03c6_Q+1.5\u03c6_RQ+EHF _{DIG2}	1.002	/ Pass	Details
SC B/2	1 STR7.4-1.35&G+1.5Q+1.5RQ+EHFDIG	1	4.574	1 STR7.4-1.35&G+1.5Q+1.5RQ+EHF012-	1	11.145	1 STR7.4-1.355G+1.5Q+1.5RQ+EHFDIG	1.000	/ Pass	Details
SC B/1a	1 STR7,4-1.35&G+1.5Q+1.5RQ+EHF _{DIGP}	1	4.567	1 STR7.4-1.35&G+1.5Q+1.5RQ+EHF012-	1	11.145	5 STR _{3.4} -1.35G+1.5\u03c6_Q+1.5\u03c6_RQ+EHF _{DII2} .	1.002	/ Pass	Details
SC C/2a	1 STR7.4-1.355G+1.5Q+1.5RQ+EHFDIG	1	4.581	1 STR7,4-1.35&G+1.5Q+1.5RQ+EHF _{DIG2}	1	11.822	$5 \text{ STR}_{5,4}$ -1.35G+1.5 ψ_0 Q+1.5 ψ_0 RQ+EHF _{DH2} -	1.002	/ Pass	Details
SC C/2	1 STR7.4-1.35&G+1.5Q+1.5RQ+EHFDIR2	1	4.574	1 STR7.4-1.35&G+1.5Q+1.5RQ+EHF012	1	11.822	1 STR7.4-1.35&G+1.5Q+1.5RQ+EHFDIr2-	1.000	/ Pass	Details
SC C/1a	1 STR7,4-1.35&G+1.5Q+1.5RQ+EHFDIG	1	4.567	1 STR7.4-1.35&G+1.5Q+1.5RQ+EHF012-	1	11.822	5 STR _{5.4} -1.35G+1.5\u03c6_Q+1.5\u03c6_RQ+EHF _{D02} .	1.002	/ Pass	Details
SC D/2a	1 STR _{7.4} -1.35&G+1.5Q+1.5RQ+EHF _{DIGP}	1	4.581	1 STR7.4-1.35&G+1.5Q+1.5RQ+EHF012-	1	12.585	$5 \text{ STR}_{5,4}$ -1.35G+1.5 ψ_0 Q+1.5 ψ_0 RQ+EHF _{DH2} .	1.003	/ Pass	Details
SC D/2	1 STR7.4-1.35&G+1.5Q+1.5RQ+EHFDr2	1	4.574	1 STR7.4-1.35&G+1.5Q+1.5RQ+EHF012	1	12.585	1 STR7.4-1.35&G+1.5Q+1.5RQ+EHFDIG	1.000	/ Pass	Details
SC D/1a	1 STR _{7.4} -1.35\G+1.5Q+1.5RQ+EHF _{DIG}	1	4.567	1 STR7.4-1.35&G+1.5Q+1.5RQ+EHF012-	1	12.585	5 STR _{3.4} -1.35G+1.5\u03c6_Q+1.5\u03c6_BQ+EHF _{Dir2} -	1.003	/ Pass	Details
SC E/2a	1 STR7,4-1.35&G+1.5Q+1.5RQ+EHF _{DIG2}	1	4.581	1 STR7.4-1.35&G+1.5Q+1.5RQ+EHFDI12-	1	13.454	5 STR _{5.4} -1.35G+1.5\u03c6_Q+1.5\u03c6_RQ+EHF _{DII2} .	1.003	/ Pass	Details
SC E/2	1 STR7.4-1.35&G+1.5Q+1.5RQ+EHFDrg	1	4.574	1 STR _{7.4} -1.35ξG+1.5Q+1.5RQ+EHF _{DIG}	1	13.454	1 STR7.4-1.35&G+1.5Q+1.5RQ+EHFDrg.	1.000	/ Pass	Details
SC E/1a	1 STR7.4-1.35&G+1.5Q+1.5RQ+EHFDr2-	1	4.567	1 STR7.4-1.355G+1.5Q+1.5RQ+EHFDIG	1	13.454	5 STR _{5.4} -1.35G+1.5\u00fc0Q+1.5\u00fc0RQ+EHF _{DIR2}	1.003	/ Pass	Details

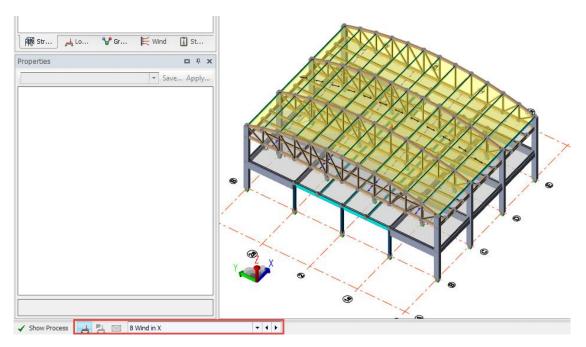
21.5 Wind Loading – Manual Input

To place any type of loading on the structure an empty loadcase has to be created.

- Step 1. In the Load tab, select Loadcases command.
- Step 2. Add a new loadcase "Wind in X" and set the Type to be Wind press OK.

oadcases 1 Self weight - excluding slabs	#	Loadcase Title	Туре		Calc Automatically	Include in Generator	Imposed Load Reductions	Pattern Load
2 Slab self weight	1	Self weight - excluding slabs	SelfWeight	~	•	-		
- 3 First Floor Imposed	2	Slab self weight	Slab Dry	×	•	-		
 4 Roof Dead 5 Roof Imposed 6 First Floor Additional Dead 7 Imposed Slab Floor 	3	First Floor Imposed	Imposed	~		~		
	4	Roof Dead	Dead	¥		-		
	5	Roof Imposed	of Impo	¥		-		
- 8 Wind in X	6	First Floor Additional Dead	Dead	¥		~		
	7	Imposed Slab Floor	Imposed	~		-		
	8	Wind in X	Wind	V		~		

Step 3. Go to Structure 3D view and select the Wind in X loadcase from the loading list option.

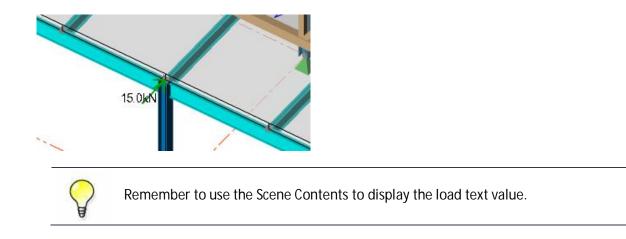


21.5.1 Nodal Loading

- Step 1. In the Load tab, click the Nodal Load command.
- Step 2. Define in the Properties window a load value of 15kN in the X direction.

atch oad	Var. Patch Load Panel	Area Load	Var. Area Load	Level Load	Slab Load	Full UDL	VDL	Trapezoidal Load	Point Load	Moment Load	Torsion Full UDL	Torsion UDL	Torsion VDL	Temperature Load ↓ Temperature Load ↓ Settlement Load Structure Loads
	Panel	Loads			Tal.			Memi	ber Loads	5			13	Structure Loads
Lo	oad Type		Nodal	Load										
	Load		[15.0), 0.0, 0.	0] kN									
	Х		15.0k	N										
	^		15.0K											
	Y		0.0kN											
E														
	Y		0.0kN 0.0kN] kNm									
	Y Z		0.0kN 0.0kN	I I 0.0, 0.0] kNm									
	Y Z Moment		0.0kN 0.0kN [0.0,	I I 0.0, 0.0 Im] kNm									

Step 3. In the Structure 3D view pick a beam to beam or beam to column intersection point to place a nodal load.

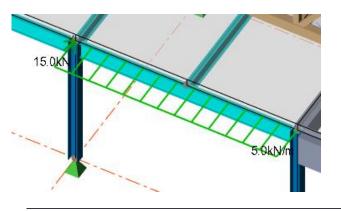


21.5.2 Element Loading

Step 1. In the Load tab and click the Full UDL command.Step 2. Define in the Properties window a load value of 5kN/m in Global X direction.

Æ	A	A	₩	A	ItIt	rffr	Π	П	=		+	6	1	fQ.
Line Load	Patch Load	Var. Patch Load	Area Load	Var. Area Load	Level Load	Slab Load	Full UDL	UDL	VDL	Trapezoidal Load	Point Load	Moment Load	Torsion Full UDL	Torsion UDL
		Panel	Loads			F <u>a</u>				Mem	ber Load	S		
Load Typ	ne .		Membe	r Load										
Member I		De	Full UDI											
Direction			Global)	(~								
Measurin	g		Along E	lement										
Load			5.0kN/	m										

Step 3. In the 3D view, click on a perimeter beam to place the full UDL on it.





When a member load is placed on an element this will induce bending which may invalidate the design (i.e. minor axis on simple beams).

21.5.3 Panel Loading

To place an area load over several elements a decomposition tool must be applied first. Wall and Roof panels are used to decompose an area load to the connecting elements. These panels are one way spanning and have to option to decompose as nodal loads or UDLs (Walls).

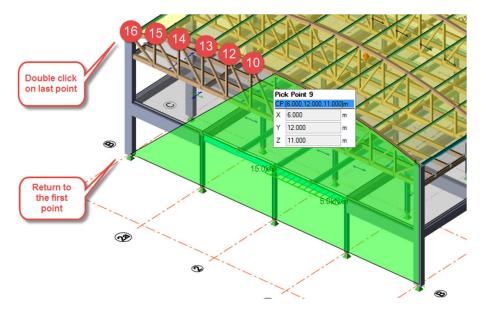
Roofs are placed as flat or sloped panels, Walls are placed vertically.

We will place a Wind Panel along gridline B.



To place a wind panel all intersection points must be placed in sequence and must be planer.

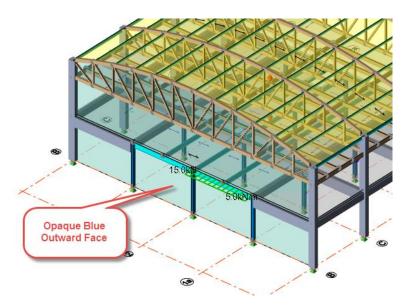
- Step 1. In the Model tab and pick the Wall Panel command.
- Step 2. Pick the intersection points in sequence to place a panel (Remember to pick every intersection point on the top boom timber truss).
- Step 3. Either double left click the last point to end the panel or single click back to the start to finish the creation of the wall panel.

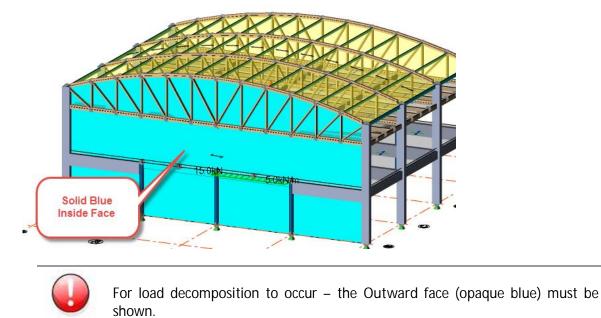


21.5.4 Wall Panel Orientation and Span Direction Decomposition

Wall panels have an inner and outer orientation which is determined by the direction which the sequence of nodes are picked.

Nodes picked in an Anti-clockwise direction will place an opaque blue indicating the Outward side.

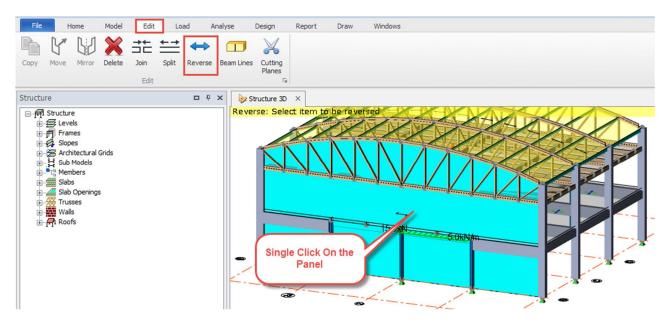




Nodes picked in a Clockwise direction will place a solid blue indicating the Inward side.

21.5.5 Wall Panel Changing Orientation

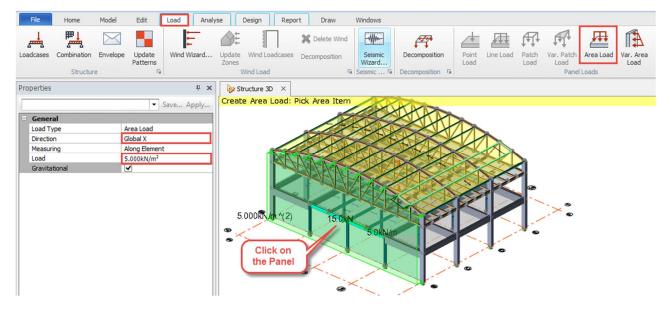
To change the orientation of the wall panel use the Reverse command in the Edit tab.



Step 1. If you have placed the wall panel with the inward face showing – reverse the orientation using the technique described above.

21.5.6 Applying Panel Loading

- Step 1. In the Load tab, pick the Area Load command.
- Step 2. Apply a load value of $5kN/m^2$ in the Global X direction on the panel by clicking on it.



This loadcase can now be merged in a combination in the usual manner.

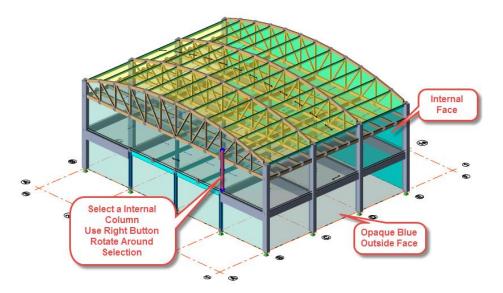
Step 3. Delete the Loadcase Wind in X.

21.6 Automated Wind Loading

21.6.1 The Wind Wizard

TSD has inbuilt an automated wind loading component, which can be used to apply lateral loading on your structure. Before this can be used, the structure need to be encased in decomposition Roof and Wall panels.

- Step 1. Place Wall Panels on the remaining vertical faces of the structure. Ensure that the Outward face orientation is facing you.
- Step 2. Check the panel orientation by rotating the model using the right mouse button.



Step 3. In the Load tab, pick the Wind Wizard command.

File	Home	Model	Edit	Load Ana	alyse	Design Repo	rt Draw	Windows				
يسلي	₽	\bowtie		1	∆ E		💥 Delete Wind		(# ? *	4		A
Loadcases	Combination	Envelope	Update Patterns	Wind Wizard	. Update Zones	Wind Loadcases	Decomposition	Seismic Wizard	Decomposition	Point Load	Line Load	Patch Load
	Structure	e	Tai .		V	Wind Load		Seismic S	Decomposition 🗔			

Step 4. Pick the following options in the wizard and press Next.

286 2		
Data Source	2	
BREVe -	UK Grid Ref.	
🔿 BREVe -	Irish Grid Ref.	
Other -	Worst Case Data	
Other -	Data for Each Direction	
Options		
Conside	r Orography	
Conside	r Tall Neighbouring Struct.	ires
Conside	r Obstructions	

Step 5. Fill in the fields as shown below for Basic Data and press Next.

Basic Data		
Site Altitude, A	121.000	m
Air Density	1.226	kg/m ³
Ground Level	0.000	m
Fundamental Basic Wind Velocity, V _{b,map}	22.5	m/s
Season factor, c _{season}	1.000	
Probability factor, c _{prob}	1.000	
Default height for internal pressure, z	11.000	m 🗸 Use Building Height

Step 6. Fill in the fields as shown below for Rougness and Obstructions and press Next.

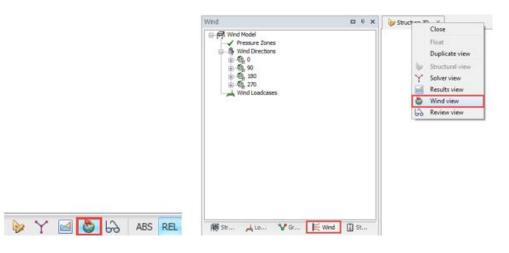
Roughness & Obstructions		
Terrain Category	Town	•
Average height of upwind buildings (h _{ave})	10.000	m
Upwind spacing of surrounding buildings (x)	20.000	m
Upwind distance from sea to site	200.0	km
Upwind distance from edge of town to site	5.5	km

Step 7. Press the Details... button to report the wind data then press Finish.

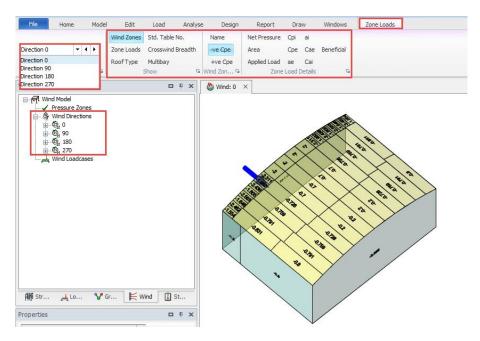
eak Velocity Pressu	res					
Results						
Direction [°]	c _a ,	v _p Max [m/s]	q _o Max [kN/m²]	-		
0.0000	1.000	35.6	0.775			
90.0000	1.000	35.6	0.775		Site Details	
180.0000	1.000	35.6	0.775	Site Details	Altitude	121.000 m
270.0000	1.000	35.6	0.775	Intermediate Factors	Air Density	1.226 kg/m ³
				⊕ 🔁 Building Directions	Site Ground Level	0.000 m
					Tall neighbouring buildings not considered	
					Orography not considered	
					Shelter effect from obstructions is not induded	
					Basic wind speed, v _{b.map}	22.5 m/s
					Probability Factor, c _{ereb}	1.000
			Add Dir.		Seasonal Factor, c _{season}	1.000
					Default Heightfor Internal Pressure, z,	11.000 m
			Del Dir.		Average height of roof tops of upwindbuildings, h	10.000 m
			Sort Dirs.		Upwind spacing of surrounding buildings, x	20.000 m
			-		Upwind distance from sea to site	200.0 km
			Details		Upwind distance from edge of town to site	5.5 km

21.6.2 The Wind View

- Step 1. Activate the Wind View mode by selecting the Icon in the lower right or right click the Structure 3D tab.
- Step 2. Select the Wind tab in the Project Workspace.



Step 3. Wind information can be displayed via the options and the ribbon and other directions can be viewed vied the drop down option or double clicking the direction in the project window.



21.6.3 Wind Loadcases

File	Home	Model	Edit	Load Analy	yse	Design Repo	rt Draw	Windows	Zone Loads		
يلي	₽			Ē			💥 Delete Wind	-Alfan-	Æ	4	
oadcases	Combination	Envelope	Update Patterns	Wind Wizard	Update Zones	Wind Loadcases	Decomposition	Seismic Wizard	Decomposition	Point Load	Li

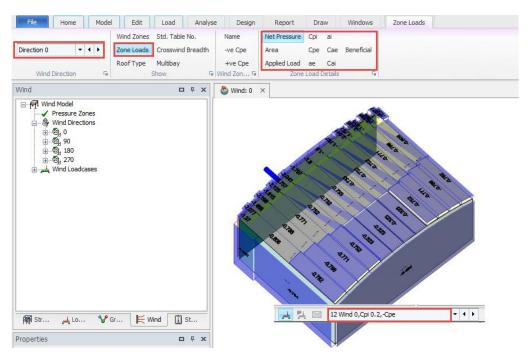
Step 1. In the Load tab and click the Wind Loadcases command.

Step 2. In the Wind Loadcases dialogue press the Auto button and press OK.

#	Name	Direction		Overall	b+h [m]	Use +ve C _{pe}	C _{pi}	^	OK
9	Wind 0,Cpi -0.3,+Cpe	0	~		5.000	~	-0.300		Cancel
10	Wind 0,Cpi -0.3,-Cpe	0	~		5.000		-0.300		
11	Wind 0,Cpi 0.2,+Cpe	0	~		5.000	~	0.200		
12	Wind 0,Cpi 0.2,-Cpe	0	¥		5.000		0.200		Add
13	Wind 0,-Cpe, All	0	~	~			0.000		Delete
14	Wind 90,Cpi -0.3,+Cpe	90	~		5.000	-	-0.300		Auto
15	Wind 90,Cpi -0.3,-Cpe	90	~		5.000		-0.300		-
16	Wind 90,Cpi 0.2,+Cpe	90	¥		5.000	~	0.200		
17	Wind 90,Cpi 0.2,-Cpe	90	¥		5.000		0.200		
18	Wind 90,-Cpe, All	90	~	~			0.000		
19	Wind 180,Cpi -0.3,+Cpe	180	~		5.000	-	-0.300		
20	Wind 180,Cpi -0.3,-Cpe	180	~		5.000		-0.300	~	
	tructural Factor - Automatically	ala data anan							

21.6.4 Reviewing the Decomposed Wind Loads

Step 1. Switch back to the Zone Loads tab and select Zone Loads option, and select a Wind Loadcase from the Loading List option.



21.6.5 Generating Combinations

Once the wind load cases have been established the lateral combinations can be created via the generator.

- Step 1. In the Load tab, click the Combination command.
- Step 2. In the Combinations dialogue, delete all previous created combinations.
- Step 3. Press the Generate... button.
- Step 4. Follow the steps through the generator and establish the EHF and Wind Combinations.
- Step 5. Only check the first combination for the EHFs and press Finish.

mbination Generator	Combinations						
nitial Parameters	Combinations						
Combinations for design of structural members (STR)	Combination	Generate	Dead	Imposed	Wind	Roof Imposed	
	STR1-1.35G+1.545Q+1.545RQ	•	1.350	1.500		1.500	
O Table A1.2(B) - Eq 6.10	STR3-1.35&G+1.5Q+1.5RQ	-	1.249	1.500		1.500	
Table A1.2(B) - Eq 6.10a8b	STR_3-1.35G+1.5\u03c6_0Q+1.5\u03c6_0RQ+EHF	-	1.350	1.500		1.500	
Indude	STR7+1.355G+1.5Q+1.5RQ+EHF	-	1.249	1.500		1.500	
GEO combinations - Table A1.2(C) - Eq 6.10	STR15-1.35G+1.542Q+1.542S+1.546W	•	1.350	1.500	1.500		
Accidental combinations - Table A2.5 - Eq.6.11a8b	STR ₁₁ -1.35&G+1.5Q+1.5V ₀ S+1.5V ₀ W+	•	1.249	1.500	1.500		
Seismic combinations - Table A2.5 - Eq 6.12a8b	STR ₁₃ -1.35¢G+1.5ψ ₀ Q+1.5ψ ₀ S+1.5W+	•	1.249	1.500	1.500		

Service

Wind/EHF Directions

Combination	Generate	Dead	Imposed	Wind	Roof
5TR:-1.35G+1.5v5Q+1.5v5RQ	4	1.000	1.000		1.000
STR ₈ -1.35&G+1.5Q+1.5RQ	9				
STRs-1.35G+1.5wpQ+1.5wpRQ+EHF	1	1.000	1.000		1.000
STR-1.35&G+1.5Q+1.5RQ+EHF	(4)				
STR ₄₀ -1.35G+1.5ψ ₂ Q+1.5ψ ₅ S+1.5ψ ₅ W+	4	1.000	1.000	0.500	
STR.,-1.35&G+1.5Q+1.5v.S+1.5v.W+1	1				1
STR:r1.352G+1.5v;Q+1.5v;S+1.5W+6	4	1.000	1.000	1.000	

Wind Lc	Dir1+	Fac _{oria}	Dir1-	Facoro	Dir2+	Fac _{orb}	Dir2-	Facore	^
9 Wind 0,Cpi -0.3,+Cpe		1.000		1.000		1.000	•	1.000	
10 Wind 0,Cpi-0.3,-Cpe		1.000		1.000		1.000		1.000	
11 Wind 0,Cpi 0.2,+Cpe		1.000		1.000		1.000		1.000	1
12 Wind 0,Cpi 0.2,-Cpe		1.000		1.000		1.000		1,000	
13 Wind 0,-Cpe, All		1.000		1.000		1.000		1.000	
14 Wind 90,Cpi -0.3,+Cpe		1.000		1.000		1.000		1.000	
15 Wind 90,Cpi -0.3,-Cpe		1.000		1.000		1.000		1.000	
16 Wind 90 Col 0.7 +Coe	17	1.000	0	1.000		1.000	1.17	1.000	Y

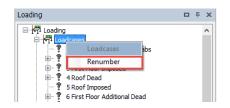
Bold text - relevant y factors will additionally be applied to load cases of this type in the combinat



Please note that all combinations must include EHF forces in the design.

#	Design Combination Title	Class		Active	Strength	Service	Ca
.35G+1.5\v0Q+1.5\v0F .35\c025G+1.5Q+1.5RQ 11	STR1-1.35G+1.5\u00fc0Q+1.5\u00fc0RQ	Gravity	¥	~	✓	✓	
1.5ψ₀Q+1.5ψ 12	STR ₃ -1.35&G+1.5Q+1.5RQ	Gravity	~	-	-		_
1.5\u03c6_0Q+1.5\u03c6 1.5\u03c6_0P+1.5\u03c6 13	$STR_{10.1}-1.35G+1.5\psi_0Q+1.5\psi_0S+1.5\psi_0W+EHF_{Dir1+}$	Lateral	~	~	~	•	4
G+1.5ψ ₀ Q+1.5ψ 14	STR _{10.2} -1.35G+1.5\u03c6_Q+1.5\u03c6_OS+1.5\u03c6_OW+EHF _{DIR1} -	Lateral	~	-	~	•	C
G+1.5Q+1.5\v0 15	STR103-1.35G+1.5\000000000000000000000000000000000000	Lateral	~	~	~	•	De
G+1.5Q+1.5ψ ₀ G+1.5Q+1.5ψ ₀ 16	$STR_{10,4}$ 1.35G+1.5 ψ_0 Q+1.5 ψ_0 S+1.5 ψ_0 W+EHF _{DH2} .	Lateral	~	~	~	•	
Q+1.5ψ ₀ 17	STR ₁₁₁ -1.35ξG+1.5Q+1.5ψ ₀ S+1.5ψ ₀ W+EHF _{DW1+}	Lateral	~	~	~		
5\u03c0_0Q+1.5 5\u03c0_0Q+1.5	STR _{11.2} -1.35ξG+1.5Q+1.5ψ ₀ S+1.5ψ ₀ W+EHF _{DIP1} .	Lateral	~	~	~		
.5ψ ₀ Q+1.5 5ψ ₀ Q+1.5 19	STR ₁₁₃ -1.35ξG+1.5Q+1.5ψ ₀ S+1.5ψ ₀ W+EHF _{DIF2+}	Lateral	~	~	~		
5&G+1.5\v0Q+1.5 20	STR _{11.4} -1.35ξG+1.5Q+1.5ψ ₀ S+1.5ψ ₀ W+EHF _{DH2} -	Lateral	~	~	~		
21	STR _{13.1} -1.35&G+1.5\v0Q+1.5\v0S+1.5W+EHF _{DW1+}	Lateral	~	-	~	•	
22	${\rm STR}_{13.2}{\rm -}1.35\xi{\rm G}{\rm +}1.5\psi_0{\rm Q}{\rm +}1.5\psi_0{\rm S}{\rm +}1.5{\rm W}{\rm +}{\rm EHF}_{\rm Dirl}.$	Lateral	~	~	~	•	
23	$STR_{13.3}\text{-}1.35\xi\text{G}\text{+}1.5\psi_0\text{Q}\text{+}1.5\psi_0\text{S}\text{+}1.5\text{W}\text{+}\text{EHF}_{\text{Dir}2\text{+}}$	Lateral	~	-	~	•	
24	STR _{13.4} 1.35ξG+1.5ψ ₀ Q+1.5ψ ₀ S+1.5W+EHF _{DIG2}	Lateral	~	~	~	~	

Step 6. Renumber the loadcases and combinations.

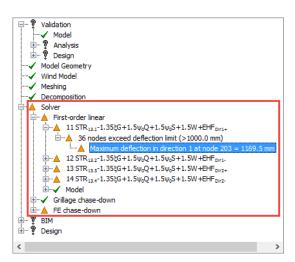


Step 7. Perform the Design All (Static) command and then check the followings:

a. Design process – Show Process window

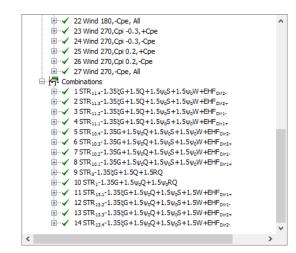
Time			Message	Duration
⊿	4 10:01:37		Welcome to Tekla Structural Designer	
⊿	-	10:02:17	Steel & Concrete Design (All)	00:19
	⊳	10:02:17	3D pre-Analysis	00:03
	\triangleright	✓ 10:02:20	3D analysis: First-order linear	Less than 1s
	\triangleright	✓ 10:02:21	Grillage chase-down	Less than 1s
		✓ 10:02:21	Calculating chase-down combinations	Less than 1s
	\triangleright	✓ 10:02:22	FE chase-down	00:01
		✓ 10:02:24	Calculating chase-down combinations	Less than 1s
		10:02:25	Running design of 4 concrete element groups	00:04
		✓ 10:02:29	Running check of 20 elements	00:02
		10:02:32	Running check of 31 elements	00:01

b. Design status – Status tab in Project Workspace



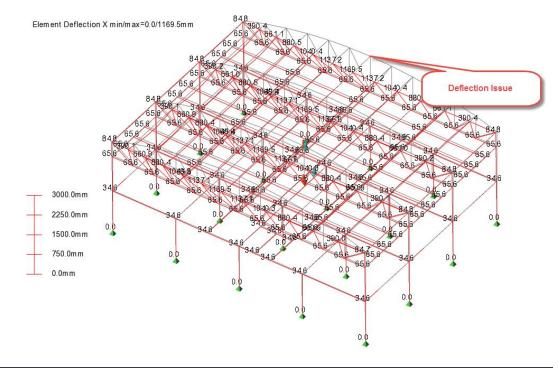


The solver is indicating a large deflection in the model.



c. Loading Summary – Loading tab in Project Workspace

d. Deflections - Results View



The structure clearly has lateral issues in the perpendicular direction to the truss try and stabilise the structure by adding other structural elements (Bracings, Walls, Etc.) or by altering the fixity of the structure.



The status of the project shows that the design is indicating a "Pass" but due to the warnings in the analysis – the design must be viewed with extreme caution.

22 Output

22.1 Predefined Reports

TSD has inbuilt predefined standard reports which can be viewed for your structure.

- Step 1. Select the Report tab on the ribbon.
- Step 2. On the Select option, click on the drop down list options to see the available reports.
- Step 3. Highlight a report and click the Show Report command.

File	Home	Model	Edit	Load	Analys	e De	sign R	Report	Draw	Windows	
e	Solver Mode Solver Mode Building Loar	Data	•	Model Report.	Membe . Report.		F	rames	Loadcases Combinations Members Filters	Design Groups Trusses Portal Frames	
itus N I VaE	Building Des Material List Beam End Fi Bracing Ford	ign ing prces	•			- {		> Struct	ure 3D	🧞 St. Base (Base)	2D
ucture inforced	Concrete B										
ucture inforced		eams Section Size	Grade	No. off	Length [m]	Mass [kg]	Surface A		olume (m ¹)		
inforced Section G	Concrete B Geometry		Grade C32/40	No. off			Surface Ar [m ²] 60.5		olume [m ¹] 6.0		
inforced Section G	Concrete B	Section Size			[m]	[kg]	[m²]		[m"]		
ucture einforced Section G	Concrete B Geometry	Section Size	C32/40	6	[m] 5.600	[kg] 15120.00	[m ²] 60.5		[m ¹] 6.0		
ucture einforced Section G	Concrete B Geometry	Section Size 300x600 300x600	C32/40 C32/40	6	[m] 5.600 6.000	[kg] 15120.00 16200.00	[m ²] 60.5 64.8		(m ¹) 6.0 6.5		
einforced Section G Recter	Concrete B Geometry	Section Size 300x600 300x600 400x900 Total	C32/40 C32/40	6 6 8	[m] 5.600 6.000	[kg] 15120.00 16200.00 41040.00	[m ²] 60.5 64.8 118.6		(m ¹) 6.0 6.5 16.4		
einforced Rector einforced	Concrete B Seometry nguisr	Section Size 300x600 300x600 400x900 Total	C32/40 C32/40	6 6 8	[m] 5.600 6.000	[kg] 15120.00 16200.00 41040.00	[m ²] 60.5 64.8 118.6	rea Vo	(m ¹) 6.0 6.5 16.4		
ucture einforced Section G Rectar einforced Section G	Concrete B Geometry Ingular	Section Size 300x600 300x600 400x900 Total olumns	C32/40 C32/40 C32/40	6 6 8 20	[m] 5.600 5.700 5.700	[kg] 15120.00 16200.00 41040.00 72360.00 Mass	[m²] 60.5 64.8 118.6 243.8 Surface A	rea Vo	[m ¹] 6.0 6.5 16.4 28.9		
ucture einforced Section G Rectar einforced Section G	Concrete B Geometry Ingular Concrete C Geometry	Section Size 300x600 300x600 400x900 Total olumns Section Size	C32/40 C32/40 C32/40 Grade	6 6 8 20 No. off	[m] 5.600 5.700 5.700 Length [m]	[kg] 15120.00 16200.00 41040.00 72360.00 Mass [kg]	[m ²] 60.5 64.8 118.6 243.8 Surface A [m ²]	rea Vo	(m ¹) 6.0 6.3 16.4 28.9 olume (m ¹)		
Section G Rectar einforced Section G	Concrete B Geometry Ingular Concrete C Geometry	Section Size 300x600 300x600 400x900 Total olumns Section Size 400x600	C32/40 C32/40 C32/40 Grade C32/40	6 6 8 20 No. off 8	[m] 5.600 5.700 5.700 Length [m] 1.500	[kg] 15120.00 16200.00 41040.00 72360.00 Mass [kg] 7200.00	[m ²] 60.5 64.8 118.6 243.8 Surface Au [m ²] 24.0	rea Vo	(m ¹) 6.0 6.5 16.4 28.9 olume (m ¹) 2.9		



The report above has been generated by adding individual components.

- 22.2 Individual Report Components Model Reports
 - Step 1. Click the Model Report command which contains overall model information.

	Chapters and Options: Drag chapters and options from left to right, which are to be included in the report.	Report Structure: Drag selected chapters up and down, to change order.	ОК
Available Styles		Picture (Structure (solver), None)	Cancel
Solver Model Data (active) Building Loading Building Analysis Checks Building Design Material Listing	Analysis Concrete Steel Timber Click a	Analysis (Structure)	View Mode
Materia Listing Jeam End Forces Bracing Forces Foundation Reactions Seismic Design Member Design Calos	Cold Formed Cold Rolled Cold Rolled General Material Frage Brang Forces		Hierarchical
Add Remove >> Active	Foundation Reactions Foundation Reactions Analysis Diagram Material Listing Fare Revision History		
Active Style	Repor		

Generate the following report:

Step 2. Click the Add button and create a report name called Sway, Drift & Storey Shears and press the >>Active button to set as default.



Step 3. In the Chapters and Options, left click on the Picture component and drag it into the Report Structure.

Available Styles	Chapters and Options: Drag chapters and options from left to right, which are to be induded in the report.	Report Structure: Drag selected chapters up and down, to change order.	OK Cancel
Solver Model Data Building Loading Building Analysis Checks Building Design Material Listing Beam End Forces Bracing Forces Foundation Reactions Seismic Design Member Design Calos Sway, Drift & Storey Shears (active)	Picture to th	ck and drag e Report	View Mode O Flat () Hierarchica
Add Remove >> Active	Material Listing		

22.2.1 Report Component Control

The Picture component just added to the report structure can be Filtered to specify the required view and any specific loading information.

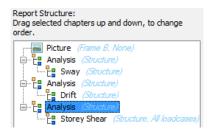
Step 4. Right click on the Picture and under Model Filter select Edit/New...

pters up and down, to change			ОК	
		, 🗌	Cancel	
Active				
Model Filter	•	~	Structure	
Loading Filter	•		Structure (solver)	
Remove Item			Edit\New	
	Active Model Filter Loading Filter	Active Model Filter Loading Filter	Active Model Filter Loading Filter	

Step 5. Press the Add button and highlight the Frame type.Step 6. Tick FRM B item, then change the name to Frame B then press OK.

Select filter		
Available filters	Filter properties	
Structure Structure (solver) Frame B Level	Type Level Frame Structure (solver) Level (solver) Frame (solver)	
Add Remove Active filter Name Frame 8	Selected items Selected items FRM B FRM C FRM D FRM E	Report Structure: Drag selected chapters up and down, to change order.

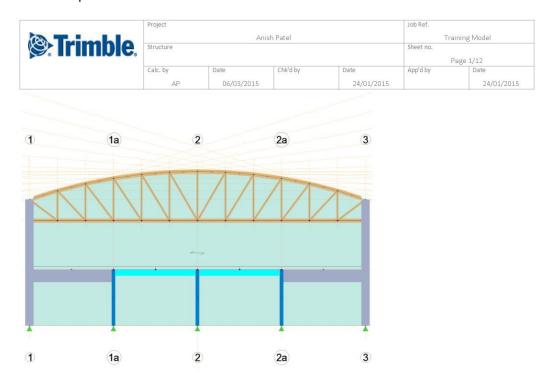
Step 7. In the Chapters and Options, expand the Analysis option and add the Sway, Drift and Storey Shears component to the Report Structure – Ensure the filter applied is for the whole structure and press OK.



Step 8. Press the Show Report command to display the report on the screen.

File	Home	Model	Edit	Load	Analyse	Design	Report	Draw	Windows
							Levels	Loadcases	Design Groups
51		er el	rs 🔻	1 4					
🔁 Select	Sway, Drift &	Storey Shear	rs 🔻	Madal	Mamban	Chau Dapart	Frames	Combinations	Trusses
韵 Select	Sway, Drift &	Storey Shear	rs 🔻	Model Report	Member Report	Show Report	Frames Planes	Combinations Members	Trusses Portal Frames

Created Report:



Trimble.	Project		Job Ref.			
		Ani	Training Model Sheet no. Page 2/12			
	Structure					
	Calc. by	Date	Chk'd by	Date	App'd by	Date
	AP	06/03/2015		24/01/2015		

Analysis

Sway

First-order linear

Ref.	Combination Dir 1	Stack Dir 1	α _{Dir 1}	Combination Dir 2	Stack Dir 2	α _{Dir 2}	Combination Dir 1/2	Twist	Status
G+ S+	1 STR _{11.4} -1.35ξ G+1.5Q+1.5Ψ ₀ S+1.5Ψ ₀ W+EHF _{Dir2} -	1	4.588	1 STR _{11.4} -1.35ξ G+1.5Q+1.5Ψ ₀ S+1.5Ψ ₀ W+EHF _{Dir2} -	1	13.454	$5 \text{ STR}_{10.4}$ -1.35G+1.5 ψ_0 Q+1.5 ψ_0 S+1.5 ψ_0 W+EHF _{Dir2} -	1.070	✓ Pass
C17	1 STR _{11.4} -1.35ξ G+1.5Q+1.5Ψ ₀ S+1.5Ψ ₀ W+EHF _{Dir2} -	1	4.561	1 STR _{11.4} -1.35ξ G+1.5Q+1.5Ψ ₀ S+1.5Ψ ₀ W+EHF _{Dir2} -	1	13.454	$5 \text{ STR}_{10.4}$ -1.35G+1.5 ψ_0 Q+1.5 ψ_0 S+1.5 ψ_0 W+EHF _{Dir2} -	1.070	✓ Pass
SC E/2a	1 STR _{11.4} -1.35ξ G+1.5Q+1.5Ψ ₀ S+1.5Ψ ₀ W+EHF _{Dir2} .	1	4.581	1 STR _{11.4} -1.35ξ G+1.5Q+1.5Ψ ₀ S+1.5Ψ ₀ W+EHF _{Dir2} -	1	13.454	$5 \text{ STR}_{10.4}$ -1.35G+1.5 ψ_0 Q+1.5 ψ_0 S+1.5 ψ_0 W+EHF _{Dir2} -	1.003	V Pass

Analysis

Drift

Ref.	Combination Dir 1	Stack Dir 1	Ratio Dir 1	Combination Dir 2	Stack Dir 2	Ratio Dir 2	Status
C1	1 STR _{11.4} -1.35ξG+1.5Q+1.5 ψ ₀ S+1.5ψ ₀ W+EHF _{Dir2} .	1	1.281	1 STR _{11.4} -1.35ξG+1.5Q+1.5 ψ ₀ S+1.5ψ ₀ W+EHF _{Dir2} .	1	1.099	V Pass
C7	$1 \text{ STR}_{11.4}$ -1.35 ξ G+1.5Q+1.5 ψ_0 S+1.5 ψ_0 W+EHF _{Dir2} -	1	1.281	$1 \text{ STR}_{11.4}$ - $1.35\xi\text{G}$ + 1.5Q + 1.5 $\psi_0\text{S}$ + $1.5\psi_0\text{W}$ + EHF_{Dir2} -	1	1.092	V Pass
C12	$1 \text{ STR}_{11.4}$ -1.35 ξ G+1.5Q+1.5 ψ_0 S+1.5 ψ_0 W+EHF _{Dir2} -	1	1.281	$1 \text{ STR}_{11.4}$ - 1.35ξ G+ 1.5 Q+ 1.5 ψ_0 S+ $1.5\psi_0$ W+EHF _{Dir2} -	1	1.086	V Pass
C17	1 STR _{11.4} -1.35ξG+1.5Q+1.5 ψ ₀ S+1.5ψ ₀ W+EHF _{Dir2} .	1	1.281	$\begin{array}{l} 1 \ \text{STR}_{11.4}\text{-}1.35 \xi \text{G}\text{+}1.5 \text{Q}\text{+}1.5 \\ \psi_0 \text{S}\text{+}1.5 \psi_0 \text{W}\text{+}\text{EHF}_{\text{Dir2}\text{-}} \end{array}$	1	1.080	V Pass

Storey Shears:

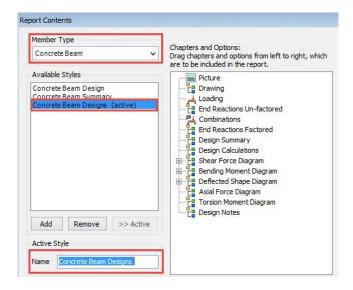
24 Wind 270,Cpi -0.3,-Cpe

Reference	Level	Σ Shear Major	$\boldsymbol{\Sigma}$ Shear Minor
	[m]	[kN]	[kN]
St. ()	11.000	0.0	0.0
St. ()	10.946	0.0	0.0
St. ()	10.783	0.0	0.0
St. ()	10.510	0.0	0.0
St. ()	10.125	0.0	0.0
St. ()	9.623	0.0	0.0
St. 3 (Eaves)	9.000	0.0	-77.2
St. 2 (Truss Bottom)	7.500	0.0	146.9
St. 1 (First Floor)	4.000	0.0	54.1
St. Base (Base)	0.000	0.0	-118.6

22.3 Structural Element Reports

Establishing the report contents for an individual concrete beam are found in the Member Reports command.

Step 1. Click the Member Reports command and select the Member Type to be Concrete Beam. Step 2. Click Add button to create a new report and set it as Active.



Step 3. Pick the following components to be included in the report – note each component will have its own filter which can be adjusted and press OK.

	Structure: ected chapters up and down, to change
	Drawing Design Summary
	Design Calculations
E	Design Notes

22.3.1 Quick Filter Report Options

Step 4. In the Report tab pick the Levels filter option.

File	Home	Model	Edit	Load	Analyse	Design	Report	Draw	Windows
					1	P	Levels	Loadcases	Design Groups
B Select	Sway, Drift &	Storey Shears	•	Model	Member	Show Report	Frames	Combinations	Trusses
				Report	Report	Show Report	Planes	Members	Portal Frames
		C	ontents			Fai		Filters	Fai

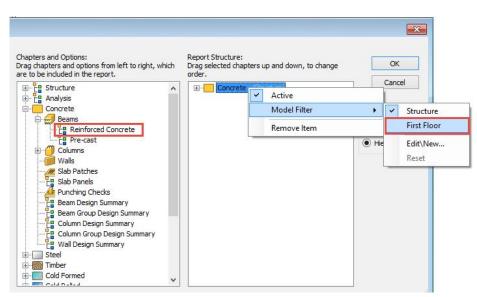
Step 5. Add a First Floor filter and give the name of First Floor - press OK.

vailable filters	Filter properties	ОК
irst Floor	Name First Floor	
	Selected items	Cancel
	St. Base (Base)	
	St. 3 (Eaves)	
	St. 2 (Truss Bottom)	
	St. 1 (First Floor)	
	St. 0	

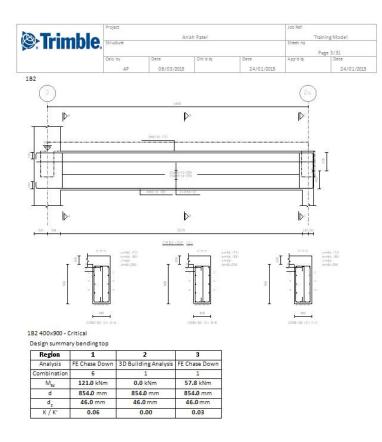
Step 6. Click the Model Report command and Add a new report called Concrete Beams and set to Active.

	-	Turner
Add	Remove	>>1

Step 7. Add the Reinforced Concrete component to the Report Structure. Step 8. Add the predefined filter of First Floor and press OK.



Step 9. Press the Show Report command to display the report on the screen.



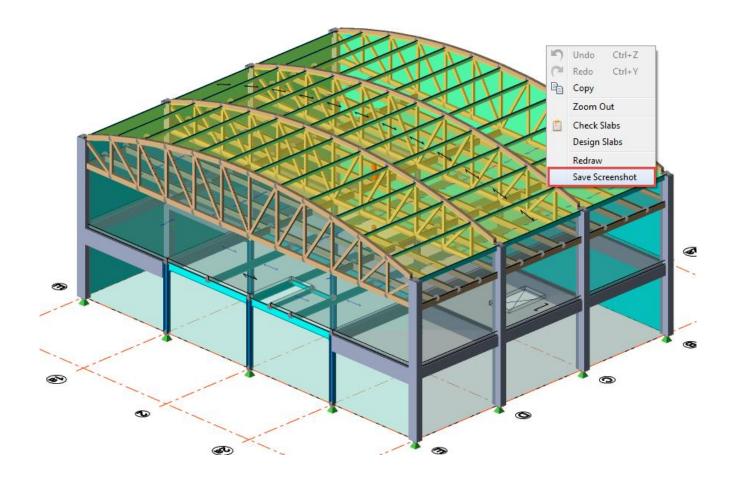
22.4 Exporting Reports

Reports can be exported to TEDDS, Excel, PDF, and Word – click the appropriate export button when viewing a report.

				•				A	W	X	
Page Setup	Edit Header	Edit Footer	First Page	Previous Page	Next Page	Last Page	Report Index	PDF	Word	Excel	Tedds
arance		Gi.	3-		Navigation		Es.		Ехр	ort	r _a

22.5 Saving Pictures

Any graphical view can be saved to a picture file by setting up the view required – right click anywhere on the screen and pick the option Save Screenshot.



22.6 Engineering Drawings

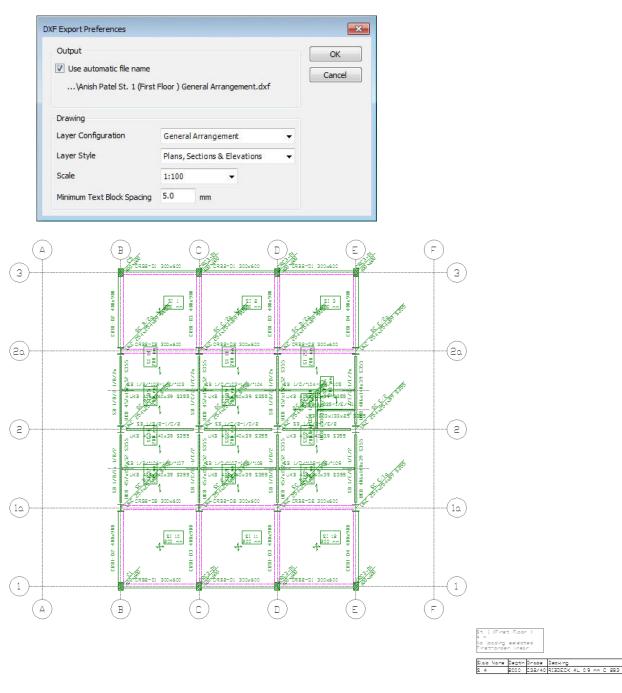
Drawings are generally created from the Draw tab. A wide range of drawing types and styles are available which can be customised to meet your specific requirements. Single member drawings can also be created from the right click context menu.

File		Home Mod	el	Edit Loa	d Analyse	Design	Foundations	Report	Draw	Windows	5		
1	24	ł		ł				甲				#	3-
Edit		Drawing Management		Schedule Management	Beam Schedule	Column Schedule	Wall Schedule	General Arrangement	Beam End Forces	Foundation Reactions	Loading Plan	Slab Detailing	Foundation Layout
Settings 5	M	anaged Drawings	5		Schedules		Fai			Drawin	ngs		La La

- Edit Drawing Settings Menu
- Drawing Management Opens a dialog for the generation and laying out of multiple drawings on to a single drawing sheet. The dialog can also be used to manage drawing revisions and history.
- General Arrangement Create multi material plans, sections and elevations
- Slab Detailing Generate a reinforcement slab detail drawing
- Beam Schedule Beam Reinforcement Engineering Drawings
- Column Schedule Column Reinforcement Engineering Drawings
- Wall Schedule Wall Reinforcement Engineering Drawings

22.6.1 Creating 2D Multi Material Drawing

- Step 1. Obtain a 2D plan view of the First Floor.
- Step 2. In the Draw tab, click the General Arrangement command.
- Step 3. Accept the defaults in the pop up dialogue.



22.6.2 Creating 2D End Reaction Drawing

Step 1. Obtain a 2D plan view of the First Floor. Step 2. Switch to the Results View.



Step 3. Select a Loadcase / Combination.

✓ Show Process 📕 🖳 🖂 1 STR₁₁₄-1.35ξG+1.5Q+1.5ψ₀S+1.5ψ₀W+ ▼ ◀ ►

Step 4. In the Draw tab, click the General Arrangement command.Step 5. Select the drawing type to be Beam End Forces and 1:50 scale - press OK.

DXF Export Preferences		
Output Output Ise automatic file name C:\\Anish Patel St. 1 (l)	OK Cancel First Floor) Beam End Forces.dxf	
Drawing Layer Configuration Layer Style Scale Minimum Text Block Spacing	Beam End Forces	
NA 1691	SB 1/B/2-1/C/2	
NA 1.631 MAN 0.0	SI 16 200 mm	
'B/1a−1/B/2	SB 1/B/°106-1/C/°107	

23 Analysis and Design of Concrete Structures

In this session, we will look at the analysis and design of concrete structures.

Step 1. Open the model TSD Concrete Design Fundamentals Model 3 - Element Design.tsmd.

23.1 Analysis and Design Options

As mentioned earlier, before any analyses or designs are undertaken, all of the analysis and design options should be checked thoroughly to ensure that elements are designed as required.

- Step 2. Go to the Analyse tab and click the Options command to review the Analysis Options, then click OK to confirm them.
- Step 3. Go to the Design tab and click the Options command to review the Design Options, then click OK to confirm them.

23.2 Analysis and Design Procedure

TSD can run multiple analyses on the model and then design all concrete frame elements (excluding slabs) for all gravity and lateral loads in one go. This is done by clicking the Design Concrete (Static) button on the Design tab.



There is also a Design Concrete (Gravity) button available, which will only deal with the vertical loads.

This process goes through a series of steps, in part controlled by the Design Options, with the key stages mentioned below.



You can monitor the progress of the design process by clicking the Show Process button in the bottom left corner of the screen before clicking a Design command.

Elements designed during this process will be designed for the worst case results from all analyses completed, based on the element's Autodesign properties, and the Design Group that it belongs to.

 Validation – the purpose of validation is to trap out errors and potential problems with the model that will likely either cause the analysis or design process to fail, or produce potentially unexpected results. If any issues are found, they will be reported as either warnings or errors, in the Status tree. Errors MUST be corrected to allow the analysis and design procedure to be completed, and will be highlighted with a red cross. The analysis and design process should complete if warnings are present, but they should be reviewed and corrected if deems necessary.



You can double click on the warning and error messages in the status tree to locate the issues in an appropriate view of the model.

The validation process will also report some information for points of interest, and stages that fully pass will be highlighted with green ticks.

- Load Decomposition slab loads are decomposed into the frame members using FE analysis.
- 3D Analysis this will analyse the whole model at once, following the analysis and design options, and can either treat the slabs as diaphragms (default), or mesh the slabs.
- Grillage Chasedown this will complete a chasedown of load using the sub-models of each floor, from top to bottom, without the slabs being meshed.
- FE Chasedown this will complete a chasedown of load using the sub-models of each floor, from top to bottom, meshing the slabs at every level.
- Design of all frame elements once the structure is fully analysed, TSD will perform either a full Auto Design or a Check Design of all frame elements in the model, based on their Autodesign property status.



The default for concrete elements is always set as Auto Design.

23.2.1 Auto Design and Check Design

When TSD performs a design, it will either do an Auto Design or a Check Design, depending on its Autodesign property status, found in the Properties window for that element.

An Auto Design will complete a new full design based on the newly updated results. There are two types of autodesign available; either Minima or Current, also specified in the element's Properties window.

Properties	口 무 :
<unsaved set=""></unsaved>	▼ Save Apply
Concrete dass	C40
Autodesign	V
Select bars starting from	Minima
Section	Minima
Automatic alignment	Current
Markey Barristowski	

- Minima option removes any current bar arrangement and starts the design from the minimum allowable bar sizes.
- Current option starts from the current bar arrangement.

A Check Design will check the element's already existing design against the updated analysis results and will report a pass or fail.

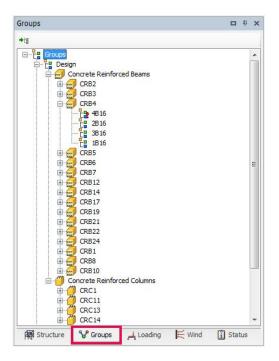
23.2.2 Design Groups

To allow for a quick and efficient design process, all concrete beams and columns are automatically placed into Design Groups.

Design Groups have no specific engineering data associated with them and are simply a unique name for a collection of similar frame elements. They are formed automatically based on span lengths, number of spans, section sizes and various other criteria.

The result of using design groups means that every element within the group will have the same reinforcement, and provided they can all be designed successfully, they will all work for all results of all the different analyses completed.

The Design Groups can be viewed using the Groups tab in the project workspace. This window allows you to do a variety of things, including editing the groups and re-grouping the elements.



The topic of Design Groups will be discussed in more detail in a later exercise.

You will probably notice that there are also Detailing Groups listed here, which relate to the generation of detail drawings. These too will be discussed in more detail later.

23.2.3 Complete the Analysis and Design

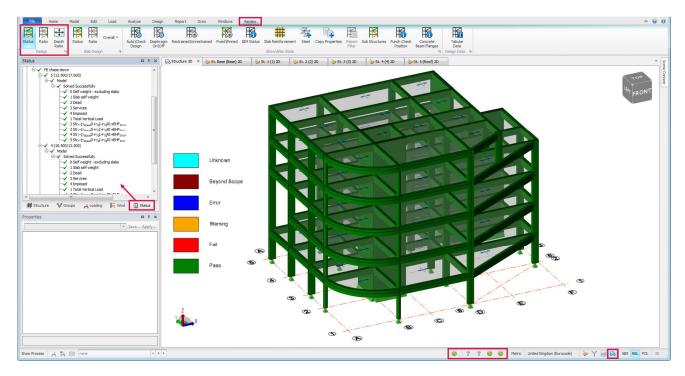
Having reviewed the Analysis and Design Options and understanding the Autodesign and Design Group functionality, the analysis of the model and design of all frame elements can now be completed. Note that slabs are designed using a separate process that will be covered later in this course.

- Step 1. Click the Show Process button in the bottom left corner the screen to open the Process window.
- Step 2. Go to the Design tab and click the Design Concrete (Static) command.

rocess			1	= >
Time		Message	Duration	
4 1	10:54:16	Concrete Design (All)	00:19	
Þ	✓ 10:54:16	3D pre-Analysis	00:01	
\triangleright	10:54:18	3D analysis: First-order linear	Less than 1s	
\triangleright	✓ 10:54:19	Grillage chase-down	00:01	
	✓ 10:54:20	Calculating chase-down combinations	Less than 1s	
\triangleright	✓ 10:54:20	FE chase-down	00:03	Ξ
	✓ 10:54:24	Calculating chase-down combinations	Less than 1s	
	✓ 10:54:24	Running design of 23 concrete element groups	00:04	
	✓ 10:54:29	Running check of 78 elements	00:04	
Þ	✓ 10:54:33	Running design of 5 shear walls	00:01	-

23.3 Reviewing the Overall Design Status

Once the Design Concrete (Static) process is complete, the View Mode of your active scene view will automatically change to the Review View mode, and the newly-created Review tab will be activated. The initial view will show the graphical Design Status of all concrete frame elements in the model to see if they have passed or failed, or if there are any other issues with them.



23.3.1 Review tab

The Review tab contains a lot of different tools to get a good overall view of the model. Some of the key options for reviewing the concrete frame element design statuses are:

- Design Status this option shows whether the frame elements in the active scene view are passing or failing the design. It also highlights any elements that have errors or warnings against them, or if the design of an element is beyond the scope. Any undesigned elements will be shown to have an Unknown status
- Design Ratio switching to this review mode allows you to see how close to its limit an element's design is. Elements are grouped together based on their design utilisation ratios
- Depth Ratio this option colour codes all the concrete beams in the model to indicate their span to depth utilisation ratios

There are also a number of Slab Design review ribbons available on the Review tab, but these will be discussed later.

Step 1. Try switching between the options mentioned above to review the design status of the model

23.4 Reviewing the Analysis Results

Once an analysis has been completed, whether in isolation or as part of the Design Concrete (Static) procedure, the results for all of the completed analyses can be interrogated by manually switching the view mode to the Results View, and a newly-created Results tab will be activated.

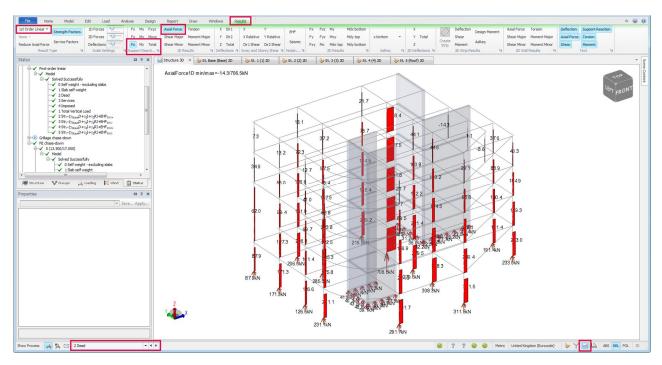
23.4.1 How to View the Results

Once in the Results View mode for a particular view, there are some settings you need to check to make sure you're displaying the results you want:

- Scene View as the Scene Mode setting is retained for each Scene View, you need to first ensure that you have the right Scene View active, and then make sure that you are in the Results View Scene Mode and have the Results tab selected.
- Analysis Result Type in the Result Type group of the Results tab, make sure you select the appropriate analysis type. Having completed the Concrete Design (All) process, the appropriate options will be 1st Order Linear, FE Chasedown and Grillage Chasedown by default.

As mentioned earlier, each of these analyses do different things so their results will vary.

Load Type – all results that are displayed are for the particular load case, combination or envelope that is selected in the Loading Drop List. If you are seeing no results after activating a particular results option from the Results tab, then you may not have a load type selected.



Once you have the view set up as required, you can view the analysis results by selecting one or more of the options on the Results tab.

Step 1. Set up your Structure 3D to display the results for Z direction axial support reactions and the axial load 1D results for the Dead loadcase, based on the 1st Order Linear analysis results.

23.4.2 Viewing Results for the Whole Model

There are a variety of different results that can be viewed by using the various options on the Results tab. These include:

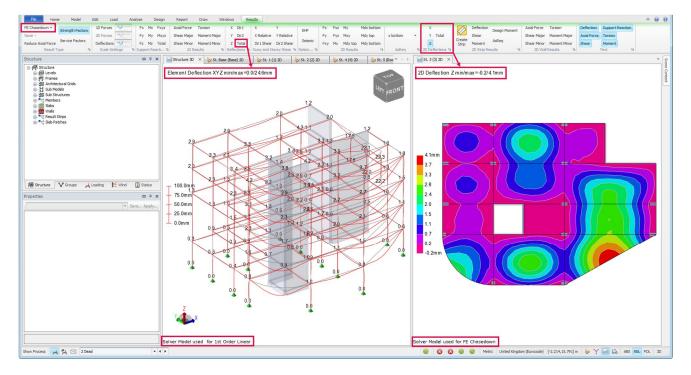
- Support Reactions view axial and moment support reactions in all 3 directions individually, combined axial or moment reactions, or total reactions.
- 1D Results displays the axial force, torsion, moments and shears in the major and minor axes for all columns and beams in the model.
- Deflections view deflections in the 3 global directions, 2 local directions, or total deflections.
- Sway and Storey Shear review the effects predominantly caused by lateral forces.
- Equivalent Horizontal Forces see the EHFs/NHFs and seismic loads applied to the model
- 2D Results these options display contours in the walls, and the slabs if you're viewing the results for the FE Chasedown analysis, and can display axial forces, moments and design moments.
- AsReq shows the area of steel required in the top and bottom of the slabs in the two directions.
- 2D Deflections displays the deflections in the walls and slabs using contours.
- 2D Wall Results shows the axial force, torsion, moments and shears in the major and minor axes for the walls, in a similar style to the 1D Results for the columns and beams.

Edit Load Analyse Design	Report Draw Windows Re	esults			
1D Forces Fx Mx Fxyz	Axial Force Torsion X Dir1	X Y EHF	Fx Fxz My Mdx bottom	x	Deflection Design Moment Axial Force Torsion
2D Forces Fy My Mxyz	Shear Major Moment Major Y Dir2		Fy Fyz Mxy Mdy top	x up - i i uai	aried Major Monerchajor
Deflections Fz Mz Total	Shear Minor Moment Minor Z Total	Dir 1 Shear Dir 2 Shear Seismic	Fxy Mx Mdx top Mdy bottom	Z Strip	Moment AsReq Shear Minor Moment Minor
Scale Settings 🕞 Support Reacti 🖼	1D Results 😼 Deflections 🖼	Sway and Storey Shear 🖼 Notion 🖼	2D Results 😼	AsReq 5 2D Deflections 5	2D Strip Results 5 2D Wall Results

Some of these results are only available, or only make sense, in certain views – i.e. 2D or 3D. You can also view multiple scene views, displaying different results at the same time, to allow comparisons.

Note that slab results can only be seen when viewing results for an analysis that includes the slabs – as default, this will just be the FE Chasedown.

Step 1. Try viewing some different results for some different scene views.



23.4.3 Slab Strips

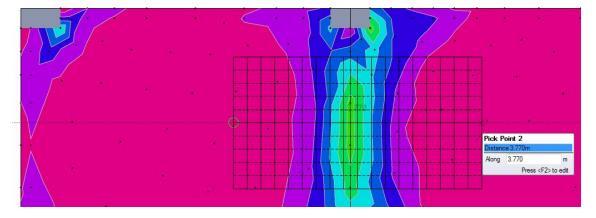
Whilst the slab design process, which will be discussed in detail later, utilises slab strips automatically and takes them much further, you may wish to cut some strips of the fly and view their results too. This can be done by clicking the Create Strip command on the Results tab.

Note that you must be in a 2D Plan View to be able to create slab strips.

Once the Create Strip command is active, the strip properties will be displayed in the Properties window.

Here, you can define the width of the strip at the start and end of the strip, the number of points and stations within the strip, as well as choose from the following Result Types:

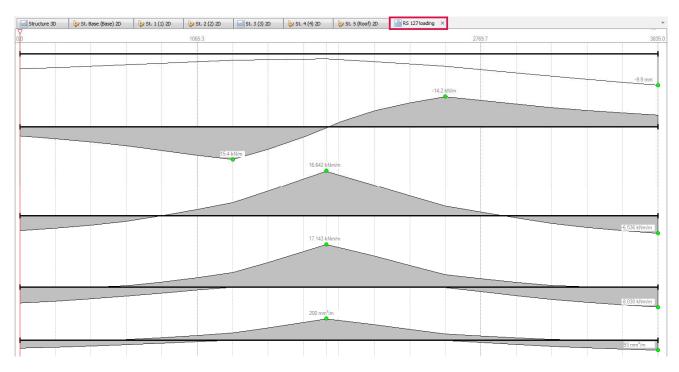
- Average the results for all points across the width of the strip are calculated using weighted averages of the shell elements local to each point. An average value is then calculated for all points across the width of the strip at that station, and the process is repeated for all stations along the length of the strip, which generates the strip results and diagrams.
- Maximum this option follows the same method as the Average option above, but uses the maximum point value across the width for the strip for each station along its length, to generate the results and diagrams.
- Centreline this option only calculates the weighted average values for the stations along the centreline of the strip, and ignores all of the points across the width of the strip.



Once you have set up the properties for the strip, it can be inserted into the plan view by first leftclicking to define the start point of the strip, then left-clicking again to define the end point.

Once the strip is created, you can view its results by selecting one of the 2D Strip Results options on the Results ribbon tab, when viewing either 2D or 3D scene views.

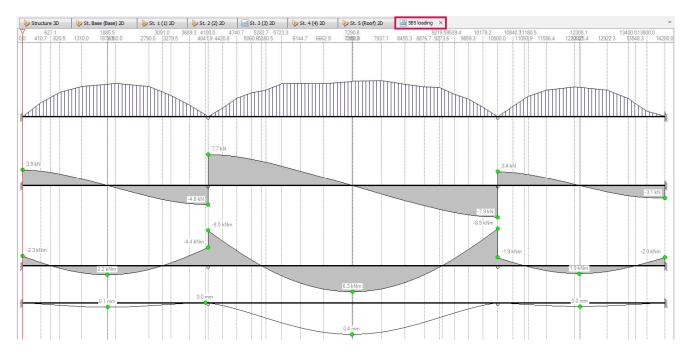
Alternatively, provided you have the Result Strips option ticked in the Scene Content, you can select the strip, right click over it, and choose Open Load Analysis View. This will create a new Scene View, displaying the relative displacement, shear, moment, design moment and the area of steel required.



Step 1. Try creating a couple of slab strips and view their results.

As well as being able to view the analysis results for the model as a whole, you can also view the results for individual frame elements – i.e. columns and beams. These element results can be accessed by selecting the element in question, right-clicking over it, and choosing Open Load Analysis View.

A new scene view will be created and a Loading Analysis tab will also automatically open. This will allow you to choose which results are displayed and which analysis they come from. The default is to display the major axis results, including loading, moment, shear and relative deflection. As with all results views, the results displayed will be based on whichever load option you have selected in the Loading drop list. The Properties window will also help control the information displayed in this view.



Step 1. Try viewing some frame element results. Step 2. Save the model.

172 (257)

24 Interactive Design of RC Columns and Walls

As you have already seen, all frame elements can designed at once by using one of the Design Concrete options on the Design tab. For a lot of the time, this will be as far as you will need to go with it. However, if you want to interrogate the designs of individual elements in more detail, or if you need to interactively design a member, then there are a variety of tools to enable you to do this.

24.1 Check Member

Once a column has been designed initially, you can do a Check Design on it by right-clicking over it in your active scene view and selecting Check Member.

A summary of the existing reinforcement placed in that column will be displayed, which will be the same reinforcement layout as all the other columns in the same design group, along with its status based on the current analysis results.

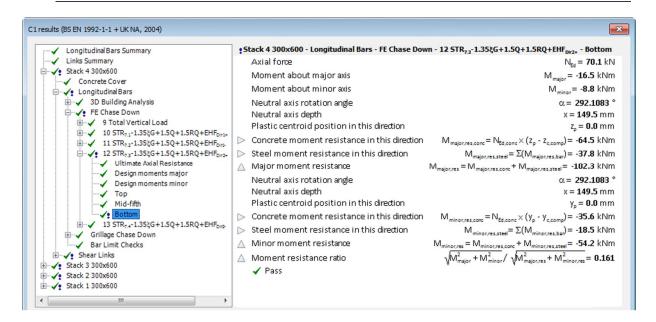
You can also view all of the design calculations and checks based on the analysis results for the specific column you're looking at by expanding the options down the left hand side.

The critical checks and results for each stack within the column being checked will be highlighted with blue exclamation marks on the left hand side of this window. The right hand pane will display the calculations and results, and clicking the arrow icons next to the text will either collapse or expand the information.

The Settings button will allow you to edit the formatting of the text in the Check Member window.

Walls can be check-designed in the exact same way, except this option is called Check Wall.

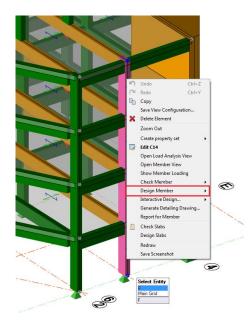
Walls are designed independently from each other and don't use Design Groups.



Make sure the Design Concrete (Static) process has been completed.
 Try using the Check commands to check a few different columns and walls.

24.2 Design Member

Along with being able to check a column's existing design, or design all frame elements in a model, you can also get TSD to design an individual column by right-clicking over it and select Design Member>static.



The Design Member command carries out an automatic design for that individual column, and then allows you to view these new design calculations and results for that member.

However, when you choose to Close this window, it applies the same reinforcement selected as part of the design to all other members in the same Design Group, and then performs a Check Design on them.

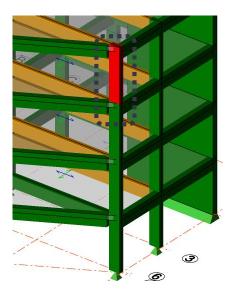


If you happen to use the Design Member command on an element that is not a critical member in its Design Group, you may find that some other members from the same group change from a pass to a fail.

- Step 1. Use the Groups tab to locate the column design group CRC11.
- Step 2. Use the Check Member command to view the current design for column non-critical column C14, which is located at the gridline intersection (F/3).
- Step 3. Close this window and use the Design Member command to redesign this member based on its own analysis results

Stack	Section	Longitudinal Bars	Analysis	Combination	Critical position	Ratio	Status
4	300x600	10H16	Grillage Chase Down	10	Тор	0.957	🖌 Pass
3	300x600	10H16	3D Building Analysis	13	Bottom	0.489	🖌 Pass
2	300x600	10H16	3D Building Analysis	10	Bottom	0.630	🖌 Pass
1	300x600	10H16	Grillage Chase Down	10	Тор	0.455	🖌 Pass

The reinforcement in this column has now been reduced from its original design, but this has resulted in column C13 now failing in its top stack. This is because C13 was the critical column for the design of group CRC11.



This failure could be removed by either:

- Using the Design Member command on the now failing column.
- Re-designing the group CRC11, as discussed in the previous example.
- Interactively designing the failing column, which will be discussed in detail shortly.
- Step 1. Use the Design Member command to design the failing column C13 and have its updated reinforcement applied to the column C14.



For walls, this option is called Design Wall, and simply does an automatic design for the individual wall being looked at.

24.3 Interactive Design...

The automatic designs of elements in TSD will generally be more than appropriate for the majority of cases. However, if you'd like to check a column or wall for specific reinforcement, or you just want to adjust the reinforcement automatically selected by the program to see how it affects the design, you can use the Interactive Design capabilities. To do this, you simply right click over a column or wall in your active scene view and choose Interactive Design....

24.3.1 Interactive Design

Once selected, the Interactive Design... option will first perform a check design on the element in question, and then allow you to edit the existing rebar for each stack of that element, based on its analysis results.

For columns, you can adjust the principal (corner) and intermediate (edge) longitudinal bar sizes and their quantity, and the link bar size and spacing. This is done by selecting the appropriate stack in the left hand pane, then selecting the required tab on the right and adjusting the various settings.

Interactive Column Design	Longitudinal Links Interaction	n Dianrams		ОК
C13 max UR = 0.807	Longitudinal Bars Principal bar size H20	Int. lengui Count [mm] ✓ 3-4 1 97.0 ✓	- 4 3	OK Cancel Check Detail Drawing Options
	Position M _{te} [kNm] M _{ret} [kNm] Ratio Nmax [kN] Ratio Smallest clear spacing [mm] A _{stmin} [mm²] A _{stmax} [mm²] A _{stmax} [mm²]	Longitudinal Bars Top Mid-fifth Bottom 298.0 120.3 237.4 369.5 368.3 354.7 0.807 ✓ 0.327 ✓ 0.669 ✓ 363.1 370.9 376.0 3985.9 0.091 ✓ 0.093 ✓ 0.094 ✓ 77.0 ✓ 720 7200 3142 ✓ 1 1	300 mm Minor Minor Major	

As soon as any of these settings are changed, the sketch, design statuses and results are instantly updated and displayed. You can also then click the Check button on the right hand side of this window to perform a check design of the edited reinforcement and view the full design calculations, as discussed earlier.

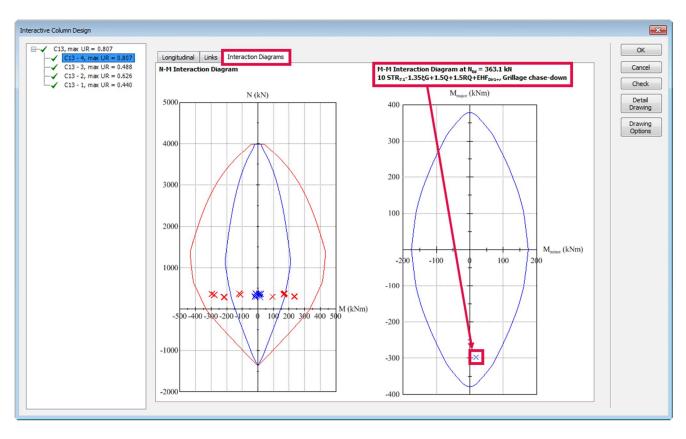
- Step 1. Open the Interactive Column Design window for the column C13.
- Step 2. Try adjusting the reinforcement in the column to see if it can be reduced.
- Step 3. Use the Check button to view the updated results.

24.3.2 Interaction Diagrams

Another useful function in this window is the ability to view the Interaction Diagrams for the column or wall by selecting the Interaction Diagrams tab in the right hand pane. Once opened, it allows for further interrogation of the column design, and shows two different diagrams based on the current reinforcement you have specified in the design. As soon as you edit the design, these diagrams will also update.

The diagram on the left shows the axial load capacity against the moment capacity for the two directions of the column. The results for all combinations and for all analyses completed are all plotted on this diagram, allowing you to quickly see the overall status of the column and how hard it's working.

The diagram on the right shows the major axis moment capacity against the minor axis moment capacity for the axial load from the critical combination. The one plot displayed on this diagram is for the same critical combination results, which is calculated as the worst case results from all combinations, considering all results from all the analyses completed.

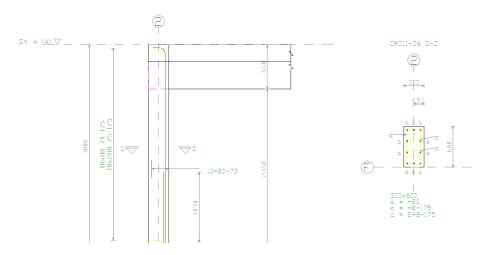


Step 4. Try adjusting the reinforcement placed in column C13 to see how this affects the Interaction Diagrams.

24.3.3 Detail Drawing

The Detail Drawing button in the Interactive Design window allows you to quickly view a detailed drawing of the column or wall. You can make adjustments to the design of the element, and then instantly create a detailed drawing of it, using whichever CAD program you have installed on your machine.

The detailed drawing generated by this method will include an elevation drawing of all stacks making up the whole column or wall, a section drawing for each stack, and a quantities table as default.



T	l loit	Vocc	Tet		+1+	Total Mass	-	
1 Y DB		/m]	101	[m]	un.	[kg]		
HB	0.3	95		187.200		73.94	1	
H20	2.4	66		174.000		429.08	1	
			Gr	rand Tor	tal	503.03]	
Mark	Type	Quan	tity	Length [mm]	To	ital Length [m]	Shape	Members
71	H50	10		2225.0		22.250	34	C13
72	H50	30		4050.0		121500	26	C13
73	HSD	10		3025.0		30,250	- 11	C13
74	Η	78		1700.0		122.400	51	C13
75	Ω.	14	<u> </u>	450.0		64,800	99	C13

Whilst you're more likely to generate detail drawings of multiple columns and walls at once using other functionality in the program, this feature can still be very useful in understanding the design of the element in question. The production of detail drawings will be discussed further later in this course.

Step 5. Generate and review the Detail Drawing for column C13.

24.3.4 Updating the Design Group

As with the Design Member command, when you click OK to confirm the interactive design of a column, the rebar selected in the new design is applied to all of the columns in the same design group. A Check Design is then performed on them and their design calculations and statuses will be instantly updated. To make this a more productive process, make sure that you interactively design the critical column(s) from the Design Group by checking the Groups tab – the critical column within a group is signified with a red exclamation mark.

This does not apply to the Interactive Design of walls, as they are not designed using Design Groups. Confirming the Interactive Design of a wall simply applies the rebar to the wall that's been designed.

Step 6. Click OK to confirm the column (C13) design.

24.3.5 Interactive Design of Walls

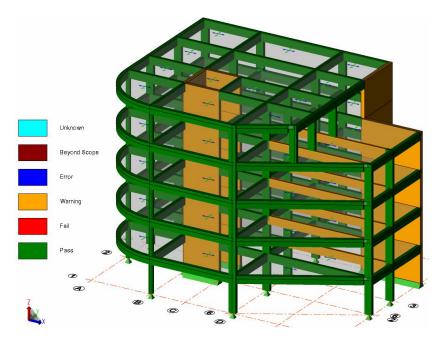
The Interactive Design of walls works in a very similar way to that of columns, but there are some slight differences.

The Interactive Wall Design allows you to specify whether you want to use end zones or not, and whether you want one or two layers of reinforcement.

Wall design also allows you to choose whether you want to use loose bars or mesh reinforcement. When using loose bars, you can select the vertical bar size, the number of rows of bars and any additional end row bars. Using mesh has the same options but allows you to choose the mesh size.

Longitudinal	Lateral	Intera	ction Di	agrams		
Duse end-zo	ones					
Number of lay	vers	2	•			
Reinforcemen	nt type	Loos	e bars	-		
Number of ro	WS	41		Centre	spacing = 75.0	mm 🖌
Vertical bar si	ze	H8	•			
Additional en	d row bars	1		Centre	spacing = 113.	0 mm 🖌

As you will have noticed with this model, all of the walls have a Warning against them and are highlighted in Amber when viewing the Results View - Design Status.



When accessing the Interactive Design window for any of the walls in the model, there are no immediately obvious reasons as to why the walls in this model have warnings, as all checks shown are passing and have green ticks next to them. This suggests that the issues are actually due to detailing problems rather than any of the actual design checks having problems.

Clicking the Check button in the Interactive Wall Design window allows you to view all of the design calculations, as well as any other additional detailing checks completed by the program.

For an example, you can see here by drilling down into these calculations, the issue is with the concrete cover for the walls. The cover specified in the wall properties is 25mm, but according to code checks, the cover needs to be 26mm or greater.

Results	
Panel 4 - Concrete Cover Shear Summary A Panel 4 Concrete Cover Concrete Cover A Concrete Cover Concrete Cover A Panel 2 A Panel 1 A Panel 1 A Concrete Cover A Panel 1 A Panel 1 A Concrete Cover A Panel 1 A Panel 1 A Panel 1 A Concrete Cover A Panel 2 A Panel 3 A Panel 3 A Panel 3 A Panel 4 A Panel 3 A Panel 4 A Panel 3 A Panel 4 A Panel 3 A Panel 4 A Panel 4 A Panel 4 A Panel 3 A Panel 4 A Panel 4 A Panel 3	$c_{nom} = 25.0 \text{ mm}$ $\phi_{largest} = 8.0 \text{ mm}$ $\phi_{cont} = 16.0 \text{ mm}$ $\Delta c_{e_{t}} = 10.0 \text{ mm}$ $\Delta c_{e_{t}} = 10.0 \text{ mm}$ $d_{g} = 20.0 \text{ mm}$ $d_{g} = 20.0 \text{ mm}$ $c_{nom,lim} = MAX[d_{g}, MAX[\phi_{colt}, 10\text{ mm}] + \Delta c_{dev}, MAX[\phi_{largest}, 10\text{ mm}] + \Delta c_{dev} \cdot \phi_{colt}] = 26.0 \text{ mm} \text{ EN 1992-1-1:2004 Section 4.4.1}$

To resolve this issue, do the following:

- Step 1. Select all concrete walls in the model.
- Step 2. In the Properties window, edit the Nominal Cover to 30mm and press Enter to confirm.
- Step 3. Update the design results for all the walls in the model by either repeating the Design Concrete (Static) process or by using the Check Wall or Design Wall commands for each wall, one by one.
- Step 4. Save the model.

180 (257)

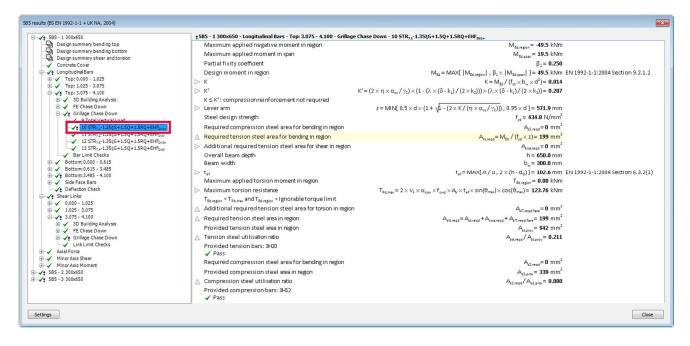
25 Interactive Design of RC Beams

As with columns, you can get TSD to complete a Check Design or an Auto Design on an individual beam member, or you can perform an Interactive Beam Design.

25.1 Check Member

In exactly the same way as with columns, you can perform a Check Design on a beam by right-clicking over it in your active scene view and selecting Check Member.

The Check Member command and window works in exactly the same way as for columns and walls, except all of the design and detailing calculations for the beam are displayed, and you can check these for each of the beam's spans.



Step 1. Use the Check Member command to perform check designs on some of the beams that are already passing their designs.

25.2 Design Member

The Design Member command also works for beams in the exact same way as it does for columns.

To perform an automatic design for a single beam, simply right click over a beam and choose Design Member. All spans of that individual beam will be designed based on the analysis results for that beam, and the reinforcement selected will then be copied to all other beams in the same design group.

Check Design will be completed on these members, and their design calculations and statuses will be updated immediately.

As with columns, if you use the Design Member command on a beam that isn't a critical beam in that design group, you may find some beams that had passed will suddenly fail.

Step 1. Locate beam 1B4 in the model from design group CRB6.

- Step 2. Use the Design Member command to design beam 1B4 some other beams should now have warnings against them.
- Step 3. Use the Design Group command in the Groups tab to re-design group CRB6.

25.3 Design...

If you'd like to check a beam for specific reinforcement, or you just want to adjust the reinforcement automatically selected by TSD to see how it affects the design, you can use the Interactive Design capabilities in a similar way to columns and walls.

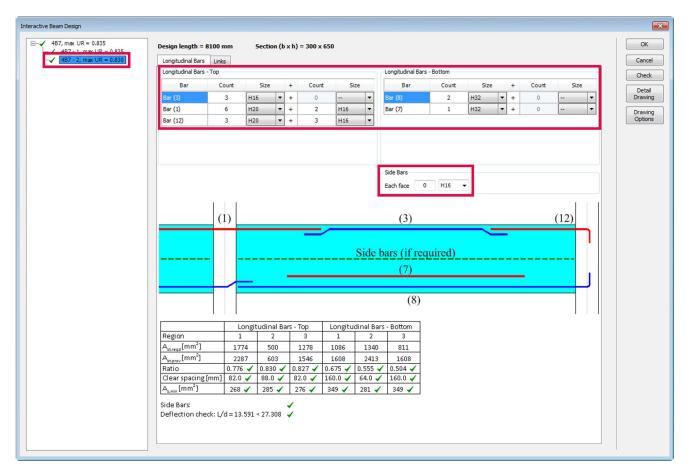
To do this, you simply right click over a beam and choose Design....

25.3.1 Interactive Design

Once selected, the Design... option will first perform a Check Design on the beam in question, and then allow you to edit the existing rebar for each span, based on that element's analysis results.

For beams, you can adjust the top and bottom longitudinal bar sizes and their quantity, the side bar size and quantity (if required), and the link bar size, number of legs and spacing.

This is done by selecting the appropriate span in the left hand pane, then selecting the required tab on the right and adjusting the various settings.



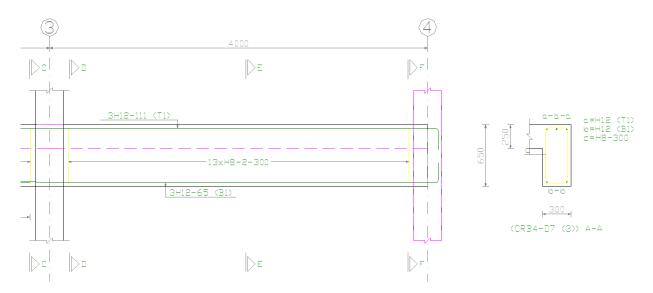
As soon as any of these settings are changed, the sketch, design statuses and results are instantly updated and displayed. You can also then click the Check button on the right hand side of this window to perform a check design of the edited reinforcement and view the full design calculations, as discussed earlier.

- Step 1. Open the Interactive Beam Design window for a few different beams in the model, except the ones with warnings.
- Step 2. Review their designs, and try adjusting the reinforcement placed in some of them to see how they affect their design status.

25.3.2 Detail Drawing

The Detail Drawing button in the Interactive Beam Design window allows you to quickly view a detailed drawing of the beam. You can make adjustments to the design of the element, and then instantly create a detailed drawing of it, using whichever CAD program you have installed on your machine.

The detailed drawing generated by this method will include an elevation drawing of all spans making up the whole beam, section drawings for each span and support regions for each beam span, and a quantities table as default.



As with the columns and walls, though you're more likely to generate detail drawings of multiple beams at once using other functionality in the program, this feature can still be very useful in understanding the design of the element in question. The production of detail drawings will be discussed further later in this course.

Step 3. Generate and review the Detail Drawing for some beams.

25.3.3 Correcting the Warnings

As you will have seen, some beams in the model have some warnings and are shaded amber when viewing the Results View - Design Status.

Opening the Design... window for any of these beams shows green ticks for all of the design results displayed, which suggests this is potentially a detailing issue.

Therefore, the Check window needs to be opened to view all of the calculations to see exactly what the issue is.

Step 4. Open the Interactive Beam Design window for beam 4B3.

Step 5. Click the Check button to view the calculations.

esults ⊕-√: 4B3 - 1 300x650	4B3 - 2 300x650 - Concrete Cover	
 ↔ 483 - 1 300x550 ▲ 483 - 2 300x550 ➡ Design summary bending top ➡ Design summary bending bottom ➡ Concrete Cover ➡ Avail Force ➡ Minor Avis Moment 	483 - 2 300x550 - Concrete Cover User nominal top face cover ▷ Nominal limiting top face cover ✓ Pass User nominal bottom face cover ▷ Nominal limiting bottom face cover △ Warning User nominal side face cover △ Warning User nominal end cover Largest top face bar diameter	$c_{nom,imp} = 30.0 \text{ mm}$ $c_{nom,imp} = MAX[MAX[\phi_w, 10mm] + \Delta c_{dev}, MAX[\phi_{top}, 10mm] + \Delta c_{dev}, \phi_w] = 22.0 \text{ mm} EN 1992-1-1:2004 Section 4.4:$ $c_{nom,im,bet} = 30.0 \text{ mm}$ $r c_{nom,im,bet} = MAX[MAX[\phi_w, 10mm] + \Delta c_{dev}, MAX[\phi_{bet}, 10mm] + \Delta c_{dev}, \phi_w] = 34.0 \text{ mm} EN 1992-1-1:2004 Section 4.4:$ $c_{nom,im,ide} = MAX[MAX[\phi_w, 10mm] + \Delta c_{dev}, MAX[\phi_{bet}, 0mm] + \Delta c_{dev}, \phi_w] = 34.0 \text{ mm} EN 1992-1-1:2004 Section 4.4:$ $c_{nom,im,ide} = MAX[MAX[\phi_w, 10mm] + \Delta c_{dev}, MAX[\phi_{bet}, 0mm] + \Delta c_{dev}, \phi_w] = 34.0 \text{ mm} EN 1992-1-1:2004 Section 4.4:$ $c_{nom,im,ide} = MAX[MAX[\phi_w, 10mm] + \Delta c_{dev}, MAX[\phi_{bet}, 0mm] + \Delta c_{dev}, \phi_w] = 34.0 \text{ mm} EN 1992-1-1:2004 Section 4.4:$
	Largest top face bar diameter Largest top face bar diameter Link diameter Allowance for deviation △ Nominal limiting end cover ▲ Warning	$v_{top} = 22.0 \text{ mm}$ $\phi_{bec} = 32.0 \text{ mm}$ $\phi_{aide} = 32.0 $

As seen above, the issue here is with the Concrete Cover checks. The automatic design has selected bars that are sufficient for the design checks but due to the selection of 32mm bars, the nominal cover specified in the beam design is not quite sufficient to meet the code criteria.

There are various different ways of correcting this issue, such as increasing the beam nominal cover, adjusting the section sizes or reducing the forces in the beam by editing the model.

However, the easiest, and probably most sensible approach here, would be to reduce the bar size being used in the beam design. For a model of this size with these sorts of element sections, it would be quite unlikely that 32mm bars would be used in reality.

Step 6. Use the Interactive Beam Design window to adjust the reinforcement provided for beam 4B3 so that it passes both the design and detailing checks.

Alternatively, you could try to set the Maximum Longitudinal Bar Size for beams in the Design Options to a smaller bar size (say 25mm), and then re-run the Design Concrete (Static) process again to make sure no beams in the model use 32mm bar size.

25.3.4 Updating the Design Groups

As with the Design Member command, when you click OK to confirm the interactive design of a beam, the rebar selected in the new design is applied to all of the beams in the same design group.

A Check Design is then performed on them and their design calculations and statuses will be instantly updated. To make this a more productive process, make sure that you interactively design the critical beam(s) from the Design Group by checking the Groups tree – the critical beam within a group is signified with a red exclamation mark.

Step 7. Click OK to confirm the beam design and ensure that all beams in the same group now pass.

As seen earlier, all frame elements are designed in one go as part of the Design Concrete (Static) process. However, slabs are designed using a separate, slightly more manual procedure, which will be covered in detail in this example.

Step 1. Open the model TSD Concrete Design Fundamentals Model 4 - Slab Design.tsmd.

26.1 Introduction to Slab Design

To design the concrete slabs in a model, whether they're part of a Flat Slab or a Beam and Slab structure, there are a number of steps required to get as simple yet economical reinforcement arrangement as possible.

When slabs are initially created, they have several properties relating to their reinforcement, known as their background reinforcement. When the slab is then designed, this reinforcement will be automatically selected based on all of the analysis results calculated anywhere within the slabs, and then placed throughout the whole slab panel.

Designing the background reinforcement in this way, to resist all forces in the slabs, would result in excessive reinforcement provisions, as the higher peak forces, and hence the higher reinforcement requirements in the slabs, normally only occur in isolated, local positions, such as near beams or at the corners of core wall areas.

Therefore, a variety of Patches need to be inserted into the model in these local areas so that the background reinforcement placed throughout the whole slab panels can be kept down, and additional reinforcement can be placed in the patch positions.

Slab Item(s): 1 items	▼ Save Apply
Slab parameters	
Slab properties	
Overall depth	250.0mm
Concrete type	Normal
Concrete dass	C32/40
Dry density	2500kg/m ³
Wet density	2600kg/m ³
Diaphragm option	Rigid
1 Design parameters	
Design parameters	
Reinforcement	
Top bars	
Outside layer	2010 - 1011
Туре	Loose bars
Rib type	Type 2
Bar type	500
Bar size	8
Bar spacing	200.0mm
Туре	Loose bars
Rib type	Type 2
Bar type	500
Bar size	8
Bar spacing	200.0mm
Outside layer in X direction	
Bottom bars	
Outside layer	
Туре	Loose bars
Rib type	Type 2
Bar type	500 8
Bar size	- First Street and Street Stre
Bar spacing	200.0mm Loose bars
Type	
Rib type	Type 2 500
Bar type Bar size	8
	8 200.0mm
Bar spacing Outside layer in X direction	200.0mm
and the second se	30.0mm
Top cover Bottom cover	30.0mm 30.0mm

There's a variety of different Patch types available, depending on the location of where the patch is to be placed, such as over columns, walls beams or in the middle of slab panels. The patches essentially just contain a number of strips running in both directions, some of which are set to design for the average design forces across their widths, and others simply gather a maximum value from across their widths.

Once the main slab design is completed, Punching Shear Checks can then also be added to specific columns and walls in a similar manner to adding patches, to see whether any additional shear reinforcement is required in these locations.

The model used for this exercise has some floors which are flat slab and others which are of beam and slab construction.

Step 1. Select any slab in the model and review its Reinforcement in the Properties window.

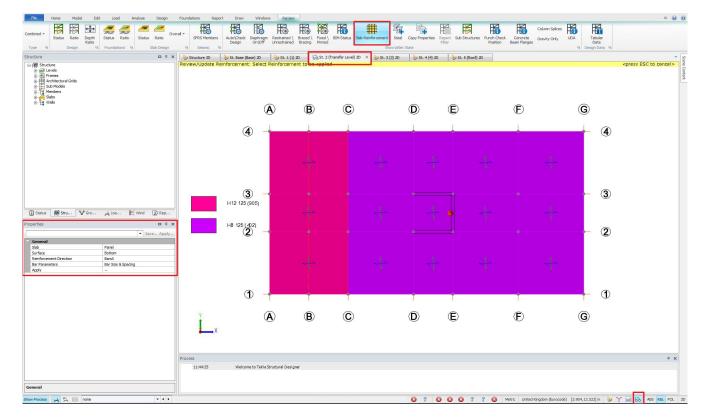
26.1.1 Overall Slab Design Procedure

Below is a brief overall view of the full slab design procedure. These points will be discussed in more detail throughout the rest of this example.

- Insert the patches in the appropriate locations
- Design the slab panels to select the background reinforcement for the whole slab panels to resist the forces in the slab areas not covered by patches
- Review and optimise the slab panel designs by rationalising the selected steel
- Design the patches to calculate the additional reinforcement required in the areas of slab covered by the slab patches
- Review and optimise the patch designs by rationalising the selected steel

26.2 Checking the Background Reinforcement with No Patches

As an initial test, all slabs in the whole model can be designed without having any patches placed in the model. This is done by clicking the Design Slabs command on the Design tab. This means the background reinforcement will be selected on every level so that each slab panel in the model will have sufficient reinforcement placed in it to be able to resist any of the moments anywhere within that panel.



Step 1. Go to the St.2 (Transfer Level) 2D view, click the Design Slabs command on the Design tab.

Once the design of the slabs is complete, the scene view should automatically switch to the Review View mode and show the all slabs passing.

However, using the Slab Reinforcement command on the Review tab, you can see the reinforcement selected, based on the options selected in the Properties window, which could be seen as a little excessive.

Step 2. Use the Slab Reinforcement command and the Properties window to see the reinforcement that has been automatically selected and placed throughout the slab panels.

26.3 Types and Applications of Patches

As the background reinforcement selected is too high, Patches need to be inserted into the areas where higher reinforcement are required. There's a variety of different Patch types available, depending on the location of where the patch is to be placed, and they're found on the Design tab. They include:

- Patch Column placed on columns in flat slab models, to deal with local peak moments
- Patch Beam placed along beams, usually to deal with hogging moments
- Patch Wall placed along walls, with options associated with the position and span of the wall
- Patch Panel placed in the middle of slab spans to deal with local peak moments

Patches contain a number of strips, some of which are set to design for the average design forces across their widths, and others simply gather a maximum value.

The patches also have a variety of properties allowing you to control options like the patch size, strip widths, bar sizes and their spacing. It is important to make sure the patch sizes are appropriate for the model.

The easiest way to do this is to insert the patches whilst viewing the results contours to ensure the peak forces are contained within the patch dimensions.

You should also ensure that both the slab panel and patch minimum bar spacing are sensible before attempting the design.

Patches are inserted into the model by first selecting the patch command you want to use, then by either left clicking on an appropriate element, or by dragging a window around multiple elements.

- Step 1. Go to the St.1 (1) 2D view and set to the Results View mode.
- Step 2. Select the Total Load combination, choose the FE Chasedown Results Type from the Results tab and review the slab design moment and area of steel requirement contours.
- Step 3. Switch to the Design tab.

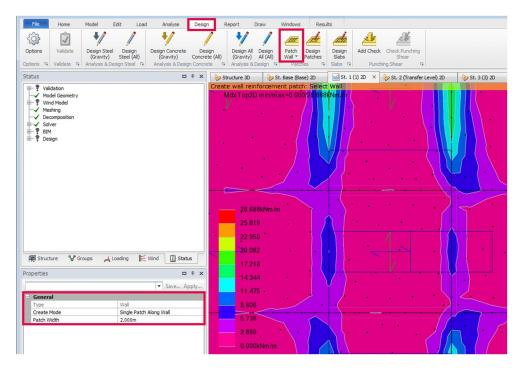
26.3.1 Wall Patches

There are different wall patches available, depending on where the walls are located and how they intersect. The different types can be selected using the Create Mode option in the Properties window once the Patch Wall command has been activated.

In this example, the Single Patch Along Wall option will be used for the two opposing horizontal walls, which are inserted by either left clicking on the individual walls, or by dragging a box around

them. The remaining wall will use the Internal Patch option, which is inserted by left clicking on the start and end points along the wall that isn't covered by the other already inserted wall patches.

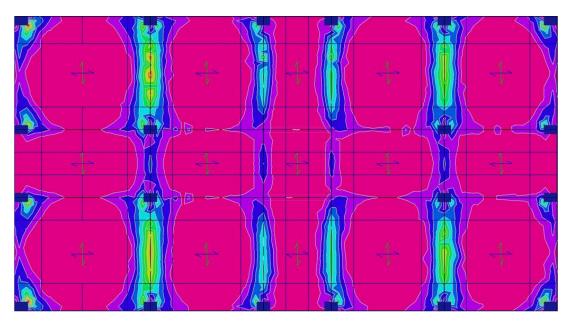
- Step 1. Select the Patch Wall command on the Design tab.
- Step 2. Set the Patch Width to be 2m.
- Step 3. Insert the patches on the 3 walls in the model, using the appropriate Create Mode settings.



26.3.2 Beam Patches

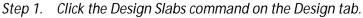
The rest of the patches inserted on St.1 will be Beam Patches. Column Patches are not required as all columns on this floor are connected to beams, so they will be covered by the beam patches.

- Step 1. Select the Patch Beam command on the Design tab.
- Step 2. Ensure the Patch Width is set as 2m and then drag a window around the whole floor to insert all of the Beam Patches at once.

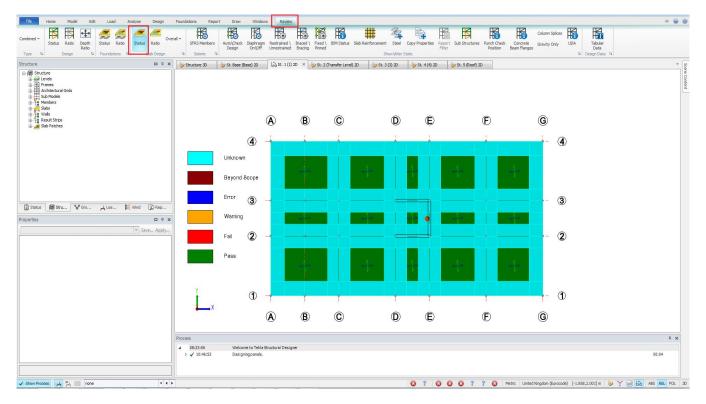


26.4 Design the Slab Panels

Now that all patches are inserted into the floor, the slab panels can be designed by clicking the Design Slabs command on the Design tab. The auto design selects the background reinforcement for the slab panels, ignoring the forces that develop within the slab patch areas, but the background reinforcement is placed throughout the whole slab panel. This process still applies to the slabs on all levels.



Step 2. Use the Review View and the Review tab to view the design Status of the slab panels.



The reinforcement selected for the panels now should be less than before the patches were added.

26.5 Review and Rationalise the Panel Design

Once the slab panels have been designed, you can view the selected reinforcement using the Slab Reinforcement command on the Review tab when in the Review View. The Slab setting in the Properties window should be set to Panel, and you can then review the reinforcement placed in the panels for the top and bottom of the slab, and in the X and Y directions.

The Apply property allows you to apply the reinforcement selected for one panel into other panels, simply by selecting the reinforcement you want to apply to a panel in the Properties window, then left clicking on the panel in the view. This allows you to rationalise the background reinforcement for all slab panels.

Once applied, a Check Design is performed on the slabs to ensure they still work.

File Home Model Edit Load Analyse Design Combined + Type File File <th>erall - Seismic Auto/Check Diaphragm I</th> <th>ndows Review Restrained \ Braced \ Fixed \ BIM Status Jurestrained Bracing Pinned</th> <th>Slab Reinforcement Clauri Vitor del te</th> <th>s Report Filter</th> <th></th>	erall - Seismic Auto/Check Diaphragm I	ndows Review Restrained \ Braced \ Fixed \ BIM Status Jurestrained Bracing Pinned	Slab Reinforcement Clauri Vitor del te	s Report Filter	
Structure 🗖 🖣 🗙	Structure 3D 🛛 🖗 St. Base (Base) 2D) 🔓 St. 1 (1) 20 × 🍃 St. 2 (Transf	ier Level) 2D 🛛 🦕 St. 3 (3) 2D 🛛 🖕 S	st. 4 (4) 2D 🛛 😺 St. 5 (Roof) 2D	ي +
Gerege Levels Gerege	Review/Update Reinforcement: Sele			4	<press cancel="" esc="" to=""></press>
I Status ∰ Stru V Gro ↓ Los K Wind D Rep Properties ■ ♥ x	H8 125 (402)	÷	4	4	4
■ General Slab Panel Surface Top Renflorcement Direction Bar Size & Spacing Ber Parametters Bar Size & Spacing Apply 835	Y L X		4	4	

Step 1. Review and rationalise the top and bottom reinforcement selected for both the X and Y directions.

26.6 Design the Slab Patches

Now that the panels are appropriately designed, the patches can also be designed by clicking the Design Patches command on the Design tab. This auto design process considers the reinforcement that has already been provided in the whole slab panels, and selects the additional reinforcement required for within the patch areas. Once complete, this should mean the patch areas have sufficient total reinforcement to resist the forces that are acting within them.

Step 1. Click the Design Patches command on the Design tab.

Step 2. Use the Review View and the Review tab to view the design Status of the slab patches.

26.7 Review and Rationalise the Patch Design

As with the slab panels, once the patches have been designed, you can view the selected reinforcement by using the Slab Reinforcement command on the Review tab when in the Review View. The Slab setting in the Properties window should be set to Patch first, and you can then review the reinforcement placed in the patches, much like you could for the panels.

The Apply property allows you to apply the reinforcement selected for one patch into other patches, simply by selecting the reinforcement you want to apply to a panel in the Properties window, then left clicking on the patch in the view. This allows you to rationalise the additional patch reinforcement to get a simple yet economic arrangement.

Buttare Control Contro Control Control	Status Ratio De	Edit Load Analyse pth tio fx Foundations fx	tus Ratio	Seismic Au	to/Check Design Diaphragm On/Off	Restrained \ Bracing	Fixed \ BIM Statu	s Slab Reinforcemer Show \Alte		perties Report Filter	Sub Structures B Punch Check Position Concrete Beam Flang	Column Splices Gravity Only		sign ta *
Image: Service	ucture		🗆 🖗 🗙 🚺	Structure 3D	St. Base (Base)	2D 63 st. 1 (1) 2D	× 😡 St. 2 (Trar	isfer Level) 2D	≽ St. 3 (3) 2D	🥪 St. 4 (4) 2D	🥪 St. 5 (Roof) 2D			Ŧ
3 state ## Stru V Gro ↓ Los E Wind D Rep berties • • • ↓ ↓ serties • • ↓ ↓ State Save Apply ↓ ↓ Serforcement Decton BarY BarY ↓ ↓ Save Bar Save Apply ↓ Ver Bars Save Save ↓ Set type Soo Save ↓ Set type Soo Save ↓	Frames Architectural Grids Sub Models Sub Models Sub Sub Sub Sub Sub Sub Sub Sub Sub		Re	view/Update F	Reinforcement: Se	lect Item to Apply	Reinforcement t	0				≺press E	SC to cance	: >
3 stats ₩ 5 void ₩ 6 void ₩ 7 v						+		1	+		+	1		
Slab Path Surface 1 fop Renforcement Direction Barry Apply Luse Bars User Bars User Bars Bit type 500 Bar type 500 Bar space 8 Bar Space 8	1 Status 🏘 Stru 😵 Gr perties		□ # ×			+		+			+	+		
Surface Top Renforcement Decton Barry Annual Renforcement Decton B	General													
Benforcement Dector Bardy Benforcement Dector Bardy Ber Stre & Spoolng Apply User Bars How Fors Step Parameters Bardy Ber By December 2000 How Fors Step Parameters Bardy Ber By December 2000 How Fors Ber By December 2000 Bardy Ber Step 8 Source												-		
Bar Star Bar Star & Spoong hoppy Uler Bars User Bars User Bars Bis type 500 Bar syne 8 Bar sen 8						1.2		1.4	1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1		1 A A			
V V	Bar Parameters							1	<u></u>		4			
kb type 1 type 2 Bar type 500 Bar size 8	Apply	User Bars				/		V	V		No.	V		
Sar type 500 Sar dze 8 y	User Bars													
tar size 8 a a a a a a a a a a a a a a a a a a	Rib type	Type 2							and the second					
	Day Arms	500												
Bar spacing 125.0mm		8		V			0			-				

Once applied, a Check Design is performed on the slabs to ensure they still work for the new arrangement.

Step 1. Review and rationalise the additional patch reinforcement selected for both the X and Y directions – this will typically only apply to the top reinforcement.

Once this stage is complete, the slab is completely designed for the first storey St.1. However, as St.1 is a unique storey, no patches exist on any other levels, so a similar process would need to be completed for all other unique levels, or levels that have been used as the Source Storey in the Construction Levels window. At present, the slabs on all other storeys are designed with no patches, so their current background reinforcement could be considered excessive.

26.8 Flat Slab Design

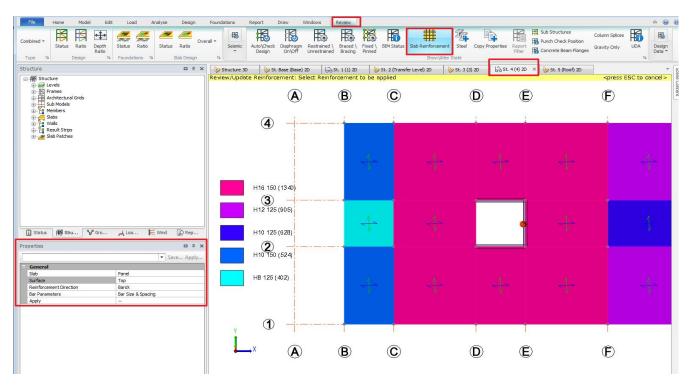
The process for designing a Flat Slab structure or floor is exactly the same as designing a beam and slab model – the only real difference is the type of patches used.

- Step 1. Switch to St.4 (4) 2D view, set to the Review View mode.
- Step 2. Select the Slab Design Status command on the Review tab.

As you should see, this level is a flat slab. The Slab Design Status should show that the slab passes the design due to the Design Slabs process having been completed earlier. However, as there are no patches inserted on this level, this means the background reinforcement has been selected to resist all forces acting anywhere within the slab panels.

Using the Slab Reinforcement command on the Review tab, you should be able to see that the selected top reinforcement is quite high in some slab panels due to the higher hogging moments over the columns and at the corners of the core wall.

Therefore, some patches need to be inserted into this floor to help reduce this background reinforcement in a similar manner to the beam and slab level.



- Step 3. Review the currently selected background reinforcement.
- Step 4. Go to the Results View mode, select the Total Load combination, choose the FE Chasedown Results Type from the Results tab and review the slab design moment and area of steel requirement contours.
- Step 5. Switch to the Design tab.

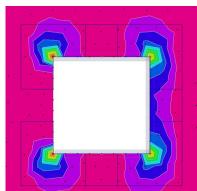
26.8.1 Wall Patches

The Wall Patches can be inserted into this floor following the same method as discussed earlier, only this time, the Internal with End Patches option will be used for the Create Mode for the two horizontal walls. These are inserted by either left clicking on the individual walls, or by dragging a box around the walls instead.

This is because on the other floor, beams connected into the walls, so the beam patches helped to cover the forces on the corners of the core wall. As this is a flat slab level, the use of wall end patches should ensure that all of the peak forces at the corner positions are covered by the wall patches.

The remaining wall will use the Internal Patch option, the same as on St.1.

- Step 1. Select the Patch Wall command on the Design tab.
- Step 2. Set the Patch Width to be 2m.
- Step 3. Insert the patches on the 3 walls in the model, using the appropriate Create Mode settings.

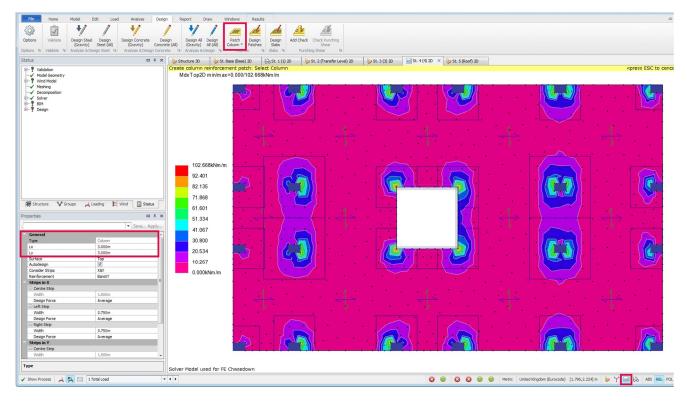


26.8.2 Column Patches

The rest of the patches on this storey will be Column Patches. These column patches can be inserted into the floor by either left clicking on individual columns, or by dragging a window around multiple columns.

As with the beam and wall patches, their dimensions can be adjusted in the Properties window to ensure all of the higher peak force contours are contained within the column patch.

For models such as this example, you will likely find larger column patches are required for the internal column positions, where the moments in the slabs will be higher.

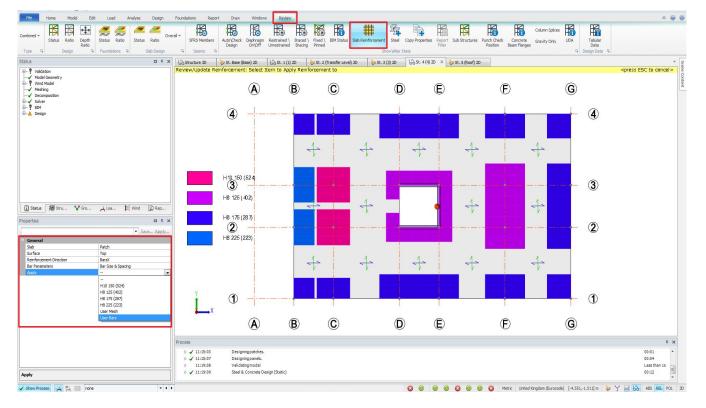


Step 4. Try inserting some column patches (3m x 3m) into the floor, adjusting their dimensions as required.

26.8.3 Completing the Design of the Floor

To complete the design of this floor, simply follow the same process as for St.1 level:

- Step 5. Design the Slab Panels to select the background reinforcement for the whole slab panels to resist the forces in the slab areas not covered by patches.
- Step 6. Review and optimise the panel designs by rationalising the selected steel.
- Step 7. Design the Patches to calculate the additional reinforcement required in the areas of slab covered by the slab patches.
- Step 8. Review and optimise the patch designs by rationalising the selected steel.



As St.3 is a duplicate of St.4, you should find that this floor has also been designed.

When duplicate levels are modelled and the slabs are designed, it works in a similar way to Design Groups. When the Design Slabs and the Design Patches commands are completed, the panels and patches on the Source floor are designed, and then their selected reinforcement is then copied to its duplicates. A check design is then completed on the duplicate floors and their design status is updated. This means that you may find that some panels and/or patches fail on the duplicate levels.

To rectify this, you simply need an auto design to be completed on the level with the failures. This can be done by either selecting the failing element, right clicking and choosing Design Member, or by right clicking anywhere in the view and choosing Design Slabs or Design Patches.

Once one of these design processes have been completed, you can then review the full design calculations for the slabs in question.

26.9 Punching Shear Checks

The final step in the flat slab design process is to complete the Punching Shear Checks.

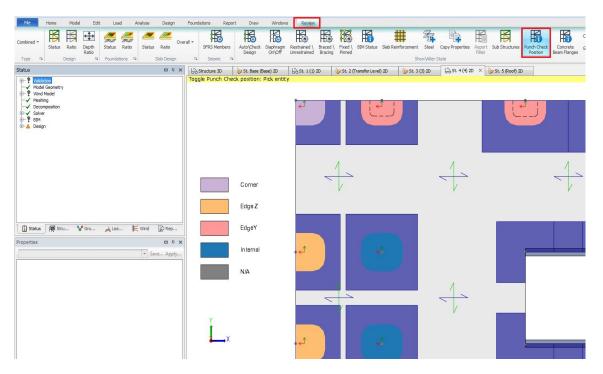
This must be completed after the main slab reinforcement has been designed so that it can be correctly considered when calculating if any additional shear reinforcement is required.

Punching checks are inserted by clicking the Add Check command on the Design tab. They're inserted in a similar method to inserting column patches – you can either left click on individual columns, or you can drag a window around multiple elements.

Step 1. Insert a few punching shear checks in different locations on St.4 using the Add Check command.

File Home Model	Edit Load	Analyse	Design	Report	Draw	Windows	Review	,						
🏟 🖸 🤸	1	+/	1	+/	1	177	A	da	1					
Options Validate Design	Steel Design	Design Concrete	Design	Design Al	Davis	Patch	Design	Design	Add Che	eck Check Pun				
(Gravi	ty) Steel (Al)	(Gravity) (Concrete (All)	(Gravity)	Design All (All)	Column *	Patches	Slabs	~	Shear				
Options & Validate & Analysis	& Design Steel G	Analysis & Design	Concrete 1	Analysis 8	Design G	Patch	nes G	Slabs G	Pu	nching Shear	5			
Status			×	Structure 3D	leg St	Dase (Dase	1 20	1 St. 1 (1)	20	St. 2 (Trans	ofer Level) 2	o l	A St. 3 (3)	20 ×
E- ? Valdation				te Panchis							U	1		
- ✔ Model Geometry 단- ♥ Wind Model - ✔ Meshing - ✔ Decomposition 단- ♥ Solver 당- ♥ BitM B)- ♥ Design			_								i			
					Ur	known								
		*	_		80	yond So	ope							
Aid Structure V Groups Properties	M Loading 📄	Wind 🚺 Status		Ļ	En	or								
- General														
Tension Reinforcement	Top		_		W	erning								
Point Load Breadth	100.0mm			_										
Point Load Depth	100.0mm													
Point Load Orientation	0.0000°				Fa	a								
Bota - User limit	1				Fa									
u0 - user reduction	0.0mm			1										
u1 - user reduction	mm0.0													
				-	Pa	155								

Once you have inserted the punching shear checks, you can use the Punch Check Position command on the Review tab to make sure TSD has specified the correct position settings for each inserted check.



If you want to change the position setting for a particular check, simply left click on the check in the view and it will change to the next setting in the list.

Keep left clicking on the check to cycle through the options until the one you want to use is selected.

Once you're happy with the position settings of the checks, click the Check Punching Shear command on the Design tab to complete the checks.

File	Home	Model	Edit	.oad Analyse	Design	R	leport	Draw	Windows				
Options 5	Validate	Design Ster (Gravity) Analysis & D	Steel (Al) (Gravity)	ete Desig Concrete esign Concrete	(All)	Design Al (Gravity) Analysis 8	All (All)		Desig Patche	n Design es Slabs	Add Check Punch	Check Punching Shear
Status					D & X	St 🖓	ructure 3D	by s	t. Base (Base)	2D	63 St. 1 (1) 2	D 👂	St. 2 (Transfer Lev
B-? Wir -√ Me	del Geometry nd Model shing composition ver sign	rouns 4	Loading	≝ wnd ी िी S	tatus						Undo Ctrl+ Redo Ctrl+ Copy Delete Element Zoom Out Check Punching Check Patches Design Patches Redraw Save Screensho	Z Y	
Properties			county	▼ Save	• • ×				·				

The checks will be highlighted in Green (Pass) or Red (Fail), depending on whether the checks passes or fails.

To view the punching check calculations, simply right click over one of the punching checks and choose the option Check Punching.

- Step 2. Click the Check Punching Shear command on the Design tab.
- Step 3. Right click over one of the punching checks to view the calculation.
- Step 4. Save the model.

27 RC Members Detail Drawings

As seen earlier, Detail Drawings of individual frame elements can be created via the Interactive Design windows. However, there are a variety of other detail drawing options available on the Draw tab.

27.1 Drawing Settings

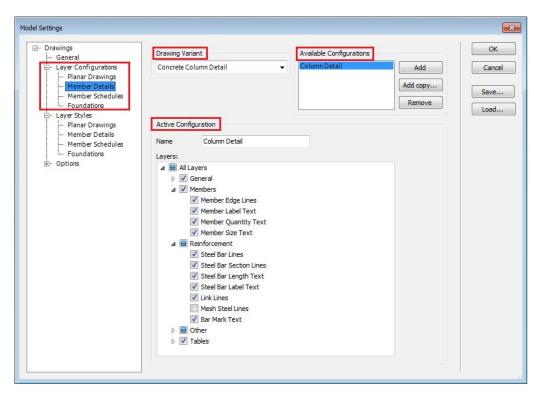
As with the Analysis and Design settings, it is important to review the Drawing Settings before generating any detail drawings. This is done by clicking the Edit... command on the Draw tab.

Step 1. Go onto the Draw tab and click the Edit... command.

27.1.1 Layer Configurations

Selecting the Layer Configurations option on the left hand side of the Model Settings window allows you to control what information is included in each of the different types of drawings that can be created.

- Select a layer configuration and choose a Drawing Variant.
- Select on an Available Configuration from the list its details into the Active Configuration area of the window.
- Active Configuration displays the Name of that type and the various Layers that will be contained in the drawing when it's created. Using the tick boxes, you can add or remove layers from the Active Configuration.



Step 1. Review the various Drawing Varients available.

27.1.2 Layer Styles

Once you have specified which layers are displayed for each detail drawing type, you can then control how those layers are displayed by adjusting the Layer Styles.

Similar to the Types page of settings, the Layer Styles page allows you to choose a Drawing Variant and Available Styles. Once the Active Style is selected, you can then edit the various settings for the Layers to suit your needs.

These settings include the Name of the layer, the Colour, the Line Type, the Font type and the Font Size.

nfigurations r Drawings ber Details ber Schedules dations		Available Beam De		Add	dd copy						
r Drawings ber Details ber Schedules dations Layers: Description	Name	Is Merger	Merged with	Color	Report	Line Type		Font	A	pply to All Font Size [mm]	
	Grid Lines	. 3.			1		T	1			2
Grid Lines				-		Dash Dot	-		1.22		9
Axis Text	Axis Text				8	Solid	1000	Txt		5.0	E
Axis Balloons	Axis Balloons					Solid	-		_		8
Section Text	Section Text					Solid	-			2.5	
Dimensions	Dimensions					Solid	-	Txt	-	2.5	
Thick Symbol Lines	Thick Symbol Lines	<u>100</u>				Solid	-				
Thin Symbol Lines	Thin Symbol Lines					Solid	-				
Small Text	Small Text					Solid	-	Txt	-	2.5	
Steel Bar Lines	Steel Bar Lines					Solid	-				
Steel Bar Section Lines	Steel Bar Section Lines	8				Solid	-				
Steel Bar Length Text	Steel Bar Length Text					Solid	-	Txt	-	2.5	
Steel Bar Label Text	Steel Bar Label Text	1				Solid	-	Txt	-	2.5	
Link Lines	Link Lines					Solid	-				
Mesh Steel Lines	Mesh Steel Lines					Solid	-		-		

Step 2. Review the various Layer Styles available.

27.1.3 Options

As well as the Types and Layer Styles, there are various other Options also available for the main concrete detail drawings. These can act like filters for individual layers.

For example, you can choose to include the Labels layer in the Types page of settings for a particular type of drawing, but you can use the Options to include or exclude specific labels that all combine to make up that single layer.

Step 3. Review the various Options available, then click OK to confirm all settings.

27.1.4 Detailing Groups

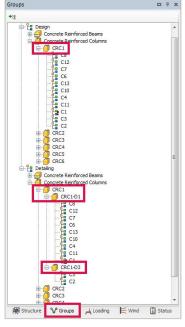
As discussed earlier, all concrete beams and columns are automatically placed into Design Groups to allow for a quick and efficient design process. At the same time, corresponding Detailing Groups are also automatically created with the same names and the same members inside them, which allows for a refined and efficient detailing process.

Although all the elements inside a design group are similar enough to allow them to have identical designs, there could be variations in the other elements surrounding and connecting to the members in that group, which would result in differences in the detail drawings created for each of the group members.

Therefore, within the Detailing Groups, there are sub-groups containing the elements with the same detailing arrangements.

For example, you could have a design group containing 10 beams, which are all designed identically as they have the same section size, number of spans, span lengths, etc. However, in the corresponding Detailing Group, there could be two sub-groups – one for the 5 beams supporting a 200mm thick slab, and the other for the 5 beams supporting 300mm thick slabs.

Step 4. Review the Detailing Groups in the Groups tab under Project Workspace.



27.2 Detail Drawings of Individual Elements

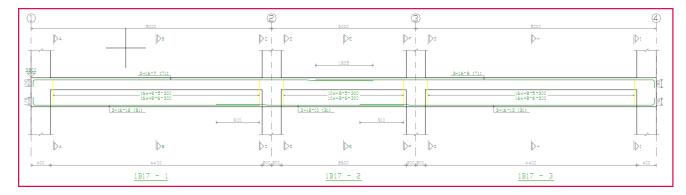
Detail Drawings of all individual frame elements (i.e. columns, walls and beams) can be created via the Interactive Design windows. These same detail drawings can also be generated by right-clicking over the element in a view and choosing Generate Detailing Drawing....

Once selected, the DXF Export Preferences window will display the file name and where it will be saved, which can be changed by unticking "Use automatic file name".

The Layer Configuration and Layer Style should be automatically selected based on the type of member you're creating the drawing for, and the corresponding Drawing Variant.

You can also adjust the Scale of the drawing, as well as the Minimum Text Blocking Spacing.

Clicking OK will create the drawing and display it using an appropriate CAD program that you have installed.



Step 1. Try generating some individual member detailing drawings.

27.3 Schedules

As well as creating detail drawings for individual members, Schedules can also be created for columns, walls and beams by clicking the appropriate schedule commands on the Draw tab.

Column and Wall Schedules can be created when in any view of the model, but Beam Schedules can only be created when viewing a plan view.

- The Column Schedule allows you to select which Column Detailing Groups you want to included.
- The Wall Schedule allows you to select which walls to be included.
- The Beam Schedule all beam detailing groups are included for the currently active plan view.

Once confirmed, a DXF Export Preferences window will open, similar to the Generate Detailing Drawing command.

Clicking OK will create the schedule and display it using an appropriate CAD program that you have installed.

Once the schedule is generated, various information will be displayed.

- The Column and Wall Schedules will display various Sections of the different members selected to help display the longitudinal bars and links, along with a Quantity Table.
- The Beam Schedule will provide information on the reinforcement provided in the beams, along with bar bending details.

St. 5 (Roof) 16 m					Туре	C	t Mass T kg/m] 0.395	otal Lengt [m] 5899.700	h Total Mass [kg] 2330.38]	
		<u>+ 400</u> +	\land /	H -400 →I	H12		0.888	2230.250 368.000	1980.46 581.07		
		< <u>-</u> 1		C7				Grand Tot		1	
						< Typ	e Quanti	Emml	Total Length [m]	Shape	Members
	ε	6	\backslash		38	H1	2 110	1775.0	195.250	34	C1, C2, C3, C4, C6, C7, C8, C10, C11, C12, C13
	3.2		X		39	H1	2 440	3825.0	1683.000	26	C1, C2, C3, C4, C6, C7, C8, C10, C11, C12, C13
					40	H1	2 110	3200.0	352.000	11	C1, C2, C3, C4, C6, C7, C8, C10, C11, C12, C13
		400×600		a a a 400×600	41	HE	2820	1550.0	4371.000	51	C1, C2, C3, C4, C6, C7, C8, C10, C11, C12, C13
St. 4 (4)		a = H12-40 b = 2H8-41-125		a = H12-40 b = 2H8-41-125	42	HS	1410	750.0	1057.500	99	C1, C2, C3, C4, C6, C7, C8, C10, C11, C12, C13
12.8 m		c = H8-42-125		c = H8-42-125	43			1975.0	79.000	34	C17, C18, C19, C20
-	ε				44	H1		4025.0	161.000		C17, C18, C19, C20
St. 3 (3)	0	AS BELDW	\sim	AS BELOW	45	H1		3200.0	128.000		C17, C18, C19, C20
9.6 m	e,				46	HE	304	1550.0	471.200	51	C17, C18, C19, C20
St. 2 (Transfer Level) 6.4 m V	3.2 m	AS BELDW	\geq	AS BELOW							

Step 1. Try creating one of each of the available Schedules.

27.4 Slab Detail Drawings

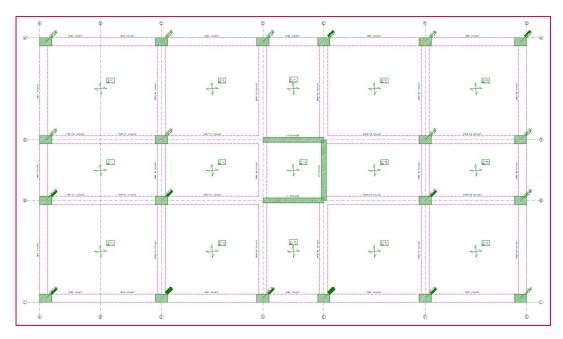
The Drawings group on the Draw tab provides options to create General Arrangements and Slab Detailing. These commands are only available when a 2D plan view is active.

27.4.1 General Arrangements

The General Arrangement command will create either a Plan, Section or Elevation of the active plan view or frame view. These typically include information such as Detailing Group labels, element section sizes and gridlines, and simply show the general layout information for the plan or frame view in question.

As with the other detail drawings and schedules, there are various options and settings relating to the General Arrangements in the Types, Layer Styles and Options areas of the Drawings Settings.

After clicking General Arrangement command, you will see the same DXF Export Preferences window as mentioned earlier, with the Type and Layer Style options, and clicking OK will generate the drawing.



Step 1. Try creating some General Arrangements of some plan and frame views

27.4.2 Slab Detailing

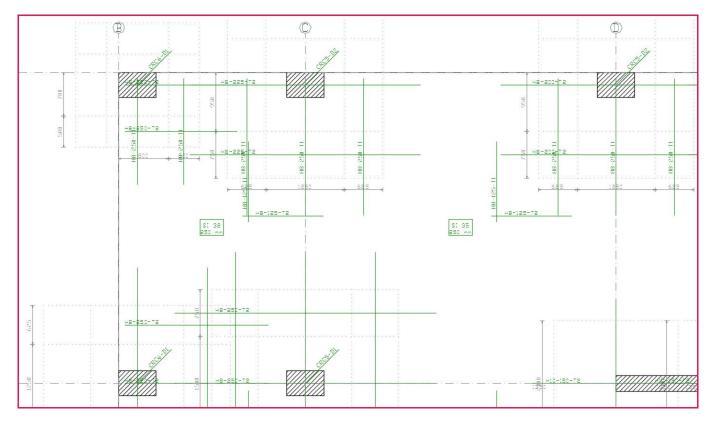
The Slab Detailing command will also create a Plan drawing of the active plan view. They will provide information on the reinforcement provided in the slab panels and patches, along with the patches themselves and dimensions. Therefore, the slab design process must be completed as required before generating the Slab Detailing drawings.

There are also various options and settings relating to the Slab Detailing in the Layer Configurations, Layer Styles and Options areas of the Drawings Settings.

After clicking Slab Detailing command, you will see the same DXF Export Preferences window as mentioned earlier, along with the Layer Configuration and Layer Style options.

The default Layer Configuration option in this window is Slab Detail (All) and will include information on all top and bottom reinforcement. Depending on your model, this could results in a cluttered drawing, so it would likely be more useful to select either the Slab Detail (Top Reinforcement) or Slab Detail (Bottom Reinforcement) options. Clicking OK to this window will generate the Slab Detailing drawing.

- Step 2. Try creating some Slab Detailing drawings for the floors that have already been designed in this model.
- Step 3. Try using some of the different Layer Configuration options to see the differences between the drawings that are created.



27.5 Drawing Management

If you'll want to generate detail drawings of multiple elements in one go, you can use the Drawing Management command on the Draw tab. In this window, the first thing to do is select the Drawing Variant.

27.5.1 Concrete Member Detailing

The Drawing Variant Concrete Beam Detail and Concrete Column Detail can be used to generate detail drawings of multiple frame elements at once. The easiest way to create these drawings is to use the Generate command.

This will populate the Available Drawings list with some standard drawings and their content.

The first drawing will be for the Typical elements in the model, which will contain multiple drawings, one for each of the Detailing Groups that contain more than one element.

There will also be some additional Available Drawings which will contain drawings for any remaining elements that are in Detailing Groups on their own.

Alternatively, you can generate completely custom drawings by clicking the Add button.

Drawing Va	ariant			ОК
E. S. S. S. S. S. S.	Beam Detail	-		Cancel
Available D	rawings			
Name		Last Revision	Add	
Typical Be St. 1 (1)	ams	Not generated yet Not generated yet	Remove	
St. 2 (Tra	nsfer Level)	Not generated yet	Generate	
			Reset ALL Marks	
Selected I Name	Drawing Typical Beams		Content	
Status:	Not generated y	et	Layout	
Items:	5		Loading	
			View Drawing	
			Create Revision	
			History	

You can check the information contained in each of these drawings by selecting them in the Available Drawings list, then clicking the Content... button.

This will open the Drawing Content window, listing any Available Members that could be included on the left hand side, and any elements or detailing groups already contained in the Drawing on the right. You can then control this content by left-clicking and dragging elements or groups from one list to the other. Once you're satisfied with the content, clicking OK button will confirm it.

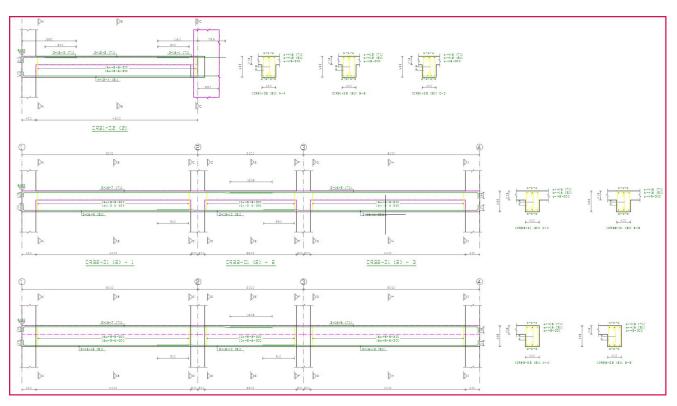
Available members	Drawing			ОК
Show unassigned members only	Variant	Concrete Beam Detail	×	Cancel
Show all members	Drawing	Typical Beams	*	
Group by By group 👻				
Concrete Reinforced Beams		e Reinforced Beams		
≟- CRB2 ≟- CRB2-D1	É- CRB	1 CRB1-D2		
1817 1819				
CRB2-D2		CRB1-D3		
		185 186		
		- 189		
		1B10 CRB1-D4		
	E.	1B13		

The Layout... button in the Drawing Management window will allow you to control the layout of the elements contained within the drawing.

awing Layout	
Direction O Horizontal O Vertical	OK Cancel
Arrangement Linear Grid	

To view a drawing, select it in the Available Drawings list and then click the View Drawing... button.

The same DXF Export Preferences window mentioned earlier will then appear, including the same Type and Layer Style options.



Clicking the OK button will then create the drawing and display it in an appropriate CAD program.

- Step 1. Try generating the standard Available Drawings for the Beams, Columns and Walls and review their content and layout settings.
- Step 2. Use the View Drawing command to create the drawings.

In the Drawing Management window, the Selected Drawing area displays information about the currently selected Available Drawing. This will show the current Status of the drawing, how many Items the drawing contains, and gives you the ability to define the Name for the drawing.

Drawing Variant		0
Concrete Beam Detail		Can
vailable Drawings		
Name	Last Revision	Add
Typical Beams	Not generated yet	Remove
St. 1 (1) St. 2 (Transfer Level)	Not generated yet Not generated yet	Generate
		Reset ALL Marks
Selected Drawing		Reset ALL Marks
Selected Drawing Name Typical Beams		Reset ALL Marks
	yet	
Name Typical Beams	vet	Content
Status: Not generated	vet	Content

If you decide to make amendments to the drawings you've created but want to keep the originals, or if you just want to make different versions of the same drawings, you can click on the Create Revision... button.

This will open the Create Drawing Revision window, which will provide similar options to the DXF Export Preferences window. It also allows you to adjust the name of the drawing file that will be created, where it will be saved and define the Next Revision Name, which automatically adds extra text to the automatic file name. There is also an option to add Notes to the revision to help clarify the differences between the existing revisions.

Revision			ОК
Previous revision name	<no revision="" yet=""></no>		Cancel
Next revision name			Cancer
Note			
Output			
Use automatic file name	.oc\Typical Beams Rev. A.d:	d	
✓ Use automatic file name C:\Users\jesan\AppData\	.oc (Typical Beams Rev. A.d: Beam Detail	xf •	
Use automatic file name C: \Users\jesan\AppData\L Drawing		xf •	
Use automatic file name C:\Users\jesan\AppData\u Drawing Layer Configuration	Beam Detail	•	

To view the revision history of a drawing, simply click the History... button in the Drawing Management window.

Step 3. Try adding and creating a custom Available Drawing.

Step 4. Try making some revisions and view the revision history.

27.5.2 Other Drawing Management Options

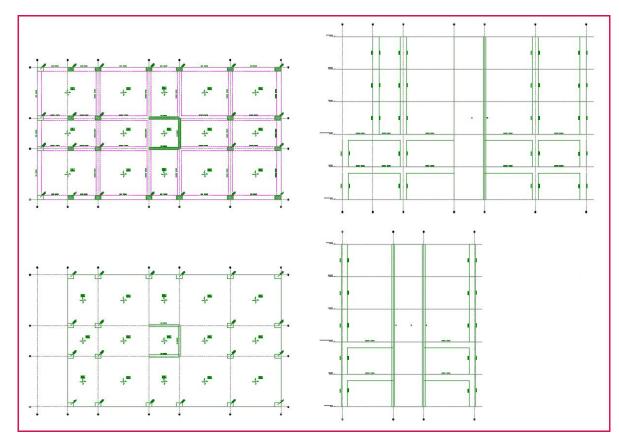
The other Drawing Variants available in the Drawing Management window do not have a Generate option to create standard Available Drawings. Therefore, the only way to create the drawings for these options is to use the Add button. Other available variants are:

Drawing Variant			ОК
General Arrangement	-		Cancel
General Arrangement			Cancer
Foundation Reactions Loading Plan Concrete Beam Detail Concrete Column Detail Concrete Wall Detail Slab Detail	n	Add Remove	
Non-concrete Beam Detail Non-concrete Column Detail Beam End Forces Base Detail Foundation Layout			

Once the Add button has been clicked, the drawing will exist, but it will contain no items.

Therefore, you need to review the Drawing Content window to add the required items to it, as discussed earlier. You can create a number of different Available Drawings for each Drawing Variant, and all other commands in this window can be used in the same manner as for the automatically generated drawings.

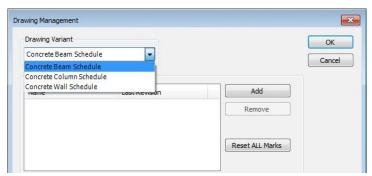
- Step 5. Try creating some custom drawings for the other Drawing Variants.
 - § General Arrangement plan views of St.2 and St.5
 - § General Arrangement frame views of Frame 2 and Frame E



27.5.3 Schedule Management

You can use Schedule Management to create custom schedules for columns, walls and beams.

You can create a number of different Available Drawings for each Drawing Variant, and all other commands in this window can be used in the same manner as Drawing Management.



Step 6. Try adding the standard Available Drawings for the Concrete Beam, Column and Wall Schedules.

28 RC Members Design Reports

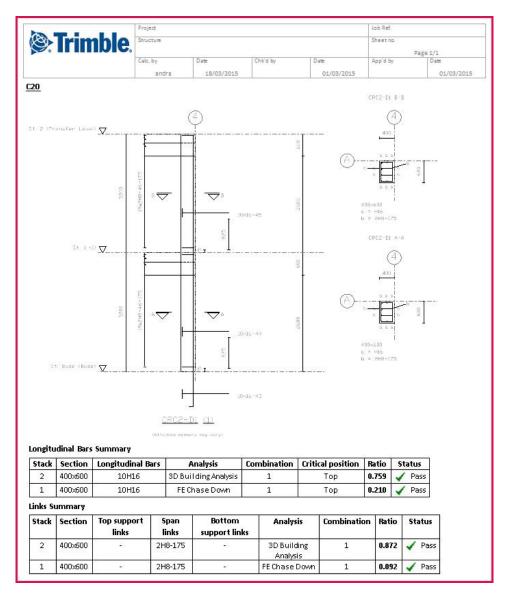
Once members have been designed, there are a variety of different Design Reports that can be generated in TSD. These can be in-depth or summaries, and for either individual frame elements, multiple elements or the model as a whole. The majority of these reports can be generated from the Report tab or from the right-click menu.

28.1 Individual Member Design Reports

To quickly generate a report for an individual frame element, you can simply right-click over the element in question in an appropriate scene view, and then choose Report for Member.

This works the same way for columns, walls and beams.

As default, the report will be a summary report, and will typically include a sketch and a brief overview of the design status and reinforcement provided.



Step 1. Try using the Report for Member command for a beam, a column and a wall.

28.1.1 Report Content

The content of the report generated using the Report for Member command is dictated by the Member Report... command, found on the Report tab.

To change the content to be included in the report, you should first select the appropriate Member Type option in the top left corner. You will then be able to see the Available Styles for that particular member report. Selecting one of the Available Styles in this list will then display the content of that report in the Report Structure list.

You can then add or remove information from the selected report style by clicking and dragging items between the Report Structure list and the Chapters and Options list.

Report Contents			×
Member Type Concrete Column Available Styles Concrete Column Summary (active) Concrete Column Design Add Remove >> Active Active Style Name Concrete Column Summary	Chapters and Options: Drag chapters and options from left to right, which are to be included in the report. Picture Picture Loading Loading Combinations Enclopes Enclopes Enclopes Bending Moment Diagram Design Calculations Bending Moment Diagram Combinent Diagram Design Notes Design Notes	Report Structure: Drag selected chapters up and down, to change order.	OK Cancel View Mode Flat Hierarchical

As you can see above, there could be numerous Available Styles for particular member reports. The default is usually the Summary version of the report.

If you want a more detailed design report when you select the Report for Member command, you can select a different report style, and then click the Active button.

The Active report style is the one used to generate the report. If you have already generated a particular report, then change the Active style, the existing report should automatically update to include the updated style content.

Once listed in the Report Structure list, you can also control the level of information displayed in the report by right clicking over the items.

There will be options to make an item Active or not (i.e. displayed in the report), you can Remove Items from the report altogether, and you could change the Settings for individual items – these setting could include the type and style of drawing to be included or the level of design calculations displayed.

- Step 1. Click the Member Report... command on the Report tab.
- Step 2. Make sure Summary styles are Active for the Concrete Wall and Concrete Beam options, and the Full Design report style is Active for the Concrete Column option.
- Step 3. For the Concrete Column option, right click over Design Calculations in the Report Structure list and choose Settings.

- Step 4. Make sure the Design Calculations Level is set to 1, then click OK to confirm these settings this will increase the amount of detail shown in the design calculations in the report.
- Step 5. View the updated existing column member report to view the changes in content levels.

28.2 Model Reports

You can also create reports for multiple elements at once using one of the standard Model Reports.

To generate a Model Report, simply choose an option from the Select drop down list on the Report tab. Once a report has been selected from this list, clicking the Show Report command will then generate the report and allow you to view its contents.

The report Member Design Calcs will allow you to view the same design reports generated earlier using the Report for Member option, but for all elements in the model in a single report.

The amount of output given for each individual member in this report is controlled by the same Member Report window also discussed earlier.

The Model Report command can also be used to control which element types are included in the report, and works in a very similar manner to the Model Report window.

- Step 1. Select the Member Design Calcs option in the Select drop down list.
- Step 2. Review the Model Report Contents window to make sure only reinforced concrete beams, columns and walls are included, then click OK to confirm, as shown below.

Available Styles Solver Model Data (active) Building Loading Building Loading Building Design Material Listing Beam End Forces Foundation Reactions Salemic Design Member Design Calcs Add Remove >> Active Active Style Name Member Design Calcs	Chapters and Options: Drag chapters and options from left to right, which are to be included in the report.	Report Structure: Drag selected chapters up and down, to change order. Concrete (Structure) Beams (Structure) Columns (Structure) Columns (Structure) Walls (Structure) Walls (Structure)	OK Cancel View Mode Flat I Hierarchical
---	---	---	---

Step 3. Click Show Report command to generate the report, then review the content.

As you can see, creating a Model Report and editing the content could result in very large reports being generated, especially if you have a large model with lots of members.

Therefore, the Filters options on the Report tab can be very useful to make sure only key required information is contained in the reports generated.

For example, clicking the Members Filters button on the Report tab allows you to Add a filter to the list of available filters. Once a filter is added, all frame members – i.e. beams and columns – will be listed on the right hand side of the window, allowing you to tick the members you want to include in the report.

To do the same for walls, you need to use the Frames Filter.

Once the filter is set up, you then go back into the Model Report window, right click over the appropriate entry in the Report Structure list and choose Model Filter > Selected Members.

The report will then automatically update after clicking OK.

				1 281		
port Structure: ag selected chapters up and down, to change OK ler. Cancel				Date 01/03/2015		
E Concrete (Structure)		ancel				
E- Columns (Selected	 Active 					
Walls (Structure)	Model Filter	•		Structure		
	Style	۲	~	Selected members		
	Remove Item			Edit\New		
-				Reset		

- Step 4. Add a Members Filter to the newly created Member Design Calcs report.
- Step 5. Reduce the size of the report by only including a few of the beams and columns, then click OK, apply the filter and update the existing report.

28.3 Other Report Options

Although we have only discussed the Design Member Calcs Model Report, there are various others that are available which could also be of use, and their content and layout can be controlled in exactly the same way as already discussed. Another useful design report is the Building Design option, which essentially shows a design summary for all frame elements, as well as the slab panels, patches and punching checks.

As with any report generated in TSD, you can use the various commands on the Report tab to adjust how the report is displayed in the program, edit the layout and labels used in the Header and Footer, navigate through the reports, and Export the reports into various other programs and formats.

29 Design Groups

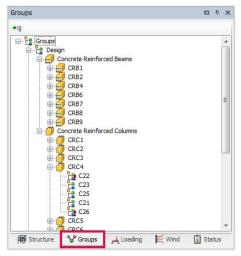
Step 1. Open the model TSD Concrete Design Fundamentals Model 2 – Design Groups.tsmd.

To allow for a quick and efficient design process, all concrete beams and columns are by defaults, automatically placed into Design Groups.

Design Groups have no specific engineering data associated with them and are simply a unique name for collection of similar frame elements.

Various criteria are used to determine which group a particular element will be automatically placed in, but they can only belong to a single design group.

For example, if you have a number of columns in a model, each with the same number of stacks, similar stack lengths, the same concrete and reinforcement material properties, and the same section sizes, rotations and alignments in the corresponding stacks, then they will likely all be placed in the same group.



If any of these options vary between the columns, then they will be placed in multiple groups.

The Design Groups are formed as you build up the model, and can be viewed using the Groups tab in the Project Workspace. This window also allows you to do a variety of other things, including editing the groups and re-grouping the elements, which will be discussed shortly.

Note that concrete walls are designed independently from each other and do not use design groups.

Step 1. Go to the Groups tab in the Project Workspace and view the Design Groups for this model.

29.2 The Design Procedure for a Design Group

When a design group is designed, either as part of the Design Concrete (All) process or when a single group is designed, there are a series of steps that are completed:

- One element in the design group is arbitrarily selected, and will have a full design completed on it for all of the results generated, by all analyses that have been undertaken.
- The reinforcement selected and placed in this first element will then also be applied to all other elements in that design group, so that they all have the same reinforcement arrangement.
- One of these other elements will then have a check design completed on it, to make sure the reinforcement is sufficient for it.
- If the reinforcement is sufficient, then it will move on to the next column and check the reinforcement for that element, and so on.

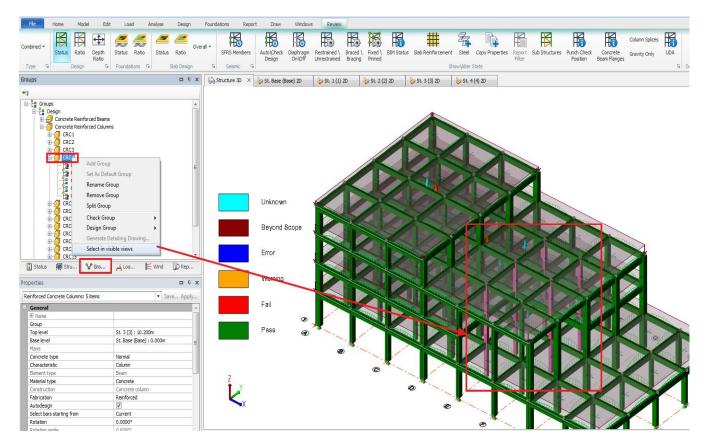
- If an element requires more reinforcement, then it will be increased and then the updated reinforcement will be applied to all other elements in the group.
- The process will then continue until all elements in the group have all been checked, resulting in them all having the same reinforcement arrangement.
- Once all elements have been designed in the group, a final check procedure will work out the utilisation ratios for all elements in the group to determine the 'critical' element, and an average utilisation ratio will be calculated for the purposes of reviewing group efficiency.
- 29.3 Working with Design Groups

As mentioned earlier, when the design of all frame elements is completed on a model, it will design all groups in the model. For a lot of the time, especially with fairly regular models, these automatically-created design groups are going to be more than sufficient, and will not need to be changed. However, the Groups tab has a number of features that allow you to interrogate, modify and re-design specific groups in the model.

29.3.1 Interrogating the Design Groups

Expanding the entries in the Groups tab allows you to see a full list of all the design groups that have been generated by the program, and the elements that make up each group.

If you want to locate all the members that are in a particular design group, ensure you're viewing an appropriate scene view, usually the Structure 3D view, then simply right click over the Design Group name in the Groups tab, and choose Select in visible views.



The same option is also available for locating the individual elements listed underneath the Design Group name.

The Critical element(s) in the Design Group are highlighted in the Groups tab with a red exclamation mark (!), so you can use the above command to find all elements in a particular Design Group and then the critical element from that group.

Note that there may be more than one critical element in a group, as there will be one marked for the critical shear and one for the critical bending.

- Step 1. Set the Structure 3D to be the active scene view.
- Step 2. Try selecting some different beam and column design groups and individual elements from those groups in the model.
- 29.3.2 Renaming Design Groups

When design groups are generated as you build up the model, they are automatically assigned group names. For beams, these will be CRB followed by a number, and for columns, it will be CRC and then a number.

To give them more meaningful names, you can manually re-name groups by right clicking over the Design Group name in the Groups tab, and choose Rename Group. You can then type in a new name, and press Enter to confirm. This will also update the related Detailing Groups names as well.

Step 3. Try selecting some design groups and then use the Rename Group command to assign more meaningful names to them.

29.3.3 Splitting Design Groups

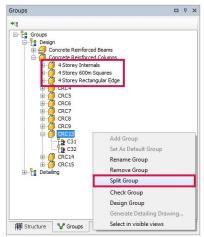
Although the automatically formed design groups will usually be perfectly suitable for the majority of the time, you may occasionally find a group made up of dimensionally very similar elements, but which have quite different forces acting on them. This could result in some of these elements having more reinforcement placed in them than they actually need for the design.

In this case, you may decide to use the Split Group command.

Doing this will place all members from the group being split into their own separate groups, effectively allowing them to be designed as ungrouped.

Groups are split by right clicking over the Design Group name in the Groups tab, and choosing Split Group.

Step 4. Try splitting the column design group CRC13 to form two new groups containing only one element each.

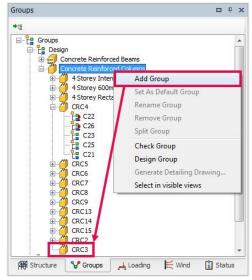


29.3.4 Adding and Modifying Existing Groups

For similar reasons to splitting groups, you may decide to create a new group and then move elements from the automatically-generated groups into the new group.

To add a new group, simply right click over one of the Design Group sub-headings, either Concrete Reinforced Beams or Concrete Reinforced Columns depending on the type of group you want to create, and then choose Add Group. This will create a new, empty design group inside that folder, as well as a corresponding Detailing Group.

To add elements to that new group, simply click and drag the element name from inside one group onto the new group folder. TSD will prevent you from moving an element into a group that contains elements that can't be grouped together, such as ones with different section sizes.



In this model, the Concrete Reinforced Column design group CRC4 contains a number of elements. The designs for these elements are obviously the same, but column C26 has some large loads applied on and near it, resulting in much higher moments in this element, which in turn is dominating the bending design for the whole group. Therefore, C26 could be moved into a new, separate group to be designed independently from the other elements.

- Step 1. Make sure the design results for the whole model are up to date.
- Step 2. Right click on C26 and choose Check Member to see an overview of the reinforcement selected for all elements in the group, and do the same for another member in this group to confirm.

Stack	Section	Longitudinal Bars	Analysis	Combination	Critical position	Ratio	Status
3	400x400	8H12	FE Chase Down	1	Тор	0.672	🖌 Pass
2	400x400	8H20	FE Chase Down	1	Bottom	0.845	🖌 Pass
1	400x400	12H25	FE Chase Down	1	Bottom	0.835	🖌 Pass

Step 3. Use the Add Group command to create a new Concrete Reinforced Column design group. Step 4. Click and drag column C26 from group CRC4 into the newly created group.

29.3.5 Designing Individual Groups

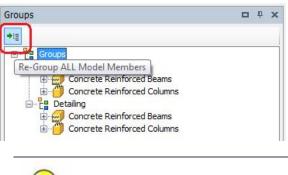
Instead of repeating the full analysis and design process again, individual design groups can be designed to take into account their updated contents. This is done by right clicking over the Design Group name and choosing Design Group. You can then check the designs of the members again to see the differences.

Step 5. Use the Design Group command to re-design CRC4 and the newly created group.Step 6. Use the Check Member command to compare the designs between the two.

Stack	Section	Longitudinal Bars	Analysis	Combination	Critical position	Ratio	Status
3	400×400	8H12	FE Chase Down	1	Bottom	0.185	🖌 Pass
2	400x400	8H12	Grillage Chase Down	1	Тор	0.286	🖌 Pass
1	400×400	8H12	Grillage Chase Down	1	Bottom	0.448	🖌 Pass

29.3.6 Re-Group All Model Members

If you have edited the Design Groups in a model and want to go back to the original groups created by TSD, you can click the Re-Group ALL Model Members button, located in the top left corner of the Groups window.



If you click this button by mistake, the Undo command will revert back to your edited groups.

216 (257)

30 Analysis and Design of Steel Structure

In this session we will take a detailed look at the analysis and design of a steel structure.

Step 1. Open the example model file 1_Design_Start.tsmd.

30.1 Combination Classes

In a typical model there can be many different combinations. These could be classified as construction stage, gravity, lateral, seismic, seismic RSA or vibration.

ombinations		Design Combination Title	Class		Active	Strength	Service	Can
 1 Manual Example 2 STR,-1.35G+1.5Q+1.5RQ 	1	Manual Example	Gravity	4		2	2	
 3 STR_{4.1}-1.35G+1.5Q+1.5RQ+EH#_{Drim} 	2	\$78,-1.356+1.5Q+1.5RQ	Gravity					1 march
- 4 STR ₄₁ *1.35G+1.5Q+1.5RQ+B4F ₂₀₅ - 5 STR ₄₁ *1.35G+1.5Q+1.5RQ+B4F ₂₀₅	3	STR 1:1.35G+1.5Q+1.5RQ+EHFDrip	Gravey		1	•	2	Ad
- 6 STR_+-1.35G+1.5Q+1.5RQ+DHP_0	4	STR42*1.35G+1.5Q+1.5RQ+EH#p-b	Lateral Seismic		2	2	8	Cay
- 7 STR4.1-1.35G+1.5Q+1.5v(S+1.5v(W+D)F2		STR ₈₅ 1.35G+1.5Q+1.5RQ+EHF _{brds}	Seismic RSA		1	•		Dek
- 8 STR _{4.2} ·1.35G+1.5Q+1.5v ₂ S+1.5v ₂ W+EHF ₂ - 9 STR _{4.2} ·1.35G+1.5Q+1.5v ₂ S+1.5v ₂ W+EHF ₂		STR_1+1.35G+1.5Q+1.5RQ+EHFore	Vibration Mass	_	1	2	2	
- 10 STR 1.35G+1.5Q+1.5w,S+1.5w,W+D+		STR4_11.35G+1.5Q+1.5w(S+1.5w)W+EHRprin	Laberal	¥				
11 STR _{8.1} -1.35G+1.5v ₂ Q+1.5v ₂ S+1.5W+EH 12 STR _{8.1} -1.35G+1.5v ₂ Q+1.5v ₂ S+1.5W+EH	8	STR4_11356+1.5Q+1.5u;5+1.5u;W+EHP	Lateral	~	9	2	2	
 12 STR_{4.2}-1.35G+1.5Q₁Q+1.5Q₂S+1.5W+D⁻⁰ 13 STR_{4.2}-1.35G+1.5Q₂Q+1.5Q₃S+1.5W+D⁻⁰ 	9	STR4.c1.35G+1.5Q+1.5wjS+1.5wjW+D+P_3%	Lateral	v	2	2	8	
- 14 STR4+1.35G+1.5u2Q+1.5u2S+1.5W+EHI		STR4+1.356+1.5Q+1.5u;8+1.5u;W+EHFpo	Cateral	*	2	2	2	
- 15 STR ₄₁ -G+1.5W+EHF _{D+1+} - 16 STR ₄₁ -G+1.5W+EHF _{Incta}	11	STR4.c1.35G+1.5v;Q+1.5v;S+1.5W+EHPp-p-	Lateral	¥.	1	2	8	
- 17 STR _{4.7} G+1.5W+EHF _{Dro}	12	STRe: 1.356+1.5v;Q+1.5v;S+1.5W+EHFpra-	Lateral	¥	9		2	
- 18 STR4G+1.5W+EHFp.p	13	STR4.:1.35G+1.5v;Q+1.5v;S+1.5W+EHFpm	Lateral	v	2	2		
	14	STR_4:1.35G+1.5v;Q+1.5v;S+1.5W+D+Fp+	Lateral	¥		1		
	15	STR _{4.2} -G+1.5W+EHF ₂₄₅₀	C Lateral	*	1	2	2	
	16	STR42'G+1.5W+EHP3rds	Laberal	¥	1	2	8	
	17	STR _{4.2} ·G+1.5W+EH# _{D-0}	Laboral	¥	1	2	2	
	13	STR++G+1.5W+D+F	Lateral	v	2	2	2	

A gravity combination is a combination that contains only gravity loads, whereas a lateral combination is one that include lateral effects such as equivalent horizontal forces (for lack of fit) and/or wind loading.



Combinations are classified in this manner to streamline the subsequent member design process.

Step 2. Review the combination classes via the Combination command in the Load tab.

30.2 Properties

The properties to be assigned to a new entity such as a steel column, steel beam or steel brace are displayed in the Properties window prior to creating the entity. These properties can also be reviewed and amended at a later date for selected entities via the Properties window.

Two important design properties that streamline each individual member design process are the Gravity only and Autodesign options.

30.2.1 Gravity only

The Gravity only option has two settings;

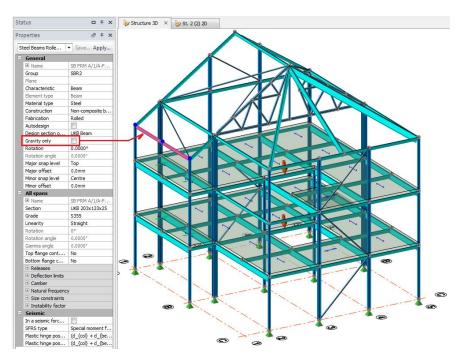
- Ticked designed for gravity combinations only.
- Unticked designed for both gravity and lateral combinations.

teel Beams Rolled Non-Compos	site: 1 items 🔹 Save Apply.					
General						
Name	SB 1/A/1-1/B/1					
User name						
Group	SBR4					
Plane	St. 1 (1) : 4,000m					
Characteristic	Beam					
Element type	Beam					
Material type	Steel					
Construction	Non-composite beam					
Fabrication	Rolled					
Autodesign						
Design section order	UKB Beam					
Gravity only						
Rotation	0.0000°					

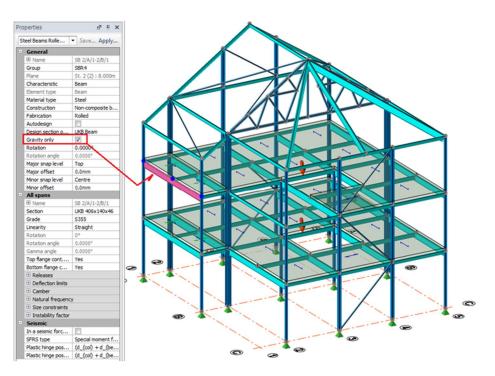
Two examples are provided below to understand why this setting might be checked.

Non-composite or composite beams within a rigid floor diaphragm are generally only designed for gravity loading - since the rigid floor diaphragm deals with any lateral load, distributing it directly to the vertical stability system. By classifying the beam as gravity only, it will only ever be subjected to gravity combinations, speeding up the autodesign or check design process (as appropriate) and reducing the amount of report output.

Columns that are not part of the vertical stability system (braced bays) and thus not being considered to resist lateral load could be set as gravity only. These columns would only ever be subjected to gravity combinations, speeding up the autodesign or check design process (as appropriate) and reducing the amount of report output.



Step 3. Select the below entities and review their Gravity only status.



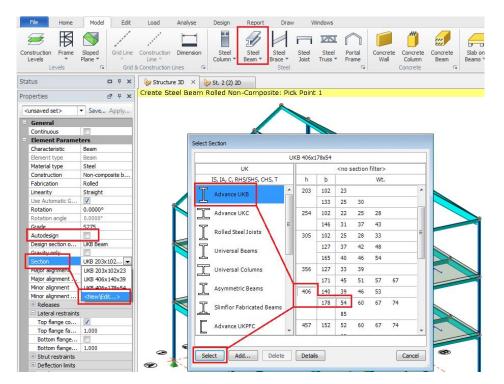
Note that the 1st and 2nd floor beams are all ticked hence they will be designed for gravity only combinations, with the exception of SB 2/B/2-2/C/2 and SB 2/C/2-2/C/3 to the perimeter of the void and the beams forming the moment frame.

All other entities that are unticked they will be designed for both gravity and lateral combinations.

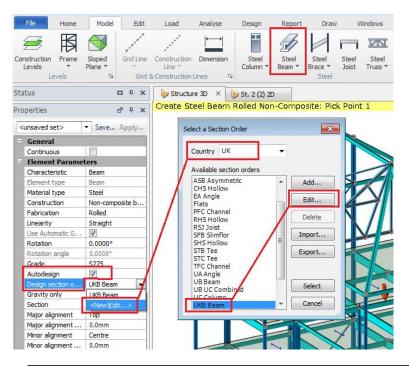
30.2.2 Autodesign

The Autodesign option specifies how you want the entity to be designed. The option has two settings:

• Unticked - the specified section will be checked during the design process for the section you have specified.



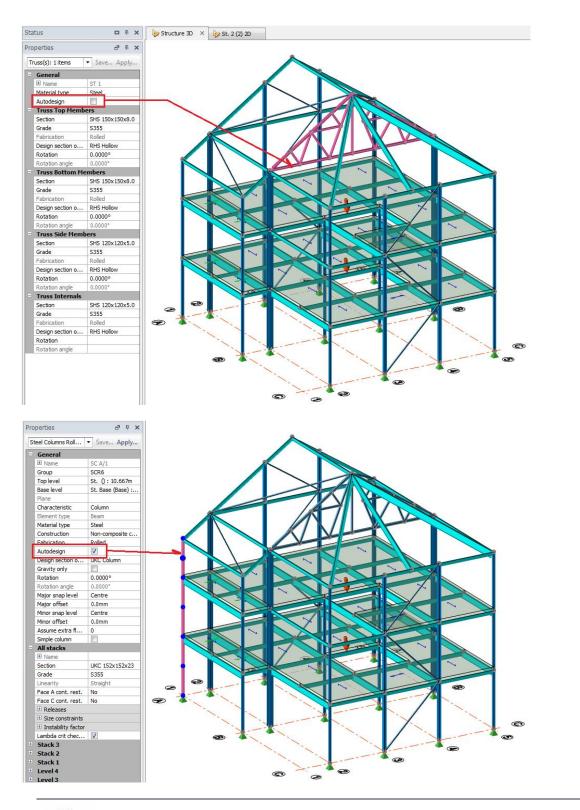
Ticked - sections from the design section order will be considered during the design process to auto size the entity.



Design Section Order - Specifies the order file from which the designed sections will be selected. i.e. a design order file comprising UKB sections, RHS sections etc. with increasing capacity. This order file can be amended so that only the sections you want to consider are included in the list.

Properties					OK
Name	UKB Beam			-	Cancel
Country	UK	Ŷ			
Sections					
Section Group	Advance UKB	8	v		
Available Sectio	ns		Sections in use		
-		Add Selected	✓ UKB 203x133x25	^	
		Add All	UKB 203x133x30 UKB 254x146x31		
		Remove Selected	 ✓ UKB 406x140x39 ✓ UKB 305x165x40 		
			✓ UKB 356x171x45 ✓ UKB 406x140x46		
		Move Up	UKB 356x171x51		
		Move Down	UKB 457x152x52		
		Sort By	UKB 406x178x54 UKB 457x152x60		
		Depth v	UKB 457x191x67		
			UKB 457x191x74		
		Sort	UKB 533x210x82		
		Keep sorted	UKB 533x210x92		
		and the second s	UKB 610x229x101	~	

Step 4. Select the below entities and review their Autodesign status.





All other entities that have the Autodesign option "ticked" and will therefore be sized based on the designated order file (UKC for columns and UKB for beams).

30.3 Design Options

TSD combines both the analysis and design into a single process via the Design tab. In the session you are primarily concerned with the design of steel. You will therefore concentrate on the Analysis and Design Steel group.

Model	Edit Load	Analyse	Design	Report Draw	v Windows			
*/	1		*/	/		*	1	-11/2
Design Stee (Gravity)	Design Steel (Static)	Design Steel (RSA)	Design Concrete (Gravity)	Design Concrete (Static)	Design Concrete (RSA)	Design All (Gravity)	Design All (Static)	Design All (RSA)
Ana	ysis & Design St	eel 🚺	Analy	sis & Design Concre	ete 🕼	Ana	lysis & Desig	n G

Analysis is still available via the Analyse tab and you will review the analysis results later via the Review view.

Prior to undertaking a design you need to review the Design Options which control how the design process will run.

Step 1. Click the Options command in the Design tab.

Step 2. Review the various folders down the left side of the dialog. These are discussed below.

30.3.1 Analysis

The Analysis options on this page allow you to choose whether a first or second order 3D Building Analysis is performed when any of the Design (Gravity), Design (Static) or Design (RSA) commands are run. An option is also provided to include sway stability analysis.

The analysis type you wish to run can be selected and will either consider geometric (P-delta) nonlinearity (second order effects) or not - dependent upon the option selected.

Anatysis	Analysis	OK
 Concrete Design Forces Design Groups 	● First-order (Elastic) analysis [Suitable when second-order effects are small enough to ignore (α _σ ≥ 10) or when they are not under consideration.]	Cancel
 Autodesign Display Limits Steel Joists 	○ Second-order (P- Δ) analysis - BS EN 1993-1-1 Cl 5.2.2 (6)B - Amp. forces method [Suitable for second-order effects for buildings where $\alpha_{cr} \ge 3.0.$]	Save
Sceer Joisis	\bigcirc Second-order (P-Δ) analysis [Suitable for all buildings with α_{or} ≥ 2.0.]	Load
	☑ Indude sway stability analysis	
	Note if non-linear members/supports exist, a non-linear analysis will automatically be run as appropriate during design	
	Auto-k _{amo} formula	

What is Geometric (P-delta) Non-linearity?

In first-order analysis the stiffness of the structure is assumed to be constant and unaffected by changes in the geometry of the structure when it is loaded. The internal forces (and displacements) are evaluated in relation to the undeformed structure.

This is the standard assumption of linear–elastic first order analysis.

In the case of first order linear-elastic analysis, the deformations (and internal forces) are proportional to the applied loads, and as a consequence the principle of superposition of effects can be used to simplify the analysis.

In second-order analysis, the effective stiffness of the structure is changed by the action of the loads upon it. The principles of superposition do not apply, as effects of actions interact.

Second-order effects, often called geometric or P-delta effects, are commonly illustrated by considering the additional displacement, forces and moments which arise from the application of actions on a deflecting structure.

A second-order analysis considers these geometric (P-delta) second order effects.

First-order (Elastic) Analysis

Runs a linear static analysis. Suitable where geometric (P-delta) secondary effects are negligible and can be ignored.

Note that if the model contains any material non-linearity such as tension or compression only elements, non-linear spring supports or non-linear axial or torsional springs then the design will automatically run the appropriate non-linear analysis – in this case 'First-order non-linear' in order to consider material non-linearity.

- First-order (Linear-Elastic) Geometric non-linearity *No*, Material non-linearity *No*.
- First-order (Non-linear) Geometric non-linearity *No*, Material non-linearity *Yes*.

Second-order (P-delta) Analysis – Amp. Forces Method

This is not a true second-order analysis and runs a linear static analysis and applies an amplification factor to the linear analysis to approximate for geometric (P-delta) secondary effects via an amplification factor.

Note that if the model contains any material non-linearity such as tension or compression only elements, non-linear spring supports or non-linear axial or torsional springs then the design will automatically run the appropriate non-linear analysis – in this case 'First-order non-linear' in order to consider material non-linearity.

- First-order (Linear-Elastic) Geometric non-linearity *No*, Material non-linearity *No*.
- First-order (Non-linear) Geometric non-linearity No, Material non-linearity Yes.

Second-order (P-delta) Analysis

This is a true second-order method and runs a second-order linear analysis which automatically takes account of geometric (P-delta) secondary effects as part of the analysis.

As with all true second order approaches it will only provide an analysis for a structure that does not collapse under the design loading. Hence care is required when using this approach to ensure a stable model.

Note that if the model contains any material non-linearity such as tension or compression only elements, non-linear spring supports or non-linear axial or torsional springs then the design will automatically run the appropriate non-linear analysis – in this case Second-order non-linear' in order to consider material non-linearity.

- Second-order (linear) Geometric non-linearity Yes, Material non-linearity No.
- Second-order (Non-linear) Geometric non-linearity Yes, Material non-linearity Yes.

Table of Analysis Options and Non-linearity consider

Analysis Type	Non-Li	nearity
	Geometric	Material
First-order linear (elastic)	No	No
First-order non-linear	No	Yes
First-order Vibration	No	No
Second-order linear	Yes	No
Second-order non-linear	Yes	Yes
Second-order buckling	Yes	No

30.4 Design Steel (Gravity)

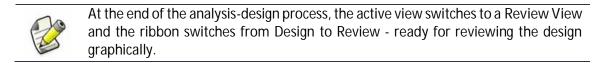
Design Steel (Gravity) is used for rapid design of the majority of steel members for gravity combinations.

The processes can be summarised as follows:

- 1. A first-order linear analysis is run *excluding pattern loading* to establish a set of design forces for the steel members. Each steel member with the member property '*Gravity only ticked*' is then either checked, or designed according to its Autodesign setting for Gravity combinations only.
- 2. 'Other' steel members with the member property '*Gravity only unticked*' and set to autodesign are designed for Gravity combinations, to give them a reasonable start size prior to consideration of the Lateral combinations.
- 3. On completion of the gravity sizing process all steel members follow the Design Options Autodesign rules and are either set to *check-design mode* (Reset *Autodesign...Always*) or retain the *auto design setting* (Reset *Autodesign...Never or When check status is at Worst*) dependent upon the option set.

30.4.1 How do I run Design Steel (Gravity)?

- First, check the individual member Autodesign property is correctly set for all members:
 - Autodesign '*ON*' new section sizes will be designed
 - Autodesign '*OFF*' existing section sizes will be checked
- Next, review the Design options and adjust if required:
- Click Design Steel (Gravity) to undertake the gravity design for the steel elements.



Step 1. Click the Design Steel (Gravity) command.

30.5 Check the Status

Always review the Status tab of the Project Workspace prior to accepting results in the Review view as this will indicate if you have any issues that need addressing.

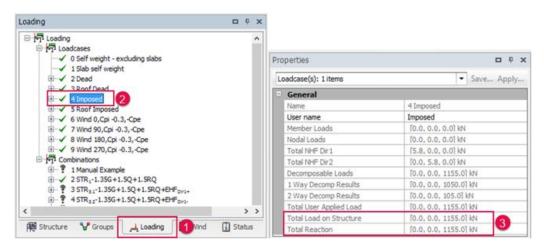
atus	□ ₽ 3
- ? Validation	
Model	
- ? Analysis	
General	
First-order linear	
E. Pirst-order non-linear	
🗄 🖗 🕈 Second-order linear	
🗄 🖗 🔋 Second-order non-linear	
🗄 🖞 🕅 🕅 🖞 First-order vibration	
🗄 🖷 👰 Second-order buckling	
🗄 🖷 🌹 Grillage chase-down	
🖽 🖤 🔋 FE chase-down	
🗄 🖤 🦹 Seismic vibration	
Design	
Model Geometry	
Wind Model	
Meshing	
Decomposition	
🕕 🕕 Solver	
🖃 🕕 First-order linear	
🖻 🕕 Model	
Solved Successfully	
🗄 🕕 Not Requested	
FE chase-down	
∃ 🚏 BIM	

Step 1. Review the Status tab for any issues.

If there are no issues reported - as you have here, then you can proceed. Otherwise you would need to investigate the issues raised here prior to looking at the design results.

30.6 Check the Loading Summary

Before preceding you should check that the total applied loads on the structure equal the total reactions.



Step 1. Review the Loadcases and Combinations in the Loading tree to ensure there is no lost load.

30.7 Review View

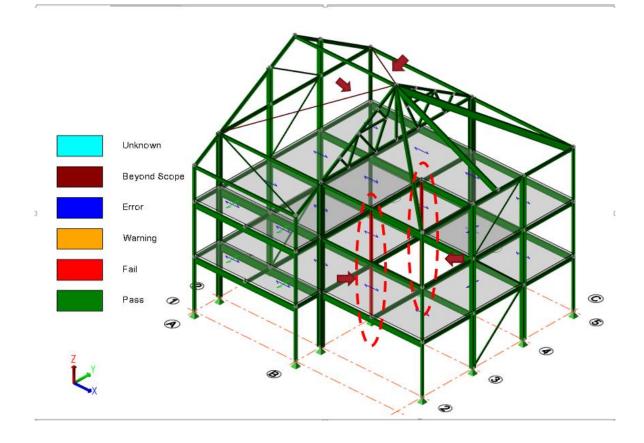
The Review View displays a colour coded version of the model so that the design status and various other parameters can be graphically interrogated and/or modified.

30.7.1 Design - Status

The design status shows graphically the steel elements design status.

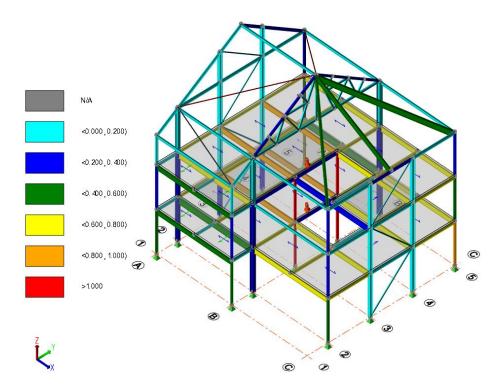
Step 1. Review your results after the Steel Design (Gravity) process – i.e. only carrying out a design on active combinations classed as gravity.

You can see the following; Two columns set with Autodesign unticked i.e. check design only - Fail. Two roof braces set with Autodesign unticked i.e. check design only - Fail.



30.7.2 Design - Ratio

The worst design utilisations ratio is displayed in colour coded form.

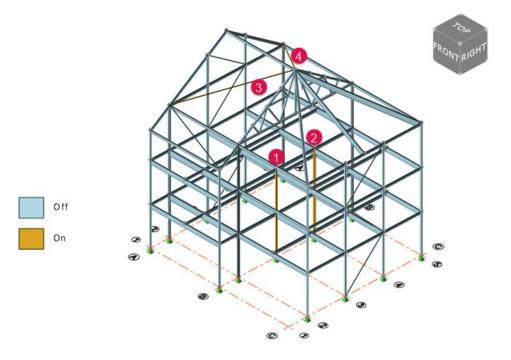


30.7.3 Show\Alter State – Auto\Check Design

The Auto\Check Design command provides a means to graphically assess and modify the Autodesign setting for all members in the model.

The two internal columns and roof braces were failing the gravity design previously so we will assign them the Autodesign status to re-size them.

Step 1. Use the Auto\Check Design view on the Review tab to set the two failing columns and roof braces to Autodesign.

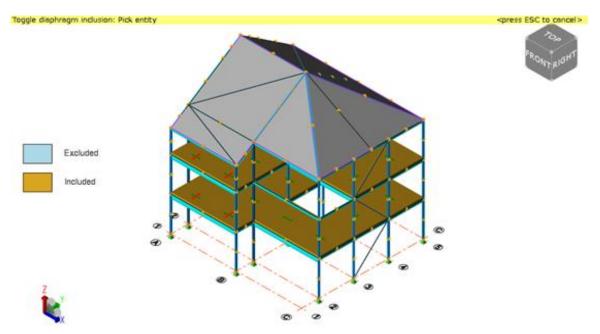


30.7.4 Show\Alter State – Diaphragm On\Off

The Diaphragm On\Off command provides a means to graphically assess and modify those slabs in the model that are constrained by diaphragms.

Each slab is colour coded to indicate it is Included or Excluded in the diaphragm.

- Clicking on an individual slab toggles its state between included and excluded.
- Dragging a box from right to left toggles the state for all slabs that are either enclosed by the box, or are cut by the box perimeter.
- Depressing the SHIFT key and dragging a line toggles the state for all slabs that cross the line.

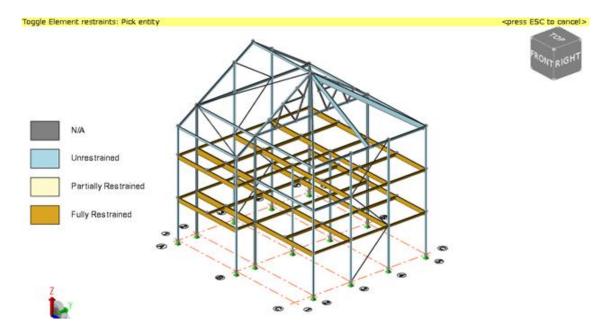


30.7.5 Show\Alter State - Restrained\Unrestrained

The Restrained\Unrestrained command provides a means to graphically assess and modify the restraint settings for all members in the model.

Each member is colour coded to indicate its restraints setting (N/A, Unrestrained, Partially Restrained, and Fully Restrained).

- Clicking on an individual member toggles its restraint setting between the types applicable to the member.
- Dragging a box from left to right toggles the restraint setting for all members totally enclosed by the box.
- Dragging a box from right to left toggles the restraint setting for all members that are either enclosed by the box, or cut by the box perimeter.
- Depressing the SHIFT key and dragging a line toggles the restraint setting for all members that cross the line.



30.7.6 Show\Alter State - Fixed\Pinned

The Fixed\Pinned command provides a means to graphically assess and modify the end fixity for all members in the model.

Each member is colour coded to indicate its end fixity setting (N/A, Pinned, Fixed, Moment, Mixed, and Cantilever).

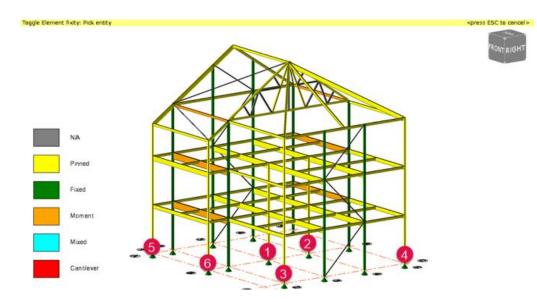
- Clicking on an individual member toggles its end fixity settings between the types that are valid for the member.
- Dragging a box from left to right toggles the end fixity settings for all members totally enclosed by the box.
- Dragging a box from right to left toggles the end fixity settings for all members that are either enclosed by the box, or cut by the box perimeter.
- Depressing the SHIFT key and dragging a line toggles the end fixity setting for all members that cross the line.



Where the end fixity is shown as 'Mixed' this indicates that the fixity at end 1 differs from that at end 2. 'Mixed' end fixity can only be specified by editing the member Properties directly.

In this model some columns are not intended to resist any lateral load. Now we will change the releases of the column stacks from fixed to pinned.

Step 1. Run the Fixed/Pinned command to quickly toggle each stack of the 6 columns identified in the screenshot below from Fixed to Pinned.



30.7.7 Show\Alter State - Steel

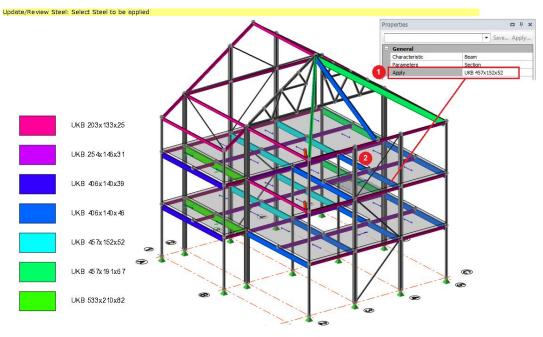
Steel element characteristics can be changed graphically such as Section, Grade etc.



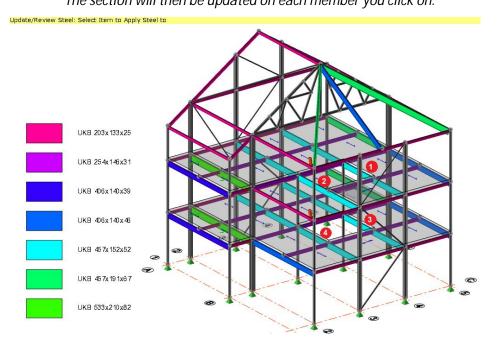
The member clicked on has to be of the same type (beam, column, or brace) as the source member.

Now we will rationalise the primary beams on the 1st and 2nd floor level so that they are all a UKB 457x152x52 section.

- Step 1. Run the Steel command from the Review tab Show\Alter State group.
- Step 2. In the Properties window, select the parameter to copy (Section, Grade or Both).
- Step 3. Select Section UKB 457x152x52 (1)
- Step 4. Click on the member (2) (as in screenshot) to which the selected section to be copied to.
- Step 5. Press Esc key to end the command.



- Step 6. Select on Member (1) (as in screenshot) containing the steel to be copied.
- Step 7. Click on the members (2), (3) & (4) to which you want to apply the steel. The section will then be updated on each member you click on.

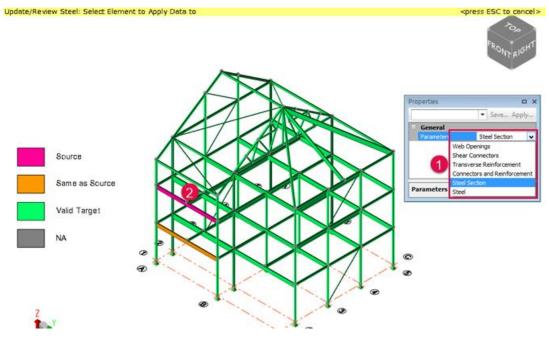


30.7.8 Show\Alter State - Copy Properties

The Copy Properties command provides a means to graphically copy a specified element parameter (e.g. web openings, connectors or transverse reinforcement) from a source member to other valid target members.

To copy a specific property from one member to another:

1. In the Properties window, select the parameter (1) of the member you want to be copied (e.g. web openings, connectors or transverse reinforcement).



2. Click on the source member that contains the property to be copied (2).

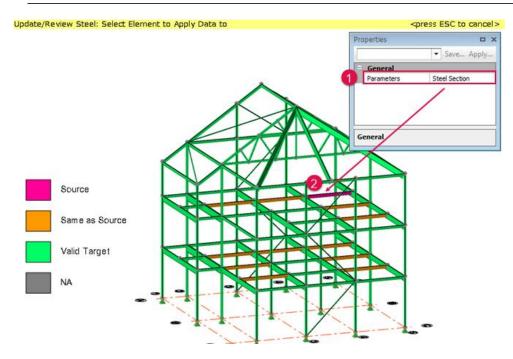
- Source this colour identifies the member that was clicked on. (If this is incorrect, press Esc then reselect.)
- Same as Source this colour identifies those members in which the selected parameter already matches the Source.
- Valid Target this colour identifies those members to which it is possible to copy the selected parameter.
- NA this colour identifies those members to which it is not possible to copy the selected parameter.

Either click on an individual target member, or box around a series of target members to copy the selected parameter to them.

Now we will rationalise the secondary beams on the 1st and 2nd floor level so all internal secondary beams are a UKB 254x146x31 section.

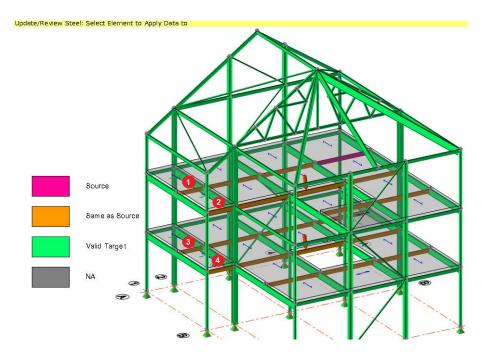
- Step 1. Run the Copy Properties command from the Review tab Show\Alter State group.
- Step 2. In the Properties window, select the parameter of the member you want to copy (e.g. web openings, connectors or transverse reinforcement, steel or steel grade).
- Step 3. Select Steel Section (1).
- Step 4. Click on the source member (as screenshot below) that contains the property to be copied (2).

Other members that already have this section are coloured 'same as source' to aid identification.



Step 5. Either click on an individual target member, or box around a series of target members to copy the selected parameter to them. (See screenshot below).

The section will then be updated on each member you click on.



- 30.8 Interrogate and Review Individual Members
- 30.8.1 Right Click Context Menu Edit Member

Displays the Edit Element Properties dialog for the currently highlighted element to allow you to edit the properties for the member.

			General Size Algnment Releases	^		Source/ Description	Distance along member/ Sub-beam length [m]	Top flange restrained/ Sub-beam continuously	Sub-beam top flange factor	Bottom flange restrained/ Sub-beam continuous	Sub-beam bottom flange factor
					1			V		V	
)	Undo	Ctrl+Z	Haunches			sub-beam	3.750	•	1.000		1.000
	Redo	Ctrl+Y	End plates		2	member	3.750	-			
ĥ	Сору		Web openings Deflection limits			sub-beam	3.750	~	1.000		1.000
	Delete Element				3	support	7.500	4		~	
	Zoom Out		Camber							1	
	Apply Property Set	1	Natural frequency Instability factor		<						>
	Create property set	-		× .	-						
Ē	Edit SB 2/A/1-2/B/1									ок	Cancel

 You can only edit a single entity in this manner.
 To edit multiple entities it must be undertaken via the Properties window for the current selection.

30.8.2 Right Click Context Menu – Open Load Analysis View

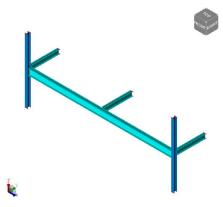
Opens a Loading Analysis View for the currently highlighted member.

Refer to the screenshot below to review the analysis results of interest.

The more Market Edit Load Analyse	Design Report Draw Windows Landing Andrem	
Bet Dobe Lines * And Heat Next Feat Benedif Actions Feat Server Pacters Hear Data Feat Server Pacters Hear	Select the result type and direction	
Status 0.8.3	5 55 55 cm 2 50 2 50 2 50 2 50 2 50 2 50 2 50 2 5	
todistan todistan todistance todis	41244	
Determin 2000 Arm Span Minke beam Orace Inft 10 Ark Strain right 30 Ark Preser right 30 Ark Preser right 51 Ark Preser right 51 Ark		
Baladow definition 22.8xm Applied fract WH 8.254/m Applied fract 8.244/m Applied fract 4.244/m Applied fract 4.244/m		4
See the loadcase of devices the sector of t		
Detaece	Pipe	
✓ Stee Prozens 🛃 🖾 2578,-5.356+1.50+1.580		O O Netto United Register (Sursceld)

30.8.3 Right Click Context Menu – Open Member View

Opens a Member View containing the currently highlighted member and any members connecting to the highlighted one.



30.8.4 Right Click Context Menu – Show Member Loading

The Member Loading dialog tabulates all the loads applied to the currently highlighted member.

Loadcase	Source	Direction	In Proj.	Span	Type	Q1[kN/m]	Q2[kN/m]	Pos[m]	Length[m]	^	Clos
1 Slab self weight	Decomposition 2	Major		1	VDL	5.0	4.9	5.625	0.677		
1 Slab self weight	Decomposition 2	Major		1	VDL	4.9	4.1	6.302	0.448		
1 Slab self weight	Decomposition 2	Major		1	VDL	4.1	1.9	6.750	0.375		
1 Slab self weight	Decomposition 2	Major		1	VDL	1.9	0.1	7.125	0.375		
2 Dead	Decomposition 2	Major		1	VDL	0.0	0.4	0.000	0.375		
2 Dead	Decomposition 2	Major		1	VDL	0.4	0.8	0.375	0.375		
2 Dead	Decomposition 2	Major		1	VDL	0.8	1.0	0.750	0.448		
2 Dead	Decomposition 2	Major		1	VDL	1.0	1.0	1.198	0.677		
2 Dead	Decomposition 2	Major		1	VDL	1.0	1.0	1.875	0.677	5	

30.8.5 Right Click Context Menu – Check Member

Carries out a check design of the highlighted member and displays the design calculations for you to interrogate.

	55) Sum	mary UKB 406x140x39(5355)							
Classification		Design Condition	#	Design Value	Design Capacity	Units	U.R.	Stat	us
Shear Major Buckling Shear Web	\triangleright	Classification	2	Class 1	-	-		1	Pas
Moment Major	\triangleright	Shear Major	2	-45.2	565.1	kN	0.080	1	Pas
Buckling Lateral Torsional	∇	Buckling Shear Web	2	-45.2	489.8	kN	0.092	-	Pa
Jeflection	\triangleright	h_/t_	= 59.500						
		Ratio	=0.092						
		✓ Pass							
			2	97.2	256.9	kNm	0.378	1	Pa
		🖌 Pass		97.2 No	256.9 Forces			Not red	
	⊳	✓ Pass Moment Major			5 5 A A B A				
		✓ Pass Moment Major Buckling Lateral Torsional	2	No	5 5 A A B A	- mm		Not red	quire
		✓ Pass Moment Major Buckling Lateral Torsional Deflection Self weight	2 - 2 2 2	No 0.7	Forces - 30.0	- mm mm	-	Not red	quire Pa
		✓ Pass Moment Major Buckling Lateral Torsional Deflection Self weight Deflection Slab	2	No 0.7 7.7	Forces - 30.0	- mm mm mm	- - 0.256	Not rec	

30.8.6 Right Click Context Menu – Design Member

Carries out a design of the highlighted member and displays the design calculations for you to interrogate.

	Summ	ary UKB 406x140x46(5355)					
Classification		Design Condition	#	Design Value	Design Capacity	Units	U.R.	Status
Shear Major Buckling Shear Web		Classification	2	Class 1	-3	-	(j=)	🖌 Pas
Moment Major	> S	ihear Major	2	-45.7	611.5	kN	0.075	🖌 Pas
Buckling Lateral Torsional		Suckling Shear Web		56.000	58.580			🖌 Pas
	100 00	/ Pass						
	100 00	Noment Major	2	97.9	315.1	kNm	0.311	🖌 Pas
	S2 2	uckling Lateral Torsional	-	No	Forces	-	-	Not require
)eflection Self weight	2	0.6		mm		
		eflection Slab	2	6.1	30.0	mm	0.205	🖌 Pas
						1000300		
		eflection Dead	2	1.3	15.0	mm	0.084	🖌 Pas
		Deflection Dead Deflection Imposed	2	1.3 4.4	15.0 20.8	mm mm	0.084	✓ Pas

30.8.7 Right Click Context Menu – Report for Member

Opens a Report View for the currently highlighted member.

ure 3D	SB 2/A/1-2/B/1		SB 2/A/1-2	/B/1 report ×							
		Pr	roject						Job Ref.		
	Trimble	51	Structure							Sheet no. Page 1/1	
		0	alc. by	Date	Chk'd by		Dete	_	App'd by	1.455	Date
			paupa	05/02/2015			14/11/2	2014			14/11/201
	<u>SB 2/A/1-2/B/1</u>	1241					144			(® T
	Ŧ			L							1
	Ť										T C
	Ţ			100							- -
	Summary UKB 406x140x4		155)				-				<u>↓</u>
				Design Capacity			Statu	IS			⊣
			155)				Statu	I s Pass			
	Design Condition	# D	155) Design Value		Units		Statu:				Ă H
	Design Condition Classification	# D	55) Design Value Class 1	Design Capacity	Units 5 kN	U.R. -	Statu:	Pass			<u>_</u>
	Design Condition Classification Shear Major	# D	855) Design Value Class 1 -45.7	Design Capacity 611.	Units 	U.R. -	Statu:	Pass Pass			<u>_</u>
	Design Condition Classification Shear Major Buckling Shear Web	# D 2	255) Design Value Class 1 -45.7 56.000	Design Capacity 611. 58.58	Units 	U.R. - 0.075	Statu:	Pass Pass Pass Pass			1
	Design Condition Classification Shear Major Buckling Shear Web Moment Major	# D 2 2 2	355) Design Value Class 1 -45.7 56.000 97.9	Design Capacity 611. 58.59 315.	Units 	U.R. - 0.075	Statu:	Pass Pass Pass Pass			1
	Design Condition Classification Shear Major Buckling Shear Web Moment Major Buckling Lateral Torsional	# D 2 2 2 2 -	355) Design Value Class 1 -45.7 56.000 97.9 No	Design Capacity 611. 58.59 315.	 Units - - - kNm s - mm 	U.R. - 0.075	Statu:	Pass Pass Pass Pass			-
	Design Condition Classification Shear Major Buckling Shear Web Moment Major Buckling Lateral Torsional Deflection Self weight	# D 2 2 2 2 2 2 2	855) Design Value Class 1 -45.7 56.000 97.9 No 0.6	Design Capacity 611. 58.59 315. Force	Units Units kN	U.R. - 0.075 0.311 -	Statu:	Pass Pass Pass Pass Jired			-
	Design Condition Classification Shear Major Buckling Shear Web Moment Major Buckling Lateral Torsional Deflection Self weight Deflection Slab	# D 2 2 2 2 2 2 2 2 2	255) Design Value Class 1 -45.7 56.000 97.9 No 0.6 6.1	Design Capacity 611. 58.59 315. Force 30.	Units Units KN KN KN KN KN KN KN M M M M M	U.R. 0.075 0.311 - 0.205	Statu:	Pass Pass Pass Jired Pass			-

30.9 How to Check Member/ Design Member/ Edit...

You can use Check Member to review the individual design results for a member.

- You could then Edit Member to make appropriate changes and Check Member again to see if the Edits were sufficient to satisfy the design requirements.
- Or you could Design Member to automatically design (size) the member.

A couple of scenarios are detailed below to help explain how the Check, Design and Edit commands could be utilised.

30.9.1 Scenario 1 – Check Member/ Design Member

In your model you could right click an entity with say a Fail and review the Check Member results.

You could then Design Member to re-design the member (i.e. increase section size etc.) to satisfy the previous fail status.

Step 1. Right click beam SB 1/A/5-1/B/5 and Check Member to review the results.

Classification		Design Condition	#	Design Value	Design Capacity	Units	U.R.	Status
Shear Major Shear Minor	\triangleright	Classification	2	Class 1	-	-	-	🖌 Pass
→ Buckling Shear Web	\triangleright	Shear Major	11	-101.2	839.2	kN	0.121	🖌 Pass
Moment Major	15.5	Shear Minor	5	No	Forces	kN	-	Not required
Moment Minor	\triangleright	Buckling Shear Web	-	50.35	58.58	-		🖌 Pass
Axial		Moment Major	11	-403.4	522.2	kNm	0.773	🖌 Pass
Axial Bending Combined		Moment Minor		No	Forces	kNm	-	Not required
Buckling Compression		Axial	-	No	Forces	kN	-	Not required
Buckling Combined		Axial Bending Combined) ⁴	No	Forces		2	Not required
Deflection	∇	Buckling Lateral Torsional	13	214.7	194.7	kNm	1.102	🗙 Fail
		Length, L _{LTB}	= 7.500 m					
	\triangleright	Design valuemajor moment, M _{v.Ed}	=214.7 kNm					
	\triangleright	Design buckling resistance, M _{b.Rd}	= 194.7 kNm	EN 1993-1-1: 2005 Cl 6.3.2.1(3)				
		Ratio	=1.102	EN 1993-1-1: 2005 Cl 6.3.2.1(1)				
		🗙 Fail						
		Buckling Compression	3	No	Forces	-	-	Not required
		Buckling Combined		No	Forces	2	-	Not required

The beam current size is UKB 457x191x67 and has a fail related to Buckling Lateral Torsional.

Step 2. Click Close to exit the Check Member dialog.

Step 3. Right click same beam and Design Member to review the design results.

The beam now has been resized to UKB 457x191x74 to satisfy the Buckling Lateral Torsional.



The analysis results are no longer up to date as the increased section will affect the analysis. The design is based on the previous analysis results.

30.9.2 Scenario 2 - Check Member/ Edit Member/ Check Member

Alternatively, you could also use Edit Member to alter some properties (i.e. increase the section size) to resolve the warning/fail issue.

Finally you could Check Member to check that the edit resolves the design warning based on the original analysis results.

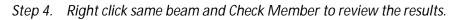
 Classification 		Design Condition	#	Design Value	Design Capacity	Units	U.R.	Status
✓ Shear Major ♀ Shear Minor	\triangleright	Classification	2	Class 1		(i i i i i i i i i i i i i i i i i i i	-	🗸 Pas
✓ Buckling Shear Web	\triangleright	Shear Major	11	-225.2	1110.6	kN	0.203	🖌 Pas
Moment Major		Shear Minor	3.73	No	Forces	kN	2.53	Not require
B Moment Minor	\triangleright	Buckling Shear Web		52.28	58.58	6	-	🖌 Pas
Axial Axial Bending Combined Buckling Lateral Torsional Buckling Compression Buckling Combined	\triangleright	Moment Major	11	-672.4	730.8	kNm	0.920	🖌 Pas
		Moment Minor	379	No	Forces	kNm	873	Not require
		Axial	-	No	Forces	kN	- 1	Not require
		Axial Bending Combined	-	No	Forces	<u></u>	112	Not require
✓ Deflection	∇	Buckling Lateral Torsional	11	-672.4	617.3	kNm	1.089	🗙 Fa
	×	Length, L _{LTB}	= 7.500 m		971. 	55) · · · ·	9 F.	
	\triangleright	Design value major moment, M _{ur}	=-672.4 kNm					
		Design buckling resistance, M		EN 1993-1-1: 2005 Cl 6.3.2.1(3)				
		Ratio	=1.089	EN 1993-1-1: 2005 Cl 6.3.2.1(1)				
		🗙 Fail						
		Buckling Compression	-	No	Forces	27		Not require
		Buckling Combined	-	No	Forces	-		Not require

Step 1. Right click beam SB 1/A/2-1/B/2 and Check Member to review the results.

The beam current size is UKB 533x210x82 and has a fail related to Buckling Lateral Torsional.

- Step 2. Double left clicking on same beam.
- Step 3. In the Properties window, increase the section size to UKB 533x210x92.

	Size		
General	Span	7500.0	mm
- Alignment Releases Lateral restraints	Steel	5355	•
Strut restraints Haunches End plates	Section	UKB 533x210x92	



Classification		Design Condition	#	Design Value	Design Capacity	Units	U.R.	Status
Shear Major	\triangleright	Classification	2	Class 1			() - ()	🖌 Pass
- 년 Shear Minor - ✔ Buckling Shear Web	\triangleright	Shear Major	11	-225.6	1180.9	kN	0.191	🖌 Pass
Moment Major		Shear Minor	-	No	Forces	kN	37	Not required
Moment Minor Axial Axial Bending Combined Suckling Lateral Torsional Buckling Compression	\triangleright	Buckling Shear Web	-	49.69	58.58	-	-	🖌 Pass
	\triangleright	Moment Major	11	-672.4	837.8	kNm	0.803	🖌 Pass
		Moment Minor		No	Forces	kNm	275	Not required
		Axial		No	Forces	kN	-	Not required
Buckling Combined		Axial Bending Combined		No	Forces	1		Not required
Deflection	∇	Buckling Lateral Torsional	nal 11 -672.4 7	739.2	9.2 kNm 0.91	0.910	D 🖌 🖌 Pass	
		Length, L _{LTB}	= 7.500 m					
	\triangleright	Design valuemajor moment, M _{v.Ed}	=- 672.4 kNm					
	\triangleright	Design buckling resistance, M _{b.Rd}	=739.2 kNm EN	1993-1-1: 2005 Cl 6.3.2.1(3)				
		Ratio	=0.910 EM	1993-1-1: 2005 Cl 6.3.2.1(1)				
		🖌 Pass		100				
		Buckling Compression	37.	No	Forces	87	0.5	Not required
		Buckling Combined	· · · ·	No	Forces	(-	(i.e.)	Not required

The beam now passed the design checks with your specified section size.

30.10 Design Steel (Static)

The Design Steel (Static) processes can be summarised as follows:

- 1. A first-order linear analysis of all unfactored loadcases is carried out to establish Serviceability Limit State requirements such as deflections.
- 2. EHF's are determined for every active combination in which they have been included.
- 3. Having established the EHF's, their contributions to frame deflections are determined using a first-order linear analysis. α_{cr} values are also established.
- 4. The analysis type that you have set in Design Options is then performed for all active combinations to establish design forces.
- 5. All steel members set to auto-design mode are designed for the appropriate design requirements. Gravity members are designed for gravity combinations, lateral members are designed for all combinations. Only active combinations are checked. If section sizes are changed the analysis-design cycle is repeated.
- 6. All steel members set to check mode are checked for the appropriate design requirements. Gravity members are checked for gravity combinations, lateral members are checked for all combinations. Only active combinations are checked.
- 7. At the end of the process, all Autodesign settings follow the Design Options Autodesign rules and are either set to check-design mode (Reset Autodesign...Always) or retain the Autodesign setting (Reset Autodesign...Never or When check status is at Worst) dependent upon the option set.

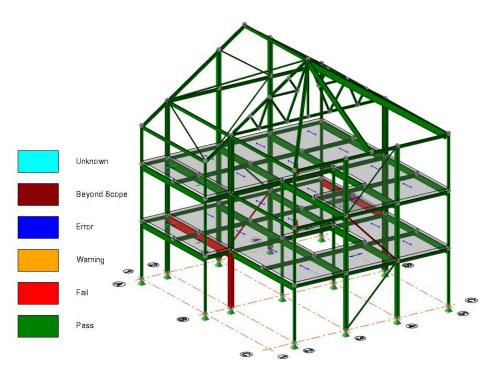
30.10.1 How to run Design Steel (Static)

- First, check the individual member Autodesign property is correctly set for all members:
 - o Autodesign 'On' new section sizes will be designed
 - o Autodesign 'Off' existing section sizes will be checked
- Next, review the Design options and adjust if required:
- Click Design Steel (Static).
 - Gravity members are designed/checked dependent upon the Autodesign setting for all active gravity combinations only.
 - Lateral members are designed/checked dependent upon the Autodesign setting for all active combinations.



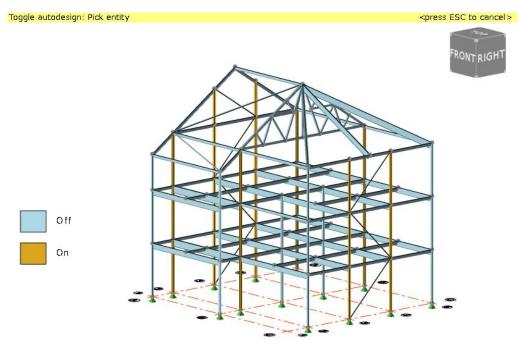
At the end of the analysis-design process, the active view switches to a Review View and the ribbon switches from Design to Review - ready for reviewing the design graphically.

Step 1. Run Steel Design (Static) and review the Design Status on the Review View.



The lateral combinations are generating horizontal forces which are being distributed into the braced bays and moment frames. These inevitably increase the design force to be resisted by the existing sections. In the case of the columns these were originally sized to satisfy just the gravity combination.

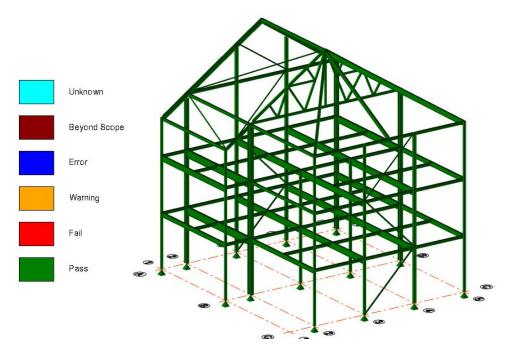
- 30.10.2 Using the Review View again
 - Step 1. Reset eight columns which attached to braced or moment resisting frames back to Autodesign 'On' using the Auto\Check Design command as the screenshot below.



- With Autodesign 'On' new section sizes will be designed.
- Since these entities are not classified as 'Gravity only' they will be sized for all the active combinations when a Steel Design (Static) is undertaken again.

- If a Steel Design (Gravity) was performed instead, they would only be sized for the active gravity combinations only.
- Step 2. Reset any failing brace members to Autodesign 'On'.
- Step 3. Reset any failing moment beam members to Autodesign 'On'
- Step 4. Re-run the Steel Design (Static) command to resize the failing members.

All the entities should now be passing.



Step 5. Save the model using File > Save As and name the model Design_Runthrough.tsmd.

- 30.11 Using Result View to Investigate Solver Warnings
- 30.11.1 Project Workspace Status

After running the design, you should always investigate any issues reported in the Project Workspace-Status, before reviewing design results in the Review View. The Results View is intended to help you investigate these instabilities.

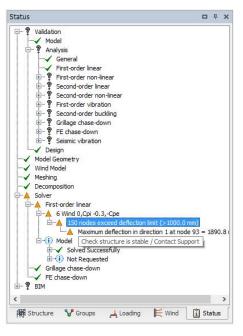
To demonstrate this, open a model that has instabilities and use the Results View to assess the forces and deflections of the model to determine the causes of the instabilities.

- Step 1. Open the model file 2_Design_Start_Instabilities.tsmd.
- Step 2. Review the loadcases and combinations that exist in the model.
- Step 3. Run a Design Steel (Gravity).
- Step 4. Review the Project Workspace Status for any issues.



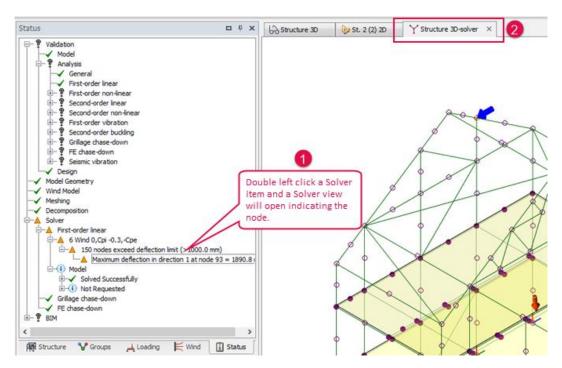
Click the '+' or '-' to expand/contract the branch of the Status tree in order to drill into the item of interest.

- A 'Warning' symbol exists against the Solver branch indicating that 150 nodes exceed a deflection limit of 1000mm for load case Wind 0, -Cpi, -Cpe and to check the structure is stable.
- An 'i ' (information) symbol exists against the Solver Model folder to indicate that the lateral combinations have not yet been considered.
- A 'ü' indicates that there are no issues with the item.



You can double left click a Solver item in the Status tree and the Solver view will open showing the underlying analysis model.

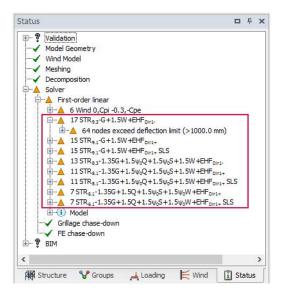
Step 5. Double left click the Solver warning to open a Solver view and highlight the node with the maximum deflection.



Step 6. Run a Design Steel (Static).

Step 7. Review the Project Workspace – Status for any additional issues.

In addition to the Wind 0 load case, all lateral combinations that produce a deflection greater than 1000mm are also being identified.



To investigate this further you need to review the deflected shape of the model. This is achieved using the Results View.

30.11.2 The Result View

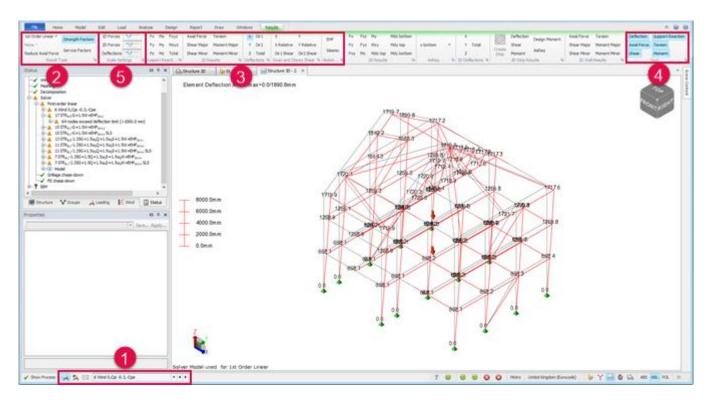
Step 1. Click on the Results View icon on the status bar to switch to the Results View mode. The Results tab will appear.



You can also right click the active view tab and select the Results View from the context menu.



- Step 2. Select the loadcase or combination you wish to view results for (1). See screenshot below.
- Step 3. Use the Results tab to set the analysis type (2), Item(s) to display (3) and text to overlay on the diagram (4).
- Step 4. Use the sliders to scale the display (5).



Step 5. Investigate the model displacement for loadcase 6 'Wind 0, Cpi -0.3, -Cpe'.

The model has deflected greater than 1000mm in the X direction indicating a mechanism - so we need to review the bracing.



The 1000mm threshold is an order of displacement that TSD considers appropriate to indicate the possibility of a mechanism. It is not an indication that the structure satisfies deflection limits imposed by the Code of Practise in use.

Clearly, there is no steel bracing in the X direction so what form of bracing do we have to use to stabilise the model? Are there moment resisting frames in the model? If so, are they correctly defined?

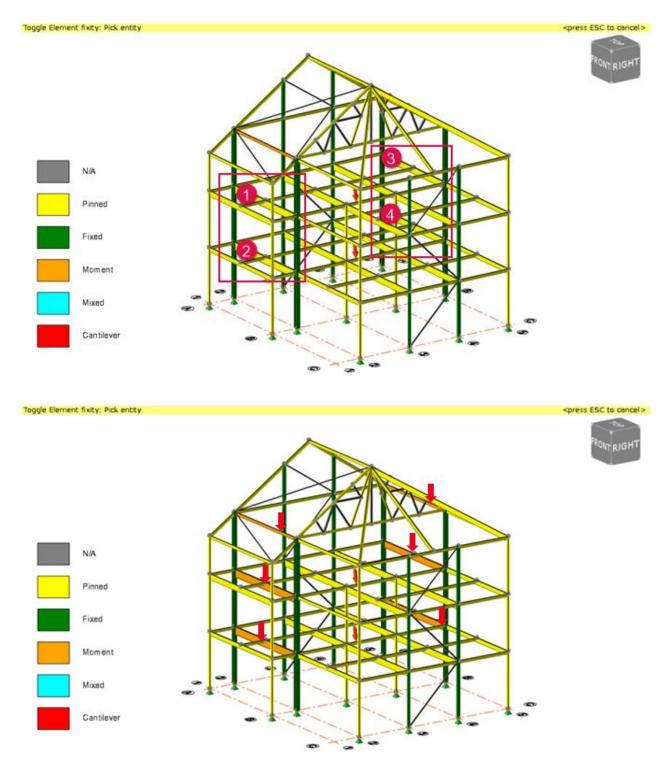
Step 6. Switch back to the Review View.



Step 7. Review the Fixed\Pinned command. This indicates that the 1st and 2nd floor beams are all pinned, hence they are not acting as moment resisting frames.

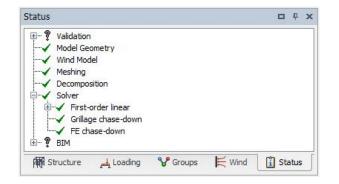


Step 8. Adjust the beams identified below to Moment by clicking on each beam the required number of times to toggle between the valid states.



Step 9. Re-run the Design Steel (Static).

The Warnings in the Project Workspace – Status have been resolved and you can now proceed with the design.



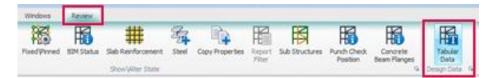
30.12 Tabular Data

Two different types of tabular data can be reported.

- Design data such as sway, design summary, material listing section resistances etc.
- Analysis data such as nodes, elements, nodal forces, element forces etc.

30.12.1 Design Tabular Data

Design tabular data can be accessed from the Review tab > Tabular Data command.



Step 1. On the ribbon set the View type (1), Material type (2), Characteristic (3) etc. and apply a Filter (4) if required.

Have Hodel Edit Land Analyse 3	eegn Report Ones Windows Revew-Data		- B
re - Colorado Series Se	Transe Parent Parent	Andread Andrea	
1 2 3	12) 10 4 4 4 5 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1	Colical Sweet	
V Model George By	Reference ContonatoriX	Stati K No Contrinator Y Stati	A.Y. N. Conducator XY Twat-Factor Details
V Medwag	9C 02 38 878,	A THE STREET STREE	eller statio-della-delle Los Debit.
Decorposition Solver	90.03 18 579 ₆₇ L396+L5Q+L5u;6+L5u;8+079 ₃₁₀	1 1.7/8 10.57%e+1.35G+1.5Q+1.5mg/8+1.5mg/8+0.9%pap 1	49.864 17 57942-0+1.501-0172-0 1.002 Details
E-10 Prot-order Insar	10 C/4 18 574, / L395+L5Q+L59,5+L59,8+0+9 ₁₁₀	1 3.76 11 574, -1.355+1.50+1.3v/8+1.5v/8+0-Fpp 1	49.864 17 5784+G=1.5W=0-Pp- 1.003 Details
Grillage chase down	10 C/1 10 D74_v1.310+1.5Q+1.5w,0+1.5w,0+0+P_{110}	1 3484 13 5% v 1350+130+130+13w/#+0+Ppp 1	45.864 17.578, -G+1.5W+0.49, 1.008 Details
T EDH	9CA2 1837947139G+13Q+136(8+136)8+136(8+1945a)	1 3.731 13.97% x+5.350+5.50+5.36(d+5.36(d+5.36(d+5.36(d+5.46)))) = 0.000 (10)	45.789 17 578 c G+1.5W+0+5m 1.002 Details
	0C0/2 18 97% v1.355+1.5Q+1.3ug9+1.3ug9+1.9ug9+0+Pre	1 3.731 10.07% +1.5Q+1.5Q+1.5w/0+D+0	+6.872 17 579 _{6.2} -6+1.5W=0+0 _{2m} 1.012 Details
	SCA/5 15 579e+1.355+1.5Q+1.5u;5+1.5u;H+DH210	1 3.694 11 579 _{8.7} 1.395+1.5Q+1.5q,0+D.9 ₃₁₀ 1	45.789 17.579 ₈₁₇ G+1.5W+019 ₂₁₀ 1.004 Details
	SC 8/3 10 5794 v 1.150+1.5Q+1.5u/S+1.5u/H+0+0+0	1 3.694 18 579 _{6.4} L350+L30+L30,5+L30,8+D49 _{3.6} 1	44.872 17 578 _{4 7} G+1.5W+019 ₈₁₀ L.003 Details
	SCA11 10 574 JULISG+1.5Q+1.3 w/S+1.3 w/H+0+Ppm	1 1.737 18.878_cr1.350+1.50+1.3u/8+1.3u/8+0+Ppp 1	45.785 (7.578, p-G+1.5W+()-Pipe 1.004 Details
	9C 0/1 18 579e+1.39C+1.5Q+1.3%5+1.5%/0+0+0+0	1 3.737 18.9%+1.395+1.50+1.5w/H+3-w/H+0+P ₂₊₆ 2	+6.872 17 579e+G+1.5W+0+Ppm 1.001 Details
	90.8/3 20.0%,+1.350+1.5Q+1.54,5+1.54,8+0+6+6	2 3.728 10.679 _{6.7} 1.356+1.5Q+1.8 ₆ 5+1.8 ₆ 5+1.8 ₆ m+D+P ₁₀₀ 2	44.872 17 57942-G+1.5W+D+P ₂₀₀ 5.000 Details
	SC 8/4 31 578, +1.315+1.5Q+1.5w,5+1.5w,8+0+6+6	1 3.86 1859e+130+130+130/0+09pp 1	44.870 17 5794, G+1.5W+010510 1.001 Details
	SCA(2) 10 57% + 1.516 + 1.5Q + 1.5u/6 + 1.9u/6 + 0.47 pm	1 3.788 10.579e+1.390+1.50+1.50e1.54e(0+0.90)+0+0+0+0+0+0+0+0+0+0+0+0+0+0+0+0+0+0+	40.788 17 57%+p-G+1.5W+D+5pm 1.000 Details
Millioner Alaster Hind Date	SCA/4 10 57%+v1.25G+1.5Q+1.5m/0+1.5m/0+0+0+0-0	1 3.700 18 5% x 1.350 + 1.50 + 1.5 x (8 + 1.5 x (8 + 15 + 1))	40.789 17 57% - G = LSW = D+P_2+0 1.003 Details

30.12.2 Analysis Tabular Data

Analysis tabular data can be accessed from the Analyse tab > Tabular Data command.

File	Home Model	Edit Load	Analyse Des	ign	Report	Draw	Windows
Options	1st Order Linear 1st Order Non-linear 1st Order Vibration	2nd Order Linear 2nd Order Non-linear 2nd Order Buckling	FE chase-down Grillage chase-dow	wn	Tabular Data	Mesh Sla Update V	bs Nall Beams
Options 🗔	1st Order Analysis 🕞	2nd Order Analysis 🕞	Chase-down	15	Solver 5	Mesh	ning G

- Step 1. Set the Result type (1) and the View type (2) to display.
- Step 2. To request for force or displacements, select a loadcase or combination from the load drop list (3).

Pierrer Headed Date Land Analyzer Der Id Onder Linear 3nd Onder Tunnerer 3nd Onder Tunnerer PE chase-daven PE chase-daven Statige chase	Tabular Data	Nest Sales	Anne 1 Sec Order Sec Order	lene *	2 et Dul Pirces •				9
nuture 0 # x	(2) Structure 20	31.2(0)	D ITT Solve	r Model Data X	1				1
A involut					ent End Forces				- 1
※部 Levels ※野 Franes ※袋 Stores	Element Number	Node Number	r. (m)	r. 040	pog	H _k [kNim]	M, (kNm)	M, [ktm]	î
 ・ 営 Architectural Grids ・ 日 Sub Models ・ 日 ・ 日	1	11	-141.7	0.0	0.0	0.0	0.0	0.0	
8 Perbers		58.	191.5		344	10.1	00	0.0	
Sabs	2	3	-281.9	0.0	0.2	0.0	0.0	0.0	
+ 🗱 Walk		61	290.7	0.0	0.2	0.0	4.4	-0.1	
in ph Roch	3	5	-368.6	0.0	0.4	0.0	0.0	0.0	
		54	367.4	0.0	-0.4	0.0	-0.7	0.0	
	4	7	-240.7	0.0	0.0	0.0	0.0	0.0	-11
		67	240.5	0.0	0.0	0.0	0.0	0.0	-11
	5	1	-37.9	0.0	0.0	0.0	0.0	0.0	-11
		\$7	37.3	0.0	0.0	0.0	0.0	0.0	-1
	6	10	-101.7	0.3	43	0.0	0.8	0.1	-11
	-	60	300.5	4.1	8.3	0.0	-0.2	0.2	-11
	7	11	-292.7	0.1	-0.6	0.0	1.4	0.1	-11
Structure Viscous Aliandrig Kinnd Distantia		63	191.5	4.1	0.0	0.0	0.0	0.2	-11
	•	66	136.4	0.0	0.0	0.0	0.0	0.0	-11
upertes 0 # X		13	-253.9	0.0	9.0	0.0	0.0	0.0	-11
+ Save., Apply	-	68	352.0	4.3	9.0	0.0	-18.0	0.6	-11
	10	15	-507.5	0.7	-35.6	0.0	0.0	0.0	-11
		11	\$04.3	4.7	10.6	0.0	21.2	1.5	-11
	11	12	-297.6	4.3	33.6	0.0	47.1	-1.3	-11
		48	195.7	6.3	-33.8	6.0	4.5	0.7	
	12	1.8	-241.9	-1.0	-39.7	0.0	29.3	-2.9	
		12	238.8	1.0	29.7	0.0	0.1	0.9	
	13	19	-251.8	6.4	6.5	0.0	0.0	6.0	
		74	249.8	4.4	4.5	0.0	-12.9	0.8	
	24	21	-486.4	6.5	-4.9	0.0	0.0	0.0	
		77	404.0	0.5	4.5	0.0	9.8	0.9	
	15	23	-041.5	4.6	25.9	0.0	-90.9	-1.5	
		75	129.2	0.6	-25.9	0.0	-1.0	0.3	
	18	24	-277.9	4.8	-29.8	0.0	36.1	-1.8	
3		78	275.5	0.8	29.9	6.0	1.7	0.2	
	17	25	-112.5	0.0	0.0	0.0	0.0	0.0	

248 (257)

31 Foundation Design

31.1 Introduction

You can model and design Pad & Strip Footing in TSD model.

Pad & Strip Footing can be auto designed based on size, depth and bottom reinforcement.

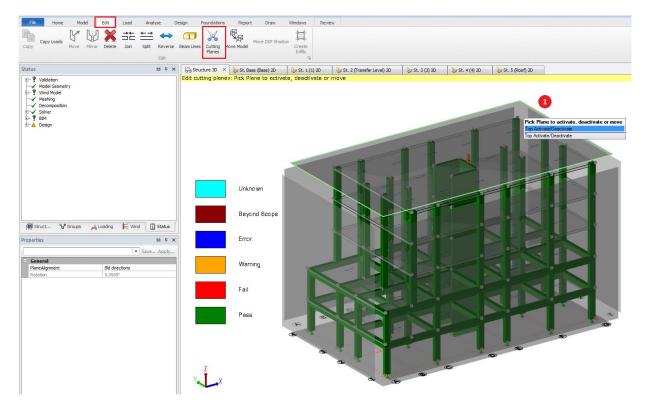
You are also able to group the design of bases which can be set in Design Options.

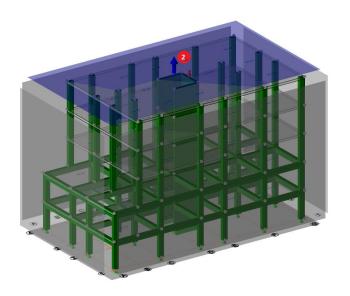
Analysis Concrete Concrete Composite Beams Design Groups Autodesign Autodesign Display Limits Steel Joists	Members to design using groups Concrete Beams Concrete Columns Steel Beams Composite Beams Trues Members
	Steel Columns Steel Braces Steel Joists
	Isolated Foundations

Foundation type Isolated Pad Base Auto-design depth Image: Constraint of the second secon

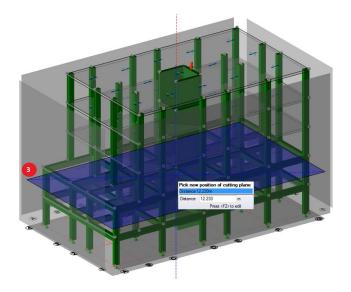
31.2 Using Cutting Planes

- Step 1. Open the model file TSD Concrete Design Model Foundation.tsmd
- Step 2. Run Design (Static).
- Step 3. Go to Edit tab and select on Cutting Planes command.
- (It is used to reduce the active 3D view to show only the lower floor level)
- Step 4. Click on the top plane (1) it will turn into blue color.

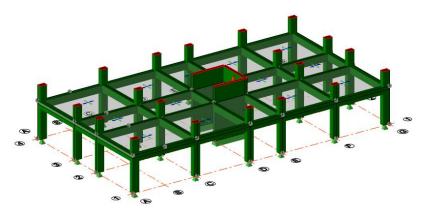




Step 5. Click on the blue arrow (2) to move the top plane.Step 6. Move the plane to above first floor level (3).



Step 7. Press Esc key or double right mouse click to stop the command.



We are going to use this view to insert the pad footing under the columns.

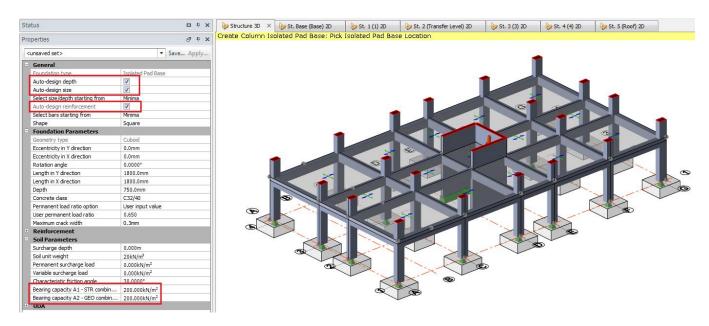
31.3 Model of Pad Footing

You can model the pad footing supporting columns in the 3D view or Base 2D view.

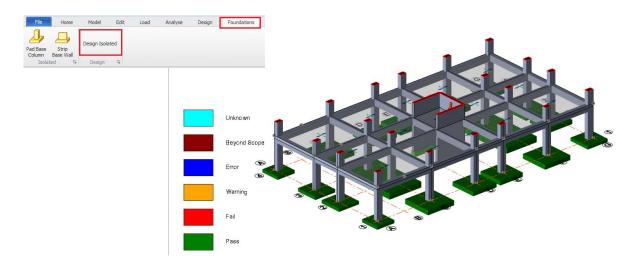
- Step 1. Set to Structure 3D view
- Step 2. Go to Foundation tab and click on Pad Base Column button.



- Step 3. Review the Properties window tick on Auto-design depth, size & reinforcement and set Bearing Capacity to be 200kN/m².
- Step 4. Drag a window (from left to right) to encompass the whole model to insert pad footing on all the columns.



31.4 Design of Pad Footing



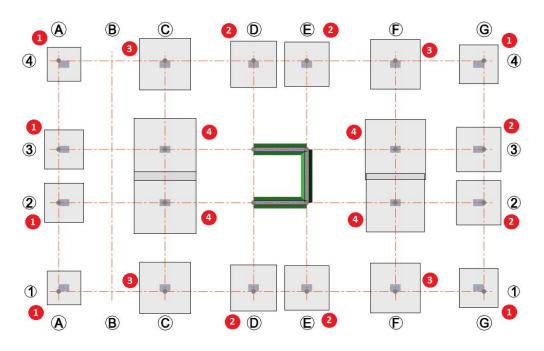
Step 1. Go to Foundation tab and click on Design Isolated button.

31.4.1 Rationalise bases

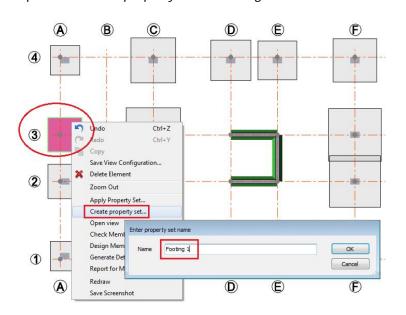
The pad footings are designed with different sizes, to rationalise them we can use property sets to group similar size bases from the first design.

Step 2. Set to St. Base (Base) 2D view.

Use Property Set to group the bases accordingly to the first designed size – see below 4 required groups to create.



Step 3. Select on the pad footing at gridline 3/A (biggest size of the group), right mouse click – on the context menu, select Create property set....
Step 4. Name the property set as Footing 1.



Step 5. Select the remaining 5 pad footings to be applied with the same property set 'Footing 1' using "Apply" button in the Properties window.

• • ×	Structure 3D	St. Base (Base) 2D	🔇 😡 St. 1 (1) 2D	St. 2 (Transfer Level) 2D	😡 St. 3 (3) 2D	😡 St. 4 (4) 2D	2 St. 5 (Roof) 2D
₽ ₽ ×							
• Save., Apply		Ē		Ô	Ē	Ê	G
	A A	Ð		U		U	9
			· · · · · · · · · · · · · · · · · · ·				
Isolated Pad Base	(4)						++
	~		1				
Cuboid		1					
0.0mm					1		
0.0mm							
0.0000*							
	(3) 🚽						
300.0mm	~						
C32/40							
User input value							
0.650				1			
0.3mm							
	(2)						
-	7						
None							
Loose bars							
Type 2							
500		1					
16			1				
200.0mm	7						
							+ +- I
16							
200.0mm							
				Ô	Ê	Ê	
40.0mm	A	U U	U	U		U	G
	Save Apply Isolated Pad Base Cuboid O.dmm O.domm O.dog Cuboid O.dmm O.dog O.dmm C32/40 User input value O.650 O.3mm None Loose bars Type 2 S00 16 200.dmm 16 200.dmm Is Cubo Su Cubo Su Cubo Su Cubo Cubo Su Cubo Cubo Su Cubo Cubo	Loolated Pad Base	Save Applyin Loolated Pad Bare Loolated Pad Bare Cubod O.0mm O.0000* Gomm O.0000* Gomm O.0000* Gomm Cubor put value O.650 O.3mm None Figure 2 Door Bars Type 2	Sva Apply Iodated Pad Base Cuboid O.mm 0.0mm 0.0mm	Sove Apply Loolated Pad Base Image: Constraint of the second of	Sov Applyin Loolated Pad Base Cubod O.0mm 0.0mm 16 200.0mm	Sore Applyin Ioolated Pad Base Ioolated Pad Base Cuboid Omm Outman Outman Outman Outman



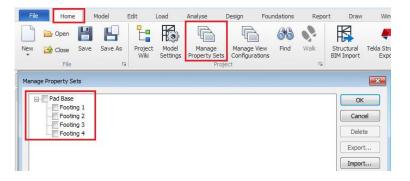
You can also select on a pad footing, use Apply Property Set... from the right click context menu. This only applies to one pad footing at a time.

Step 6. Repeat the same process to create 3 more property sets with pad footing at gridline 1/D, 1/C & 2/C naming them as Footing 2, Footing 3 & Footing 4 respectively.

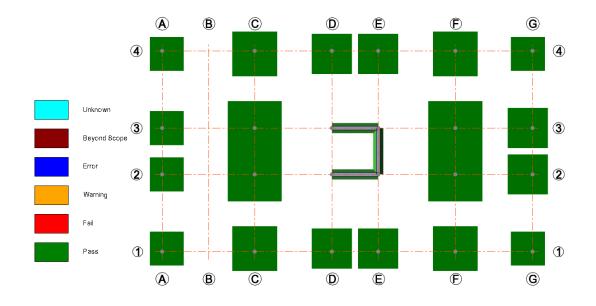


The footing to be used to create property size has to be the biggest size (i.e. most critical one) to ensure those footings applied with its same property set are able to pass when re-designed them.

You can check via Home tab > Manage Property Sets for all the property sets created.



- Step 7. Use Apply Property Set... to apply each property set created to the remaining pad footings.
- Step 8. Go to Design tab and click on Options button in Design Groups option, tick 'Isolated Foundations' to set to grouping design.
- Step 9. Update the analysis result by running 1st Order Linear Analysis again.
- Step 10. Go to Foundation tab and click on Design Isolated button to run design for the newly grouped pad footings.



31.5 Model of Strip Footing

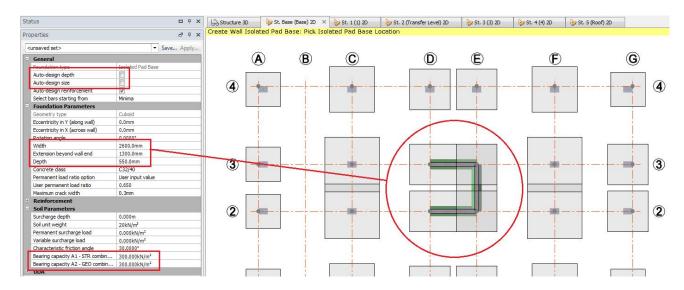
You can model the strip footing supporting walls in 3D view or Base 2D view.

- Step 1. Set to St. Base (Base) 2D view.
- Step 2. Go to Foundation tab and click on Strip Base Wall button.

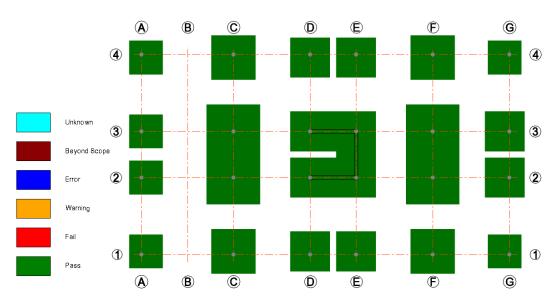


In the earlier exercise, we have learnt how to set auto design for pad footing where TSD proposed a suitable footing size. In this exercise, we are going to manually define the strip footing size and then get TSD to design for the required reinforcement only.

- Step 3. In the Properties window Untick on Auto-design depth & size, tick on Auto-design reinforcement and set Bearing Capacity to be 300kN/m².
- Step 4. Set Width to be 2600mm, Extension to be 1300mm and Depth to be 550mm.
- Step 5. Click on the wall to create strip footing.



Tekla Structural Designer Fundamental Training

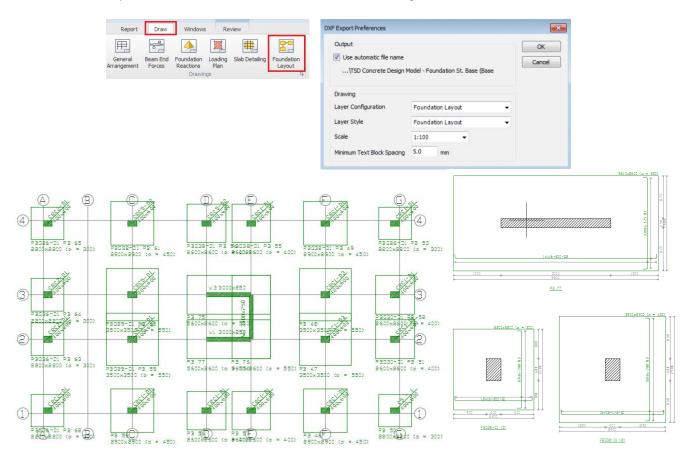


Step 6. Go to Foundation tab and click on Design Isolated button.

Now all the footings have been successfully designed and next we can look at how to get the foundation drawings.

31.6 Foundation Drawings

Step 1. Go to Draw tab and click on Foundation Layout button.



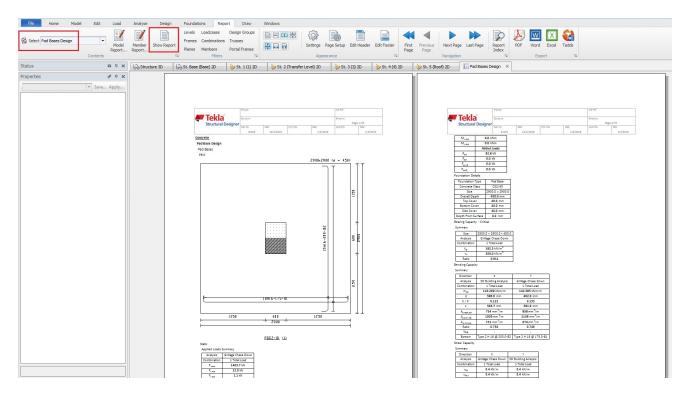
31.7 Foundation Design Report

- Step 1. Go to Report tab and click on Model Report...
- Step 2. In the Report Contents page, click on Add button Name as Pad Bases Design.
- Step 3. From the Chapters and Options column, select on Concrete > Pad Base Design and drag it into Report Structure column.

leport Contents			×
Available Styles	Chapters and Options: Drag chapters and options from left to right, which are to be included in the report.	Report Structure: Drag selected chapters up and down, to change order.	OK Cancel
Active Style Solver Model Data (active) Building Loading Building Loading Building Loading Building Design Material Listing Beam End Forces Bracing Forces Found ation Reactions Seismic Design Member Design Calcs Pad Bases Design Add Remove >> Active Active Style Name Pad Bases Design		Concrete (Nuclure) Pad Base Design (Stucture) Pad Base Design Summary Static & RSA(Structure) Pad Base Group Design Summary Static &	View Mode Flat Hierarchical

Step 4. Click OK to exit.

Step 5. In the Select list, choose Pad Bases Design and then click on Show Report.



32 Changing the Design Code (For Information only)

32.1 Introduction

TSD allows you to choose from a range of international design codes of practice.

Each new project will initially adopt the codes that have been specified in Settings from the Home tab when create a new blank file, however it is also possible to change codes in mid project.

32.2 Head Code

A head code is selected which configures the choices available for the following codes:

Action Codes	Resistance Codes
General Loading	Steel Design
Wind Loading	Concrete Design
Snow Loading	Composite Design
Seismic Loading	Timber Design
Combinations	Masonry Design
	Foundation Design
	Seismic Design and Detailing

32.3 How to Configure the Default Design Codes to be applied to New Projects?

- Step 1. Click Settings on the Home tab.
- Step 2. Select a suitable settings set from the drop list and make it Active.
- Step 3. Use the Design Codes page of the Settings dialog to choose the Codes required.

Having set the codes as required, TSD will retain these as the default codes to apply for each new project until you decide to amend them.

To start a new project with the chosen design codes:

Step 4. Click New on the Home tab.

32.4 How to Change Design Codes in an Existing Project?

- Step 1. Click Model Settings from the Home tab.
- Step 2. In the Design Codes page of the dialog choose the Codes required.

32.5 Model Changes

If you change design codes in mid project, you need to be aware of the following model changes:

- Any existing wind load data will be lost you will therefore need to recreate them.
- Any existing load combinations will be lost you will therefore need to recreate them.
- Steel material properties will be amended.
- Rebar material properties will be amended.
- Concrete material properties will be amended.
- Composite stud data is lost you will therefore need to re-assign the data.
- Effective lengths will be set to 1.0L
- All existing design checks will be deleted and a re-design will be necessary.